Manual

SoftMaker Office 2024 and NX

PlanMaker

© 1987-2025 SoftMaker Software GmbH

Welcome! 22

Technical support ... 23
About this manual ... 24
System requirements ... 24

Installation and program startup 26

Installing on Windows ... 26
Installing on macOS ... 27
Installing on Linux ... 28
Installing on Android devices ... 28
Installing on iOS devices ... 28

Application window 29

Title bar ... 30
User interface: Ribbon ... 31
User interface: Classic menus ... 33
User interface on Android/iOS ... 34
Context menu ... 36
Edit toolbar ... 37
Document tabs ... 38
Worksheet register ... 39
Status bar ... 39

Basics 42

The structure of a worksheet ... 43
Moving within a worksheet ... 44
Entering data ... 44
Deleting data ... 44
Undoing changes ... 45
Starting a new document ... 46
Opening a document ... 48
Printing a document ... 50
Saving a document ... 50
Exiting the application ... 51

The PlanMaker tour 52

Some theory to begin with ... 52
First steps with PlanMaker ... 53
Entering values and formulas ... 54
Going three-dimensional ... 57
Changing the appearance of worksheets ... 57

Charts 59
Outlook 59

Editing worksheets 61

Entering data in cells 63
Undoing changes 66
Repeating commands 66
Selecting cells and cell content 67
Selecting in the Windows, Mac or Linux version 67
Selecting in the Android/iOS version 69
Deleting cells and cell content 70
Deleting the content of cells 70
Deleting whole cells 71
Deleting empty rows 72
Deleting duplicate rows 72
Inserting new cells 73
Inserting copied cells 74
Moving and copying 75
Paste (special) 77
Filling cells automatically 81
Working with multiple worksheets 85
Creating new worksheets 86
Working with the worksheet register 86
Managing worksheets 87
Calculating in three dimensions 89
Naming cell ranges 89
Assigning and editing names for cell ranges 90
Defining names automatically from the selection 91
Creating a list of all named ranges 92
Using named ranges 92
Applying names to existing formulas 93
Transposing cells 94
Splitting text into multiple columns 94
Sorting cells 95
Filtering cells by their content 99
AutoFilter 99
Special filter 104
Analyzing sheets 107
Showing formulas instead of results 107
Syntax highlighting 108
Watch window for cell content 109
Formula auditing 110
Displaying the relationships between cells 111
Detecting errors in calculations 112
Detecting invalid data in cells 113

Updating sheets 114
Using comments 116
Goal seek 118
Scenarios 120
Creating scenarios 120
Showing scenarios 121
Managing scenarios 122
Merging scenarios 122
Creating a scenario summary 123
Consolidating data 124
Consolidating data by position 125
Consolidating data by labels 128
Editing and updating consolidations 130
Tables in worksheets 132
Creating tables in worksheets 133
Ribbon tab "Table" 134
Removing tables from worksheets 135
Editing tables in worksheets 135
Formatting tables in worksheets 137
Using the total row of tables in worksheets 139
Tables in worksheets and cell references 141
Pivot tables 143
Creating a new pivot table 145
Creating a pivot table with existing data 146
Creating a pivot table with imported data 148
Starting with the pivot table field list 149
Exercises for structuring the field list 150
Working with the field list: Fields section 155
Working with the field list: Areas section 156
Further options in the field list 158
Configuring pivot tables with the field list 159
Field settings (Row/Column labels area) 160
Value settings (Values area) 165
Pivot table settings 168
Defer layout update 171
Sorting and filtering pivot tables 172
Grouping pivot tables 174
Updating pivot tables and changing data area 178
Deleting, copying, moving pivot tables 179
Freezing rows and columns 182
Inserting special characters 184

Formatting worksheets 186

Cell size 187
Changing cell sizes via the mouse 188

Changing cell sizes via commands ... 188
Showing and hiding rows/columns ... 189
Number format ... 190
List of all number formats ... 192
Working with custom number formats ... 195
Structure of a custom number format ... 196
Borders ... 201
Shading ... 203
Alignment ... 205
Protection ... 209
Character format ... 209
Font and font size ... 211
Text styles ... 212
Font color ... 214
Superscripts and subscripts ... 215
Letter spacing and character pitch ... 216
Kerning ... 216
Resetting the character formatting ... 217
Paragraph format (text frames only) ... 217
Paragraph indents (text frames only) ... 218
Line spacing (text frames only) ... 218
Paragraph alignment (text frames only) ... 219
Spacing above/below a paragraph (text frames only) ... 220
Character styles ... 221
Creating character styles ... 221
Applying character styles ... 222
Changing character styles ... 222
Scope of character styles ... 223
"Normal" character style ... 224
"Based on" character styles ... 224
Cell styles ... 226
Applying cell styles ... 226
Creating cell styles ... 227
Editing cell styles ... 229
"Normal" cell style ... 229
Document templates ... 231
Creating document templates ... 231
Using document templates ... 232
Editing document templates ... 232
"Normal.pmvx" document template ... 233
AutoFormat ... 234
Conditional formatting ... 235
Creating new formatting rules ... 235
Types of conditional formatting ... 237
Managing formatting rules ... 240

Removing conditional formatting ... 242
Input validation ... 243
Settings for input validation ... 244
Detect invalid data ... 247
Format painter ... 248
Page setup ... 250
Setting page format ... 250
Setting page format options ... 252
Setting up header and footer ... 256
Adjusting page breaks manually ... 260

Searching and replacing 262

Search ... 262
Replace ... 263
Repeating a search/replace ... 264
Advanced search options ... 264
Jumping to specific cell addresses ... 266
Jumping to specific elements of a table ... 266

Objects 268

Objects – basic procedures ... 268
Inserting objects ... 269
Selecting objects ... 269
Object mode ... 270
Changing position and size of objects ... 271
Rotating and flipping objects ... 272
Changing the order of objects ... 273
Duplicating objects ... 274
Changing the names of objects ... 274
Aligning and distributing objects ... 274
Grouping objects ... 275
Hiding objects ... 276
Changing the AutoShape of objects ... 276
Combining shapes ... 277
Editing shapes retroactively ... 279
Rotation of the inner text ... 281
Vertical alignment of the inner text ... 281
Changing text area for the inner text ... 281
Showing the contents of a cell in objects ... 281
Changing object properties via the dialog box ... 283
Object properties, Format tab ... 283
Object properties, Fill tab ... 284
Object properties, Lines tab ... 286
Object properties, Shadow tab ... 287

Object properties, 3D tab 288
Object properties, Effects tab 288
Object properties, Properties tab 289
Object properties, AutoShapes tab 289
Additional tabs 290
Changing the default settings for objects 290
Text frames 292
Inserting text frames 292
Changing text frame properties via the ribbon tab 292
Changing text frame properties via the dialog box 294
Pictures 297
Inserting pictures 297
Scanning pictures 298
Inserting pictures from the gallery/camera (Android and iOS) 298
Compressing pictures 299
Changing picture properties via the ribbon tab 300
Changing picture properties via the dialog box 302
OLE objects 305
Inserting OLE objects 305
Editing OLE objects 307
Editing links to OLE objects 307
Changing OLE object properties via the ribbon tab 308
Changing OLE object properties via the dialog box 310
Using the SoftMaker Equation Editor 311
Drawings 313
Inserting drawings 313
Adding text to AutoShapes 315
Changing drawing properties via the ribbon tab 316
Changing drawing properties via the dialog box 318
AutoShapes tab (available only for AutoShapes) 318
Inner text tab (available only for AutoShapes with text) 319
Text tab (available only for TextArt objects) 320
Transformation tab (available only for TextArt objects) 320
QR codes/Barcodes 321

Charts 323

Inserting charts 324
Editing charts 324
Changing the chart type 326
Changing the arrangement of data series 331
Showing/hiding chart elements 331
Showing/hiding axes and axis titles 332
Showing/hiding chart titles 333
Showing/hiding data labels 334
Showing/hiding error bars 335

Showing/hiding gridlines 335
Showing/hiding the legend 336
Showing/hiding lines 337
Showing/hiding the trendline 338
Showing/hiding up/down bars 339
Editing chart elements 340
Chart area 342
Plot area 343
Walls (three-dimensional charts only) 344
Floor (three-dimensional charts only) 344
Corners (three-dimensional charts only) 345
Data series and data points 345
Trendlines 349
Category axis (X axis) 351
Value axis (Y axis) 352
Series axis (Z axis) 354
Axis title 355
Chart title 356
Gridlines 357
Legend 357
Changing chart properties via the dialog box 359
Chart properties, Format tab 359
Chart properties, Properties tab 359
Chart properties, Chart type tab 359
Chart properties, Data source tab 360
Chart properties, Series tab 360
Chart properties, Elements tab 361
Chart properties, 3D view tab 362
Chart properties, Radar tab 362
Updating charts 363
Moving charts to another worksheet 363
Saving charts as pictures 364

Forms 365

Working with form objects 366
Inserting form objects 366
Editing form objects 367
Using and evaluating form objects 367
Form objects and Excel macros and scripts 368
Form objects in detail 368
Checkboxes 368
Changing the properties of checkboxes 369
Radio buttons 370
Changing the properties of radio buttons 370
Dropdown lists 371

Changing the properties of dropdown lists ... 371
Listboxes ... 372
Changing the properties of listboxes ... 373
Pushbuttons ... 374
Changing the properties of pushbuttons ... 374
Spinners ... 374
Changing the properties of spinners ... 375
Scrollbars ... 375
Changing the properties of scrollbars ... 376
Labels and groupboxes ... 377
Changing the properties of labels and groupboxes ... 377
Options of the contextual ribbon tab "Object" ... 378

Language tools 379

Setting the language ... 379
Spell check ... 380
Manual spell check ... 380
Checking the spelling as you type ... 382
Editing user dictionaries ... 382
Hyphenation ... 384
Hyphenation in text frames ... 384
Hyphenation in table cells ... 385
Auto correction ... 386
Options tab ... 386
Auto correction tab ... 390
First letter tab ... 391
Initial Caps tab ... 391
SmartText ... 392
Creating SmartText entries ... 392
Inserting SmartText entries ... 393
Editing SmartText entries ... 394
Research ... 395
Read aloud ... 396

Document management 398

Quick paths ... 398
Document summary ... 400
File manager ... 401
Searching with the file manager ... 402
Backup copies (file versions) ... 405

Outlines 408

Grouping cells ... 409
Showing/hiding grouped cells ... 410

Changing outline settings 410

Using hyperlinks 412

Outputting documents 415

Print preview 415
Printing a document 416
Exporting a document as a PDF file 420
Sending a document by e-mail 424

Protecting cells and documents 426

Sheet protection 426
Step 1: Setting up protection settings for cells 426
Step 2: Enabling sheet protection 427
Disabling sheet protection 428
Workbook protection 429
Enabling workbook protection 429
Disabling workbook protection 430
Document protection 430
Enabling document protection 430
Disabling document protection 432

Working with other file formats 433

Saving and opening files in other file formats 433
Notes on the text format 435
Notes on the Excel format 438
Opening and saving Excel files 438
Differences between PlanMaker and Excel 440
Notes on the HTML format 440

Working with Arabic text 442

Enabling extended support for Arabic text 442
Changing the text direction in table cells 442
Changing the text direction in text frames 443
Changing the direction of worksheets 443

Scripts (BasicMaker) 445

Working with multiple documents 446

Creating or opening a document 447
Switching between open documents 447
Arranging documents on the screen 448

Closing a document 449

Customizing PlanMaker 450

Changing PlanMaker settings 451
Settings, View tab 451
Settings, General tab 453
Settings, Edit tab 456
Settings, Appearance tab 458
Settings, Language tab 462
Settings, Files tab 463
Settings, Backup tab 464
Settings, System tab 465
Settings, Fonts tab 468
"Manage" button 469
Exporting/importing settings 470
Changing document properties 473
Document properties, Summary tab 473
Document properties, Colors tab 473
Document properties, Statistics tab 475
Document properties, Options tab 475
Document properties, Calculate tab 477
Document properties, Protection tab 480
Document properties, Fonts tab 480
Changing worksheet properties 481
Customizing the document display 483
Zoom level 483
Full screen view 484
Dark document mode 485
Customizing the ribbon 486
Showing and hiding the ribbon 487
Customizing icons and groups in the ribbon 488
Changing the position of the Quick access toolbar 492
Customizing icons on the Quick access toolbar 492
Customizing toolbars 494
Showing and hiding toolbars 494
Positioning toolbars on the screen 496
Managing toolbars 497
Customizing toolbar icons 498
Using the toolbar menu 502
Creating user-defined icons 503
Customizing shortcut keys 503
Editing lists for automatic filling 506
Installing dictionaries 508
Installing additional dictionaries 508
Choosing a different dictionary for a language 509

Formulas and functions 511

Formula basics 512
Operators in formulas 512
Entering formulas 513
Relative versus absolute cell references 515
External cell references 516
Entering external cell references 517
Updating and managing external cell references 518
Error values 520
Working with arrays 522
Working with database functions 524
Functions from A to Z 526
ABS (absolute value) 526
ACOS (arccosine) 526
ACOSH (inverse hyperbolic cosine) 527
ACOT (arccotangent) 528
ACOTH (inverse hyperbolic cotangent) 528
ADDRESS (cell address as text) 529
AND (logical AND function) 530
AREAS (number of areas) 531
ASIN (arcsine) 531
ASINH (inverse hyperbolic sine) 532
ATAN (arctangent) 532
ATAN2 (arctangent 2) 533
ATANH (inverse hyperbolic tangent) 534
AVEDEV (average deviation) 534
AVERAGE (arithmetic mean) 535
AVERAGEA (arithmetic mean) 535
AVERAGEIF (average if condition is true) 536
AVERAGEIFS (average if conditions are true) 537
B (compatibility function) 538
BASE (convert decimal number into another base) 539
BESSELI (modified Bessel function In(x)) 540
BESSELJ (Bessel function Jn(x)) 541
BESSELK (modified Bessel function Kn(x)) 541
BESSELY (Bessel function Yn(x)) 542
BETADIST (beta distribution) 542
BETA.DIST (beta distribution) 543
BETAINV (percentiles of the beta distribution) 544
BETA.INV (percentiles of the beta distribution) 544
BIN2DEC (binary number to decimal number) 545
BIN2HEX (binary number to hexadecimal number) 546
BIN2OCT (binary number to octal number) 547
BINOMDIST (binomial distribution) 548

BINOM.DIST (binomial distribution) 549
BINOM.DIST.RANGE (binomial distribution) 550
BINOM.INV (binomial distribution) 551
CEILING (round up to a multiple of base) 552
CELL (information about a cell) 553
CHAR (character from ANSI code) 554
CHIDIST (chi-square distribution) 555
CHIINV (percentiles of the chi-square distribution) 555
CHISQ.DIST.RT (chi-square distribution) 556
CHISQ.INV.RT (percentiles of the chi-square distribution) 557
CHISQ.TEST (chi-square test for independence) 557
CHITEST (chi-square test for independence) 558
CHOICE (x>0, x=0, x<0?) 558
CHOOSE (choose value from list) 559
CLEAN (remove non-printable characters) 560
CODE (ANSI code of a character) 560
COLUMN (column number of cells) 561
COLUMNS (number of columns in a range) 561
COMBIN (combinations) 562
COMPLEX (create complex number) 563
CONCAT (concatenate strings from list) 563
CONCATENATE (concatenate text strings) 564
CONFIDENCE (confidence interval) 565
CONFIDENCE.NORM (confidence interval) 566
CONVERT (unit conversion) 567
CORREL (correlation coefficient) 571
COS (cosine) 571
COSH (hyperbolic cosine) 572
COT (cotangent) 572
COTH (hyperbolic cotangent) 573
COUNT (number of cells filled with numbers) 574
COUNTA (number of cells filled) 574
COUNTBLANK (number of empty cells) 575
COUNTIF (count if condition is true) 575
COUNTIFS (count if conditions are true) 576
COUNTP (PlanMaker 97 compatibility function) 578
COVAR (population covariance) 579
COVARIANCE.P (population covariance) 579
COVARIANCE.S (sample covariance) 580
CREATEDATE (date the document was created) 581
CRITBINOM (binomial distribution) 582
CUMIPMT (cumulative interest) 583
CUMPRINC (cumulative principal) 584
CURRENCY (format number as currency) 585
DATE (create a date value) 586

DATEDIF (date difference) 586
DATEVALUE (convert text into date value) 588
DAVERAGE (database function) 588
DAY (day of a date) 589
DAYS (days between two dates) 589
DAYS360 (days between two dates) 590
DAYSP (compatibility function) 591
DAYSPERMONTH (days per month) 592
DAYSPERYEAR (days per year) 593
DB (fixed-declining balance depreciation) 594
DCOUNT (database function) 594
DCOUNTA (database function) 595
DDB (double-declining balance depreciation) 596
DEC2BIN (decimal number to binary number) 596
DEC2HEX (decimal number to hexadecimal number) 597
DEC2OCT (decimal number to octal number) 598
DECIBEL (decibel value of two quantities) 599
DECIMAL (convert number from any base into decimal) 600
DECIMALS (fractional part of a number) 601
DEGREES (convert radians to degrees) 601
DELTA (test for equality) 602
DEVSQ (average square deviation) 603
DGET (database function) 603
DMAX (database function) 604
DMIN (database function) 604
DOLLARDE (convert to dollar price decimal) 605
DOLLARFR (convert to dollar price fraction) 606
DPRODUCT (database function) 606
DSTDEV (database function) 607
DSTDEVP (database function) 608
DSUM (database function) 608
DVAR (database function) 609
DVARP (database function) 610
EDATE (date before/after n months) 610
EFFECT (effective interest rate) 611
EOMONTH (end of month in n months) 612
ERF (Gaussian error function) 612
ERFC (complement to Gaussian error function) 613
ERROR.TYPE (error type) 613
EUROCONVERT (convert EU currencies) 614
EVEN (round up to next even number) 616
EXACT (compare texts) 616
EXP (power of e) 617
EXPONDIST (exponential distribution) 618
EXPON.DIST (exponential distribution) 618

FACT (factorial) 619
FACTDOUBLE (double factorial) 620
FALSE (logical value FALSE) 620
FDIST (F distribution) 621
F.DIST.RT (F distribution) 621
FILENAME (file name of the document) 622
FIND (search for text in a text string) 623
FINV (percentiles of the F distribution) 623
F.INV.RT (percentiles of the F distribution) 624
FISHER (Fisher transformation) 625
FISHERINV (inverse of the Fisher transformation) 625
FIXED (format number as text with fixed decimals) 626
FLOOR (round down to a multiple of base) 627
FORECAST (forecast using linear regression) 628
FORMULATEXT (display formula contained in cell) 629
FREQUENCY (frequency) 629
FTEST (F-test) 630
F.TEST (F-test) 631
FV (future value) 631
FVSCHEDULE (future value) 632
GAMMA (gamma function) 633
GAMMADIST (gamma distribution) 634
GAMMA.DIST (gamma distribution) 634
GAMMAINV (percentiles of the gamma distribution) 635
GAMMA.INV (percentiles of the gamma distribution) 636
GAMMALN (logarithm of the gamma function) 636
GAUSS (standard normal distribution) 637
GCD (greatest common divisor) 637
GEOMEAN (geometric mean) 638
GESTEP (number greater or equal to threshold value?) 639
GETPIVOTDATA 639
GROWTH (values of an exponential regression) 640
HARMEAN (harmonic mean) 641
HEX2BIN (hexadecimal number to binary number) 642
HEX2DEC (hexadecimal number to decimal number) 643
HEX2OCT (hexadecimal number to octal number) 644
HLOOKUP (look up row-wise) 645
HOLIDAY (dates of movable Christian holidays) 646
HOUR (hour) 647
HYPERLINK (hyperlink) 648
HYPGEOMDIST (hypergeometric distribution) 649
HYPGEOM.DIST (hypergeometric distribution) 650
IF (if-then-else condition) 651
IFEMPTY (return a value if cell is empty) 652
IFERROR (return a value if error) 652

IFNA (return defined value for #N/A) 653
IFS (first condition met) 654
IMABS (absolute value of a complex number) 655
IMAGINARY (imaginary coefficient of a complex number) 655
IMARGUMENT (angle of a complex number) 656
IMCONJUGATE (conjugate a complex number) 657
IMCOS (cosine of a complex number) 657
IMDIV (division of complex numbers) 658
IMEXP (exponential of a complex number) 658
IMLN (natural logarithm of a complex number) 659
IMLOG10 (base-10 logarithm of a complex number) 659
IMLOG2 (base-2 logarithm of a complex number) 660
IMNEG (negative value of a complex number) 660
IMPOWER (power of a complex number) 661
IMPRODUCT (product of complex numbers) 661
IMREAL (real coefficient of a complex number) 662
IMSIN (sine of a complex number) 663
IMSQRT (square root of a complex number) 663
IMSUB (difference of complex numbers) 664
IMSUM (sum of complex numbers) 664
INDEX (cell in a particular row/column of a range) 665
INDIRECT (create reference from text) 666
INT (round down to nearest integer) 667
INTERCEPT (intercept point of a regression line) 667
INTERSECTION (intersection of two ranges) 668
IPMT (interest payment) 669
IRR (internal rate of return) 670
ISBLANK (is empty?) 671
ISERR (is an error value other than #N/A?) 672
ISERROR (is an error value?) 672
ISEVEN (is an even number?) 673
ISFORMULA (is a formula?) 673
ISLOGICAL (is a logical value?) 674
ISNA (is not available?) 675
ISNONTEXT (is no text?) 675
ISNUMBER (is numeric?) 676
ISNUMBERP (PlanMaker 97 compatibility function) 677
ISODD (is an odd number?) 678
ISOWEEK (ISO week number) 678
ISOWEEKNUM (ISO week number) 679
ISPMT (interest payment) 680
ISREF (is a valid reference?) 681
ISTEXT (is text?) 681
KURT (kurtosis) 682
LARGE (k-th largest number) 683

LASTPRINTED (date last printed) 684
LASTSAVED (date last saved) 684
LCM (least common multiple) 685
LEFT (left part of a text string) 685
LEN (length) 686
LINEST (statistics of a linear regression) 687
LN (natural logarithm) 689
LOG (logarithm) 689
LOG10 (base-10 logarithm) 690
LOGEST (statistics of an exponential regression) 691
LOGINV (percentiles of the log-normal distribution) 692
LOGNORMDIST (log-normal distribution) 693
LOGNORM.DIST (log-normal distribution) 694
LOGNORM.INV (percentiles of the log-normal distribution) 695
LOOKUP (search cell range) 695
LOWER (convert text to lowercase) 698
MATCH (relative position in a range) 699
MAX (maximum) 699
MAXA (maximum) 700
MAXIFS (conditional maximum) 701
MDETERM (matrix determinant) 702
MEDIAN (Median) 703
MID (part of a text string) 703
MILLISECONDS (milliseconds) 704
MIN (minimum) 705
MINA (minimum) 705
MINIFS (conditional minimum) 706
MINUTE (minute) 707
MINVERSE (inverse matrix) 708
MIRR (modified internal rate of return) 709
MMULT (product of matrices) 709
MOD (remainder of a division, Excel method) 710
MODE (most frequently occurring value) 711
MODE.SNGL (most frequently occurring value) 712
MODP (remainder of a division, PlanMaker method) 713
MONTH (month of a date) 714
MROUND (round to a multiple of base) 714
MSOLVE (solution of matrix equation Ax=B) 715
MULTINOMIAL (multinomial coefficient) 716
N (convert value into number) 717
NA (error value #N/A) 718
NEG (negative value) 719
NEGBINOM.DIST (negative binomial distribution) 719
NEGBINOMDIST (negative binomial distribution) 720
NETWORKDAYS (number of workdays) 721

NOMINAL (nominal interest rate) 722
NORMDIST (normal distribution) 722
NORM.DIST (normal distribution) 723
NORMINV (percentiles of the normal distribution) 724
NORM.INV (percentiles of the normal distribution) 725
NORMSDIST (standard normal distribution) 726
NORM.S.DIST (standard normal distribution) 727
NORMSINV (percentiles of the standard distribution) 727
NORM.S.INV (percentiles of the standard distribution) 728
NOT (logical NOT function) 729
NOW (current date and time) 729
NPER (number of periods) 730
NPV (net present value) 731
NUMBERVALUE (convert text into a number, locale) 732
OCT2BIN (octal number to binary number) 734
OCT2DEC (octal number to decimal number) 735
OCT2HEX (octal number to hexadecimal number) 735
ODD (round up to next odd number) 736
OFFSET (shifted reference) 737
OR (logical OR function) 738
PEARSON (Pearson correlation coefficient) 738
PERCENTILE (percentiles of a data set) 739
PERCENTILE.EXC (percentiles of a data set) 740
PERCENTILE.INC (percentiles of a data set) 741
PERCENTRANK (percent rank in a data set) 742
PERCENTRANK.EXC (percent rank in a data set) 742
PERCENTRANK.INC (percent rank in a data set) 743
PERIOD (duration of fixed-interest investments) 744
PERMUT (permutations) 745
PHI (standard normal distribution) 746
PI (constant value of pi) 747
PMT (payment) 747
POISSON (Poisson distribution) 748
POISSON.DIST (Poisson distribution) 749
POLYNOMIAL (polynomial coefficient) 751
POWER (power) 751
PPMT (payment on the principal) 752
PROB (probability) 753
PRODUCT (product) 754
PROPER (convert text to upper/lower case) 755
PV (present value) 755
QUARTILE (quartiles of a data set) 756
QUARTILE.EXC (quartiles of a data set) 757
QUARTILE.INC (quartiles of a data set) 758
QUOTIENT (quotient of a division) 759

RADIANS (convert degrees to radians) 759
RAND (random value) 760
RANDBETWEEN (random value) 761
RANK (rank in a data set) 761
RANK.AVG (rank in a data set) 762
RANK.EQ (rank in a data set) 763
RATE (interest rate per period) 764
REPLACE (replace text in a text string) 765
REPT (repeat text string) 766
RIGHT (right part of a text string) 766
ROMAN (Roman numeral) 767
ROOTN (n-th root) 768
ROUND (round to n digits) 768
ROUNDDOWN (round down to n digits) 769
ROUNDUP (round up to n digits) 770
ROW (row number of cells) 771
ROWS (number of rows in a range) 771
RRI (interest for fixed-interest investments) 772
RSQ (square of Pearson) 773
SEARCH (search for text in a text string) 773
SECOND (second) 774
SERIESSUM (sum of a power series) 775
SHEET (index of a worksheet) 775
SHEETNAME (name of a worksheet) 776
SHEETNUMBER (compatibility function) 777
SHEETS (number of worksheets) 777
SIGN (sign of a number) 778
SIN (sine) 779
SINH (hyperbolic sine) 779
SKEW (skewness of a distribution) 780
SLN (straight-line depreciation) 781
SLOPE (slope of a linear trend) 781
SMALL (k-th smallest number) 782
SORTM (sort) 783
SORTV (sort) 784
SQRT (square root) 786
SQRTPI (square root of x*Pi) 786
STANDARDIZE (standardized value) 787
STDEV (standard deviation of a sample) 787
STDEVA (standard deviation of a sample) 788
STDEVP (standard deviation of entire population) 789
STDEV.P (standard deviation of entire population) 790
STDEVPA (standard deviation of entire population) 790
STDEV.S (standard deviation of a sample) 791
STEYX (standard error of a linear regression) 792

SUBSTITUTE (replace text in a text string) 793
SUBTOTAL (calculations without hidden cells) 794
SUM (Sum) 795
SUMIF (sum if condition is true) 797
SUMIFS (sum if conditions are true) 798
SUMPRODUCT (sum of products) 799
SUMSQ (sum of squares) 800
SUMX2MY2 (sum of x^2 - y^2) 801
SUMX2PY2 (sum of x^2 + y^2) 802
SUMXMY2 (sum of (x - y)^2) 802
SWITCH (categorize initial value) 803
SYD (sum-of-years' digits depreciation) 804
T (convert value into text string) 805
TAN (tangent) 806
TANH (hyperbolic tangent) 807
TDIST (t-distribution) 807
T.DIST (t-distribution) 808
T.DIST.2T (t-distribution two-tailed) 808
T.DIST.RT (t-distribution right-tailed) 809
TEXT (convert number into text string) 810
TEXTJOIN (concatenate strings with separators) 810
TIME (create time) 811
TIMEDIFF (time difference) 812
TIMEVALUE (convert text into time) 813
TINV (percentiles of the t-distribution) 813
T.INV (percentiles of the t-distribution left-tailed) 814
T.INV.2T (percentiles of the t-distribution two-tailed) 815
TODAY (current date) 816
TRANSPOSE (transpose matrix) 816
TREND (values of a linear regression) 817
TRIM (remove spaces from text) 818
TRIMMEAN (mean without marginal values) 819
TRUE (logical value TRUE) 820
TRUNC (truncate a number) 820
TTEST (t-test) 821
T.TEST (t-test) 822
TYPE (type of a value) 822
UPPER (convert text to uppercase) 823
USERFIELD (output user info) 824
VALUE (convert text into a number) 825
VAR (variance of a sample) 826
VARA (variance of a sample) 827
VARP (variance of entire population) 827
VAR.P (variance of entire population) 828
VARPA (variance of entire population) 829

VAR.S (variance of a sample) 830
VLOOKUP (look up column-wise) 830
WEEKDAY (weekday) 832
WEEKNUM (week number) 833
WEIBULL (Weibull distribution) 834
WEIBULL.DIST (Weibull distribution) 835
WORKDAY (date after x workdays) 836
XIRR (internal rate of return) 837
XNPV (net present value) 837
XOR (logical XOR function) 838
YEAR (year of a date) 839
ZTEST (z-test) 839
Z.TEST (z-test) 840

Appendix 841

Ribbon commands and corresponding menu commands 841
Shortcut keys 857
Shortcut keys in Windows and Linux versions 857
Shortcut keys in Mac version 861
Command-line parameters 866

Index 868

Welcome!

Welcome to PlanMaker! You have purchased an extremely powerful spreadsheet application that is also easy to use.

In developing this program, we have made every effort to integrate and coordinate PlanMaker's functions in such a manner that you will obtain the maximum benefit with the minimum amount of work.

PlanMaker gives you enough freedom to meet your individual needs. You can avail of the numerous setting options to set up PlanMaker exactly to your liking.

Note: This manual was written in order to describe how to use the program via the new **ribbon user interface**. A description of its use via **classic menus with toolbars** can only be found in older manuals.

Tip: A table in the appendix shows you which ribbon command corresponds to which menu command: Ribbon commands and corresponding menu commands

Some features of PlanMaker

- Available for **Windows**, **Mac**, **Linux, Android** and **iOS**
- Well-designed, ergonomic **user interface** – either with a modern "ribbon" or classic menus with toolbars
- PlanMaker knows **more than 400 computational functions** – from simply calculating totals to numerous financial and statistical functions.
- The creation of attractive worksheets is facilitated by the various **design options**. Simple columns of numbers can be transformed into clear and appealing worksheets at the touch of a button using the **"Tables in worksheets"** feature. You can use **cell styles** and **character styles** to apply frequently needed formatting at the touch of a button.
- The integrated **chart module** lets you present numbers clearly in charts.
- ... and so much more!

PlanMaker is being developed continually. If, when working, you discover the need for a feature that is not yet available, or if you have any other suggestions, please contact us as we want PlanMaker to meet the needs of users!

Additional features in SoftMaker Office Professional and NX Universal

The even more powerful **SoftMaker Office Professional** and **SoftMaker Office NX Universal** include an advanced version of PlanMaker which offers the following additional features:

- **Read aloud** – have your document content read aloud to you
- **Compress pictures** compresses inserted pictures to reduce the size of the file
- **QR code/Barcode** – generate your own QR codes or linear barcodes

Android version and iOS version: Unlock locked commands

PlanMaker is also available for Android and iOS devices and contains virtually all features of the Windows version.

Some commands are locked in the Android/iOS version (recognizable by the lock in the command icon), but they can be unlocked if necessary.

To unlock the commands, click on the command icon. The program opens a dialog box with the following options: Users of **SoftMaker Office NX Home** and **NX Universal** can use their product key to unlock commands. Users of other versions have the option of unlocking the features via Google Play/Apple App Store (subject to a fee).

Technical support

If you have any questions, our technical support team will be happy to assist you. You can reach us as follows:

Website

Our website provides the latest program updates, tips and tricks, free downloads and much more.

For more information, see: www.softmaker.com

Support forums

In our support forums, you can ask our technical support team any technical questions you may have, while also communicating with other users.

These forums can be found at: forum.softmaker.com

Inquiries to our support team

If you have any problems with our software, we will happily assist you.

You can post your questions here: www.softmaker.com/en/support/customer-support

About this manual

PlanMaker has a wide variety of features – but don't worry: you don't have to master all of them! While you're getting started, just use those you need. If you would like to make use of PlanMaker's more advanced features at a later stage, you can then read the relevant pages of the manual.

This manual is structured as follows:

- The chapter **Installation and program startup** deals with the installation of PlanMaker. You also learn how to start the program.
- The chapter **Application window** describes the individual components of PlanMaker's main window.
- The chapter **Basics** provides information on PlanMaker's most important functions and explains basic spreadsheet terminology.
- The chapter **The PlanMaker tour** deals with the spreadsheet application in tutorial form and shows you how to use PlanMaker by means of some practical examples.
- The chapter **Editing worksheets** marks the start of the reference section of this manual. These chapters are organized by subject and provide a detailed description of all functions of PlanMaker.

System requirements

In order to run this software, you require the following hardware and software:

Windows version

- Windows 11, 10, 8 or 7 (32-bit or 64-bit)

Mac version

- macOS as of version 10.14 (Mojave)
- Intel or ARM-based CPU

Linux version

- Any x86 Linux (64-bit)

Android version

- Android 9.0 or higher
- ARM-compatible CPU (32-bit or 64-bit)

iOS version

- iOS 14.0 or higher

Installation and program startup

This chapter provides information on installing and starting PlanMaker.

It is divided by operating system into the following sections:

- Installing on Windows
- Installing on macOS
- Installing on Linux
- Installing on Android devices
- Installing on iOS devices

Please read the section for your operating system.

Installing on Windows

Download

If you obtained the software by *downloading* it from our website, you will find installation instructions in the e-mail that you automatically received after purchasing the software.

CD-ROM

If you obtained the software on a *CD-ROM*, please start the installation program provided in the root folder of the CD. Then follow the installation program's instructions to install the software.

Starting the programs

To start the installed programs, use the **Start** menu (the button with the Windows logo) in the lower left corner of the screen. You will find your SoftMaker Office applications in a sub-folder called **SoftMaker Office**.

Entering address upon initial start: When you start PlanMaker for the first time, you will be asked to enter your name, address, etc. This information is not used for registering the program. In fact, it allows you to insert your name, address, etc., in documents or document templates using the USERFIELD function. You can always change this information at a later stage (see Settings, General tab).

Installing on macOS

You will find information about installing the software in the e-mail that you automatically received after purchasing it.

Is an error message displayed when you start the installation program?

Depending on your system settings, the following error message may be displayed when you try to start the installation program:

This occurs when your system's security settings specify that only programs that have been downloaded from the App Store may be run.

Clicking on the question mark in the upper right corner of the message reveals how you can still run the installation program. You then see a help window in which the following procedure is recommended:

1. Close the error message.
2. Hold down the **Ctrl** key and click on the installation program.
3. A context menu is displayed. Choose the command **Open**.
4. You then see a message which asks you if you really want to open the program. Confirm this by clicking on the **Open** button.

The installation program will now start.

Starting the programs

After the installation is complete, icons for each of the installed programs will be displayed in both the Launchpad and the Dock at the bottom of the screen. To start one of the programs, simply click on its icon.

Installing on Linux

You will find information about installing the software in the e-mail that you automatically received after purchasing it.

Starting the programs

In most Linux distributions, the installation program automatically creates shortcut icons for all SoftMaker Office applications in the menu. To start any of the applications, click on the corresponding icon.

Alternatively, the following scripts are available for starting the programs:

- **textmaker24** starts TextMaker.
- **planmaker24** starts PlanMaker.
- **presentations24** starts Presentations.

You can execute these scripts in a shell, for example.

Entering address upon initial start: When you start PlanMaker for the first time, you will be asked to enter your name, address, etc. This information is not used for registering the program. In fact, it allows you to insert your name, address, etc., in documents or document templates using the USERFIELD function. You can always change this information at a later stage (see Settings, General tab).

Installing on Android devices

If you download an application to your Android device via the **Google Play Store**, you don't need to do anything, as the application will be automatically installed immediately after downloading it.

Installing on iOS devices

If you download an application to your iOS device via the **Apple App Store**, you don't need to do anything, as the application will be automatically installed immediately after downloading it.

Application window

The following pages contain an introduction to the individual components of the PlanMaker user interface, both for the *ribbon interface* and for the *classic menu interface*.

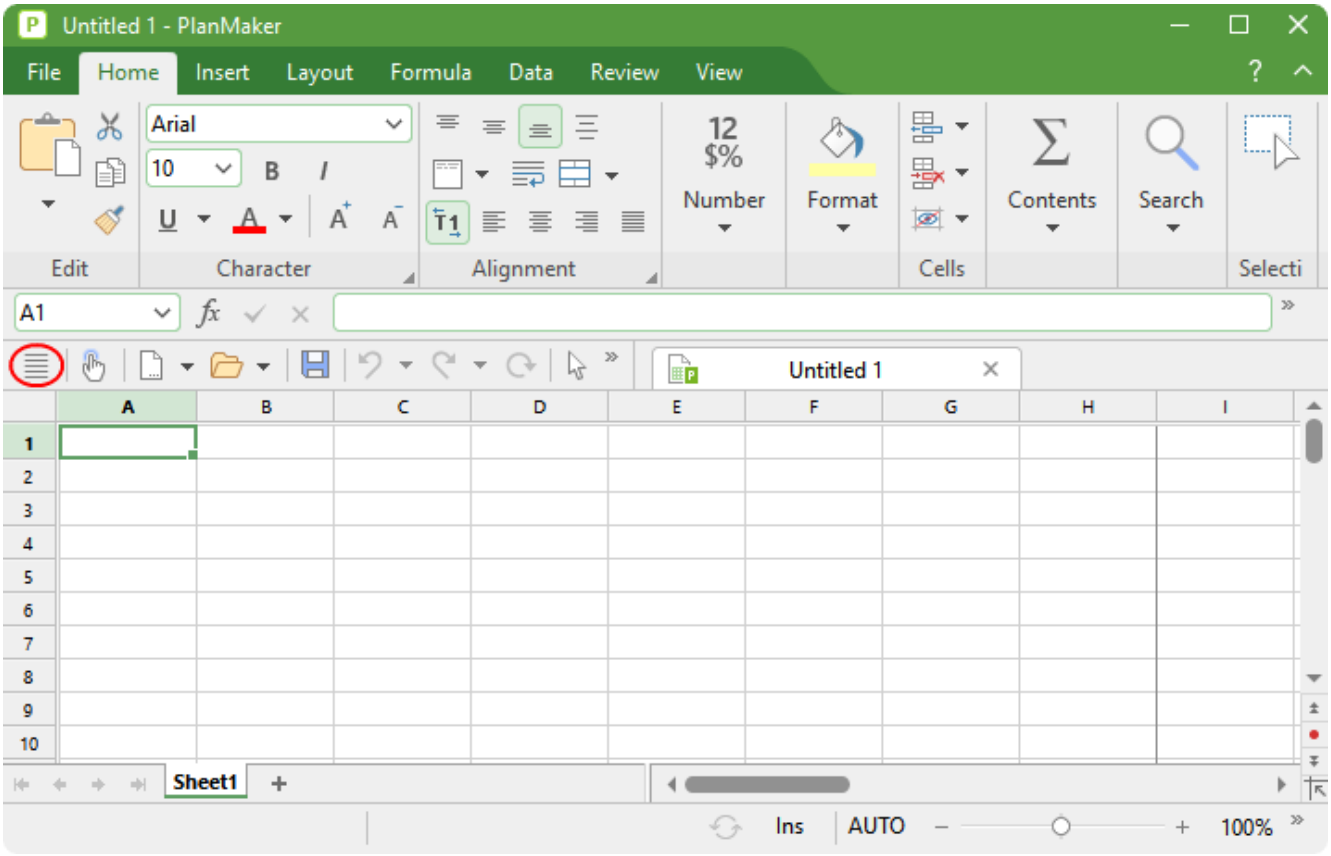
In all other chapters, this manual is designed for the user interface with the **ribbon**.

If you would like to continue working with the **classic menu interface**, a table in the appendix shows you which ribbon command corresponds to which menu command: Ribbon commands and corresponding menu commands.

Tip: You can switch the user interface between *Ribbon* and *Classic menus* at any time. To do so, choose the command **File | Options** in the ribbon (or choose **Tools > Options** in the classic menu). On the **Appearance** tab, click on the **User interface** button.

Alternatively: Right-click on a free space in the ribbon or toolbar, and choose **User interface**.

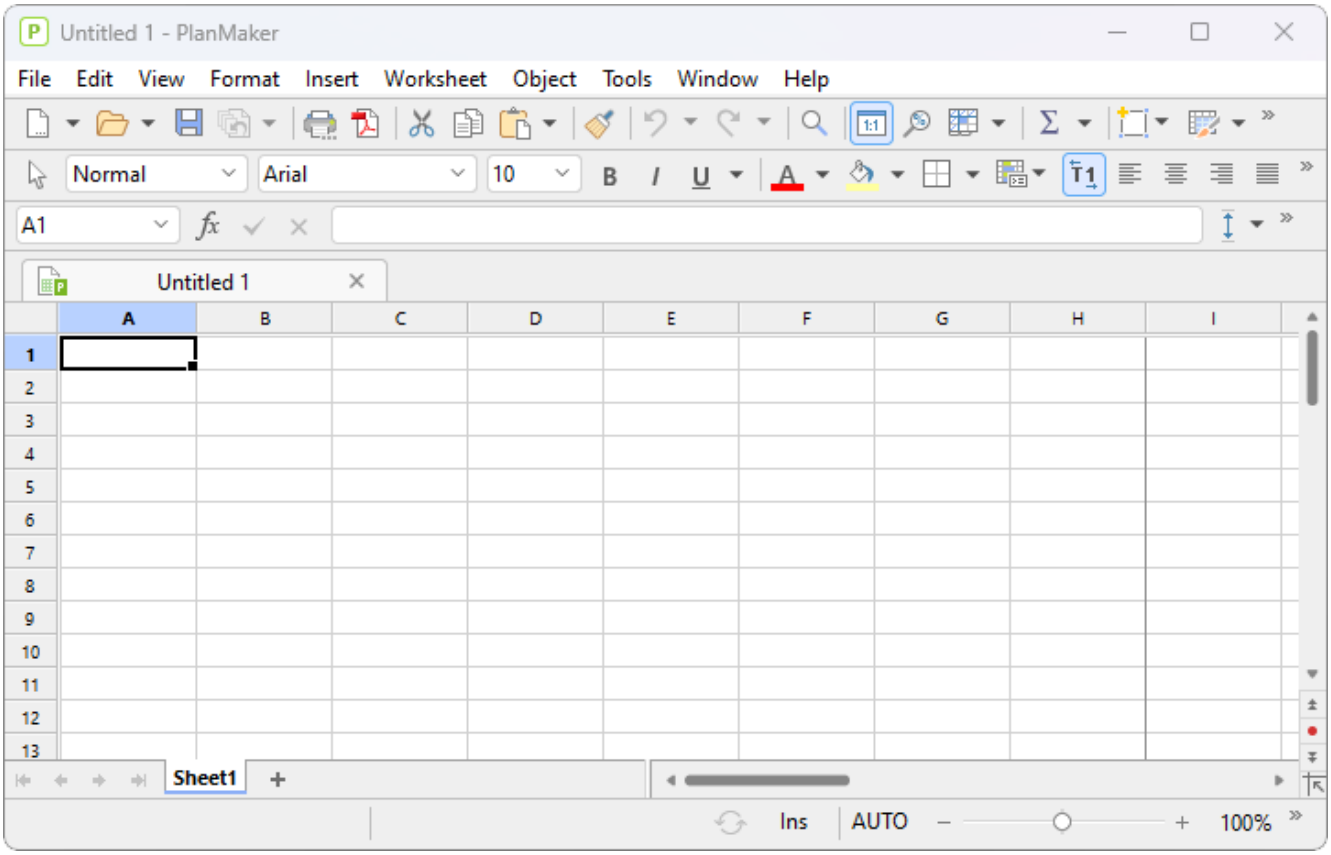
Program window with the "Ribbon" user interface



PlanMaker with the "Ribbon" user interface (Windows version)

Tip: Even when using the ribbon user interface, the classic menu commands are always accessible in the ribbon view. To access them, use the "hamburger menu" (the icon ☰ to the left of the Quick access toolbar, highlighted in red). There, you will find all familiar menu commands from the classic menu user interface.

Program window with the "Classic menus" user interface



PlanMaker with the "Classic menus" user interface (Windows version)

Note: Most of the illustrations in this manual were produced with the **Windows** version of PlanMaker. On other operating systems, some of the controls look slightly different, but their functionality is the same.

Title bar

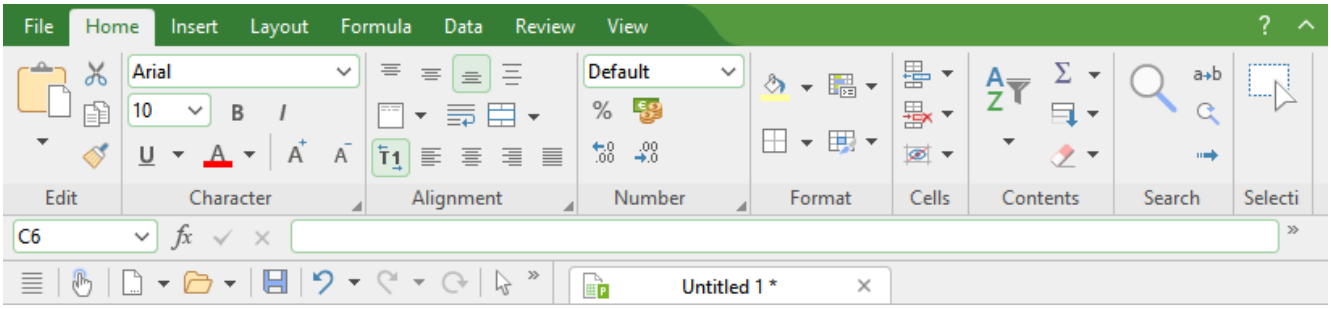
In the top line of the program, you will find the *title bar*.

This displays the program name and the name of the document that you are currently editing.

If the document contains changes that have not yet been saved, an asterisk is displayed after its name.

User interface: Ribbon

If, in the settings, you have selected that you would prefer to use the *Ribbon* interface instead of *Classic menus*, the upper area of the program window will look like this on your device:



The "Ribbon" and the Quick access toolbar (below)

The ribbon

Ribbons are a modern type of user interface that combines menus and toolbars to simplify the use of the software.

As you can see, the ribbon has several tabs whose tab headers are displayed at the top: **File**, **Home**, **Insert**, etc. Each of these "ribbon tabs" contains logically combined groups (for example, on the ribbon tab **Home**: *Edit, Character, Alignment*, etc.) with related commands for a certain task area.

To switch to another ribbon tab, click on its tab header.

Tip: You can also switch between the ribbon tabs with a shortcut key: Use **Ctrl+F12** to move to the next tab and **Ctrl+Shift+F12** to move to the previous tab. You can also scroll with the middle mouse wheel while pointing to the top bar with the tabs.

Each icon represents a specific command. If you click on it, the corresponding command is chosen. In the group **Character** on the ribbon tab **Home**, for example, you will find commands for formatting. You can use them to both read and change the most common text formatting (font, bold, italics, etc.) for table contents.

If you select a section of text or a cell beforehand, formatting changes will affect only the selected part.

To choose a different font, for example, click on the small arrow to the right of the font name to open the list, and then choose a font.

Other icons are switches that you can enable or disable by clicking on them – for example, the **B** for bold.

There are also split buttons, where you can either click on the icon to directly apply the displayed command or use the corresponding small arrow to the right of the icon to open its dropdown menu. This is the case, for example, when choosing the font color.

Tip: If you hover the mouse cursor over an icon (without clicking), the program will display a text box called a "tooltip", which describes the icon's function. This requires Show tooltips to have been enabled in the settings.

The group arrows ◢ in the bottom right corner of each command group indicate that there are other options and commands available for that group. Simply click on this arrow to open the dialog box belonging to the group, where you can make further settings.

The Quick access toolbar

The *Quick access toolbar* is displayed below the ribbon. It provides a selection of frequently used commands.

- **Touch mode**

  If you enable this button, all icons in the ribbon will be slightly enlarged. This is useful when operating the software with your finger (for example, on a tablet).

- **Dark/light document mode** (only available in NX Home and NX Universal)

  Switch between the usual white or a dark display of the worksheets.

- **Create a new file**

  This button opens a new document. For more information, see Starting a new document.

- **Open a file**

  This button opens an existing document. For more information, see Opening a document.

- **Save file**

  This button saves the document that is open in the current window. For more information, see Saving a document.

- **Undo**

  This button undoes the last change made to the current document. You can also choose this command several times in succession, see Undoing changes.

- **Redo**

  This button restores the last undone change(s). For more information, see Undoing changes.

- **Repeat last command**

  This button repeats the execution of the last command chosen with exactly the same settings. For more information, see Repeating commands.

- **Object mode**

  Switch between edit and object mode to make it easier to edit either cells or objects. For more information, see Object mode.

To the right of the Quick access toolbar, there is a double arrow ». You can use its menu to add/remove common buttons directly from the Quick access toolbar. You can also change the position of the Quick access toolbar here and add additional icons via the command **Customize** (for Quick access toolbar and ribbon, see below).

To the left of the Quick access toolbar, you will find the **"hamburger-menu"** ☰. Even if you have selected the "Ribbon" as the user interface, the "hamburger menu" is still available in the Quick access toolbar in case you would like to access the menu commands of the classic menu interface.

Creating your own ribbon and Quick access toolbar

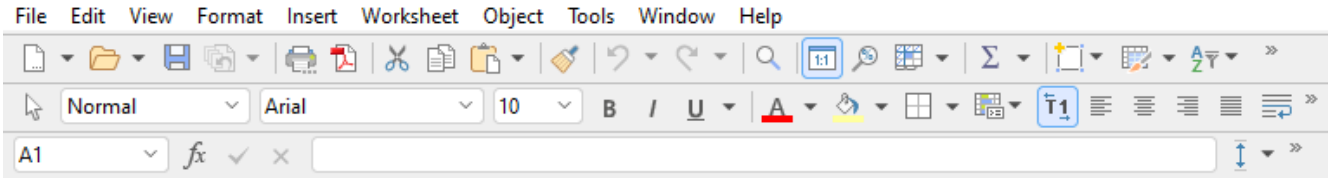
You can change the *ribbon* and the *Quick access toolbar* at will and add additional icons. To do so, either choose **File | Customize** in the ribbon or right-click on a free space in the ribbon. In the following menu, you will find the entries *Customize ribbon* or *Customize Quick access toolbar* to open a corresponding dialog box.

You can also hide the ribbon display completely (*Minimize ribbon*) or change the position of the Quick access toolbar.

For more information, see Customizing the ribbon.

User interface: Classic menus

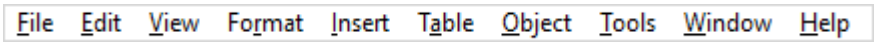
If, in the settings, you have selected that you would prefer to use *Classic menus* instead of the *Ribbon* interface, the upper area of the program window will look like this on your device:



Classic menus with toolbars

The menu bar

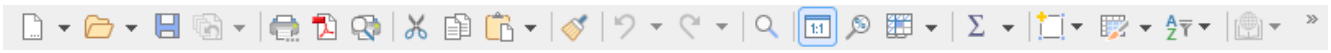
The *Menu bar* is found directly under the title bar.



It contains all of PlanMaker's commands in the form of clearly arranged *menus*. Click on a menu item to open a menu and choose a command.

The Standard toolbar

The *Standard toolbar* is displayed below the menu bar. It contains icons for PlanMaker's most frequently used commands.

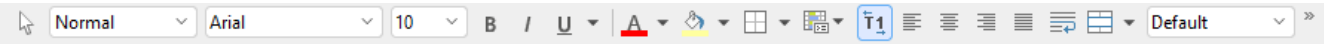


Toolbars such as the Standard toolbar enable quick access to the functions of a program. Each icon represents a specific command. If you click on it, the corresponding command is chosen.

Tip: If you hover the mouse cursor over an icon (without clicking), the program will display a tooltip that describes the icon's function. This requires Show tooltips to have been enabled in the settings.

The Formatting toolbar

The *Formatting toolbar* can be found below the toolbar. You can use it to both read and change the most common text formatting (font, bold, italics, etc.) for table contents.



If you select a section of text or a cell beforehand, formatting changes will affect only the selected part.

To choose a different font, for example, click on the small arrow to the right of the font name to open the list, and then choose a font.

Other icons in the Formatting toolbar are switches that you can enable and disable by clicking on them – for example, the **B** for bold.

There are also split buttons, where you can either click on the icon to directly apply the displayed command or use the corresponding small arrow to the right of the icon to open its dropdown menu. This is the case, for example, when choosing the font color.

Additional toolbars

There are additional toolbars in PlanMaker that you can enable and disable as you choose. To do so, either choose the menu command **View > Toolbars** or *right-click* on one of the displayed toolbars. You then see a menu in which you can select toolbars that you would like to have displayed.

Customizing toolbars

You can customize the default toolbars to your liking and even create your own toolbars. For more information, see Customizing toolbars.

User interface on Android/iOS

In addition to the *Ribbon* and *Classic menus* user interfaces, the Android/iOS version provides another option: the *Toolbars* user interface.

The **toolbars** interface is designed in a manner that makes it easier to use PlanMaker on smartphones and tablets. It constitutes a mixed form of the functionalities of ribbon and classic menus. On smartphones, PlanMaker is started by default with this **toolbars** interface the first time you open it, while it is opened by default with the **ribbon** interface on tablets.

To switch the interface, choose the command **File | Options** in the ribbon interface or choose **Tools > Options** in the classic menu interface. In the following dialog box, the **Appearance** tab contains the dropdown list **User interface**. Here, you can choose between toolbars, ribbon or classic menus with different colors.

Tip 1: In the *toolbars* interface, you reach this dialog box by opening the "hamburger menu" to the left of the toolbar and choosing the command **Tools > Options**.

Tip 2: The hamburger menu also contains all other menu commands of the classic menu interface.

Using the "toolbars" user interface

The **toolbars** user interface contains a bar with icons at the bottom of the program window, and each icon represents a group of commands. If you click on a group icon, another bar opens with the corresponding group commands.

Behind the **File** group icon , for example, you will find the commands *Open document*, *Save document*, *Print document*, etc. To hide the displayed bar with the commands, simply click on the group icon again.

By default, the toolbar contains the following groups:

	Group	Commands (among others)
	File	Open document, Save document, Print document, etc.
	Edit	Undo, Format painter, Copy & Paste, Search, etc.
	Worksheet	Filter and sort, Cells (insert, delete, hide), Manage worksheets
	Character	Font, Font size, Font color, Text styles (bold, italics, etc.)
	Formatting	Number formats, Cell formatting, Cell styles, Conditional formatting, etc.
	Insert	Table style, Objects (picture, chart, AutoShape, etc.), Comments, Hyperlinks
	Layout	Page setup, Headers and footers, Align objects, etc.
	View	Object mode, Zoom, Freeze top row/first column, etc.

"Keyboard" button

In **Options** , the **System** tab contains an option **Show/hide keyboard automatically**. Once this setting is enabled, it automatically opens the on-screen keyboard when you type something into the document.

If you disable the option, this behavior is suppressed and a freely movable **Keyboard** button is displayed instead on the document. If you click on the button, the keyboard is shown. By clicking on the button again, you hide the keyboard.

Note: This setting is possible in any user interface for smartphones and tablets. For *tablets*, the Keyboard button is only freely movable on the document in the **toolbars** interface if you disable the above option. In the **Ribbon** and **Classic menus** interfaces, the Keyboard button is displayed instead in a fixed manner in the upper left area of the command bar.

Changing input mode

PlanMaker supports the use of certain input modes of Android/iOS devices.

- **Android**

  The command **Input method** enables you to switch to a different keyboard mode on an Android device. PlanMaker accesses the on-screen keyboard settings of your Android device for this purpose.

In the classic menus interface, you change the mode with the menu command **View > Input method**. In the ribbon or toolbars user interfaces, you can choose this command via the hamburger menu ☰.

- **iOS**

  If **Scribble** mode is enabled in PlanMaker, you can write directly in a document on an iOS tablet using the Apple Pencil. Your handwritten entries will then be converted directly into text. For other edits – for example, those of objects – you have to disable the mode again.

  In the classic menus interface, you enable or disable the mode with the menu command **View > Scribble**. In the ribbon or toolbars user interfaces, you can choose this command via the hamburger menu ☰.

  Note: The command **View > Scribble** is only available for iOS tablets, but not for smartphones. Please also ensure that you have enabled the option *Apple Pencil* > *Scribble* in the system settings of your iOS tablet.

Unlocking locked commands

Some commands are locked in the Android/iOS version (recognizable by the lock in the command icon), but they can be unlocked if necessary.

To unlock the commands, click on the command icon. The program opens a dialog box with the following options: Users of **SoftMaker Office NX Home** and **NX Universal** can use their product key to unlock commands. Users of other versions have the option of unlocking the features via Google Play/Apple App Store (subject to a fee).

Customizing command groups

You can customize the default groups as desired by adding/removing commands. Note: However, you cannot add or delete an entire group in the **toolbars** user interface.

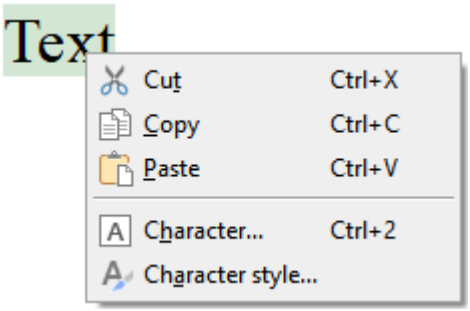
In the hamburger menu ☰, choose the command **Tools > Customize**. This opens the dialog box *Customize user interface*, which is structured as follows:

- On the left side, you will find all of the commands that are available in PlanMaker. On the right side, the dropdown list **Customize** contains the groups that are in the toolbar. The list below it displays the commands that are currently in the selected group.
- You can add new commands to each group (or remove them, change their position or insert separators). This works in a similar manner to that described for the **Classic menus with toolbars** user interface in the Customizing toolbar icons section. Difference: Instead of "*toolbar* icons", you customize "*group* icons".

Context menu

Regardless of the user interface in which you work, there is also another way to choose commands: via the **context menu**.

This menu contains different commands depending on the current situation. If, for example, you select some text and then **right-click** to open the context menu, it offers commands for cutting, copying or formatting that text.



Depending on what you have previously selected (text, objects, tables, etc.), you will be offered a variable – contextual – selection of commands when opening the context menu. This makes it much easier for you to quickly navigate to situational commands without a long search.

Android/iOS: In these versions, you can also open the context menu with your finger: Just tap on the screen and hold your finger there for about a second.

Edit toolbar

In the ribbon interface as well as in the classic menus you will find the *Edit toolbar* below the ribbon (or below the standard toolbar and formatting toolbar in the classic menu interface).

The Edit toolbar contains the following options:

Cell address

The address of the currently active cell is displayed on the far left.

Icons and input field for editing cell contents

Three icons and an input field can be found to the right of the cell address.

Click in the input field to edit the content of the current cell. Alternatively, you can edit the cell content *directly* by simply clicking in the cell and starting to type. The latter option is usually faster.

For more information on entering and editing data in cells, see Entering data in cells.

The buttons next to the cell address have the following functions:

Icon	Function
fx	This button corresponds to choosing the ribbon command **Formula | Function** (menu command **Insert > Function**)
✓	This button accepts the entry in the input field (corresponds to pressing the **Enter key**↵)

✕	This button rejects the entry in the input field (corresponds to pressing the **Esc** key)

For more information on entering formulas, see Formulas and functions.

Expanding the input field

The input field of the Edit toolbar is usually displayed as one line. For longer formulas, however, it is helpful if the input field expands so that you can see the entire formula.

You can set the behaviour of the input field as desired. To do so, click on the button ↕ to the right of the input field to open a menu in which you can choose from the following options:

- **Automatic**

  The input field is displayed as one line by default. If the space is no longer sufficient to display the formula in one line, the input field is automatically expanded to four lines.

- **One line**

  The input field is always displayed as one line, even if the formula exceeds one line.

- **Four lines**

  The input field is always displayed as four lines, even if the formula fits in one line.

Tip: These options can also be found in the dialog box that opens upon choosing the command **File | Options** and clicking on the **Edit** tab.

Note: If the aforementioned button is not available, click on the » icon on the far right of the Edit toolbar. Select **Add/remove buttons** and enable **Input field height**.

Document tabs

Android/iOS: This function is not available for *smartphones*. For *tablets*, actions are possible with the tabs within the same program window.

A bar with *document tabs* can be seen above the document. It displays one tab for each open document.

The document tabs bar

The tabs can be used to perform the following actions:

- To make a document become the active document, *left-click* its tab. This enables you to rapidly switch between the open documents.

- To close a document, click on its tab with the *middle* mouse button.

  Alternatively, left-click on the **x** icon displayed on the far right of each tab.

- You can change the order of the tabs by dragging a tab to the desired position on the tab bar while holding down the left mouse button.

- To release a tab and display it in a separate program window, drag it to a position outside the current program window while holding down the left mouse button.

  Of course, this can also be done in reverse: A separately opened program window can be dragged into the tab bar of an existing PlanMaker program window.

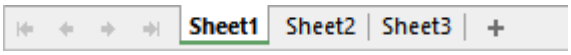
By the way, documents can also be started in a new program window when they are opened. For more information on working with multiple documents, see Working with multiple documents.

Worksheet register

The worksheet register is displayed below the document.

A PlanMaker document can consist of multiple worksheets which are stacked on top of each other like a stack of paper. They are called *worksheets*. You can use the *worksheet register* to switch between the individual worksheets of a document, create new sheets, and delete sheets.

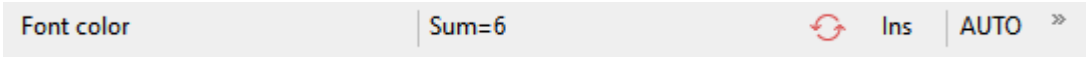
For more information, see Working with multiple worksheets.



The above figure shows the worksheet register for a workbook with three worksheets. The currently active sheet "Sheet1" is highlighted.

Status bar

The *Status bar* is found at the bottom of the program window.



Note: If you do not see the status bar, you will have to change the settings for it as described in Settings, Appearance tab (for the ribbon interface) or as described in Showing and hiding toolbars (for the classic menu interface).

You can read the following information in the status bar:

Example	Explanation

Font color, Chart1, etc.	Different information is displayed on the far left. If you hover the mouse cursor over a command icon, the function of this command is displayed here. If you select an object, its name is displayed here. If you move or resize an object, the current position/size is displayed, etc.
Sum=6	The *online calculation* is displayed next to it. If you select several cells, the calculation of the cell contents, for example, the sum, is automatically displayed here. Various calculation functions are possible and can be opened by just clicking on the display. (See also the option **Calculation in status bar** in the Settings, Edit tab.)
	This icon lights up if the worksheet contains a circular reference. This usually indicates an incorrectly formulated formula – for example, a formula in cell A1 that refers to cell A1 itself. If you click on this icon, the cell frame jumps to the cell with the circular reference. If you click on it again, it jumps to the next cell with such a reference (if more of them exist).
Ins	This option shows whether Insert mode (Ins) or Overwrite mode (Ovr) is enabled. This allows you to determine what should happen to the existing cell contents when editing a cell: **Ins:** Insert mode is enabled – newly entered text will be inserted in front of existing text. **Ovr:** Overwrite mode is enabled – newly entered text will be written over existing text. The default setting is Ins. You can use the **Ins** key on your keyboard to switch between these two modes (or by clicking on the Ins/Ovr display). The **Ins** key can also be disabled. For more information, see Settings, General tab.
AUTO	This indicates PlanMaker's current input mode: **AUTO:** Normal input mode is enabled – numbers are automatically recognized as numbers, and formulas are automatically recognized as formulas. This is the default setting. **TEXT:** Text input mode is enabled – all entries are converted into text (including numbers and formulas). To switch between these two modes, either click on the text "AUTO" in the status bar or press the key combination **Ctrl+Shift+F4**. ***Important:*** If you switch to TEXT input mode, all numbers and formulas you enter will be automatically preceded by a single quotation mark ('). This converts them into text, and they can *no* longer be used for calculations. (In order to turn converted numbers/formulas back to normal numbers/formulas, switch back to AUTO mode and remove the leading quotation mark.)

The status bar also contains a zoom slider that allows you to change the zoom level for the document:

To do so, either drag the slider with your mouse or click on the plus or minus icon (see Zoom level).

Tip: Configuring the status bar

If you click on the double arrow » on the far right of the status bar, you will find the following options:

- You can use the option **Add/remove buttons** to show or hide the individual preset elements of the status bar.

- The option **Customize** opens the dialog box *Customize user interface*, where you can freely configure the status bar from all commands available in PlanMaker.

  In the dialog box, select the item "Status bar" in the list *Customize* (top right). You can then add or remove buttons from the status bar and change their order. For more information, see Customizing icons and groups in the ribbon.

- **Reset this toolbar** restores all changes made to the status bar back to their original state.

Basics

This chapter provides information on PlanMaker's most important basics and explains basic spreadsheet terminology.

This chapter covers the following topics:

- **The structure of a worksheet**

  First, you will learn how worksheets are actually structured.

- **Moving within a worksheet**

  You will then learn how to navigate within a worksheet.

- **Entering data** and **Deleting data**

  In the next two sections, you will learn methods for entering and deleting data.

- **Undoing changes**

  Here, you will learn how to undo changes to the document.

- **Starting a new document**

  This section will show you how to create a new document.

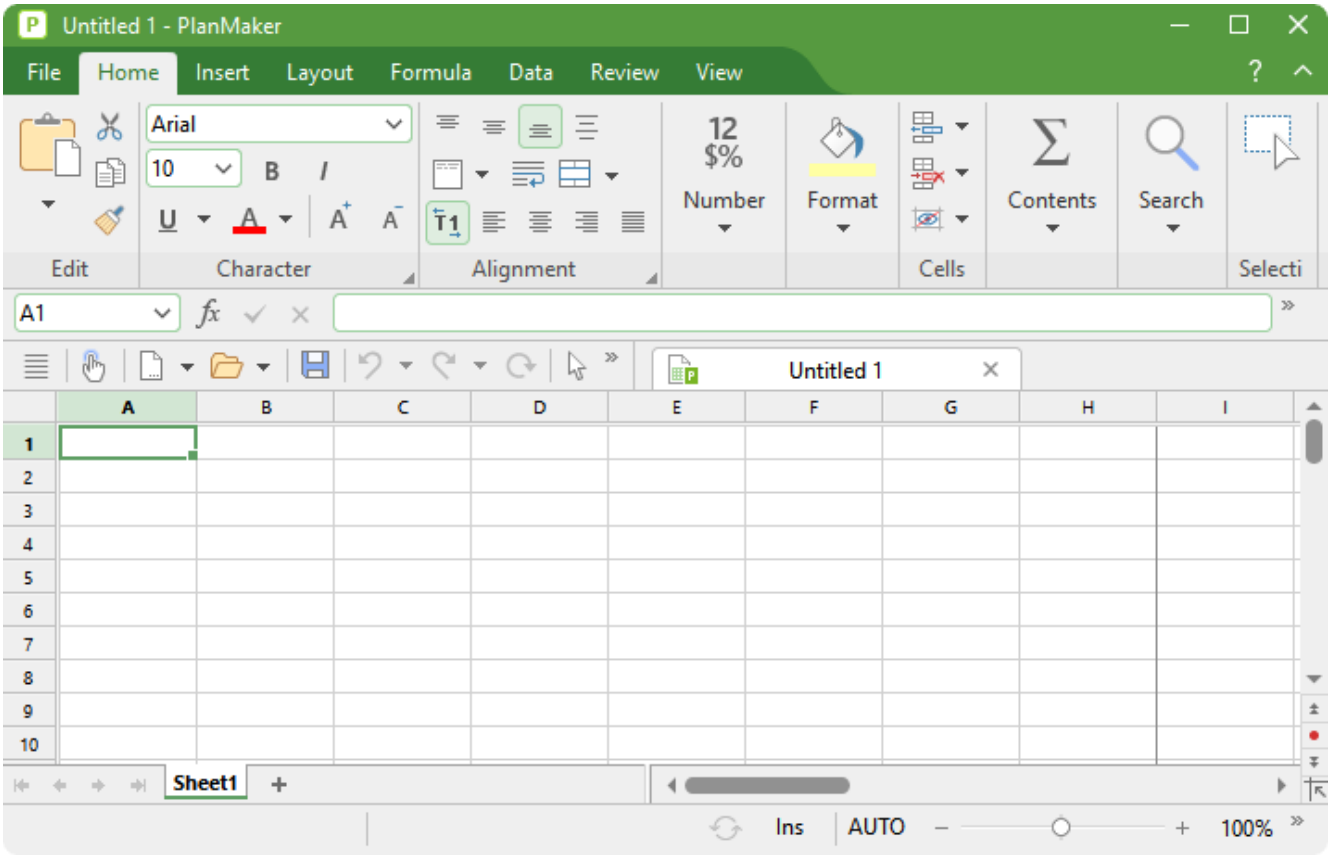
- **Opening a document**, **Printing a document** and **Saving a document**

  These three sections deal with opening, printing and saving documents.

For more information, see the following pages.

The structure of a worksheet

In order to use PlanMaker directly while working through this chapter, you should launch it now.



As you can see from the figure, worksheets are divided into rows and columns. Each worksheet can contain up to 1 million rows and 16384 columns. They are labeled as follows:

- The rows are numbered consecutively.
- Conversely, the columns are labeled with letters: A to Z, then AA to AZ, then BA to BZ, etc.

The matrix of rows and columns results in a grid composed of "boxes". These boxes are called the *cells* of the worksheet.

Each of the cells has its individual coordinates: "B5", for example, denotes the cell in the fifth row of column B (the second column). "A3" is the cell in the third row of column A (the first column) and so on. These coordinates are referred to as the cell *address*.

The cell address is very important because calculations often refer to other cells: If, for example, you want to have twice the value of cell A1 in cell A2, you would enter =A1*2 in cell A2.

Twice the value of A1 is now displayed in A2. If you change the value in A1, the resulting calculation in cell A2 is automatically updated.

Moving within a worksheet

If you enter data in PlanMaker, it is always inserted into the *active* cell. A thin dark frame known as the *cell frame* indicates the active cell:

You can move the cell frame to another cell as follows:

Mouse: Click on the desired cell.

Keyboard: You can use the arrow keys of the keyboard to move the cell frame.

Moving via the keyboard is also possible: Press the **Tab** key to move to the next cell on the right and **Shift+Tab** to the next cell on the left. Press the **Enter key**↵ to move down and **Shift+Enter** to move up.

For a list of all keys available for this purpose, see Shortcut keys.

Entering data

In order to enter data in a cell, move to the desired cell using the arrow keys or the mouse, and start typing.

To complete your input, press the **Enter key**↵.

If you made an error during input, press the **Esc** key instead of the **Enter key**↵. PlanMaker will discard what you just typed and restore the original cell content.

Alternatively, you can enter data via the following buttons of the Edit toolbar:

- To accept the input, click on the ✓ button.
- To cancel the input, click on the ✗ button.

To subsequently edit the contents of a cell, navigate to the cell and press the **F2** key. Alternatively, you can simply double-click on the cell.

For more information on this topic, see Entering data in cells.

Deleting data

There are several ways to delete data from a document:

- **Deleting the content of a cell**

  To delete the entire content of a cell, navigate to that cell and then press the **Del** key on the keyboard (or choose the ribbon command **Home** | group **Edit** | **Cut**).

- **Deleting parts of the cell content**

  To delete only part of the cell content, double-click on the cell. Alternatively, you can select the cell and press the **F2** key. You can now edit the cell content, for example, delete parts of it using the **Del** key or the **Backspace key**⌫.

- **Deleting the contents of multiple cells**

  To delete the contents of multiple cells at once, *select* them before pressing the **Del** key. For information on selecting cells, see Selecting cells and cell content.

For more information on "Deleting cells", see Deleting cells and cell content.

Undoing changes

In the Quick access toolbar, you can use the **Undo** button on the Quick access toolbar. This restores the last action you undid. You can thus undo the cancellation of changes.

For more information on this topic, see Undoing changes.

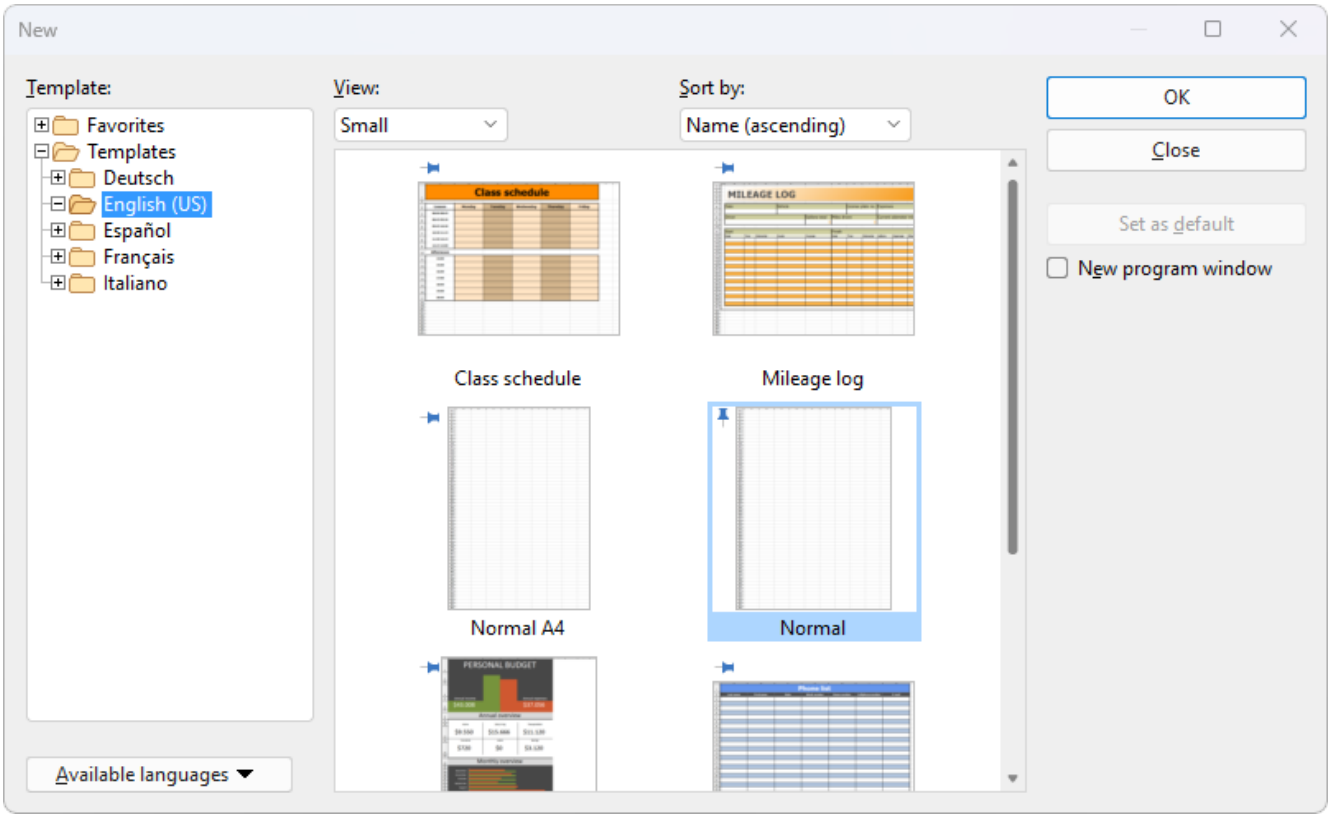
Starting a new document

To start a new document, choose the command **File | New** . (Click directly on the icon itself.) Incidentally, this command is also in the Quick access toolbar.

(Note: If you click on the icon's arrow instead, you'll see the templates you previously used. For more information, see "List of document templates used" below.)

Tip: Use the key combination **Ctrl+N** to quickly open a new document with the current default template, while skipping the dialog box shown below.

After you click on the **File | New** icon , the following dialog box appears:



The dialog box of the command **File | New**

If you just want to create a normal worksheet without any additional formatting, choose the default template `Normal.pmvx`.

Tip: If you hover over a template (without clicking on it), more information about the template will be displayed. This requires Show tooltips to have been enabled in the settings.

As soon as you confirm with **OK**, the new document will be created.

Using document templates

The above dialog box contains the following options:

Template: In addition to the default template `Normal.pmvx`, you will also see some predefined document templates that you only have to fill in after opening them.

The *Favorites* folder always contains the default template as well as all "pinned" templates. They are permanently displayed at the top in the list of the command **File | New** for quick access. (For more information, see the paragraph below "List of document templates used".)

Pinned templates are recognizable by the vertical pin icon. By clicking on the pin icon, you can pin or unpin a template.

View: The templates can be displayed in the dialog box as small, medium or large preview images in the tile view. Alternatively, a list view in text form is also possible.

Sort by: You can view the templates sorted by name or by date.

Available languages: Here, you can select the language(s) in which document templates are to be displayed.

Set as default: Change the default template by clicking on a template and then pressing this button. In future, a new document will always be opened with this template.

For more information on creating your own document templates, see Document templates.

Option "New program window"

The option **New program window** means the following: If it is enabled, the new document will be displayed in its own program window. If the option is disabled, the new document will be created with a new tab in the existing program window.

This setting is also saved and is applied each time a document is opened, both when new documents are created and when existing documents are opened.

(Note: In the Android/iOS version for *smartphones*, a separate program window is always opened, while in the version for *tablets*, a new tab is always opened in the same program window.)

List of document templates used

If you click on the small arrow of the **File | New** button, the program opens a list of previously used document templates, which you can apply directly:

Default template: The current default template is displayed here. If you just want to select the default template, you can also click on it here in the list and skip the dialog box.

Unpinned templates: This is displayed if you have already used templates other than the default template. You can thus use these templates directly from the list for a new document without performing a long search.

If you click on the pin icon of a template, the status changes to **Pinned templates**: These are permanently displayed at the top of the list (and in the *Favorites* folder of the dialog box, see above). You can unpin the template by clicking on the pin icon again.

If you click on the cross icon, you remove an entry completely from the list. The command **Delete all unpinned items** deletes all entries from the list – except the pinned templates and the default template.

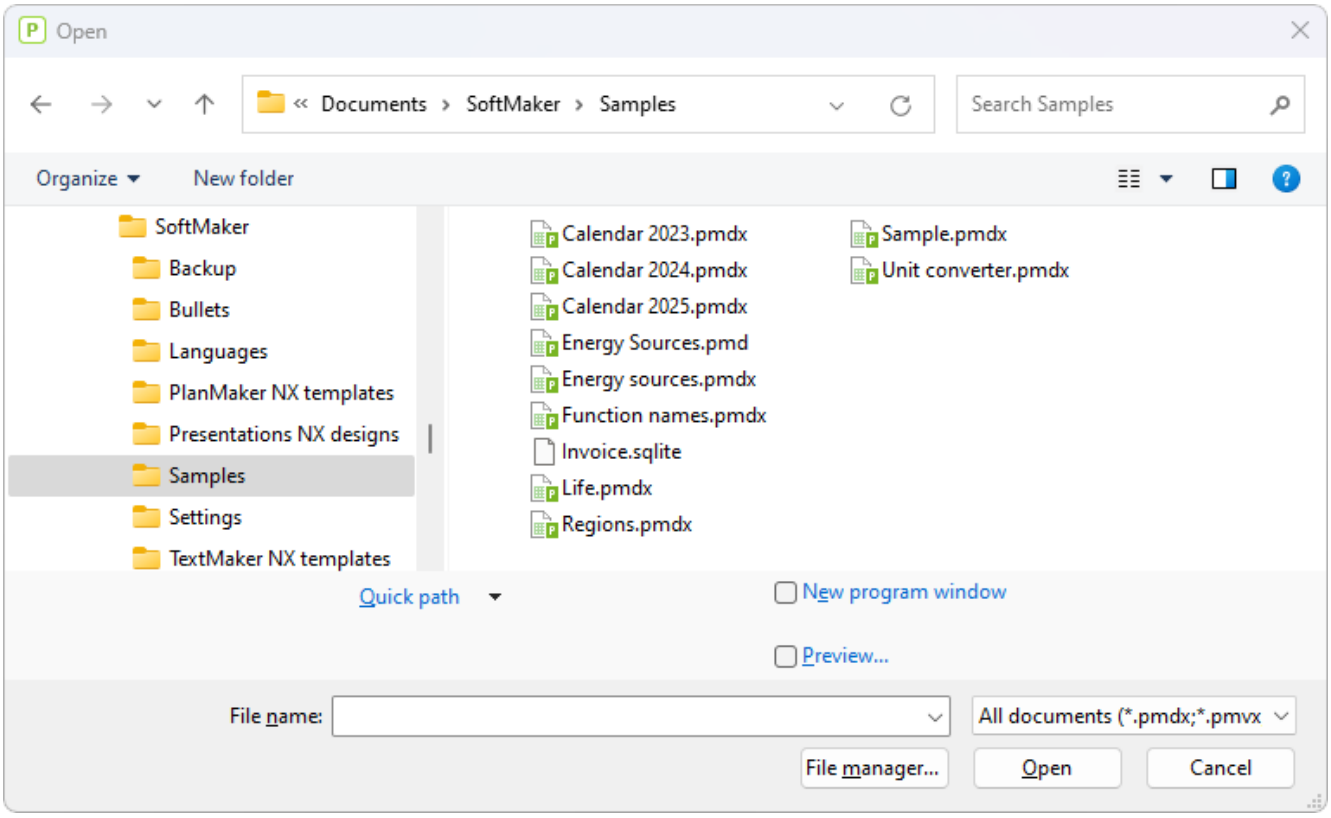
The command **More** will take you again to the dialog box described above.

Opening a document

To open an existing document, choose the command **File | Open** by clicking directly on the icon itself. Incidentally, this command is also in the Quick access toolbar.

(Note: If you click on the icon's arrow instead, you'll see a list of recently used files. See explanations in the paragraph "List of recently opened files" below.)

After you have clicked directly on the **Open** icon, the program will display a dialog box which may look as follows:



The dialog box of the command **File | Open**

This dialog box may look slightly different depending on the operating system, but its function always remains the same. Use this dialog box to tell PlanMaker which document to open.

To do so, you could simply type in the name of the file you want to open manually. However, the dropdown list with the files is more convenient because all files are listed here in the current folder and can be easily selected.

Option "New program window"

The option **New program window** means the following: If it is enabled, the new document will be displayed in its own program window. If it is disabled, the document will be opened with a new tab in the existing program window.

This setting is also saved and is applied each time a document is opened, both when new documents are created and when existing documents are opened.

(Note: In the Android/iOS version for *smartphones*, a separate program window is always opened, while in the version for *tablets*, a new tab is always opened in the same program window.)

Previewing a document

When the option **Preview** is enabled, a little box displaying a preview of the currently selected document is displayed to the right of the dialog box.

Opening other file formats

In addition to opening documents created in PlanMaker's default file format, you can also open files created with other software. All you have to do is select the desired format from the list of file types.

For more information, see Working with other file formats.

Using Quick paths

You can use the **Quick path** button to create and choose *quick paths* in order to quickly switch to a specified folder when opening or saving files.

For more information, see Quick paths.

File manager

The **File manager** button opens the integrated file manager. This displays a list of your documents and allows you to open, print, view or delete them, as well as perform searches.

For more information, see File manager.

List of recently opened files

If you click on the small arrow below the **File | Open** button, you will see a list of recently opened files. Simply click on one of these files to open it again.

In this file list, you can also select the following options:

If you click on the pin icon of a file, it will appear in the list as a **pinned file** and will be displayed permanently at the top of the list. **Unpinned files** will eventually be removed from the list of recently opened files if the number of entries exceeds the limit that was set. For more information, see "Recently used files in the File menu" in Settings, Files tab.

Click on the pin icon again to unpin the file or click on the cross icon to remove an entry from the list. The command **Delete all unpinned items** deletes all entries from the list – except the pinned files.

The command **Browse documents** will take you again to the dialog box described above.

Printing a document

To print the current document, choose the ribbon command **File | Print**.

The program will display a dialog box in which you can specify the parts of the document to be printed.

For more information on outputting documents, see Outputting documents.

Saving a document

When you have completed a document, you should save it to retain it permanently. To do so, click on the ribbon command **File | Save**.

Tip: This command can also be found by default in the Quick access toolbar below the ribbon.

If the document does not yet have a name, PlanMaker automatically prompts you to assign a file name to it before saving it.

Saving under a different name or in a different location

PlanMaker also has a ribbon command called **File | Save as**. This also saves your document, but you can first assign it another name or select another folder in which to save it.

Saving in a different file format

You can use the ribbon command **File | Save as** to also save a document in the file format of another program. To do so, select the desired format in the list **Save as type**. For more information, see Working with other file formats.

Saving all open documents

If you have multiple documents open at the same time, you can use the ribbon command **File | Save all** to save all of them at once. PlanMaker checks whether the documents have been changed since they were last saved and saves only those that have been changed.

Exiting the application

To exit PlanMaker, choose the command **File | Close** and then **File | Exit**. Alternatively, just click on the **X** icon on the far right of the title bar or tab.

If one of the open documents has been changed since it was last saved, PlanMaker automatically asks you whether you want to save it before closing it.

The PlanMaker tour

Welcome to the PlanMaker Tour!

On the following pages, you will obtain more in-depth information on PlanMaker than in the Basics chapter. Users who are unfamiliar with spreadsheets should read the next section thoroughly.

More advanced users are encouraged to browse through this section as needed. It contains some useful tips that will make it easier for you to use PlanMaker.

Note: Most of the illustrations in this manual were produced with the **Windows** version of PlanMaker. On other operating systems, some of the controls look slightly different, but their functionality is the same.

Some theory to begin with

The topic "Calculating and computers" is best explained by means of a brief review. It was precisely through the field of "computing" that computer technology originally found its way into homes and offices. The first tentative advance was made by pocket calculators. They are definitely a great help, but their disadvantages are a good indicator of the usefulness of a spreadsheet:

If, for example, you're planning on buying a computer, you'll need to compare various systems based on their hardware components, as you'll need a PC, a monitor and a printer. One vendor may offer a competitively priced computer, another vendor might offer an inexpensive monitor, etc. Which is the best option?

To determine this fact, you have to add up the various prices. To do so, you pull out your pocket calculator, add number by number, write down the result – and become frustrated: With every new offer, the game starts all over again with new values, even though the method of calculation never changes.

Such calculations can be performed more conveniently by means of a spreadsheet.

The first technique you will encounter when working with PlanMaker is the use of *"variables"*. Variables help you to generate abstract formulas without the use of actual number (or dollar) values. For example:

Computer + monitor + printer = total price

Thus, your formula would look as follows:

a + b + c = d

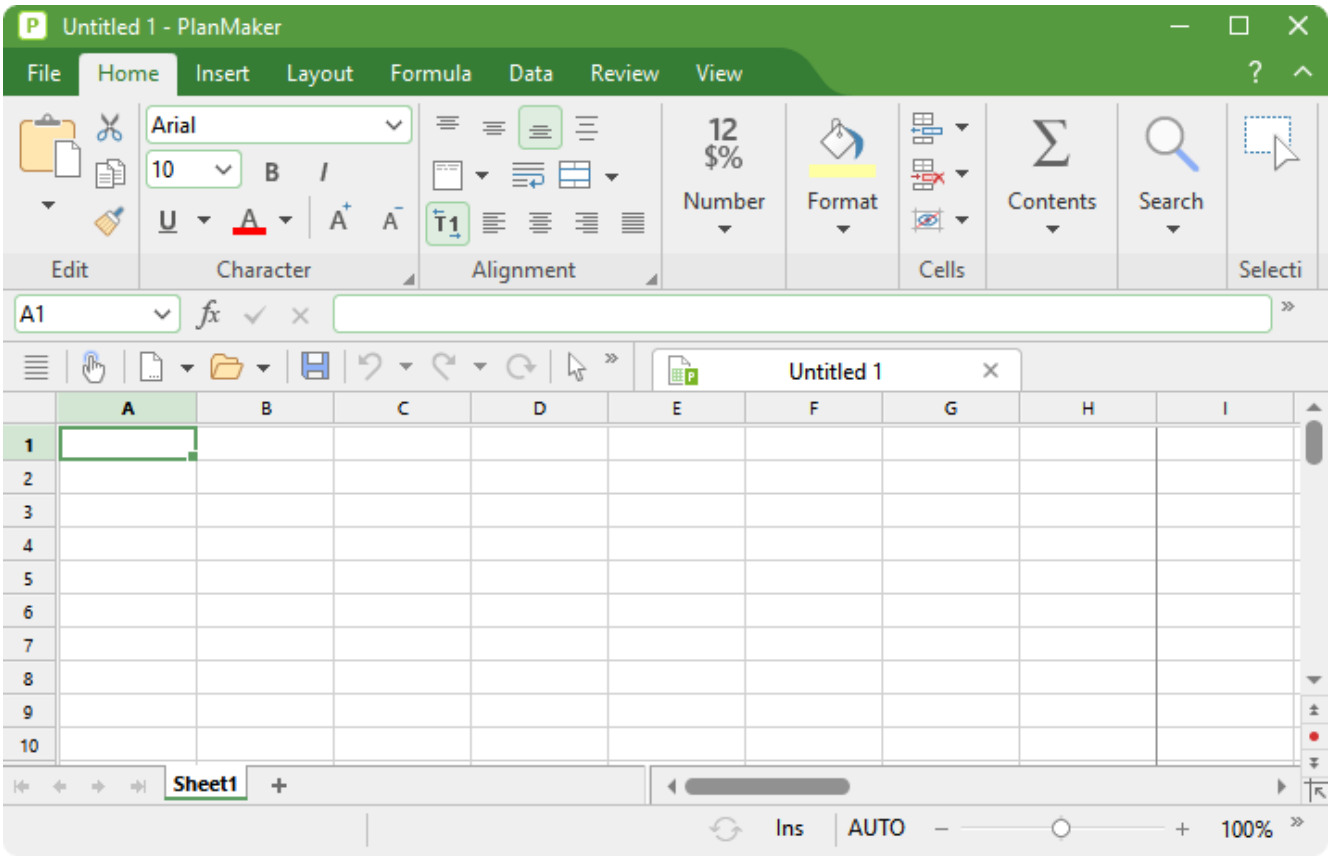
Such a formula is only of practical use in calculating *without* a spreadsheet if, for example, you have a programmable calculator – you can enter the formula in it once, and then execute it many times. The calculator asks for the values one by one, and then automatically calculates the total value. However, this is not much of an improvement over repeated individual calculations...

First steps with PlanMaker

With these preliminary considerations in mind, let's look at how such tasks can be performed with PlanMaker.

To do so, start PlanMaker now.

On Windows, PlanMaker looks as follows:



The worksheet takes up most of the program window. The top of the worksheet is bordered by a bar with letters and by a bar with numbers on the left.

Let's take a closer look at the worksheet:

The structure of a spreadsheet

As you already know from the Basics chapter, a worksheet is divided into *rows* and *columns*. They are also indicated by the gray lines that are displayed in the worksheet.

The cell in the top left corner is surrounded by a dark border. This is called a *cell frame*. It indicates which cell is currently active.

When you type something, it is always entered in the active cell.

You can use the arrow keys on your keyboard, e.g., ↓ and →, to move the cell frame from one cell to another.

When doing so, take a look at the display panel on the left above the worksheet: PlanMaker displays the *address* of the currently active cell. The address always consists of one or two letters and a number:

"A1" means "column 1, row 1".

"A5" means "column 1, row 5".

"D5" means "column 4, row 5", etc.

The fact that each cell has a unique address is the key to all functions of a spreadsheet: This allows you to enter calculations in any cell that relate to other cells – for example, to form the sum of cells. You don't even need to deal with variables like a, b, c, as you just use the coordinates of the cells.

This is like handing someone a worksheet and asking them, for example, "Please add the numbers in the second, third and fourth rows of the second column".

Entering values and formulas

That's enough theory for now! Let's now perform our first calculation.

First, we enter the price of the computer; under that, the price of the monitor; and under that, the price of the printer.

Thus, navigate to cell B2 and type the value 799 there. The value will appear both in the selected cell and in the Edit toolbar above the worksheet.

Note: When entering numbers, do not make the mistake of typing the letter "O" or "o" instead of the number "0". PlanMaker makes a clear distinction between letters and numbers. The letter "O" is not a number for the program. PlanMaker would accept the input but interpret it as text, and thus could not perform calculations with it.

Then press the **Enter key**↵ to complete your input. The cell frame automatically moves down one row to cell B3. Type the value 425 here and the value 199 in the row below.

Tip: If you entered an incorrect value and have already pressed the **Enter key**↵, you can still correct your mistake. Use the arrow keys to move the cell frame to the cell with the error, and enter the correct value. Then press the **Enter key** again, and the old cell content will be overwritten by the new input. You can also edit the content of already filled cells by pressing the **F2** key.

Entering formulas

Let's enter our first formula.

In order to calculate the total cost of our computer equipment, we have to add up the unit prices that we just typed. This is very easy:

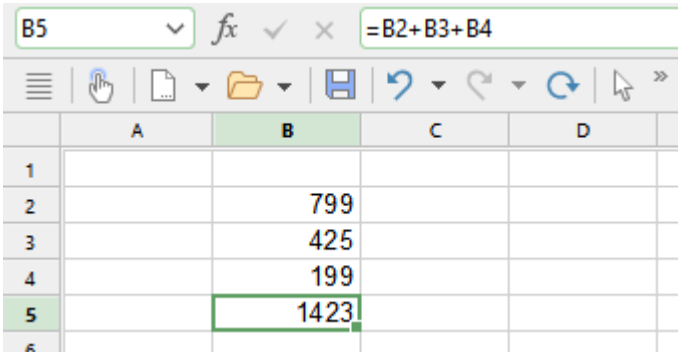
Go to cell B5 and type an equal sign = at first. The equal sign shows PlanMaker that you would like to start entering a *formula* in this cell.

Then enter the formula. For this purpose, you use the cell addresses as "variables". Enter the following:

```
=B2+B3+B4
```

Note: Cell addresses are not case-sensitive, i.e., you can enter them in either uppercase or lowercase.

When you press the **Enter key**↵, you will immediately see the result of your first formula in the cell:



Let's see what happens if you change the numbers in the cells. For example, navigate to the cell that contains 425 and enter 259 or some other value instead. As soon as you press the **Enter key**↵ again after replacing the cell content, the result of the calculation will be updated immediately.

Regardless of what cells B2, B3 and B4 contain, PlanMaker will always add them up. If you get a quote for a computer system in which only the price of the printer has changed, for example, you only need to change that particular value, and the new total price will be displayed in cell B5.

The "SUM" function

The above example is one method of adding up several numbers. Although the method is adequate for a few numbers, it is clearly too cumbersome for adding 50 numbers - that would be a very long formula! Fortunately, there are alternative methods: the *computational functions* of PlanMaker.

In order to find out more about them, make cell B5 the current cell; it contains the formula you entered previously.

First, delete the old formula by pressing the **Del** key on the keyboard or by simply overwriting the existing cell content. Then enter the following formula:

```
=SUM(B2:B4)
```

After pressing the **Enter key**↵, you will see the result in the cell: the sum of the cells B2 to B4.

PlanMaker has a whole range of computational functions – and one of them is SUM. As its name suggests, the SUM function calculates the sum of the relevant values. The expression in parentheses after the function name tells PlanMaker where to start and stop summing.

You have directed PlanMaker to start adding in cell B2 and stop in cell B4. In this case, only the number in B3 is in between, but the SUM function would also work with a larger range, such as B2:B123.

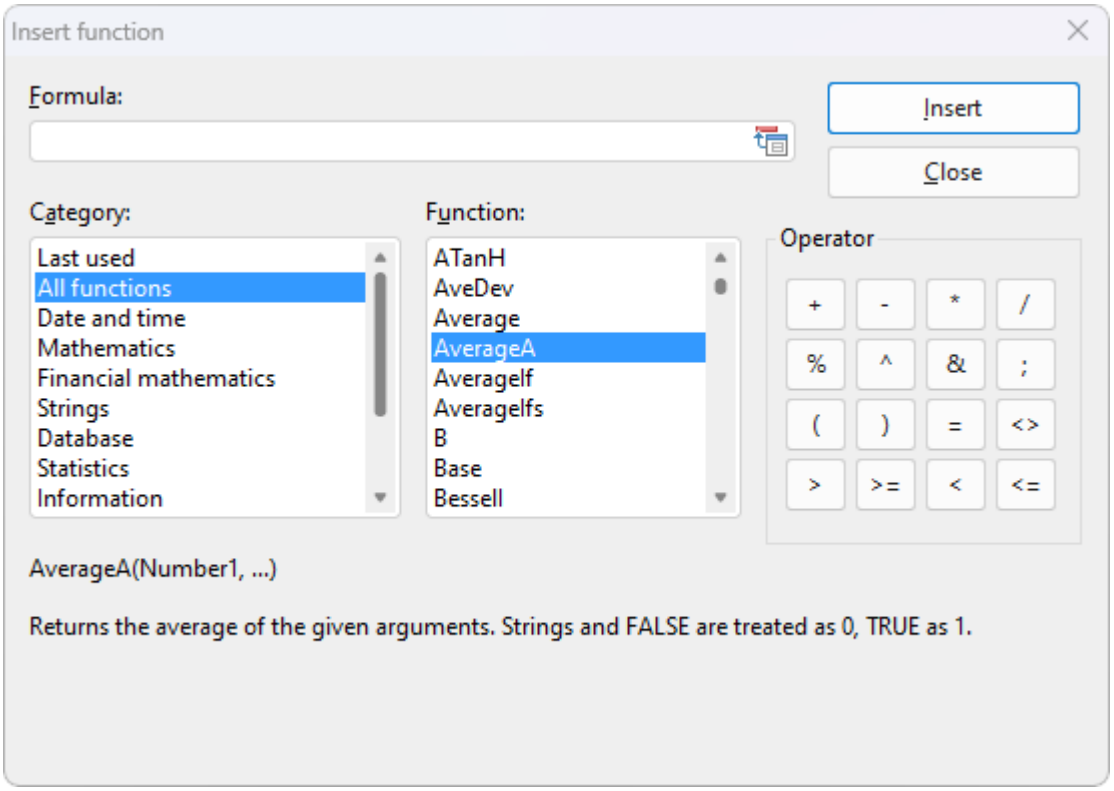
The notation *Start cell:End cell* also works across rows and columns. If, for example, you enter B2 as the start cell and C4 as the end cell, these two coordinates in the worksheet form the corners of a rectangle. The formula =SUM(B2:C4) would thus sum *all* numbers included within this rectangle.

Variety of formulas

To get an idea of the multitude of computational functions available in PlanMaker, you can now choose the command **Function**.

You will find the command on the ribbon tab **Formula | Function** or as **Insert function** with an identical icon in the Edit toolbar. Even faster: Simply press **F7**.

The program will now display a dialog box with a list of all functions that PlanMaker supports.



Tip: You can open a help page for each arithmetic function. Click on the desired function in the above dialog box, and then press the **F1**key.

Let's try another function. We'll calculate the *average value* of our three numbers in the worksheet:

To do so, exit the dialog box and delete the contents of B5 again.

Then choose the command **Function (Insert function)** . Select the "All functions" category in the list on the left side of the subsequent dialog box. Then scroll down through the list on the right to the "Average" function. Double-click on the "Average" function.

In the input field **Formula** of the dialog box, PlanMaker has now automatically inserted the `Average()` row. To complete the formula, enter B2:B4 again between the parentheses.

Alternatively, you can simply select the desired cell range directly in the worksheet by left-clicking on cell B2 and then dragging the mouse down to cell B4. If the dialog box is covering the cells that you want to select, simply drag it out of the way while holding down the mouse button.

The completed formula should look as follows:

```
=Average(B2:B4)
```

If you click on **Insert**, this formula will be transferred to cell B5 and calculated immediately.

You have now learned about two of the more than 400 computational functions of PlanMaker. For a comprehensive list of all functions, see Functions from A to Z.

Going three-dimensional

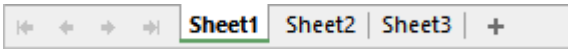
PlanMaker spreadsheets do not necessarily have to consist of a single worksheet. You can also stack several worksheets on top of each other – just like a stack of papers. Thus, PlanMaker documents are also called *workbooks*. Each workbook can contain up to 2048 *worksheets*.

The main advantage of this is that calculations in one worksheet can also refer to cells in other worksheets of a workbook, thus enabling you to perform "three-dimensional" calculations as such.

For example: You record the balance sheets of several branches of your company on three worksheets. You could now create a fourth worksheet and use it to calculate, for example, the sum of the results of the other three sheets.

To add a new worksheet, choose the ribbon command **Insert** | group **Tables** | **Sheet** . The program opens a dropdown menu in which you can choose the command **Insert**.

Tip: You can create a new worksheet faster with the + icon in the worksheet tab.



The worksheet tab at the bottom of the document

To switch between the existing sheets, simply click on the desired worksheet in the worksheet tab with the mouse.

For more information on this topic, see Working with multiple worksheets.

Changing the appearance of worksheets

Let's go back to our first sample worksheet. Although the worksheet is very useful for performing calculations, it is not very appealing in terms of its design. At the same time, PlanMaker has extremely powerful options for preparing worksheets in visual terms.

Let's try some of them out:

Adding a heading

What our worksheet definitely lacks is a heading. Thus, let's simply enter a relevant text in a cell above the numbers and increase its font size.

To do so, click on cell B1 to make it the active cell. Then enter the following text:

```
My first worksheet↵
```

Changing character formatting

Let's now choose a different font for the heading and make it a lot bigger.

For this purpose, move the cell frame back to cell B1 with the heading. Then expand the font list on the ribbon tab **Home** | group **Character** by clicking on the small arrow to the right of the font name.

PlanMaker now displays a list of all fonts installed on your computer. Select the Tahoma font, for example.

Then open the font size list

to the right of (or below) the font list, and select the 24 point size.

You can also set the font color

if you want, and this command group also contains three buttons labeled **B**, *I* and U for enabling/disabling bold, italics and underlining.

Changing number formatting

PlanMaker provides extensive options for formatting numbers. It not only enables you to change the character format of cell content (font, font size, etc.), but you can also modify their *number format*.

Let's try this out: To display the values in cells B2 to B5 with a currency symbol, proceed as follows:

First, *select* cells B2 to B5. To do so, click on cell B2 and then – with the mouse button still held down – move the mouse cursor to cell B5.

Android/iOS: If you are using these versions, please note that selecting cells works slightly differently there. For more information, see Selecting cells and cell content.

When the desired cells are selected, choose the ribbon tab **Home** and then go to the command group **Number** and click on the group arrow ◢ in the bottom right corner. The program opens a dialog box with numerous settings options. We are only interested in the **Number format** tab: On this tab, select the entry "Currency" in the list and confirm with **OK**.

Result: PlanMaker now adds a currency symbol to the numbers in the selected cells and displays them with two decimal places, as is usual with monetary amounts.

There are many other number format options that you can use to change the appearance of numbers. For example, the number of displayed decimal places can be set, numbers can be displayed as percentages, etc. Important: Applying a different number format to a value only changes how it is displayed and does not change the value itself.

You have now become familiar with a small part of PlanMaker's options for improving the visual display of worksheets. As you can imagine, there are many other options for visually enhancing worksheets and making them clearer. For more information, see Formatting worksheets.

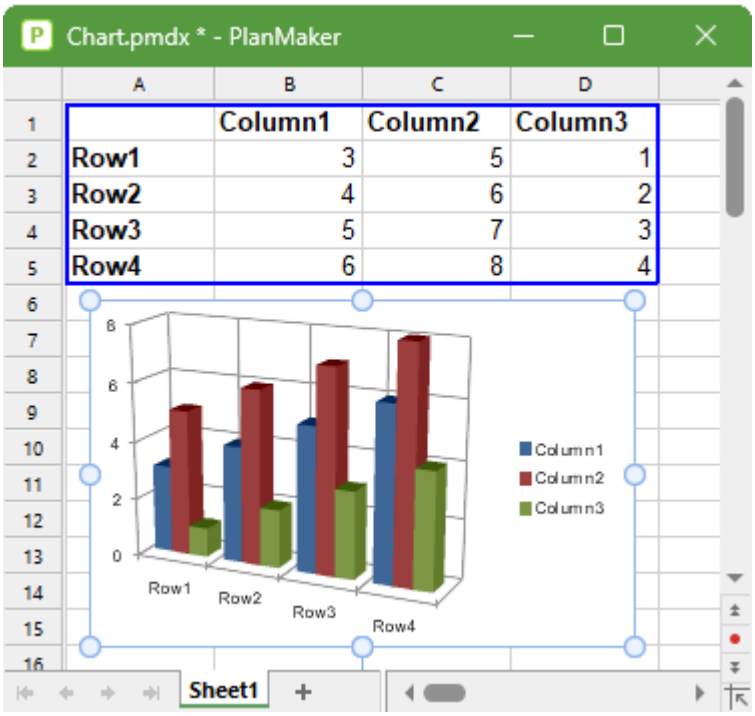
Charts

Before we end this PlanMaker tour, let's take a quick look at *charts*.

It is often difficult to interpret the meaning of spreadsheets consisting of long columns of numbers. PlanMaker thus has its own chart module for the optical preparation of data from your worksheets.

To insert a chart, first select the values to be displayed.

Then choose the ribbon command **Insert | Chart frame** (click directly on the icon itself). A dialog box appears in which you only have to select the desired chart type and the completed chart appears in the document.



For more information on this topic, see Charts.

Outlook

This is where our brief introduction to PlanMaker ends. You now know many of the program's most important basics, and everything else you will learn in the *reference section* of the manual, starting with the next chapter.

This part of the manual provides detailed descriptions of all of PlanMaker's features by subject matter. Of course, you don't have to read it chapter by chapter to learn all the capabilities of PlanMaker at once. Just read the chapters that you really need, and gradually explore PlanMaker's features.

Have fun with PlanMaker!

Editing worksheets

This chapter marks the beginning of the *reference section* of the manual. This part of the manual provides detailed descriptions of all of PlanMaker's features organized into chapters by subject matter.

In this first chapter, you will learn how to enter data and edit worksheets. This chapter covers the following topics:

- **Entering data in cells**

  The first section describes what you have to consider when entering text, numbers, calculations, etc., in worksheet cells.

- **Undoing changes**

  The **Undo** button in the Quick access toolbar can be used to undo the last changes.

  The **Redo** command in the Quick access toolbar is its counterpart and reverses the **Undo** command.

- **Repeating commands**

  The command **Repeat** in the Quick access toolbar repeats the last command that you used with exactly the same settings.

- **Selecting cells and cell content**

  This section describes how to select cells – for example, by using the mouse or the keyboard.

- **Deleting cells and cell content**

  You can either use the **Del** key on the keyboard to delete only the cell content or choose the ribbon command **Home** | group **Cells** | **Delete** to remove cells completely from the worksheet (the rest of the worksheet moves up accordingly).

- **Inserting new cells**

  The ribbon command **Home** | group **Cells** | **Insert** inserts new cells into your worksheet and automatically moves the existing cells down (or right).

- **Moving and copying**

  Selected worksheet areas can be copied or moved with the commands of the ribbon tab **Home** | group **Edit** or simply by dragging them via the mouse. Formulas are automatically adjusted when you move or copy cells.

- **Paste (special)**

  The ribbon command **Home** | group **Edit** | **Paste** | **Paste special** is a more powerful variant of the command **Paste**. You can specify exactly what is to be pasted, for example, just the values or just the formatting.

- **Filling cells automatically**

  You can choose the ribbon command **Home** | group **Contents** | **Fill** to easily fill cells with consistent values or ranges of values.

- **Working with multiple worksheets**

  You can create multiple worksheets in a single document and stack them like a pile of paper, thus enabling you to perform "three-dimensional" calculations.

- **Naming cell ranges**

  If you assign a name to a range of cells using the ribbon tab **Formula** | group **Named areas** | **Edit names**, you can use this name in formulas, for example, =SUM(sales) instead of =SUM(A1:A5).

- **Transposing cells**

  The ribbon command **Data** | group **Edit** | **Transpose** reverses the row and column orientation of a selected cell range.

- **Splitting text into multiple columns**

  Choose the ribbon command **Data** | group **Edit** | **Text to columns** to split text that is in one worksheet cell into multiple worksheet columns.

- **Sorting cells**

  Choose the ribbon command **Data** | group **Filter** | **Sort** to sort ranges of cells according to one or more criteria.

- **Filtering cells by their content**

  Choose the ribbon command **Data** | group **Filter** | **AutoFilter** or **Special filter** to filter a cell range according to one or more criteria. All rows in this range that do not match the filter conditions will then be hidden.

- **Analyzing sheets**

  This section introduces some tools that can be very useful for analyzing worksheets and troubleshooting. These tools include a command for displaying formulas, syntax highlighting, a watch window for cell content, and various functions for analyzing the formulas in a worksheet.

- **Updating sheets**

  The calculations in *all* worksheets are kept up to date automatically. Whenever you change the content of a cell, PlanMaker recalculates the worksheets. If required, you can disable this behavior and use the ribbon command **Formula** | group **Update** | **Recalculate** to update worksheets manually.

- **Using comments**

  Choose the ribbon command **Insert** | **Comment** to add comments to cells. The comment is displayed when you hover over such a cell with the mouse cursor.

- **Goal seek**

  Choose the ribbon command **Data** | group **Analyze** | **Goal seek** so that an approximation method is used to determine the value which a particular component of a calculation must assume in order to produce the desired result.

- **Scenarios**

  The ribbon command **Data** | group **Analyze** | **Scenario manager** allows you to create and display different *scenarios*. You can use them to observe how the calculations in a worksheet change when the values in specific cells are altered. This enables you to perform all kinds of "what-if" analyses.

- **Consolidating data**

  The ribbon command **Data** | group **Analyze** | **Data consolidation** can be used to evaluate data from one or more cell ranges, for example, to calculate their total sums.

- **Tables in worksheets**

  This function is a particularly useful option, as it formats an entire worksheet area in one of numerous predefined formats at the touch of a button. Select a cell range and choose the ribbon command **Insert** | **Table** to create a "table in a worksheet".

  This has the following effects:

  The cell range is automatically formatted with a *table style*. You can switch to a different table style at any time, and thus change the appearance of the entire cell range at once. In addition, an *AutoFilter* is applied to the "table" automatically. Furthermore, other useful features such as a *total row* are available.

- **Pivot tables**

  Large amounts of complex data can be summarized very clearly according to certain criteria using pivot tables. Select an existing data area and use the ribbon command **Insert** | **Pivot table** to create meaningful presentations of your data with minimal effort. The section includes exercises for illustration purposes.

- **Freezing rows and columns**

  Choose the ribbon command **View** | group **Windows** | **Freeze cells** to freeze the first rows or columns of a worksheet on the screen. This will make them stay in place when you scroll through the worksheet. This is ideal for row or column headings.

- **Inserting special characters**

  The ribbon command **Insert** | group **Text** | **Symbol** opens a dialog box with all available characters of a font. It can be used to insert special characters that are not readily accessible via the keyboard.

For more information on these topics in the above order, see the following pages.

Entering data in cells

In order to enter data in a cell, navigate to the desired cell with the arrow keys or mouse, and simply start typing.

To complete your entry, press the **Enter key**↵.

If you made an error during input, press the **Esc** key instead of the **Enter key**↵. PlanMaker will discard what you just typed and restore the original cell content.

Alternatively, you can enter data via the following buttons of the Edit toolbar:

- To accept the input, click on the button.
- To cancel the input, click on the button.

To subsequently edit the contents of a cell, navigate to the cell and press the **F2** key. Alternatively, simply double-click on the cell.

Notes on entering numbers

- Always enter numbers with the decimal separator specified in your computer's regional settings, for example, either the decimal comma usually used in Europe (0,7) or the decimal point used in the USA, UK and Ireland (0.7).
- Numbers can also be entered in *scientific (exponential) notation*. For example, 4E03 corresponds to 4 * 10^3, which equals 4000. Alternatively: 4E-03 corresponds to 4 * 10^-3, which equals 0.004.
- Numbers can also be entered as *fractions*. For example, 2 1/2 corresponds to 2.5.
- Numbers can also be entered as *percentages*. For example, 70% corresponds to 0.7.

On the ribbon tab **Home**, you can change the format of numbers in worksheet cells at any time using the command group **Number**. For example, you can set PlanMaker to display numbers as percentages or as a currency with a currency symbol. For more information, see Number format.

Notes on entering dates and times:

- Dates can be entered in various formats, for example, 09/25/18 or 9/25/2018, etc.
- You can enter dates between January 1st, 1900 and December 31st, 2500.
- If you enter a date with the year represented by only its *last two digits*, it is interpreted as follows:

  00 to 29 -> 2000 to 2029

  30 to 99 -> 1930 to 1999

  PlanMaker treats 1/1/29 as 1/1/**20**29 and 1/1/30 as 1/1/**19**30.
- Times can be entered in numerous formats. For example, you can enter 09:30 as follows:

  9:30

  09:30

  09:30:00

  09:30:00 AM

  etc.
- You can combine a date and a time in a single cell by entering the date, a space, and the time. For example, when you enter 9/25/18 6:00, PlanMaker knows that you mean September 25, 2018 at 6:00 am.

Note: Dates and times are always stored as *numbers* in spreadsheets so that calculations can be performed with them. The number 0 represents 1/1/1900, the number 1 stands for 1/2/1900, etc. The time is stored in the decimal places of the number.

However, you don't have to worry about this fact. Simply enter dates and times in your preferred manner. PlanMaker automatically converts your input into a number and displays it formatted in a date/time format. To learn more about changing the format of dates and times, see Number format.

Tip: Use the key combination **Ctrl+.** (period) to enter the current date in a flash and **Ctrl+Shift+.** to enter the current time.

Notes on entering logical values (Boolean values):

- To enter a logical value in a cell, type either TRUE or FALSE there.

Notes on entering calculations:

- Formulas must always begin with the equal sign, e.g., =2+5.
- The command **Insert function** in the Edit toolbar can be used to easily insert *arithmetic functions*. Of course, you can also input formulas manually, for example, =SUM(2, 5).
- For more information on performing calculations, see Formulas and functions.
- For descriptions of all computational functions, see Functions from A to Z.

Notes on entering text:

- Text can be easily typed into a cell.
- In some rare cases, you may want to "force" PlanMaker to consider a *number* as text. This can be achieved by putting a single quotation mark in front of the number. If, for example, you enter '42 (including the single quotation mark), PlanMaker will treat this as text and not as a number.
- ***Tip:*** If you have entered some text entries directly below each other in a column, you can open a dropdown list of these entries by pressing the key combination **Alt+Shift+↓** in a cell directly above or below the entries.

  The program then displays a list of all text entries directly above or below this cell. If you select one of these entries and press the **Enter key**↵, it will be copied to the cell. This can save you a lot of time when you have to enter a large number of identical text entries.

  Note: This feature works only with text entries – not with numbers, dates or formulas.
- ***Another tip:*** Furthermore, when you type text into a cell, PlanMaker automatically makes suggestions from the existing text entries found in the cells above and below this cell.

  Example: You have entered the values "New York", "Rio", and "Tokyo" in a column. If you now go to the cell below these entries and type the letter "N", the program will automatically suggest "New York". If you type the letter "T", "Tokyo" will be suggested, etc.

  To accept the suggestion, press the **Enter key**↵. To reject it, simply continue typing or press the **Del** key.

  Note: If you do not want PlanMaker to make suggestions while you are typing, you can always disable this feature.

  Choose the ribbon command **File | Options**, switch to the **Edit** tab and disable the option **Autocomplete cells**.

Undoing changes

In the Quick access toolbar, you can use the **Undo** button to undo the last changes made to a document. If, for example, you format cells in a different font, all you have to do is choose the command **Undo** and it will be undone again.

This not only works for formatting, but for virtually all changes – for example, you can undo typing or the deletion of text.

The command **Undo** can be applied repeatedly if necessary. For example, you can choose it five times to reverse the last five changes. (Tip: You can set the maximum number of actions that can be reversed in the Settings, General tab.)

If you click on the small arrow next to the **Undo** button, you will see a list of recent changes to which you can revert. If you click on an entry in the list, several steps can be undone at the same time.

Redoing undone actions

There is also a counterpart to the command **Undo**: the **Redo** button on the Quick access toolbar. This restores the last action you undid. You can thus undo the cancellation of changes.

This command can also be chosen several times in succession. If, for example, you choose the command **Undo** five times, the last five changes will be undone. If you then choose the command **Redo** five times, you will restore the initial state.

Again, you can use the small arrow next to the **Redo** button to select entries from the list of recent actions to restore multiple steps at the same time.

Repeating commands

The command **Repeat** in the Quick access toolbar repeats the last command that you used with exactly the same settings.

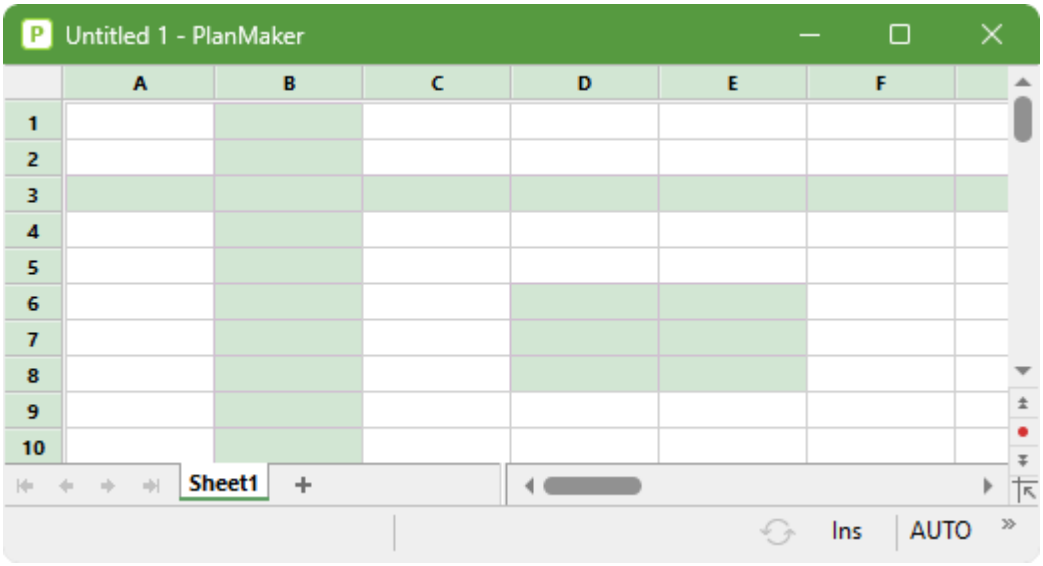
This is especially useful when applying formatting. If, for example, you have assigned a different font to a cell via the ribbon tab **Home** | group **Character**, you can then navigate to other cells and simply use the command **Repeat** there. These cells are then formatted in the same font – as if you had chosen the last command once again with the same settings.

There are many other commands that can be repeated in this manner as often as desired.

Selecting cells and cell content

For some PlanMaker commands, you can first *select* a range of cells before you execute them. The command will then be applied to *all* cells within the selection.

If, for example, you select some cells and then press the **Del** key on the keyboard, *all* selected cells will be deleted at once.



You can even select non-contiguous cell ranges (using the Ctrl key and the mouse).

The selection procedure differs depending on the operating system in use. Thus, this section is split into two parts:

- Selecting in the Windows, Mac or Linux version
- Selecting in the Android/iOS version

Selecting in the Windows, Mac or Linux version

In the Windows, Mac or Linux version of PlanMaker, you can select cells and cell content as follows:

Selecting via the mouse

To select something via the mouse, proceed as follows:

- **Selecting cell ranges**

  You can select arbitrary cell ranges by pressing and holding down the left mouse button while you drag the mouse cursor over the cells to be selected.

- **Selecting whole rows and columns**

  An entire column can be selected by clicking on its *column header*. Column headers are buttons displayed above the worksheet (labeled A, B, C, etc.).

  There are also *row headers* for each row, which are located to the left of the worksheet and are labeled 1, 2, 3, etc. If you click on a row header, the entire row will be selected.

  If you left-click and drag the mouse cursor over several column or row headers, you can select multiple columns or rows at the same time.

- **Selecting the entire worksheet**

  The button in the upper left corner of the worksheet window (above 1 and to the left of A) has a special function: clicking on it will select the entire worksheet.

  Alternatively, you can do the same via the ribbon command **Home | Select all** .

- **Selecting a single cell**

  To select a single cell, click on it, drag the mouse cursor to an adjacent cell, and drag it back again (while holding down the left mouse button).

- **Selecting non-contiguous cell ranges**

  You can also select non-contiguous cell ranges. To do so, hold down the **Ctrl** key and and then select the desired cell ranges with the mouse button. (Mac users: **Cmd** key)

- **Selecting only part of the cell content**

  To select only *part* of the content of a cell, proceed as follows:

  First, double-click on the cell to edit its content. Then, drag the mouse cursor over the characters that you want to select (while holding down the left mouse button).

- **Selecting objects**

  To select an object (for example, a picture, a drawing, etc.), simply click on it with the mouse. A frame will then appear around the object to indicate that it is now selected.

  To select multiple objects, switch to object mode (see Object mode).

Selecting via the keyboard

Alternatively, you can use the keyboard for selecting. To do so, proceed as follows:

- **Selecting cell ranges**

  To select a range of cells, hold down the **Shift**⇧ key, and select the cells by navigating with the arrow keys.

  If, for example, you navigate to cell B5, hold down the **Shift**⇧ key and press ↓ five times, cells B5 to B10 will be selected. For a list of all key combinations for moving and selecting within a worksheet, see Shortcut keys.

- **Selecting a single cell**

  To select a single cell, navigate to that cell, hold down the **Shift**⇧ key, use any arrow key to select an adjacent cell and then return to the original cell.

From the cell to be selected and while holding down the **Shift key**⇧, first press the → key and then the ← key.

- **Selecting non-contiguous cell ranges**

  Non-contiguous cell ranges can only be selected with the mouse (see above).

- **Selecting only part of the cell content**

  To select only *part* of the content of a cell, proceed as follows:

  Navigate to the cell and press the **F2** key to edit its content. Then, use the arrow keys together with the **Shift**⇧ key to select text in the usual manner.

- **Selecting objects**

  Objects can only be selected with the mouse (see above).

Selecting in the Android/iOS version

Selecting works a little differently on Android/iOS devices. You can use either your finger or a mouse. Proceed as follows:

- **Selecting cells**

  On Android/iOS devices, you can select worksheet cells as follows:

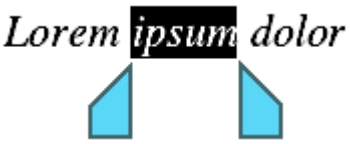
  Tap on the start cell once (you should *not* see a blinking text cursor in the cell now). After a short pause, tap on this cell again, and keep your finger pressed down while you drag it to the desired end cell.

  Tip: Use the ribbon command **Home | Select all** to select the *entire worksheet*.

- **Selecting only part of the cell content**

  To select only *part* of the content of a cell, proceed as follows:

  First, double-tap on the cell to edit its content. Then double-tap the word in the cell with which you want to start the selection. The word should now be selected. This is indicated by the large "handles" that appear in front of and behind the word:

  These two handles represent the beginning and end of the selection. To extend the selection to additional text, simply drag the two handles to the desired positions.

- **Selecting objects**

  To select an object (for example, a picture or drawing, etc.), simply click on it. A frame will then appear around the object to indicate that it is now selected.

  To select all objects, switch to object mode (see Object mode).

Deleting cells and cell content

There are two methods for deleting data from a worksheet:

- **Deleting the content of cells**

  If you use the **Del** key on the keyboard, only the *contents* of the selected cells will be deleted.

- **Deleting whole cells**

  If you use the ribbon command **Home** | group **Cells** | **Delete**, however, the selected cells will be *completely* deleted from the worksheet. The other cells in the worksheet will move up accordingly.

In addition, PlanMaker provides two commands specifically for deleting *empty rows* and *duplicate rows*:

- **Deleting empty rows**

  To delete all *empty* rows within a certain range of rows, select these rows, and then choose the ribbon command **Data** | group **Edit** | **Remove empty rows**.

- **Deleting duplicate rows**

  To delete all *duplicate* rows within a certain range of rows, select these rows, and then choose the ribbon command **Data** | group **Edit** | **Remove empty rows** | **Remove duplicate rows**.

For more information, see the following pages.

Deleting the content of cells

To delete the content of cells, proceed as follows:

1. Select the relevant cells.
   For more information on how to select cells, see Selecting cells and cell content.
2. Press the **Del** key on the keyboard (or the ribbon command **Home** | **Cut**).

Only the *contents* of the cells are removed and not the cells themselves (see next section Deleting whole cells).

Selective deletion

If you use the ribbon command **Home** | group **Contents** | **Delete** *(Delete special)* instead, you can determine exactly what will be deleted.

The program displays a dropdown menu with the following options:

- **All**

  This option deletes everything – cell contents as well as formatting, comments, conditional formatting and input validation of selected cells.

- **Formatting**

  This option deletes only the formatting (number format, character format, borders, etc.).

- **Contents**

  This option deletes only the contents of the selected cells. Everything else (formatting, comments, etc.) is retained.

- **Comments**

  This option only removes comments that have been added to the selected cells, leaving both content and formatting intact (see Using comments).

- **Conditional formatting**

  This option removes all conditional formatting of the selected cells (see Conditional formatting).

- **Input validation**

  This option removes all input validation of the selected cells (see Input validation).

- **Reset character formatting**

  This option removes all character formatting that you have applied to the selected cells (see also Resetting the character formatting).

- **Remove link**

  This option removes the hyperlink, but the cell or text passage to which it is assigned remains intact (see also Using hyperlinks).

Deleting whole cells

The ribbon command **Home** | group **Cells** | **Delete** not only deletes the content of the selected cells but also removes them completely from the worksheet.

If you click on this command, the program displays a dropdown menu in which you can determine how the other cells should be moved up after the deletion process.

The following options are available:

- **Delete cell, move cells up**

  If you select this option, the selected range of cells will be deleted and the cells below will move up.

- **Delete cell, move cells to the left**

  If you select this option, the selected range of cells will be deleted and the cells to the right of it will move to the left.

- **Remove complete rows**

  This option deletes the complete *rows* within the selected range of cells and not just the selected cells. The rows below move up accordingly.

- **Remove complete columns**

  This option deletes the complete *columns* within the selected range of cells and not just the selected cells. The columns to the right of them move to the left accordingly.

- **Delete worksheet**
  This option removes the worksheet (including its content). For more information, see Managing worksheets.

Deleting empty rows

To delete all *empty* rows within a certain range of rows, select the range from which you want to remove empty rows.

Then choose the ribbon command **Data** | group **Edit** | **Remove empty rows** .

PlanMaker scans this range for rows that are completely empty (i.e. where none of their cells contain a value). All corresponding rows are deleted.

Note

Normally, you select a range of *complete* rows before using this command. However, you can also select a cell range that doesn't consist of complete rows. PlanMaker will then ask you if you want it to delete the corresponding rows *completely*:

- If your answer is **Yes**, all corresponding rows will be deleted completely.
- If your answer is **No**, only cells within the selection will be deleted. The cells to the left and right of the selection will remain unchanged.

Deleting duplicate rows

To delete all *duplicate* rows within a certain range of rows, select the range from which you want to remove these rows.

Then choose the ribbon command **Data** | group **Edit** | **Remove empty rows** | **Remove duplicate rows**.

PlanMaker scans this range for rows that are completely identical (i.e., where the contents of *all* cells in one row are identical to the corresponding contents in another row). It then deletes these duplicate rows.

Note

Normally, you select a range of *complete* rows before using this command. However, you can also select a cell range that doesn't consist of complete rows. PlanMaker will then ask you if you want it to delete the corresponding rows *completely*:

- If your answer is **Yes**, all corresponding rows will be deleted completely.
- If your answer is **No**, only cells within the selection will be deleted. The cells to the left and right of the selection will remain unchanged.

Inserting new cells

The ribbon command **Home** | group **Cells** | **Insert** inserts new cells, rows or columns in a worksheet.

If you choose this command, the program displays a dropdown menu in which you can select how to insert the cells.

The following options are available:

- **Insert cell, move cells down**

  The program inserts the exact number of cells as were selected before choosing this command. If nothing was previously selected, only one cell will be inserted.

  The cells below the insertion will be moved down.

- **Insert cell, move cells to the right**

  The program inserts the exact number of cells as were selected before choosing this command. If nothing was previously selected, only one cell will be inserted.

  The cells to the right of the insertion will be moved further to the right.

- **Insert complete rows**

  PlanMaker will insert as many *complete rows* as were included within the selection. If nothing was previously selected, only one row will be inserted.

  The rows below the insertion will be moved down.

- **Insert complete columns**

  PlanMaker will insert as many *complete columns* as were included within the selection. If nothing was previously selected, only one column will be inserted.

  The columns to the right of the insertion will be moved further to the right.

Note: When inserting new cells, the cell formatting of the previous row/column is applied to them. Furthermore, when inserting complete rows/columns, the row height/column width of the previous row/column is also applied to them.

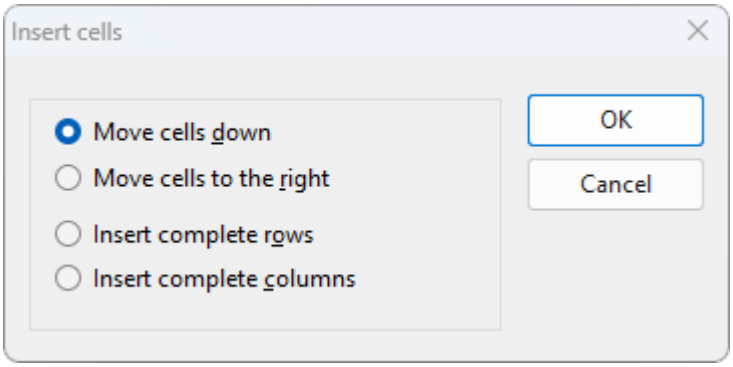
The dropdown menu of the command **Insert** also contains the following options:

- **Insert copied cells**, see next section
- **Insert worksheet**, see Creating new worksheets

Inserting copied cells

You sometimes want to insert copied cells in a place that already contains values. If you only want to move these values rather than overwrite them when pasting something from the clipboard, you have the following option:

The ribbon command **Home** | group **Cells** | **Insert** | **Insert copied cells** inserts cells, rows or columns from the clipboard into the worksheet.



The program opens a dialog box in which you can select how the cells should be inserted:

- **Move cells down**

  The copied cells from the clipboard are pasted into the selected place.

  The cells below the insertion will be moved down.

- **Move cells to the right**

  The copied cells from the clipboard are pasted into the selected place.

  The cells to the right of the insertion will be moved further to the right.

- **Insert complete rows**

  The copied rows from the clipboard are pasted into the selected place.

  The rows below the insertion will be moved down.

- **Insert complete columns**

  The copied columns from the clipboard are pasted into the selected place.

  The columns to the right of the insertion will be moved further to the right.

Moving and copying

All operating systems supported by PlanMaker have an extremely useful feature: the *clipboard*.

The clipboard works as follows: You select something in the worksheet and then cut or copy the selected content *to the clipboard*. You can then reinsert the selected content elsewhere in the worksheet. The clipboard thus enables cells (as well as objects) to be deleted, copied or moved.

All necessary commands can be found on the ribbon tab **Home** | group **Edit**:

Command		Explanation
Cut		The ribbon command **Home**	**Cut** deletes the content of a selection – but not permanently. This content is moved to the clipboard and can be pasted from there at any time.
Copy		The ribbon command **Home**	**Copy** copies the content of the selection to the clipboard.
Paste*		To paste the contents of the clipboard, choose the command **Home**	**Paste**. Position the cursor in the desired place, and then choose this command (click directly on the icon itself). You can also insert the content of the clipboard multiple times in this manner.
Paste special		If you click on the small arrow below the icon rather than directly on it at **Home**	**Paste**, a dropdown menu opens with additional options for pasting the content. For more information, see Paste special.

* For information on how to paste from the clipboard without overwriting the existing values in the selected place, see Inserting copied cells.

This works in the same manner for objects such as pictures, charts and drawings.

Pasting into a selected cell range

If you copy cells to the clipboard and then *select* a range of cells before choosing the ribbon command **Home** | **Paste**, PlanMaker will *fit* the pasted cells into this selection.

This is shown as follows:

- If the selected cell range is *smaller* than the cell range to be pasted, all cells that do not fit will be omitted.
- If the selection is *larger* than the cell range to be pasted, the cells will be repeated until the selected range is completely filled.
- There's one exception:

  If the selected cell range consists of just one row, PlanMaker will still paste *all* columns of the copied cells.

  The same applies when the selected cell range consists of just one column.

Moving and copying with the mouse ("drag and drop")

Android/iOS: This feature is not available in the Android or iOS version.

You can also move or copy cells by simply selecting them, dragging them with the mouse to a different place and dropping them there. This "drag and drop" technique enables you to move or copy cell contents very quickly.

To do so, proceed as follows:

1. Select the relevant cells.
2. Press the selected cells with the left mouse button and wait about one second until a small dashed rectangle appears on the mouse cursor.
3. While still holding the mouse button down, drag the selected cells to the target position.
4. When you release the mouse button, the cells will be *moved* to the target position.

   If you hold down the **Ctrl** key while releasing the mouse button, the cells will instead be *copied* to the new position.

   If you hold down the **Alt** key while releasing the mouse button, the program displays a dialog box in which you can choose whether the cells are to be copied or moved. When you choose "Copy", you can also specify whether only the contents of the cells or also their formatting should be copied.

Copying and moving formulas with relative coordinates

If you copy or move cells that contain formulas, the cell addresses in those formulas are automatically updated to the new place in the worksheet.

An example:

Let's suppose that cell A7 contains a formula that adds up the values from the four cells above:

```
=SUM(A3:A6)
```

You now copy this formula to position B7. PlanMaker will automatically update the formula so that it will work again with the four cells above the total row. Cell B7 will thus contain the following formula:

```
=SUM(B3:B6)
```

If you do not want this to happen, use *absolute* cell addressing instead of the usual *relative* cell addressing. For more information, see Relative versus absolute cell references.

Paste (special)

There is a more powerful variant for the ribbon command **Home** | group **Edit** | **Paste** .

If, for example, you have copied text or complete cells and click on the small arrow below the **Paste** icon, you can use the dropdown menu to specify in which form the content from the clipboard should be pasted.

In detail:

When you use the commands **Cut** or **Copy** to save information to the clipboard, it is saved in multiple formats simultaneously. If, for example, you have copied text, it will appear in the clipboard in both formatted and unformatted form.

Normally, you don't need to worry about this because PlanMaker automatically selects the most appropriate format when you paste the content of the clipboard into the document by clicking *directly on the icon* **Paste**. Alternatively, you can select the format yourself by clicking *on the arrow of the icon* **Paste** .

The following options are available here:

Pasting content via the dropdown menu of the command

If you click on the small arrow of the ribbon command **Home** | group **Edit** | **Paste** , you will get a dropdown menu with various entries for direct selection:

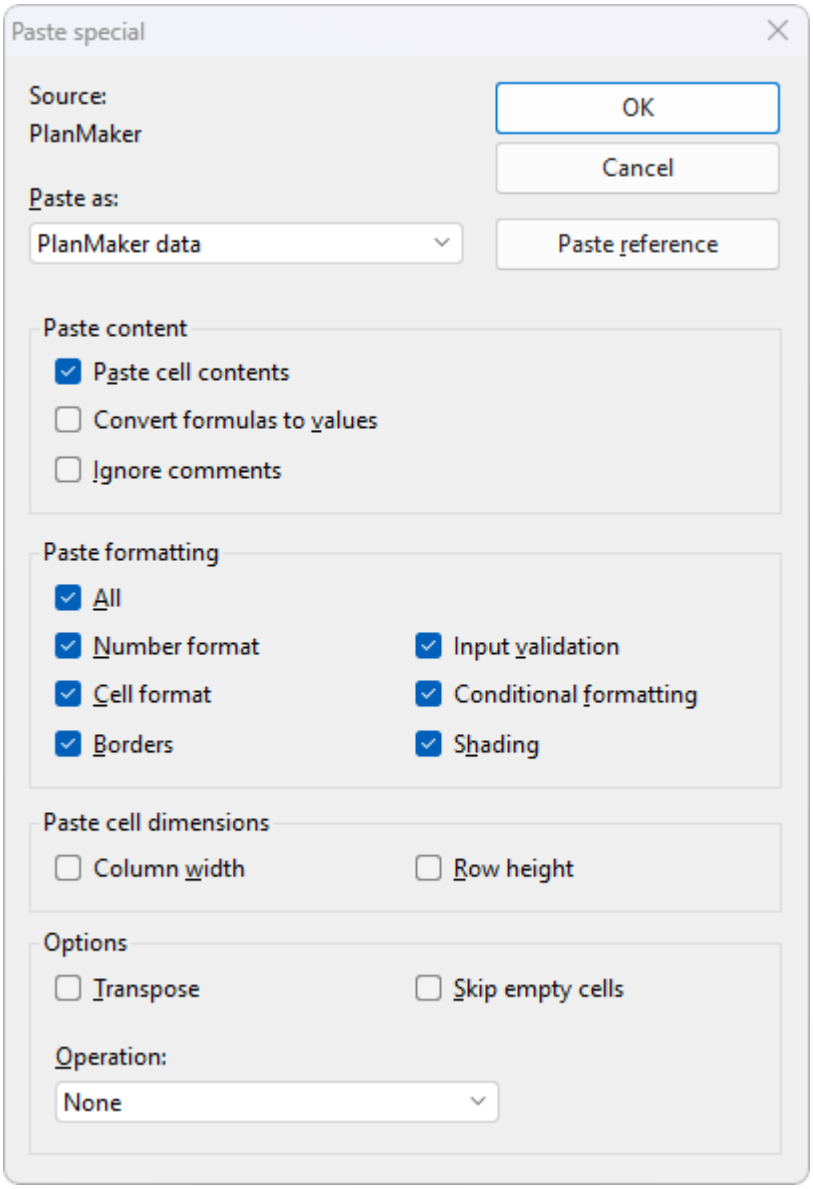
- **Paste** : The original formatting of the copied content is retained. This results in the same program behavior as when you click directly on the **Paste** icon above the small arrow.
- **Paste unformatted text** : This option appears contextually if you have copied text. If you select this entry, the original formatting of the copied text will be removed, and it will adopt the formatting of the passage where it is pasted. This option can also be found in the dialog box **Paste special** (see below).
- **Paste picture** : This option appears contextually if you have copied a picture. Use this entry to paste the picture from the clipboard.
- *Paste formats, Paste values, Other paste options*: appear contextually if you have copied complete worksheet cells. You will be offered some common paste options that you will also find in the dialog box **Paste special** (see below).
- **Paste special**: This option opens a dialog box with differentiated options for pasting contents from the clipboard (see below).

Pasting content via the dialog box with additional options

If you select the **Paste special** entry from **Home** | group **Edit** | **Paste** , the program displays a dialog box in which you can specify exactly how the content from the clipboard is to be pasted.

The dialog box displays a list of all formats in which the information currently on the clipboard is stored. If you select the desired format from the **Paste as** list and then confirm with **OK**, the content of the clipboard will be pasted in the selected format.

Furthermore, this dialog box offers several additional options, but they are only available if worksheet cells have been copied to the clipboard:



The following additional options are available:

"Paste content" section

Here, you can specify how the *content* of the cells in the clipboard should be pasted:

- **Paste cell contents**

  If this option is enabled, the content of the copied cells will also be pasted. If it is disabled, only their formatting will be pasted.

- **Convert formulas to values**

  If this option is enabled, PlanMaker will paste the results of formulas in the copied cells rather than the formulas themselves. So if you copy a cell containing the calculation "=A1*2" to the clipboard, PlanMaker will calculate its current value and paste the result as a numeric value.

- **Ignore comments**

  If this option is enabled, comments that have been added to the copied cells will be omitted when pasting.

"Paste formatting" section

These options determine whether the *formatting* of the copied cells is to be retained when pasting:

- **All**

  The cells retain their entire formatting when they are pasted. (Selecting this option enables all the other options listed below.)

- **Number format**

  The cells retain their number format when they are pasted.

- **Cell format**

  The cells retain their cell format when they are pasted.

- **Input validation**

  The cells retain their input validation settings when they are pasted.

- **Conditional formatting**

  The cells retain their conditional formatting settings when they are pasted.

- **Borders, Shading**

  The cells retain their borders/shading when they are pasted.

"Paste cell dimensions" section

These options determine whether the *cell sizes* of the copied cells should also be transferred to the target cells when pasting:

- **Column width**

  The cells retain their column width when they are pasted.

- **Row height**

  The cells retain their row height when they are pasted.

"Options" section

This section contains the following additional options:

- **Transpose**

  If this option is enabled, the row and column order of the cells will be reversed when they are pasted. Thus, rows will become columns and columns will become rows.

- **Skip empty cells**

  If this option is enabled, all empty cells will be skipped when the copied cells are pasted. Thus, an existing cell will keep its current content and formatting if you paste an empty cell over the existing cell.

- **Operation**

  This option can be useful when you paste cells over *existing* cells. It determines whether the existing values should be combined with the copied values. For this purpose, the four basic arithmetic operations are available:

  None: Do not combine the copied values and the existing values. Just overwrite the existing values. This is the default setting.

  Add to destination: Add the values in the clipboard to the existing values.

  Subtract from destination: Subtract the values in the clipboard from the existing values.

  Multiply with destination: Multiply the existing values by the values in the clipboard.

  Divide the destination: Divide the existing values by the values in the clipboard.

  Example: Copy a cell that contains the value 2 to the clipboard. Then select a cell that contains the value 8, and choose the command **Home | Paste | Paste special**. If you choose the option **Add to destination** in the dialog box, the cell will then contain the value 10 (8+2=10).

"Paste reference" button

The **Paste reference** button ignores any settings made in this dialog box, as it has the following special function:

When you copy cells to the clipboard, choose the command **Home | Paste | Paste special** and then click on the **Paste reference** button, *cell references* to the addresses of these cells will be pasted instead of the *cell contents*.

If you copy cell B15 to the clipboard, this button will insert the cell reference =B15.

Tip: This works even if cells have been copied to the clipboard from another document. In this case, PlanMaker will automatically insert an *external* cell reference. For more information, see External cell references.

Filling cells automatically

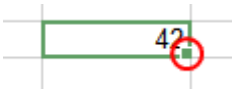
You can use the **Fill** function to copy the contents of one cell to other cells in an instant or to create an increasing or decreasing number series across several cells based on this cell content.

There are two ways to use this function:

Filling with the mouse

The easiest way to fill cells automatically is via the mouse. Proceed as follows:

1. Enter the initial value in a cell.
2. Click on that cell (to make it the currently selected cell).
3. Hover over the small square in the bottom right corner of the cell frame that surrounds the current cell (outlined in red in the figure):



4. Drag this square in the desired direction while holding down the mouse button. Release the mouse button when the target position has been reached.

All cells in the selected area have now been filled automatically.

What the cells are filled with depends on the content of the initial cell:

- If the initial cell contains **text**, PlanMaker will fill all cells with this text.
- If the initial cell contains a **number** (e.g., 42), PlanMaker will fill successive cells with incrementing values (in this case, 43, 44, 45, etc.).

Tip 1: If you do not want PlanMaker to fill the cells with a series of numbers but just want to have all cells filled with 42, press and hold down the **Ctrl** key while dragging with the mouse.

Tip 2: By double-clicking on the small square, you can also fill a series if there are values in the column to the left of it. The series then expands to the end of the values in the column to the left of it. In this case, too, you can use the **Ctrl** key to fill the series with the same values.

If you press and hold down the **Alt** key while dragging, the program will display the following dialog box in which you can specify exactly how the cells should be filled.

Filling via buttons/dialog box

To fill cells automatically via buttons/dialog box, proceed as follows:

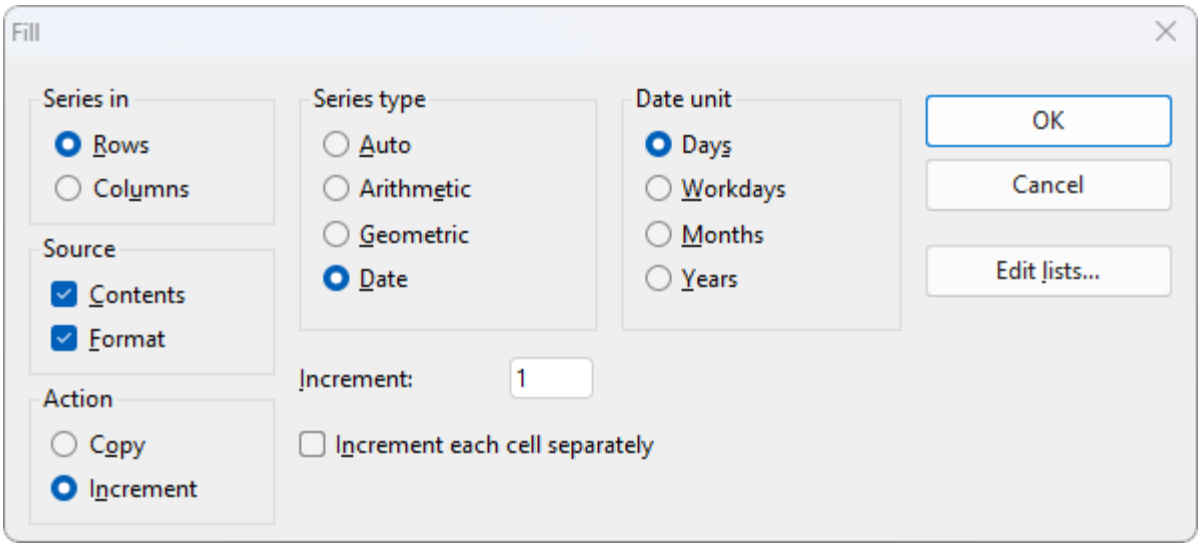
1. Enter the initial value in a cell.

2. Select the cell with this initial value and all cells that you want to fill.
3. Choose the ribbon command **Home** | group **Contents** | **Fill** .
4. In the dropdown menu, specify the fill direction (starting from the cell with the initial value and moving in the direction of the selected cells): **Down**, **Right**, **Up** or **Left**.

   As soon as you have selected a direction, the corresponding cells are filled with the initial value (copied as value and format but not incremented).
5. If you want to make further settings (for example, increment values instead of copying them), select the entry **Series** in the dropdown menu to open the dialog box. Here, you can specify exactly how the cells should be filled (see below).

   As soon as you confirm with **OK** in the dialog box, the selected cells are filled according to your settings.

In the dialog box, you can make the following settings:



- **Series in**

  Here, select whether to fill in the direction of the **rows** (to the right) or in the direction of the **columns** (downwards).

  Note: If you have selected only one row or only one column, this option is omitted and PlanMaker automatically presets rows or columns.

- **Source**

  Enable the option **Contents** if you want the cells to be filled with values.

  Enable the option **Format** if you also want the formatting of the source cell to be applied to the new cells.

  Both options are enabled by default.

- **Action**

  This option corresponds to pressing the **Ctrl** key when filling cells with the mouse. It lets you determine whether to copy values or to increment them from cell to cell:

  If, for example, the initial cell contains 42 and you select **Copy**, all cells will be filled with 42.

If you select **Increment**, successive cells will be filled with a series of numbers (in this case, 43, 44, 45, etc.).

The numeric increment can be specified using the other options in this dialog box:

- **Series type**

  Here, you specify how the number series is to continue:

  Auto:

  This option corresponds to an arithmetic series: Each additional value will be increased by the increment. However, PlanMaker determines the increment automatically.

  If, for example, you continue two cells with the values 10 and 20, the next cells will be filled with 30, 40, 50, etc. For more information, see Tips and tricks in the next section.

  Arithmetic:

  Here, each additional value will be increased by the increment.

  If, for example, the initial cell contains 1 and you specify an increment of 2, the following cells will be filled with 3, 5, 7, 9, 11, etc.

  Geometric:

  Here, each additional value will be multiplied by the increment.

  If, for example, the initial cell contains 1 and you specify an increment of 2, the following cells will be filled with 2, 4, 8, 16, 32, etc.

  Date:

  Here, each additional value in the selected **Date unit** (see below) will be increased by the specified increment. This type of series is only useful if the initial cell contains a date.

  If, for example, the initial cell contains the date 1/1/2018, the **Increment** is set to 1 and the **Date unit** is set to "Months", the new cells will be filled with 2/1/2018, 3/1/2018, 4/1/2018, etc.

- **Date unit**

  This option is only available if the **Series type** is set to "Date".

  It allows you to specify the date unit of the increment:

  Days: increment by days

  Workdays: increment by days, excluding Saturdays and Sundays. If, for example, the initial value is a Friday, the next value would be the Monday (not Saturday).

  Months: increment by months

  Years: increment by years

- **Increment**

  Here, you specify the amount by which each value will increase from cell to cell. See above for examples.

  Positive or negative increment values can be used.

  This option is not available if the **Series type** is set to "Auto", as PlanMaker automatically determines the increment in this case.

- **Increment each cell separately**

  When this option is enabled, the specified increment will be added to each of the currently selected cells individually.

  Example:

  You select three cells containing the values 1, 2, 3, and try to fill the cells below them.

  If this option is disabled, this sequence is simply continued and the cells are filled as follows: 4, 5, 6, 7, 8, 9, 10, 11, 12, etc.

  If you enable this option, the cells are filled as follows: 2, 3, 4, then 3, 4, 5, then 4, 5, 6, etc.

- **Edit lists** button

  You can also create your own lists for the **Fill** function.

  If, for example, you create a list with the content "Red", "Green" and "Blue", and then subsequently start a fill operation from a cell that contains the text "Red", PlanMaker will continue the series and fill with Green, Blue, Red, Green, Blue, etc.

  For more information on how to create such lists, see Editing lists for automatic filling.

Tips and tricks for automatic filling

The many options that are available for the **Fill** function make its application seem more difficult than is actually the case. This section offers some tips and tricks on how to use this extremely useful feature most effectively.

Tip: The quickest way to fill a range of cells is to use the mouse to drag the small rectangle in the bottom right corner of a cell in the appropriate direction. PlanMaker then sets the series type to "Auto". This means it will automatically determine which values to use to fill the cells (see following examples).

- **Filling with text**

  When filling with text, you do not need to worry about any options, as text can only be copied anyway.

  If you want to fill multiple cells with the same text, simply enter it into a cell, click on that cell, and then drag the small rectangle in the bottom right corner of the cell frame over the cells you want to fill with it.

- **Filling with numbers**

  If you use the mouse to fill cells with numbers, PlanMaker will automatically determine the increment.

  If, for example, you drag the small rectangle from a cell with 42, the other cells will be filled with the values 43, 44, 45, etc.

  If the initial cell value is 10 and the cell value below it is 20, and you select *both* cells and drag the small rectangle downwards, PlanMaker automatically recognizes how to continue this series and fills the other cells with 30, 40, 50, etc.

- **Filling with formulas**

  If you use the mouse to fill cells with formulas, PlanMaker automatically attempts to update the formulas in the new cells with incremented cell addresses.

  For example, the formula =A1*2 in the initial cell will be updated to =A2*2, =A3*2, etc., in the other cells.

- **Filling with dates**

  Filling with dates is identical to filling with numbers:

  Enter the start date in the initial cell and drag the small rectangle over the desired cells. PlanMaker will add one day to each additional cell.

  If necessary, you can use the ribbon command **Home** | group **Contents** | **Fill** | **Series** to specify via the dialog box whether to increment by days, months or years.

- **Filling with structures**

  As you have already learned, you can also select multiple cells and continue them. What if the selected cells have no automatically recognizable increment or completely different values?

  In this case, PlanMaker will attempt to find a pattern within these values. If, for example, you enter the values 1, 2, text, PlanMaker will continue the series with 3, 4, text, 5, 6, text, 7, 8, text, etc.

  If the values are solely strings, they will be simply repeated. If the initial cells are filled with Red, Green, Blue, the following cells will be filled with Red, Green, Blue, Red, Green, Blue, etc.

- **Filling with custom lists**

  If you choose the ribbon command **Home** | group **Contents** | **Fill** | **Series**, you can create your own lists for the **Fill** function using the **Edit lists** button in the dialog box that opens.

  If, for example, you create a list with the content "Red", "Green" and "Blue", and then subsequently start a fill operation from a cell that contains the text "Red", PlanMaker will continue the series and fill with Green, Blue, Red, Green, Blue, etc.

  Note: If necessary, you can also add a separate **Edit lists** button for direct access to a ribbon tab. For more information on this topic and all other information on creating your own lists, see Editing lists for automatic filling.

Working with multiple worksheets

PlanMaker worksheets do not necessarily have to consist of a single worksheet. You can also stack several worksheets on top of each other – just like a stack of papers. Thus, PlanMaker documents are also called *workbooks*. Each workbook can contain up to 2048 *worksheets*.

The main advantage of this is that calculations in one worksheet can also refer to cells in other worksheets of a workbook, thus letting you create "three-dimensional" calculations as such. For more information, see Calculating in three dimensions.

In this section, you will learn everything you need to know about working with multiple worksheets in a document. The following topics are covered:

- **Creating new worksheets**

  To create a new worksheet, choose the ribbon command **Insert** | **Sheet** | **Insert**.

- **Working with the worksheet register**

  The *worksheet register* displayed at the bottom of the document window allows you to switch between the worksheets of a document with a single mouse click. You can also use it to create, move and manage worksheets.

- **Managing worksheets**

  The context menu of the worksheet register and the ribbon command **Insert | Sheet** both contain commands for managing worksheets (Rename, Copy, Move, etc.).

- **Calculating in three dimensions**

  In the last section, you will learn how to perform calculations across multiple worksheets.

The following pages provide detailed information on these topics.

Creating new worksheets

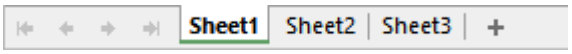
To add a new worksheet, choose the ribbon command **Insert** | group **Tables** | **Sheet** . The program opens a dropdown menu in which you can choose the command **Insert**.

Tip: You can create a new worksheet faster with the icon in the worksheet tab.

PlanMaker creates the worksheet and makes it the active worksheet.

Working with the worksheet register

The *worksheet register* at the bottom of the document window displays tabs for each worksheet in the current document.



The active worksheet is highlighted.

You can use the worksheet register to perform the following actions:

- **Switching between worksheets**

  To switch to a different worksheet, simply click on its tab in the worksheet register.

  Note: If there are more worksheet tabs than can be displayed at the same time, you can scroll through the list using the arrows displayed to the left of the register.

  Tip: You can also switch between the worksheets via the keyboard: Use **Ctrl+Tab** to move to the next worksheet and **Ctrl+Shift+Tab** to move to the previous worksheet.

- **Changing the order of worksheets**

  The worksheet register can also be used to move worksheets, thus allowing you to change their order.

  To do so, left-click for about one second on the tab of the sheet that you want to move and, while still holding down the mouse button, drag the tab to the desired position in the worksheet register.

- **Editing multiple worksheets at the same time**

  You can select multiple worksheets in the worksheet tab and edit them *simultaneously*. To select multiple worksheets, press and hold the **Ctrl** key while you click on their tabs in the register. (Mac users: **Cmd** key)

  If multiple worksheets are selected, changes made to one of the worksheets will affect all of the other selected worksheets in exactly the same way.

  Examples:

  If you change the formatting of cells, your changes will also be applied to the same cells in the other selected worksheets.

  When you edit or delete the content of a cell, the same cell in the other selected worksheets will be changed in the same manner.

  Changes to the cell size (ribbon tab **Layout** | group **Cells**) also affect all selected worksheets.

  Tip: To select multiple worksheets at the same time, use the **Shift key** instead of **Ctrl**. To **deselect**, click again with the mouse and **Ctrl** or **Shift** on the tab(s).

- **Managing worksheets**

  When you right-click on one of the tabs in the worksheet register, the program will display a context menu that contains all commands for creating and managing worksheets. For more information, see the next section Managing worksheets.

- **Creating a new worksheet**

  The tab to the right of the worksheet register with the + icon serves as a "shortcut" for creating new worksheets: When you click on it, PlanMaker immediately creates a new worksheet.

Managing worksheets

If you choose the ribbon command **Insert** | group **Tables** | **Sheet** , a dropdown menu opens with commands for managing worksheets.

Tip: These commands (and a few more) also appear in the context menu by *right-clicking* on the desired tab in the worksheet register at the bottom.

The menu contains the following commands:

- The command **Insert** creates a new worksheet.
- The command **Copy** creates a copy of the current worksheet. To do so, the program displays a submenu in which you can choose where the copy should be placed (within the other worksheets).

  If you want to place the copy in a different (currently open) document or in a new document, choose the option **To another document**.

- The command **Move** moves the worksheet to a different location. As with the command **Copy**, the program opens a submenu in which you can choose the location.
- The command **Delete** removes a worksheet along with its contents.

  Alternatively, you will find the command under **Home** | group **Cells** | **Delete** | **Delete worksheet**.
- The command **Rename** lets you change the name of a worksheet.

  Tip: You can also simply double-click on the worksheet in the worksheet register at the bottom.
- The command **Hide** ensures that the worksheet is hidden.

  Alternatively, you can use the command **Home** | group **Cells** | **Visibility** | **Hide sheet**.

  Hidden worksheets are invisible. They cannot be displayed or edited, and they do not even appear in the worksheet register. However, PlanMaker continues to use them for calculations.
- The command **Show** makes hidden worksheets (see command **Hide**) visible again.

  Alternatively, you can use the command **Home** | group **Cells** | **Visibility** | **Show sheet**.

  This opens a dialog box that shows all hidden worksheets. In the list, click on the sheet that you want to show again. Use the **Select all** button to select all hidden sheets from the list. Press **OK** to make the selected worksheets visible again.
- The commands **Select all sheets** and **Deselect sheets** (in the context menu of the worksheet register) select or deselect all worksheet tabs in the worksheet register.
- You can choose the command **Page Setup** (in the context menu of the worksheet register) to make settings such as page size, page margins, etc., for a worksheet. The dialog box corresponds to the ribbon tab **Layout** | command group **Page Setup** ◢. For more information, see Page setup.
- The command **Copy page setup** (in the context menu of the worksheet register) allows you to copy the page setup of a worksheet – i.e., all settings made with **Page setup** – to another worksheet. The program opens a submenu with the following commands:

  To all sheets: Copies the page setup from the current worksheet to all other sheets.

  From...: Copies the page setup from any other worksheet to the current worksheet.
- You can choose the command **Properties** to make general settings for a worksheet. For more information, see Changing worksheet properties.

Tip: As previously described in Working with the worksheet register, you can select multiple worksheets in the worksheet register. Some of the aforementioned commands (for example, **Hide**) can then be applied to all selected sheets at the same time.

Note: If you enable *workbook protection* for a document, you can no longer make any changes to the worksheet register in it. The ribbon commands under **Insert** | **Sheet** are no longer available. For more information, see Workbook protection.

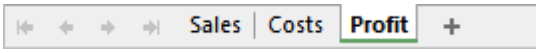
Calculating in three dimensions

The specific advantage of documents with multiple worksheets is that you can also perform calculations across multiple worksheets.

To do so, specify the name of the desired worksheet before the cell address (separated by an exclamation point). For example, "Sheet1!A1" refers to cell A1 on worksheet "Sheet1".

An example

Let's assume that you want to create a profit and loss statement. For this purpose, you have created three worksheets named "Sales", "Costs" and "Profit".



You now want the value in cell D2 of the "Costs" worksheet to be deducted from the value in cell A8 of the "Sales" worksheet and to appear in cell C9 of the "Profit" worksheet.

To do so, switch to the "Profit" worksheet. Navigate to cell C9 in this worksheet and enter the following formula:

```
=Sales!A8-Costs!D2
```

After pressing the **Enter key**↵, cell C9 in the "Profit" worksheet will contain the result of your calculation.

Naming cell ranges

Choose the ribbon command **Edit names** to assign *names* to cell ranges.

If, for example, your worksheet has four columns with the sales figures for spring, summer, autumn and winter, then just name these columns accordingly. You can now perform the following calculation:

```
=SUM(Spring)
```

The ribbon tab **Formula** | group **Named areas** contains all commands that you need for defined cell ranges.

For more information on working with cell ranges, see the following pages. The following topics are covered:

- Assigning and editing names for cell ranges
- Defining names automatically from the selection
- Creating a list of all named ranges
- Using named ranges
- Applying names to existing formulas

Assigning and editing names for cell ranges

Assigning names

To assign a name to a cell range, proceed as follows:

1. Select the cell range that you want to name.
2. Choose the ribbon command **Formula** | group **Named areas** | **Edit names** .
3. In the dialog box, enter the desired name in the **Name** field.
4. Click on the **Add** button.

You have now assigned the specified name to the selected cell range.

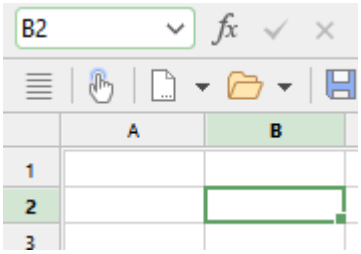
This name can now be used in formulas to represent the cells in the specified range. If you assign the name "January" to the cell range from A2 to A10, the formula =SUM(January) is equivalent to =SUM(A2:A10).

Note: Names for cell ranges may only consist of letters, numbers, underscores and periods. The first character must be a letter or an underscore.
Furthermore, names must not be structured like cell references. For example, the name "A2" is not allowed because PlanMaker would not be able to distinguish it from the cell address A2.

Tip: A faster method

To assign names even faster, proceed as follows:

In the top left corner of the worksheet, there is an editable dropdown list that normally displays the current cell address.



Use this dropdown list to quickly assign a name to a cell range as follows:

1. Select the cell range that you want to name.
2. Use the mouse to click into this editable dropdown list.
3. Enter the new name for the selected cell range.
4. Press the **Enter key**↵.

You have now assigned the specified name to the selected cell range.

Note: In this dropdown list, you can also display the existing named ranges at any time.

Renaming or deleting names

To rename or delete a named cell range, proceed as follows:

1. Choose the ribbon command **Formula** | group **Named areas** | **Edit names** .
2. In the list of names, select the desired name.
3. To change the name, click on the **Rename** button. To delete it, click on the **Delete** button.

The name has now been changed or deleted.

Note: If you delete a name, all formulas that use this name will now return a #NAME? error value instead of a result. If, for example, a cell range was named "January", all references to this named range, such as =SUM(January), will display a #NAME? error value.

Defining names automatically from the selection

If you select cells that contain values as well as row or column headings, you can let PlanMaker generate named ranges automatically from these cells. PlanMaker automatically retrieves the names from the headings.

Proceed as follows:

1. Select the desired cell range. It must contain both the values and the headings to be used for naming.
2. Choose the ribbon command **Formula** | group **Named areas** | **Create names** .
3. PlanMaker displays a dialog box in which you specify the row and/or column containing the headings to be used for naming (see example).
4. Confirm with **OK**.

The names will now be created from the selected headings.

An example

You have created the following table:

	January	February	March
London	100	450	333
Paris	345	543	564
Madrid	123	543	555

Now select all values *and* all headings, and choose the ribbon command **Formula** | **Create names**. Then specify the row and/or column from which names are to be created:

- If you select **Top row**, PlanMaker will assign the names "January", "February" and "March" to the cells below them.
- If you select **Left column**, PlanMaker names the cells to the right of "London", "Paris" and "Madrid" with the corresponding name.
- If both options are selected, PlanMaker will create all six names listed above.

Creating a list of all named ranges

You can instruct PlanMaker to insert a list of all named cell ranges into the worksheet. This is useful if you want an overview of all named cell ranges in the worksheet and the corresponding cell addresses.

Proceed as follows:

1. Select the cell where the list is to be inserted.
2. Choose the ribbon command **Formula** | group **Named areas** | **Export names** .

A list of all named ranges will now be inserted into the worksheet.

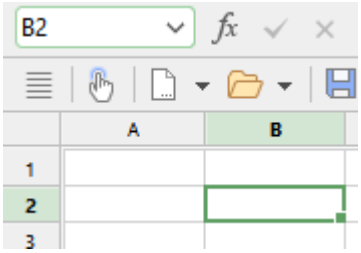
Using named ranges

If you have named cell ranges using the ribbon command **Formula** | group **Named areas** | **Edit names** , you can perform various operations much more efficiently.

You use named ranges in practice as follows:

Quickly selecting a named range

In the top left corner of the worksheet window, you will see a dropdown list that displays the address of the currently selected cell.



If you open this dropdown list, a list of all named ranges will be displayed. If you click on a name, the corresponding cell range will be immediately selected.

Using names in formulas

In formulas, range names can be used instead of the cell addresses they represent. This not only saves time, but it also makes formulas clearer.

An example:

You have entered sales figures for January in cells A2 to A10. You have also assigned the name "January" to this cell range.

To sum up the sales for January, you simply type:

=SUM(January)

This method is also more easily comprehensible than the default naming convention of =SUM(A2:A10).

Of course, you can now also name the sales for February, March, etc., accordingly.

Tip: The dialog box of the ribbon command **Formula | Function** also contains an entry called "Named ranges" in the **Category** list. If you select this category, all named ranges will be listed in the **Function** list so that you can easily insert them into formulas.

Applying names to existing formulas

The ribbon command **Apply names** on the ribbon tab **Formula |** group **Named areas** is a useful option for using named ranges: It scans all formulas in the selected range. If it finds a cell reference within a formula for which a name has already been defined, it replaces it with the name.

You use this command as follows:

1. Select the range of cells whose formulas you want to update. If you do not select anything, the *entire* worksheet will be affected.
2. Choose the ribbon command **Formulas |** group **Named areas | Apply names**.
3. In the **Names** list, select all names that you want to apply. Alternatively, click **Select all** to select all names.
4. Confirm with **OK**.

PlanMaker now scans all cell references within the selected range. Any cell reference that has a name assigned to it will be replaced by the corresponding name.

An example:

The name "January" is assigned to cells A2 to A10. One of the cells in the worksheet contains the formula =SUM(A2:A10). If you now use the ribbon command **Formula |** group **Named areas | Apply names**, the formula in this cell will be converted to =SUM(January).

The option "Also apply to relative references"

The aforementioned dialog box contains the option **Also apply to relative references**. This option should normally remain enabled.

This option means the following:

In PlanMaker, names always refer to cell ranges with *absolute* addressing (like $A$1:$A$10). However, relative addressing is usually used in calculations (for example, A1:A10).

If this option is enabled, PlanMaker will also convert relative cell references.

If the option is disabled, PlanMaker will *only* convert absolute cell references.

An example:
Let's assume that the range from A1 to A10 (more precisely: $A$1:$A$10) was named "January". If you apply the **Apply names** function to the formula =SUM($A$1:$A$10), this formula will always be converted to =SUM(January). If you apply it to the formula =SUM(A1:A10), however, the formula will be converted only if the option **Also apply to relative references** is enabled.

For more information on this topic, see Relative versus absolute cell references.

Transposing cells

The ribbon command **Data** | group **Edit** | **Transpose** reverses the row and column order of the selected cells. Thus, rows will become columns and columns will become rows.

Select the desired cell range before you use this command.

Important: Please note that transposing does not adapt any formulas or range names to the changes made to cell addresses by transposing, neither inside nor outside the transposed range!

Splitting text into multiple columns

In the iOS version, the command is locked. (To unlock the command, see Welcome.)

By choosing the ribbon command **Data** | group **Edit** | **Text to columns**, you can split text that is in one worksheet cell into multiple worksheet columns. If, for example, a cell contains a sentence, this command will move each single word into its own column.

You apply this command as follows:

1. Navigate to the cell that contains the text to be distributed across columns.

(You can also select multiple cells or even an entire column.)

2. Choose the ribbon command **Data** | group **Edit** | **Text to columns** .
3. The program displays a dialog box with several options (see below).

   PlanMaker usually makes all the settings in it automatically, and you don't have to change anything.
4. Confirm with **OK**.

The text in the cell will then be distributed across multiple columns.

Options of the dialog box

Tip: The best settings for this dialog box are determined automatically upon analysis of the cell contents and usually do not need to be changed.

The dialog box for this command has the following options:

- **Target range**

  If desired, you can enter a different target range here. If you do so, the distributed text will then be placed in the specified target position rather than in its original position.

  You only have to specify the address of the cell in the top left corner of the target position.

- **Separator**

  If required, you can specify which character PlanMaker should consider as a text separator here. If, for example, the cell contains the sentence "This is a test", PlanMaker automatically selects a space as a separator and thus distributes each word to a column.

- **Text marker**

  If the text in the cell contains text markers, use this option to specify which character has been used (for example, quotation marks).

  Background info:

  The text export feature of some programs surrounds each data field with such a text marker in order to make it easy to determine where each field begins and ends.

Sorting cells

You can use the sort commands on the ribbon tab **Data** | group **Filter** to sort a cell range.

Tip: You will also find the commands on the ribbon tab **Home**| group **Contents** | **Sort and filter**.

Simple sorts: Quickly via the direct commands

If you only want to apply the sort to a certain column, you can do this directly via the commands **Sort ascending** and **Sort descending**.

You can also select a cell range across multiple columns; in this case, the left column of the selected range is sorted. The selected values to the right of it follow their left value in the same row.

Select the cell range you want to sort, and then on the ribbon tab **Data** | group **Filter**, choose one of the following commands:

- **Ascending**
  The data of the selected column is sorted in ascending order (A-Z).
- **Descending**
  The data of the selected column is sorted in descending order (Z-A).

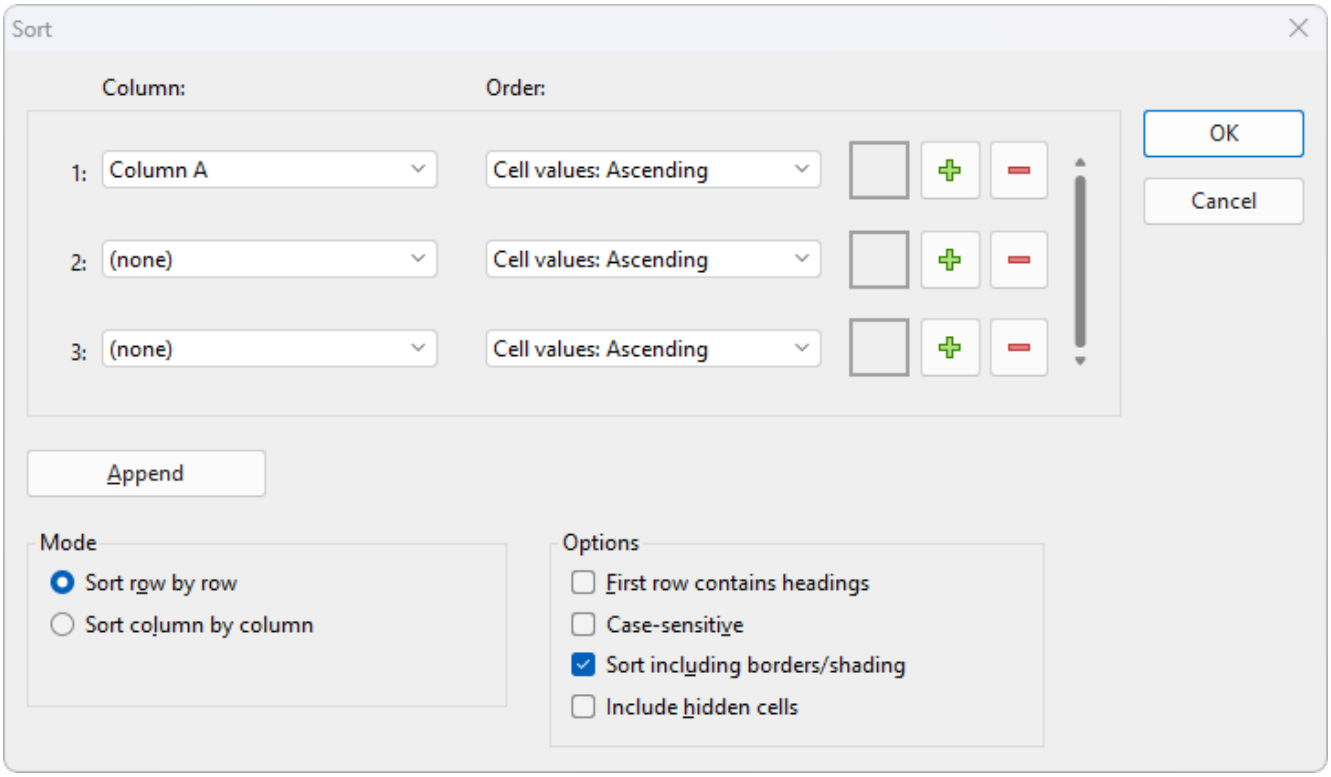
If, on the other hand, you want to apply different sort criteria to the columns of a selected cell range or select other user-defined options, choose the command **Sort** to open the dialog box (see below).

Sort with different criteria: Via the dialog box

You can choose the ribbon command **Sort** to open a dialog box with additional sort options.

Proceed as follows:

1. Select the cell range to be sorted.
2. Choose the ribbon command **Data**| group **Filter** | **Sort** .
3. PlanMaker opens the following dialog box:



4. For **1:**, select the **Column** by which you want to sort.

5. To the right of it, you can also change the **Order** of the sort: You can sort the cell values in **Ascending** (A to Z) or **Descending** order (Z to A).

   Note: If different shades or font colors have been applied to cells, they can also be sorted. See "Sorting cells by shading or font color" below.

6. If required, you can select additional columns by which to sort for **2:** and **3:**.

   If, for example, you select a column containing family names for **1:** and a column with first names for **2:**, the cell range will be sorted by the family name and then, in groups of identical family names, by the first name.

7. Make any further settings as required, see below.

As soon as you confirm with **OK**, the cell range will be sorted accordingly.

Options of the dialog box

The dialog box offers the following options:

- **Column** and **Order**

  Here, select the desired **Column**(s) by which to sort. You can also specify the order for each column: **Ascending** goes from A to Z, and **Descending** goes from Z to A for the cell values.

  Note: If different shades or font colors have been applied to cells, they can also be sorted. See "Sorting cells by shading or font color" below.

  Up to 3 columns can be specified by default. You can even add additional columns if you need more than 3 sort criteria. Up to 64 columns are possible. Proceed as follows:

  Add a column: Click on the **Plus** icon to insert another column.

  Append a column: Click on the **Append** button (below the list) to append a column at the bottom.

  Remove a column: Click on the **Minus** icon to remove the respective column. (This only works if there are more than 3 columns.)

- **Sort row by row** or **Sort column by column**

  These options determine whether PlanMaker sorts by row or by column.

- **First row contains headings**

  If the first row or column of the selected cell range contains a heading, you should enable this option. PlanMaker then omits it from the sorting.

  Example:

  You have selected a list of addresses that you want to sort by row. The first row of your selection contains headings such as "Name", "Street", "City", etc. The actual addresses are in the rows below. You should enable this option so that this row is not sorted and remains the first row.

- **Case-sensitive**

  If you enable this option, sorting distinguishes between uppercase and lowercase letters. For example, all words that begin with a lowercase letter end up in front of the words that begin with a capital letter:

  Disabled: Apples, bananas, Cherries. **Enabled:** bananas, Apples, Cherries.

- **Sort including borders/shading**

  If this option is enabled, cells retain their assigned borders and shading when moved by the sorting operation.

  If disabled, the selected cell range keeps its original formatting as far as borders and shading are concerned.

- **Include hidden cells**

  If the selected range contains cells that have been hidden (see Showing and hiding rows/columns), these are usually *not* sorted. Enable this option if you want hidden cells to be included in the sort.

Sorting cells by shading or font color

Not only can you sort a cell range by cell values in ascending/descending order, but you can also sort a range by cell colors (Shades) or text colors (Font colors), provided they have been applied to cells.

Note: The colors do not have their own defined order according to which they can be sorted automatically. Thus, you can decide for yourself which of the available colored cells you would like to move up or down within the cell range.

Proceed as follows:

1. Select the cell range and use the ribbon command **Data** | group **Filter** | **Sort** to open the dialog box as described above.
2. For **1:**, select the **Column** that you want to sort by color (for example, column A).
3. Under **Order** to the right, you can select whether you want to reorder cells according to their cell color or text color, and whether these cells should be sorted to the top or to the bottom.
4. Then click on the button with the black arrow (to the left of the plus and minus signs) and select the desired color there. Cells of this color are sorted to the very top/to the very bottom.

   Note: The button with the black arrow appears only if you have specified sorting by cell color/text color in **Order**. In addition, there must actually be cells in the selected **Column** that have the relevant colors.
5. If you specify the same column again for Column 2, Column 3, etc. (for example, column A), you can determine the next color to follow in that column. Make sure that the same sort criterion is set for **Order**.

   Otherwise, you can also select other columns for Column 2, Column 3, etc., to which you want to apply further sort criteria.

As soon as you confirm with **OK**, the cell range will be sorted accordingly.

Filtering cells by their content

You can select a cell range and then "filter" the data in it using a filter condition. PlanMaker will hide all rows that do not meet this condition.

To filter data, you can choose the ribbon tab **Data** | group **Filter** to access the following four commands:

- The command **AutoFilter** offers you different options for filtering rows: If you select a cell range and then choose this command, an arrow will be displayed from now on above each column of this cell range. If you click on one of these arrows, a menu will open with all the cell contents of this column – and various conditions by which you can filter.
- The command **Special filter** opens a dialog box in which you can formulate one or more filter conditions. After clicking on **OK**, the filter is enabled: All rows in the selected range that do not meet the filter condition are hidden.
- The command **Show all** makes all hidden rows visible again, thus eliminating the effect of both the *AutoFilter* and the *Special filter*.
- You can use the command **Reapply AutoFilter** to also update subsequently added values in the cell range of an active *AutoFilter* (not possible for *Special filter*).

Tip: You will also find these commands on the ribbon tab **Home** | group **Contents** | **Sort and filter**.

For more information on AutoFilter and Special filter, see the following pages.

AutoFilter

Use the command **AutoFilter** to filter the rows of a cell range:

If you select a cell range and choose the command **Data** | group **Sort and filter** | **AutoFilter**, an arrow will be displayed from now on above each column of this cell range. If you click on one of these arrows, a menu will open with all the cell contents of this column – and various conditions by which you can filter.

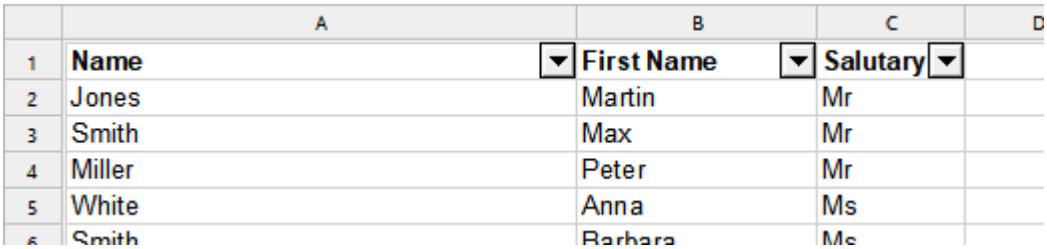
Note 1: The AutoFilter can only be inserted once per worksheet; two separate filters cannot be set on one worksheet. Otherwise, you cancel the previously applied filter by executing the AutoFilter command again. If you have created Tables in worksheets, they have their own AutoFilters – even on the same worksheet.

Note 2: Newly added or updated values are not automatically sorted by the previously set filter conditions. To integrate changed data into existing AutoFilters, use the command Reapply filter.

To apply the AutoFilter, proceed as follows:

1. Select the desired cell range. Important: The first row of this range must contain headings for the data below.
2. Choose the command **Data** | group **Filter** | **AutoFilter** .

The AutoFilter function is now enabled. You can tell by the fact that a downward arrow is displayed in the first row of each column in the selected range.



	A	B	C	D
1	**Name**	**First Name**	**Salutary**	
2	Jones	Martin	Mr	
3	Smith	Max	Mr	
4	Miller	Peter	Mr	
5	White	Anna	Ms	
6	Smith	Barbara	Ms	

Clicking on one of these arrows will open a menu which includes a list of all cell contents that occur in the corresponding column. By default, all values are selected in this list, thus there is no filtering.

To filter the data in the cell range, use this menu as follows:

- **Sort ascending** : This option sorts the filter results of the applied AutoFilter range in ascending order.
- **Sort descending** : This option sorts the filter results of the applied AutoFilter range in descending order.
- **Sort by color:** : This option sorts the filter results of the applied AutoFilter range by cell color or text color, if these were used in cells. In the submenu, select the cell color/text color of the cells to be sorted upwards. The last selected color appears at the top. Clicking on **More** opens a dialog box with differentiated settings for sorting. For more information, see Sorting cells.
- **Filter by color:** This option lets you filter a cell range by visual criteria (cell color/text color), if applied to cells.
- **More filters** (contextual): **Text filters, Number filters** and **Date filters** open a submenu with additional filters (see below).
- **(All):** This menu entry is a useful placeholder: It allows you to add/remove all values the column contains with just one click.

  The check mark in front of the entry indicates that all cell contents are currently included in the filter.

  If you now click on the entry **(All)**, all cell contents will be immediately removed from the filter (and the check mark will disappear).If you click on it again, all cell contents will be added to the filter again (and the check mark will reappear).

  If the filter does not contain the contents of all cells but contains those of at least one cell, a gray area will be displayed instead of the check mark.
- **(Blank):** If you have empty cell contents in your column, you can use this selection to show/hide all empty rows.
- **List of cell contents:** The most important part: All cell contents included in the column are listed here. By clicking on a value, you can add it to the filter (a check mark will appear in front of it) or remove it again by clicking on it again (the check mark will disappear).

Note: For the last three aforementioned options *(All)*, *(Blank)* and *List of cell contents*, please always note the following: To confirm your selection, you have to press **OK**.

Example

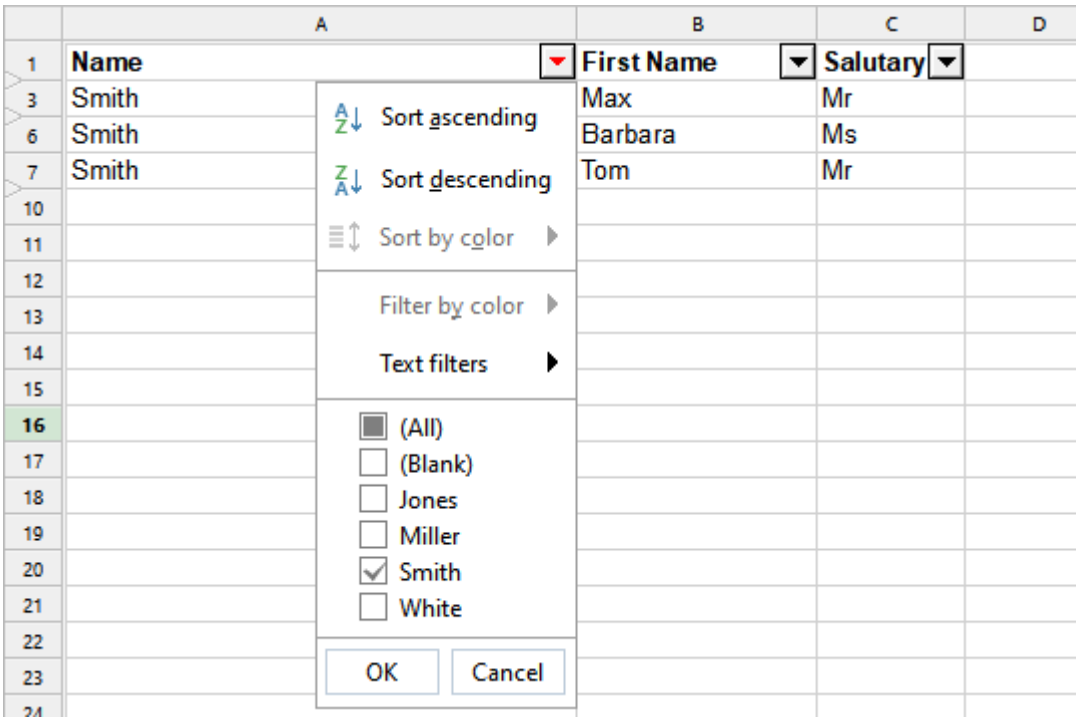
If, for example, you want to filter a cell range in such a manner that only those rows that contain the name "Smith" are shown in a column, proceed as follows:

Select the cell range (including the column headings) and choose the ribbon command **Data** | group **Filter** | **AutoFilter** to enable the AutoFilter.

An arrow now appears next to each column heading. Click on the arrow of the column heading "Name" to open the AutoFilter menu for this column.

First, click on the entry **(All)** in this menu in order to remove the check mark. This deselects all cell contents from the filter. Then, select the menu entry "Smith" and confirm with **OK**. You have now created a filter condition "Name equals Smith" using the AutoFilter function. PlanMaker will hide all other rows that do not match this condition.

The red color of the arrow indicates that an AutoFilter is applied.



Filter result after the name "Smith" has been selected in the AutoFilter list

If you would also like to see all Millers listed, simply select the menu entry "Miller". To remove the Millers again, click on the entry "Miller" once more to deselect it. Press **OK** to confirm in each case.

If necessary, apply additional filter conditions in the other columns also (e.g., "Ms" for all female matches of a given name) to narrow down the results even further.

As you can see, the values displayed in a column can be included or removed from the filter in any desired combination.

More filters: Text filters, Number filters, Date filters

Depending on the format category of the filtered columns, PlanMaker automatically sets additional filter options for *Text filter*, *Number filter* or *Date filter* in the AutoFilter menu. The *Text filter* is offered for text-only values and the *Date filter* for date-only values. If the formats are mixed, the *Number filter* is applied.

Use the offered filter in each case to obtain specialized filter conditions, for example:

Text filter

- **Equals...** Show only rows with exact matches.
- **Does not equal...** Hide rows with exact matches.
- **Greater than...** You will see the rows with the text values that are alphabetically after the filter value you entered.
- **Greater than or equal to...** See above, but including the entered value.
- **Less than...** You will see the rows with the text values that are alphabetically before the filter value you entered.
- **Less than or equal to...** See above, but including the entered value.
- **Starts with...** Show only rows with specific word beginnings.
- **Doesn't start with...** Hide rows with specific word beginnings.
- **Ends with...** Show only rows with specific word endings.
- **Doesn't end with...** Hide rows with specific word endings.
- **Contains...** Show only rows that contain specific strings as part of the text.
- **Doesn't contain...** Hide rows that contain specific strings as part of the text.

Number filter

- **Greater than...** This works like the operator > . You will see the rows with values that are greater than the filter value entered.
- **Greater than or equal to...** This works like the operator ≥ . You will see the rows with values that are greater than or equal to the filter value entered.
- **Less than...** This works like the operator < . You will see the rows with values that are less than the filter value you entered.
- **Less than or equal to...** This works like the operator ≤ . You will see rows with values that are less than or equal to the filter value you entered.
- **Between...** Here, define the range of numbers which contains the values of the rows that are to be displayed.
- **Not between...** Here, define the range of numbers which contains the values of the rows that are to be hidden.
- **Top 10...** Show only those rows where, for example, one of the 10 highest (or 10 lowest) values is in this column. You can customize this selection when the dialog box *Top 10...* has opened: In the left field, choose

between *Top* or *Bottom* values. In the middle field, you have the option of setting the number of top/bottom values. In the right field, you can choose between absolute values (*Items*) and relative values (*Percent*).

An example:
If you want to obtain 50% of the lowest values from 60 specified values, then set the following:
Left field: Lowest Middle field: 50 Right field: Percent

- **Only empty:** This option only displays rows where the value in this column is empty.
- **Non-empty:** This option only displays rows where the value in this column is not empty.
- **Above average:** This option only displays rows where the value is above the average of the values in this column.
- **Below average:** This option only displays rows where the value is below the average of the values in this column.

Date filter

- **Equals...** This displays only rows with exact date matches.
- **Does not equal...** This hides rows with exact date matches.
- **Before...** This displays only rows in which the date values are earlier than the entered date value.
- **Before or equal...** See above, but including the entered value.
- **After...** This displays only rows in which the date values are later than the entered date value.
- **After or equal...** See above, but including the entered value.
- **Between...** This displays rows where the date values are within a defined date range.
- **Not between...** This hides rows where the date values are within a defined date range.
- **Day, Week, Month, Quarter, Year:** Here, you can make further selections to quickly narrow down the desired date ranges.

  Note: If the AutoFilter has been applied to date values, you will notice in the arrow menu of the AutoFilter that the single days have already been sorted there at year and month level. Click on the plus sign in front of a year/month level to expand the level and view the single values grouped within it. If only single values are selected in a date level, a gray area is displayed in front of this date level (month/year) instead of a check mark. It is only when all existing values of a date level are selected that a check mark appears in front of this level. If no value of a date level is selected, you will see a white area in front of it. This allows you to always quickly see whether all, none or single values of a date level have been selected.

There are also the following options for each of the offered filter methods Text filters, Number filters or Date filters:

- **Custom filter:** This opens a dialog box in which you can define individual filter conditions.
- **Delete filter:** This option is only available if criteria have been set via the Text filter, Number filter or Date filter selection. Press *Delete filter* to remove only the criteria applied via these filters.

Showing all hidden rows again

If you want all rows that were hidden by AutoFilter to become visible again, choose the ribbon command **Data |** group **Filter | Show all**.

Reapplying AutoFilter

If values in the cell range set by AutoFilter have changed, you can use the ribbon command **Data** | group **Filter** | **Reapply AutoFilter** to update the selection you have already defined.

For example, you have specified that all rows with the name "Smith" should not be displayed and further entries with this name have been subsequently added. You can choose the command **Reapply AutoFilter** to filter out such subsequently created entries again without having to redefine the conditions of the filter.

Of course, this function is particularly useful for dynamic formula values and dynamic date values.

Disabling the AutoFilter

To completely disable the AutoFilter function, choose the ribbon command **Data** | group **Filter** | **AutoFilter** once again. The downward arrows displayed at the top of the cell range will disappear, and all cells will be displayed again.

Special filter

In addition to the **AutoFilter** introduced in the previous section, there are further options for filtering the data in a cell range in a much more complex manner based on one or more combinable conditions: with the ribbon command **Data** | group **Filter** | **Special filter**.

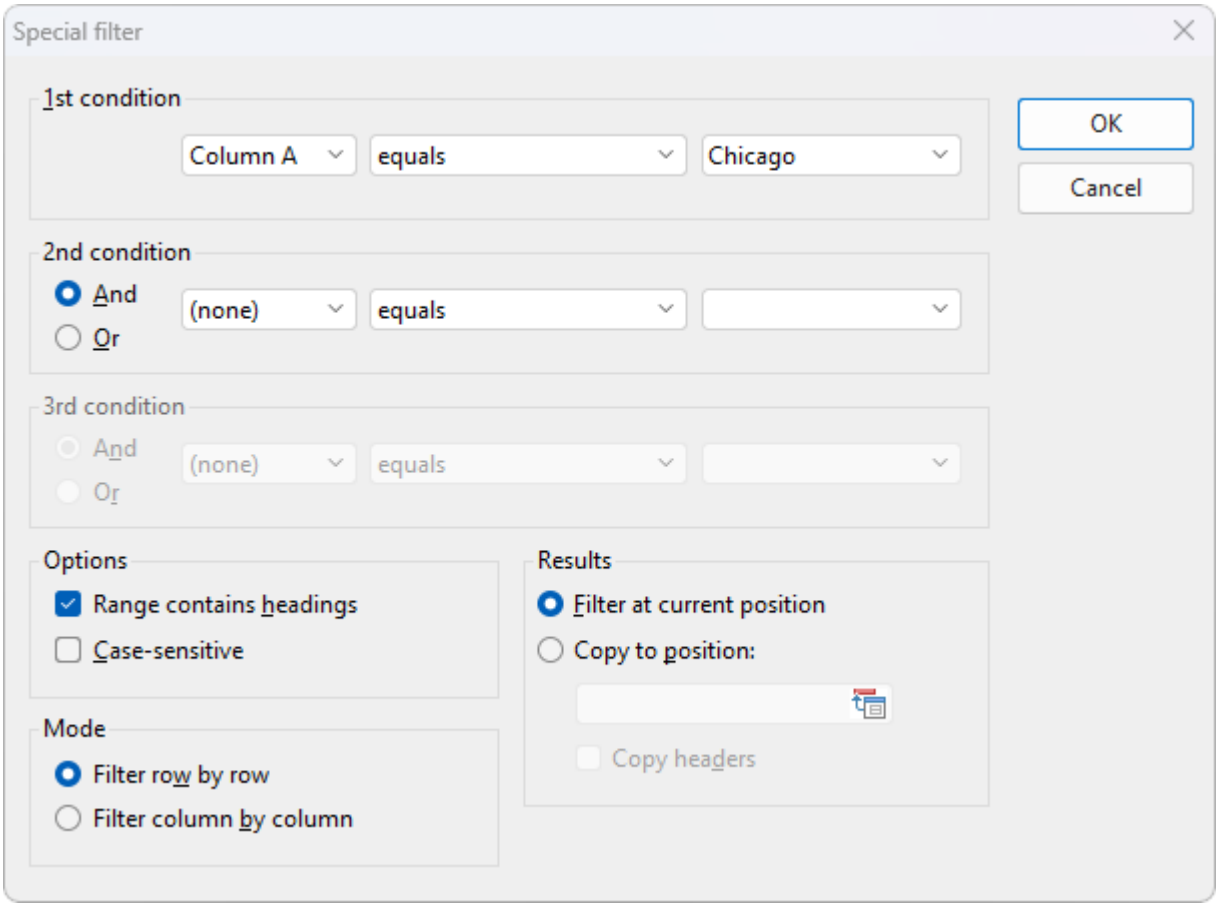
Proceed as follows:

1. Select the desired cell range.
2. Choose the command **Data** | group **Filter** | **Special filter** .
3. In the following dialog box, define one or more filter conditions (see below).
4. Confirm with **OK**.

All rows that do not match the filter conditions will now be hidden.

Formulating filter conditions

To specify one or more filter conditions in the dialog box of the command **Special filter**, proceed as follows:



In the **1st condition** section, first select the column on the left to which a condition is to be applied. In the middle, select the arithmetic operator. On the far right, enter the value against which you want to compare.

Some examples:

- The condition "Column D equals Chicago" only shows entries where column D contains the text "Chicago".
- The condition "Column E greater than or equal to 100000" filters out all entries for which column E contains a value less than 100000.

If one condition is not sufficient for formulating your filters, you have the option of linking up to three conditions together by also filling in the sections **2nd condition** and **3rd condition**.

Using "wildcard characters": In conditions, the characters ***** and **?** can be used as "wildcards" where ***** represents *any* number of arbitrary characters, and **?** represents a *single* arbitrary character. For example, "M*er" would return "Mister", "Miller", "Mary's mother", etc., whereas "?ouse" would return "mouse", "house", "louse", etc.

Options of the dialog box

The dialog box for the special filter offers the following options:

- **Range contains headings**

  If the first row and/or column of the selected cell range contains headings, enable this option. PlanMaker will then ignore this row or column when filtering.

- **Case-sensitive**

  If this option is enabled, PlanMaker will distinguish between uppercase and lowercase letters in conditions. For a condition such as "COLUMN A equals Smith", the condition will match only if the cell contains the text "Smith". Rows with "SMITH" and "smith" will not be included in the filtering results.

- **Mode**

  This determines whether rows or columns will be filtered.

  If you select the option **Filter row by row**, PlanMaker will filter out all *rows* that do not match the filter condition.

  If, on the other hand, you select the option **Filter column by column**, PlanMaker will filter out all *columns* that do not match the filter condition.

- **Results**

  This determines whether the filter will be applied to the original data or a copy of it:

  Filter at current position – Select this option and the original data will be filtered. Rows/columns that do not satisfy the filter condition will be hidden at the exact point where you set the filter.

  Copy to position – If you select this option instead, PlanMaker will create a copy of the original data at a cell address that you specify. This copy contains only the filtered data, and the original data remains unchanged in its place.

  In the input field under this option, enter where the copy should be inserted. You can either specify a single cell address (which will be the starting point of the output range) or the exact range of cells in which the copy should be placed. Copying to other worksheets is also possible. Caution: If the copy does not fit into this area, it will be truncated accordingly (exception: you specify a single cell address as the starting point).

Showing all hidden rows again

If you want all rows hidden by the filter to become visible again, choose the ribbon command **Data |** group **Filter | Show all**.

Analyzing sheets

PlanMaker provides several tools for analyzing cell contents and formulas in a worksheet. They can be very useful for troubleshooting.

For this purpose, the following tools are available:

- **Showing formulas instead of results**

  If you choose the ribbon command **View | Show formulas**, cells containing a calculation will display the formula rather than the result.

- **Syntax highlighting**

  If you choose the ribbon command **View | Syntax highlighting**, cells will be color-coded according to their content. For example, cells containing an error are displayed in red; cells containing a formula are displayed in green, etc.

- **Watch window for cell content**

  If you choose the ribbon command **Formula | Show watch list**, you can display a *watch window* for cell contents at any time. It allows you to observe the current values in the specified cells.

- **Formula auditing**

  The commands of the ribbon tab **Formula |** group **Traces** provide tools for graphically representing the relationships between cells. If, for example, the current cell contains a formula that refers to other cells, you can let PlanMaker display arrows pointing to these cells.

  Furthermore, errors can be detected in the worksheet or invalid data can be marked with red circles.

For more information on these functions, see the following pages.

Showing formulas instead of results

If you enable *Show formulas*, cells containing a calculation will display the formula rather than the result. This can be useful when analyzing the formulas in a worksheet and for troubleshooting.

Enabling and disabling "Show formulas"

To enable the "Show formulas" tool, choose the ribbon command **View | Show formulas** (in the group **View**).

If you choose this command again, the "Show formulas" tool will be disabled.

Syntax highlighting

If *Syntax highlighting* is enabled, all cell contents will be color-coded according to their content. For example, cells containing an error are displayed in red; cells containing a formula are displayed in green, etc.

This allows you to see at a glance where formulas are located and where fixed values are entered, where errors occur, etc.

Enabling and disabling syntax highlighting

To enable syntax highlighting, choose the ribbon command **View | Syntax highlighting** (in the group **View**).

If you choose this command again, it will be disabled.

Meaning of foreground color

When syntax highlighting is enabled, cell contents are displayed in the following foreground colors:

Cell content	Foreground color
Error value	Red
Formula	Green
Fixed positive number	Navy
Fixed negative number	Light blue
Fixed logical value	Brown
Fixed text string	Black

Meaning of background color

Cells that contain an *error* are thus displayed in a red foreground color. In the event of serious errors, the cell is also given a colored background. The background color indicates the error type:

Background color	Error type
White	**The formula returns an error value.** The calculation in this cell returns an error value because, for example, it contains a division by zero. **Tip:** If you click on the cell, an error message will be displayed in the status bar. For more information, see Error values. Correct the formula accordingly.
Cyan	**Syntax error in the formula.** The formula in this cell is incorrectly formulated because, for example, a closing parenthesis is missing or a function was not called with the correct arguments. **Tip:** If you click on the cell, an error message will be displayed in the status bar. Correct the formula accordingly. *If you don't correct the formula, it will be replaced by the text string #NULL! when the*

document is saved!

Light gray — **Non-exportable function.** The formula in this cell uses an arithmetic function that cannot be saved in the file format in which you saved the document.

Tip: Replace the function with another function that is compatible with the desired file format. For more information, see the corresponding function description in this manual.

Magenta — **Function possibly imported incorrectly.** The formula in this cell uses an older PlanMaker arithmetic function whose syntax or function might be incompatible with recent versions of PlanMaker. This error only occurs when importing PlanMaker documents with time functions.

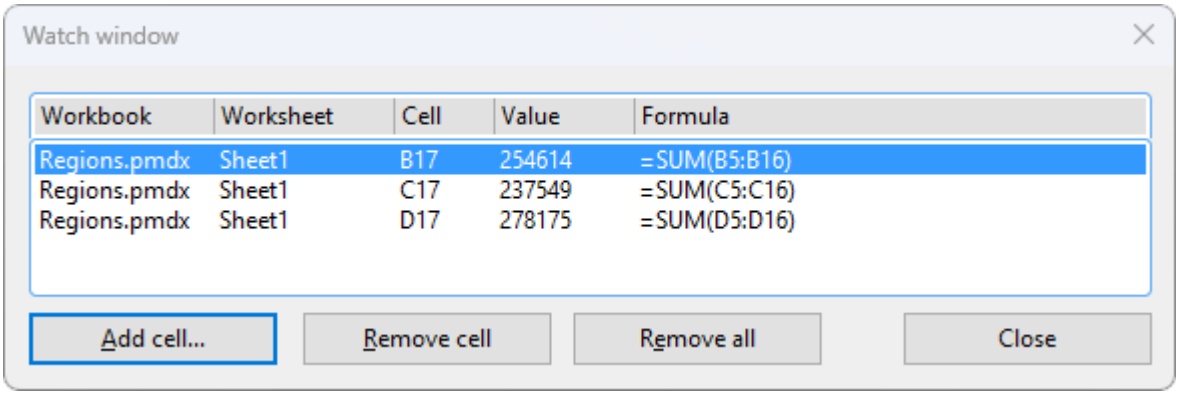
Tip: See the corresponding function description in this manual.

Yellow — **Unknown function.** The formula in this cell uses an arithmetic function unknown to PlanMaker. This error might occur when importing documents created with other applications (e.g. Microsoft Excel).

Tip: Replace the function with a corresponding PlanMaker function. For a list of all PlanMaker functions, see Functions from A to Z.

Watch window for cell content

You can display a *watch window* for cells at any time. It allows you to observe the current values in the specified cells.



Enabling and disabling the watch window

To display the watch window, choose the command **Show watch list** on the ribbon tab **Formula** | group **Watch**.

If you choose this command again, the window will disappear.

Adding cells to the watch window

To keep track of the value in a cell, add it to the watch window as follows:

1. In the watch window, click on the **Add cell** button.

2. Enter the address of the desired cell and confirm with **OK**.

   Tip: You can also enter an entire cell range here. If, for example, you enter A1:A3, the cells A1, A2 and A3 will be added.

The contents of the specified cells are now displayed in the watch window. Whenever the worksheet is recalculated, the watch window will also be updated.

Removing cells from the watch window

To remove a cell from the watch window, select it in the watch window and click on the **Remove cell** button.

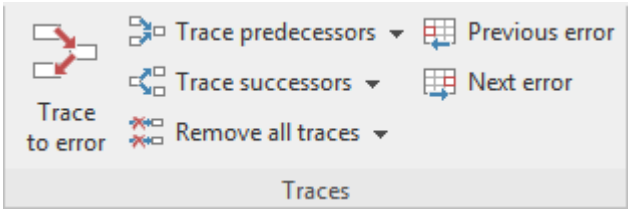
To remove all cells at once, use the **Remove all** button.

Formula auditing

The command group **Formula | Traces** and the command group **Review | Input validation** provide tools for graphically representing the relationships between cells.

If, for example, the current cell contains a formula that refers to other cells, you can let PlanMaker display arrows pointing to these cells. Furthermore, errors can be detected in the worksheet or invalid data can be displayed with red circles.

The command group **Formula | Traces** contains functions for the following topics:



- **Displaying the relationships between cells**

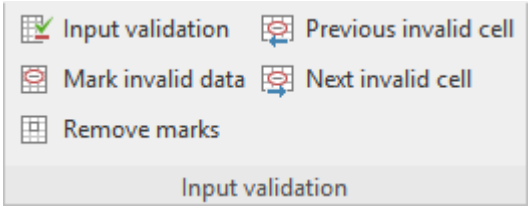
  PlanMaker has commands that can be used to graphically represent the relationships between cells. For example, PlanMaker can display arrows for all cells to which the formula in the current cell refers. Thus, if a cell contains the formula =A1*A2, arrows to the cells A1 and A2 will appear.

  Furthermore, there are commands for selecting all cells to which the formula in the current cell refers, and vice versa, i.e., for selecting all cells that refer to the current cell.

- **Detecting errors in calculations**

  You can also jump to the previous or next cell that contains an error value and display arrows to the source of the error.

You can choose the command group **Review | Input validation** to use the following functions:



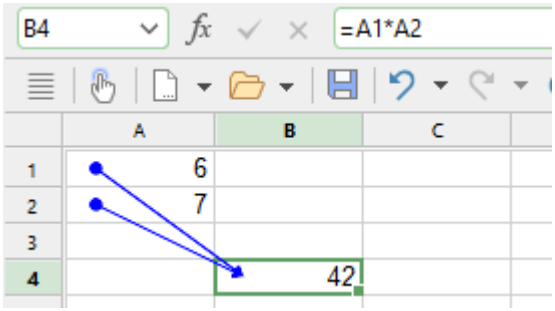
- **Detecting invalid data in cells**

  Here, you will find all cells whose content violates the input validation settings for the cell.

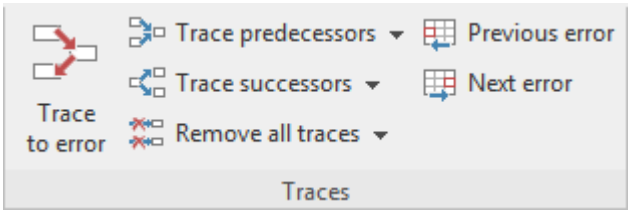
The following pages provide detailed information on these topics.

Displaying the relationships between cells

The command group **Formula | Traces** provides commands with which the relationships between cells can be graphically represented.



To do so, select the cell that you want to analyze, and then choose one of the following commands in the command group **Formula | Traces**:



Note: The commands **Trace to error**, **Previous error**, **Next error** are described in the next section Detecting errors in calculations.

- **Trace predecessors**

  This command displays arrows for all cells to which the formula in the current cell refers.

  Example: If the current cell contains the formula =A1*A2, this command will display arrows from cells A1 and A2 to the current cell (see figure above).

- **Trace predecessors | Select predecessors**

  This command selects all cells to which the formula in the current cell refers.

  Example: If the current cell contains the formula =A1*A2, this command will select cells A1 and A2.

- **Remove all traces | Remove traces to predecessors**

  This hides all arrows that were added with the command **Trace predecessors** for the current cell.

- **Trace successors**

  This displays an arrow to each cell that refers to the current cell.

  Example: If a cell contains the formula =B5*2, this command will display an arrow pointing to this cell when you choose the command in cell B5.

- **Trace successors | Select successors**

  This selects all cells that refer to the current cell.

  Example: If any cell contains the formula =B5*2, this cell will be selected when you choose the command in cell B5.

- **Remove all traces | Remove traces to successors**

  This hides all arrows that were added with the command **Trace successors** for the current cell.

- **Remove all traces**

  This removes all arrows added with the commands **Trace predecessors** or **Trace successors** for the entire workbook.

  Click directly on the icon itself or use the icon's arrow and select **Remove all traces** in the dropdown menu.

Notes and tips

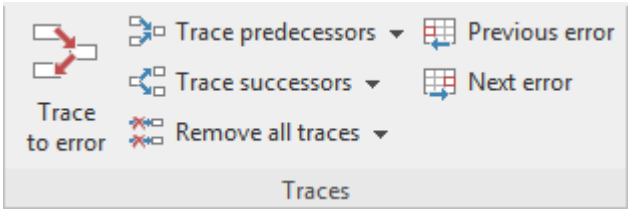
- If desired, you can choose the commands **Trace predecessors** and **Trace successors** several times in succession in a cell. Each call displays another level of relationships.
- The arrows that can be displayed with the above commands are *clickable*. If you click on such an arrow, PlanMaker will jump to the cell at the other end of the arrow.
- If the current cell refers to a cell that is in *another worksheet*, only a small arrow will be displayed in the top left corner of the cell. You can also click on this arrow to quickly jump to the corresponding cell.
- When you edit, delete or move the contents of a cell, all arrows to that cell are automatically removed.
- All displayed arrows also appear in the printout.
- However, the arrows are not saved in the document. Thus, when you save a document and then reopen it, all arrows are removed.

Detecting errors in calculations

The command group **Formula | Traces** provides commands for finding and analyzing cells that return an error value.

Note: A cell returns an *error value* when an error occurs in the calculation it performs. If, for example, a cell tries to perform a division by zero, the error value #DIV/0 will be displayed in the cell. For more information on error values, see Error values.

To find and analyze such errors, choose the command group **Formula | Traces**, and then use one of the following commands:



Note: The commands **Trace predecessors**, **Trace successors** and **Remove all traces** are described in the previous section Displaying the relationships between cells.

- **Trace to error**

  If the current cell contains an error value, this command will display arrows to the cells that cause the error value (if applicable).

  Example: If a cell returns a #DIV/0 error value and you choose this command in this cell, arrows will appear on the cells that perform this division by zero.

- **Previous error**

  This jumps to the previous cell that contains an error value.

- **Next error**

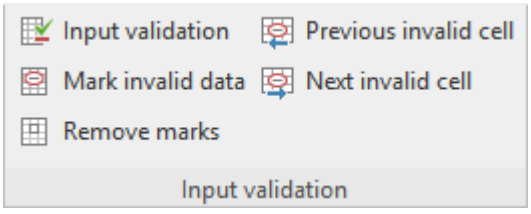
  This jumps to the next cell that contains an error value.

Detecting invalid data in cells

The command group **Review | Input validation** provides commands that can be used to detect cells that contain an invalid value.

Note: A cell contains an invalid value if the command **Input validation** has been used to specify that the cell can only contain certain values, and the current cell content does not match this check. For more information, see Input validation.

To search for invalid data, choose the command group **Review | Input validation** and then use the following commands:



- **Mark invalid data**

  This command inserts a red circle around each cell (in the current worksheet) that contains an invalid value.

- **Remove** (invalid data) **marks**

  This command removes all red circles (from the current worksheet) that were added with the above command.

- **Previous invalid cell**

  This command jumps to the previous cell that contains an invalid value.

- **Next invalid cell**

  This command jumps to the next cell that contains an invalid value.

Notes

- The marks for invalid data also appear in the printout.
- However, the marks are not saved in the document. Thus, when you save a document and then reopen it, all marks are removed.

Updating sheets

All calculations of all worksheets are *automatically* kept up to date. Whenever you change the content of a cell, PlanMaker recalculates the entire workbook.

Thus, you usually don't have to concern yourself with this issue at all.

Disabling automatic updating

If necessary, you can also disable the automatic updating of all worksheets. This can be useful if, for example, you want to edit a complex workbook on a slow device.

For this purpose, change the setting of the **Recalc automatically** option for the document properties accordingly.

1. Choose the ribbon tab **Formula** | group **Update** | **Recalculate** and click on the arrow of the icon to open its menu.

   Alternatively: On the ribbon tab **File** | group **File management** | **Recalculate** or also in the Document properties, Calculate tab.

2. Remove the check mark in front of **Recalc automatically** to disable it.

From now on, the **Recalc only before ...** setting is enabled:

There will be an automatic recalculation of the worksheets only when **saving** or **printing** the document, or when **copying** (or cutting) cells – depending on which option(s) you choose here. If all three options are disabled, calculations will no longer be updated automatically.

Starting recalculation manually

Even if you have disabled the automatic recalculation, you can still update the worksheets manually at any time. For this purpose, you can use the following keys or ribbon commands:

- If you press the **F9** key or choose the ribbon command **Recalculate** | **Recalculate workbook**, PlanMaker updates the calculations on *all* worksheets.

  Tip: This is also triggered if you simply click on the **Recalculate** icon directly.

- On the other hand, if you press **Shift+F9** or choose the ribbon command **Recalculate** | **Recalculate sheet**, PlanMaker only updates the calculations on the *current* worksheet.

  Note: The latter can return outdated results if the current worksheet performs calculations with cells in other worksheets.

Updating external cell references

Note that *external cell references* are not updated every time you recalculate the workbook.

(External cell references are cell references that refer to cells in another workbook – for example ='C:\My Folder\[My Workbook.pmdx]Sheet1'!A1.)

Such cell references are automatically updated only when you open the document that contains the external references. You can also manually update external references at any time.

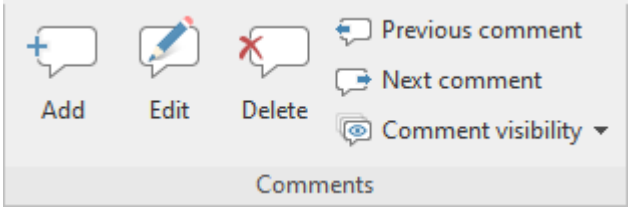
For more information on working with external references, see External cell references.

Additional notes

For information on updating *charts* or *pivot tables*, please see the relevant sections Updating charts and Updating pivot tables and changing data area.

Using comments

If you choose the command group **Comments** on the ribbon tab **Review**, you can add a comment to the current cell – for example: "The data from the previous year still has to be entered here!".



To work with comments, proceed as follows:

Adding comments to cells

1. Click on the desired cell to which you want to add a comment.
2. Then choose the ribbon command **Review** | group **Comments** | **Add** .
   Alternatively: **Insert** | **Comment**, or even faster with the key combination **Shift+F2**.

   Note: Comments appear with your name if you have specified it in the "Name" field for the **User info** (see Settings, General tab). If nothing is entered here, PlanMaker will now prompt you once for the current document.

   Tip: You can edit the displayed name in the comment window just like any other content in this window.
3. The program will display a yellow box in which you can type the comment. When you have finished typing, click outside the yellow box or press the **Esc** key.

Comments are saved in the document, thus they are available again when you reopen the document.

Viewing comments

Cells with an attached comment have a red triangle in their top right corner. If you hover your mouse over such a cell, the comment will be displayed. It will disappear if you move the mouse out of the cell.

Editing comments

To change a comment, select the cell that contains the comment.

Then choose the ribbon command **Review** | group **Comments** | **Edit** and make the desired changes in the yellow comment window.

Tip: Alternatively, double-click on the red triangle in the cell or press the key combination **Shift+F2** to edit a comment.

When you have finished making changes, click anywhere outside the comment or press the **Esc** key.

Deleting comments

To remove comments from cells, select the relevant cells (you can also select several cells at the same time).

Then choose the ribbon command **Review** | group **Comments** | **Delete** . The comments will then be deleted.

Alternatively, you can also choose the command **Home** | group **Contents** | **Delete** | **Comments**.

Finding comments

The command group **Review** | **Comments** contains two commands that you can use to jump back and forth between all comments in the document:

- **Previous comment** jumps to the previous comment
- **Next comment** jumps to the next comment

Visually customizing the comment window

- **Resize:** Use the round "handles" on the frame of the comment window in order to resize it. When you drag one of these handles with the mouse, the size of the object is changed accordingly.
- **Change position:** Move the comment window by clicking with the mouse on its frame and dragging it to the desired position while holding down the mouse button.

Tip: The comment window can be customized like any other object with different colors, shades, etc. If you click on the frame of the comment window, you can edit it similar to a text frame using the contextual ribbon tab **Object** or the dialog box *Object properties*. For more information, see Text frames.

Comment visibility

You usually only see the comment window when you point to the relevant cell.

Thus, use the **Comment visibility** button on the ribbon tab **Review** | group **Comments** if you want to display comments permanently.

Note: Comment visibility also affects whether comments appear in the printout. Only the permanently displayed comments are printed if the option "Print: Comments" is selected. This option can be found in the dialog box of the command **File** | group **Print** | **Page setup** on the **Options** tab. For more information, see Setting page format options.

The following commands are available in the dropdown menu of the **Comment visibility** button:

- **Show all comments**

  If you have added comments to several cells, all comment windows will be displayed (including on other worksheets in the document). If you click on the command again, they will be hidden again.

  This option can also be found in the Document properties, Options tab as **Always show comments**.

- **Show this comment**

  Only the comment of the selected cell will be shown or hidden.

The **Comment visibility** button also has the following command:

- **Restore comment positions**

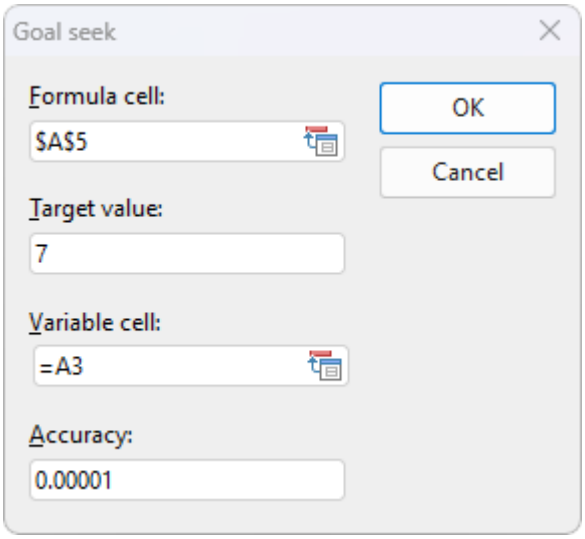
  If the position of one or more comment windows has been moved (by moving them with the mouse or by rearranging the cells), you can use this command to reset the comment windows to their original position. To do so, select the cells whose comment windows you want to move back to their original position.

Goal seek

The ribbon command **Data** | group **Analyze** | **Goal seek** can help you to determine what value a certain component of a calculation must assume in order to obtain a predefined result from the calculation.

Note: This is done by mathematical approximation, and as with all approximation methods, the result does not necessarily have to be exact.

Before you choose this command, navigate to the cell that contains the calculation.



Upon choosing the command, a dialog box appears. This is used as follows:

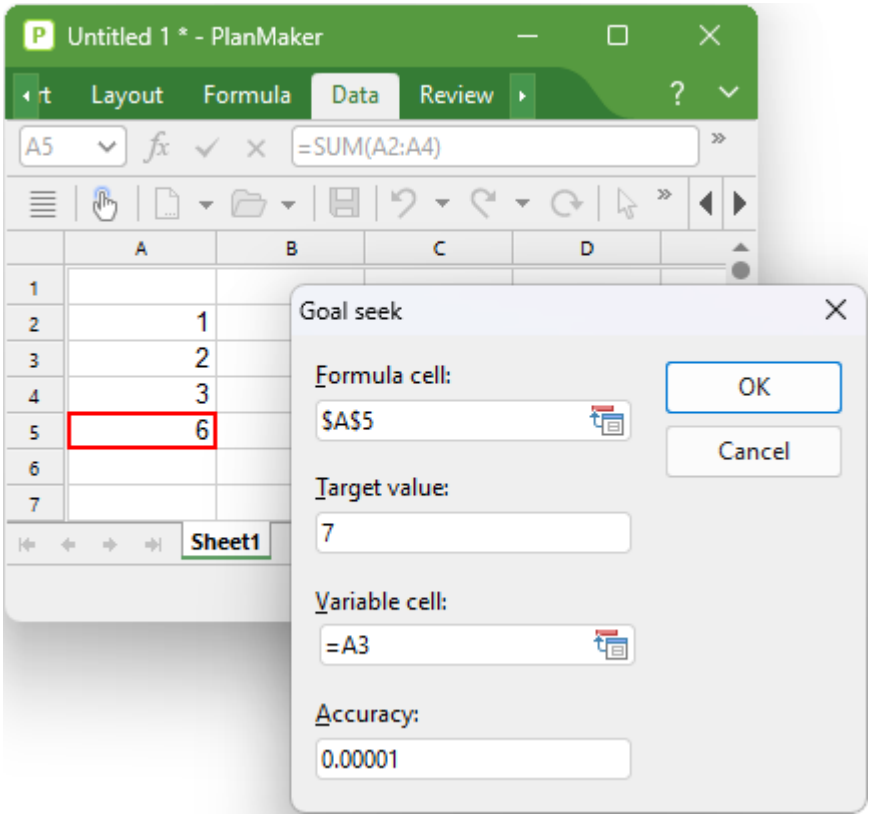
- In **Formula cell**, enter the address of the cell that contains the formula. This is generally preset by PlanMaker.
- In **Target value**, enter the desired result of the calculation.
- In **Variable cell**, enter the cell address of the cell in which PlanMaker will test different values until the formula cell returns the desired result.
- In **Accuracy**, you can set the accuracy of the approximation. PlanMaker will attempt to approximate values until the difference to the target value is less than the parameter **Accuracy**.

If PlanMaker is unable to find a sufficiently close result after a few seconds, it will determine that the value cannot be calculated through approximation and will abort the search.

An example

In the worksheet shown below, the SUM function has been used in cell A5 to add the numbers 1, 2 and 3 in the cells A2, A3 and A4. The result in cell A5 is 6. We now want to know to what value cell A3 must be changed so that the result of the addition is 7.

For this purpose, move the cell frame to the cell with the calculation whose result is to be changed – in this case, cell A5 with the SUM formula – and apply the goal seek.



The figure illustrates how to fill in the values in the dialog box. When you click on **OK**, PlanMaker starts the approximation process and displays result 3 in a dialog box.

This means that the content of cell A3 must be changed from 2 to 3 so that the calculation has the result 7. If you click on **Use** in the dialog box, the value found will be applied to cell A3.

Scenarios

In the iOS version, the command is locked. (To unlock the command, see Welcome.)

The ribbon command **Data** | group **Analyze** | **Scenario manager** allows you to create and display "scenarios".

Scenarios can be used to observe how changing the values in specific cells affects the calculations in a worksheet. This enables you to perform all kinds of "what-if" analyses.

Example: How does the total cost of a loan change if the term is 24, 36 or 96 months?

In this section, you will learn everything you need to know about scenarios. The following topics are covered:

- Creating scenarios
- Showing scenarios
- Managing scenarios
- Merging scenarios
- Creating a scenario summary

For more information, see the following pages.

Creating scenarios

To create scenarios, first complete a worksheet with the desired values and calculations.

Then proceed as follows:

1. Choose the ribbon command **Data** | group **Analyze** | **Scenario manager** .
2. Click on **Add** in the dialog box that opens.
3. Under **Scenario name**, enter a name that describes the purpose of the scenario.
4. Under **Changing cells**, enter the address of the cell whose content is to be changed in the scenario.

   You can also specify multiple cells. Separate the cell addresses with a comma – for example: A2,A4,A10. Cell ranges (like B5:B10) are also allowed.
5. Click on the **Next** button to continue.
6. Another dialog box appears. There, enter the value that is to be inserted into the "changing cell" in this scenario.

   If you have defined multiple changing cells in the scenario, this dialog box displays a list of all these cells. Click on each list entry and enter the desired value next to it.
7. Click on **Finish** to complete the creation of this scenario.

8. Repeat steps 2. to 7. for all other scenarios that you want to create.
9. When you have finished, exit the dialog box with the **Close** button.

The desired scenarios have now been created. For information on how to display them, see the next section.

Options of the dialog box

The dialog box for adding scenarios offers the following additional options:

- **Comment**

  Here, you can enter a descriptive comment for the scenario.

- **Prevent changes**

  If this option is enabled, the scenario cannot be edited or deleted if you enable sheet protection.

- **Hide**

  If this option is enabled, the scenario is no longer displayed in the list of scenarios when you enable sheet protection.

For more information on using sheet protection, see Sheet protection.

Showing scenarios

If you have created scenarios as described in the previous section, you can display them at any time:

Proceed as follows:

1. Choose the ribbon command **Data** | group **Analyze** | **Scenario manager** .
2. In the dialog box, select the scenario to be displayed from the **Scenarios** list.
3. Click on the **Show** button.

> **Tip:** You can do so even quicker if you simply double-click on the scenario in the list.

The selected scenario will then be displayed.

By switching between different scenarios, you can now compare how changing the cells specified in the scenario affects the calculations that refer to those cells.

Note: When you display a scenario, the values in the specified "changing cells" are physically replaced by the values set in the scenario. The old cell contents will thus be lost.

Managing scenarios

The command **Data** | group **Analyze** | **Scenario manager** can also be used to manage the scenarios defined in the current worksheet.

To do so, use the following buttons in the dialog box for this command:

- **Add**

  This adds a new scenario.

  For more information, see Creating scenarios.

- **Delete**

  This deletes the scenario currently selected in the list.

- **Edit**

  This opens a dialog box in which you can edit the selected scenario. If you click on **Next**, the program will display another dialog box in which you can change the values for the changeable cells.

- **Merge**

  This merges scenarios that have been defined in other worksheets (or workbooks) into the current worksheet.

  For more information, see Merging scenarios.

- **Summary**

  This creates a scenario summary that directly compares the values in the changeable cells and the values in any "result cells" for all scenarios.

  For more information, see Creating a scenario summary.

Merging scenarios

The command **Data** | group **Analyze** | **Scenario manager** can also be used to merge scenarios that have been defined in other worksheets (or even other workbooks) into the current worksheet.

Note: When using this function, ensure that all workbooks whose scenarios are to be merged have exactly the same structure and use the same changing cells in their scenarios.

To merge scenarios, proceed as follows:

1. Open the workbooks that contain the scenarios to be merged.
2. Switch to the workbook and worksheet into which you want to merge the scenarios.

3. Choose the ribbon command **Data** | group **Analyze** | **Scenario manager** .
4. Click on **Merge** in the dialog box.
5. Select the workbook and worksheet that contain the scenario(s) to be merged.
6. Confirm with **OK**.
7. Repeat steps 4. to 6. for all other scenarios that you want to merge.

The selected scenarios are now available in the current worksheet.

Creating a scenario summary

The command **Data** | group **Analyze** | **Scenario manager** can also be used to create a *scenario summary* for all scenarios defined in the current worksheet.

A scenario summary lists all scenarios clearly next to each other. For each scenario, both the changing cells and any "result cells" (containing the calculations to be observed) are displayed.

Scenario summaries are useful for comparing the changing values and the results of scenarios at a glance.

To create a scenario summary, proceed as follows:

1. Create the desired scenarios (as described in Creating scenarios).
2. Choose the ribbon command **Data** | group **Analyze** | **Scenario manager** .
3. Click on **Summary**.
4. Enter the address of the desired *result cell*. This is the cell which contains the calculation that you want to observe in your scenarios.

   You can also specify multiple cells. Separate the cell addresses with a comma – for example: A2,A4,A10. Cell ranges (like B5:B10) are also allowed.
5. Confirm with **OK**.

PlanMaker will now create a new worksheet and generate the scenario summary in it.

Note: Scenario summaries are *not* automatically updated when you change cell contents or scenario settings. To manually update a scenario summary, delete the worksheet containing the summary and then generate the summary again.

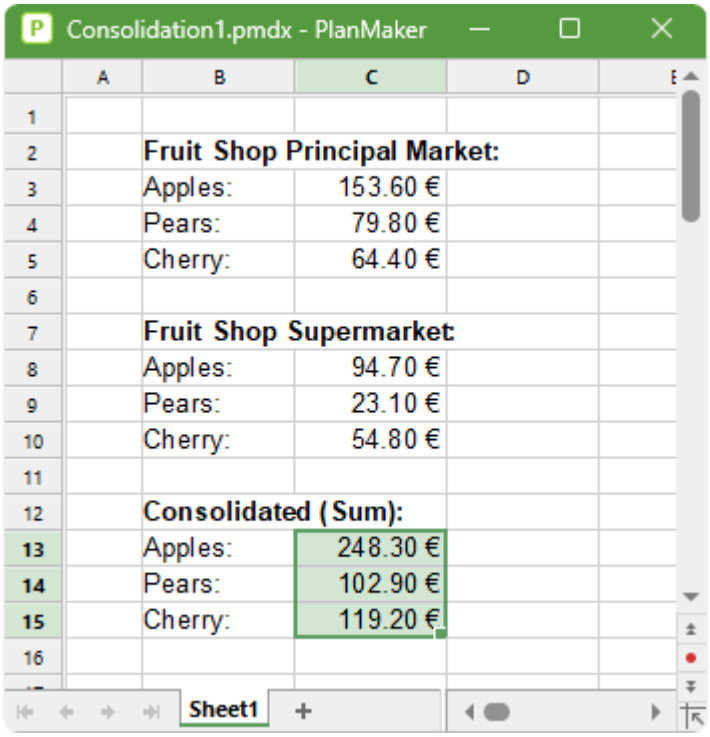
Consolidating data

In the iOS version, the command is locked. (To unlock the command, see Welcome.)

You can use the ribbon command **Data** | group **Analyze** | **Data consolidation** to evaluate data from one or more cell ranges, for example, in order to calculate their total sums.

To do so, you first define the "source ranges" for consolidation. They can be any cell ranges with the data to be evaluated. They can all be in the current worksheet or spread across multiple worksheets or even multiple files. For the evaluation, you can choose from among several statistical functions, such as sum, minimum, average, etc.

The result of the evaluation is inserted – as fixed values – into the specified "target range". Note: The results will not be updated if you change the values in any of the source ranges. This command is thus primarily useful if you want to evaluate the *current* state of data and no longer want to take changes into account (useful, for example, for monthly reports).



In this section, you will learn everything you need to know about working with consolidations. The following topics are covered:

- **Consolidating data by position**

  In the simplest case, data is consolidated by its *position*. This works as follows:

  You have entered the data into, for example, three source ranges which should be *identical* in size and structure. In each of the ranges, the individual data is thus in the same (relative) position.

When consolidating, the first cell (top left) in the first range is then added to the first cell in the second and third ranges. The result appears as the first cell in the target range. The same is then done with all other cells in the source ranges.

- **Consolidating data by labels**

  Furthermore, this function can also consolidate by *labels*. This works as follows:

  There may be one or more source ranges that have one thing in common: Each individual value has a *label* – for example, in the cell to the left of it.

  When you start a consolidation with such source ranges and enable the option **Labels in left column**, PlanMaker calculates the sum of all values that have the same label on their left.

  The same label may occur as often as desired. The size of the source ranges and the order of data are also irrelevant. PlanMaker only uses the labels to determine which values are to be added.

- **Editing and updating consolidations**

  Whenever you perform a consolidation using the ribbon command **Data** | group **Analyze** | **Data consolidation**, PlanMaker remembers each setting that you made in the dialog box "Consolidate" (separately for each worksheet).

  This enables you to choose this command again at any time, change some settings if required, and then perform the consolidation again.

For more information on these topics in the above order, see the following pages.

Consolidating data by position

As mentioned in the introduction of the section Consolidating data, the command **Data consolidation** allows you to *consolidate* data from one or more cell ranges, for example, in order to calculate their total sums.

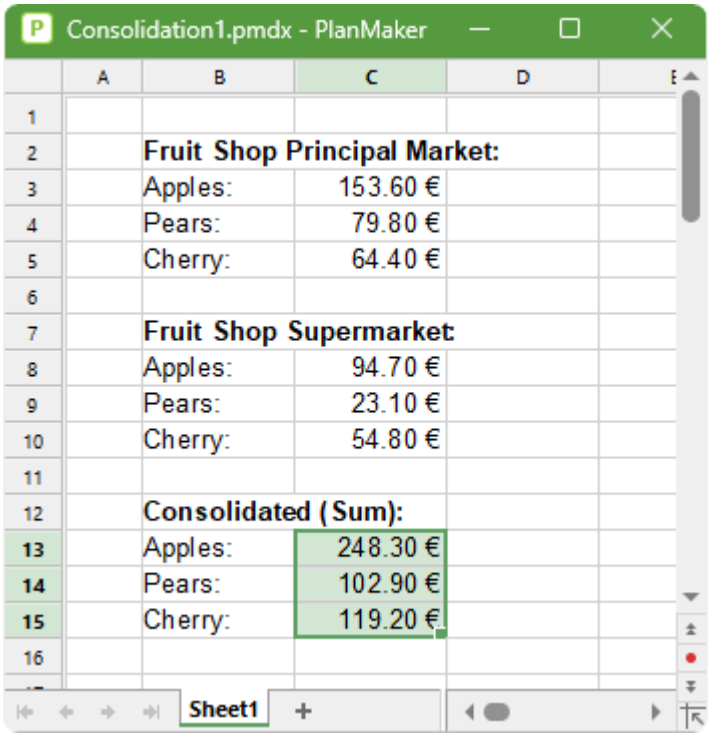
In the simplest case, data is consolidated by its *position*. This works as follows:

You have entered the data into, for example, three "source ranges" which should be identical in size and structure. In each of the ranges, the individual data is thus in the same (relative) position.

When consolidating, the first cell (top left) in the first range is then added to the first cell in the second and third ranges. The result appears as the first cell in the "target range". The same is then done with all other cells in the source ranges.

Example

The daily revenues of two fruit shops are to be consolidated by means of a sum function, divided according to fruit varieties.



For this purpose, the revenues of the two shops were entered into a worksheet. In the figure, this data is shown in the cell ranges C3:C5 and C8:C10.

The command **Data consolidation** was then chosen and these two ranges were added as source ranges. The target range was set to C13:C15 (selected in the figure) and consolidation was started.

Result: In the target range, the sums of the daily revenues appear (again per fruit variety, just as in the source ranges).

Performing a consolidation by position

To consolidate data by position, proceed as follows:

1. Enter the data to be consolidated into individual cell ranges of exactly the same size and structure. The order of the individual pieces of data must be identical in each range.

   The individual cell ranges can all be in the same worksheet, in multiple worksheets or even in several different documents.

2. Choose the ribbon command **Data** | group **Analyze** | **Data consolidation** .

3. Click into the **Source ranges** input field. There, enter the address of the first cell range with the data to be consolidated. (See also notes below.)

   Tip: Alternatively, you can also click directly into the worksheet while the dialog box is open and simply select the cell range in the worksheet with the mouse to transfer it to the dialog box.

4. Click on the **Add** button.

5. To add additional source ranges, repeat steps 3. and 4.

6. For **Target range**, enter the address of the cell range in which you want the result of the consolidation to be inserted.

   Tip: It is sufficient to specify the address of the cell in the top left corner of the target range. PlanMaker will then determine its size automatically.

   Tip: You can also simply click on the desired cell directly in the worksheet to transfer its address to the dialog box.

7. For **Function**, select the arithmetic function to be used for the consolidation.

8. The options **Labels in ...** must be *disabled* for this type of consolidation. For more information, see Consolidating data by labels.

9. Click on **Apply** to start the consolidation.

The data from the source ranges is now consolidated using the selected arithmetic function, and the result is inserted in the target range.

Note: The result of a consolidation is inserted into the worksheet as *fixed* numbers. These numbers will *not* be updated if you change the values in any of the source ranges.

This command is thus primarily useful if you want to evaluate the *current* state of data and no longer want to take changes into account (useful, for example, for monthly reports). For more information, see also Editing and updating consolidations.

Notes on specifying the source ranges

If you click in the **Source ranges** input field in the dialog box described above, a source range can be added using the following options:

- **Source range from the current worksheet**

  To add a cell range that is located in the current worksheet to the source ranges, simply enter its address or name.

  Example: D2:G5

  Tip: Alternatively, you can also click directly into the worksheet while the dialog box is open and simply select the cell range in the worksheet with the mouse.

- **Source range from a different worksheet**

  To add a cell range that is located in a different worksheet, enter its address preceded as usual by the other worksheet's name and an exclamation mark.

  Example: Sheet3!D2:G5

  Tip: You can also simply select the cell range with your mouse. First, click on the desired worksheet in the worksheet register.

- **Source range from a different document**

  To add a cell range that is located in a different document, enter its address as described in External cell references).

  Example: 'C:\My Folder\[My Workbook.pmdx]Sheet3'!D2:G5

Tip: You don't have to manually enter the first part of the address (folder and file name). Simply click on the **Browse** button in the dialog box, and the program will then display a file dialog box in which you can select the desired file.

Don't forget to click on the **Add** button after entering the address for each source range.

Consolidating data by labels

As mentioned in the introduction of the section Consolidating data, the command **Data consolidation** allows you to *consolidate* data from one or more cell ranges, for example, in order to calculate their total sums.

In addition to consolidating data by its position (see previous section), data can also be consolidated by its *labels*. This works as follows:

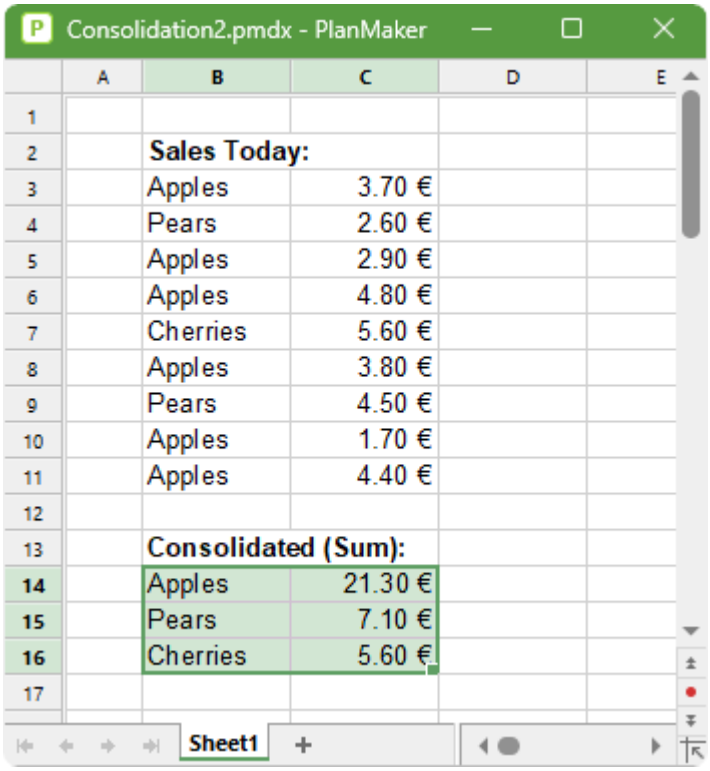
The data to be evaluated can be stored in any number of "source ranges" – all of which should have one thing in common: a label has been added to each value (for example, in the cell to the left of the value).

When you start a consolidation with such source ranges and enable the option **Labels in left column**, PlanMaker calculates the sum of all values that have the *same* label on their left.

The same label may occur as often as desired. The size of the individual source ranges and the order of the data within them are irrelevant. PlanMaker only uses the labels to determine which data is to be consolidated.

Example

A fruit shop offers different types of fruit. Each individual sale during the course of a day is recorded in a worksheet. You now want to calculate how much of each type of fruit was sold in total.



Of course, the individual types of fruit occur in a completely random order, but this is irrelevant to the consolidate command:

Simply choose the command **Data consolidation**. Add the cell range with the individual sales as the source range (B3:C11 in this case). Note: This must contain the numbers *and* their labels! Then select any target range for the result (B14:C16 in the figure) and confirm.

Result: The totals of the sales of the individual types of fruit appear in the target range.

Performing a consolidation by labels

To consolidate data by its labels, proceed as follows:

1. Enter the data to be consolidated into one or more cell ranges. The size and structure of these cell ranges don't matter. However, the values to be consolidated should all have a label – either in the column to the left of them and/or in the row above them.

   The individual cell ranges can all be in the same worksheet, in multiple worksheets or even in several different documents.

2. Choose the ribbon command **Data** | group **Analyze** | **Data consolidation** .

3. Click into the **Source ranges** input field. There, enter the address of the cell range with the data to be consolidated. See also notes at the end of the previous section.

   Tip: Alternatively, you can also click directly into the worksheet while the dialog box is open and simply select the cell range in the worksheet with the mouse to transfer it to the dialog box.

> **Important:** Each source range must contain both the values themselves and their *labels*. The labels must be in the far left column or in the top row of the range.

4. Click on the **Add** button.
5. To add additional source ranges, repeat steps 3. and 4.
6. For **Target range**, enter the address of the cell range in which you want the result of the consolidation to be inserted.

   Tip: It is sufficient to specify the address of the cell in the top left corner of the target range. PlanMaker will then determine its size automatically.

   Tip: You can also simply click on the desired cell directly in the worksheet to transfer its address to the dialog box.
7. For **Function**, select the arithmetic function to be used for the consolidation.
8. For **Options**, specify the position of the labels in the source ranges:

   Labels in left column: The labels are stored in the far left column of each source range. (The associated data must then be entered directly to the right of it.)

   Labels in top row: The labels are stored in the top row of each source range. (The associated data must then be entered directly below.)

   You can also enable both options if you want to evaluate source ranges where labels are in their far left column *and* in their top row.

   If you enable the option **Sort labels**, the results of the consolidation in the target range will be sorted according to the labels.
9. Click on **Apply** to start the consolidation.

The data from the source ranges is now consolidated using the selected arithmetic function, and the result is inserted in the target range.

Note: The result of a consolidation is inserted into the worksheet as *fixed* numbers. These numbers will *not* be updated if you change the values in any of the source ranges.

This command is thus primarily useful if you want to evaluate the *current* state of data and no longer want to take changes into account (useful, for example, for monthly reports). For more information, see also Editing and updating consolidations.

Editing and updating consolidations

Whenever you perform a consolidation using the command **Data consolidation**, PlanMaker remembers each setting that you made in the dialog box "Consolidate" (separately for each worksheet).

This enables you to choose this command again at any time, change some settings if required, and then perform the consolidation again.

Tip: The settings are even saved in the document, so PlanMaker will remember them the next time you open the document. (Small restriction: The **Target range** is not saved in *xls files* because this file format does not support it. However, it works for *xlsx* files, *pmd* files and *pmdx* files.)

Changing settings and performing the consolidation again

To change the settings of a consolidation and then reinsert its updated result, proceed as follows:

1. Choose the ribbon command **Data** | group **Analyze** | **Data consolidation** .
2. Make the desired changes to the consolidation settings. You can even add new source ranges or remove existing ones.
3. Click on **Apply** to apply the changes.

The consolidation in the current worksheet will then be performed again, and its updated result will be inserted into the target range. Existing data will be overwritten.

Updating the results of a consolidation

If you just want to update the result of a consolidation (without changing any settings), proceed as follows:

1. Choose the ribbon command **Data** | group **Analyze** | **Data consolidation** .
2. Click on **Apply**.

The consolidation in the current worksheet will then be performed again with exactly the same settings as the last time.

Tables in worksheets

The ribbon command **Insert | Table** allows you to create "tables" in a worksheet.

Note: The fact that this feature is named "table" may be a bit confusing, since the term "table" is often used to represent the content of a worksheet in general. However, since Microsoft has called this feature a "table" in Excel, we decided to keep this term in PlanMaker for compatibility reasons.

	A	B	C	D	E
1	Type	Pieces	Price/unit	Price	
2	Apples	4	0.50 €	2.00 €	
3	Oranges	2	0.75 €	1.50 €	
4	Bananas	3	1.00 €	3.00 €	
5	Pears	2	0.75 €	1.50 €	
6	**Total**			**8.00 €**	
7					
8					

How exactly does this feature work?

When you select a cell range and choose the ribbon command **Insert | Table**, PlanMaker will create a "table" from it.

This has the following effects on the cell range:

- **Automatic formatting with a "table style"**

  The selected cell range is formatted automatically with a *table style*. The default table style alternately applies blue and white shading to the cells. However, you can also use a different style or subsequently change the style and thus change the formatting of the entire "table" at the touch of a button.

- **AutoFilter applied automatically**

  In addition, an AutoFilter is applied to the cell range that was used to create the "table". The AutoFilter adds little arrows to the right of each of the column headings in the table. If you click on one of these arrows, the program will open a menu in which you can set a filter on the cell contents of the corresponding column.

- **Total row**

  In "tables", you can easily activate a *total row* that is displayed below the table. The total row can display calculations that sum up the values contained in the table (for example, the sum of each column).

- **Special cell references**

  When you have created a "table" in a worksheet, you can also use special cell references. For example, the cell contents in a column of a table can be addressed via its column heading – as in:

  =SUM(Table3[Spring])

The following pages contain everything you need to know about the "table" feature. The following topics are covered:

- Creating tables in worksheets
- Ribbon tab "Table"
- Removing tables from worksheets
- Editing tables in worksheets
- Formatting tables in worksheets
- Using the total row of tables in worksheets
- Tables in worksheets and cell references

Creating tables in worksheets

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to have "tables in worksheets". Here, you will learn how to create such a table:

To create a "table in a worksheet", proceed as follows:

1. Optional: In a range of adjacent cells, enter the data from which you want to create a table.

   Tip: It is useful if you have provided this cell range with a *column heading* in the first row of each column. (If there are no column headings, PlanMaker will generate automatic headings named *Column1*, *Column2*, etc.)

2. Select the cell range (including the column headings).
3. Choose the ribbon command **Insert | Table** by clicking directly on the icon itself.

   Tip: If you click on the icon's arrow rather than directly on the icon itself, PlanMaker will open a list of predefined styles which can be applied directly.

4. The program displays a dialog box with the following options:

   Source range: This is the cell range from which the table is to be created. The currently selected range is suggested by default. As a rule, you only have to confirm this default range.

   Source range contains titles: Here, you can specify whether the first row of the source range consists of column headings, as recommended above. When you disable this option, PlanMaker will generate automatic column headings named *Column1*, *Column2*, etc., and indents these headings with an additional row above the source range.

As soon as you confirm your selection in the dialog box, a "table" is created for the source range. More specifically, the following happens:

- **No changes are made to the table cells and their contents**

  The cells in the source range remain normal table cells even after a table has been created. Thus, you can still edit them, format them, etc.

- **Automatic formatting with a "table style"**

  However, it is immediately noticeable that the cells in the source range have been automatically reformatted: By default, the first row (containing the column headings) is shaded in a medium blue, the rows below are shaded alternately in light blue and white (unless you have selected a different style). For more information, see tip from step 3.

  This automatic formatting is performed via a *table style*. Whenever you create a table, it is automatically reformatted with the default table style. Alternatively, numerous other table styles with different colors and styles are available. On the contextual ribbon tab **Table**, you can switch to another style at any time using the commands **Table styles** or **Properties**.

- **AutoFilter applied automatically**

  There is a small arrow to the right of each column heading in the first table row. These arrows indicate the presence of an *AutoFilter* that has been automatically applied to the source range of your table.

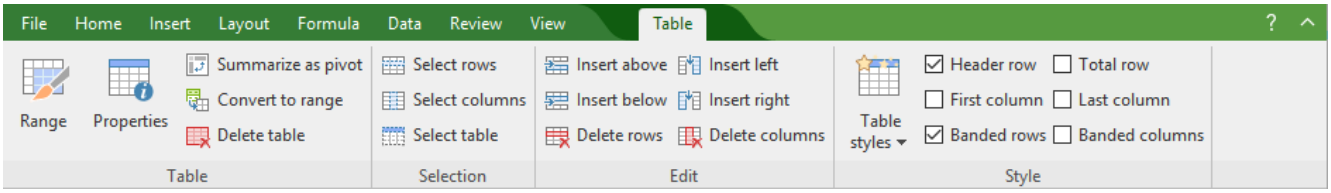
  By default, the filter is disabled and *all* data in the table is displayed. If you click on one of the arrows, the menu of the AutoFilter opens for the corresponding column. In this menu, you can apply different types of filters or even sort the table data.

  For more information on AutoFilters, see AutoFilter.

Ribbon tab "Table"

As soon as you have created a "table in worksheets" using the ribbon command **Insert | Table**, you can switch to the contextual ribbon tab **Table** for editing.

This ribbon tab always appears automatically when you place the cursor in the created table.



The contextual ribbon tab "Table"

The ribbon tab has the following buttons, from left to right:

- **Range:** Here, you can change the cell range that the table contains. For more information, see Editing tables in worksheets.
- **Properties:** This command opens a dialog box in which you can format inserted tables and change other properties of the table. For more information, see Formatting tables in worksheets.
- **Summarize as a pivot table:** Use this command to create a pivot table directly from your table data. For more information, see Pivot tables.
- **Convert to range:** This converts the table back to a "normal" cell range. For more information, see Removing tables from worksheets.
- **Delete table:** This completely deletes the table and all its contents. For more information, see Removing tables from worksheets.

- **Select rows, columns, table:** These commands can be used to select complete rows, columns or all cells of an inserted table. For more information, see Editing tables in worksheets.
- **Insert above, below, left, right** These commands can be used to insert a new row or column into a table (and *only* into this table). For more information, see Editing tables in worksheets.
- **Delete rows, columns:** These commands can be used to delete rows or columns within a table (and *only* within this table). For more information, see Editing tables in worksheets.
- **Table styles** (and all other commands in the "Style" group): This command can be used to apply formatting options to inserted tables. For more information, see Formatting tables in worksheets.

On the following pages, you will find detailed descriptions of the commands of the ribbon tab **Table**.

Removing tables from worksheets

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "tables in worksheets". Here, you will learn how to remove such tables:

In this case, you have the following options:

A) Converting a table back to a normal cell range

To convert a table back to an "ordinary" cell range, move the cell frame to any cell within the table.

Then choose the command **Convert to range** on the contextual ribbon tab **Table | group Table**. (Alternatively, use the context menu by right-clicking in the table.)

In the following dialog box, select the checkbox **Remove all formatting from the cells**.

This will remove the "magic" of the table feature from all cells that were part of the table – including the automatic formatting and the AutoFilter.

B) Deleting a table and its entire content

To completely delete a table and all its contents, move the cell frame to any cell within the table.

Then choose the command **Delete table** on the contextual ribbon tab **Table | group Table**. (Alternatively, use the context menu by right-clicking in the table.)

This will delete the entire table – including all cell contents.

Editing tables in worksheets

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "tables in worksheets". Here, you will learn how to edit such tables:

In this case, you have the following options:

Adding another row or column

An additional row or column can be easily added to "tables in worksheets" as follows:

- If you enter a value in a cell that is immediately to the right of the table, the table automatically expands to include the column with this value.
- When you enter a value in a cell that is just below the table, the table automatically expands to include the row with this value.

  (Note: This only works if the Total row for the table is disabled.)

  Tip: The above two methods for expanding tables can be disabled if you do not want them. To do so, open the dialog box of the ribbon command **File | Options**, choose the **Edit** tab and disable the option **Expand tables automatically**.

- When you press the **Tab** key in the last cell of a table (the bottom right cell), another row is added to the table automatically. This works here even if the "Total row" is enabled, as it is then automatically moved by one row.

Changing the size of a table

You can also change the size of a table at any time as follows:

- A little arrow is displayed in the bottom right corner of a table. When you drag this arrow around with your mouse, the table is resized accordingly.
- If you choose the command **Range** on the contextual ribbon tab **Table**, PlanMaker opens a dialog box in which you can edit the cell range covered by the table.
- You can also use the context menu by right-clicking in the table and selecting the entry **Modify table range** to edit the cell range.

Inserting rows or columns

To insert a new row in a table (and *only* in this table), proceed as follows:

1. Move the cell frame to the desired row in the table.
2. Then choose **Insert above** or **Insert below** on the contextual ribbon tab **Table** | group **Edit**.
   Alternatively: Right-click in the table to open the context menu and choose the command via **Insert** here.

This will add a new row above or below the current row in the table.

Note: These two commands will insert the new row only into the table itself and not into the cells that surround this table.

Inserting new columns works accordingly with the commands **Insert left** or **Insert right**.

Deleting rows or columns

To delete a row from a table (and *only* this table), proceed as follows:

1. Move the cell frame to the row in the table that you want to delete.
2. On the contextual ribbon tab **Table**, choose the command **Delete rows** .
   Alternatively: Right-click in the table to open the context menu and choose the command via **Delete** here.

This will delete the row (including its content) from the table.

Note: This command removes the row only from the table itself and not from the cells that surround this table.

Deleting columns works accordingly with the command **Delete columns** .

Selecting cells in a table

Cells in a table can be selected as usual with a mouse or keyboard, of course.

The commands on the contextual ribbon **Table** | group **Selection** are available as additional options for selecting within tables. Here, you can choose from the following commands:

- **Select rows:** This selects the current table row.
- **Select columns:** This selects the current table column.
- **Select table:** This selects the entire table.

There is another command which is not available on the ribbon by default, but you can easily access it by right-clicking on the table via the context menu:

- **Select** ▸ **Select column data:** This selects only those cells in the current column that contain data. Column headings or cells in the total row will not be included in the selection.

  Alternatively, you will also find the command by default in the *classic menu interface* at:
  Worksheet > Table > Select > Select column data

  Tip: If necessary, you can add this command as a separate button to a ribbon tab (ideally to the ribbon tab "Table"). To do so, choose the ribbon command **File | Customize | Customize ribbon** to open the dialog box for customizing the user interface, and search there for the command "Select data". Then proceed as described in Customizing icons and groups in the ribbon.

Formatting tables in worksheets

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "tables in worksheets". Here, you will learn how to format such tables and change other properties of the table:

First of all, the cells in a table can be formatted just like any other cell, using the entire palette of commands and options that PlanMaker provides for this purpose.

There is also the ribbon command **Table | Properties** especially for tables. This provides formatting options that are only applicable to tables.

You can use this command as follows:

1. Move the cell frame to any cell within the table that you want to format.
2. Choose the command **Properties** on the contextual ribbon tab **Table**.
   Alternatively, right-click on the table to open the context menu and select the entry **Table properties**.
3. A dialog box appears with the options described below. Make the desired changes and then confirm with **OK**.

In this dialog box, you can make the following settings:

Applying a different table style

Tip: You can also apply this option directly via the **Table styles** icon on the ribbon tab **Table** | group **Style**.

The first tab of the dialog box contains the **Style** list. Here, you can apply a different *table style* to the table.

Applying a different table style changes the entire appearance of a table. First and foremost, this concerns the coloring of the table (via shades). The small thumbnails in the list show which colors are included in each style.

Showing/hiding header row and total row

Tip: You can also apply these options directly using the commands on the ribbon tab **Table** | group **Style**.

Below the **Style** list, there are more options for customizing the appearance of a table. The first two options have the following function:

- **Header row**

  This option enables/disables the *header row* (displayed above the table). The header row displays a column title for each of the columns.

  In the header row, you can also see a small arrow displayed to the right of each of the column titles. These arrows indicate the **AutoFilter** that is applied automatically to each table. To filter data, click on one of these little arrows. The program opens a menu in which you can set different types of filters for the values in the corresponding column.

  For more information on using AutoFilters, see AutoFilter.

- **Total row**

  This option enables/disables the *total row* (displayed below the table). The total row displays calculations that sum up the values contained in the table columns.

  By default, only the sum of the values in the last column is displayed. If, however, you click on one of the cells in the total row, a small arrow will appear to the right of it. Clicking on this arrow opens a menu in which you can choose what kind of calculation the cell should perform. For example, you can display the sum, average, or other calculations for any columns in the table.

  For more information, see Using the total row of tables in worksheets.

Additional formatting options for table styles

Tip: You can also apply these options directly using the commands on the ribbon tab **Table** | group **Style**.

The other options on the **Style** tab have the following purpose:

- **Banded rows** and **Banded columns**

  This option enables/disables alternating colored shading for the rows or columns of the table. Their color is defined in the *table style*, which you can also set in this dialog box (as described above).

- **First column** and **Last column**

  If you enable this option, the first and/or the last column in the table will be highlighted.

  For most table styles, this is done by using boldface for the text in the respective column. However, some table styles also change the background color – this depends on the respective table style.

Extended properties on the "Properties" tab

On the **Properties** tab of the same dialog box, you will find the following advanced properties for the table:

- **Name**

  Here, you can change the name of the table, which was automatically generated when the table was created and also included in the list of named ranges at the same time. The name can also be changed there. For more information, see Naming cell ranges.

  The name of a table can be used in cell references (see Tables in worksheets and cell references).

  Please note that the following rules apply to table names:

  1. Table names must be unique.

  2. Table names must always begin with a letter or an underscore.

  3. Table names must not contain blank spaces or other invalid characters. (For maximum compatibility with other programs, only use letters, numbers, and underscores.)

  4. Table names must not be structured like cell references (for example, A1).

- **Title** and **Description**

  Here, you can enter a title for the table and a short description for its contents.

  This is useful for people who, due to impaired vision, use special tools such as a screen reader to have the individual components of the worksheet read to them.

Using the total row of tables in worksheets

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "tables in worksheets". Here, you will learn about the "total row" of such tables and how you can adjust it:

Another advantage of "tables in worksheets" is the possibility of displaying a **total row** with summary calculations if desired. You can enable or disable the total row at any time. When it is enabled, it appears in the form of an additional row below the table.

	A	B	C	D	E
1	Type	Pieces	Price/unit	Price	
2	Apples	4	0.50 €	2.00 €	
3	Oranges	2	0.75 €	1.50 €	
4	Bananas	3	1.00 €	3.00 €	
5	Pears	2	0.75 €	1.50 €	
6	Total			8.00 €	
7					

By default, the total row displays only the "Total" text on the left and the sum of the values of the column on the far right of the table. However, you can also perform other calculations in it by editing the total row (see below).

Enabling/disabling the total row

Tip: The total row can also be enabled directly on the ribbon tab **Table** | group **Style**.

To enable the total row for a table via the dialog box, proceed as follows:

1. Move the cell frame to any cell within the table.
2. Choose the command **Properties** on the contextual ribbon tab **Table**.
   Alternatively, right-click on the table to open the context menu and select the entry **Table properties**.
3. Enable the option **Total row** on the **Style** tab and confirm with **OK**.

The total row will now be displayed below the table.

If you do not want the total row to be displayed, disable this option again.

Editing total rows

By default, the total row displays only the "Total" text and the sum of the values of the column on the far right of the table.

However, you can adjust this as desired. The cells in the total row can be edited – just like any other cell – so you can enter your own formulas or text here at any time.

A little tool is available for adding calculations that total the values in a column: When you move the cell frame to any cell in the total row, a little arrow appears to its right. If you click on this arrow, PlanMaker opens a menu offering different types of statistical functions (sum, average, etc.). If, for example, you select the "Sum" function here, PlanMaker inserts a formula into the cell that calculates the sum of the corresponding column. For information on the formula used, see SUBTOTAL.

Tables in worksheets and cell references

As described in the introduction of the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "tables in worksheets". Here, you will learn more about the specifics of using cell references to the cells in such a table.

Apart from the usual cell references, tables and their contents can also be addressed using *names* and special *identifiers*:

Using names

Names in cell references to table contents can be used as follows:

- **Addressing tables by their name**

  When you use the name of a table in a cell reference, it represents the entire table (with all of its cells, except the column headings and the total row).

  Example:

  =SUM(Table3) returns the sum of all cells in the table named "Table3".

  Tip: The name of a table can be changed via the dialog box of the command **Properties** on the contextual ribbon tab **Table**.

- **Addressing table columns by their column heading**

  To address a single column of a table in a cell reference, enter the name of the table, followed by the column heading of the desired column enclosed in square brackets.

  Example:

  =SUM(Table3[Spring]) returns the sum of all cells in the column with the column heading "Spring" in the table named "Table3".

- **Addressing multiple adjacent table columns**

  You can also address multiple adjacent table columns. To do so, use the notation [[FirstColumn]:[LastColumn]].

  Example:

  =SUM(Table3[[Spring]:[Autumn]]) returns the sum of all cells in the columns "Spring" to "Autumn" in the table.

- **Addressing multiple non-adjacent table columns**

  To address multiple non-adjacent table columns, separate them with a comma instead of a colon.

  Example:

  =SUM(Table3[Summer],Table3[Winter]) returns the sum of the columns "Summer" and "Winter".

  Note that the name of the table must be specified before *each* column.

Using special identifiers

In addition to names, cell references to table contents can also use special *identifiers* that represent a certain component of this table.

The identifier must be enclosed within square brackets when entered after the table name:

For example, **Table3[#Totals]** is a cell reference to all cells in the *total row* of the table named "Table3".

The following identifiers are available:

- **#All:**

  This represents the entire table *including* column headings and total row, for example, Table3[#All].

- **#Data:**

  This represents all cells in the table that contain the actual data. These are all cells *except* the column headings and the total row.

- **#Headers:**

  This represents the cells in the *header row* of the table (the row containing the column headings).

- **#Totals:**

  This represents the cells in the *total row* of the table. (Note: If the total row is disabled, the cell reference will return a #REF! error value.)

- **@[ColumnName]:**

  This represents the cell that is in the *same row* of the specified column. This is especially useful for calculations *within* a table.

  If, for example, you use a cell reference like **[@Spring]** in a cell in a table, it represents the cell that is in the same row of the column with the column heading "Spring".

  Example:

  =SUM([@[Spring]:[Winter]]) returns the sum of those values in the columns "Spring" to "Winter" that are in the same row as the cell in which this calculation was entered.

Note: PlanMaker does not distinguish between uppercase and lowercase letters in identifiers.

Pivot tables

In addition to the normal tables (see chapter Tables in worksheets), there are also *Pivot tables*. You can use the ribbon command **Insert | Pivot table** to create a pivot table based on your existing source data.

What exactly is a pivot table?

A pivot table is an optimal tool for providing large amounts of data in a meaningful form. The generated pivot data can be structured, summarized, filtered and analyzed according to different perspectives. The source data remains unaffected.

Reports using pivot tables are particularly suitable for data sets with many different elements and criteria, which are predominantly similar and interdependent, such as sales reporting.

What is the advantage of pivot tables compared to "normal" tables?
In addition to clarity, pivot tables are much better suited to data analysis than "normal" tables. If you would like to ask complex questions about extensive data, for example, *"How many sales did an employee make for which articles at what time?"*, it would be quite laborious if you had to extract this information directly from a normal table. You would have to sort the data records first, then filter them, use commands and finally prepare them. Pivot tables reduce the number of work steps and enable a much better presentation of the desired information.

Data resources can be edited directly within your PlanMaker worksheet and also by importing data from external PlanMaker and Microsoft Excel files.

Pivot tables illustrated by an example

The functionality of pivot tables is illustrated here with a simplified example, which is used throughout the chapter for description purposes.

Here, you can see sample data records in PlanMaker from which a pivot table is to be created. The company employees had various expenses for their break room supplies throughout the year. Some employees and some articles appear several times in the relevant columns.

	A	B	C	D	E
1	**Date**	**Employee**	**Article**	**Amount**	
2	01/10/2018	Anna	Coffee	$35	
3	02/15/2018	Toni	Cream	$15	
4	04/13/2018	Anna	Snacks	$20	
5	05/15/2018	Toni	Tea	$15	
6	06/18/2018	Maria	Coffee	$30	
7	06/20/2018	Anna	Cream	$12	
8	08/11/2018	Anna	Snacks	$16	
9	08/18/2018	Toni	Sweetener	$7	
10	10/28/2018	Anna	Coffee	$40	
11	11/15/2018	Toni	Cream	$10	
12					

Source data for the pivot table

Note: Number formatting of the source data is not applied to pivot tables. For more information on formatting values in the created pivot table, see Value settings (Values area).

As an evaluation, the following pivot table could be created: Here, the *employees* are clearly listed with the sums of their purchased articles and the *grand total* of all expenses.

Row labels	Sum of Amount
Anna	123
Maria	30
Toni	47
Grand total	**200**

Variant 1 of a pivot table (view of the employees)

However, this pivot table can also be created: Here, the purchased *articles* are arranged in a totaled overview.

Row labels	Sum of Amount
Coffee	105
Cream	37
Snacks	36
Sweetener	7
Tea	15
Grand total	**200**

Variant 2 of a pivot table (view of the articles)

Above all, the possibility of grouped arrangement shows the major advantages of pivot tables: Here, *employees* and *articles* are displayed together and nested in one overview.

Row labels	Sum of Amount
Anna	**123**
Coffee	75
Cream	12
Snacks	36
Maria	**30**
Coffee	30
Toni	**47**
Cream	25
Sweetener	7
Tea	15
Grand total	**200**

Variant 3 of a pivot table (view of employees with grouped articles)

It can be seen from these examples that, depending on the question addressed to the existing data, different statements can be made with pivot tables.

The following sections clarify how to obtain different perspectives on pivot data and what settings you can make to obtain a suitable presentation of your results:

- Creating a new pivot table
- Starting with the pivot table field list

- Configuring pivot tables with the field list
- Sorting and filtering pivot tables
- Grouping pivot tables
- Updating pivot tables and changing data area
- Deleting, copying, moving pivot tables

Creating a new pivot table

In the iOS version, the command is locked. (To unlock the command, see Welcome.)

To create a pivot table, you must first have source data records to which a pivot table can be applied.

Choose the ribbon command **Insert** | group **Tables** | **Pivot table** to create a pivot table from existing source data.

Please note a few important rules for the existing source data. Some of the requirements are not mandatory, but they are still helpful for practical use:

Requirements for source data

- The source data must have a separate heading for each column in the top row.
- Headings must appear in this single row, not one below the other.
- Headings must be unique, and duplicate names for the headings must not occur.
- There must be no empty rows or columns and no merged cells in the source data.
- The data in the respective columns must be of the same data type (number, text or date – not mixed).
- If possible, the data should be available in its raw form; no formulas, summarizations, subtotals or other prepared data should have been applied to it previously.

You can either use existing records in the current workbook as source data or import data records from another file (possible formats: PlanMaker and Excel).

Please read the next two sections:

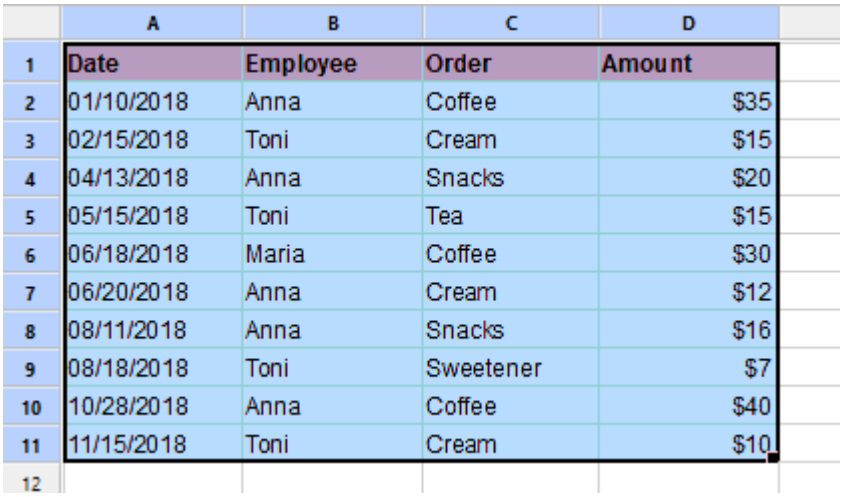
- Creating a pivot table with existing data
- Creating a pivot table with imported data

Tip: Pivot tables can also be generated from Tables in worksheets. To do so, select a cell in the table and choose the command **Summarize as pivot** (in the group **Table**) on the contextual ribbon tab **Table**. Alternatively, you will also find this command in the context menu by right-clicking in the table. Then proceed with the next steps to create a pivot table as described in the next section.

Creating a pivot table with existing data

If the source data is already in the open workbook, proceed as follows:

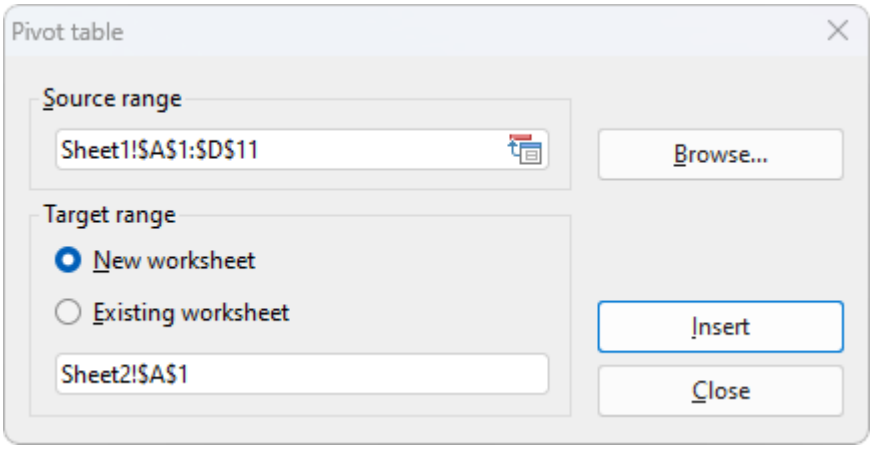
1. Select the entire data area. You can also select just one cell from the source data. PlanMaker will then automatically extend the selection to the entire contiguous area.



	A	B	C	D
1	Date	Employee	Order	Amount
2	01/10/2018	Anna	Coffee	$35
3	02/15/2018	Toni	Cream	$15
4	04/13/2018	Anna	Snacks	$20
5	05/15/2018	Toni	Tea	$15
6	06/18/2018	Maria	Coffee	$30
7	06/20/2018	Anna	Cream	$12
8	08/11/2018	Anna	Snacks	$16
9	08/18/2018	Toni	Sweetener	$7
10	10/28/2018	Anna	Coffee	$40
11	11/15/2018	Toni	Cream	$10
12				

2. Choose the ribbon command **Insert** | group **Tables** | **Pivot table** .

   PlanMaker opens the following dialog box:



3. **Source range:** This is the cell range with the existing data from which the pivot table is to be generated. Here, the source range is already defined by your previous selection and can thus be applied. If you still want to change the source range, you can enter the correct cell range in the input field or simply select the correct cell range directly in the worksheet with the mouse.

Tip: If you have previously given the data area a name (see Naming cell ranges), you can simply enter this name in the input field. An advantage of this procedure is that you only have to adjust the named range if the source data changes.

You can also import external data using the **Browse...** button. For more information, see Creating a Pivot table with imported data.

4. **Target range:** Here, you can decide where the pivot table should be created. Now choose between the following options:

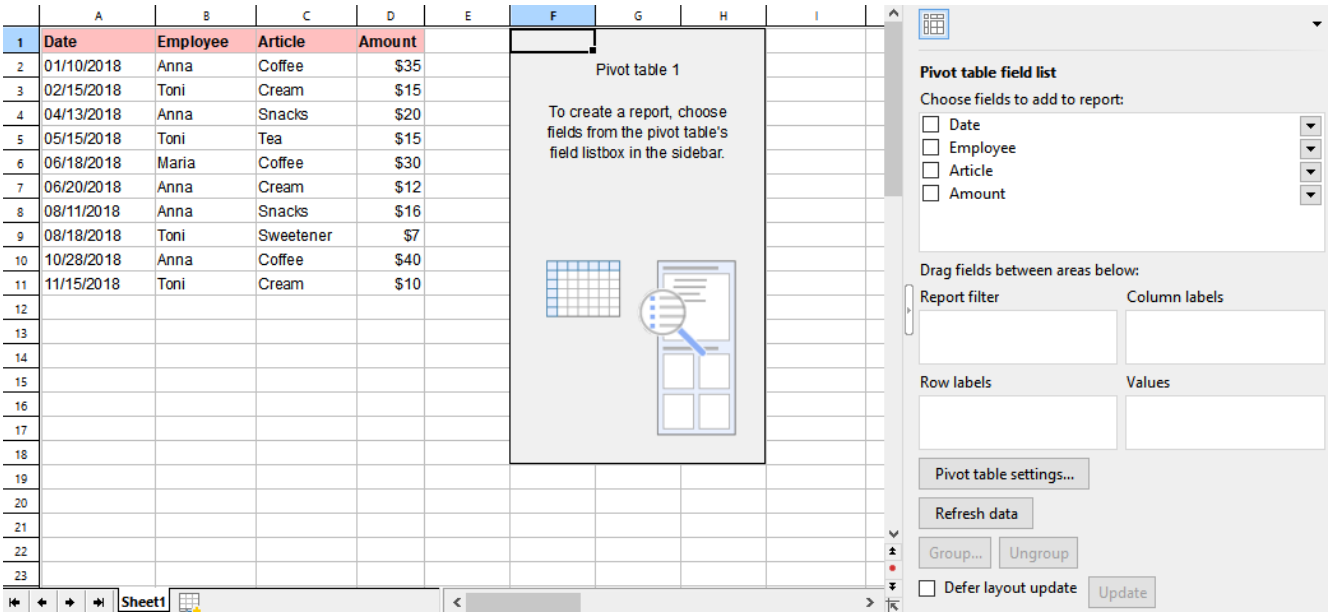
   New worksheet: The pivot table will be created in a new worksheet that is automatically generated by PlanMaker. You can adjust the proposed target in the lower input field.

   Existing worksheet: The pivot table will be created in an existing worksheet. This can be the worksheet containing the source data or another worksheet that has already been created. Please make sure that you first select the option *Existing worksheet* and only then click on a cell in a **free area** in the desired worksheet. Alternatively, type the target range into the lower input field.

   Tip: For pivot tables with large amounts of data, we recommend that you use a new worksheet rather than the worksheet with the source data, as it can quickly lead to a lack of space otherwise. If there is only a small amount of data – as in our example – it is useful and illustrative to display the pivot table next to or below the source data in the same worksheet.

5. Confirm with the **Insert** button to create the pivot table.

If you created the pivot table in the same worksheet as the source data, the result should look as follows:



On the left, you can still see the **Source data**, and the (still blank) **Pivot table report** next to it. If you click in this report, the **Pivot table field list** – or just **Field list** – appears in the right sidebar. It is the central control of the pivot table. By selecting the fields from the field list, you fill the blank pivot table report with content according to your requirements.

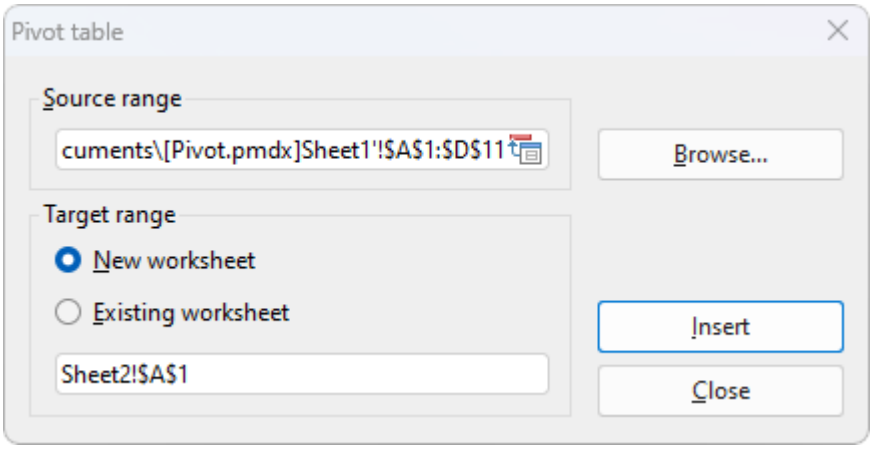
For more information about its structure and working with the field list, see Starting with the pivot table field list.

Creating a pivot table with imported data

You can also import data records from other PlanMaker files or Microsoft Excel files to create a pivot table. To do so, proceed as follows:

1. Choose the ribbon command **Insert** | group **Tables** | **Pivot table** .

   PlanMaker opens the following dialog box:



2. **Source range**

   In the file browser, locate the file with your source data and confirm with **Open**.

   The input field below *Source range* displays the file path with the file name and a proposed worksheet with a cell range. Here, you have to adjust the desired cell range precisely. PlanMaker does not automatically extend the cell range to contiguous data records when importing from external files.

   Example: Your source data is in the file *Pivot.pmdx* in the worksheet *Sheet1*, and the cell range of your source data records is from *A1 to D11*.

   The syntax in the input field is then: 'filepath\[Pivot.pmdx]Sheet1'!$A$1:$D$11

   Tip: If you have previously named the range of source data in your external file (see Naming cell ranges), you can avoid entering the cell range exactly. A further advantage of this procedure is that you only have to adjust the named range when making changes to the data records. Choose the name of the named range in the input field using the following syntax: 'filepath\[filename]'!name

3. **Target range:** Here, you can decide where the pivot table should be created:

   New worksheet: The pivot table will be created in a new worksheet that is automatically generated by PlanMaker. You can adjust the proposed target in the lower input field.

   Existing worksheet: The pivot table will be created in an existing worksheet. This can be the current worksheet or another worksheet that has already been created. Please make sure that you first select the option *Existing worksheet* and only then click with the mouse on a cell in a **free area** in the desired worksheet. Alternatively, type the target range into the lower input field.

4. Confirm with the **Insert** button to create the pivot table.

You should now see a (still blank) **Pivot table report** in the worksheet and the **Pivot table field list** or just **Field list** on the right in the sidebar. It is the central control of the pivot table. By selecting the fields from the field list, you fill the blank pivot table with content according to your requirements.

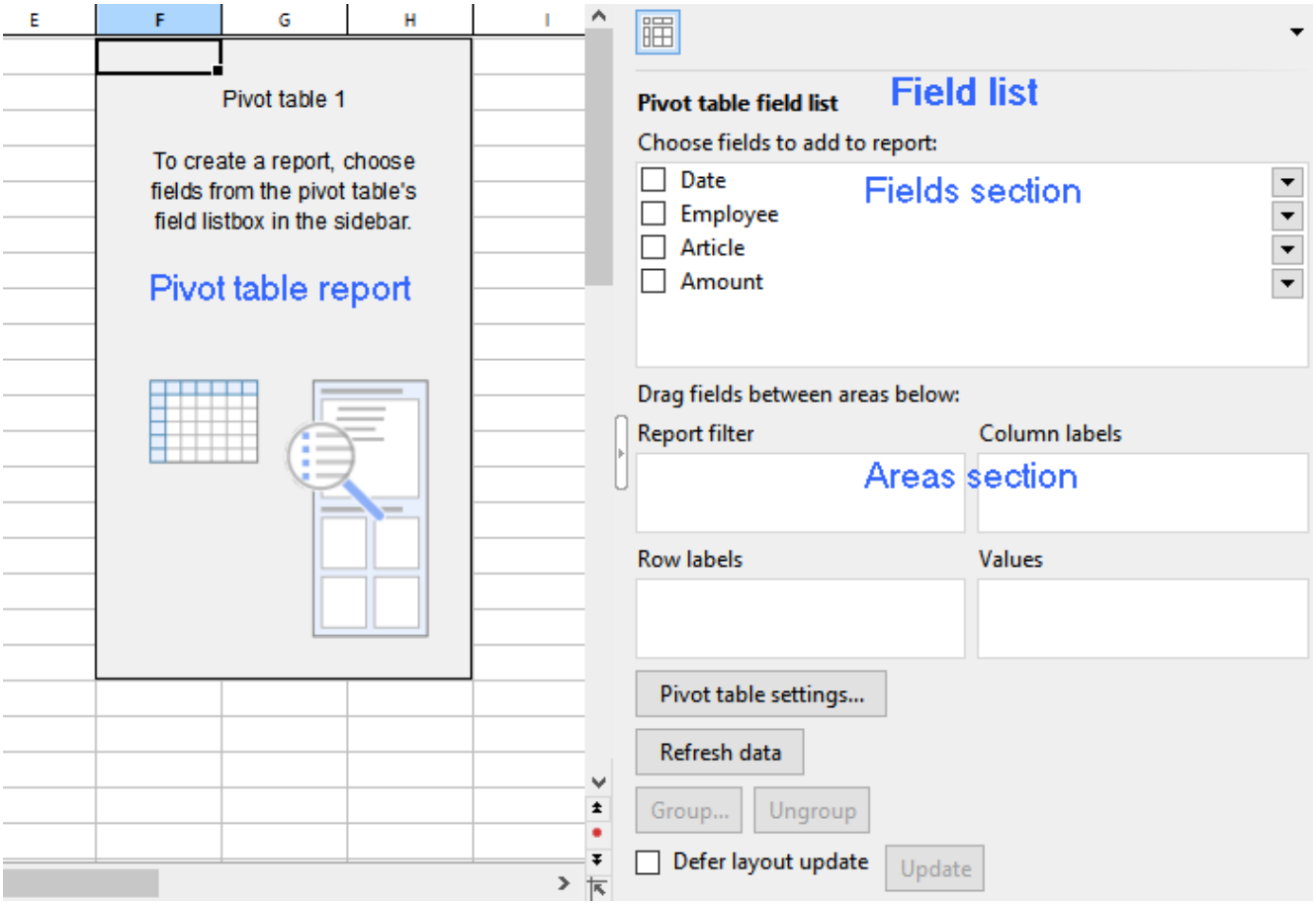
The following sections explain its structure and working with the pivot table field list.

Starting with the pivot table field list

Once you have created a pivot table, the **Field list** appears in the sidebar on the right with the upper **Fields section** and the lower **Areas section**. It is the central element of a pivot table because it is where you control which data is displayed in the target range of the pivot table (the **Pivot table report**) and how the data is arranged.

Note: The field list appears whenever you click on the pivot table report.

Structure of the field list



On the left, you have the pivot table report that is still blank; on the right, you have the field list with the "Fields section" above and the "Areas section" below.

Fields section: In the upper part of the field list, the headings from the columns of the source data are grouped in their original sort order. The order of the columns from left to right in the source data corresponds to the order from top to bottom in the fields section. These elements are called **"Fields"** here and not "Columns".

Areas section: You can drag the fields from the upper fields section into the lower part of the field list, and thus design the structure of the pivot table depending on the arrangement of these fields. Once you have placed at least one field in one of the 4 areas of the areas section, results appear in the *pivot table report*.

The areas section is divided into the following 4 individual areas:

- **Row labels**

  If you drag a field into this area, the data from this field is grouped by *rows* in the pivot table report. In this context, grouping means that identical items from a column of the source table appear once only and are totaled. You can also drag several fields into this area; the added fields then also form further subgroups by row. The field that appears in the row labels at the top is the higher-level primary sort criterion. The fields below are treated as lower-level sort criterion and are displayed in the pivot table report according to this structure.

- **Column labels**

  The same procedure applies here as for row labels, but grouping is by *columns*.

- **Values**

  This is the most important area: The records of the field that was dragged here are totaled in the pivot table report, for example, amounts are added up. This is why fields with numbers usually appear here.

- **Report filter**

  This allows you to show and hide selected records in the pivot table report based on the field that is inserted into the report filter. For more information, see also Sorting and filtering pivot tables.

Tips for displaying the field list in the sidebar:

- Adjust the **Field list width** as needed by dragging the mouse along the inner edge of the sidebar.
- To **show or hide the field list**, click on the narrow vertical button at the edge of the sidebar.
- Use the ribbon command **View** | group **Windows** | **Pivot table sidebar** | **Show at left** or **Show at right** to choose at which side of the document the field list should be displayed.

In the next sections, you will find practical information on how to use the field list.

Exercises for structuring the field list

You have already created a pivot table using the ribbon command **Insert** | group **Tables** | **Pivot table** . The field list with the fields section and the areas section is on the right-hand side, but the target range of the pivot table (the *pivot table report*) is still blank.

This section explains how to drag elements step-by-step from the fields section to the areas section in the field list in order to display meaningful results with different characteristics in the pivot table report.

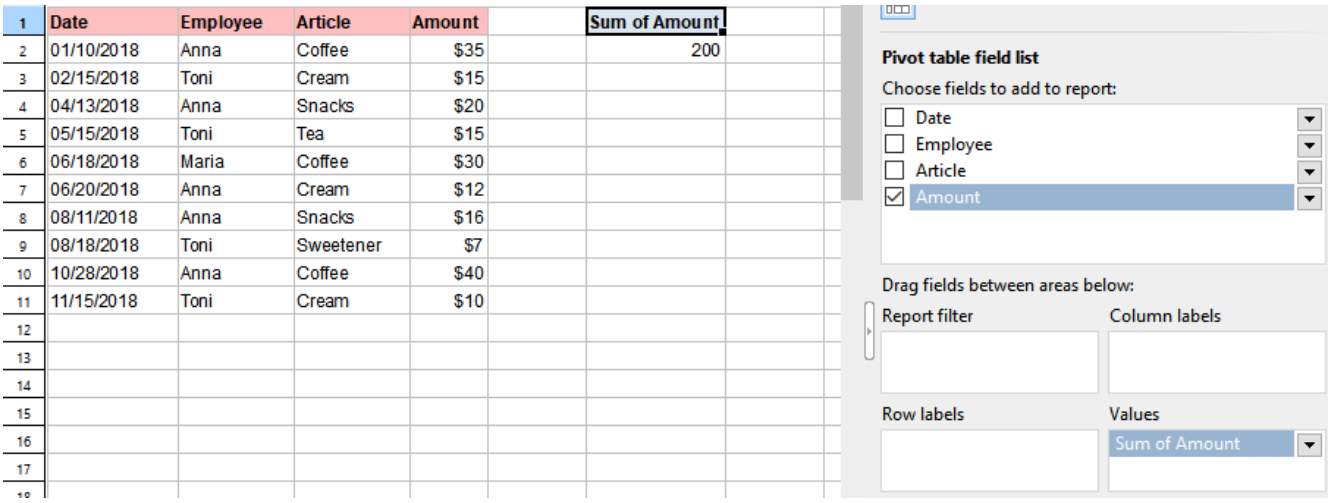
If you are already familiar with the structure of a pivot table, you can skip this section.

Tip 1: You can drag the fields to the desired area with the mouse. You can also set a check mark in front of the relevant field in the fields section. PlanMaker then automatically assigns the field to an area in the areas section. Fields with numbers are placed in the values area; all other formats, such as text and date, are placed in the row labels area.

Tip 2: Fields can be dragged back and forth not only between the fields section and the areas section, but also between the individual areas of the areas section and within the areas themselves, so you can easily swap the position of the fields.

Exercise 1: Adding a field to the "Values" area

In the fields section, set a check mark in front of the *Amount* field or use the left mouse button to drag the field down directly to the **Values** area of the areas section.

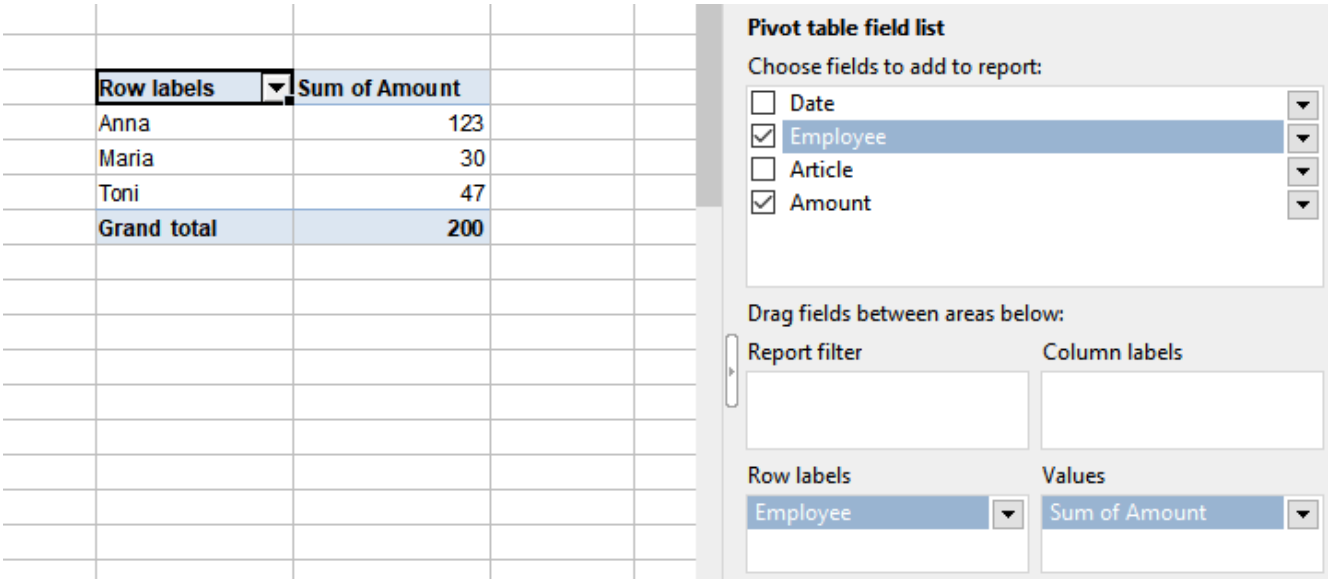


The unchanged source data (left), the result of the pivot table (middle) and the field list with the selected fields for Exercise 1 (right)

- The field element "Sum of amount" will appear in the **Values** area of the field list.
- In the pivot table report, the value field with the header "Sum of amount" will now be displayed instead of the blank report.
- All values from the *Amount* field are summed up to a grand total in the pivot table report.

Exercise 2: Adding a field to the "Row labels" area

Now set an additional check mark in front of the *Employee* field in the fields section or use the left mouse button to drag the field down directly to the **Row labels** area.

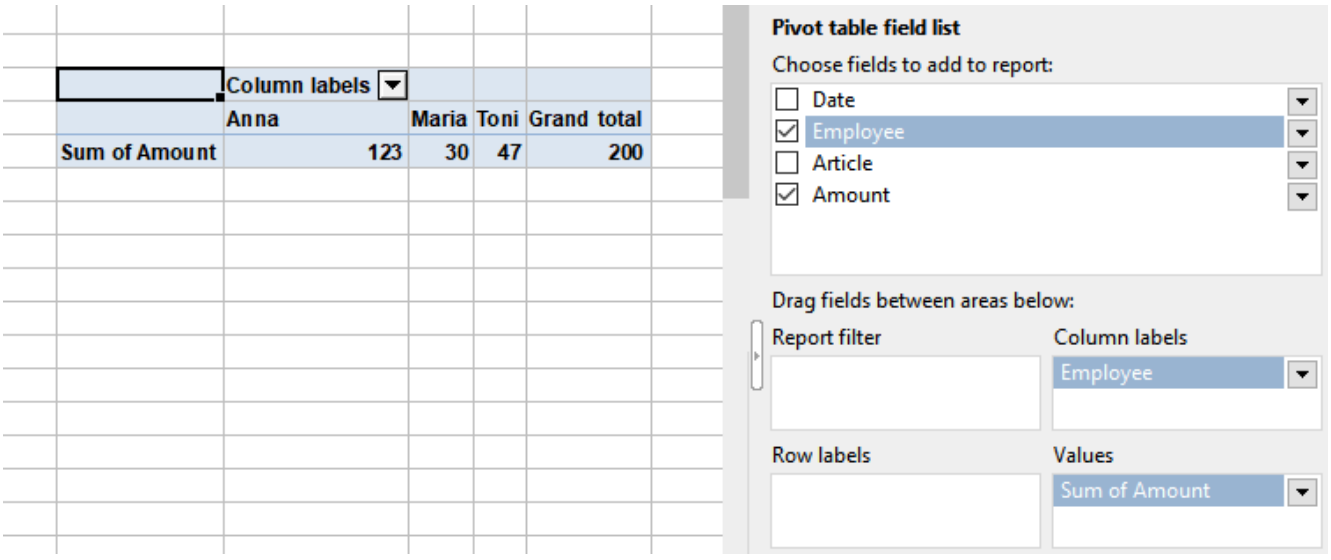


The result of the pivot table (left) and the field list with the selected fields for Exercise 2 (right)

- The field element "Employee" appears in the **Row labels** area of the field list.
- In the pivot table report, there is now a row field with the header "Row labels". Below the row field, the employees are arranged by row and each name only appears once.
- The value field with the header "Sum of amount" in the pivot table report displays the expenses totaled and broken down by employee.

Exercise 3: Adding a field to the "Column labels" area

You can also display the result by columns instead of by rows. To do so, drag the same *Employee* field to the **Column labels** area.



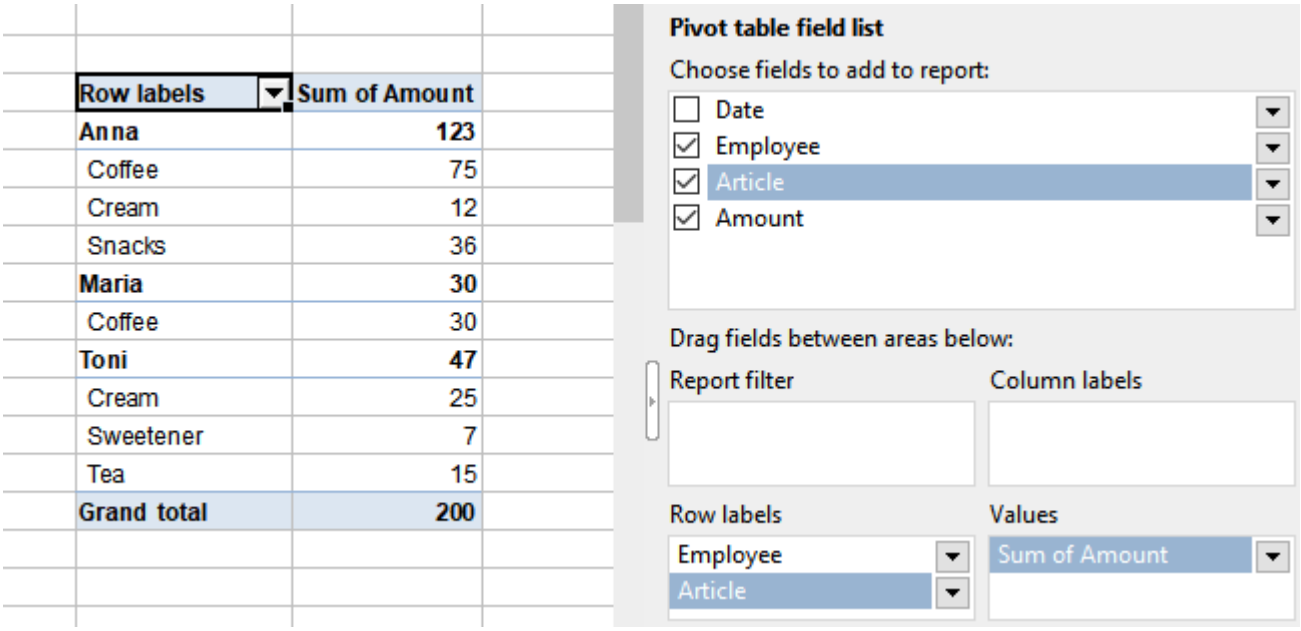
The result of the pivot table (left) and the field list with the selected fields for Exercise 3 (right)

- The field element "Employee" appears in the **Column labels** area of the field list.

- In the pivot table report, a column field with the header "Column labels" is displayed and the employees are arranged by column. Each name only appears once here.
- The expenses are summed in the value field "Sum of amount" of the pivot table report and are broken down by employee.

Exercise 4: Adding a second field to the "Row labels" area

Drag the *Employee* field back into the **Row labels** area, and set an additional check mark in front of the *Article* field.



Row labels	Sum of Amount
Anna	**123**
Coffee	75
Cream	12
Snacks	36
Maria	**30**
Coffee	30
Toni	**47**
Cream	25
Sweetener	7
Tea	15
Grand total	**200**

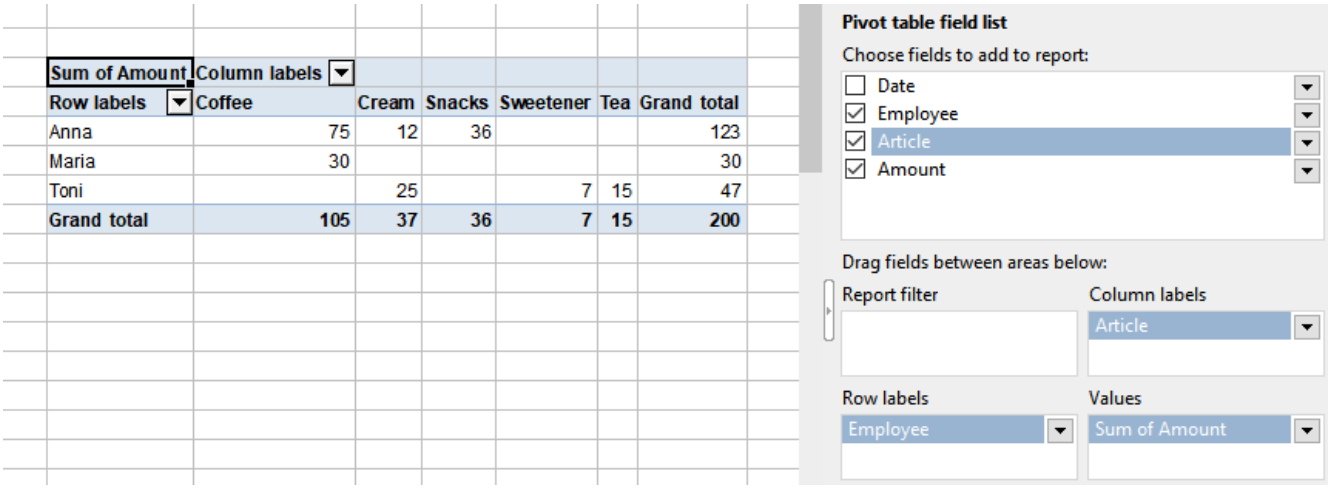
The result of the pivot table (left) and the field list with the selected fields for Exercise 4 (right)

- In the pivot table report, employees are grouped with their totaled article types under "Row labels". Since the *Employee* field is at the top of the row labels area, it is the primary criterion and is thus displayed as the higher-level field in the pivot table report.
- The amounts for purchased article items are broken down by employee in the "Sum of amount" value field in the pivot table report, and partial results are displayed as subtotals per employee.

> **Note:** You may have the article items in the pivot table report grouped as an indented column to the right of the employees. The structure of the results shown here is based on PlanMaker's default settings. For information on customizing the form, see Configuring pivot tables: Field settings.

Exercise 5: Using "Column labels" and "Row labels" together

Drag the *Article* field to the **Column labels** area, while the *Employee* field remains in the **Row labels** area.



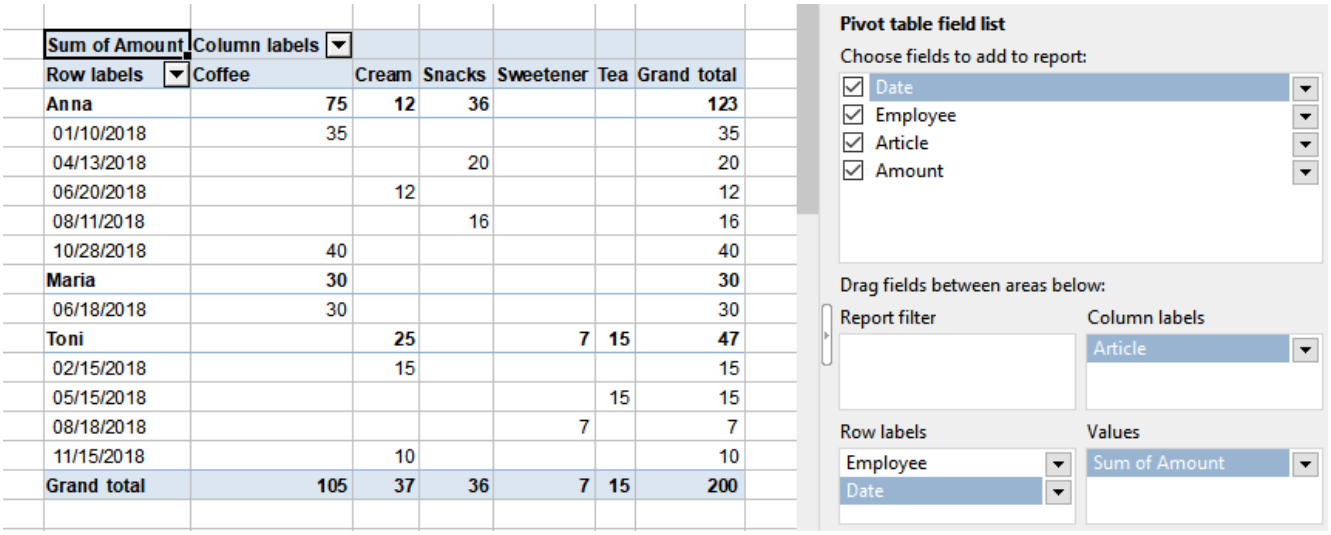
Sum of Amount	Column labels					
Row labels	Coffee	Cream	Snacks	Sweetener	Tea	Grand total
Anna	75	12	36			123
Maria	30					30
Toni		25		7	15	47
Grand total	105	37	36	7	15	200

The result of the pivot table (left) and the field list with the selected fields for Exercise 5 (right)

- You will see the same results as in Exercise 4 but presented as a cross-classified table. The intersecting fields without existing values (for example, Anna did not buy sweetener) are displayed as empty cells.

Exercise 6: Using "Column labels" and "Row labels" together with multiple fields

Finally, drag the *Date* field to the **Row labels** area below the *Employee* field.



Sum of Amount	Column labels					
Row labels	Coffee	Cream	Snacks	Sweetener	Tea	Grand total
Anna	**75**	**12**	**36**			**123**
01/10/2018	35					35
04/13/2018			20			20
06/20/2018		12				12
08/11/2018			16			16
10/28/2018	40					40
Maria	**30**					**30**
06/18/2018	30					30
Toni		**25**		**7**	**15**	**47**
02/15/2018		15				15
05/15/2018					15	15
08/18/2018				7		7
11/15/2018		10				10
Grand total	105	37	36	7	15	200

The result of the pivot table (left) and the field list with the selected fields for Exercise 6 (right)

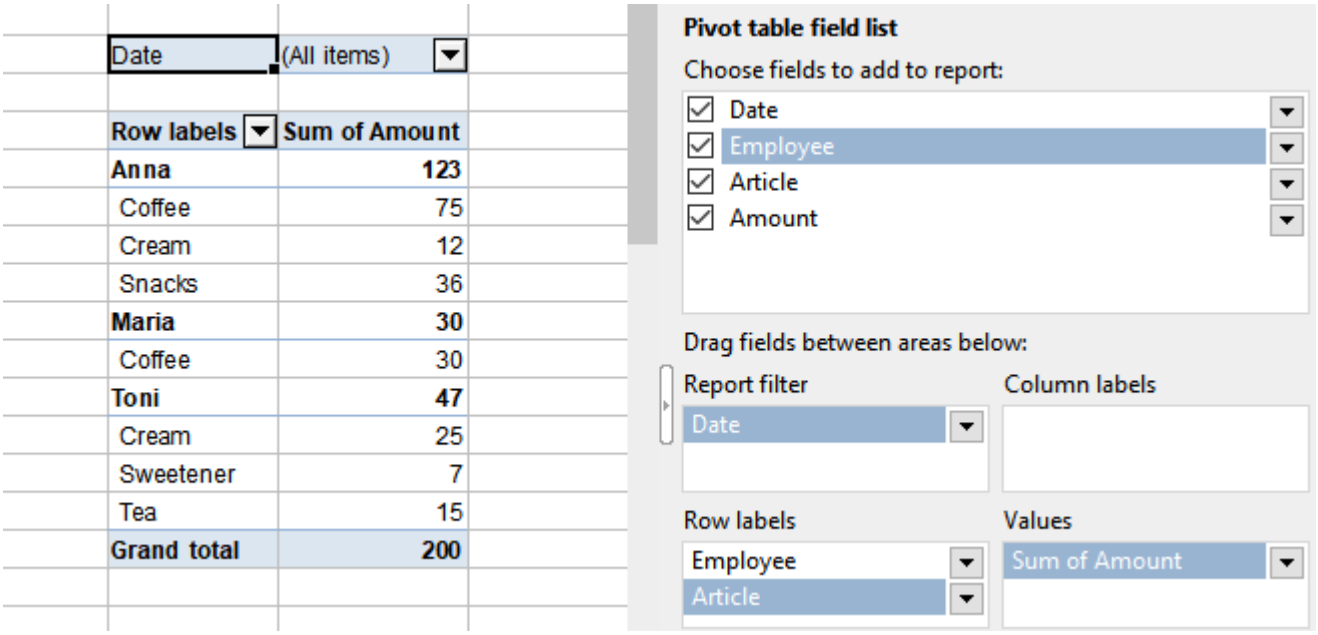
- The date of the article purchased by employees is also displayed in the pivot table report, grouped directly below the employees in this example. A grouping of the items as an indented column to the right of the employees could also appear, depending on the settings in PlanMaker.

Note 1: You can drag multiple fields into the row and/or column labels for evaluation purposes, but your pivot table will also quickly become increasingly unclear as a result.

Note 2: If there are multiple fields in an area, the order of the fields determines the structure of the outline levels displayed in the pivot table report. You can change this order at any time by dragging the field up or down within the area.

Exercise 7: Adding a field to the "Report filter" area

Drag the *Date* field from the **Row labels** area to the **Report filter** area and the *Article* field back to the **Row labels** area.



Date	(All items)
Row labels	**Sum of Amount**
Anna	**123**
Coffee	75
Cream	12
Snacks	36
Maria	**30**
Coffee	30
Toni	**47**
Cream	25
Sweetener	7
Tea	15
Grand total	**200**

The result of the pivot table (left) and the field list with the selected fields for Exercise 7 (right)

- A **Report filter** has been added for the *Date* field, which is integrated as a new row above the pivot report.
- Above the pivot report, you can open a dropdown menu to the right of *(All items)* and filter by date values displayed in the list. First, all date values are selected, indicated by the check mark in front of them. You can filter out the undesired date items from the pivot report by removing the check marks.

Working with the field list: Fields section

The fields section at the top of the field list displays the headings of the columns of your source data. You can make the following changes to the fields section:

- **Display field names alphabetically in the fields section**

  The field names are first automatically sorted according to the order of the columns from your source data. You can also display the fields in alphabetical order to find specific field names more quickly. To do so, click on the **Pivot table settings** button below the areas section. On the **Filter** tab, go to the **Field list** section, select the option **Sort A to Z** and confirm with **OK**.

  Note: The alphabetical order that you enter here does not affect the display in the pivot table report. For information on the alphabetical sorting of elements directly in the pivot table report, see Sorting and filtering pivot tables.

- **Rename field names in the fields section**

  Open the dropdown menu to the right of the field name and select **Field settings** there. In the dialog box that opens, you can assign a **Custom name**.

  You can also rename a field if it is in the values area or row labels/column labels area of the areas section. For more information, see Working with the field list: Areas section.

- **Move fields from the fields section to the areas section**

  To move a field from the fields section to the areas section, you have the following options:

  A. Enable the checkbox to the left of the field name in the fields section. PlanMaker automatically assigns the field to an area of the areas section according to its format: Numbers are placed in the *Values* area, while text and date formats end up in the *Row labels* area.

  B. You can also drag the field with the left mouse button to the desired area below.

  C. Alternatively, click on the dropdown menu to the right of the field name. Here, you can select the area in which you want to move the field directly.

Working with the field list: Areas section

After creating a pivot table, you will first see a blank pivot table report in your worksheet and the field list in the sidebar on the right. By adding the fields from the upper fields section to the 4 individual areas of the lower areas section, you can observe how the assignments in the field list affect the previously blank pivot report.

Tip: If possible, first drag the field with the amounts into the values area of the areas section. This is the best way to follow the next steps for structuring the pivot table.

Adding, moving, removing fields

Adding fields to the areas section

To move a field from the fields section to the areas section, you have the following options:

A. Enable the checkbox to the left of the field name in the fields section. PlanMaker automatically assigns the field to an area of the areas section according to its format: Numbers are placed in the values area, while text and date formats end up in the row labels area.

B. You can also drag the field with the left mouse button to the desired area below.

C. Alternatively, click on the dropdown menu to the right of the field name. Here, you can select the area in which you want to move the field directly.

Moving fields within the areas section

To move fields from one area to another area within the areas section, you have the following options:

A. Drag the field from one area to another with the left mouse button.

B. Click on the dropdown menu to the right of the field name (either in the fields section above or in the areas section below). Here, you can select in which area you want to move the field directly.

C. Use the left mouse button to drag the field again from the upper fields section to the desired new area of the areas section. This will automatically remove it from the previous area (except for the values area, where copies of a field are possible).

Removing fields from the areas section

To remove fields from the areas section at any time, you have the following options:

A. Use the left mouse button to drag the field back up to the fields section.

B. Disable the checkbox to the left of the field name in the fields section.

C. Click on the dropdown menu to the right of the field name (either in the fields section above or in the areas section below) and select **Remove field**.

> **Note:** Removing fields from the areas section or moving fields between the areas will undo any changes made to the field (exception: renaming in the row labels/column labels area).

Tips and rules for using the areas section

Using fields multiple times in the areas section

- For the *Row labels*, *Column labels* and *Report filter* areas, the following applies: A field can only be dragged into one of these areas once. If you also want to insert it into another area, it will be removed from the area where it was previously.
- This does not apply to the *Values* area: You can insert a field again here, even if it already exists in another area. The same field can even be inserted several times as a copy. What would be the advantage of doing so? You could display a field element first as a sum and then in a different format (for example, average). An additional element ∑ *Values* appears automatically in the column labels, which you can also move to the row labels area (the format is retained) but not to the report filter area.

> **Note:** If you drag the field ∑ *Values* back into the fields section, all fields in the values area will be automatically removed and all changes made to these fields will be discarded.

Changing the order of the fields within an area of the areas section

- The order of the fields within an area can be changed by using the mouse to drag the field to the correct position within the area.
- Alternatively, click on the dropdown menu to the right of the field name. Here, you can select where you want to move the field (up, down, to the start or to the end).

Renaming a field name in the areas section

A field name can be renamed in the areas section so that it appears in the pivot table with a different name.

- Custom name in the **Row labels** or **Column labels** area:

  If the field is in the row labels or column labels area, click on the dropdown menu to the right of the field name and select **Field settings**. In the **Field** tab, enter a custom name.

- Custom name in the **Values** area:

  If the field is in the values area, select **Value settings** from the dropdown menu to the right of the field, and enter a custom name in the **Summarize values** tab.

  Note: For renaming in the **Values** area, the following applies: The new name is discarded when the field is removed from this area. However, you can also rename the field in the fields section above (see Working with the field list: Fields section); the renaming done there remains. In the **Values** area, however, the addition "Sum/Count of..." is still displayed.

- In the **Report filter** area, you cannot rename fields. However, you can also rename the field in the fields section above.

Further options in the field list

The field list includes other operating options that have not yet been described but are explained in more detail in the following sections:

- **Pivot table settings** button:
  Section Pivot table settings
- **Refresh data** button:
  Section Updating pivot tables and changing data area
- **Defer layout update** along with the **Update** button:
  Section Defer layout update
- **Field settings** (via dropdown menu of fields in the row labels/column labels area):
  Section Field settings (Row/Column labels area)
- **Filter labels** (via dropdown menu of fields in the row labels/column labels area):
  Section Sorting and filtering pivot tables
- **Filter values** (via dropdown menu of fields in the report filter area):
  Section Sorting and filtering pivot tables
- **Value settings** (via dropdown menu of fields in the value area):
  Section Value settings (Values area)

Configuring pivot tables with the field list

In the field list, you have various options for changing the structure and appearance of your pivot table, which are explained in this section:

- **Field settings**

  The **Field settings** can be found in the *Row labels* or *Column labels* areas in the areas section of the field list. For this purpose, click on the dropdown menu to the right of each field. In the field settings, you can define if and how partial results are displayed and in which form the layout appears (outline form or tabular form); you can also change the field name here.

- **Value settings**

  Fields in the *Values* area have different setting components from the field settings in the row/column labels. They can also be opened in the *Values* area via the dropdown menu of the respective field and are called **Value settings** here. In the value settings, you will find a selection of evaluation functions and calculations as well as the desired number format for the displayed results. The field name can also be renamed here.

- **Pivot table settings**

  In the **Pivot table settings**, you can make advanced layout adjustments, define filter methods, select styles, set data usage and assign a name to the pivot table. You will find the pivot table settings located as a button below the areas section.

- **Defer layout update**

  You will find the option **Defer layout update** along with the **Update** button at the bottom of the field list. It allows you to "freeze" the continuous recalculation, and you can rearrange the fields between the areas without time-consuming calculations.

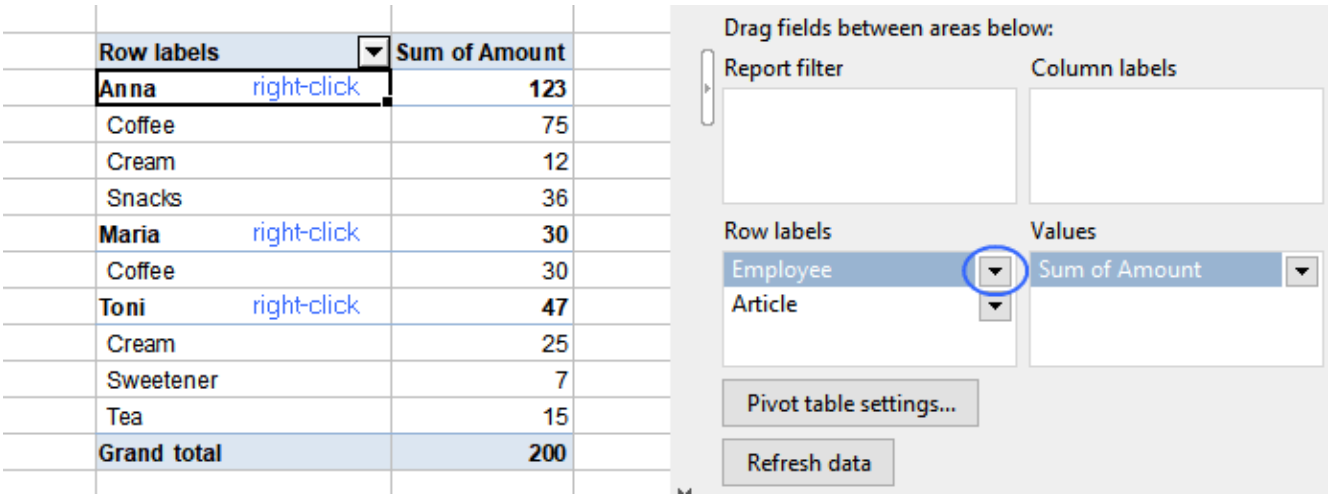
Please note: Changes to **Field settings** and **Value settings** are discarded when a field is removed from its area (back to the fields section or to another area of the areas section). Changes to **Pivot table settings** are retained.

Field settings (Row/Column labels area)

If you open the dropdown menu to the right of a field in the **Row labels** or **Column labels** areas in the lower areas section of the field list, you will find the option **Field settings**. Alternatively, you can select this option in the context menu by right-clicking directly in the Pivot table report (select a cell of the desired field below the header *Row labels/Column labels*).

This section describes the adjustments you can make with the **field settings**.

This is illustrated by the familiar sample data shown in the figure below: The *Amount* field is dragged into the **Values** area of the areas section, and the *Employee* and *Article* fields are dragged into the **Row labels** area.



The pivot table report with default values for field settings. Highlighted in blue: selecting the field settings for the "Employee" field

Note: For the purposes of simplicity, the descriptions in this section refer to the arrangement of the fields in the *Row labels* area. If you drag the fields to the *Column labels* area instead, the results will be basically the same. However, they will be displayed transposed (that is, from left to right instead of from top to bottom).

Field settings, "Subtotals" tab

If you have arranged several fields in a row labels or column labels area, you will obtain partial results for the higher-level field element (Employee) in the pivot table report, as shown in the figure above (for Anna, Maria, Toni). These subtotals will be automatically calculated as sums if a number field has been identified in the values area. In the **Subtotals** tab, you can also change the display of the subtotals to other evaluation functions (for example, count, average, etc., instead of sums) or hide the display of subtotals completely.

Important: The field settings affect only the selected field of the row/column labels area in the pivot table.

Click on the dropdown menu to the right of the field (Employee) in the **Row labels** area (areas section of the field list), and select the **Field settings**. In the open dialog box, select the **Subtotals** tab.

Here, you have the following options:

- **Auto** (default): *Sums* are displayed for the subtotals (Employee, as in the figure above). If the value field (Amount) contains values with other formats rather than numbers, the *Count* function is automatically used for the subtotals.
- **None:** The subtotals (Employee) are not displayed.

Row labels	Sum of Amount
Anna	
Coffee	75
Cream	12
Snacks	36
Maria	
Coffee	30
Toni	
Cream	25
Sweetener	7
Tea	15
Grand total	200

The Pivot table report without subtotals

- **Custom**

  You can choose from various evaluation functions for displaying subtotals by clicking on the option **Custom**:

  Sum: The subtotals are displayed as the sum of the individual values.

  Count: This displays the number of individual values that occur. This works in the same manner as the COUNTA function.

  Average: This displays the average value of the individual values that occur.

  Maximum: This displays the highest individual value that occurs.

  Minimum: This displays the lowest individual value that occurs.

  Product: This displays the multiplied value of the individual values that occur.

  Count numbers: This displays the number of values that are numbers. This works in the same manner as the COUNT function.

  Standard deviation (sample): This provides an estimate of the standard deviation of a population based on a sample. This works in the same manner as the STDEV.S function.

  Standard deviation (population): This returns the standard deviation of a population. This works in the same manner as the STDEV.P function.

  Variance (sample): This provides an estimate of the variance of a population based on a sample. This works in the same manner as the VAR.S function.

  Variance (population): This provides the variance of a population. This works in the same manner as the VAR.P function.

Tip 1: You can also combine several custom functions. Each function is then displayed as a separate row for the subtotal.

Tip 2: If you want to change both the subtotals and also the other values to a certain display, you can make the required changes in the *Values* area with the option **Value settings**. For more information, see Value settings (Values area).

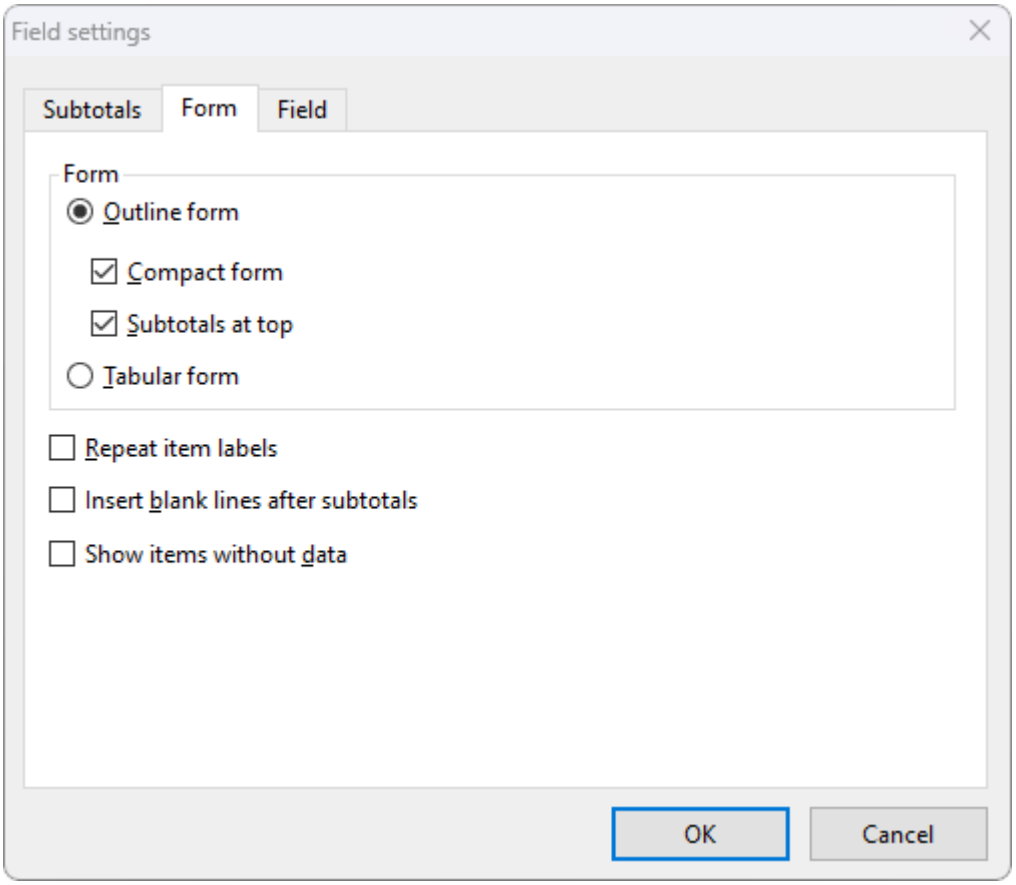
Tip 3: If you want to show or hide the grand total for columns/rows, you will find the options available under Pivot table settings.

Field settings, "Form" tab

In the example figure at the beginning of the section, you can see the pivot table report displayed in the outline view. You can use the **Form** options to customize this view.

Note: The **Form** can only be customized for fields in the *Row labels* area. For fields in the *Column labels* area, the outline forms cannot be set.

In the **Row labels** area (in the areas section of the field list), click on the dropdown menu to the right of the *Employee* field and select **Field settings**. In the open dialog box, select the **Form** tab.



The Field settings dialog box with the default values in the "Form" tab

Form section

Here, you can select between the view in **Outline form** or in **Tabular form**:

- In the **Outline form** – with **Compact form** also enabled – several fields are displayed together in one column, and the items of the lower-level element are indented in the column. Here, you can also select whether the subtotals should appear above or below the grouping.

  The following figures illustrate the different display variants, depending on the enabled options:

  Outline form: *Compact form* enabled; *Subtotals at top* enabled

Row labels	Sum of Amount
Anna	**123**
Coffee	75
Cream	12
Snacks	36
Maria	**30**
Coffee	30
Toni	**47**
Cream	25
Sweetener	7
Tea	15
Grand total	**200**

  Outline form: *Compact form* disabled; *Subtotals at top* enabled

Row labels	Article	Sum of Amount
Anna		**123**
	Coffee	75
	Cream	12
	Snacks	36
Maria		**30**
	Coffee	30
Toni		**47**
	Cream	25
	Sweetener	7
	Tea	15
Grand total		**200**

Outline form: *Compact form* disabled; *Subtotals at top* disabled

Row labels	Article	Sum of Amount
Anna		
	Coffee	75
	Cream	12
	Snacks	36
Anna Total		**123**
Maria		
	Coffee	30
Maria Total		**30**
Toni		
	Cream	25
	Sweetener	7
	Tea	15
Toni Total		**47**
Grand total		**200**

- The **Tabular form** always displays a separate column for each field, with the corresponding field heading:

Row labels	Article	Sum of Amount
Anna	Coffee	75
	Cream	12
	Snacks	36
Anna Total		**123**
Maria	Coffee	30
Maria Total		**30**
Toni	Cream	25
	Sweetener	7
	Tea	15
Toni Total		**47**
Grand total		**200**

Further options in the "Form" tab

There are three other options in the lower section of the Form tab that you can use to give the pivot table report a more structured look as needed.

Please note: The options *Repeat item labels* and *Insert blank lines after subtotals* only affect higher-level field elements (Employee). The option *Show items without data*, however, only affects lower-level field elements (Article).

- **Repeat item labels**

  The item labels are displayed repeatedly in each row. This option only works for the *Tabular form* or in the *Outline form* if you disable the compact form.

Row labels	Article	Sum of Amount
Anna	Coffee	75
Anna	Cream	12
Anna	Snacks	36
Anna Total		123
Maria	Coffee	30
Maria Total		30
Toni	Cream	25
Toni	Sweetener	7
Toni	Tea	15
Toni Total		47
Grand total		200

Enabled option "Repeat item labels" for **Tabular form**

- **Insert blank lines after subtotals**

  The clarity of the pivot table report is improved if a blank row appears after each group of subtotals. Enable the option for this purpose (not available for fields in the column labels area).

- **Show items without data**

  Enable this option to also display those items that did not contain associated data.

  Example: Toni did not buy any *Coffee*. If the option is enabled, this article will still be displayed with an empty cell in "Sum of Amount".

Field settings, "Field" tab

You can assign a **Custom name** to the field here so that it appears in the pivot table with a custom name. Enable the checkbox and type the new name into the input field. For more information, see also Working with the field list: Areas section.

Value settings (Values area)

If you open the dropdown menu to the right of a field in the **Values** area in the lower areas section of the field list, you will find the option **Value settings**. Alternatively, you will find this option via the context menu by right-clicking directly in the Pivot table report (select a cell below the header title *Sum of...*).

The value settings dialog box contains the tabs **Summarize values** and **Show values as**, which provide you with various summary functions and calculation options.

Value settings, "Summarize values" tab

The results in the pivot table report are automatically displayed as a sum if a number field was identified in the *Values* area. If the field has values other than numbers, the results are calculated as a count.

You can customize the display of the results. The specified selection applies to the subtotals of the higher-level field and also to the subgrouped fields.

To do so, click on the dropdown menu to the right of the desired field in the **Values** area (in the areas section) and select the option **Value settings**. In the dialog box that opens, choose the **Summarize values** tab.

In the upper section of the tab, you will find "Summarize values by" with the following options for displaying the values:

- **Sum:** The results of the individual values are displayed as a sum.
- **Count:** This displays the number of individual values that occur. This works in the same manner as the COUNTA function.
- **Average:** This displays the average value of the individual values that occur.
- **Maximum:** This displays the highest individual value that occurs.
- **Minimum:** This displays the lowest individual value that occurs.
- **Product:** This displays the multiplied value of the individual values that occur.
- **Count numbers:** This displays the number of values that are numbers. This works in the same manner as the COUNT function.
- **Standard deviation (sample):** This provides an estimate of the standard deviation of a population based on a sample. This works in the same manner as the STDEV.S function.
- **Standard deviation (population):** This returns the standard deviation of a population. This works in the same manner as the STDEV.P function.
- **Variance (sample):** This provides an estimate of the variance of a population based on a sample. This works in the same manner as the VAR.S function.
- **Variance (population):** This provides the variance of a population. This works in the same manner as the VAR.P function.

Tip 1: If you only want to change the subtotals to a specific display, you can make the required changes in the *Row/Column labels* area with the option **Field settings**. For more information, see Field settings (Row/Column labels area).

Tip 2: If you want to show or hide the grand total for columns/rows, you will find the options available under Pivot table settings.

Furthermore, you will find the option **Custom name** in the tab. Here, you can rename the field so that it appears in the pivot table with a different name. For more information, see also Working with the field list: Areas section.

Value settings, "Show values as" tab

In the **Show values as** tab, you will find several options for displaying the results of the pivot table, such as percentage values.

Tip: You can also drag the same field several times into the *Values* area. If you then apply different output settings to the copy of the field using **Show values as** (for example, standard values for one field and percentage values for the other field), you can clearly compare the different output values in the pivot table report.

In the upper section of the tab, you will find the section **Show values as** with options for calculating the values. Depending on the calculation type, additional selectable entries appear in the *Base field* and *Base item* lists below.

- **Standard (no calculation):** This displays the value as usual.
- **% of grand total:** Each item is displayed as a percentage of the grand total.
- **% of column total:** For each column, the total is set as a 100% reference, and each item in the column is shown as a percentage of that total.
- **% of row total:** For each row, the total is set as a 100% reference, and each item in the row is shown as a percentage of that total.
- **% of:** A column or row field is set as a 100% reference, and all other values are shown as a percentage of this reference value. The desired entries must be selected from the Base field list and Base item list.
- **% of parent row total:** This displays the percentage of an item relative to the higher-level item (parent) in the row structure.
- **% of parent column total:** This displays the percentage of an item relative to the higher-level item (parent) in the column structure.
- **% of parent total:** This displays the percentage of an item relative to any (selectable) higher-level item in the structure. The desired entry must be selected from the Base field list.
- **Difference from:** This displays the absolute difference of the values from the selected reference value. The desired entries must be selected from the Base field list and Base item list.
- **% difference from:** This displays the percentage difference of the values from the selected reference value. The desired entries must be selected from the Base field list and Base item list.
- **Running total in:** This displays the cumulative absolute values of the successive items. The desired entry must be selected from the Base field list.
- **% running total in:** This displays the cumulative percentage values of the successive items. The desired entry must be selected from the Base field list.
- **Rank smallest to largest:** This displays the rank of selected values in a specific field. The smallest item in the field is ranked 1. The desired entry must be selected from the Base field list.
- **Rank largest to smallest:** As above, but the largest item in the field is ranked 1.
- **Index:** This displays the weighting of the values in relation to the grand row total and grand column total. The higher the index value, the greater the weighting of the value on the grand row/column total. The following formula applies to the index value:

  $$\frac{\text{value in cell x grand total of the pivot table}}{\text{grand row total x grand column total}}$$

You can use the **Number format** button on this tab to *permanently* adjust the display of your desired number format. For information on the various number formats, see List of number formats.

Note: Changes to the **Number format** will only be permanently retained if you do so here. If you change the number format using the cell formatting only, the changes are discarded when the data is refreshed. The number formatting of the source range is also not adopted in the pivot table.

Pivot table settings

In the field list below the areas section with the 4 areas, you will find the **Pivot table settings** button. You can also access the command via the context menu by right-clicking directly in the pivot table report.

You can make further adjustments to the structure of your pivot table here, which you cannot find in the field settings or value settings, since pivot table settings are of a general nature. Unlike the field settings and value settings, the changed options in the pivot table settings will remain in effect when you remove the fields from the areas section.

Layout tab

- **Show totals for rows/columns:** This shows or hides grand totals for rows/columns.
- **Indent row labels...character(s):** If you have arranged several fields in the *Row labels* area and the outline form with compact format is set in the Field settings, the lower-level field is displayed indented below it. Here, you can now specify the extent of the indentation from the left edge of the column (default is 1 character).
- **Autofit column width:** If this option is enabled, the column width of the pivot table automatically adjusts to the text of the columns so that the text is not truncated by the next column. This has its advantages, but the disadvantage is that manual adjustments to the column width may be discarded when changes are made to the fields. Disable this option to retain the column width that you set manually.
- **Preserve cell formatting:** If this option is enabled, the formatting you made manually in the pivot table (for example, using a different font, shading, border) will be retained. If the option is disabled, such formatting changes will be discarded as soon as you update the pivot table or reopen the workbook.

  Note: Number formatting must be set using the **Number format** button (in the Value settings on the **Show values as** tab) in order to be permanently retained.

- **Classic pivot table format:** If you are confused by the headers "Row labels" and "Column labels" in the pivot table report and would prefer to display the real names of the fields instead, enable this option for a classic representation (known from older versions of Microsoft Excel up to 2003).
- **Show the values row:** Enable the checkbox to insert an additional row with the heading "Values" above the pivot table report. Note: The values row is displayed only if there are at least two fields in the *Values* area (at the bottom of the field list) and cannot be enabled for the classic pivot table format.

Filter tab

- **Report filter** section: You can combine the following options *Display fields* and *Fields per row* to rearrange the placement of report filters in the pivot table report.

  Display fields – If there are at least two fields in the report filter area, you use this option to specify how the fields in the report filter area are arranged in the pivot table report. You can choose between *First down, then right* or *First right, then down*.

  Fields per row – Here, you can also specify for the report filters of the pivot table report how many fields are displayed per column before a new column starts.

- **Field list** section:

  Sort A-Z/Sort in data source order – The field names in the field list are sorted according to the order of the columns from your source data. You can also display the fields in alphabetical order to find specific field names in the fields section more quickly.

  Note: The order specified here does not affect the display in the pivot table report. For information on the alphabetical sorting of elements directly in the pivot table report, see Sorting and filtering pivot tables.

- **Display** section

  Show field captions and filter dropdowns – The field names and filter symbols for row labels and column labels are displayed/hidden in the pivot table report.

- **Use fill lists when sorting:**

  If this option is enabled, the field items of the pivot table are sorted using the lists defined for automatic filling. On the one hand, these are the lists predefined in PlanMaker (for example Mon;Tue;Wed;... for the sequence of weekdays) and, on the other hand, lists you create yourself. For more information on how to create such lists, see Editing lists for automatic filling.

  If, based on our example table, you would like the items of the *Article* field to always be displayed sorted in a certain order Tea;Cream;Snacks;Coffee;Sweetener, first create a corresponding custom fill list. Then click on the **Refresh data** button located below the areas section of the field list.

  Note: You can only use text for fill lists but not numbers.

  If the option is disabled, the lists will not be used for automatic filling and the data will be sorted alphabetically (thus, Coffee;Cream;Snacks;Sweetener;Tea in this example).

Style tab

You can customize the visual appearance of the pivot table report in various ways, just as you do with normal table styles. Select a style source *(Excel 2007, Excel 2003, Excel 97)* from the upper left list in the **Style** section, and choose one of the pivot styles with different formatting from the list below. For the Excel 2007 style templates, the selection *light/dark/medium* is also available in the upper right list.

The options *Banded rows*, *Banded columns*, *Row headers* and *Column headers* are displayed automatically adapted to the style template. For the Excel 2007 style templates, you can manually modify these options to your liking.

Enabling the option **Banded rows/Banded columns** causes alternating background colors of the rows/columns, while enabling the option **Row headers/Column headers** highlights the first row/column of the pivot table.

The change of the design is displayed directly in the pivot table report as a preview and has to be confirmed with **OK** in order to be finally adopted.

Data tab

- **Special values** section:

  Show for error values – Cells with incorrect calculations return the known error values also in the pivot table report (for example, #N/A). For more information, see Error values). Enable this option and enter an individual text in the right input field to display self-defined output values that represent the error values.

  Show for empty cells – If no data can be assigned to an item (this usually occurs with combinations of row and column labels for cross-classified tables), an empty cell is displayed in the pivot table report. If you disable the checkbox, a 0 is displayed instead of empty cells. If the checkbox is enabled, you can also enter an individual text in the input field next to it to display a self-defined output value instead of an empty cell or 0.

- **Pivot table data** section:

  Save source data with file – If this option is enabled, the source data is stored in an internal cache and is available from the cache for the pivot table independently of your source data. In a way, a copy of the source table is created. This is particularly useful for external source data: If it no longer exists, you can still work on your pivot table with the data from the pivot cache.

  If you disable this option, your data will not be stored in the pivot cache. When you re-open the file, you have to click on the *Refresh data* button (see Updating pivot tables and changing data area) to initialize the pivot cache for the pivot table with the source data. In the case of external source data, the following applies: If it no longer exists, your pivot table cannot be updated either.

  Refresh when opening the file – If this option is enabled, the pivot table data is automatically updated to current source data when you open the file. If you disable this option, you can continue working with your old pivot data until you trigger the *Refresh data* button. For more information, see Updating pivot tables and changing data area.

  Please note: Updating the data with the option *Refresh when opening the file* or with the *Refresh data* button will replace the pivot cache with updated source data, thus overwriting the previous data in your pivot table.

Text tab

On the **Text** tab, you can enter a custom **Name**, a **Title** and a **Description** for your pivot table. The name is automatically generated by PlanMaker in sequence when a pivot table is created (*Pivot table 1*, *Pivot table 2*, etc.) and is shown as a title in the blank pivot table report. An entry is also created in Named ranges, but it cannot be renamed there. Furthermore, the assigned *Name* cannot be used to identify the pivot table from the GETPIVOTDATA function; thus, you must explicitly create a new entry for the pivot table in the *Named ranges*.

Apart from that, the entries from the **Text** tab do not appear anywhere in the pivot table. *Title* and *Description* can be useful as internal information. To rename the fields that actually appear in the pivot table, see Working with the field list: Fields section.

Defer layout update

At the bottom of the field list, you will find the option **Defer layout update**. It is combined with the **Update** button on the right, but is not related to the option **Refresh data**. For more information, see Updating pivot tables and changing data area.

When you reposition fields in the field list, for example, add/remove/move fields in the areas section, the pivot table report is recalculated each time. This can take quite some time with large reports. Thus, if you want to temporarily disable continuous recalculation, please proceed as follows:

1. Place a check mark in the **Defer layout update** checkbox. You can then reorder the fields of your pivot table, but you will not initially see the resulting changes in the pivot table report. In addition, all other editing options for this state are disabled.
2. Once you have finished repositioning the fields, click on the **Update** button to see the result of the change.
3. Remove the check mark from the **Defer layout update** checkbox.

Note: If you disable the option **Defer layout update** again without clicking on **Update**, all arrangements of the fields that you have made since the option was enabled will be discarded.

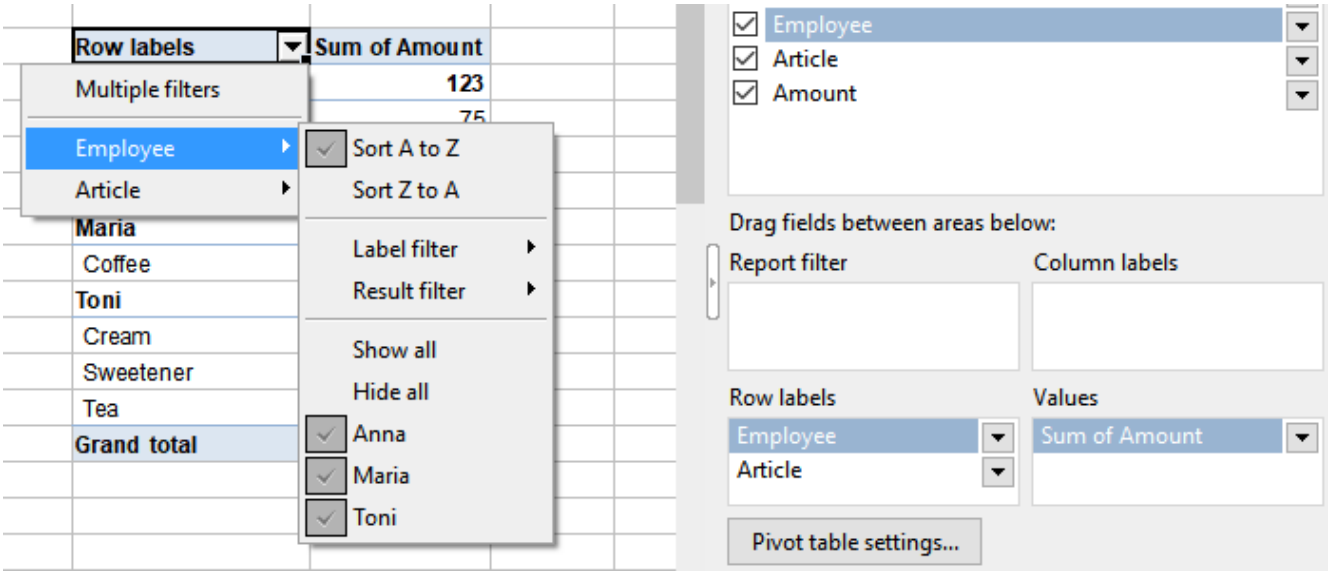
Sorting and filtering pivot tables

Too much data makes it difficult to see the relevant content. Thus, there are various options for sorting and filtering table fields in pivot tables:

- Sorting field items alphabetically in the pivot table report via the dropdown menu
- Sorting field items in the pivot table report by fill lists
- Filtering in the areas section with the option *Filter labels (Filter values)*
- Filtering by adding *Report filters*
- Setting additional special filters directly in the pivot table report via the dropdown menu

Please note: For all presented options of sorting and filtering, the changes will be discarded as soon as you have removed the field from the areas section or moved it to another area.

Sorting field items alphabetically in the pivot table report



You can sort the field items of a pivot table report alphabetically by opening the dropdown menu in the report itself at the header *Row labels/Column labels* and choosing from the list:

- **Sort from A to Z** (alphabetical order)
- **Sort from Z-A** (reverse alphabetical order)

If there are several fields in the row/column labels area (as in the figure above), you can sort each field separately. If you move the mouse in the dropdown menu to the desired field, the program will open a submenu with the sorting options of this field.

Note: For the alphabetical display of the fields only in the field list, see Working with the field list: Fields section.

Sorting field items in the pivot table report by fill lists

You can have the field items of the pivot table sorted according to the lists defined for automatic filling. To do so, first create the relevant custom fill lists as described in Editing lists for automatic filling.

To use these fill lists for sorting the field items, enable the option **Use fill lists when sorting** in the Pivot table settings, Filter tab. If this option is not enabled, the field items will be sorted alphabetically, as described above.

Filtering in the areas section with the option Filter labels (Filter values)

In the *Row labels/Column labels* areas of the field list, go to the desired field whose items you want to filter. Open the dropdown menu to the right of the field and select the option **Filter labels** (do the same for fields in the *Report filter* area, but the option is called **Filter values** here). You can also select the option directly from the pivot table report by right-clicking on the desired field and opening the context menu.

In the **Filter** dialog box that opens, you can select the following:

- Select **individual list items** that you want to show/hide by placing/removing a check mark in front of them.
- The **Show all** button selects all items.
- The **Hide all** button deselects all items.

Confirm your selection by pressing **OK**.

In the pivot table report, the black dropdown arrow next to the header changes to a red arrow, indicating that a filter is active. The filtered out values are no longer displayed in the pivot table report.

Filtering by adding report filters

The **Report filter** is located in the lower areas section of the field list and offers you a particularly simple and clear option for filtering data.

Drag the fields by which you want to filter to the report filter area. A cell with the field name is then inserted above the pivot table report and another cell *(All items)* with a dropdown arrow is integrated to the right of it. The items of the field can be filtered via this dropdown menu. You can select **individual list items** here or select/deselect the complete list of items with **Show all/Hide all**.

The red dropdown arrow to the right of the cell indicates that a filter is active. The filtered out values are no longer displayed in the pivot table report.

You can drag several fields from the field list to the Report filter area and filter any combination of individual items from the various fields for the pivot table report.

Tip: The positioning of the report filter fields in the pivot table report can be changed using the Pivot table settings, Filter tab.

Setting additional special filters directly in the pivot table report

In the pivot table report, open the dropdown menu to the right of the header *Row labels/Column labels*. If there is only one field in the Row labels/Column labels area, the choice of filter will be displayed directly for the selected field.

If there are multiple fields in the Row/Column labels area, you can apply separate filters to each field in the pivot table report. If you move the mouse to the desired field in the dropdown menu of the header, the program will open a submenu with the filter selection of the field.

- You can select **individual list items** or select/deselect the entire list of items with the options **Show all/Hide all**.

  Note: If the list has too many entries, the entry **More** appears at the end of the list. Clicking on this entry will take you to the dialog box **Filter** with the complete list of all items.

Furthermore, additional special filter conditions are available in the dropdown list. For detailed descriptions of the individual filter conditions, see AutoFilter):

- **Label filter** or **Date filter:** Depending on whether you have selected a field with numbers/text or date values, one of the two filters is automatically offered. Depending on the filter, you can filter numbers/text or date values according to specific conditions.
- **Result filter:** The result filter has special filter conditions that you can use to narrow down the amounts in the value fields.

  Note: When applying filters to multiple fields, please note the order in which you set the filters. The next filter that is applied refers only to the remaining results of the previously set filter.

- **Multiple filters:** The filters in pivot table reports can be applied additively, that is, you can set another filter on the currently effective filter of the same field. To do so, however, you must enable the option *Multiple filters* to combine filters for individual list items, label/date filters and result filters for the same field. If this option *Multiple filters* is not enabled, a newly applied filter replaces the old one.
- **Clear filter:** If you want to remove the applied filters from the label filter/date filter or result filter, select *Clear filter* from the dropdown list of the relevant filter.

  Tip: If you want to remove all filters at once, disable the entry **Multiple filters**. In the following dialog box **Reset all filters?**, confirm with **OK** to remove all filters.

In the pivot table report, the red dropdown arrow next to the header indicates that a filter is set. The filtered out values are no longer displayed in the pivot table report.

Grouping pivot tables

You can use the **Group** function to display data even more clearly. For example, group the dates in your pivot table report by years, quarters or months to make time periods more meaningful.

You will find the command for this as the **Group...** button at the bottom of the pivot table sidebar.

An example:

You have created a pivot table from the familiar sample data (see Creating a new pivot table).

Then, proceed as follows:

1. In the pivot table sidebar, drag the *Date* field to the **Row labels** area and the *Amount* field to the **Values** area.
2. In the pivot table report, select a cell with a date value.

   Note: Only one cell may be selected at a time.

3. Click on the **Group...** button in the pivot table sidebar.
4. In the dialog box that now appears, select the entry *Quarters* from the list, for example.
5. In the dialog box, confirm with **Group**.

The pivot table report then shows the following result: The amounts are summed quarterly.

	Date	Employee	Article	Amount		Row labels	Sum of Amount
2	01/10/2018	Anna	Coffee	$35		Qtr1	50
3	02/15/2018	Toni	Cream	$15		Qtr2	77
4	04/13/2018	Anna	Snacks	$20		Qtr3	23
5	05/15/2018	Toni	Tea	$15		Qtr4	50
6	06/18/2018	Maria	Coffee	$30		Grand total	200
7	06/20/2018	Anna	Cream	$12			
8	08/11/2018	Anna	Snacks	$16			
9	08/18/2018	Toni	Sweetener	$7			
10	10/28/2018	Anna	Coffee	$40			
11	11/15/2018	Toni	Cream	$10			
12							

The "Group..." function in detail:

Depending on the contents of a selected cell, you group *date values*, *numeric values* or *text*. Multiple groupings in the same pivot table are possible, both for *Row labels* and *Column labels*.

After you have selected a cell in the pivot table and clicked on the **Group...** button, one of the following dialog boxes opens, each with different options:

- The dialog box **Group by dates** appears if the selected cell contains date values:

  With the **Auto** options enabled, the date values are preset to a range with the first and last date of all records and are thus grouped for this period. If you want to customize the time period, disable the **Auto** checkbox in front of the input fields **Starting at** or **Ending at** and enter the desired values there.

  In the list below, select the time unit (*Months*, *Quarters*, etc.) by which you want to group. Click with the mouse on the corresponding entry from the list, which is then highlighted in blue and is thus selected. To deselect, click again on the entry highlighted in blue. Tip: You can also select multiple time units by just

clicking on another entry. For example, it may make sense to select *Days* as well as *Quarters*. The days are then shown in the pivot table for the relevant quarter.

Once you have made your choice, press **Group** in the dialog box.

Note: The input field **Number of days** is only available if you have selected *Days* as the only unit of time in the list above. This option causes the pivot table to display the specified number of days as a time interval from...to and is especially useful for displaying the results of calendar weeks. To do so, enter the number 7 and disable the two Automatic options above. For **Starting at**, select the Monday before the oldest date of your source data (this would be 01/08/2018 in the example above) and, for **Ending at**, the Sunday after the most recent date (1/18/2018 in the example).

- The dialog box **Group by numbers** appears if the selected cell contains numeric values:

  With the **Auto** check boxes enabled, the numeric values are preset to a range with the smallest and largest numeric value of all records. The numeric values in this range are thus grouped. If you want to customize the range, disable the **Auto** checkbox in front of the input fields **Starting at** or **Ending at** and enter the desired values there.

  You use the option **By** below to determine at which intervals the numeric values are to be grouped. You thus receive a summarization to value classes. If, for example, you enter 0 for **Starting at**, 40 for **Ending at**, and 10 for **By**, you will obtain the classes 0-19, 10-19, 20-29, 30-39 by which your records will be grouped.

  Finally, press **Group** in the dialog box.

- The dialog box **Group text** appears if the selected cell contains text:

  In the *Labels* list on the left, select the items that you want to group by clicking on them with the mouse. The entry now appears highlighted in blue. To deselect an item, simply click on it again.

  Important: In the input field *Group name*, first type a name. It is only then that you can then use the **New group** button to create a grouping for the selected items. You can then add more groups in the same way by selecting items from the left list again and entering a group name (ensure that you confirm with **New group**).

  In the *Groups* list on the right, you will see the result of the created group(s): The group name at the top and the items it contains below.

  If you want to rename a group, select the corresponding group in the *Groups* list on the right. ***Important:*** In the *Group name* field, type the new name first, and then click on the **Rename group** button.

  Use the buttons >> and << to add or remove items from the groups. To do so, select the desired group in the *Groups* list on the right, and then select the item that you want to add (from the list on the left) or remove (from the list on the right).

  Finally, click on **OK** in the dialog box.

Ungrouping

To reset grouped data, select a cell of the grouping in your pivot table report. Click on the **Ungroup** button in the pivot table sidebar. Only the grouping of the field to which the selected cell belongs is reset.

Alternatively, you can click on the **Group...** button to open the dialog box again and click on the **Ungroup** button there.

Only for cells with *text*: If you have created several groups for a field with text, you can remove a single group via the dialog box by selecting a group in the dropdown list on the right and using the **Ungroup** button. The

Ungroup all button, on the other hand, ungroups all groupings for this field (corresponds to the **Ungroup** button in the sidebar).

Updating pivot tables and changing data area

The following generally applies to pivot tables: If you subsequently add new data to the source table, this initially has no effect on the pivot table. Due to PlanMaker's default values for *pivot table settings* (see Pivot table settings, Data tab), the data of the original source table is stored in an internal pivot cache after creating the initial pivot table. The pivot table accesses the data from this cache.

It is only when you *Refresh data* that the pivot cache will be overwritten and the pivot table will be updated.

In the **Pivot table settings, Data tab**, you will find the option **Refresh when opening the file**. If you enable this option, the pivot cache is regularly overwritten with the current data of the source range when you open the file. If you want to continue working with your old pivot data after opening the file, leave this option disabled.

You can manually update your pivot table at any time by clicking on the **Refresh data** button.

Please note: By updating the data with the *Refresh data* button or with the option *Refresh when opening the file*, the pivot cache will be replaced by updated source data, thus overwriting the previous data in your pivot table.

"Refresh data" button

You will find the **Refresh data** button below the areas section in the field list.

Depending on how you added new data in the source data area, this affects the pivot table differently when it is refreshed:

- If you overwrite data within the source data area, the changed data will be used in the refreshed pivot table.
- If you insert a new row or column within the source data area, the data area will **not** be automatically extended. Thus, the refreshed pivot table only takes into account the data that is in the previous source data area. You thus must adjust the source data area again (see "Changing the source data area" below).

  The same applies if you add a new row or column at the end of the source data area.

Refresh all pivot tables

If you want to update all existing pivot tables – even with different source ranges – use the ribbon command **Formula** | group **Update** | **Update data** | **Update pivot tables**.

Changing the source data area

iOS: In this version, the command to change the source data area is not freely available.

If your source data has changed to such an extent that it no longer corresponds to the original source range of the pivot table (new records have been inserted/appended or the file with imported data has been moved), the referenced data area for the source data must be changed.

To do so, proceed as follows:

1. Place the mouse cursor in the pivot table report.
2. Choose the ribbon command **Insert** | group **Tables** | **Pivot table** again.

   Alternatively: On the ribbon tab **Data** | group **Analyze** | **Pivot table** or simply by right-clicking in the Pivot table report with the command **Modify pivot table areas** from the context menu.
3. The program opens a dialog box that is similar to the one for creating a new pivot table but with one difference: The *Insert* button is called *Change* here. In addition, the originally referenced source range is highlighted with a blue border directly in the worksheet with the source data.
4. In the dialog box, enter the new data area in the input field (source range); the previous source range is offered by default. Alternatively, you can drag the desired source range with the mouse if you have the worksheet with the source data in front of you. To do so, grab the blue selection at the bottom right with the left mouse button.

   Note: For external data, the adjustment is only possible in the input field (source range).
5. Confirm with the **Change** button. Before you do so, you can enter in the lower input field whether you also want to move the target range to another position; the previous target range is offered by default.

Deleting, copying, moving pivot tables

You can also delete, copy and move your created pivot tables. You will find out how to do this in the following paragraphs.

Tip: To delete, copy or move, it is always necessary to select the entire area of the pivot table report exactly. The easiest way to do this is to place the mouse in a cell of the report and use the key combination **Ctrl+A** to select it. Alternatively, choose the ribbon command **Home** | group **Selection** | **Select all**.

Deleting pivot tables

Deleting pivot tables works in a similar manner to the standard procedures for deleting. Note, however, that the pivot table report must be selected as a whole; individual cells cannot be deleted separately.

Thus, select the entire pivot table report and delete it using one of the following methods:

- Press the **Del** key on the keyboard
- Use the ribbon command **Home** | group **Contents** | **Delete** or open the context menu by right-clicking with the mouse on the pivot table report and selecting **Delete special**.

  In the submenu of these commands, you can use **All** or **Contents** to completely delete the pivot table. Select the entry **Formats** if you only want to remove the applied formats from the pivot table.

Copying pivot tables

You can copy pivot tables at any time and duplicate them in the same worksheet in a different place or in a new worksheet. The copied pivot table will have the same values as the original. Even after you click on the *Refresh data* button, the copied pivot table adopts changed data from the source table.

In the copied pivot table, however, you still have the option of configuring it independently of the copied original with regard to field arrangement, formatting, etc. This gives you different views of the reports for the same source data.

Select the entire pivot table report to copy it and proceed according to one of the following methods:

- Use the ribbon command **Home | Copy** and then **Home | Paste**
- Right-click on the pivot table report to open the context menu and select **Copy** and then **Paste**
- Drag the pivot table report with the mouse to the desired position and press the **Ctrl** key to drop it before releasing the mouse.
- Alternatively, simply use the key combination **Ctrl+C** and **Ctrl+V**

You can now decide which report you want to work on by clicking in the respective pivot table report. The program displays the relevant field list that corresponds to the selected report.

Moving pivot tables

Select the entire pivot table report and use the left mouse button to drag the selected range to a free position in the worksheet. Ensure that the report does not overlap the source data area.

If you want to move the pivot table report to another worksheet, proceed as follows:

- Choose the ribbon command **Insert** | group **Tables | Pivot table**. Alternatively, right click in the pivot table to open the context menu and choose the command **Modify pivot table areas**. In the dialog box, enter the target range and the desired worksheet (see Updating pivot tables and changing data area).
- Alternatively, copy the pivot table report as described above, switch to a free space in the desired worksheet and paste the copy from the clipboard. Then remove the remaining pivot table report from its former location.

Some restrictive notes on copying and moving source/target range

Copying and moving data ranges – both the source range and the target range (the pivot table report) – is not fully possible in the same way as is usual when working with normal tables. You should note the following rules:

- **Copy & paste** the source range/target range: Although you can first copy the source range of your data and paste it to another position, the pivot table is still referenced to the position of the original source range and not to the new position. This means that the former cell range is still addressed when the data is refreshed, and if it has been removed, an error message appears. This restriction also applies if you copy and paste the source range together with the target range. Copying and pasting only the target range to another position can be easily done.
- **Drag & drop** the source range/target range: If you drag the selected source range to another position with the mouse, the aforementioned restriction does not apply. The pivot data will be referenced to the new cell range and will be correctly addressed when the data is refreshed. Dragging and dropping only the target range or together with the source range can also be easily done.
- **Cut & paste** the source range/target range: Cutting is neither possible for the source range nor for the target range. An error message appears when you use this command.

In the following table, you can see the options once again clearly summarized:

Action	Source range	Target range
Copy & paste	✘	✓
Drag & drop	✓	✓
Cut & paste	✘	✘

Freezing rows and columns

You can choose the ribbon command **View** | group **Windows** | **Freeze cells** to fix the first rows and/or columns of a table on the screen. This means that they no longer move when scrolling through the table, but permanently stay in place.

If, for example, you have entered labels for rows or columns of a worksheet, you can freeze those labels so that they are always visible.

Enabling freezing

To freeze rows or columns, you can use the following options:

- **Only freeze the top row:** Use the small arrow of the icon **Freeze cells**, and select the entry **Freeze top row**. It does not matter which cell you have previously selected.
- **Only freeze the first column:** Use the small arrow of the icon **Freeze cells**, and select the entry **Freeze first column**. It does not matter which cell you have previously selected.
- **Freeze the top row and first column:** Select the aforementioned two commands in succession to combine freezing the top row and the first column.
- **Freeze any number of lines:** If you want to freeze the first rows (without freezing columns), proceed as follows:

  In the row header on the far left, select the row *directly below* the rows that you want to freeze. Then click directly on the **Freeze cells** icon.

  Alternatively: Use the icon's small arrow and select the entry **Freeze at current position**.
- **Freeze any number of columns:** If you want to freeze the first columns (without freezing rows), proceed as follows:

  In the column header at the top, select the column to the *right* of the columns that you want to freeze. Then click directly on the **Freeze cells** icon.

  Alternatively: Use the icon's small arrow and select the entry **Freeze at current position**.
- **Freeze any number of rows and columns:** If you want to freeze several first rows and columns, place the cell frame in the cell *right below* the range that you want to freeze. Then click directly on the **Freeze cells** icon.

  Alternatively: Use the icon's small arrow and select the entry **Freeze at current position**.

The rows and/or columns are now frozen. They remain in their original location when you scroll through the worksheet.

Tip: You can also use the icon in the bottom right corner of the document window to freeze rows or columns. Simply click on this icon and then drag it to the desired location.

Disabling freezing

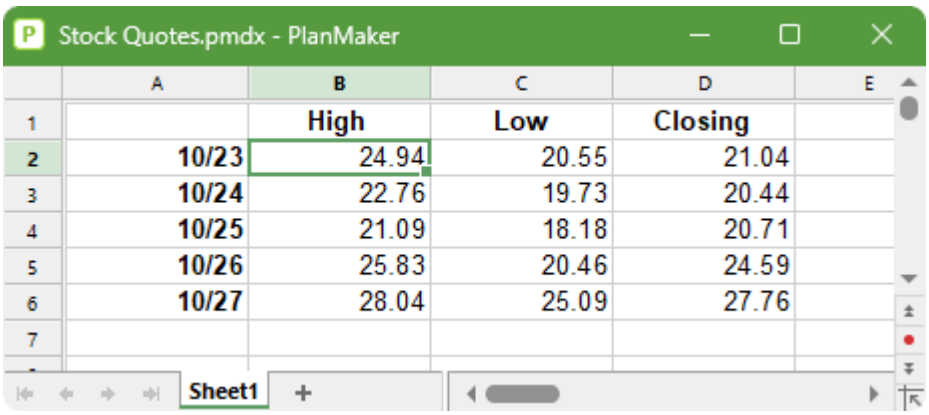
If you have frozen rows or columns, the **Freeze cells** icon is darker in order to indicate that freezing is enabled. To disable it, click directly on the icon itself – the rows/columns will no longer be frozen.

Alternatively: Use the icon's small arrow and select the entry **Unfreeze cells**.

Tip: A single mouse click on the icon in the bottom right corner of the document window will also disable freezing.

An example

Assume you have the following worksheet:



Note that both the columns and the rows are labeled. To freeze the labels, proceed as follows:

- The *column* labels (High, Low, etc.) are in the first row of the table.

  To freeze them, select the second row by clicking on its row header (the button labeled "2" to the left of the row).

  Then choose the ribbon command **View | Freeze cells**.

- The *row* labels (10/23, 10/24, etc.) are in the first column of the worksheet.

  To freeze them, select the second column (column B) by clicking on its column header (the button labeled "B" above the column).

  Then choose the ribbon command **View | Freeze cells**.

- To freeze *both* rows and columns, click on cell B2 and choose the ribbon command **View | Freeze cells**.

Tip: Since the top row and first column are involved in this example, you might as well use the commands **Freeze top row** and **Freeze first column** by clicking on the arrow of the icon **Freeze cells**.

If you want to unfreeze, click on the **Freeze cells** icon again.

Inserting special characters

Some characters, such as the copyright sign, the degree sign and the accented characters used in some languages, cannot be entered into text directly with the keyboard. With PlanMaker, you can easily insert these *special characters* via the built-in character table.

To do so, proceed as follows:

1. Open the dialog box via the ribbon command **Insert | Symbol** (in the group **Text**) by clicking directly on the icon.

   Tip: If you click on the icon's arrow rather than directly on the icon itself, a small selection of the last used special characters will open for quick use. If you click on **More** here, you will open the dialog box again.

2. The dialog box displays all characters available in the currently selected font. You can select a different font from the dropdown list **Font** if necessary.

3. You can use the dropdown list **Set** to jump to a specific section of the character set. For example, you can select **Cyrillic** to view the Cyrillic characters (provided the selected font contains such characters).

   Note: Many fonts contain only a few of the available character sets. A particularly large reservoir of characters is available in the standard fonts supplied with your operating system. In Windows, for example, these include such fonts as Arial, Tahoma and Times New Roman.

4. Select the character to insert with the arrow keys and press **Enter**↵ or double-click on the character to insert it into the text.

   Tip: If you know the Unicode of the character you want to insert (for example, 20AC for the Euro sign €), you can enter it in the **Code** field. The program will then jump directly to the corresponding character.

5. You can now insert more characters in the same way or exit the dialog box with the **Close** button.

Tip for Windows users: Most versions of Windows include two icon fonts named *Symbol* and *Wingdings*. These fonts contain numerous useful symbols (phone symbols, bullets, etc.).

Using shortcut keys

If you need to insert a special character frequently, you can always give it a shortcut key. The advantage of doing so is that you can then apply the character very quickly with one press of a key.

To assign a shortcut key to a special character, proceed as follows:

1. Choose the ribbon command **Insert | Symbol** to open the dialog box.

2. Select the special character to which you want to assign a shortcut key.

3. Click on the **Change** button next to the option **Shortcut key**.

4. Another dialog box appears. In the **Press new shortcut key** input field, press the desired key combination (starting with **Ctrl** or **Alt**).

   We recommend that you always use key combinations that include the keys **Ctrl** *and* **Shift** since they are not usually assigned. If you press a key combination that is already assigned, its current assignment will be displayed just below the **Press new shortcut key** input field. You should then press the **Backspace key**⌫ to delete your shortcut key and use a different key combination. Otherwise, you will overwrite the key combination of another PlanMaker command.

5. **Don't forget:** Click on **Add** to assign this shortcut to the character.
6. Confirm with **OK** and exit the main dialog box with **Close**.

From now on, you can insert this special character by pressing the relevant key combination.

Note: The shortcut key only remembers the selected character but not the selected font.

Inserting special characters via their character code

There is another method for inserting special characters: by typing in its hexadecimal character code (Unicode) and then pressing the key combination **Ctrl+Alt+Shift+X**.

If, for example, you type 20AC and then press this key combination, you will get a euro sign € because it has the character code 20AC in the Unicode character table.

Tip: The Unicode value will be displayed in the aforementioned dialog box when you click on a special character.

Inserting special characters via Alt + numeric keypad

Here is yet another method for inserting a special character, via its ASCII code: Press and hold down the **Alt** key and enter the respective sequence of numbers (only possible via the numeric keypad!). Please also ensure that the numeric keypad is enabled by pressing the **Num** key.

If, for example, you type Alt+0128, you will obtain a euro sign € as soon as you release the Alt key.

Tip: You can display the ASCII code of a special character in Windows via the search term "Character map".

Mac/Linux: On some systems (including macOS and several Linux distributions), these shortcut keys do not work.

Applying selected formatting and special characters via the "Character" icon

The ribbon command **Insert | Character** (in the group **Text**) opens a dropdown menu with some special formatting and special characters, such as:

- Soft hyphen / non-breaking hyphen
- Non-breaking space
- Dashes and spaces of different widths
- Ellipsis and some other useful special characters.

Tip: If you click on the icon ⌨ on the far right of the character in the dropdown menu, you can assign your own shortcut key to the character.

For more information on changing the keyboard mapping, see Customizing shortcut keys. You can edit the shortcut keys for special characters described in this section by selecting the "Characters" entry in the upper left list *Categories* in the dialog box.

Formatting worksheets

You can use charts, colors, borders, and other design options to enhance the appearance of your worksheets and increase their legibility.

In this chapter, you will learn everything about designing appealing worksheets. It covers the following topics:

- **Cell size**

  To change the cell size, use the commands of the ribbon tab **Layout** | group **Cells**. Alternatively, simply drag the row header (or column header) using the mouse.

- **Number format**

  PlanMaker can display numbers in many different number formats: from simple numbers with or without decimal places via currency, percentage and date formats to freely definable formats. Use the commands of the ribbon tab **Home** | group **Number** to select the appropriate number format.

- **Borders**

  You can use the ribbon command **Home** | group **Format** | **Borders** to add a border or individual lines to cells.

- **Shading**

  The background of cells can be shaded. The ribbon command **Home** | group **Format** | **Shading** lets you apply a variety of colors, shading and patterns to the background of cells.

- **Alignment**

  You can use the commands of the ribbon tab **Home** | group **Alignment** to align cell contents (left-aligned, right-aligned, etc.) and rotate them.

- **Protection**

  Furthermore, you can use the dialog box of the ribbon tab **Home** | group **Alignment** ◢ (**Protection** tab) to *protect* cells so that they cannot be overwritten or changed, for example. For more information, see also Sheet protection.

- **Character format**

  The commands of the ribbon tab **Home** | group **Character** let you specify the character format of cells or individual characters. This includes settings such as the font, font styles **(bold**, *italics*, etc.) and the font color.

- **Paragraph format (text frames only)**

  If you have inserted *text frames* into the worksheet, you can change the paragraph formatting (line spacing, indents, etc.) of the included text at any time. This can be done using the dialog box *Paragraph* of the ribbon tab **Home** | group **Alignment** ◢ (only available in text frames).

- **Character styles**

  Character formats that you use very frequently can be stored in a character style, which can then be applied as often as you like.

- **Cell styles**

  You can also create cell styles. Both the character format (font, font size, etc.) and the cell format (number format, borders, shading, alignment and cell protection) can be saved in these cell styles.

- **Document templates**

  PlanMaker even lets you create your own document templates. Document templates can include your own character styles, your own cell styles and any kind of content.

- **AutoFormat**

  The command **AutoFormat** enables you to format a range of cells by applying predefined formats.

  Note: The command **Insert | Table** is even more powerful for applying different table styles. For more information, see Tables in worksheets.

- **Conditional formatting**

  You can also apply *conditional* formatting to cells. This will only be applied if a certain *condition* is met. Thus, for example, you can specify that a cell should be colored red whenever it contains a value greater than 1000.

- **Input validation**

  When you add input validation to cells, only the specified types of values (for example, only numbers) within the specified limits (for example, only values between 10 and 20) will be considered valid in these cells.

- **Format painter**

  With the ribbon command **Home** | group **Edit | Format painter**, you can quickly transfer the formatting of one cell to other cells using a "brush".

- **Page setup**

  The page setup command allows the user to determine the appearance of the document when it is printed. It includes settings such as paper size, page margins, headers and footers, and much more. Use the ribbon command **File** | group **Print | Page setup** to change these settings.

For more information on these topics in the above order, see the following pages.

Cell size

On the following pages, you will learn how to change the size of cells.

Important: If a cell displays hashes (for example, ########) instead of the cell content, the column is too narrow to display the cell content. If this occurs, simply widen the column accordingly.

Changing cell sizes via the mouse

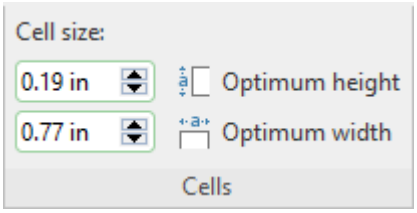
Across the top of the table, there are buttons labeled A, B, C, etc. To the left of the table, there are buttons labeled 1, 2, 3, etc.

These *column headers* and *row headers* allow you to change the column width or row height using the mouse:

- To change the *column width*, move the mouse cursor to the right border of the corresponding column header. The mouse cursor will change to a double arrow. If you now press and hold down the left mouse button, you can change the column width by dragging with the mouse.
- To change the *row height*, use the row headers in the same manner.

Changing cell sizes via commands

On the ribbon tab **Layout** | group **Cells**, you can change the column widths and row heights of your worksheet via commands.



The command group **Cells** contains the following entries:

Cell size (width and height)

Use the lower field to specify the exact width of the column. To do so, enter the desired value in the input field. Alternatively you can use the small arrow keys to increase or decrease the value.

With the field above you can set the height of the row accordingly.

Optimum width (or optimum height)

The **Optimum width** button automatically adjusts the width of a table column to fit its contents.

You can use this command as follows:

- If you select an *entire column* and choose this command, the width of the column will be adjusted to the cell that needs the most space.

> **Tip:** The same can be achieved much faster by simply double-clicking on the column header.

- If you select *some cells* within a row and then choose this command, only these cells will be used for the calculation of the optimum column width.
- If you select *some cells* within a column and then choose this command, only these cells will be used for the calculation of the optimum column width.
- If an *individual cell* is selected and you choose this command, the column width will be adjusted to the contents of that current cell.

The same applies to the **Optimum height** button, which allows you to automatically adjust the height of a row.

Showing and hiding rows/columns

You can hide rows/columns, thus they become invisible. This is useful if their content is not required at the moment or should remain hidden.

Tip: If you enable *sheet protection* for the current worksheet, users will not be able to unhide hidden row/columns in the worksheet. For more information, see also Sheet protection.

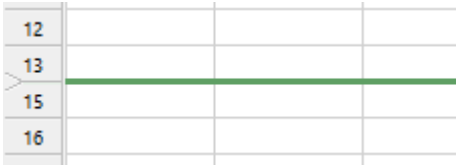
You can hide rows/columns as follows:

Using the row header to hide rows

The quickest way to hide rows is via their row header. *Row headers* are the buttons labeled with the row number to the left of the worksheet.

For example, to hide row 14, drag the bottom border of its row header upwards with the mouse until the row disappears completely.

The row is now hidden. Instead of its row header, a small triangle is displayed:



To show row 14 again, click on this triangle.

Alternatively: Right-click on the row header to open the context menu. Here, you will find the commands **Hide** and **Show**.

Hiding rows using the "Home | Visibility" button

You can also show and hide rows using the following ribbon commands:

Select the rows to be hidden, and choose the ribbon command **Home** | group **Cells** | **Visibility** | **Hide rows**.

The selected rows will be hidden.

To show them again, select the cell range that contains the hidden rows and choose **Home** | group **Cells** | **Visibility** | **Show rows**.

Hiding columns

Hiding and showing columns works in the same manner. Here, you select the column headers and use the ribbon commands **Home** | group **Cells** | **Visibility** | **Hide columns** and **Show columns**.

Number format

On the ribbon tab **Home** | group **Number**, you can change the number format of cells.

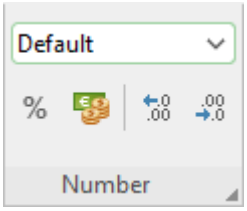
The *number format* determines how numbers are represented in these cells. If, for example, you enter the number 12.3456 in a cell and then apply one of the "Currency" number formats to it, the number will be displayed with the selected currency symbol and rounded to two decimal places (e.g. as €12.35).

Note: Numbers will only be *displayed* rounded, but they are actually not rounded. However, they still retain all of their decimal places. To actually round numbers, use arithmetic functions like the ROUND function.

To change number formats of cells, you can either simply use the buttons of the command group **Number** directly or open the dialog box of the command group **Number** ◢ for additional options.

Applying common number formats directly via the buttons of the command group "Number"

Common number formats can be used via the buttons of the ribbon tab **Home** | group **Number**.



Here, you have the following options:

- In the top dropdown list, you will find different categories of number formats such as *Default*, *Currency*, *Accounting*, etc. You can apply these directly via a mouse click. For detailed descriptions of the individual formats, see List of number formats.

  If you would like to change additional settings, select **More** in this dropdown list to open the dialog box with additional options (see below).

- You can use the icon % to display a number directly as a *percentage* (multiplied by 100). Click on this icon again to return to the default value.

- You can use the icon to quickly convert a number into a currency value, for example, *Dollar*. Click on this icon again to return to the default value.

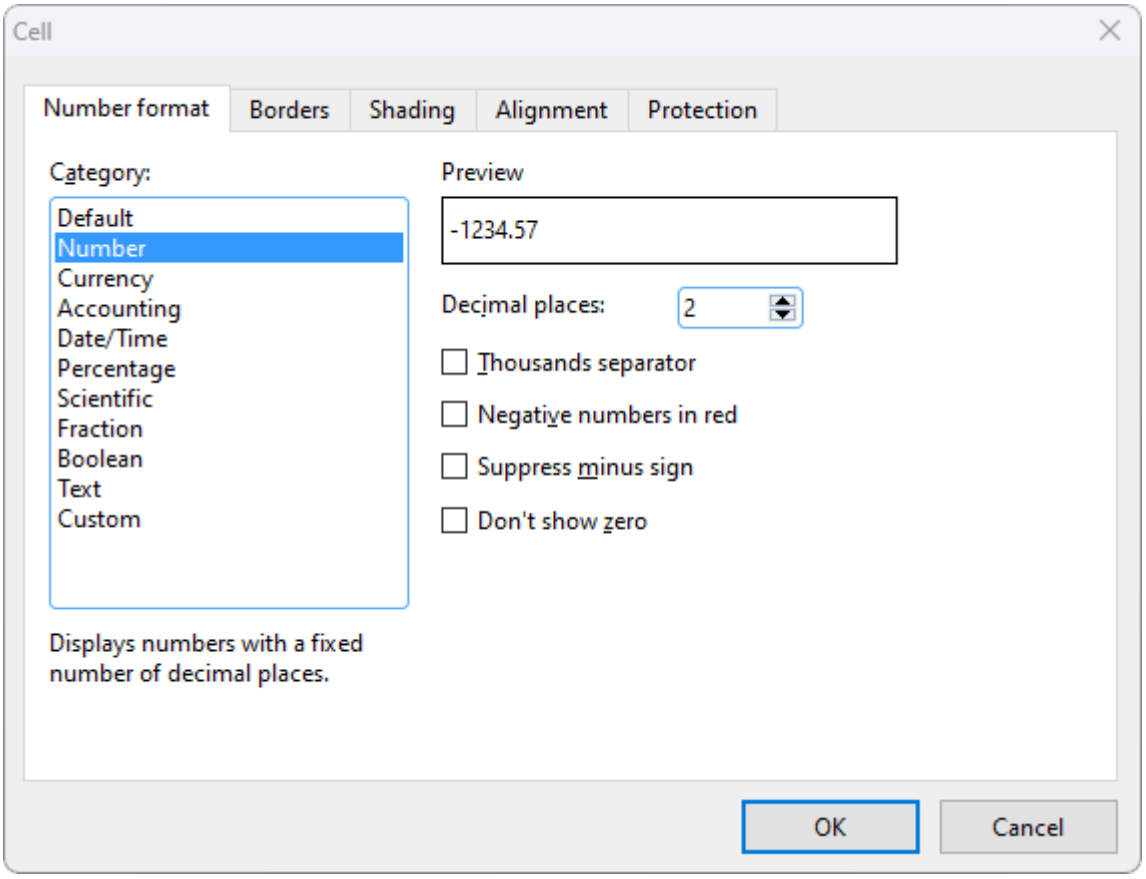
- With the two icons and , you can easily increase or decrease the number of decimal places (the numbers will only be *displayed* rounded, but they actually retain all of their decimal places).

Additional options: Via the dialog box of the command group "Number"

For advanced options (for example, custom number formats), use the dialog box. To do so, click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Number**.

Alternatively: Right-click to open the context menu and select the entry **Cell**.

In the following dialog box, switch to the **Number format** tab.



The following settings are available in the dialog box:

- **Category**

  Here, you can select the desired type of number format. For more information on the individual formats, see List of number formats.

- **Format**

  Some number formats also have several subtypes from which to choose. Select the desired subtype in this list.

- **Decimal places**

  Some formats allow you to specify the number of decimal places to be displayed.

Note: Numbers will only be *displayed* rounded, but they are actually not rounded. However, they still retain all of their decimal places.

- **Thousands separator**

  Some formats allow you to specify if numbers should be displayed with thousands separators (for example, 5,000,000) or without (for example, 5000000).

- **Negative numbers in red**

  Some formats allow you to specify if negative numbers should be displayed in red.

- **Suppress minus sign**

  Some formats allow you to specify if the minus sign of negative numbers should be suppressed.

- **Don't show zero**

  Some formats allow you to specify if the content of cells containing the number zero should be suppressed.

When you confirm with **OK**, the numbers in the selected cells will be formatted accordingly.

List of all number formats

As described in the previous section, you can use the direct commands of the command group **Number** on the ribbon tab **Home** to set the *number format* of a cell in order to determine how numbers should be displayed in this cell. For advanced options, you can also open the dialog box by clicking on the group arrow ◢ in the bottom right corner of the command group **Number**.

The following number formats are available:

Default

This displays the number as it was entered.

More precisely: The number is displayed right-aligned; otherwise, no changes will be made to its formatting. It is displayed with its actual number of decimal places (but with a maximum of five decimal places).

Example: 42

Number

This is the same as the **Default** number format, except that you can specify several formatting options (see previous section, options of the dialog box). For example, you can set the number of decimal places to be displayed.

Example: 42.00

Note: The number will only be *displayed* rounded, but it is actually not rounded. Thus, it still retains all of its decimal places. To actually round the number, use arithmetic functions like the ROUND function.

Currency

This displays the number with a currency symbol and two decimal places.

Only available via the dialog box: You can select the desired currency from the dropdown list **Currency**.

Example: €42.00

Note: Depending on the country settings of your system, the display may differ.

Accounting

This is the same as the **Currency** number format (see above), except that numbers are displayed in accounting format.

The difference to the currency format is that negative amounts have the minus sign in the left edge of the cell, separated from the number. Currency symbols are also aligned flush with each other.

Example: - €42.00

Note: Depending on the country settings of your system, the display may differ.

Date/Time

This displays the number as a date and/or time.

Only available via the dialog box: The dropdown list **Format** offers numerous different date and time formats, depending on your system's regional settings.

Directly via the dropdown list of the command group **Number**:

- **Short date:** The date is displayed in the *11/22/2018* format
- **Long date:** The date is displayed in the *Thursday, November 22, 2018* format
- **Time:** The time is displayed in the *22:30:00* format

Percentage

This displays the number as a percentage.

Apart from adding a percent sign (%) to the number, the number will also be multiplied by 100 when displayed. For example, 0.5 will be displayed as 50%.

Example: 4200%

Scientific

This displays the number in scientific notation.

Examples: 5E+03 (equals 5 * 10^3, which is 5000)

4.2E-01 (equals 4.2 * 10^-1, which is 0.42)

Fraction

This displays the number as a fraction.

Only available via the dialog box: The format of the fraction can be selected in the dropdown list **Format**.

Note: The number will be displayed as a rounded value if necessary. If, for example, you select the format **As halves** and then enter the number 2.3, it will be displayed as 2 1/2.

Examples: 2 1/2

5 9/10

Boolean

This displays the number as a Boolean value (logical value).

If the number is zero, FALSE will be displayed. If it is *any* other value, TRUE will be displayed.

Examples: TRUE

FALSE

Text

This number format presents a special case: It forces PlanMaker to treat a number as *text* – not as a number.

The number will be left-aligned and displayed exactly the way it was entered. Unlike all other number formats, numbers will *not* be automatically reformatted after they are entered. Leading zeros will not be removed; dates will not be reformatted automatically, etc.

This can be useful, for example, when entering numbers with leading zeros (for example, phone area codes or zip codes). If you enter a number like "00043682" into a cell, PlanMaker will usually remove the three leading zeros automatically. If you first apply the **Text** number format to the cell, the leading zeros will not be removed.

Caution: For compatibility reasons, this number format has been implemented in the same way as in Microsoft Excel. Thus, it will lead to the same unexpected results in calculations:
If you apply this format, for example, to cells A1 and A2, the result of the calculation A1+A2 will be the sum of these two cells. However, the calculation SUM (A1:A2) will return zero. The reason for this is that text generally has the "value" zero for most calculation functions!

Custom

Only via the dialog box: This displays the number in a custom number format.

For more information, see the next section.

Working with custom number formats

If required, you can also define and apply *your own* number formats in the dialog box of the command group **Number** on the ribbon tab **Home**.

Defining and applying a new number format

To create a new custom number format, proceed as follows:

1. Select the table cells to which you want to apply the new number format.
2. On the ribbon tab **Home** | group **Number**, click on the group arrow ◢ in the bottom right corner to open the dialog box, and switch to the **Number format** tab.
3. In the **Category** list, select **Custom**.
4. Click into the input field below **Format**.
5. Enter a text string that specifies the desired number format.

   For a list of available format codes, see Structure of a custom number format.
6. Confirm with **OK**.

The number format is created (and also applied to the currently selected cells).

Tip: If you want to create a new number format that is similar to one of the predefined number formats, the following tip can save you a lot of work: After opening the dialog box, first click on the desired predefined number format in the **Category** list. This will transfer the corresponding format codes into the input field for the format codes (step 5.). All you have to do now is make your desired changes, and that's it.

Applying a custom number format

Once created, a custom number format can be applied to as many other cells as you like. Proceed as follows:

1. Select the desired table cells.
2. Open the dialog box on the ribbon tab **Home** | group **Number** ◢ and switch to the **Number format** tab.
3. In the **Category** list, select **Custom**.
4. In the **Format** list, all custom number formats for the current document are displayed. Select the desired format.
5. Confirm with **OK**.

The number format will now be applied to the selected cells.

Deleting a custom number format

If you no longer need a specific custom number format, you can delete it at any time. Proceed as follows:

1. Open the dialog box on the ribbon tab **Home** | group **Number** ◢ and switch to the **Number format** tab.
2. In the **Category** list, select **Custom**.

3. In the **Format** list, select the format to be deleted.
4. Click on the **Delete** button.
5. Close the dialog box (preferably using the **Cancel** button, since clicking on **OK** would apply the currently selected format to the current cell).

Note: The **Delete** button is only available for custom number formats. You cannot delete any of the predefined number formats.

Structure of a custom number format

When you create a new number format (as described in the previous section), you have to enter a text string containing *format codes*. These codes specify how numbers are displayed.

An example:

#.00

These codes will cause numbers to be displayed as follows: All digits to the left of the decimal point, then a decimal point, then exactly two digits to the right of the decimal point.

The following format codes are available:

Format codes for numbers and text

To format numbers and text, you can use the following format codes:

Code	Result
#	Display a single digit of the number. Display nothing if there is no digit at this position of the number. Example: #.## applied to the number 1.2 returns 1.2.
0	Display a single digit of the number. Display a zero if there is no digit at this position. Example: #.00 applied to the number 1.2 returns 1.20.
?	Display a single digit of the number. Display a space if there is no digit at this position. Example: #.?? applied to the number 1.2 returns 1.2 (with a space behind the 2). ***Important:*** If at least *one* of the above three codes is located to the *left* of the decimal point, *all* digits to the left of the decimal point will be displayed. Example: 0.00 applied to the number 123.456 will not return 3.46, but 123.46.
. (period)	Display a decimal separator at this position. Note: If your system uses a different character (for example, a comma) as a decimal separator instead of a period, please use this character instead of a period.
, (comma)	Display the number with a thousands separator. This code can be placed at an *arbitrary* position in the format string: ,0.00 applied to the number 5000000 returns 5,000,000.00. Note: If your system uses a thousands separator other than a comma, use the corresponding character instead of a comma.

%	Display a percent sign at this position and multiply the number by 100. Example: 0% applied to the number 0.5 returns 50%.
E+ or **e+**	Display the number in *scientific notation*. Example: 0E+00 applied to the number 50000 returns 5E+04. Tip: *Engineering notation* (using only exponents that can be divided by 3) is also available. For example, #00E+00 applied to the number 50000 returns 50E+03 rather than 5E+04.
E- or **e-**	Same as E+, except that the exponent's sign is only displayed if it is negative. (It is always displayed for E+.) Example 1: 0E-00 applied to the number 50000 returns 5E04. Example 2: 0E-00 applied to the number 0.0005 returns 5E-04.
General	Wildcard for the entire cell content (formatted with the "Default" number format)
@	Wildcard for the entire cell content as text
\	Do not try to interpret the following character, just display it. Example: To output a # character, use the code \# (since the # sign is a valid format code).
"*Text*"	Do not interpret the text between the quotation marks as format code, just display it. Example: "MyText"
* (asterisk)	Repeat the following character. For example, *x fills the entire cell with the character "x". Example: The format code "Total:"* 0.00 displays the text "Total:" at the left of the cell and the number (with two decimal places) at the right. The space in between is filled with blanks.
_ (underscore)	Display a space that is as wide as the following character. Example: _- returns a blank space that is exactly as wide as a minus sign.
[Red]	Display the cell content in red. Other color codes which are available: [Black] [White] [Red] [Green] [Blue] [Cyan] [Magenta] [Yellow]

Format codes for dates and times

For dates/times, only the following format codes are valid:

Note: The format codes listed below are **case-sensitive!**

Code	Result
D	Day (1-31)
DD	Day, with 2 digits (01-31)
DDD	Name of day, abbreviated ("Sun" to "Mon")
DDDD	Name of day ("Sunday" to "Monday")

M	Month (1-12)
MM	Month, with 2 digits (01-12)
MMM	Name of month, abbreviated ("Jan" to "Dec")
MMMM	Name of month ("January" to "December")
MMMMM	Name of month, first letter only ("J" for January to "D" for December)
YY	Year, with 2 digits (e.g., 18)
YYYY	Year, with 4 digits (e.g., 2018)
Q	Quarter (1-4)
QQ	The word "quarter" in the language that is set in the system's regional settings
h	Hour (0-23)
hh	Hour, with 2 digits (00-23)
m	Minute (0-59)
mm	Minute, with 2 digits (00-59)
s	Second (0-59)
ss	Second, with 2 digits (00-59)
0	Tenths of a second (for example, hh:mm:ss.0)
00	Hundredths of a second (for example, hh:mm:ss.00)
000	Thousandths of a second (for example, hh:mm:ss.000)
AM/PM	Display times in 12-hour format (with *AM* or *PM*). This has to be placed at the *end* of the format codes! Example: h:mm:ss AM/PM
am/pm	Same as AM/PM, but with *am* or *pm* (lowercase)
a/p	Same as AM/PM, but with *a* or *p*
[h]	Number of hours in "endless" time format*
[m]	Number of minutes in "endless" time format*
[mm]	Same as [m], but with 2 digits*
[s]	Number of seconds in "endless" time format*
[ss]	Same as [m], but with 2 digits*

[$-*n*] Optional: This lets you specify the country code of the language to be used for day and month names. Examples: [$-0409] represents English (US), [$-0809] is for English (UK), [$-0407] is for German, [$-040A] is for Spanish, [$-040C] is for French, [$-0410] is for Italian.
If you do not specify a country code, PlanMaker automatically uses the system's default language.

* The *"endless"* time format is characterized by the fact that it does *not* reset times to 0:00 after 24 hours. This is particularly useful for *calculations* with times. For example, when you add 20:00 and 5:00, the result would normally be 1:00. If you format this calculation as [h]:mm, however, the result will be 25:00.

  The endless time format also supports negative time values. For example, 7:00 - 10:00 does not result in 21:00, but in -3:00.

  The same applies to minutes and seconds, which are also not reset after 60 minutes/seconds. Thus, you can even use the endless time format to convert times to minutes or seconds. For example, when you format 02:00:00 as [mm]:ss, 120:00 will be displayed, as there are 120 minutes in 2 hours. When you format it as [s], 7200 (the number of seconds) will be displayed.

Defining separate sections for positive numbers, negative numbers, zero and text

If required, you can split the text string with the format codes into up to four sections (separated by semicolons). This allows you to define your own number formats for positive numbers, negative numbers, the number zero and text – as follows:

Positive; Negative; Zero; Text

The number of sections that you include has the following effect:

Number of sections	Effect
Only 1 section:	The format codes apply to all values.
2 sections:	The codes in the 1st section apply to positive values.
	The codes in the 2nd section apply to negative values.
3 sections:	The codes in the 1st section apply to positive values.
	The codes in the 2nd section apply to negative values.
	The codes in the 3rd section apply to the number zero.
4 sections:	The codes in the 1st section apply to positive values.
	The codes in the 2nd section apply to negative values.
	The codes in the 3rd section apply to the number zero.
	The codes in the 4th section apply to text.

Example (with two sections):

0.00; -0.00[Red]

This number format has the following effect:

1st section: *0.00* causes positive numbers to be displayed with two decimal places.

2nd section: *-0.00[Red]* causes negative numbers to also be displayed with two decimal places. A minus sign is displayed in front of the number (see note!). The number is also colored in red.

Note: If you define a number format with two (or more) sections and you want negative values to be displayed with a minus sign, you must include a minus sign in the section for negative numbers.

For number formats with only one section, the minus signs are automatically displayed for negative numbers, even without explicitly defining them in the number format.

Defining separate sections using conditions

Alternatively, you can split the text string with the format codes into multiple sections by specifying a *condition* for each section.

The individual sections must by separated by semicolons. To define the conditions, simply add the desired comparison operation enclosed within square brackets to each section – for example, conditions such as [>0] or [>=100] or [=42].

You can define up to two sections with conditions. After these sections, you should add an additional section *without* a condition, which will then define the number format for those values that do not meet any of the conditions.

An example:

[<=0][Blue]0.00; [>30][Red]0.00; 0.00

These format codes display all values smaller than or equal to zero in blue – and values larger than 30 are displayed in red. If neither of the two conditions applies, the value is displayed in black.

Tip: "Conditional formatting" provides a much more convenient and powerful method for performing such tasks. For more information, see Conditional formatting.

Borders

You can use the ribbon command **Home** | group **Format** | **Borders** to completely surround cells with borders or add individual lines to cells. Gridlines (the lines between the cells) can also be applied here.

Common border lines: Via the icon of the command group "Format"

A selection of common border lines can be found via the icon **Borders** under **Home** | group **Format**.

Proceed as follows:

1. Select the desired cells.
2. Click on the icon's arrow for the dropdown menu and select an entry with the desired properties by clicking on it.

   Note: You can also combine individual lines to your liking by reopening the dropdown menu of the **Borders** icon and adding another attribute, for example, first positioning a line at the top and then an additional line at the bottom.

The table cells are now provided with the relevant lines.

Tip: If you then want to add exactly the same type of border to other cells, simply select them and click on the icon itself (not the arrow). PlanMaker will then reapply the most recently selected type of border.

Removing border lines: Click on **None** in the dropdown menu. All border lines will then be completely removed.

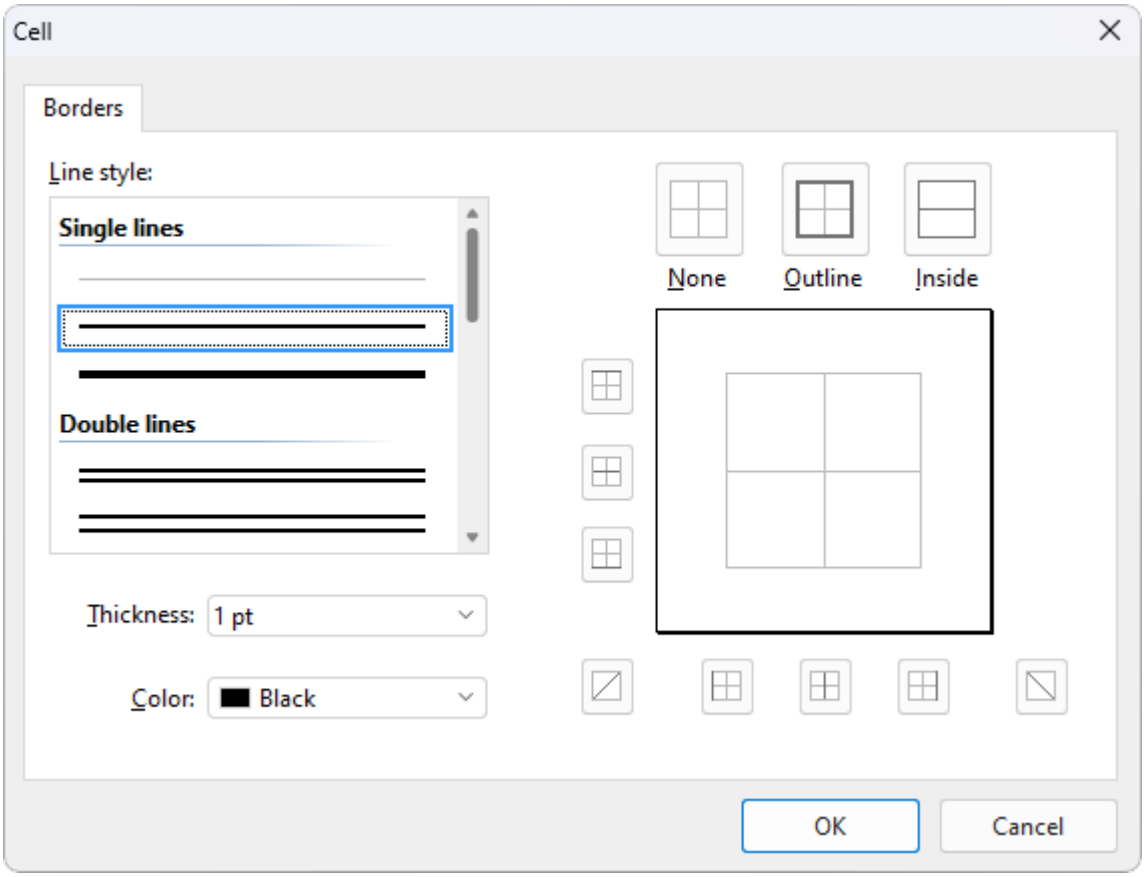
If you want to edit/remove individual border lines or define additional properties for borders and lines, click on **Borders** in the dropdown menu. A dialog box opens with other options (see below).

Other options: Via the dialog box

For additional options, such as the color or different thickness of lines, there are various options available in the dialog box.

You open the dialog box by clicking on the arrow of the icon and selecting **Borders** in the dropdown menu.

Alternatively: On the ribbon tab **Home** | group **Alignment**, click on the group arrow ◢ and switch to the **Borders** tab.



Essentially, you use this dialog box as follows:

1. First, you specify *what* type of border line you want to apply (line style, thickness, color).
2. Then specify *where* this type of border line should be placed. To do so, simply click on the desired lines (or their buttons) in the right half of the dialog box.

Let's take a closer look at the individual steps:

To add border lines to cells, for example, proceed as follows:

1. Select the desired cells.
2. On the ribbon tab **Home** | group **Alignment**, click on the group arrow ◢ for the dialog box and switch to the **Borders** tab.
3. First, specify *what* type of border line you want to apply. You have the following options:

   Line style (single, double or dashed lines)

   Thickness of the line

   Color of the line

   Note: In addition to the colors offered here, you can define new colors at any time. For more information, see Document properties, Colors tab.
4. Then specify *where* you want to apply this type of border line (top, bottom, left, right, etc.).

For this purpose, a preview box surrounded by a set of buttons is available in the right half of the dialog box. Use it as follows:

A. In the preview in the right half of the dialog box, simply click directly on all lines to which a border line is to be applied.

B. Alternatively, you can use the buttons that are displayed to the left and below the preview for this purpose. Each button represents a specific line (see icon on the button).

C. The buttons above the preview are useful shortcuts:

The **Outline** button applies the currently set border line to all outer borders.

The **Inside** button does the same for all inner lines (the "gridlines" between the cells).

The **None** button removes all lines at once.

5. If required, you can add as many lines as you like by simply repeating step 4.

Of course, you can always change the line style, thickness or color (step 3) before applying it (step 4).

6. When everything is as required, confirm with **OK**.

The table cells are now provided with the relevant lines.

Editing or removing existing border lines in the dialog box

You can change the style, thickness or color of existing border lines or also remove them at any time. To do so, open the dialog box again and proceed as follows:

- **Changing:** To change the appearance of a line, first select the desired settings (style, thickness, color). Then, in the preview, click on the line (or its button) to which you want to apply these settings.
- **Removing:** To remove a border line, click on that line (or its button) *twice* in the preview. Clicking once applies the border line, while clicking it again removes it.

  Tip: The **None** button displayed above the preview removes *all* lines at once.

Shading

You can use the ribbon command **Home** | group **Format** | **Shading** to apply a colored shade or pattern to cells.

Applying directly via the icon of the command group "Format"

If you click on the small arrow next to the icon **Shading**, the program will open a dropdown menu with *Standard colors* and *Color variations*. You can apply these directly to the selected cells by hovering over the colors with the mouse cursor (you can see the effect on the cell in the preview) and then clicking on the desired color.

Tip: The last selected color is displayed in the icon as a small horizontal bar. If you want to apply the same color to other cells, simply select them and click on the icon itself (not the arrow). PlanMaker will then reapply the last selected color.

This also applies to other shading settings, such as *Patterns* or *Shades*, which you can only find in the dialog box. You open the dialog box by clicking on the icon's arrow and choosing **More**. Detailed information on the options of the dialog box can be found below.

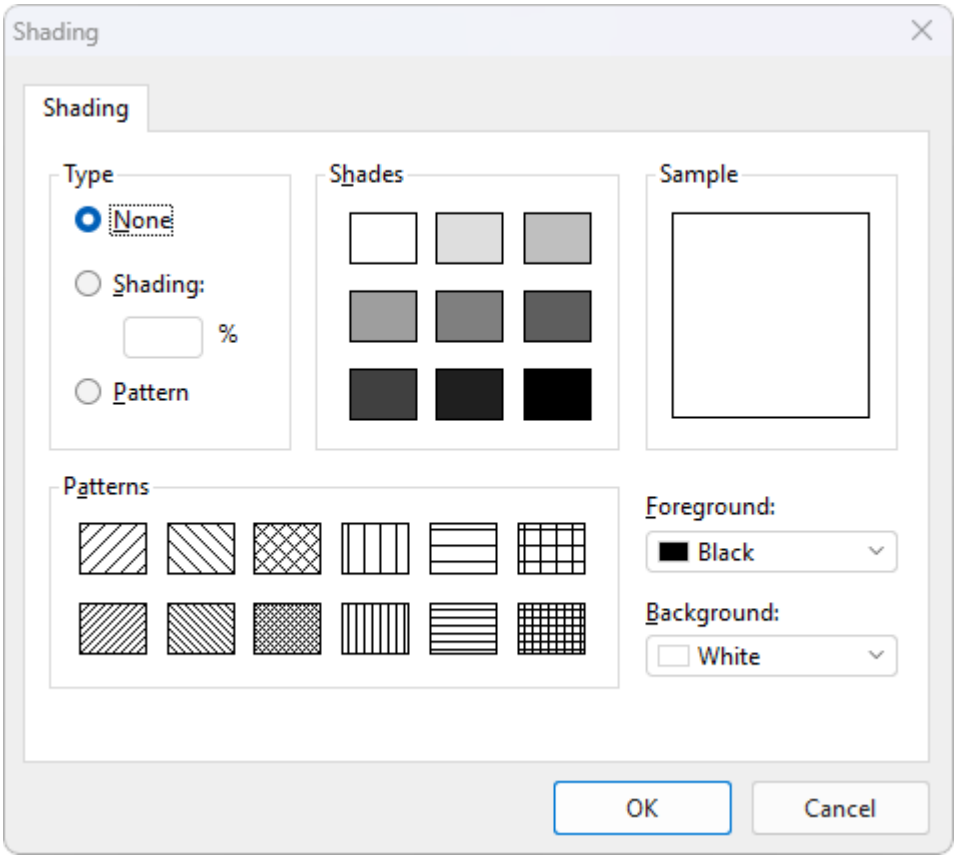
Removing shading: Click on **Transparent** in the dropdown menu to remove existing shading.

Other options: Via the dialog box

The dialog box contains additional shading options. The option **Shades** allows you to mix colors individually, and you can also apply different **Patterns** in the dialog box.

You open the dialog box by clicking on the arrow of the icon and choosing **More** in the dropdown menu.

Alternatively: On the ribbon tab **Home** | group **Alignment**, click on the group arrow and switch to the **Shading** tab.



In the dialog box, proceed as follows:

1. Select the desired cells.
2. On the ribbon tab **Home** | group **Alignment**, click on the group arrow for the dialog box and switch to the **Shading** tab.
3. Apply the desired shading (see below).
4. Confirm with **OK**.

You can choose between the following types of shading:

- **Adding a shade**

  A shade is a color shading that is mixed from the selected foreground color and background color.

  To add a shade, set the **Type** to **Shading** and then make the desired settings:

  First, set the desired **Foreground** and **Background** colors. (The background color is set to white by default.)

  In the section **Shades**, different mixtures of these two colors are now offered. Select one of these suggestions. Alternatively, you can enter an exact percentage for the shade in the input field under **Shading**. Values between 0 (100% background color) and 100 (100% foreground color) are allowed.

- **Adding a pattern**

  To add a pattern, click on one of the patterns in the section **Patterns**.

  You can also select a different **Foreground** and **Background** color for the pattern.

- **Removing a shade or pattern**

  If you want to remove existing shades or patterns, select **None** for **Type**.

Then confirm with **OK**.

Alignment

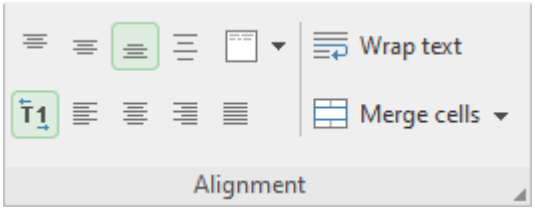
On the ribbon tab **Home** | group **Alignment**, you can specify the alignment of the cell contents within the cell.

Before you execute the commands, you must first select the desired cells. To change a single cell, simply navigate to it.

Then proceed as follows:

Aligning directly via the icons of the command group "Alignment"

Most of the options for aligning cell contents can be applied directly via the icons on the ribbon tab **Home** | group **Alignment**.



The following functions are available here:

Icon	Function
	Top vertical alignment: Align the cell content with the top edge of the cell.

Centered vertical alignment: Align the cell content vertically with the center.

Bottom vertical alignment: Align the cell content with the bottom edge of the cell.

Justified vertical alignment: The cell content is evenly distributed between the top and bottom cell border (only affects cells that contain multiple lines of content and have the option **Wrap text** enabled).

Rotate: Here, you can rotate the entire cell (including its contents) around the selected angle. The command *Stacked* corresponds to the option *Vertical text* in the dialog box (see below): The cell content is not read from left to right, but from top to bottom. To disable it, click on *Stacked* again.

Wrap text: The cell content will be automatically wrapped to the next line (inside the cell) if the cell width is insufficient to display it in a single line.

Merge cells: This option lets you merge a cell with its neighboring cells. PlanMaker treats merged cells as if they were one single cell. To do so, select the cell and any number of directly adjacent cells, and click on the icon itself to merge the cells.

Note: Whenever you merge cells, the content of all selected cells is automatically deleted – except for the cell in the upper left corner of the selected range.

To split merged cells, click on the icon again.

If you click on the icon's arrow, you can choose from various options for merging cells:

Merge cells: This corresponds to clicking on the icon itself, and all selected cells are merged normally.

Merge and center: The selected cells are merged and the cell content is centered horizontally over several columns (vertically at the bottom).

Merge and turn on line wrap: Cells are merged while wrapping the cell contents in the created merged cell.

If you selected cells across multiple columns and rows, you can choose between the commands **Merge by row** or **Merge by column** to merge the selected cells to only one column or only one row.

Here in the menu also, you can use the entry **Split cells** to separate merged cells.

Default horizontal alignment: Text is left-aligned, numbers are right-aligned. Logical values and error values are centered.

Left horizontal alignment: Align the cell content with the left edge of the cell

Centered horizontal alignment: Align the cell content centered

Right horizontal alignment: Align the cell content with the right edge of the cell

Justified horizontal alignment: The cell content is evenly distributed between the left and right cell border (only affects cells that contain multiple lines of content and have the option **Wrap text** enabled)

Most of these options can also be found in the dialog box of the command group **Alignment** (see below). In addition, the options enabled via the direct ribbon commands are displayed as enabled in the dialog box. If, for example, the ribbon command *Wrap text* is enabled for a cell, a check mark also appears in the dialog box in front of the option *Wrap text*.

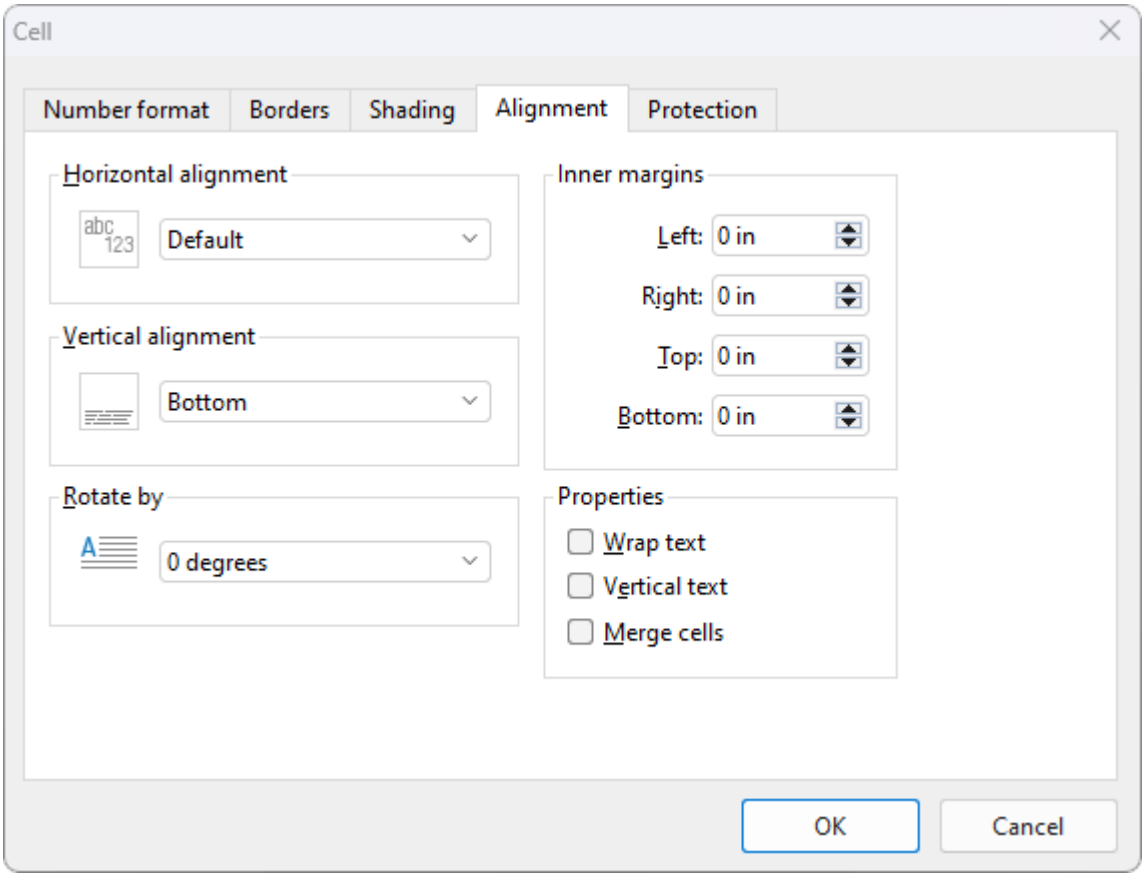
Shortcut keys

There are also shortcut keys for horizontal alignment:

Command	Windows/Linux	Mac
Left align	**Ctrl+L**	**Cmd+L**
Center	**Ctrl+E**	**Cmd+E**
Right align	**Ctrl+R**	**Cmd+R**
Justify	**Ctrl+J**	**Cmd+J**

Aligning via the dialog box of the command group "Alignment"

You open the dialog box on the ribbon tab **Home** by clicking on the group arrow ◢ in the bottom right corner of the command group **Alignment**. In the dialog box, you will find additional options that cannot be set directly via the icons of the command group **Alignment** as described above (for example, *Inner margins*).



The dialog box contains the following options on the "Alignment" tab:

Horizontal alignment

This determines the horizontal alignment of the cell content within the cell boundaries:

Default: Text left-aligned, numbers right-aligned, and logical values and error values centered

Left: Align the cell content with the left edge of the cell

Centered: Align the cell content horizontally with the center

Right: Align the cell content with the right edge of the cell

Justified: The cell content is evenly distributed between the left and right cell border (only affects cells that contain multiple lines of content and have the option **Wrap text** enabled)

Center across columns: The content of one cell is centered across multiple columns. For this purpose, select the cell and any number of adjacent cells to the right of it (these cells have to be empty!). Then select the **Center across columns** alignment.

Vertical alignment

This determines the vertical alignment of the cell content within the cell boundaries:

Bottom: Align the cell content with the bottom edge of the cell

Centered: Align the cell content vertically with the center

Top: Align the cell content with the top edge of the cell

Justified: The cell content is evenly distributed between the top and bottom cell border (only affects cells that contain multiple lines of content and have the option **Wrap text** enabled).

Rotate by ...

Here, you can rotate the entire cell (including its contents) around the selected angle.

Text direction

For text in Arabic script, you can change the text direction of the cell to right-to-left here. If you select the default setting **Context-sensitive**, PlanMaker sets the correct writing direction automatically.

For more information, see also Working with Arabic text.

Inner margins

Here, you set the inner margins of cells.

Line break

If this option is enabled, the cell content will be automatically wrapped to the next line if the cell width is insufficient to display it in a single line.

By default, this option is disabled.

Tip 1: You will also find this option via the context menu by right-clicking on a selected cell.
Tip 2: You can use the key combination **Ctrl+↵** to wrap to the next line manually.

Vertical text

If this option is enabled, the cell content will be displayed vertically (top to bottom instead of left to right).

Merge cells

This option lets you merge a cell with adjacent cells. To do so, select the cell and any number of directly adjacent cells and enable this option.

PlanMaker treats merged cells as if they were one single cell.

Note: Whenever you merge cells, all cells except the upper left cell lose their contents.

To split merged cells, select them again and disable this option.

Protection

On the **Protection tab** of the dialog box **Cell**, you can provide cells with cell protection, hide cell contents or formulas or disable printing.

Open the dialog box by clicking on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Alignment** (or group **Number**). Switch to the **Protection** tab.

Alternatively, open the dialog box by right-clicking on the selected cells for the context menu and select the entry **Cell**.

For more information, see Sheet protection.

Character format

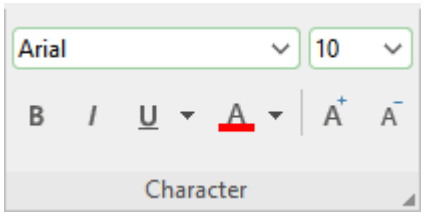
You can use the commands on the ribbon tab **Home** | group **Character** to change the character format (such as font, font size, font color, text styles, etc.).

The effect of this command depends on what you selected before you chose it:

- If you do not select anything specifically, the command will affect the cell on which the cell frame is currently located.
- If you select multiple cells, all selected cells will be affected by the change.

- You can also press the **F2** key in a cell and then select characters within the cell content, which will only change the selected characters.

Common commands: Via the icons of the command group "Character"



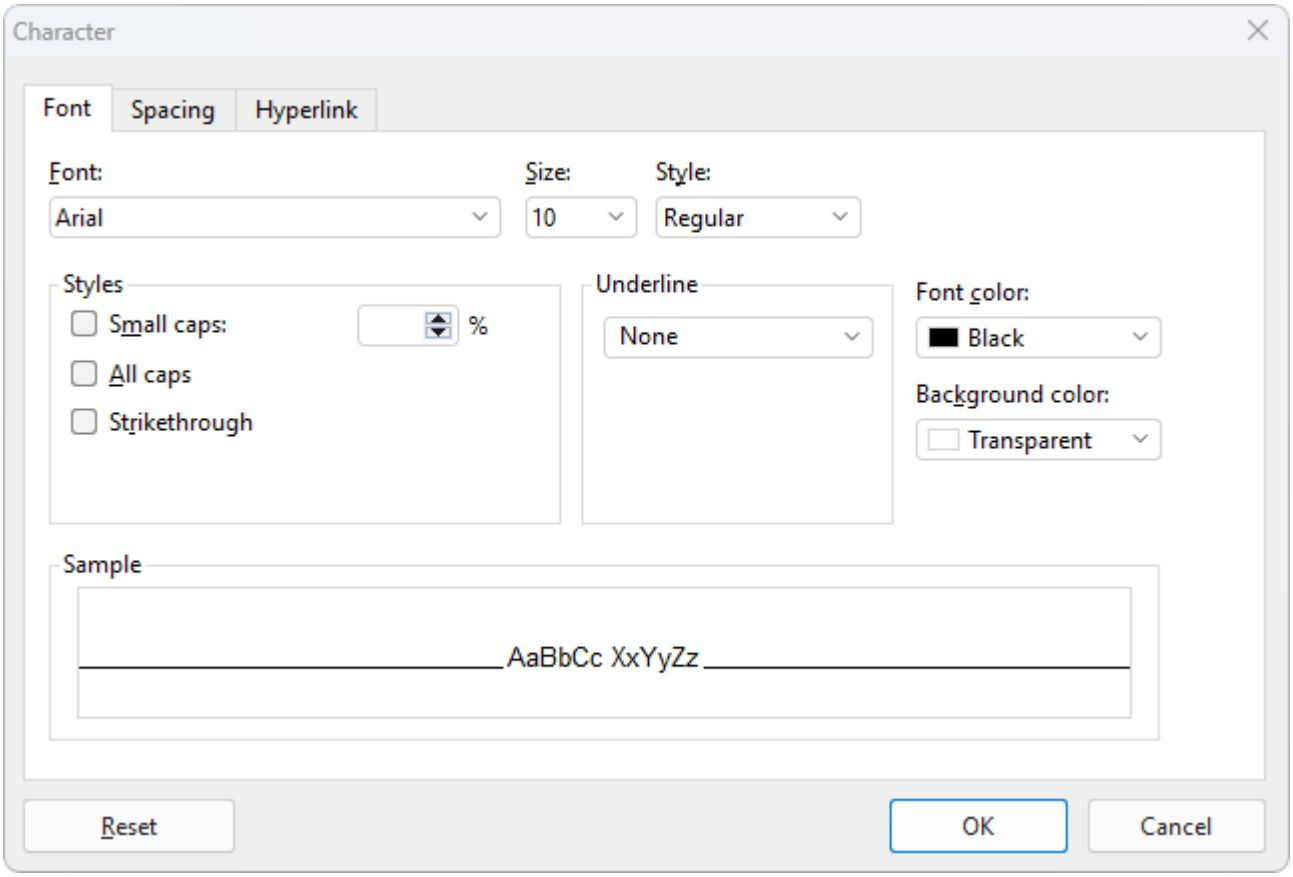
Frequently used settings can be made directly via the icons of the command group:

- Font and font size
- Text styles (bold, italic, underlined)
- Font color

Additional options: In the dialog box of the command group "Character"

If you click on the group arrow in the bottom right corner of the ribbon tab **Home** | group **Character**, the program opens a dialog box in which you will find additional options for character formatting.

The displayed dialog box has several tabs that you can switch between by clicking on one of the tab headers.



The individual tabs contain the following formatting options:

- **Font** tab

  For selecting properties like Font and font size, Text styles (bold, italic, underlined, etc.), Font color (and background color)

- **Spacing** tab

  For changing properties like Superscripts and subscripts, Letter spacing and character pitch, and enabling/disabling Kerning

- **Hyperlink** tab

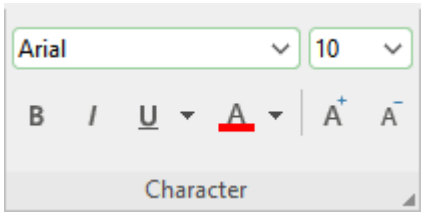
  For inserting and editing hyperlinks (for example, to web pages). For more information, see Using hyperlinks.

For more information on formatting, see the following pages.

Font and font size

To change the font or font size, see the commands on the ribbon tab **Home** | group **Character**. Proceed as follows:

Via the buttons of the command group "Character"



1. Select the desired cells or text.

   Android/iOS: If you are using these versions, please note that selecting cell contents works slightly differently there. For more information, see Selecting cells and cell content.

2. Click on the small arrow to the right of **Font** or **Font size**.

3. Set the desired font and font size:

   Font: Select the desired font from the dropdown list via a mouse click.

   Font size: The dropdown list offers the most common font sizes, which you can select via a mouse click. However, you can enter a different size manually at any time. Font sizes can be entered with a precision of a tenth of a point – thus font sizes such as 12.7 are also allowed.

   Tip: The icons in the group **Character** can be used to increase or decrease the font size by one level.

In the dialog box of the command group "Character"

If you only want to change the font and font size, these options in the dialog box are exactly the same as the buttons described above. Thus, it is only advisable to use the dialog box if you also want to use other character formatting options that are not available as buttons.

On the ribbon tab **Home** | group **Character**, you open the dialog box via the group arrow ◢ in the lower right corner and select the **Font** tab in the dialog box.

Text styles

Text styles are character formatting options such as bold, italics, etc.

PlanMaker offers the following text styles:

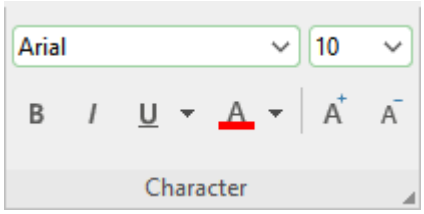
- ***Italic*:** A slanted variant of the font.
- **Bold:** A stronger ("more pronounced") variant of the font.
- **Underline:** The text is underlined (single, double, words only single, words only double, etc.).

- **Small Caps:** Smaller uppercase letters are used instead of lowercase letters.
- **ALL CAPS:** All letters are output as uppercase letters.
- **~~Strikethrough~~:** The text is struck through.
- **Superscripts** and **subscripts:** for example, r^2 and H_2O. For more information, see Superscripts and subscripts.

  Note: The 4 latter options are only available via the dialog box of the command group "Character" and not directly via the icons of the command group.

Common commands: Via the icons of the command group "Character"

The most common text styles can be applied on the ribbon tab **Home** | group **Character** directly via the icons of the group.



To apply text styles, proceed as follows:

1. Select the desired text.
2. Click on the icon for the text style you would like to enable or disable: **B** stands for bold, *I* for italics and U for simple underlining.

Text styles can be combined (almost) as desired.

Other options: Via the dialog box of the command group "Character"

For additional text styles that you cannot find via the buttons (e.g. **Small Caps**), open the dialog box.

On the ribbon tab **Home** | group **Character**, click on the group arrow ◢ in the bottom right corner and switch to the **Font** tab in the dialog box.

- In the area **Styles**, you will find small caps, all caps and strikethroughs. These can be enabled or disabled there via a simple mouse click.
- In the area **Underline**, you can use the dropdown list to choose between different options for underlined text.
- To enable bold and/or italics, open the dropdown list **Style** (to the right of font size) and select the desired entry: Regular, *Italic*, **Bold** or ***Bold Italic***.

Shortcut keys

There are also shortcut keys for some text styles:

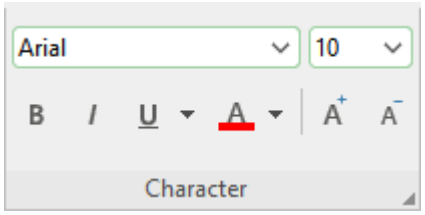
Command	Windows/Linux	Mac
Bold	**Ctrl+B**	**Cmd+B**
Italic	**Ctrl+I**	**Cmd+I**
Underline	**Ctrl+U**	**Cmd+U**

Font color

You can color both text and its background.

The commands for this can be found on the ribbon tab **Home** | group **Character**.

Font color: Via the icon of the command group "Paragraph"



For the **Font color** , proceed as follows:

1. Select the desired cells or text, and click on the small arrow to the right of the icon (thus, *not* on the icon itself).
2. Select the desired color from the dropdown menu.

   Tip: To format more text in the same color, select the text and then click on the icon itself because the last color used will be applied again.

The **Font color** can be found both as a direct button in the command group and in the dialog box. The **Background color**, however, can only be changed via the dialog box (see below).

Background color: In the dialog box of the command group "Character"

Insofar as only the font color is concerned, the selection in the dialog box corresponds exactly to the button described above. However, in addition to the font color, you can also change the **background color** in the dialog box.

Proceed as follows:

1. Select the desired cells or text.
2. Click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Character**. In the dialog box, switch to the **Font** tab.
3. Open the dropdown list **Background color** and select a different color instead of the default **Transparent**.

The text is highlighted with this color.

Tip: If you change the background color in this manner, it will only affect the individual *characters* within the cell. If you want to color the *entire cell*, however, you should use the ribbon command **Home** | group **Format | Shading** instead.

Notes

Color lists only offer a few standard colors, but you can always compose your own colors. To do so, click on the entry **Define color...** at the end of the dropdown list. (For more information, see also Document properties, Colors tab.)

Superscripts and subscripts

You can also add a superscript to characters (for example, r^2) or a subscript to characters (for example, H_2O). The commands for this can be found in the dialog box of the ribbon tab **Home** | group **Character**.

Proceed as follows:

1. Select the desired cells or text.

   If you only want to change individual characters within a cell, first press the **F2** key to select individual characters.

2. Click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Character**.

3. In the dialog box, switch to the **Spacing** tab.

You can now enable the options **Superscript** or **Subscript**.

If required, also enter the percentage of the superscript or subscript text for **Position**. You can also use a percentage in the field **Size** to determine how much the font size should be reduced. For example, enter 100 percent if you do not want the font to be reduced.

Shortcut keys

You can apply superscript/subscript even faster via the following shortcut keys:

Command	Windows/Linux	Mac
Superscript	**Ctrl+Shift+(Num+)**	**Cmd+Shift+(Num+)**
Subscript	**Ctrl+Shift+(Num-)**	**Cmd+Shift+(Num-)**
Remove superscript or subscript	**Ctrl+Shift+(Num*)**	**Cmd+Shift+(Num*)**

Note: (Num+) (Num-) (Num) represent the keys on the numeric keypad, usually located on the right side of the keyboard.*

Tip for Windows users: If you simply want to insert a superscript square number (m^2) or cubic number (m^3) directly, the key combination **AltGr+2** or **AltGr+3** is also available. Use the number bar above the letters rather than the numbers on the numeric keypad.

Letter spacing and character pitch

You can even change the letter spacing and character pitch in PlanMaker.

Letter spacing is the horizontal distance between characters. If values are less than 100%, the characters are closer together. For values greater than 100%, the characters are further apart.

Changing the **character pitch** changes the width of the characters themselves rather than the distance between them, thus the characters are distorted.

To change these settings, click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Character**. In the dialog box, switch to the **Spacing** tab and enter the required values for **Letter spacing** or **Character pitch**.

Note: Some printers do not support changing the character pitch for internal printer fonts. The font then appears wider or narrower on the screen, but not on the printout.

Kerning

Certain pairs of letters look better when the spacing between these letters is reduced or increased a bit. Such adjusting is known as *kerning*.

The following figure illustrates what is meant by kerning:

Above: without kerning. Below: with kerning

The upper text (without kerning) shows that the letters "V" and "A" are usually too far apart. This was corrected for the lower text (with kerning).

This adjusting occurs fully automatically. All you have to do is enable kerning:

1. Select the desired cells (or the desired text).
2. On the ribbon tab **Home** | group **Character**, click on the group arrow ◢ to open the dialog box and switch to the **Spacing** tab.

3. Enable the option the **Use kerning**.

PlanMaker now automatically adjusts the spacing between all letters where this would improve the text appearance.

Note: Not all fonts provide *kerning information* in their font data. This information is required to determine which letter pairs to adjust and how. Almost all font collections published by **SoftMaker** include extensive kerning information.

Resetting the character formatting

PlanMaker allows you to remove character formatting at any time. Proceed as follows:

1. Select the desired cells or text.
2. On the ribbon tab **Home** | group **Contents** | **Delete** , choose the command **Reset character formatting**.

PlanMaker now removes any character formatting that you have applied via the dialog box or the buttons on the ribbon tab **Home** | group **Character**.

Note: In the dialog box of the ribbon tab **Home** | group **Character**, you will also find a **Reset** button on each tab. However, the use of this button differs from the above function: You can only use it to undo all changes made *in the open dialog box* provided that you have not yet confirmed them with OK.

If the changes have already been confirmed with OK, they can only be undone via the **Reset character formatting** icon described above.

Paragraph format (text frames only)

If you have inserted *text frames* into your worksheet, you can change the paragraph formatting (line spacing, indents, etc.) of the included text at any time. For this purpose, use the commands of the ribbon tab **Home** | group **Alignment**.

Note: The "Paragraph" dialog box, which appears by clicking on the group arrow on the ribbon tab **Home** | group **Alignment**, is only available in *text frames*. For more information on working with text frames, see Text frames.

Paragraph formatting includes the following options:

- Paragraph indents
- Line spacing
- Paragraph alignment

- Spacing above/below a paragraph
- Hyphenation frequency (see Hyphenation)

For more information on these topics in the above order, see the following pages.

Paragraph indents (text frames only)

Note: Paragraph indents can only be applied to text inserted into a *text frame*. To indent text in *table cells*, you can use the option **Inner margins** on the **Alignment** tab when you open the dialog box of the ribbon tab **Home** | group **Alignment** ◢. Please note: A different dialog box will open depending on whether you are in a *text frame* or in a *table cell*.

You can use *indents* to change the left and right margins of paragraphs in text frames to move text in or out. The indent for the first line of a paragraph can be specified separately.

To use indents, select the desired paragraphs in the text frame. Then open the dialog box by clicking on the group arrow ◢ of the ribbon tab **Home** | group **Alignment**.

Alternatively, open the dialog box by right-clicking on the selected text and selecting the entry **Paragraph**.

In the **Indents** section, you can set the **Left** indent, the **Right** indent and the **First line** indent of a paragraph. Enter the desired values here. Negative values are also possible for the **First line** indent.

Line spacing (text frames only)

Note: Line spacing can only be changed for text inserted into a *text frame*.

Line spacing determines the spacing between the individual lines of a paragraph in a text frame.

To change the line spacing, proceed as follows:

1. Place the text cursor in the desired paragraph of the text frame (or select several paragraphs).
2. Open the dialog box by clicking on the group arrow ◢ of the ribbon tab **Home** | group **Alignment**.
   You will find the options for changing the line spacing in the section **Line spacing**:
3. First, use the dropdown list to select the method you want to use to specify the line spacing (see below).
4. Then enter the spacing in the edit box to its right.

When you confirm with **OK**, the line spacing will be changed according to your settings.

Methods of specifying the line spacing

In the dropdown list **Line spacing**, you can choose which method to use to define the line spacing.

The list has the following entries:

- **Single:** Automatic single line spacing

  Here, the optimum line spacing is determined *automatically*:

  If you increase the font size in the paragraph, the line spacing will be increased accordingly.

  If you decrease the font size, however, the line spacing will be decreased automatically.

- **Multiple:** Multiple automatic line spacing

  This corresponds to the option **Single** and thus also provides automatic line spacing. However, you can simply increase or decrease line spacing if necessary by specifying the desired number of lines in the input field to the right of this option.

  Some examples:

  Enter "1.5" for **lines** to obtain one and a half times the automatic line spacing.

  Enter "2" to get twice the automatic line spacing.

  Enter "1" to get automatic single line spacing (corresponds to selecting the option **Single**).

- **Exactly:** Exact line spacing

  Here, you can set the line spacing to an *exact* value in points. Here, the line spacing will *not* be automatically adjusted to the font size.

- **At least:** Automatic line spacing with a given minimum

  This setting corresponds to the option **Single**, but prevents any reduction below the specified minimum value.

  Thus if you enter 12 points as the minimum value, the program uses automatic single line spacing by default. If the automatic line spacing is less than 12 points (because you are using a very small font size), an exact line spacing of 12 points is used instead.

By default, the line spacing is set to **Single**.

Paragraph alignment (text frames only)

On the ribbon tab **Home** | group **Alignment**, you can also change the alignment of text paragraphs that were inserted into a text frame.

The following types of paragraph alignment are available:

- Left align
- Center
- Right align
- Justify (distribute words evenly between the left and right margins)

You can adjust the paragraph alignment directly using the icons under **Home** | group **Alignment**:

Alternatively, you can use the path via the dialog box by clicking on the group arrow ◢ under **Home** | group **Alignment**. In the open dialog box, you will find the corresponding commands in the *Alignment* list.

Changing the text direction (for Arabic script)

For text in Arabic script, there is an additional option called **Text direction**, which allows you to change the text direction of the paragraph to right-to-left. For more information, see also Working with Arabic text.

Spacing above/below a paragraph (text frames only)

Note: The spacing above/below a paragraph can only be set for text that was inserted into a *text frame*. To set the spacing between *cells*, you can change their row height at any time.

In addition to line spacing, you can also define how much space should be kept above the first line and below the last line of a paragraph for text in a text frame.

To do so, open the dialog box by clicking on the group arrow ◢ of the ribbon tab **Home** | group **Alignment**. The following options are available in the **Paragraph spacing** section:

Option	Result
Before	Here, specify how much spacing should be maintained from the end of the previous paragraph.
After	Here, you can enter the spacing that PlanMaker should maintain up to the start of the next paragraph.

Note: These values do not affect the amount of space *between* lines within a paragraph (see Line spacing).

Character styles

You can use the command **Character style**, which can be accessed via the context menu, to create and modify "character styles".

In a *character style*, you can save a frequently used character format (font, font size, text styles, etc.) and then apply it repeatedly to any characters or entire cells.

If, for example, you want to regularly print small print sections in a different font and smaller size, simply create an appropriate character style – lets call it "Small print" – and apply it to the desired text passages.

Note: In PlanMaker you can also apply cell styles (see Cell styles). The difference to character styles is that character styles only store the character format, whereas cell styles store *both* character format *and* cell format (number format, alignment, borders, etc.). Furthermore, character styles are also applicable to individual characters within a cell, while cell styles are applicable only to entire cells.

For more information, see the following pages.

Creating character styles

To create a new character style, proceed as follows:

1. Double-click on a cell (alternatively: press **F2**).
2. Right-click to open the context menu and select the entry **Character style**.
3. A dialog box opens. In the list on the left side of the dialog box, select the style on which you want your new character style to be based. In the event of doubt, use the "Normal" style.

   Note: The style of the character in which the text cursor was located before choosing the command **Character style** is always preselected here in the dialog box. For more information, see "Based on" character styles.
4. Then click on the **New** button.
5. Give the character style any name you like and confirm with **OK**.
6. The program will display a dialog box, which is very similar to the dialog box for the ribbon tab **Home** | group **Character** ◢. Here, you can specify the desired character format for this style (see Character format).
7. Once all the desired settings have been made, click on **OK** to define the style.
8. Exit the dialog box "Character style" with **Close**.

The new character style is now defined and ready for use. To learn how to apply character styles, see the next section.

Using shortcut keys

If you need a style frequently, you can assign it a shortcut key at any time. The advantage of this is that you can then enable the style very swiftly with one press of a key.

To assign a shortcut key to a style, proceed as follows:

Perform steps 1. to 5. as described above. In the dialog box on the **Style** tab, position the text cursor in the **Shortcut key** input field and press the desired key combination (starting with **Ctrl** or **Alt**).

Note: If you press a key combination that is already *assigned*, the current assignment is displayed below the input field **Shortcut key**. You should then press the **Backspace key**⌫ to delete your shortcut key and use a different key combination. Otherwise, you will overwrite the key combination for another style or even one of PlanMaker's commands.

We recommend that you always use key combinations that include the keys **Ctrl** *and* **Shift** since these are not usually assigned.

Applying character styles

To apply a character style, proceed as follows:

1. Double-click on the cell and select the desired characters within the cell.
2. Right-click to open the context menu and select the entry **Character style**.
3. In the dialog box, select the desired style.
4. Click on **Apply**.

The formatting of the selected characters immediately changes to that which you specified in the character style.

Tip: If you have assigned a shortcut key to the style, you can apply the style even faster: Select the desired cell contents and press the assigned key combination.

Changing character styles

Of course, you can change character styles again at any time.

Important: When you change the formats stored in a character style, *all* cells using this style are changed accordingly. However, this automatic reformatting applies in a limited way to characters where you manually changed the formatting *after* you assigned the style (see next section).

To change a character style, proceed as follows:

1. Double-click on a cell.
2. Right-click to open the context menu and select the entry **Character style**.
3. Click on the **Edit** button in the dialog box.
4. Make the desired changes to the style.
5. Click on **OK**.
6. Exit the dialog box with **Close**.

Deleting or renaming character styles

The **Delete** button in the dialog box of the command **Character style** deletes the selected style in the current document. To give a style a new name, click on **Rename** and type the new name. This also only affects the current document.

Note: After you *delete* a character style, the formatted characters retain their formatting. These characters are now directly assigned the character style on which the deleted style was based. For more information, see "Based on" character styles. Those formatting options that were explicitly set in the deleted style and now remain with the characters of the document are treated as if the characters were formatted manually – and are thus retained even if you change formatting later in the style.

Deleting or renaming character styles in a document template

To delete or rename the styles in a document template, open the document template, make the changes there and save the template.

Scope of character styles

Did you change a character style and nothing happened in some places that were assigned this style? This happens because these sections of text were subsequently reformatted manually.

An example:

Let's assume that you define a style with the Arial font and apply it. If, for example, you now change the style's font to Times New Roman, all characters that are formatted in this style will change accordingly.

However: If you subsequently selected a completely different font for some of these characters via the command group **Home | Character**, the change to the character style *no* longer affects these characters. That is to say, manually applied character formatting takes precedence over styles.

If you want the affected characters to be "freed" from their manual formatting, choose the command **Reset character formatting** on the ribbon tab **Home |** group **Contents | Delete**. The characters now have the formatting specified in the character style again.

"Normal" character style

The character style called "Normal" is present by default in every document and has a special significance: it is the default style for all cell contents. When you start typing something in a new document, the "Normal" character style is automatically used for it.

If you change the "Normal" character style – for example, the font – the font for *all* cells will be changed (except for cells that you have formatted manually in a different font via the ribbon tab **Home | Character**). Furthermore, subsequent entries will appear in this default font which you have redefined as the "Normal" character style for the document.

"Based on" character styles

When you create a new character style, as described at the beginning of this chapter, it is usually based on the "Normal" character style. This means that the new character style is based on the formatting of the "Normal" style. Changes to the formatting of the "Normal" style thus affect all other styles.

If you have already created character styles and used them in the document, the following happens when you choose the command **Character style**: In the list on the left side of the dialog box, the style of the currently selected character is selected – this may be the "Normal" character style or another one – and is thus also preset as the basis for a new style. In the list, change the style that you want to use as the basis for a new character style, if necessary, before clicking the **New** button.

Please note: Subsequent changes to the formatting of the style basis will also change all styles based on it. Only the formatting that you have already customized there will remain unchanged.

If, for example, you select a new font for the "Normal" style, the font will also change in all other character styles – unless a different font was explicitly selected there.

In addition, you can reassign the style basis to a character style at any time with the option **Based on**.

To have a character style based on a style *other* than the one you previously selected, proceed as follows:

1. Double-click on a cell.
2. Right-click to open the context menu and select the entry **Character style**.
3. In the list on the left side of the dialog box, select the character style whose basis you want to change and click on the **Edit** button.
4. In the following dialog box, switch to the **Style** tab.
5. *Important:* In the dropdown list **Based on**, select the style basis to which the selected character style is to be linked.
6. You can now also change the desired formatting of the character style, etc.

Relative font size

If you have one style based on another, you can even specify the *relative* font size if you want. For example, you can specify that the font size of style X should always be 80% of the font size of style Y.

To do so, proceed as follows:

Perform steps 1. to 4. as described above and enable the option **Scale** on the **Style** tab. Enter the desired factor as a percentage – for example, 80 – and confirm with OK.

The font size of a style edited in this manner will always be 80% of the size specified for the style on which it is based.

Cell styles

You can use the ribbon command **Home** | group **Format** | **Cell styles** to apply "Cell styles".

Cell styles are a combination of cell attributes (number format, orientation, borders, etc.) and a character format (font, font size, etc.) that you can save. Once saved, you can repeatedly apply this cell style to any cell.

If, for example, you want to frequently format cells with the number format "Percent", red background and bold font, simply create a corresponding cell style and apply it to the desired cells.

Note: In PlanMaker, you can also apply character styles (see Character styles). The difference to cell styles is that character styles only store the character format (font, font size, etc.), whereas cell styles store *both* character format *and* cell format. Furthermore, character styles are also applicable to individual characters within a cell, while cell styles are applicable only to entire cells.

For more information, see the following pages.

Applying cell styles

To apply a cell style, navigate to the cell that you want to format. Alternatively, you can select multiple cells to change their formatting together.

Then proceed as follows:

Applying directly via the dropdown menu of the icon

If you click on the **Cell styles** icon on the ribbon tab **Home** | group **Format**, the program opens a dropdown menu with various predefined styles (and custom styles if created). Here, you can directly select and apply a suitable cell style.

Further down in this dropdown menu, you can use the option **Styles in use** to display the styles already used in the current document by enabling the checkbox.

For advanced options, click on the entry **Manage cell styles** in this dropdown menu to open the dialog box (see below and the following pages).

Advanced options: Via the dialog box

1. On the ribbon tab **Home** | group **Format**, click on the **Cell styles** icon and select **Manage cell styles** in the dropdown menu.

   The program displays a dialog box with a list of all cell styles.

   Tip: You can use the option **Show** to determine whether *all* available styles should be displayed in the list – or only those already *used* in the current document.

3. Select the desired cell style in the **Styles** list.

4. Click on **Apply**.

The formatting of the selected cells immediately changes to that which you specified in the cell style.

To clarify, a cell style can include the following formatting options:

- Number format
- Font (font and font size, text styles, etc.)
- Cell alignment
- Shading
- Borders
- Cell protection

In the section **Format includes** of the above dialog box, you can (optionally) specify exactly which of these formatting options should be used when you apply the selected cell style (see also next section).

Creating cell styles

To create a custom cell style, proceed as follows:

1. On the ribbon tab **Home** | group **Format**, click on the **Cell styles** icon and select **Manage cell styles** in the dropdown menu.

   The dialog box *Cell style* opens.

2. Optional: In the **Styles** list in this dialog box, select the cell style on which your new style should be based (see also "Based on" below). Alternatively, simply select the default style "Normal".

   Tip: You can use the option **Show** to determine whether *all* available styles should be displayed in the list – or only those already *used* in the current document.

3. Click on the **New** button.

4. Give the cell style any name you like and confirm with **OK**.

5. The new cell style has now been created, and you can edit it individually as follows:

   To change the character format, click on the **Character** button. The program will display a dialog box, which is very similar to the dialog box for the ribbon tab **Home** | group **Character** (see Character format).

   To change the cell format, click on the **Cell** button. The program will display a dialog box, which is very similar to the dialog box for the ribbon tab **Home** | group **Number** (see Number format and the following section).

6. Click on **Close** to exit the dialog box (or on **Apply** first if you want to see the result of the cell style directly for the selected cells).

The style is now defined and ready for use via the ribbon command **Home | Cell styles**.

You can create a different set of cell styles for each document; the styles are stored in the document.

The selection "Based on"

When you create a new cell style, it is usually based on the "Normal" cell style. This means that the new cell style is based on the formatting of the "Normal" style. Changes to the formatting of the "Normal" style thus affect all other styles.

If you have already created cell styles and used them in the document, the following happens when you choose the command **Manage cell styles**: In the list on the left side of the dialog box, the style of the currently selected cell is selected as the basis. This may be the "Normal" cell style or another one. In the list on the left, change the style that you want to use as the basis for a new cell style, if necessary, before clicking on the **New** button.

You can use the selection **Based on** to subsequently change the assignment of the style basis for the created cell styles:

In the list on the left of the dialog box, first select the style to which you want to assign another style base. The **Based on** list shows you the basis on which the selected style is currently based. Here, you can change the style basis.

Note: If you want to change the **Based on** assignment for a cell style and you have not yet created a custom cell style, then this option cannot be selected yet because only the default styles are available so far. They have the "Normal" style as their basis, and this is a fixed assignment.
The option **Based on** will only be freely available to you once you have defined your custom new style and then selected it from the Styles list on the left.

Please note: Subsequent changes to the formatting of the style basis will also change all styles based on it. Only the formatting that you have already customized there will remain unchanged.

The "Format includes" section

In the **Format includes** section of the dialog box, you can specify exactly which of the formatting options to include when you apply the cell style.

For example, cell styles such as "Currency" should only provide the cells with the number format "Currency" – and not change anything else. That's why all options except "Number format" are disabled in this style.

The settings in this section can be made individually for each style.

Note: For the default style "Normal", these options cannot be changed.

Using shortcut keys

If you need a cell style particularly frequently, you can assign it a shortcut key at any time. The advantage of this is that you can then enable the style very swiftly with one press of a key.

To assign a shortcut key to a style, use the **Cell styles** icon to access the entry **Manage cell styles**. In the opened dialog box, select the desired style in the list. Then click in the **Shortcut key** input field and press the desired key combination (starting with **Ctrl** or **Alt**).

Note: If you press a key combination that is already *assigned*, the current assignment is displayed below the input field **Shortcut key**. You should then press the **Backspace key**⌫ to delete your shortcut key and use a different key combination. Otherwise, you will overwrite the key combination for another style or even one of PlanMaker's commands.

We recommend that you always use key combinations that include the keys **Ctrl** *and* **Shift** since these are not usually assigned.

Editing cell styles

Of course, you can change cell styles again at any time.

Important: When you change the formatting options stored in a cell style, *all* cells using this style are changed accordingly.

To change a cell style, proceed as follows:

1. On the ribbon tab **Home** | group **Format**, click on the **Cell styles** icon and select **Manage cell styles** in the dropdown menu.
2. The dialog box *Cell style* opens. Select the style that you want to change.
3. Click on **Character** to change its character format or on **Cell** to change its cell format.
4. When you have finished, exit the dialog box with the **Close** button.

Deleting or renaming cell styles

The **Delete** button in the dialog box of the command **Manage cell styles** deletes the selected style in the current document.

To give a style a new name, click on **Rename** and type the new name. This also only affects the current document.

Note: Some of the cell styles are predefined and cannot be deleted or renamed.

"Normal" cell style

The cell style called "Normal" is present by default in every document and has a special significance: it is the default style for all cells. As long as you do not apply other cell styles, every cell in a document is always formatted in the "Normal" cell style.

If you change the "Normal" cell style – for example, the number format – the number format for *all* cells will be changed (except for cells that you have formatted manually in a different number format via the ribbon tab

Home | Number). Furthermore, subsequent entries will appear in this number format, which you have defined as the new default format in the "Normal" cell style for the document.

If you want to apply a different cell style to certain cells instead of the default "Normal" style, select the cells and follow the instructions in Applying cell styles.

Document templates

As you learned in the previous two sections, character styles and cell styles are always saved in the document in which they were created. If you want to use such styles in other documents also, you should save them in a *document template*.

Each time you choose the ribbon command **File | New** , PlanMaker allows you to select a document template as the basis for the new document. If, at this point, you select a document template that you have created, the new document will have the same character and cell styles that you saved in the template.

Note: Document templates may also contain text, values, calculations, etc., in addition to your favorite character and cell styles. This enables you to create a set of templates for invoices, annual reports, balance sheets or other frequently used documents.

Simply choose the command **File | New**, select the template and complete the worksheet.

Note: The default folder for document templates is defined in the Settings, Files tab. Normally, this setting should not be changed.

For more information, see the following pages.

Creating document templates

To create a new document template, proceed as follows:

1. Start a new document or open a document or document template that you want to use as the basis for the new template.
2. Create the desired character and cell styles and/or enter text, values, etc., to create the template.
3. Choose the ribbon command **File | Save as**.
4. Select the entry **PlanMaker template** from the dropdown list **Save as type**.
5. At this point, PlanMaker switches automatically to the folder where the document templates are located.
   Android/iOS: This step is not supported here. Navigate manually to the folder "SoftMaker/PlanMaker NX templates".
6. Type a name for the template in the **File name** input field.
7. Confirm that you want to save it.

The new template will now be saved.

Using document templates

To use a document template, simply start a new document with the ribbon command **File | New**. You can specify on which document template this is to be based.

Proceed as follows:

1. Choose the ribbon command **File | New** to open the dialog box. (Click directly on the icon itself.)
2. You can select the required document template in the dialog box.
3. Confirm with **OK**.

The system now creates a new document for which all character and cell styles saved in the document template become available. If the document template also contains filled cells, they are displayed accordingly and can be edited normally.

Editing document templates

Changing a document template is no different from editing a normal document. You open the template, make your changes and save the document template.

Important: Of course, changing a document template will affect *all* documents that are subsequently created on the basis of this template.

To change a document template, proceed as follows:

1. Choose the ribbon command **File | Open** (click directly on the icon itself).
2. Select **PlanMaker templates** from the dropdown list of file types.
   Note: The default folder for document templates is defined in the Settings, Files tab. Normally, this setting should not be changed.
3. Select the template to be changed and confirm that it has been opened.
4. Make the desired changes to the cells and/or styles of this document template.
5. Choose the ribbon command **File | Save** to save the changed template.

Of course, you could also save the template under a different name via the ribbon command **File | Save as** if you did not want to overwrite the original template.

"Normal.pmvx" document template

The `Normal.pmvx` document template is the default template for new documents. If you choose the command **File | New**, this template will always be set. It is completely empty and contains only the default "Normal" character style as well as the default "Normal" cell style.

Accordingly, the `Normal.pmvx` template is a suitable basis for creating a completely new template from scratch.

Important: You should not usually change the `Normal.pmvx` default template. If you want to do this anyway, you should be aware that these changes will affect *all* documents that are subsequently created based on this template.

If, for example, you want to change the page format for all documents that will be subsequently created based on the `Normal.pmvx` template, open this template, change the page format and save the template.

You can also choose a different default template for new files:

To do so, choose the ribbon command **File | New** (click directly on the icon itself), select the desired template in the dialog box and click on **Set as default**. From now on, PlanMaker will always offer this template as the default template when you create a new document.

Template folder for the "Normal.pmvx" document template

The default folder for the "Normal.pmvx" document template and all other document templates is defined in the Settings, Files tab. Normally, this setting should not be changed.

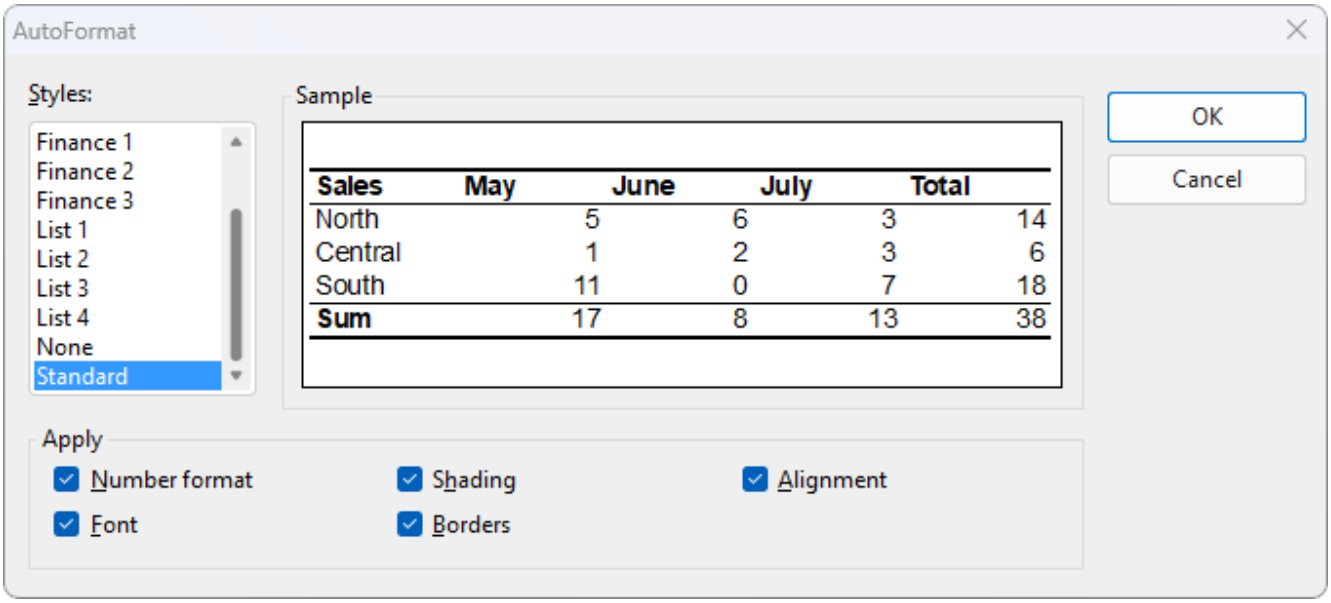
AutoFormat

You can use the command **AutoFormat** to change the layout of entire table areas at once.

Tip: By inserting "Tables in worksheets" using the ribbon command **Insert | Table**, you can apply different table styles even better. You will find a wider range of styles, additional features and better handling via this function. You can then further customize the table style on the ribbon tab **Table |** group **Style**. For more information, see Tables in worksheets.

Note: The command AutoFormat is not available on the ribbon by default. However, you can add this command as a separate button on a ribbon tab. To do so, choose the ribbon command **File | Customize | Customize ribbon** and then proceed as described in Customizing icons and groups in the ribbon.

The menu command is still to be found in the *classic menu interface*: **Format > AutoFormat**:



The dialog box for this command shows a list of predefined "AutoFormat styles" whose appearance can be previewed in the **Sample** field. To apply an AutoFormat style, simply select a table area in the worksheet, choose the above command, select the desired style and click on **OK**.

The options in the **Apply** section of the dialog box let you choose which formatting options to apply. If, for example, you disable all options except **Borders**, only the borders of the AutoFormat style will be applied; all other formatting of the table will remain unchanged.

Conditional formatting

You can use the ribbon command **Home** | group **Format** | **Conditional formatting** to apply *conditional* formatting to cells. This will only be applied if a certain *condition* is met. Thus, for example, you can specify that a cell should be colored red whenever it contains a value greater than 1000.

You will find out everything you need to know about it on the following pages. The following topics are covered:

- **Creating new formatting rules**

  To apply conditional formatting to cells, you select the relevant cells and create a *formatting rule* for them – for example: "If the cell content is greater than 1000, display the cell in red."

  You can apply any number of formatting rules to a cell – for example, you can create a second rule that formats the cell in boldface if it contains a value below zero, etc.

- **Types of conditional formatting**

  This section deals with the different types of conditional formatting.

- **Managing formatting rules**

  You can use the command **Manage rules** to centrally manage all conditional formatting in the document, including editing and deleting it.

- **Removing conditional formatting**

  The last section covers the removal of conditional formatting.

Creating new formatting rules

To apply conditional formatting to cells, you select the relevant cells and then create a *formatting rule* for them. Formatting rules always consist of two parts:

- a condition
- ... and the formatting to be applied if this condition is met.

Example: "If the cell content is greater than 1000, display the cell in red."

Application example

To define (and also apply) such a rule, proceed as follows:

1. Navigate to the desired cell. Of course, you can also select multiple cells to apply conditional formatting to them all at once.
2. Choose the ribbon command **Home**| group **Format** | **Conditional formatting** and select **New rule** from the dropdown menu to open the dialog box.

Tip: For quick use of common formatting rules, you can also apply certain options directly from the dropdown menu of the command **Conditional formatting** (see "Tip: Applying formatting rules directly from the dropdown menu" below).

3. In the dialog box, first select the type of condition to be used for **Type**.

   In our example, you would select the option **Format only cells that contain**. (For more information on all available options, see Types of conditional formatting).

4. You can now specify the desired condition.

   In our example, this would be the condition "cell value is larger than 1000". Thus, select the options **Cell value** and **greater than**. Then enter **1000** on the right.

5. Finally, click on the **Format** button, and specify the formatting options to be applied if this condition is met.

   In our example, switch to the **Font** tab, set the font color to **Red** and confirm with **OK**.

6. All necessary settings are now made. Click on **OK** to confirm and create the new rule.

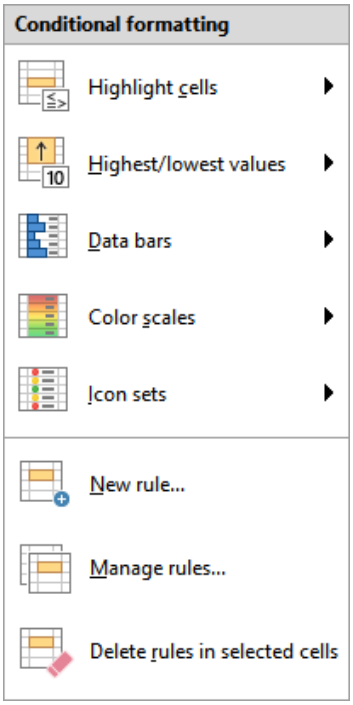
The new formatting rule is now created – and is applied immediately to the selected cells. This has the following effect:

- If the cell content is smaller than or equal to 1000, the cell will be displayed in its original format.
- If, on the other hand, the value is greater than 1000, the conditional formatting will be applied, thus the cell contents will be displayed in red.

Note: You can create as many formatting rules for a cell (or cell range) as you like. For example, you can add a second rule that formats the cell in boldface if it contains a value less than zero.

Tip: Applying formatting rules directly from the dropdown menu

If you choose the ribbon command **Home** | group **Format** | **Conditional formatting** , you can apply some frequently used formatting rules directly from the dropdown menu.



The following types of formatting rules are available at the top of this dropdown menu:

- **Highlight cells:** see description in the next section "Types of conditional formatting", paragraph Format only cells that contain ...
- **Highest/lowest values:** see description in next section "Types of conditional formatting", paragraph Format only upper or lower values
- **Data bars, Color scales, Icon sets:** see description in next section "Types of conditional formatting", paragraph Format all cells based on their values

In the submenus, click on the entry **More rules** to open the dialog box with even more differentiable setting options if required.

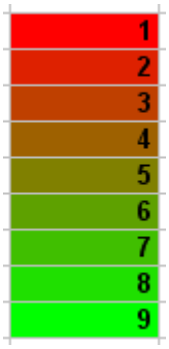
Types of conditional formatting

The previous section described how to apply conditional formatting to cells. In the dialog box that appears, you can choose between the following types of formatting rules:

Format all cells based on their values

This type of conditional formatting does not actually use a condition at all. In fact, it reformats *all* of the previously selected cells – based on the values they contain.

If, for example, you choose a **2-color scale** from red to green, the lowest value will be highlighted in red and the highest value in green. The colors for the values in-between will be calculated automatically. The result will be a color gradient as follows:



There are several different subtypes for this type of formatting rule. They can be selected via the option **Format style**. This contains the following entries:

- **2-color scale**

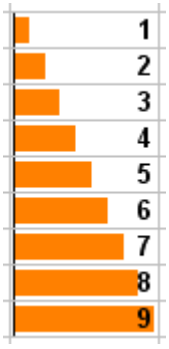
  This is described in the example above.

- **3-color scale**

  This is the same as the 2-color scale, but it also lets you set the color for the midpoint.

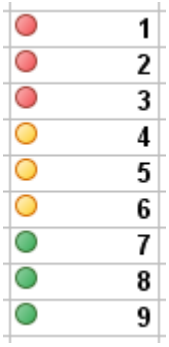
- **Data bars**

  Here, bars that correspond to the relative size of the value are displayed in the background of the cells – similar to a bar chart:



- **Icon sets**

  This displays an icon in each cell, indicating the size of the respective value – for example, a red signal light for the lower third of the values, a yellow one for the middle third and a green light for the upper third:



Format only cells that contain ...

This type of rule reformats only those cells within the current selection that meet the specified *condition*.

Proceed as follows: First, specify the desired condition, using the controls and input fields in the dialog box.

Then, click on the **Format** button and specify the formatting options to be applied for all cells that meet the condition.

An example can be found in the previous section – where we created a formatting rule "If the cell content is greater than 1000, display the cell in red".

Format only upper or lower values

This type of rule reformats only those cells that contain the *highest* or *lowest* values within the current selection.

First, specify which values to reformat – for example, the top 3 values or the top 10% of the values.

Then click on the **Format** button and specify the formatting options to be applied to the corresponding cells.

Format values above or below average

This type of rule reformats only those cells that contain values above or below the *average* of the current selection.

First, specify which values to reformat – for example, all values above the average.

Then click on the **Format** button and specify the formatting options to be applied to the corresponding cells.

Format unique or duplicate values

This type of rule reformats all *unique* values (or *duplicate* values) within the currently selected cells.

First, specify which values to reformat:

- all **unique values** (values that occur only once)
- or all **duplicate values** (values that occur at least twice)

Then click on the **Format** button and specify the formatting options to be applied to the corresponding cells.

Use a formula to determine which cells to format

Here, only those cells are reformatted for which the specified *formula* returns TRUE.

To do so, enter the desired formula in the dialog box. Note that only formulas that return a logical value (thus, TRUE or FALSE) are permitted. See also notes below.

Then click on the **Format** button and specify the formatting options to be applied to the corresponding cells.

Some notes:

- **Creating suitable formulas**

  You can use any kind of formula for the condition. You only have to ensure that the result of the formula is a logical value (thus, TRUE or FALSE).

  Examples:

  If you enter the formula "SUM($A$1:$C$3) > 42", the conditional formatting will be applied if the sum of the cell range A1:C3 is greater than 42.

  If you enter the formula "ISEVEN(ROW())", the conditional formatting will be applied if the current cell is located in a row with an even row number.

- **Using absolute and relative cell addresses**

  Note that the formula can also use *relative* cell references in addition to *absolute* cell references (as in the above example). These are treated as follows:

  An *absolute* cell reference – for example, $A$1 – always refers to cell A1.

  On the other hand, a *relative* cell reference – for example, A1 – refers to the cell in the upper left corner of the selection. That means:

  If you apply conditional formatting to an individual cell, A1 refers to the relevant cell.

  If you previously selected a cell range, A1 refers to the cell in the upper left corner of the selection, A2 to the cell below it, etc.

Managing formatting rules

On the ribbon tab **Home** | group **Format | Conditional formatting** , you can choose the command **Manage rules** to manage the conditional formatting in a document, including editing and deleting it.

Use this command as follows:

1. Navigate to the cell whose formatting rules you want to edit. Of course, you can also select multiple cells.
2. Choose the ribbon command **Home** | group **Format | Conditional formatting | Manage rules**.
3. A dialog box appears.

   Note: By default, this dialog box displays only the rules for the *current* cell (or selection). If you want to see *all* rules created in the entire worksheet instead, set the option **Source** to **Current worksheet**. For more information, see below.
4. Make the desired changes to the formatting rules (see below).
5. Close the dialog box by clicking on **OK**.

Using the dialog box

The dialog box for this command has the following options:

- **Source**

  Here, you can determine which formatting rules should be displayed in the dialog box:

Only selected cells: This shows only rules that affect the currently selected cells (or the current cell if no cells are currently selected)

Current worksheet: This shows all rules for the current worksheet

Sheet <name>: This shows all rules for a different worksheet (if available)

- **Rules** list

  The **Rules** list displays all rules for the current selection or the entire worksheet (depending on what you selected for **Source**).

  Rules that affect the currently selected cells are displayed in boldface.

  Stop: You usually don't have to concern yourself with the checkboxes in the **Stop** column. They are only necessary for compatibility reasons with older documents. (Background info: In older versions of PlanMaker and Excel, conditional formatting for a cell that had multiple formatting rules was "stopped" as soon as *one* of these rules applied. All further rules for this cell were then ignored. This is no longer the case in current versions of PlanMaker and Excel: Here, *all* rules defined for a cell are always observed.)

- Arrow buttons next to the **Rules** list

  The arrow buttons next to the **Rules** list enable you to change the ranking order of the formatting rules.

  To change the ranking of a rule, select it in the list and then click on the up button (increase ranking) or down button (decrease ranking).

  This may be necessary if a cell range has been assigned competing formatting rules. Simple example: A rule wants to color the cell green, but another rule wants to color it red.

  The *higher* a rule is in the list, the higher its ranking.

  Note: Please note that the list does *not* reflect the order in which the rules are processed – on the contrary: The list is processed from bottom to top, so that the rules with the highest ranking are applied last.

- **Apply to**

  Here, you can change the cell range to which you want to apply a rule.

  To do so, select the rule in the **Rules** list and enter the desired cell range in this input field.

- **New** button

  This creates a new conditional formatting rule for the cells currently selected in the worksheet, just like the ribbon command **Home** | group **Format** | **Conditional formatting** | **New rule**.

  PlanMaker displays again the dialog box for creating rules, which was described in Creating new formatting rules.

- **Edit** button

  This lets you edit the conditional formatting rule currently selected in the **Rules** list.

  Note: The rule is then changed in *all* cells to which it has been applied, regardless of which cells are currently selected in the worksheet.

  PlanMaker displays again the dialog box for creating rules, which was described in Creating new formatting rules.

- **Delete** button

  This deletes the rule currently selected in the **Rules** list.

  Note: This will remove the rule from *all* cells to which it has been applied, regardless of which cells are currently selected in the worksheet.

  For more information on Removing conditional formatting, see the next section.

- **Go to** button

  This closes the dialog box and jumps to the cell range for the rule that is currently selected in the **Rules** list.

Removing conditional formatting

Conditional formatting can be removed in two ways:

A) Deleting formatting rules

When you delete a formatting rule, it will be removed from all cells to which it has been applied, regardless of which cells are currently selected in the worksheet.

Proceed as follows:

1. Choose the ribbon command **Home** | group **Format** | **Conditional formatting** | **Manage rules**.
2. A dialog box appears.

   Note: By default, this dialog box displays only the rules for the *current* cell (or selection). If you want to see *all* rules created in the entire worksheet instead, set the option **Source** to **Current worksheet**.
3. Select the rule to be deleted in the **Rules** list.
4. Click on the **Delete** button.

The selected rule will now be deleted – and thus removed from all cells it affected.

B) Removing all conditional formatting from a cell range

There's another method for removing conditional formatting, which removes *all* conditional formatting from the selected cells.

Proceed as follows:

1. Select the cells from which you want to remove all conditional formatting.
2. Choose the ribbon command **Home** | group **Format** | **Conditional formatting** | **Delete rules in selected cells**.

   (Alternatively, choose **Home** | group **Contents** | **Delete** and select **Conditional formatting**, which does exactly the same thing.)

This will "free" the selected cells from any conditional formatting rules applied to them.

All formatting rules that are no longer in use will be deleted automatically.

Input validation

When you apply *input validation* to a cell, you can achieve the following:

- Ensure that only certain types of values (for example, numbers) within certain limits (for example, only values between 10 and 20) are considered valid in this cell.
- Display an info text whenever the user goes to this cell (for example, "Please enter only numbers between 10 and 20.").
- Display an error, warning or information message if the user nevertheless enters values that do not match the input validation.

This is useful when you are creating tables that will be filled in by other people and you want to prevent invalid values from being entered.

Applying input validation (example)

If, for example, you would like to limit the input of a certain cell to numeric values between 10 and 20, you would proceed as follows:

1. Select the desired cells.
2. Choose the ribbon command **Review | Input validation** .
3. On the **Settings** tab, you can specify which types of values are permitted and set limits for the allowed values.

   In our example, you would set the following:

   Allow: "Decimal numbers"

   with values: "between"

   Minimum: 10

   Maximum: 20
4. On the **Input message** tab, you have the option of entering an info text that will be displayed whenever the user goes to the cell.

   For example, you could add the message, "Please only enter numbers between 10 and 20." Of course, you can also refrain from entering an input message.
5. On the **Error message** tab, you can specify the message that should appear when the user types invalid values into the cells.

   The icons in the **Type** area allow you to choose the type of error message that is displayed. (See explanation below.)

Important: The type of message also determines whether the user is *actually* allowed to enter invalid values. PlanMaker strictly rejects invalid entries only when the type **Error message** (first icon) is selected!

Furthermore, you can enter the actual text of the message. If you do not enter anything, a default message will be issued.

For our example, set the type to **Error message** by clicking on the first icon and enter the desired message (for example, the title "Invalid entry" and the text "Only numbers between 10 and 20 are allowed here.")

6. Confirm with **OK**.

Only values between 10 and 20 will now be permitted in the selected cells. If the user enters something else, an error message will appear.

Note: Input validation only becomes active when the user *types* in values. If the user pastes invalid values into the cell through a *copy and paste* operation (for example, with the ribbon command **File | Paste**), however, input validation will not intervene!

For more information on the options of the dialog box *Input validation*, see the next section Settings for input validation.

Disabling input validation again

If you want to disable the input validation again, select the relevant cells and proceed as follows:

On the ribbon tab **Home** | group **Contents** | **Delete** , choose the command **Input validation** in the dropdown menu.

Alternatively, you can use the ribbon command **Review** | **Input validation** again. In the dialog box on the **Settings** tab, select "All values" again under **Allow:**.

Settings for input validation

As described in the previous section, you can limit the allowed values using the dialog box of the ribbon command **Review** | **Input validation**. The following options are available here:

"Settings" tab

- **Allow:**

  You use these options to define which types of values are allowed:

Option	Explanation
All values	Allow all types of values (thus disable the input validation)

Integer numbers	Allow only integer numbers
Decimal numbers	Allow only decimal numbers
List entries	Allow only values from a predefined list of permitted values. Method 1: Enter a comma-separated list of the allowed values in the **Source** field – for example, dog,cat,mouse or 1,2,3,4,5,6. Method 2: Alternatively, you can enter an equal sign followed by a cell range that contains all valid entries. If, for example, cells C1 to C3 contain the values "dog", "cat" and "mouse" and you enter =C1:C3 in **Source**, only those three values will be allowed. The **Options** button can be used to set additional options (related to Method 2).
Date	Allow only date values
Time	Allow only time values
Text length	Allow only entries (including numbers) with the specified text length
User-defined	Allow only entries which satisfy the specified condition Enter the condition in the **Formula** field. (For more information on this, see the end of this section.)

- **with values**

  Here, you can further limit the allowed values. If, for example, you select the option **between**, you can specify minimum and maximum values between which the values must lie.

- **Minimum, Maximum, etc.**

  Here, you define the limits of the entries. If you enter 10 as the **Minimum** and 20 as the **Maximum**, only values between 10 and 20 will be accepted.

  You can also use formulas here. If you enter C1 as the **Minimum**, the contents of the cell must be greater than or equal to the cell contents of cell C1. If you enter SUM(D2:D8), the contents of the cell must be greater than or equal to the sum of cells D2 to D8.

- **Use dropdown**

  This option is only available when **List entries** is selected under **Allow**. If this option is enabled, an arrow will appear on the right edge of the cell when the user goes to it. If you click on this arrow, the program will display a dropdown list of all permitted values from which the user can select.

- **Ignore empty cells**

  If this option is enabled, input validation is ignored for empty cells. If the user deletes the contents of a cell, no input validation occurs in the cell until the user enters another value.

"Input message" tab

On this tab, you can enter an info text for filling in the cells (for example, "Please only enter numbers between 10 and 20."). This information will be displayed when the user goes to the cell.

This tab has the following options:

- **Show input message when cell is selected**

  This option determines whether the input message should be shown or not.

- **Title**

  Here, you can specify the title of the message.

- **Message**

  Here, you can enter the text of the message.

"Error message" tab

On this tab, you can specify the error, warning or information message which should appear when the user enters data that does not match the input validation.

This tab has the following options:

- **Show error message after invalid data is entered**

  This option determines whether the error message should be shown or not.

  Note: Normally, you should leave this option enabled, since input validation is of little use if the user is not informed that he is entering invalid data.

- **Type**

  Here, you can specify the type of message. Simply click on the desired icon.

  Important: The type of message also determines whether the user is *actually* allowed to enter invalid values. PlanMaker strictly rejects invalid entries only when the type **Error message** (first icon) is selected!

  The following types of messages are available:

  Error message (first icon): PlanMaker displays an error message. The user is *forced* to enter a valid value; invalid values will not be accepted.

  Warning message (second icon): PlanMaker displays a warning message. The user can *decide* whether to still proceed with the invalid entry.

  Information message (third icon): PlanMaker only displays an information message. The user must only *acknowledge* this message and the invalid data will be accepted (as long as the user does not click on **Cancel**).

- **Title**

  Here, you can specify the title of the message.

  If you do not enter anything here, a default text will be used.

- **Message**

  Here, you can enter the text of the message.

  If you do not enter anything here, a default text will be used.

Tip: If you would like to have the specified minimum and maximum values displayed in the message, simply use the wildcards %min and %max. Example: "Error – Only values between %min and %max are allowed here."

Examples

Finally, some examples of how to use this feature:

- **Allow only values within the specified range**

  On the **Settings** tab, first select the desired type of values. If only dates are to be allowed, set the option **Allow** to "Date". If only numbers are to be allowed, set the option to "Decimal numbers" – and so on.

  You can then use the option **With values** to set the allowed limits. If, for example, only values greater than 1 are to be allowed, select the "greater than" option and enter 1 as the **Minimum**.

- **Allow only values of a certain length**

  If you would like to prevent the user from entering values with lengths greater than 40 characters, select the following settings:

  Allow: "Text length"

  with values: "smaller than or equal to"

  Maximum: 40

  Note: Numbers, periods, commas, signs, etc., are considered to be just as much a character as letters.

- **Using your own formulas for input validation**

  Advanced users can also select "User-defined" under **Allow** and then specify any formula for input validation.

  To do so, use a formula that returns a *logical* value (thus, TRUE or FALSE). If the result of the formula is TRUE, PlanMaker will consider the entry to be valid; otherwise it will be considered invalid.

  Example: If you enter the formula SUM($A$1:$C$3) > 42, the input will be considered valid only if the sum of the A1:C3 cell range is greater than 42.

  Note: Notice that *relative* cell references can also be used in such formulas in addition to *absolute* cell references (as in the example above):

  Absolute cell references such as $A$1 always refer to cell A1.

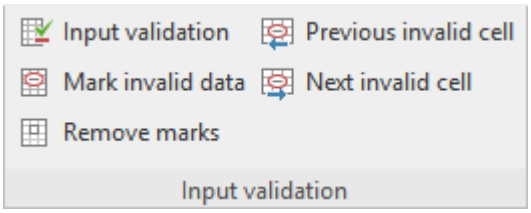
  Relative cell references such as A1, on the other hand, refer to the cell in the upper left corner of the selected cells. If you apply input validation to only one cell, A1 refers to this cell. If you previously selected a cell range, A1 refers to the cell in the upper left corner, A2 refers to the cell below, etc.

Detect invalid data

On the ribbon tab **Review** | group **Input validation**, you will find commands which can be used to detect cells that contain an invalid value.

Note: A cell contains an invalid value if the command **Input validation** has been used to specify that the cell can only contain certain values, and the current cell content does not match this check.

To find and analyze such errors, use the following commands in the command group:



- **Input validation**

  This command opens the dialog box *Input validation* for defining allowed values. For more information, see previous sections.

- **Mark invalid data**

  This command inserts a red circle around each cell (in the current worksheet) that contains an invalid value.

- **Remove** (invalid data) **marks**

  This removes all red circles (from the current worksheet) that were added with the above command.

- **Previous invalid cell**

  This command jumps to the previous cell that contains an invalid value.

- **Next invalid cell**

  This command jumps to the next cell that contains an invalid value.

Notes

- The marks for invalid data also appear in the printout.
- However, the marks are not saved in the document. Thus, when you save a document and then reopen it, all marks are removed.

Format painter

You can use the command **Home | Format painter** to instantly transfer the cell format from one cell to other cells.

To do so, proceed as follows:

1. First, click on the cell whose format you want to transfer to other cells.
2. Click on the **Format painter** icon on the ribbon tab **Home|** group **Edit**.

The mouse cursor will now change into crosshairs with a little brush:

3. Drag the mouse (with the mouse button held down) over the cell ranges to which the format is to be transferred. You can apply this function to any number of cell ranges or individual cells (simply click on individual cells).
4. When you are finished, click on the **Format painter** icon again or simply press the **Esc** key.

Both the character format (font, text styles, etc.) and the cell format (number format, alignment, etc.) will be applied to the selected cells.

Transferring the format of entire cell ranges

You can also transfer the format of entire cell *ranges* to other cells. To do so, proceed as described above, but in step 1., select the cell range whose format you want to transfer.

If, for example, you select three cells that are formatted with red, green and blue backgrounds and transfer their formatting to another cell range, that range will be formatted alternately with red, green, and blue backgrounds.

Shortcut keys

You can also use the keyboard to perform the **Format painter** function:

Command	Windows/Linux	Mac
Format painter (copying step)	**Ctrl+Shift+C**	**Cmd+Shift+C**
Format painter (pasting step)	**Ctrl+Shift+V**	**Cmd+Shift+V**

Page setup

In this section, you will learn about the options available for formatting a worksheet for printing.

The following topics are covered:

- **Setting page format**

  On the ribbon tab **Layout** | group **Page setup**, you can set the *page format* for the current worksheet. This includes settings such as page size, page orientation (portrait or landscape) and page margins.

- **Setting page format options**

  In the dialog box of the ribbon tab **Layout** | group **Page setup** ◢, you will find various settings on the **Options** tab for page numbers, print order, etc., and you can specify which parts of the worksheet should be printed.

- **Setting up header and footer**

  You can use the command **Header and footer** on the ribbon tab **Layout** | group **Page setup** to add a header and/or footer to the document, which will be printed at the start or end of each printed page of the worksheet.

- **Adjusting page breaks manually**

  Page formatting also includes the option of manually controlling the automatic page break if necessary. This can be done with the command **Page break** on the ribbon tab **Layout** | group **Page setup**.

See the following pages for more information.

Setting page format

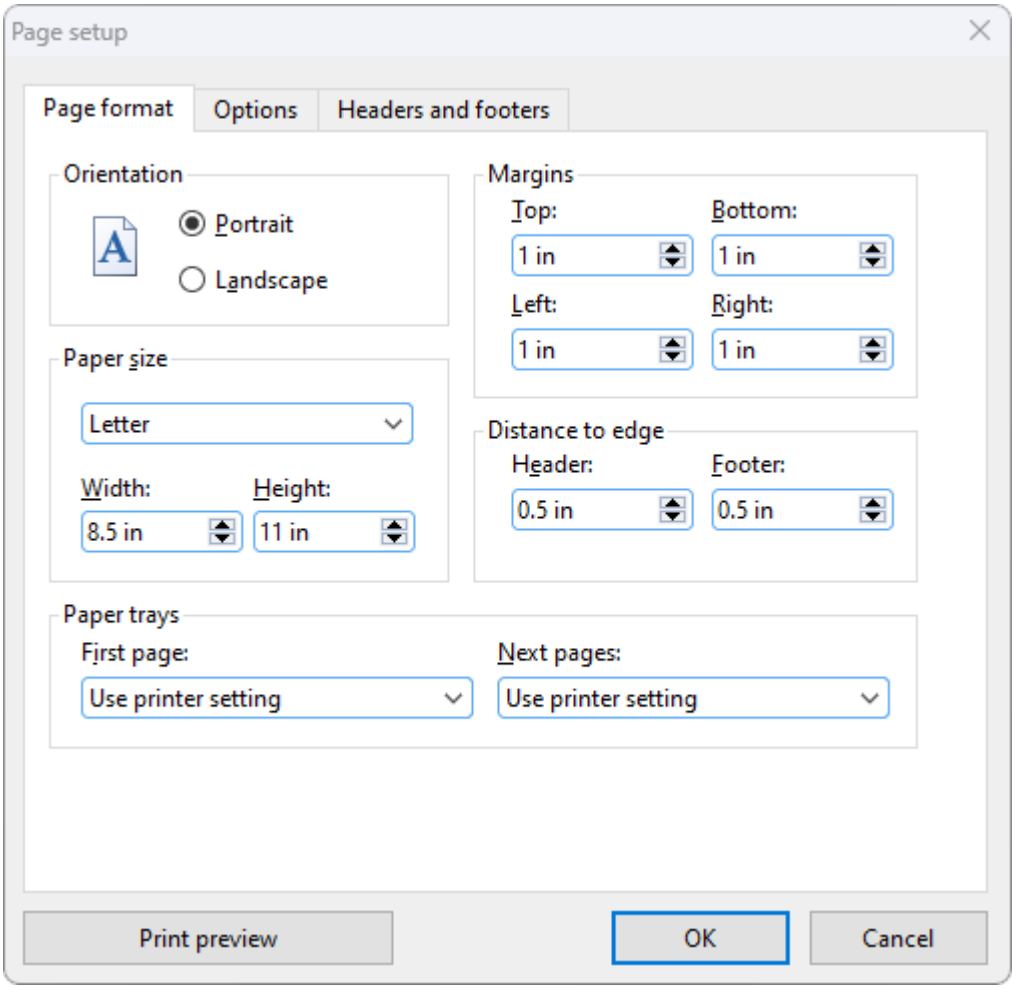
On the ribbon tab **Layout** | group **Page setup**, you can define the page format of a worksheet. This includes settings such as page size and orientation (portrait or landscape).

Note: You can set up a different page format for each worksheet in a document. For example, you can choose portrait for one worksheet and landscape for another.

Applying via the dialog box of the command group "Page setup"

On the ribbon tab **Layout** | group **Page setup**, click on the group arrow ◢ in the lower right corner to open the dialog box and switch to the **Page format** tab.

Alternatively, you can open the dialog box with the ribbon command **File** | group **Print** | **Page setup**.



Then make the desired settings:

- **Orientation**

  Here, you can specify the **Orientation** of the document on the printed page: **Portrait** or **Landscape**.

- **Paper size**

  You can specify the paper size for the printed pages here. The dropdown list **Paper size** contains all paper sizes that are supported by the current printer. If you want to specify a size that is not listed, you can do so by manually specifying values in the fields **Width** and **Height**.

- **Margins**

  You can set the page margins here

- **Distance to edge**

  Here, you can determine how much space should be left between headers and footers and the upper or lower edge of the sheet.

  Note: Headers and footers are always printed at the top and bottom of the page, respectively. If they do not appear on the printout, check whether a) the Distance to edge is too large or b) the top/bottom margin is too small.

- **Paper trays**

  Only available in Windows version: If your printer has multiple paper trays, you can set the tray which the printer should use. If you have selected the default setting **Use printer setting**, PlanMaker does not affect the paper feed. If, for example, you want the first page of your document to be printed on paper from tray 1 and the remaining pages on page from tray 2, you should change this setting accordingly.

Tip: If you right-click on a worksheet tab at the bottom, you can choose the context menu command **Copy page setup** to copy all configured page setups from one worksheet to another. See also Managing worksheets.

Applying via the icons of the command group "Page setup"

Common settings of the dialog box can also be found via the icons on the ribbon tab **Layout** | group **Page setup**. Click on the black arrow of the respective icon to open the corresponding dropdown menu.

Margins (corresponds to the options for **Margins** in the above dialog box):
You will find a selection of predefined values here. If required, select **More** in the dropdown menu to access the dialog box for customizations.

Page orientation (corresponds to the options for **Orientation** in the above dialog box):
portrait or landscape

Page size (corresponds to the options for **Paper size** in the above dialog):
Here you will find a selection of predefined values. If required, select **More** at the bottom of the dropdown menu or click on the icon itself to perform customizations via the dialog box.

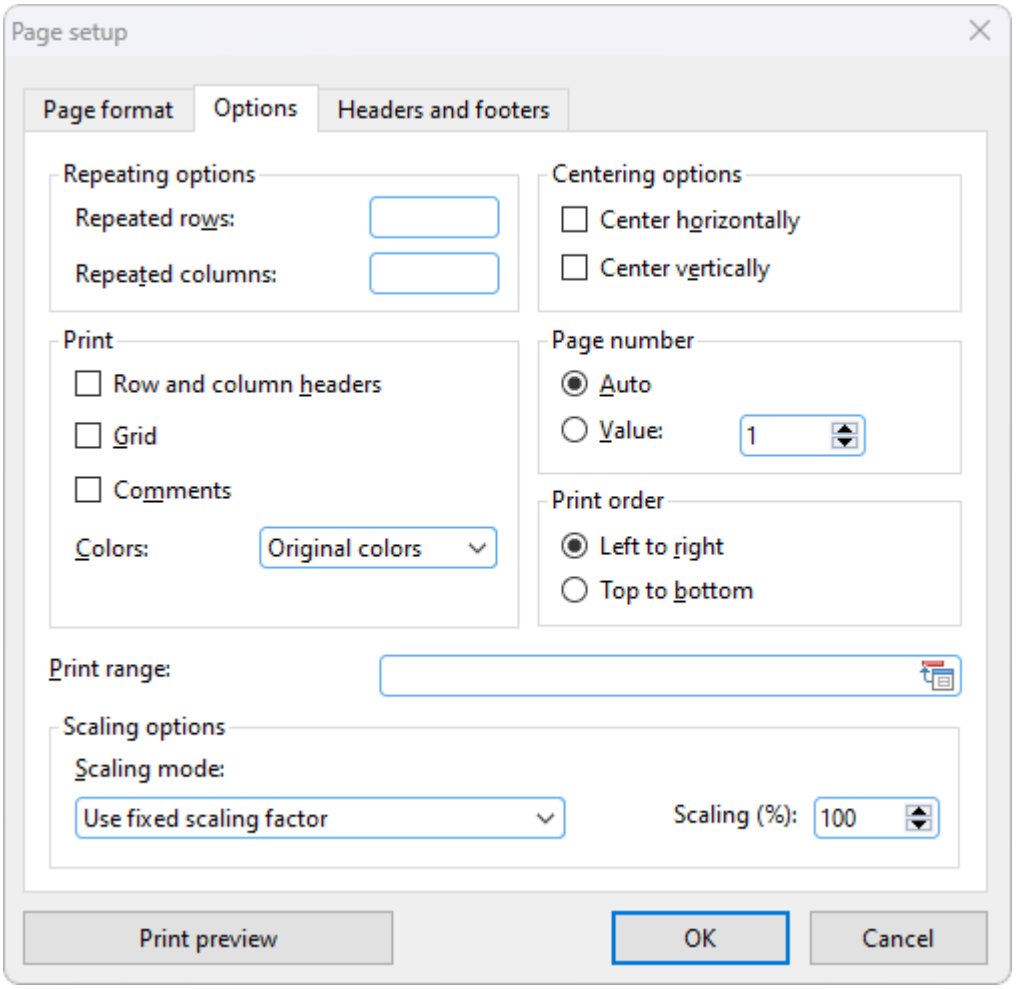
Setting page format options

On the **Options** tab of the dialog box "Page setup", you can make settings that affect the printout of the current worksheet.

Note: These options must be set separately for each worksheet.

You open the dialog box by clicking on the group arrow ◢ in the bottom right corner of the ribbon tab **Layout** | group **Page setup**. Switch to the **Options** tab.

Alternatively, you can open the dialog box with the ribbon command **File** | group **Print** | **Page setup**.



Here, you will find the following settings:

Repeated rows/Repeated columns

If the worksheet contains row labels or column labels, and you want them to be repeated on all printed pages, enter them here.

Use the following notation: *FirstRow:LastRow* (or *FirstColumn:LastColumn*, respectively)

If, for example, the first row of a long list of data contains a label to be repeated on all pages, enter 1:1 in the **Repeated rows** field to ensure this row is repeated on every page of the printout.

If, on the other hand, you want columns A to C to be repeated on each page, enter A:C in **Repeated columns**.

Print

Select any of these options to include the following worksheet components in the printout:

- **Row and column headers:** Print the row headers located to the left of the worksheet and labeled 1, 2, 3, etc. or the column headers located above the worksheet and labeled A, B, C, etc.
- **Grid:** Print the gray gridlines between the cells.
- **Comments:** Print the comments added with the ribbon command **Insert | Comment**.

Note: Only comments that you have enabled for permanent display via the **Comment visibility** button will be printed. For more information, see Using comments.

In addition, you can use the option **Colors** to specify if the printout should be rendered in color:

- **Original colors:** The printout is rendered in its original colors.
- **Grayscale:** Colored text is rendered in black and white, while colored objects (pictures, charts, etc.) are converted to shades of gray.
- **Outlines only:** Colored text is rendered in black and white, while colored objects are reduced to their outline in black and white.

Centering options

If the option **Center horizontally** is enabled, the worksheet's content will be horizontally centered on the page when printed.

If the option **Center vertically** is enabled, the worksheet's content will be vertically centered on the page when printed.

Page number

This option lets you define the starting number for page numbers:

Auto: Pages are numbered starting with 1.

Value: Pages are numbered starting with the specified value.

Tip: To add page numbers to a worksheet, insert a **Page number** field into the header or footer (see next section).

Print order

Here, you define the order in which PlanMaker should output worksheets that are wider than a printed page.

- **Left to right**

  Print from left to right, and then from top to bottom.

- **Top to bottom**

  Print from top to bottom, and then from left to right.

Print range

Here, you can define a *print range* for the current worksheet.

When a print range is defined, only those cells that are within this cell range will appear when the worksheet is printed. The rest of the worksheet will be omitted.

By default, this input field is empty, thus no print range is defined and the *entire* worksheet will be printed. If you type in a cell range (or the name of a named range) here, however, only this range will be generally output when the worksheet is printed.

A print range can be defined individually for each worksheet of the document.

Important: When you save a document, this setting will be *permanently* stored in the document. Thus, if you subsequently open and print this document again, the printout will *still* contain only those cells within the print range. If you want the entire worksheet to be printed again, simply clear the **Print range** input field in this dialog box to remove the print range.

Tip for working with print ranges:

Alternatively, there is a slightly more convenient method: You can also define a print range for a worksheet by selecting the desired cell range and then choosing the command **Define print range** directly from the ribbon tab **Layout |** group **Page setup**.

To do so, either click on the icon itself – or click on the icon's small arrow and select the command **Define print range** in the dropdown menu. If you select **Remove print range** here, you can remove the print range again.

Note: To ensure that only the defined print range is printed, the option **Ignore print range** must be disabled in the dialog box of the command **File | Print**. This is the default setting. For more information, see Printing a document.

Tip for checking the print range:

To visually check the selected print range, you can use the option **Page break preview** on the ribbon tab **View**. The selected print range is indicated by the white area and can be further adjusted by dragging the wrap lines with the mouse. For more information, see Adjusting page breaks manually.

Scaling options

These options enable you to scale *printouts* of the document up or down.

By default, a fixed scaling factor of 100% is set (which means that the original size is used).

To change the scaling, first select a **Scaling mode**, and then set the desired value as follows:

- **Use fixed scaling factor:** Here, you can set the scaling factor manually (in percent). Only values between 20 and 200 percent are permitted. If, for example, you set the scaling factor to 50%, the printout will be scaled down to half its original size. If you set it to 200%, the printout will be doubled in size.

For the other 4 scaling modes, the scaling factor is calculated automatically according to your settings:

- **Specify height in pages:** Here, you specify the desired height of the printout in pages. PlanMaker will determine the corresponding scaling factor automatically.
- **Specify width in pages:** Here, you specify the desired width of the printout.
- **Specify height and width in pages:** Here, you specify the desired maximum height and width of the printout.
- **Specify number of pages in total:** Here, you specify how many pages the printout should have in total.

Note: PlanMaker does not allow the automatically determined scaling factor to be less than 20%. If this value is not reached, PlanMaker automatically uses the minimum allowed value of 20%. The same applies to the maximum value of 200%.

Setting up header and footer

If desired, you can add *headers* and/or *footers* to a worksheet. Headers/footers are repeated on the top/bottom of *each* printed page of a worksheet. You can also insert useful fields here, such as page number, file name, print date, etc.

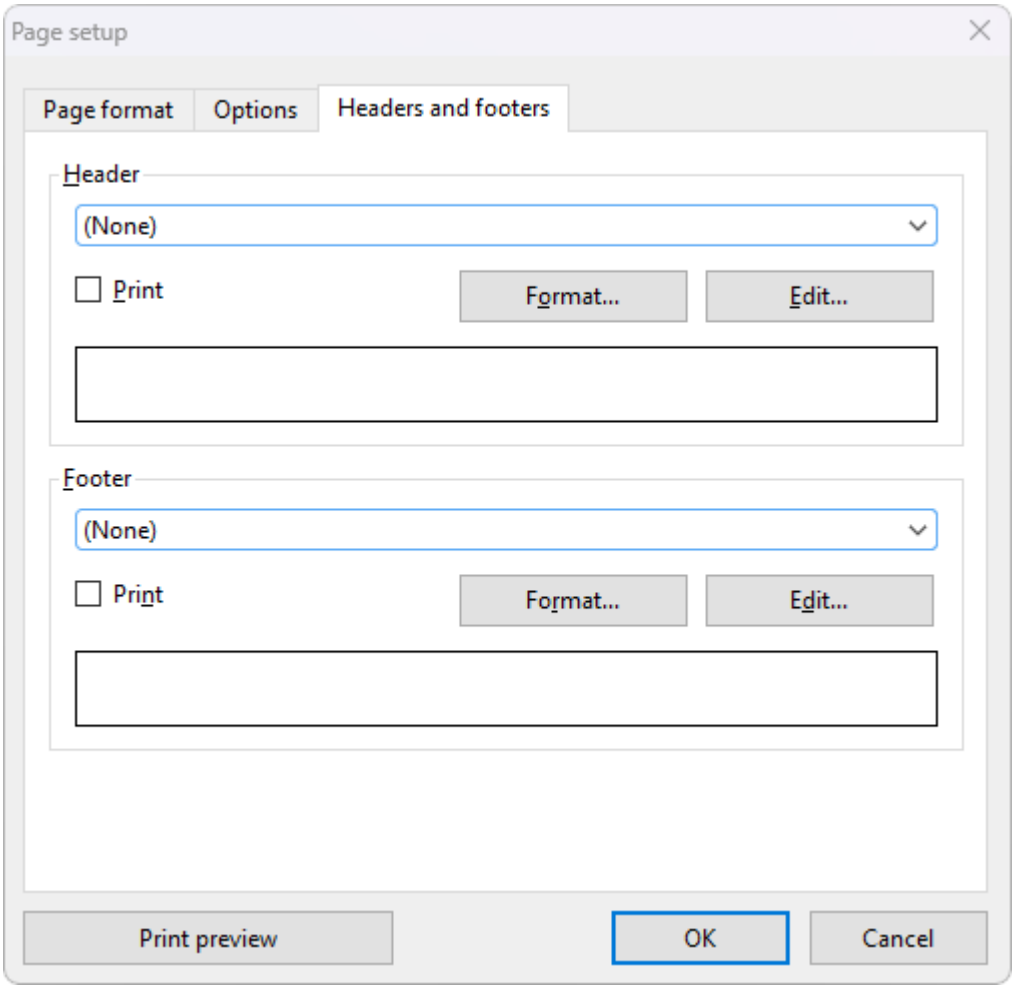
Note: Each worksheet of a document can have its own headers/footers.

To edit the header of the current worksheet, for example, proceed as follows:

1. On the ribbon tab **Layout |** group **Page setup**, choose the command **Header and footer**.

   Alternatively, you will find the command on the ribbon tab **Insert |** group **Text**.

   PlanMaker opens the following dialog box:



2. To apply one of the predefined headers, select it from the dropdown list box at the top of the dialog box.

   To customize the header to your liking, click on the **Edit** button (see "Editing headers and footers" below).

   To change the format of the header, click on the **Format** button (see "Formatting headers and footers" below).

3. Confirm with **OK**.

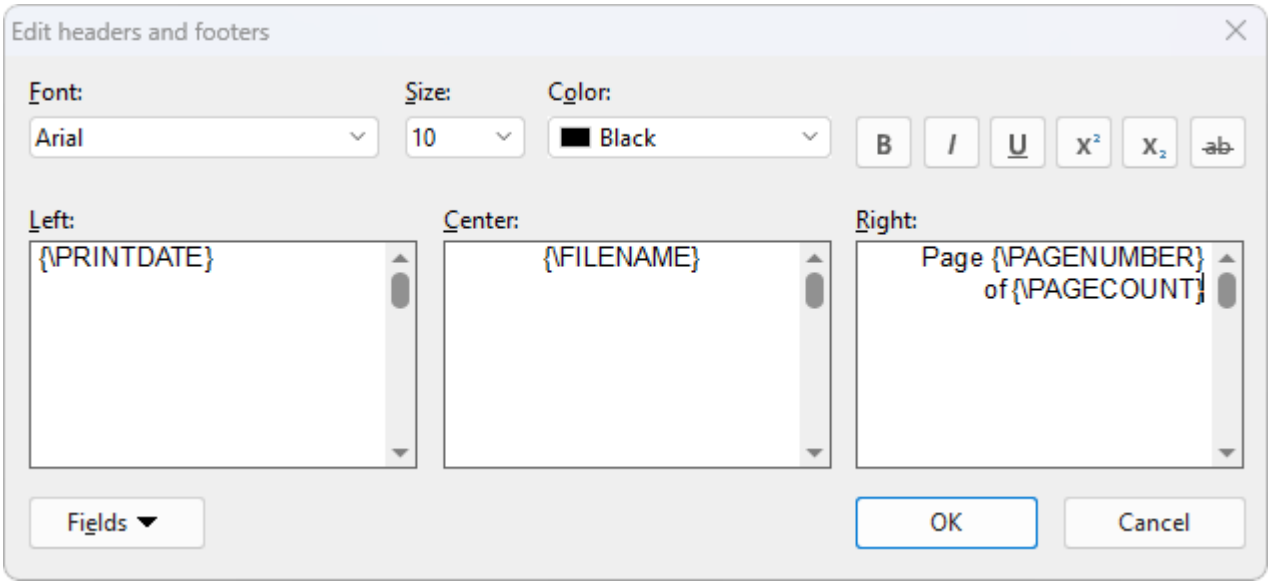
The header has now been changed accordingly.

Editing footers can be performed in the same manner as headers (using the dialog controls at the bottom of the dialog box).

Tip: Headers/footers are not displayed in the worksheet on the screen – they appear in the *printout* only. However, you can view the appearance of the header and footer with the **Preview** button or also at any time with the ribbon command **File** | group **Print** | **Print Preview**.

Editing headers and footers

When you click on the **Edit** button in the aforementioned dialog box, you can edit the worksheet's headers and footers as desired.



Some notes:

- **Headers/footers can contain up to three sections**

  The input fields in this dialog box are split into three parts. This is due to the fact that headers/footers can consist of up to three sections: a left section (left-aligned), a middle section (centered) and a right section (right-aligned). This allows you to display, for example, the date on the left, the file name in the middle and the page number in the right section of the header/footer.

  Of course, you can also fill in only one of the three sections, for example, if you only want to have the file name displayed on the left.

- **Using fields for page numbers, file name, etc.**

  To insert fields into headers/footers, click on the **Fields** button.

  Fields are wildcards that symbolize a specific piece of information. If, for example, you insert a field for the page number, the wildcard {\PAGE NUMBER} is in the input field. When printing, this symbolic information is then automatically replaced by the page number.

  The following fields are available:

Field name	Output
Document name	File name of the document (for example, "My Table.pmdx")
Document path	File path of the document (for example, "C:\Tables")
Document name and path	File name and path of the document (for example, "C:\Tables\My Table.pmdx")
Worksheet name	Name of the current worksheet
Page number	Page number
Number of pages	Total number of pages

Print date	Current date (date of printing)
Print time	Current time (time of printing)
Date last changed	Date the document was last changed (and saved)
Creation date	Date the document was created

To insert one of these fields, click on the **Fields** button and then click on the desired field.

- **Changing the character format**

  To change the character format (font, font size, etc.) of text in the header/footer, use the controls located at the top of the dialog box.

  Note: First, select the text to be formatted (in one of the three input fields).

Formatting headers and footers

In addition to changing the character format of headers/footers (see above) you can also apply borders and shades or change the alignment and margins. To do so, click on the **Format** button in the *main* dialog box of the ribbon command **Layout | Header and footer**.

The program displays another dialog box containing the following tabs:

- **Shading tab**

  Here, you can add a colored shade or a pattern to the headers/footers. The controls in this dialog box are identical to those described in Shading.

- **Borders tab**

  Here, you can add a border or individual border lines to the headers/footers. The controls in this dialog box are identical to those described in Borders.

- **Alignment tab**

  Here, you can change the inner margins and the vertical alignment of the headers/footers.

Tip: The distance between headers/footers and the top/bottom edge of the page cannot be changed here, as they must be changed in the *main* dialog box "Page setup" on the **Page format** tab.

Disabling headers/footers temporarily

If you want to print a document occasionally without its headers or footers, choose the command **Layout | Header and footer** and disable the option **Print** in the dialog box. The headers or footers will then be retained but will no longer appear in printouts.

If you want the headers/footers to be printed again, simply enable the option **Print** again.

Adjusting page breaks manually

Page formatting also includes the possibility of manually controlling the automatic page break if required.

Normally, PlanMaker includes the page break without your intervention: It automatically inserts a page break at the end of a printed page (depending on the page format you have set, see Setting page format). If you take a closer look at a worksheet, you will notice that some of the gridlines between cells are darker. These lines indicate where PlanMaker has inserted automatic page breaks.

Note: On the ribbon tab **View** | group **View**, enable the option **Show page breaks** if the dark gridlines for page breaks are not displayed.

Inserting a manual page break

If necessary, you have the option of manually inserting "hard" page breaks which override PlanMaker's automatic page breaks. To do so, choose the ribbon command **Layout** | group **Page setup** | **Page break**. In the dropdown menu of this command, you will find the following options:

For horizontal page break

- **Insert before row** inserts a horizontal page break above the current row.
- **Delete before row** deletes such a page break. Navigate to a cell located directly below the page break before selecting this command.
- With the options **Move break one row up/down**, you can change the position of the page break by one row at a time. Navigate to a cell located directly below the page break before selecting this option.

  Note: The latter options are only enabled after you have inserted a horizontal page break.

For vertical page break

- **Insert before column** inserts a vertical page break to the left of the current column.
- **Delete before column** deletes such a page break. Navigate to a cell located directly to the right of the page break before selecting this option.
- With the options **Move break one column left/right**, you can change the position of the page break by one column at a time. Navigate to a cell located directly to the right of the page break before selecting this option.

  Note: The latter options are only enabled after you have inserted a horizontal page break.

The command **Delete all page breaks** removes all manually inserted page breaks again.

Tip: Using the "Page break preview"

To visually check the page breaks, you can enable the option **Page break preview** on the ribbon tab **View** | group **View**. You will then see printout pages with their page numbers (displayed as "watermarks" which are not printed) clearly arranged with page breaks.

The area that is filled with data is automatically marked in white by PlanMaker and thus – without further adjustments – leads to the displayed printout pages.

Adjusting manual page break with the mouse

You can manually adjust the vertical/horizontal page breaks in the **Page break view** by dragging a page break (recognizable by the green dashed line) while holding down the left mouse button. The page break line will change to a red line, indicating that the automatic page break will become a manual one. The page numbers are recalculated in the process.

The manual page break adjusted in this manner does the following:

- By moving a page break with the mouse, for example, you can now enlarge the range that will be on a page when printing.
- In the dialog box **Page setup** (see Setting page format options), the **Scaling mode** is automatically set to the option *Use fixed scaling factor* and the value for *Scaling (%)* is adjusted to the changed page ranges.

  Explanation: The value for *Scaling (%)* is never scaled up by moving the page breaks with the mouse. However, you can enter a higher scaling value here so that a small page range is enlarged when displayed on a printed page.
- Above this (in the same dialog box), a *Print range* is defined automatically if you have moved the *outer edges* of the white shaded area with the mouse.

Resetting manual page breaks

To reset the page breaks (that you have changed with the mouse) to their original state, you can proceed as follows:

Open the dialog box **Page setup** (see Setting page format options). Here, set the value for *Scaling (%)* back to 100% and, if necessary, remove the newly created *Print range* in the input field above.

Searching and replacing

This chapter covers information on searching (and replacing) in tables:

- **Search** and **replace**

  Use PlanMaker's search options to search a table for a specific search term – and replace the occurrences of the search term with something else if necessary.

- **Jumping to specific cell addresses**

  Use the command **Go to …** to jump to specific cells, cell ranges, or named ranges.

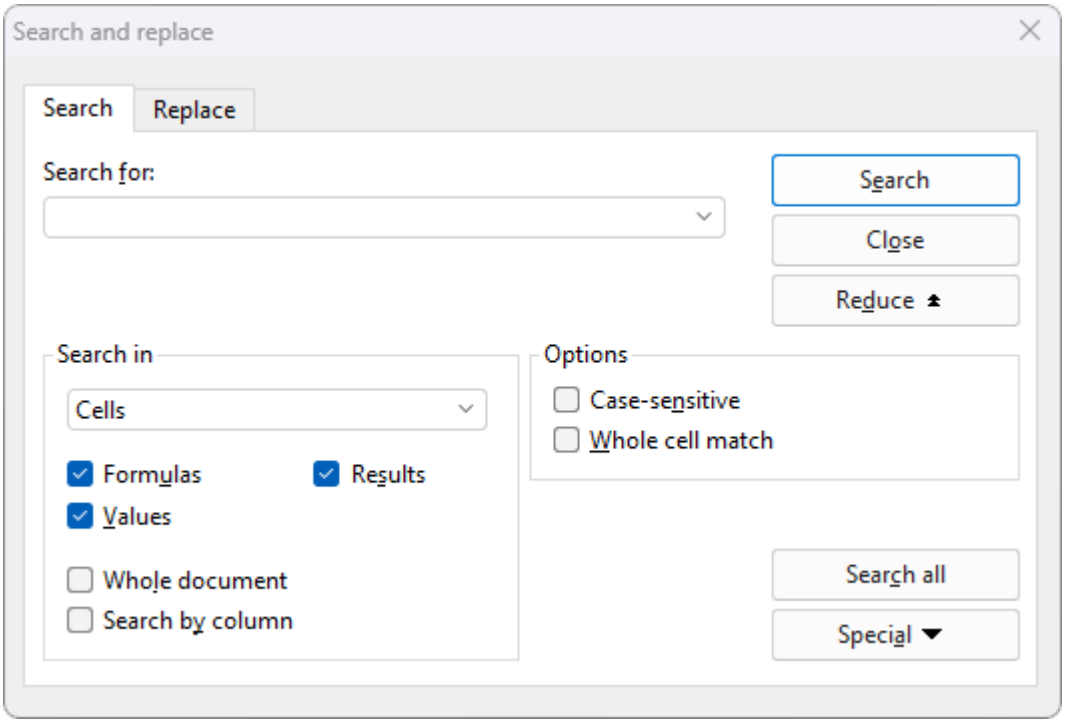
- **Jumping to specific elements of a table**

  Use the three buttons in the lower right corner of the document window to jump to specific parts of a table (for example, to the next cell with an error value or comment).

See the following pages for more information.

Search

You can search a document for a specific term using the ribbon command **Home | Search** .



Proceed as follows:

1. If you want to limit the search to a specific cell range, select the cell range first. If nothing is selected, the search is performed on the entire worksheet.
2. Choose the ribbon command **Home | Search** .
3. Under **Search for:**, enter the search term.
4. Enable the desired search options (see Advanced search options) if required.
5. Click on the **Search** button to start the search.

PlanMaker scans the document for the search term and highlights the first cell that contains it.

You now have the following options:

A. You can click on **Search again** to proceed to the next occurrence of the search text.

B. You can close the dialog box by choosing **Close**.

Replace

Sometimes, you not only want to find a search term, but you also want to replace it with another term. The ribbon command **Home | Replace** (in the **Search** group) is used for this purpose.

Proceed as follows:

1. If you want to limit the search to a specific cell range, select the cell range first. If nothing is selected, the search is performed on the entire worksheet.
2. Choose the ribbon command **Home | Replace** .
3. Under **Search for:**, enter the search term.
4. Under **Replace with:**, enter the term that is to replace the search term.
5. Enable the desired search options (see Advanced search options) if required.
6. Click on the **Search** button to start the search.

PlanMaker scans the document for the search term and highlights the first cell that contains it.

You now have the following options:

A. You can click on **Replace**, whereupon the search term is replaced with the replacement term and PlanMaker jumps to the next occurrence of the search term.

B. You can click on **Search again**, whereupon PlanMaker jumps to the next occurrence of the search term – without replacing the term.

C. You can click on **Replace all**, whereupon PlanMaker replaces *all* occurrences of the search term.

D. You can close the dialog box by choosing **Close**.

Repeating a search/replace

To repeat a search or replace operation or to continue it after canceling, choose the ribbon command **Home | Search again** or simply press the **F3** key.

PlanMaker continues the search and displays the next occurrence of the search term.

Advanced search options

In the dialog box of the ribbon command **Home | Search** or **Home | Replace**, you can click on the **Enlarge** button, which expands the dialog box and offers additional options.

Clicking on **Reduce** makes the dialog box smaller again. This allows you to decide for yourself whether you would prefer to have many settings options or whether you would rather see as much of the worksheet as possible when searching.

The enlarged *Search and replace* dialog box offers the following additional options:

"Search in" section

Here, you can specify how to scan the document:

- **Search in**

  Use this dropdown list to select which parts of the worksheet to scan: **Cells**, **Comments** or **Frames** (text frames and drawings containing text).

- **Formulas**

  If this option is enabled, all formulas in cells will be scanned.

  For example, you can search for "Sum" to find all cells that use the SUM function.

- **Results**

  If this option is enabled, all results of formulas will also be scanned.

  For example, when you search for "4", cells containing the calculation =2+2 will also be found.

  (This option is only available for the search command but not for the replace command.)

- **Values**

  If this option is enabled, all fixed values will be scanned (thus, manually entered numbers, strings, etc.).

- **Whole document**

  If this option is enabled, all worksheets in the document will be scanned.

  If it is disabled, only the current worksheet will be scanned (or all worksheets that are currently selected in the worksheet register).

- **Search by column**

  If this option is enabled, the worksheet will be scanned column by column.

  If it is disabled, the worksheet will be scanned row by row.

"Options" section

Here, you can specify the following options for searching:

- **Case-sensitive**

  The case of letters in the search term is ignored by default. Thus, if you search for "house", not only "house", but also "House", "HOUSE", "HouSe", etc. will be found.

  If you enable this option, however, the program will only find those occurrences of the search term where the cases of the letters correspond exactly to the cases of the letters in the search term. Thus, if you search for "house", only "house" will be found.

- **Whole cell match**

  By default, PlanMaker finds all cells that *contain* the search text in any position. If you enter "win" in the **Search for** field, PlanMaker will also find cells containing "windows" or "sales, winter 2018".

  If you enable this option, however, the *entire* cell content must match the search term. When searching for "win", only cells with the exact content "win" will be found.

"Special" button

The **Special** button allows you to use the following wildcard characters in your search terms:

- **Single arbitrary character** (^?) – this placeholder represents a single arbitrary character. For example, when you search for "fo^?d", PlanMaker finds occurrences of "fold", "fond", "food", etc.
- **Multiple arbitrary characters** (^*) – this placeholder represents any number of arbitrary characters. For example, when you search for "f^*r", PlanMaker finds occurrences of "fair", "flyer", "freighter", etc.
- **Caret** (^^) – searches for the character "^" (caret).
- **Line break** (^a) – searches for manual line breaks (inserted with **Ctrl+↵**).

To insert such a wildcard into the search term, click on the **Special** button and then select the desired wildcard by clicking on it.

You can repeat and combine these wildcards as desired within your search term. Furthermore, you can use them in combination with normal text.

"Search all" button

You can click on the **Search all** button to obtain a list of all occurrences of the search term. When you click on one of its entries, PlanMaker jumps to the corresponding occurrence in the document.

Note: This button is only available when the option **Search in** is set to **Cells**.

Jumping to specific cell addresses

The command **Home | Go to** (in the group **Search**) is used to set the cell frame to a specific cell in the table.

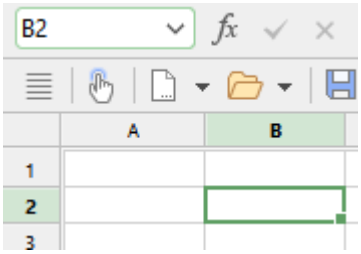
The program displays a dialog box in which you can enter the desired cell address. Both an individual cell address (for example, D2) and a cell range (for example, D2:D4) are permitted.

In the list below the input field, all areas of the table named with the ribbon command **Formula | Named area | Edit names** are still listed so that they can also be accessed immediately.

If you confirm with **OK**, PlanMaker will immediately select the corresponding cell or cell range.

Tip: A faster method

There is also a faster method: In the Edit toolbar (above the table window), there is a dropdown list on the left, which usually shows in which cell the cell frame is currently located.



Here, you can enter the address of a cell or cell range – just like in the dialog box of the command **Home | Go to**. When you press the **Enter key**↵, PlanMaker highlights the corresponding cell(s).

In addition: If you click on the small arrow to the right of the list, it will open and display all named ranges for selection. If you click on a name, the corresponding cell range will be immediately selected.

Jumping to specific elements of a table

Use the three buttons in the lower right corner of the document window to jump to specific parts of a table (for example, to the next cell with an error value or a comment).

You use these buttons as follows:

- If you click on the button with the red dot, a small menu will appear. Then select the type of Go to operation you want to perform from the menu (see below).
- If you click on the button with the up arrow, PlanMaker will jump to the previous position. If, for example, you selected "Error value" in the menu, the previous cell with an error value will be displayed.
- If you click on the button with the down arrow, PlanMaker will jump to the next position, for example, to the next cell with an error value.

You can thus determine the purpose of the search via the menu that opens after clicking on the red dot. This menu has the following items:

Menu item	Explanation
Formula	Jump to cells that contain a formula
Array formula	Jump to cells that contain an array formula
Constant	Jump to cells that contain a fixed value
Error value	Jump to cells that contain an error value (see Error values)
Faulty cell	Jump to cells that have been imported incorrectly – for example, cells using an Excel arithmetic function that PlanMaker doesn't support.
Comment	Jump to cells that contain a comment
External reference	Jump to cells that contain an external reference to another document
Conditional formatting	Jump to cells that use conditional formatting
Input validation	Jump to cells that use input validation
Search all worksheets	If this option is enabled, PlanMaker searches all worksheets instead of just the current one.

Objects

You can insert different types of *objects* into your documents – such as charts, pictures, text frames, drawings, etc.

In the following chapter, you will learn everything you need to know about the general use of objects:

- **Objects – basic procedures**

  Here, you will learn about basic functions such as inserting, selecting and editing objects. You will also learn about more unusual options, such as how to hide objects, group them, etc. The other settings that you will find in the dialog box *Object properties* will also be explained in a separate section.

There will then be a detailed presentation of the individual types of objects:

- Text frames
- Pictures
- OLE objects
- Drawings
- QR codes/Barcodes
- Charts (see chapter Charts)
- Forms (see Form objects in detail)

Objects – basic procedures

In these sections, you will find basic functions for working with objects:

- Inserting objects
- Selecting objects
- Object mode
- Changing position and size of objects
- Rotating and flipping objects
- Changing the order of objects
- Duplicating objects
- Changing the names of objects

Other less frequently used features for working with objects will be covered in the following sections:

- Aligning and distributing objects
- Grouping objects

- Hiding objects
- Changing the AutoShape of objects
- Combining shapes
- Editing shapes retroactively
- Rotation of the inner text
- Vertical alignment of the inner text
- Changing text area for the inner text
- Showing the contents of a cell in objects

The options of the dialog box *Object properties* will then be explained:

- Changing object properties via the dialog box

Inserting objects

To insert a picture, for example, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects** | **Picture frame** .

   (To do so, click directly on the icon itself or use the icon's arrow and select the entry **From file**.)
2. A dialog box appears. Select the desired picture file in it and confirm it.
3. The picture frame is now inserted.

   You can then move or resize the frame:

   To move it, click on the frame with the mouse, and then drag it to the desired position (while still holding down the mouse button).

   To adjust its size, drag one of the round handles surrounding the frame.

The process for inserting other types of objects is similar.

Note: As soon as you have inserted an object, an additional contextual ribbon tab **Object** (or picture, chart, etc.) will appear on the far right and will enable you to conveniently edit the object properties via the corresponding buttons.

For information on the different types of objects and how to adjust their properties, see Text frames, Pictures, Charts, etc.

Selecting objects

If you want to edit an object, you must first *select* it. For most types of objects, it is sufficient to simply click on the desired object. A frame will then appear around the object to indicate that it is now selected.

Note: This does not work with text frames and other objects that have transparent fills. To select such an object, click on its border or switch to *object mode* (see next section), where *all* types of objects can be easily selected by clicking on them.

Once an object has been selected, you can only edit it and can no longer enter anything in the table cells. If you want to edit table cells again, simply click on the desired cell with the mouse. This will deselect the object.

Object mode

Object mode provides the easiest way to work with objects.

Normally, you work with PlanMaker in *edit mode*. In this mode, you can enter data or calculations in table cells, edit them, format them, etc.

If you switch to *object mode*, you will no longer be able to edit table cells. However, this mode offers many functions that make working with objects easier. In object mode, for example, you can select *all* types of objects with a click of the mouse.

Note: As long as you are in object mode, you only have functions for creating and editing objects. You can no longer edit cells, and most of the commands are unavailable.

Switching between edit mode and object mode

There are several ways to switch between edit mode and object mode:

- Choose the ribbon command **View | Object mode** to access object mode. Choose the command again to return to edit mode.
- Click on the icon in the **Quick access toolbar** to switch between edit and object mode.
- You can also *right-click* anywhere in the document. This will open a context menu in which you can choose the command **Object mode** or **Exit object mode**.
- The quickest method is to hover over any *free* position within the document and double-click on it with the *right* mouse button to switch between edit and object mode.
- You can also return to edit mode from object mode at any time by double-clicking anywhere in the text where *no* object is located.

Selecting objects in object mode

In object mode, you can select any type of object simply by clicking on it with the mouse.

This is especially useful for objects with transparent fills (for example, text frames) because such objects can only be selected in edit mode by a targeted mouse click on the *margin* of the object. In object mode, however, you can simply click on such objects anywhere to select them.

To select *several* objects at once, use the mouse to draw a frame around the desired objects. Alternatively, you can click successively on several objects while holding down the **Shift key**⇧ to select them together.

Tip: Use the ribbon command **Home | Select all** to select all objects.

Jumping from object to object

In object mode, you can also "jump back and forth" between objects. Use the following buttons:

Key	Function
Tab	Select next object
Shift+Tab	Select previous object

Changing position and size of objects

To change the position or size of an object, proceed as follows:

Changing the position of objects

To change the position of an object, click on it to select it. You can now either drag it with the mouse to the desired position or move it with the arrow keys on the keyboard.

You will also find such a handle on all objects for moving the object with the mouse. This makes it easier for you to handle small objects.

Note: By default, objects are anchored to the cell beneath their upper left corner. You can move objects as desired, but when an object is dropped at its new position, it will be automatically anchored again to the cell beneath its upper left corner.

This means that an object will move down if additional rows are inserted above it and move up if rows above are deleted. The same happens if you change the height of rows above the object, for example.

If necessary, you can change this behavior by right-clicking on the object's frame to open the context menu and selecting the last entry **Properties**, whereupon a corresponding dialog box appears. Switch to the **Format** tab and change the option for **Position**. If, for example, you choose **Independent of cell position and size**, the object will no longer move down when you insert or delete cells above it. For more information, see Object properties, Format tab.

Changing the size of objects

The **size** of an object can be changed as follows:

- **Dragging the handles on the selection frame**

  Click on the object to select it. The object will then be surrounded by a blue selection frame which also contains some round "handles". When you drag one of these handles with the mouse, the size of the object is changed accordingly.

- **Using the dialog box**

  You can also resize objects via the dialog box. Right-click on the object to open the context menu and select the last entry **Properties**. On the **Format** tab, these settings can be specified exactly in numeric values. For more information on this dialog box, see Object properties, Format tab.

- **Via the ribbon tab**

  If you have selected an object, you will also find input fields for the **Width** and **Height** in the group **Resize** on the far right of the object-specific ribbon tab. Type the desired values here or increase/reduce the values with the small arrow keys next to the input field.

  If the option **Keep aspect ratio** is enabled, changes in height automatically have a proportional effect on width (and vice versa).

  For OLE objects, an additional option named **Keep scaling** is available. If it is enabled, any changes made to the object's size *within its source application* will automatically affect the object in PlanMaker. If it is disabled, the object will always remain the same size in PlanMaker. (Note: Not all OLE source applications support this feature.)

Preventing changes to the position and size of objects

By *locking* an object, you can prevent it from being moved or resized with the mouse or the arrow keys on the keyboard.

Select the object, right-click on it to access the context menu and then select the last entry **Properties** to open the dialog box. On the **Properties** tab, enable the option **Locked**.

Now the object can no longer be moved with the mouse, and it cannot be resized by dragging its boundary lines. The object also cannot be moved with the arrow keys of the keyboard. The position and size can only be changed by entering values in the dialog box.

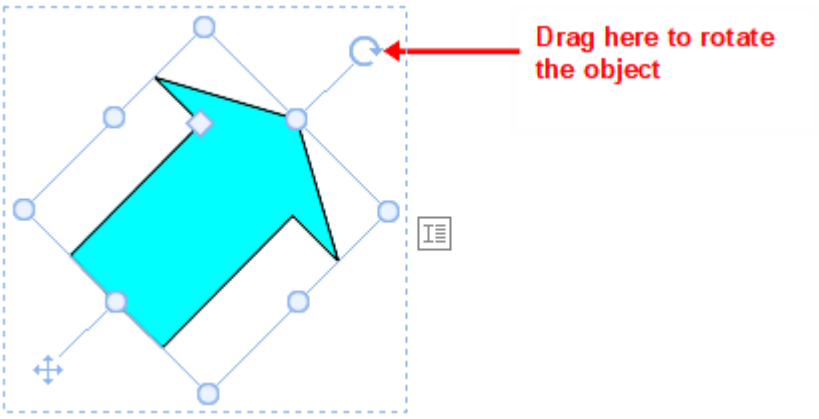
If you disable the option **Locked** again, the position and size of the object can also be changed with the mouse.

Rotating and flipping objects

Note: Only *drawings* and *pictures* can be rotated and flipped; this is not possible for other types of objects.

When you select an object that can be rotated, an additional *green* handle will appear on one of its borders. If you drag this handle with the mouse, the object will rotate.

Tip: If you hold down the **Shift key**⇧, the rotation occurs at exactly 22.5° angles.



Furthermore, there are the following ways in which you can also flip (mirror) drawings and pictures in addition to rotating them:

- You can use the **Rotate object** button on the object-specific ribbon tab.
  Note: An object-specific ribbon tab appears on the far right as an additional tab when you select an object.
- You can do so via the context menu (right-click on the object) by selecting the entry **Rotate or flip**.
- You can also set the desired rotation angle manually in the dialog box. Right-click on the object to open the context menu and select the last entry **Properties**. In the dialog box, switch to the **Format** tab and enter the angle in the **Rotation** section. In the **Flip** section, you can flip the object vertically or horizontally.

Changing the order of objects

If several objects overlap, you can determine the display order of these objects (which object is in the foreground, which is in the background, etc.).

To do so, proceed as follows:

1. Right-click on the desired object.
2. Choose the command **Order** from the context menu. The program opens a submenu with the following commands:

   Bring to front: This option brings the object in front of all other objects.

   Send to back: This option sends the object behind all other objects.

   Bring forward one level: This option brings the object forward by one level.

   Send backward one level: This option sends the object back by one level.

Tip: You can also execute these commands on the object-specific ribbon tab on the far right via the **Bring to front** and **Send to back** buttons (in the group **Position**).

Duplicating objects

To obtain a duplicate of an object, you can copy it to the clipboard in the usual way and then paste it back from there.

Alternatively, you can also use the **Duplicate** button on the object-specific ribbon tab. This command immediately creates a copy of the selected object.

Note: An object-specific ribbon tab appears on the far right as an additional tab when you select an object.

Tip: Duplicating an object is even faster with the mouse: If you drag an object while holding down the **Ctrl** key, you immediately obtain a duplicate of that object.

Changing the names of objects

Every object in a document must have a unique name.

Normally, you do not have to worry about these names as they are automatically assigned by PlanMaker: If you insert a rectangle into an empty document, it is automatically called "Rectangle1". If you insert a picture, it is called "Picture1", the next picture is called "Picture2", etc.

Tip: The name of the currently selected object is displayed on the left in the status bar.

However, you can change the names of objects at any time. If, for example, you have created a form that contains a list for entering the place of residence, you would naturally name this list "residence". To do so, right-click on the list to access the context menu and select the last entry **Properties** to open the dialog box. Switch to the **Properties** tab and change the entry in the **Name** field.

Remember that names must be unique. If you attempt to assign a name to an object that is already in use for another object, PlanMaker will reject it and display an error message.

Note: Newly inserted "Tables in worksheets" are also given a unique name, which can be changed via the **Table properties**. For more information, see Formatting tables in worksheets.

Aligning and distributing objects

To align or evenly distribute two or more objects, select them and click on the **Align objects** button on the object-specific ribbon tab.

Note: An object-specific ribbon tab appears on the far right as an additional tab when you select an object.

In the displayed dropdown menu of **Align objects**, select the desired alignment or distribution:

- **Align left:** This option aligns objects to the left edge of the object placed furthest left.
- **Align center:** This option aligns the objects horizontally centered to each other.
- **Align right:** This option aligns the objects to the right edge of the object placed furthest right.
- **Align top:** This option aligns the objects to the top edge of the top object.
- **Align middle:** This option aligns the objects vertically centered to each other.
- **Align bottom:** This option aligns the objects to the bottom edge of the bottom object.
- **Distribute horizontally:** This option distributes objects evenly between the left edge of the object furthest left and the right edge of the object furthest right.
- **Distribute vertically:** This option distributes objects evenly between the top edge of the top object and the bottom edge of the bottom object.

The last two commands only work if at least *three* objects are selected.

Tip: You will also find the commands for aligning and distributing objects in the context menu by right-clicking on the selected objects and selecting the entry **Align or distribute**.

Grouping objects

When you *group* several objects, you combine them into a unit that can be selected and manipulated like a single object.

In practice, this means the following:

If you click on an object that belongs to a group, the entire group is selected. You can then move the group, and all objects of the group will also be moved.

Once the group is selected, clicking subsequently on a single object of the group will enable you to edit that object separately. It can then be moved, rotated, enlarged/reduced, provided with text or even deleted within the grouping.

To group objects, proceed as follows:

1. Select the objects to be grouped. To do so, click on the objects successively while holding down the **Shift key**⇧.

   Alternatively: Switch to **Object mode** and drag a rectangle around all objects to be selected while holding down the mouse button.

2. Click on the **Group** icon (in the **Objects** group) on the object-specific ribbon tab. Alternatively: Right-click on the selected objects and choose **Group** in the context menu.

Ungrouping objects

To ungroup objects, proceed as follows:

1. Select the group by clicking (once) on one of the objects in it.
2. Click on the **Group** icon again or choose the command **Ungroup** via the context menu.

Hiding objects

If necessary, you can *hide* an object so that it will not appear on the screen and/or on a printout.

Select the desired object and right-click to open the context menu. Select the last entry **Properties** to open the dialog box, switch to the **Properties** tab and proceed as follows:

- If you disable the option **Visible**, the object will no longer be displayed on the screen.
- If you disable the option **Printable**, the object will no longer appear in the printout.

Tip: Show hidden objects anyway

If you have inadvertently made an object invisible, you can always choose the ribbon command **File | Properties** and enable the option **Show hidden objects** on the Options tab. Now all hidden objects are displayed again. Thus, you can now select the object and enable the option **Visible** to make it visible again.

Changing the AutoShape of objects

Objects can be replaced afterwards with another shape using the ribbon command **Change AutoShape**, while retaining the existing formatting.

Tip: For more options for changing the shape of objects, see Combining shapes and Editing shapes retroactively.

Replacing the shape of drawings and text frames

To replace the shape of drawings (AutoShapes, lines, curves, TextArt objects) or text frames with another shape, proceed as follows:

1. Select the desired object.
2. Click on the **Change AutoShape** button on the contextual ribbon tab **Object | group Edit**.

   Alternatively via the dialog box: Right-click on the object to open the context menu and select the last entry **Properties**. In the dialog box, switch to the "AutoShapes" tab.
3. Select the desired new shape in the list.

The contours of the object will then change to the selected shape.

Replacing the shape of pictures

To replace the shape of pictures with another shape, proceed as follows:

1. Select the desired object.
2. Click on the small arrow of the **Cropping** icon on the contextual ribbon tab **Picture** | group **Size** and select the entry **Change shape** in the dropdown menu that appears.

   Alternatively via the dialog box: Right-click on the object to open the context menu and select the entry **Change shape** there.
3. Select the desired new shape from the list.

The contours of the picture frame will then change to the selected shape.

Tip: The command **Photo frame** on the contextual ribbon tab **Picture** | group **Format** provides another method of changing the shape of pictures. Here, you can choose from different variants. Some of these variants display handles that enable you to further adjust certain parameters of the shape by dragging them. You can also use the small square in the center to display the shadow with a varying offset.

Combining shapes

Note: This feature is only available for *drawings* and *text frames*; it is not possible for other types of objects.

In PlanMaker, you can combine two or more shapes into one overall shape using the command **Combine objects**, and different variants of the merge can be selected. This allows you to quickly create individual drawing objects.

Important: The new overall shape always assumes the formatting of the *first selected object*.

Proceed as follows:

1. Select two or more objects (drawings/text frames) that you want to combine.

   To do so, click on the objects successively while holding down the **Shift key**⇧.

   Alternatively: Switch to **Object mode** and drag a rectangle around all objects to be selected while holding down the mouse button.
2. Choose the command **Combine objects** (in the group **Edit**) on the contextual ribbon tab **Object**.

   Alternatively, you will find the command in the context menu after right-clicking on the selected objects.
3. In the dropdown menu, select the desired variant for combining the objects (see below).

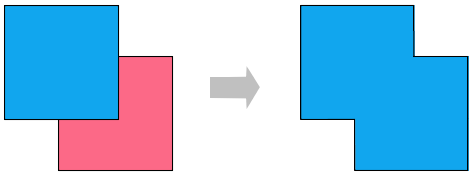
Variants of the command "Combine objects"

As mentioned at the start, the *first selected object* is the reference object whose formatting will be applied to the new overall shape when each variant of the command **Combine objects** is chosen. This is the fill color, for example, in the following illustrations.

The following variants are available:

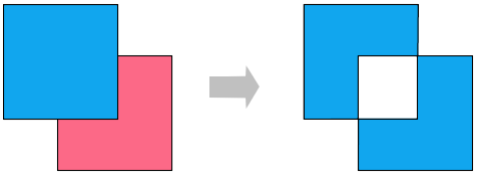
- **Union**

  This option creates a new common shape with a consistent outer contour of all selected shapes:



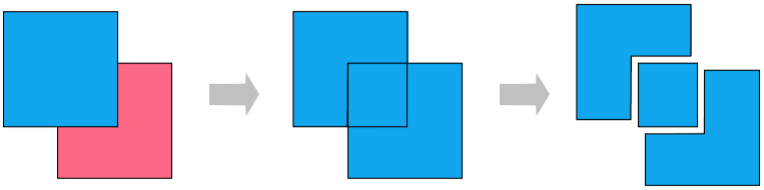
- **Combine**

  The non-overlapping parts of the shapes are preserved, while the overlapping areas become transparent:



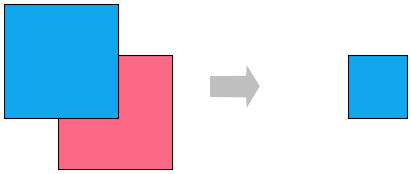
- **Fragment**

  New separate shapes are formed along the intersecting lines of overlapping shapes. These can then be moved individually – as shown on the far right in the following figure:



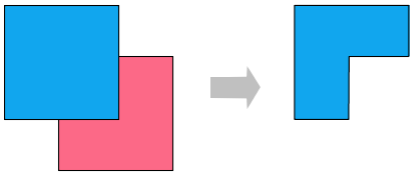
- **Intersect**

  This is the counterpart to **Combine**. Only the overlapping area is preserved, and the non-overlapping areas disappear:



- **Subtract**

  The overlapping areas are removed from the first selected shape, and the following selected shapes themselves disappear:



You can move the new shape like a "normal" object, resize it, add text to it, etc. There are also black corners or vertices that allow you to customize the contours of the shape. For more information, see the next section Editing shapes retroactively).

Editing shapes retroactively

There are several methods of retroactively adjusting the shape of objects:

- Drawings/text frames can be given a different AutoShape via the ribbon command **Change AutoShape** (in the group **Edit**). For more information, see Changing the AutoShape of objects.
- You can also use the command **Combine objects** (in the group **Edit**) to combine two or more shapes into one overall shape. For more information, see Combining shapes.
- Additional handles are displayed for some types of AutoShapes. If you drag one of these handles, a parameter controlling the shape of the object is changed. For stars, for example, the length of the points can be changed, and for rounded rectangles, the rounding of the corners can be changed.

  For pictures whose shape has been changed via the ribbon command **Photo frame** (in the group **Format**), these handles also appear for some variants.

In this section, the command **Edit points** is used to describe a very individual method of customizing certain objects (drawings, text frames or pictures) to your liking – by editing points of the object via the "Bezier tools". Technically, you use this command to convert an AutoShape into a curve.

To do so, proceed as follows:

1. Select the object and choose the command **Edit points** on the contextual ribbon tab **Object** | group **Edit**.

   Tip: For pictures, you will find the command only via the context menu. (Right-click on the object.)

   After switching to "Bezier Mode" via the command **Edit points**, you will see small black corners or vertices on the object's contour line (Note: Bezier mode remains constantly for the object once enabled).
2. Drag the black points via the mouse to change the shape.
3. Optional: For shapes with curves, a blue line (tangent) with a small square (handle) also appears when you left-click on one of the black points. Use the mouse to take hold of the blue square in order to change the angle of inclination and length of the curve. See also the next section "Other options via the context menu".

Note: If you ensure that you are positioned exactly on the points (or the line segments, see below) with the mouse, the mouse cursor will then change to crosshairs .

Other options via the context menu

You will find additional editing options in the context menu by right-clicking on one of the small black points or on a line segment of the object.

For *points*, you can choose between the following options in the context menu:

- **Auto point**

  The angle and length of the tangent are set to a default value for automatic smoothing. No tangent for adjusting the angle and length appears accordingly for this option.

- **Straight point**

  A blue tangent appears. Both tangent halves are firmly connected, by angle and length. If one handle is moved, the other handle changes analogously in terms of angle and length.

- **Smooth point**

  A blue tangent appears. Only the angle of the two tangent halves is connected, but not the length. If one handle is moved, the other handle changes analogously in terms of angle. The length (and thus the gradient of the curve) can be set independently.

- **Corner point**

  A blue tangent appears. Both tangent halves can be moved independently, both by angle and length.

- **Delete point**

  This option removes the selected point from the contour line.

For *line segments*, the context menu provides the following options:

- **Straight segment**

  The selected line segment between two points is straightened.

- **Curved segment**

  The selected line segment between two points becomes a curved line.

- **Add point**

  This option adds another editing point to the selected line segment.

- **Open curve/Close curve**

  You use **Open curve** to remove the line segment between two points. The option **Close curve** closes an open shape and inserts a new line segment between the open ends.

The entry **Curve properties** opens a dialog box where you can edit the properties of the created shape (fill, line color, etc.) as with any other object. For more information, see Changing object properties via the dialog box. Many of these options can also be found directly on the contextual ribbon tab **Object** (or **Picture**).

Rotation of the inner text

The ribbon command **Object** | group **Inner Text** | **Text rotation** is only available for objects with inner text. You can thus use this command with Text frames and also with AutoShapes to which text has been added. For more information, see the relevant sections.

Vertical alignment of the inner text

The ribbon command **Object** | group **Inner Text** | **Vertical alignment** is only available for objects with inner text. You can thus use this command with Text frames and also with AutoShapes to which text has been added. For more information, see the relevant sections.

Changing text area for the inner text

For Text frames and also for AutoShapes to which text has been added, you can resize the text area for the inner text in PlanMaker.

To do so, proceed as follows:

1. Click on the frame of the object to select it.
2. Choose the command **Change text area** on the contextual ribbon tab **Object** | group **Inner text** (or via the context menu by right-clicking on the text frame).

   Note: For drawings (AutoShapes, curves, etc.), you must first add text to the shape so that you can use the commands for the inner text.
3. The frame of the inner text is now selected and some round navy "handles" appear. Drag one of these handles via the mouse to resize the text area.

When your adjustments are complete, choose the command **Change text area** again or simply click anywhere in the document.

Showing the contents of a cell in objects

A link to a cell can be made in PlanMaker for Text frames, AutoShapes and also certain chart elements (chart title, axis title). This enables you to transfer the contents of a cell – for example, the result of a calculation – to an object for illustration purposes.

Note: This only works if there is no text in the object. Thus, delete existing text from the object beforehand.

The cell contents are displayed dynamically. So if the linked cell is overwritten with new data, the corresponding content will also change in the object.

Proceed as follows:

1. Click on the frame of an object to select it.

   (For chart elements, proceed differently. For more information, see chapter Chart title, Axis title)

2. Choose the command **Add/edit link** on the contextual ribbon tab **Object |** group **Inner text**.

   Alternatively: Right-click on the frame for the context menu, select **Link to a cell**.

3. A dialog box opens. Now click directly in the document on the cell whose content is to be displayed in the selected object. Alternatively, you can type the cell address into the input field of the dialog box (for example, "=Table1!A1").

   Tip: It is also possible to link to cells on another worksheet in the document.

4. Click on **Add**.

The cell content of the selected cell is now displayed in the object. If the object is selected, you will see the cell address that is linked to the object in the Edit toolbar.

If you want to change the font or alignment of the displayed text, select the object and use the commands on the ribbon tab **Home |** group **Character/Alignment** or on the ribbon tab **Object |** group **Inner Text**.

Changing object properties via the dialog box

The *properties* of an object include, for example, its size, fill color, borders, etc. These properties can be edited centrally using the dialog box **Object properties**.

Note: Many object properties can also be conveniently edited directly via the icons of the contextual ribbon tab **Object** (or **Picture**, **Chart**, etc.). Information about these ribbon tabs can be found in the respective section of the corresponding object. However, some other options can only be set via the dialog box.

To change the properties of an object via the dialog box, right-click on the object frame to open the context menu and select the last entry **Properties**. A corresponding dialog box appears.

Tip: For most types of objects, double-clicking on the object provides a quick alternative way to open this dialog box.

The dialog box has several tabs that you can switch between by clicking on one of the tab headers at the top.

You will find detailed information on each tab and the options it contains on the following pages.

Object properties, Format tab

You can use the **Format** tab to make settings with respect to the format of the object, such as resizing it.

Available options:

- **Size** and **Scaling** sections

  You can resize the object here. To do so, enter the desired **Width** and **Height**. Alternatively, you can change an object's size by specifying the width as a percentage of the original width in **Horz. scaling** and the height as a percentage of the original height in **Vert. scaling**.

  If the option **Keep aspect ratio** is enabled, changes in height automatically have a proportional effect on width (and vice versa).

  There is an additional option named **Keep scaling** for OLE objects. If it is enabled, any changes made to the object's size *within its source application* will automatically affect the object in PlanMaker. If it is disabled, the object will always remain the same size in PlanMaker. (Note: Not all OLE source applications support this feature.)

- **Position** section

  Here, you can choose to what extent the object's position and size are dependent on the table cells it covers.

  Depends on cell position and size: The object behaves as if it were anchored to the cell covered by the top left corner of the object. Thus, if you insert new rows above this cell, the object will be pushed down accordingly.

Furthermore, the object's size depends on the height and width of the cells it covers. If, for example, you increase the width of these cells, the object will grow accordingly.

Depends only on cell position: The object also behaves here as if it were anchored to the cell covered by the top left corner of the object. The object moves when you insert rows above it, but the size of the object does not change when you resize the cells it covers.

Independent of cell position and size: The object always retains its original position and size, regardless of the cells it covers.

- **Rotation** section

  This option is only available for drawings and pictures.

  Here, you can rotate the object. To do so, enter the angle at which you want to rotate the object clockwise.

- **Flip** section

  This option is only available for drawings and pictures.

  Here, you can flip the object vertically or horizontally.

Object properties, Fill tab

Note: This tab is only available for certain types of objects. For pictures, changes to the fill affect only the transparent parts of the picture.

You can use the **Fill** tab to specify with what the object should be filled.

First, select the desired type of fill in the dropdown list **Fill type**. You can then make various settings for each fill type.

The following fill types and settings are available:

Tip: You can also apply some of these options directly on the object-specific ribbon tab via the command **Fill color** in the group **Format**.

- **No fill**

  If you select the first fill type in the list, the object is not filled and thus remains transparent.

- **Solid color**

  This fill type fills the object with a uniform color shade. Select the desired color in the list **Colors**.

  Tip: Color lists only offer a few standard colors, but you can always compose your own colors. To do so, click on **More colors** and then proceed as described in Document properties, Colors tab.

  If desired, you can also use the option **Transparency** to change the transparency of the fill. You can enter any value between 0% (no transparency) and 100% (full transparency). If, for example, you set the transparency value to 25, the fill will have 25% transparency.

- **Pattern**

  This fill type fills the object with a pattern. In the list **Patterns**, select the type of pattern and then set the desired color for the foreground and background.

  If desired, you can also use the option **Transparency** to change the transparency of the fill. You can enter any value between 0% (no transparency) and 100% (full transparency).

- **Picture**

  This fill type fills the object with a picture file. You can use the **Open** button to determine the picture file which you want to use for the fill. The most recently used picture files are displayed in the list **Pictures** and can be selected there by clicking on them with the mouse.

  Additional options:

  Rotate with object: If you enable this option, the picture will also be rotated whenever you rotate the object to be filled.

  Mirror type: This option allows you to mirror the picture horizontally or vertically.

  Transparency: If desired, you can use this option to change the transparency of the fill. You can enter any value between 0% (no transparency) and 100% (full transparency).

  Tile as texture: If this option is enabled, the picture is repeated in the object like a tile pattern.

  You can then use the settings in the **Tile options** section to change the size and position of the tiles: **X scale** and **Y scale** change the size (in percent), while **X offset** and **Y offset** change the position. The option **Alignment** can be used to determine to which edge of the object the tiles should be aligned.

  If the option **Tile as texture** is disabled, only **Offsets** (relative to the edges of the object) can be set for the picture.

  Save: You can use this button to export the currently selected picture, that is, to save a copy of it under any name on your hard disk.

- **Linear gradient, Rectangular gradient, etc.**

  The bottom five fill types are different types of color gradients. First, select the desired type of gradient in the dropdown list **Fill type** and then select one of the subtypes in the list **Variants**.

  In the **Options** section, you can specify the appearance of the gradient even more precisely:

  X offset and **Y offset** enable you to move the center of the gradient. The **Angle** option allows you to set the angle of rotation.

  > **Tip:** You can also change these settings by using the mouse to move or rotate the cross-hair displayed in the **Sample** field.

  If you enable the option **Double gradient**, for example, a black-white gradient will become a black-white-black gradient.

  In the **Colors** section, you can change the colors for the gradient as follows:

  To change a color, first click on one of the small triangles below the bar representing the gradient. The left triangle stands for the initial color, while the right triangle stands for the final color:

Then select the desired color from the dropdown list **Color** below.

You can use the option **Transparency** to change the transparency of the selected color if desired. You can enter any value between 0% (no transparency) and 100% (full transparency).

Additional colors can be added to the gradient by double-clicking on the desired position in the bar and selecting a color. In the same way, namely by double-clicking on the triangle, additional colors can also be deleted from the gradient.

Object properties, Lines tab

Note: This tab is only available for certain types of objects.

Use the **Lines** tab to specify the type of lines to use for the object. For drawings, this applies to the lines used to draw the object; for all other objects, it applies to the lines that border the object.

Available options:

Tip: Many of these options can also be found directly on the object-specific ribbon tab in the command group **Format**.

- **Variants**

  Here, you can select the desired line style.

  The entries in this list are merely default variants. If required, you can specify the appearance of the lines more precisely via the options of the dialog box.

- **Color**

  This option lets you specify the color of the lines.

- **Dashed**

  Here, you can specify whether solid or dashed lines should be used.

- **Thickness**

  Here, you can specify the exact thickness of the lines (in points).

- **Transparency**

  If desired, you can use the option **Transparency** to change the transparency of the lines. You can enter any value between 0% (no transparency) and 100% (full transparency).

- **Start point** and **End point** (only available for lines, curves and connectors)

  Here, you can specify whether an icon should be drawn at the start point or end point of the line. If, for example, you select an arrow icon for the end point, the line will look like an arrow. If necessary, you can also change the **Width** and the **Height** of the icon.

Object properties, Shadow tab

Note: This tab is only available for certain types of objects.

Use the **Shadow** tab to add a shadow to the object.

Available options:

Tip: You can also apply many of these options directly on the object-specific ribbon tab via the command **AutoShape effects** in the group **Format**.

- **Variants**

  Here, you can select the desired type of shadow.

  The entries in this list are only default variants; the other options in the dialog box allow you to specify the appearance of the shadow even more precisely if necessary.

- **Scaling**

  Here, you can change the size of the shadow (relative to the size of the object).

- **Offset**

  Here, you can change the position of the shadow (relative to the object).

- **Perspective**

  Here, you can change the **Skew angle** of the shadow.

  For certain variants of the "Perspective Shadows" type, the **Horizon** can also be changed, which distorts the shadow in perspective. If **Horizon** is a positive value, the shadow is in front of the object; if it is negative, the shadow is behind the object.

- **Color**

  Here, you can change the color of the shadow.

- **Blur**

  If you enter a value greater than zero here, a blur effect is added to the shadow. The larger the value, the softer the edges of the shadow are drawn.

- **Transparency**

  Here, you can change the transparency of the shadow. You can enter any value between 0% (no transparency) and 100% (full transparency). If, for example, you set the transparency value to 25, the shadow will have 25% transparency.

Object properties, 3D tab

Note: This tab is only available for text frames and drawings.

Use the **3D** tab to add a three-dimensional effect to the object.

Select the desired type of 3D effect in the **Variants** section. The entries in this list are only default variants; the other options on the **3D** tab allow you to specify the appearance of the effect even more precisely if required.

In the preview field **Sample**, you can follow the result of your adjustments and finally confirm with **OK** if you want to apply the settings.

Object properties, Effects tab

Note: This tab is only available for certain types of objects.

On the **Effects** tab, you can add various effects to the object.

The following effects and options are available:

- **Reflection**

  If you enable the option **Use reflection effect**, the object is displayed as if it was located on a reflective surface. The following variations are possible:

  Visible part of object: This option specifies how much of the object should be visible in the reflection (in percent).

  Starting transparency: The reflection is softly faded out downwards. This value determines how transparent the reflection should be at the top (in percent).

  Y offset: This option moves the reflection upwards or downwards.

- **Soft edges**

  If you enable the option **Use soft edges effect**, the borders of the object will be given a blur effect. The following variations are possible:

  Width: This option specifies the width of the blur effect.

- **Glow**

  If you enable the option **Use glow effect**, the borders of the object will be surrounded by a glow effect. The following variations are possible:

  Width: This option specifies the width of the glow effect.

  Color: This option specifies the color of the glow effect.

Object properties, Properties tab

On the **Properties** tab, you can change the general settings for an object.

Available options:

- **Name**

  PlanMaker automatically assigns a unique name to every object in a document. For example, rectangles are called rectangle1, ...2, ...3, etc. If desired, you can enter another name here.

  Tip: The name of the currently selected object is displayed on the left in the status bar.

  For more information on using names for objects, see Changing the names of objects.

- **Visible**

  This option is enabled by default. It causes the object to be displayed on the screen. (For more information, see also Hiding objects.)

  If you disable this option, the object becomes invisible.

- **Printable**

  This option is enabled by default. It causes the object to appear in the printout when you print the worksheet. (For more information, see also Hiding objects).

- **Locked**

  This option is enabled by default. It means that the size, position and all other properties of the object can no longer be changed if you enable sheet protection. For more information, see also Sheet protection.

- **Text locked**

  This option is only available for text frames and drawings that contain text.

  This option is enabled by default. It prevents the text contained in this object from being edited when you enable sheet protection. For more information, see also Sheet protection.

- **Link**

  Here, you can enter a hyperlink – to a website, for example. To do so, click on the **Select** button, select the type of link and specify the desired target. The target is opened when the user clicks on the object.

  In the **Jump target** field, you can also specify the target position in the document to be opened.

In the case of pictures, the **File name** of the picture is displayed on this tab, while some information about the picture that was used (resolution, color depth, etc.) is displayed at the top right.

Object properties, AutoShapes tab

Note: This tab is only available for pictures, text frames and AutoShape drawings.

On the **AutoShapes** tab, you can assign an AutoShape to the object. Thus you can turn a rectangle into a speech balloon or any other shape you like, for example.

To change the shape, simply click on the desired AutoShape in the list.

Tip: You will also find this option as the command **Change AutoShape** directly on the contextual ribbon tab **Object**. For more information, see Changing the AutoShape of objects.

For more information on AutoShapes, see Inserting drawings.

Additional tabs

There are additional tabs with more options for some object types. For information on these tabs, see the section on the relevant object.

Changing the default settings for objects

You can change the default settings for objects (drawings, pictures, etc.) at any time. This is particularly relevant for drawings (AutoShapes and lines). For certain other objects, there are restrictions on applying the default settings.

If, for example, you don't like the default line thickness for drawings, simply set it to a different value. You can also alter other default settings – for example, the default fill for drawings as well as the settings for shadows and 3D effects.

Note: Changes to the default settings for objects only affect objects that you insert *afterwards*. Existing objects remain untouched.

To change the default settings for objects, use the **Set as default** button in the dialog box **Object properties**.

For example, the default line thickness for drawings can be changed as follows:

1. Insert a new drawing (or click on an existing drawing).
2. Right-click on the object to access the context menu and select the last entry **Properties** to open the dialog box.
3. Make the desired settings. To change the default setting for the line thickness, for example, switch to the **Lines** tab and simply set the desired line thickness.
4. Important: Click on the **Set as default** button and not on **OK**.
5. The program displays another dialog box in which you can also limit which default settings should be changed if necessary. As a rule, there is no need to change anything in this dialog box.
6. If you confirm with **OK**, the settings currently made in the dialog box will be set as the default settings for new drawings.

7. Confirm again with **OK** to exit the dialog box completely.

All drawings that you insert from now on will use the line thickness set in this way by default.

Note: The default settings for objects can be set separately for each document.

This concludes the general description of objects. The following sections present the various types of objects in detail:

- Text frames
- Pictures
- OLE objects
- Drawings
- Charts (see chapter Charts)
- Forms (see Form objects in detail)

Text frames

You can use *text frames* to insert text boxes into a document. You can enter, edit and format text as desired in a text frame.

Note: Text frames, by the way, are just AutoShapes (in this case, simple rectangles) to which text has been added.

In this section, you will learn everything you need to know about working with text frames. The information covers the following topics:

- Inserting text frames
- Changing text frame properties via the ribbon tab
- Changing text frame properties via the dialog box

See the following pages for more information.

Inserting text frames

To insert a text frame, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects** | **Text frame** .
2. The text frame is now inserted.

   You can then move or resize the text frame:

   Important: Click with the mouse on the *border* of the text frame in order to move it. While still holding the mouse button down, drag the text frame to the desired position.

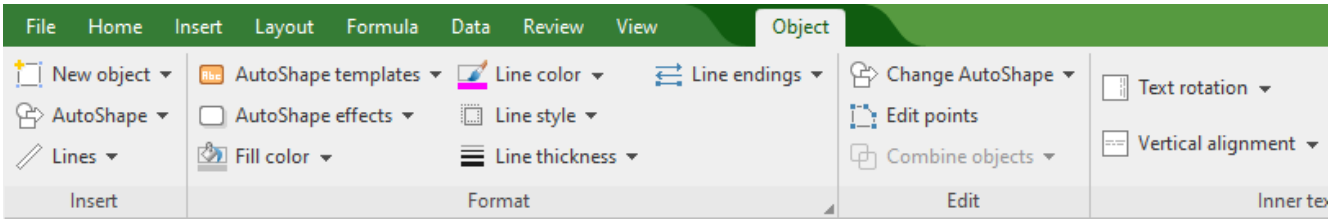
   To adjust its size, drag one of the round handles surrounding the frame.

The text frame is now created, and you can start entering text in it immediately.

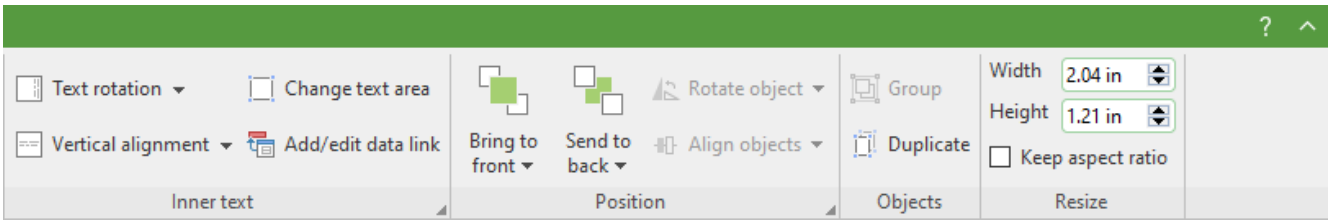
When you have finished entering text, click on any cell to leave the frame. You can now edit worksheet cells again. Conversely, you move from the body text back to the text frame by clicking into the frame.

Changing text frame properties via the ribbon tab

Numerous properties of text frames can be changed directly via the buttons of the contextual ribbon tab **Object**.



The contextual ribbon tab "Object", left section



The contextual ribbon tab "Object", right section

This ribbon tab appears automatically when you select a text frame by clicking on its border. It contains the following buttons (from left to right):

- **New object:** This option allows you to insert a new object into the document and different object types are available.
- **AutoShape:** This option allows you to insert a new AutoShape into the document.
- **Lines:** This option allows you to insert simple lines, curves and also connectors (which you can "dock" to the text frame) into the document.
- **AutoShape templates:** This option allows you to apply various predefined color formats to the text field.
- **AutoShape effects:** Here, you can apply effects such as shadows, reflections, etc. This option largely corresponds to the options of the tabs *Shadow*, *3D* and *Effects* of the dialog box "Object properties". For more information, see the next section.
- **Fill color:** The fill can be changed here. For more information, see Object properties, Fill tab.
- **Line color, Line style, Line thickness, Line endings:** These options allow you to edit borders for the text field or the inserted lines. For more information, see Object properties, Lines tab.
- **Change AutoShape:** This option allows you to assign a different AutoShape to the text frame (formatting is retained). For more information, see Changing the AutoShape of objects.
- **Edit points:** This option allows you to give the text frame an individual shape with the "Bezier tools". For more information, see Editing shapes retroactively.
- **Combine objects:** This option allows you to combine two or more shapes from different variants into one overall shape. For more information, see Combining shapes.
- **Text rotation:** This option allows you to rotate the inner text by the specified angle.
- **Vertical alignment:** This option determines how the inner text should be aligned between the top and bottom edges of the frame.
- **Change text area:** This option allows you to adjust the size of the inner text area. For more information, see Changing the text area.

- **Add/edit link:** This option allows you to show the content of the selected cell in a text frame. For more information, see Showing the content of a cell in objects.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Rotate object:** This option is not available for text frames.
- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option creates an immediate copy of the selected object.
- **Width, Height**: These options change the size of the object. For more information, see Changing position and size of objects.

Many (but not all) of these functions and some additional options can be found in the dialog box *Object properties*, which you can access, for example, via the context menu. For more information, see the next section.

Changing text frame properties via the dialog box

To edit the properties of a text frame via the dialog box, first select it. To do so, click on the border surrounding the text frame or simply click on it in object mode (**View | Object mode**). Then right-click to open the context menu and select **Properties**, whereupon a corresponding dialog box appears.

Also possible: Click on the group arrow ◢ in the bottom right corner of any command group of the contextual ribbon tab **Object**.

Tip: If you are using object mode, you can also access this dialog box instantly by double-clicking on the text frame.

The following settings can be made in the dialog box:

Format, Fill, Lines tabs, etc.

These tabs are available for almost all object types. They allow you to change the following settings:

- **Format:** On this tab, you can change the size and positioning of the text frame. For more information, see Object properties, Format tab.
- **Fill:** This option allows you to change the fill. For more information, see Object properties, Fill tab.
- **Lines:** This option allows you to apply border lines. For more information, see Object properties, Lines tab.
- **Shadow:** This option allows you to apply a shadow. For more information, see Object properties, Shadow tab.

- **3D:** This option allows you to apply a 3D effect. For more information, see Object properties, 3D tab.
- **Effects:** This option allows you to apply different effects. For more information, see Object properties, Effects tab.
- **Properties:** This option allows you to change general settings. For more information, see Object properties, Properties tab.

In the case of text frames, there are two additional tabs with further options:

Inner text tab

You can use this tab to change settings for the text in the text frame.

Note: An AutoShape must contain text in order to be treated as a text frame. Thus, select the AutoShape and type something in it, and then the **Inner text** tab will also be available.

Available options:

- **Inner margins** section

  Here, you can change the margins between the text frame and the text it contains. If you enable the option **Automatic**, PlanMaker will automatically determine suitable margins.

- **Overlapping objects** section

  This setting determines what should happen if the text frame overlaps with another frame.

  If the default setting **Ignore object** is enabled, PlanMaker will do nothing. In the area of overlap, the contents of both frames will thus be printed on top of each other.

  If, on the other hand, you enable the option **Wrap text**, PlanMaker will automatically wrap the text in this frame so that it flows around the area of overlap.

- **Rotate by...** section

  Here, you can rotate the text by the specified angle.

- **Vertical alignment** section

  This setting determines how the inner text should be aligned between the top and bottom edges of the frame:

Option	Explanation
Top	The text is aligned to the top edge of the frame. (This is the default setting.)
Centered	The text is centered between the top and bottom edges of the frame.
Bottom	The text is aligned to the bottom edge of the frame.
Justified	The lines of text are vertically justified and are thus evenly distributed so that the text starts exactly at the top edge of the frame and ends exactly at the bottom edge.

- **Data link** section

For text frames (and for AutoShapes), you can create a link to a cell. This will display the contents of this cell in the object. For more information, see also Showing the contents of a cell in objects.

AutoShapes tab

A text frame is actually an *AutoShape* that has been filled with text. AutoShapes are predefined shapes (such as rectangles, ellipses, arrows, speech bubbles, etc.).

By default, a simple rectangle is used as an AutoShape for text frames. However, you can assign a different AutoShape to the text frame at any time. Simply select the desired shape from the **AutoShapes** tab.

For more information on AutoShapes, see Drawings.

Pictures

In this section, you will learn everything you need to know about working with pictures. The information covers the following topics:

- Inserting pictures
- Scanning pictures
- Inserting pictures from the gallery/camera (Android and iOS)
- Compressing pictures
- Changing picture properties via the ribbon tab
- Changing picture properties via the dialog box

See the following pages for more information.

Inserting pictures

To insert a picture, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects** | **Picture frame** .
   (To do so, click directly on the icon itself or use the icon's arrow and select the entry **From file**.)
2. A dialog box appears. Select the desired picture file in it and confirm it.
3. The picture is now inserted.

   If you want, you can then move or resize the picture:
   To move it, click on the picture with the mouse, and then drag it to the desired position (while still holding down the mouse button).
   To adjust its size, drag one of the round handles surrounding the picture.

Options of the dialog box

The options in the above dialog box mean the following:

- **Save within document**

  If the option is enabled, PlanMaker saves a copy of the picture in the document and uses this copy instead of the original.
  Note: If this option is enabled, the file size of the document may increase significantly.

  If it is disabled, however, PlanMaker saves only a reference to the original picture file in the document. Changes to the original picture file are thus always available in an updated form in PlanMaker.

Tip: You can see the location of the original picture file in the picture properties. Right-click on the picture to open the context menu, and select **Picture: Properties**. In the dialog box, switch to the **Properties** tab and take a look under **File name**.

Note: If the file path of the picture file changes, however, it will no longer be displayed in the document.

- **Copy to document's folder**

  If the option is enabled, PlanMaker creates a copy of the picture file in the folder in which the document is located and saves a reference to this copy instead of to the original.

  This option is available only if the document has already been saved.

Scanning pictures

Note: *Scanning pictures* directly into a document is possible only with the **Windows** version of PlanMaker.

With PlanMaker for Windows, you can scan originals directly into a document. You must have connected a scanner to your computer and installed the corresponding software.

To scan something into the current document, proceed as follows:

1. Turn on the scanner and insert the original that is to be scanned.
2. On the ribbon tab **Insert** | group **Objects** | **Picture frame**, click on the arrow of the icon and select the entry **From scanner** from the dropdown menu.

PlanMaker now enables your scanner's software. Make the desired settings (see scanner manual) and start the scanning process. When this is complete, the scanned result will appear as a picture in the PlanMaker document.

Selecting another source

If several scanners are connected to your PC, you can use the command **Insert** | **Picture frame** | **Select source** to decide from which of these sources you want to scan when you use the command **Insert** | **Picture frame** | **From scanner**.

Inserting pictures from the gallery/camera (Android and iOS)

Note: This function is available only in the **Android** and **iOS** version.

In the Android/iOS version, you can also insert pictures via the *Gallery* or *Camera* application on your device.

To do so, proceed as follows:

Inserting a picture from the gallery

1. Position the text cursor where you want to insert the picture.
2. Choose the command **Insert** | group **Objects** | **Picture frame** | **From gallery**.
3. Your Android/iOS device automatically opens its *Gallery* app. Select the desired picture by tapping on it.

The picture is now inserted.

Inserting a picture from the camera

1. Position the text cursor where you want to insert the picture.
2. Choose the command **Insert** | group **Objects** | **Picture frame** | **From camera**.
3. Your Android device/iOS automatically opens its *Camera* app. Take a picture.

The picture is now inserted.

Compressing pictures

This feature is only included in **SoftMaker Office Professional** and **NX Universal**.
In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

You can use the command **Compress pictures** to reduce the memory size of images in PlanMaker. In this way, you reduce the file size of the document and save capacity on your device or when sharing the document.

You can use the command effectively if your document contains pictures that were embedded in the document when you inserted them using the option **Save within document** (see Inserting pictures).

Note: Pictures that were instead saved in the document as a reference to an external picture file are not affected by compression. However, when executing the command, you have the option of subsequently embedding these pictures in the document and compressing them here at the same time.

Proceed as follows:

1. In your document, select one or more pictures that you want to compress. An additional ribbon tab **Picture** appears at the top right of the ribbon.

   Tip: If you want to compress *all* pictures of the document, just choose the ribbon command **File** | group **File management** | **Compress all pictures** and continue with step 2.a.

2. Choose the ribbon command **Picture** | group **Adjust** | **Compress pictures**.

   2.a If your selection contains only pictures that have been saved in the document, the dialog box "Compress pictures" opens immediately.

   2.b If your selection contains pictures that have *not* been saved in the document, a message appears before the dialog box opens, asking whether you want to embed these pictures now (see description below).

3. In the **Resolution** section of the dialog box, choose between the predefined qualities or enter a custom resolution.

   In the **Options** section, you will find two more settings (see description below).

4. Confirm with **OK**.

After saving the document, you will notice a change in the file size.

Other options

- **Delete cropped areas of the pictures**

  If you have cropped an image in your document using the command **Pictures | Cropping** and you do not need the truncated remainder, enable this option. Once you press **OK**, the cropped image area will be deleted and only the crop will be retained.

- **Embed and compress external pictures**

  This option allows you to embed and compress pictures in the document that you did not insert using the option **Save in document**, but which are only saved in the document as a reference to the original external file.

  To explain this option, the following cases may occur:

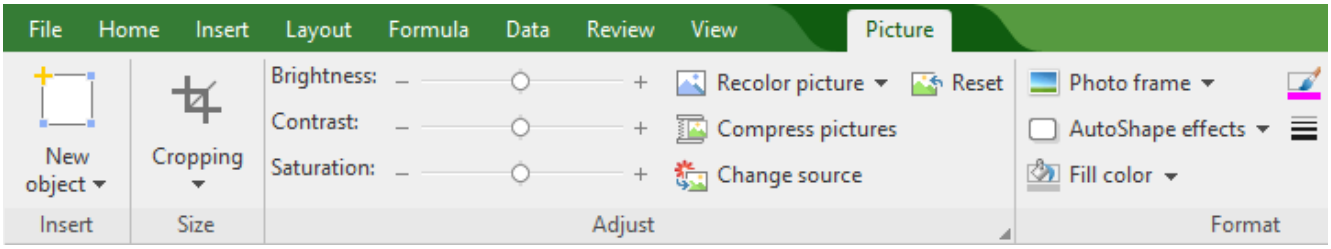
  a. You have selected only pictures that were not saved in the document. After choosing the command **Compress pictures**, a message appears asking if you want to subsequently embed the pictures. If you click on **Yes**, this option is already preselected in the following dialog box.

  b. If pictures are selected that have already been saved in the document, this message also appears. If you click on **Yes** (or also **No**), this option is freely selectable in the following dialog box.

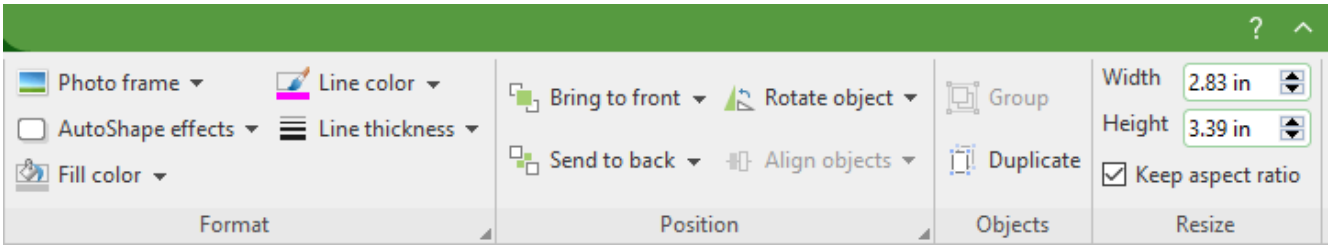
  c. If you have selected only pictures that have already been saved in the document, this message does not appear. This option is not selectable in the dialog box.

Changing picture properties via the ribbon tab

Numerous properties of pictures can be changed directly via the buttons of the contextual ribbon tab **Picture**.



The contextual ribbon tab "Picture", left section



The contextual ribbon tab "Picture", right section

This ribbon tab appears automatically when you select a picture. It contains the following buttons (from left to right):

- **New object:** This option allows you to insert a new object into the document and different object types are available.
- **Cropping:** You can enable/disable *crop mode* via this button. In this mode, additional handles appear on the borders of the picture. Drag these handles with the mouse to crop the picture accordingly. To exit the mode, click on the button again. You can also use the button's small arrow to select a cropping from different shapes or to remove the cropping.

  See also the next section ("Picture tab" in the dialog box).
- **Brightness, Contrast, Saturation:** Drag the slider or click on + and − to change the respective values for the picture. For more information, see also the next section ("Picture tab").
- **Recolor picture:** This option provides a list of predefined variations of the picture – for example, other color modes (grayscale, black/white, etc.) and variations where the image was colored. For more information, see also the next section ("Picture tab").
- **Compress pictures:** Reduce the memory size of images in your document. For more information, see also Compressing pictures.
- **Change source:** Replace the picture with another one (formatting is retained).
  This corresponds to the **File** button in the dialog box. For more information, see the next section ("Picture tab").
- **Reset:** Reset the *image-specific* changes made to the picture, such as brightness, shading, rotate object, etc., to their default values. The *non-image-specific* settings are not reset, for example, line thickness and photo frame.
- **Photo frame:** This option allows you to apply a photo frame effect to a picture.
- **AutoShape effects:** Here, you can apply effects such as shadows, reflections, etc. This option largely corresponds to the options of the tabs *Shadow*, *3D* and *Effects* of the dialog box "Object properties". For more information, see the next section.
- **Fill color:** The fill can be changed here. For more information, see Object properties, Fill tab.
- **Line color, Line thickness:** These options allow you to apply border lines. For more information, see Object properties, Lines tab.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Rotate object:** This option allows you to rotate a picture in 90° increments or flip it vertically/horizontally. For more information, see Rotating and flipping objects.

- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option allows you to create an immediate copy of the selected picture.
- **Width, Height**: These options change the size of the picture. For more information, see Changing position and size of objects.

Many (but not all) of these functions and some additional options can be found in the dialog box *Object properties*. You open the dialog box by double-clicking on the picture. For more information, see the next section.

Changing picture properties via the dialog box

The properties of an object can be edited centrally via the **Object properties** dialog box. An object's properties include, for example, its size, margins and numerous other settings options.

To edit the properties of a picture, right-click on the picture to open the context menu and select the entry **Picture: Properties**, which opens the corresponding dialog box.

Also possible: Click on the group arrow ◢ in the bottom right corner of any command group of the ribbon tab **Picture**.

Tip: You can also open this dialog box by double-clicking on the picture.

The following settings can be made in this dialog box:

Format, Fill, Lines tabs, etc.

These tabs are available for almost all object types. They allow you to change the following settings:

- **Format:** On this tab, you can change the size and positioning of the picture and rotate it. For more information, see Object properties, Format tab.
- **Fill:** This option allows you to change the fill. It is only visible for pictures with transparent areas. For more information, see Object properties, Fill tab.
- **Lines:** This option allows you to apply border lines. For more information, see Object properties, Lines tab.
- **Shadow:** This option allows you to apply a shadow. For more information, see Object properties, Shadow tab.
- **3D:** This option allows you to apply 3D effects. For more information, see Object properties, 3D tab.
- **Effects:** This option allows you to apply different effects. For more information, see Object properties, Effects tab.

- **Properties:** This option allows you to change general settings. It also displays information about the picture you are using. For more information, see Object properties, Properties tab.
- **AutoShapes:** Here, you can assign a different shape from AutoShapes to the picture. For more information, see Object properties, AutoShapes tab.

For pictures, there is an additional tab with further options:

Picture tab

On the **Picture** tab, you can make settings that are specific to pictures:

Tip: Some of these options can also be found directly as commands on the contextual ribbon tab **Picture** | group **Adjust**.

- **Variants**

  This list offers some predefined variations of the picture – for example, different color modes (grayscale, black & white, etc.) as well as colored variants of the image.

  Simply click on the desired variant. The options in the dialog box are then automatically set to the corresponding settings.

  Tip: You will also find these variants via the command **Recolor picture** directly on the contextual ribbon tab **Picture**.

- **Options** section

  Here, you can change brightness, contrast, saturation and gamma (an alternative way to change brightness).

- **More** button

  This button opens a dialog box with the following additional options:

  If you enable the option **Mix with color** and select a color below, the picture will be colored accordingly.

  The options in the **Cropping** section can be used to crop the picture. This is useful if you only need part of the picture. If, for example, the upper quarter of the picture is to be truncated, first select **Use relative values** and enter the number "25" (percent) for **Top**.

  If you select **Use absolute values**, the entries will be truncated in absolute units of length rather than as a percentage. If you enter negative values, the areas of the picture will not be truncated, but will be extended with a frame.

  Tip: You will also find a corresponding command **Cropping** directly on the contextual ribbon tab **Picture**. For more information, see the previous section Changing picture properties via the ribbon tab.

- **Transparency** section

  Here, you can make settings for the transparency of the picture:

  Use picture settings: PlanMaker reads from the picture file which parts of the picture have been marked as transparent and displays the picture accordingly. Note: Only pictures in GIF or PNG format contain such transparency information.

  No transparency: The picture will not be displayed transparently, even if it is a picture file with transparency information.

  Color: This option lets you select the color to be displayed transparently. If, for example, you select white, all white areas of the picture will become transparent.

- **Sample**

  In the small toolbar below the sample, you will find the option **Pick transparent color** (the pipette icon). You can use it to make certain colored parts of the picture transparent, for example, to remove a distracting background from the picture. To do so, use the pipette icon to click on an area of the thumbnail that you want to make transparent.

  The other toolbar buttons help you to fine-tune the picture (**Zoom in**, **Zoom out**, **Pan picture**), but they do not affect the appearance of the picture in the document.

  Note: You must first select a tool in the toolbar by clicking on it with the mouse. You can then work on the thumbnail with the enabled tool.

- **File** button

  You can use this button to replace the picture with another one and the formatting will be retained.

  Tip: This corresponds to the command **Change source** on the contextual ribbon tab **Picture**.

- **Export** button

  This option is available only for pictures that are saved in the document. You can use this button to export the picture, that is, to save a copy of it on your hard disk under any name. The dialog box "Save as" appears for this purpose.

  If you enable the option **Create a link to file** here, PlanMaker also replaces the picture saved in the document with a reference to the destination file. Thus, the picture is no longer saved within the document.

- **Compress picture** button

  This button allows you to reduce the memory size of pictures in your document.

  Tip: This corresponds to the command **Compress pictures** on the contextual ribbon tab **Picture**. For more information on this command, see Compressing pictures.

OLE objects

Note: The use of OLE objects is only possible with the **Windows** version of PlanMaker.

With PlanMaker for Windows, you can embed objects created with other applications (drawings, pictures, etc.) in your document.

Example: You can use the ribbon command **Insert | OLE object frame** (click directly on the icon itself) to start the Windows *Paint* program from PlanMaker and draw something in it. If you exit Paint, the drawing will appear in the PlanMaker document. It is now embedded in the document.

Advantage: You only have to double-click the drawing in the PlanMaker document and Paint will then be restarted automatically. If you now edit the drawing in it and exit Paint again, your changes will be imported by PlanMaker.

For this to work, the application that you choose must support *OLE*. OLE stands for Object Linking and Embedding. The object that is imported from the other application into PlanMaker is called an *OLE object*.

In this section, you will learn everything you need to know about OLE objects. The information covers the following topics:

- Inserting OLE objects
- Editing OLE objects
- Editing links to OLE objects
- Changing OLE object properties via the ribbon tab
- Changing OLE object properties via the dialog box
- Using the SoftMaker Equation Editor

See the following pages for more information.

Inserting OLE objects

Note: The use of OLE objects is only possible with the **Windows** version of PlanMaker.

By choosing the ribbon command **Insert | OLE object frame** (click directly on the icon itself), you can embed an object that you have created with another application into a PlanMaker document. After embedding such an object, you can still edit it with the application that you originally used to create it.

The command opens a dialog box with a list of all applications that can provide OLE objects. The entries which are displayed here depend on the OLE-enabled programs that are installed on your system.

If you select "Paintbrush Picture", for example, a window from Window's *Paint* program (called *Paintbrush* in some versions of Windows) will open. Create a picture in it and then exit Paint. The picture is now embedded in your document. It can be edited again at any time – simply double-click on the picture in the document.

To insert an OLE object into a document, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects** | **OLE object frame** . To do so, click directly on the icon itself.
2. A dialog box appears. From the list **Object Type**, select the source application from which you want PlanMaker to receive the OLE object.
3. When you confirm with **OK**, PlanMaker will start the application that corresponds to the selected object type.
4. Create the object in the application.
5. Exit the application.
6. The application may now ask if you want to update the object in the PlanMaker document. Confirm this with "Yes".
7. The OLE object created in the other application is now created in the PlanMaker document.

   You can then move or resize the object:

   To move it, click on the object and then drag it to the desired position (while still holding the mouse button down).

   To adjust its size, drag one of the round handles surrounding the object.

Tip: If you click on the icon's arrow under **Insert** | **OLE object frame** instead of on the icon itself, you will find a small selection of useful applications for quick access: *TextMaker object* and *Presentations object*. For other applications, select **More** to access the dialog box described above.

Creating an OLE object from a file

If you don't want to create a new OLE object, but want to embed an existing file from another application as an OLE object, you still choose the ribbon command **Insert** | **OLE object frame**, but you select the option **Create from File** in this case.

The appearance of the dialog box changes: Instead of the list **Object Type**, the **File** input field is displayed. Here, enter the full file name or click on the **Browse** button to open a dialog box that will enable you to search your hard disk for the file.

If you specify a file and confirm with **OK**, the file will be inserted as an OLE object.

In doing so, Windows checks the file name extension to see if it is associated with a program that can be started and it responds accordingly. If the file name extension is not registered, only an icon will appear for the embedded file. If, on the other hand, Windows can associate the name extension with an OLE-enabled program, the content of the embedded file will be displayed.

Creating a link to the original file

For the option **Create from File** in the aforementioned dialog box, you can also select whether the file to be inserted should be *embedded* or *linked*. If you leave the option **Link** disabled, PlanMaker will insert a copy of the file into the document as an OLE object and save this copy in the document – thus the object will be *embedded* in the usual way.

If, on the other hand, you enable the option **Link**, PlanMaker will insert the OLE object in the form of a link to the original file. This does not change anything in terms of use. If you subsequently change the original file of the OLE object with another program and reopen the PlanMaker document, however, the OLE object will have also changed accordingly – the OLE object will thus be *linked* to the original file.

Editing OLE objects

Existing OLE objects can be edited as follows:

Select the OLE object and click on the **OLE object** icon on the contextual ribbon tab **OLE object**. Select **Edit** from the dropdown menu.

Tip: Alternatively, simply double-click on the OLE object to edit it. The object will be opened immediately in the associated application, where it can be edited.

Another alternative: You can also use the context menu to edit an OLE object.

Proceed as follows:

1. Click on the OLE object to select it.
2. Right-click on the OLE object to open the context menu and select the menu item with the object name (for example, **Bitmap object**). Selecting this menu item opens a submenu containing all OLE commands that are available for this object. There is a command **Edit** for editing the object. Choose this command.
3. The source application associated with the OLE object will now start. Make the desired changes.

To return to PlanMaker, simply exit the object's source application. When exiting some applications, you will be asked if you want to update the object in the PlanMaker document. Answer "Yes" – your changes will be discarded otherwise.

Editing links to OLE objects

If you have created an OLE object from a file and enabled the option **Link**, you can edit this link to the source file. To do so, right-click on the OLE object to open the context menu and choose the command **Link**.

Alternatively, simply click on the **Edit link** icon on the contextual ribbon tab **OLE object**.

The program will display a dialog box which lists all links in the current document. When you select a link, the file name and type of the associated source file are displayed in the lower part of the dialog box.

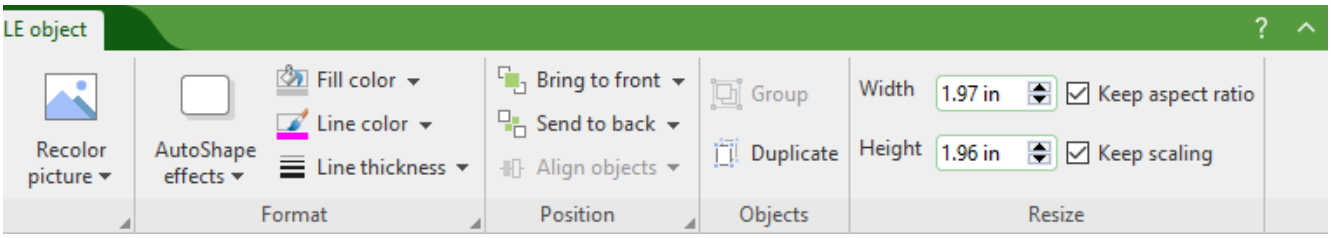
The buttons of the dialog box allow you to perform the following actions:

Button	Function
Update now	This button updates the OLE object. Use this button if the source file has been changed by another program and the changes are not visible in the PlanMaker document.
Open source	This button starts the application associated with the OLE object and lets you edit the object in it. *Tip:* Double-clicking on the object has the same effect.
Change source	This button lets you select a different file as the source for the OLE object.
Break link/ Remove	This severs the link to the source file. The object is then embedded permanently in the PlanMaker document. Changes to the source file no longer have any effect on the document. However, the object can still be edited by double-clicking on it.
Update: **Automatic** or **Manual**	If you have selected the option **Automatic**, changes that you make to the source file using the **Open Source** button are always updated automatically for the OLE object. If **Manual** is selected, changes to the source file are not updated for the OLE object until you click on the **Update Now** button

Changing OLE object properties via the ribbon tab

Numerous properties of OLE objects can be changed directly via the buttons of the contextual ribbon tab **OLE object**.

The contextual ribbon tab "OLE object", left section



The contextual ribbon tab "OLE object", right section

This ribbon tab appears automatically when you select an OLE object. It contains the following buttons (from left to right):

- **New object:** This option allows you to insert a new object into the document and different object types are available.
- **OLE object:** You can *Edit*, *Open* or permanently *Convert* the selected object here.
- **Edit link:** If you have created an OLE object from a file and enabled the option **Link**, you can edit this link to the source file. For more information, see Editing links to OLE objects.
- **Cropping:** You can enable/disable *crop mode* via this button. In this mode, additional handles appear on the borders of the object. Drag these handles with the mouse to crop the object accordingly. To exit the mode, click on the button again. You can also use the button's small arrow to select a cropping from different shapes or to remove the cropping.

  See also the next section ("Picture tab" in the dialog box).
- **Brightness, Contrast, Saturation:** Drag the slider or click on + and − to change the respective values for the object. For more information, see also the next section ("Picture tab").
- **Recolor picture:** This option provides a list of predefined variations of the object – for example, other color modes (grayscale, black/white, etc.) and variations where the object was colored. For more information, see also the next section ("Picture tab").
- **AutoShape effects:** Here, you can apply effects such as shadows, reflections, etc. This option largely corresponds to the options of the tabs *Shadow*, *3D* and *Effects* of the dialog box "Object properties". For more information, see next section.
- **Fill color:** The fill can be changed here. For more information, see Object properties, Fill tab. However, this function is not supported by all OLE applications.
- **Line color, Line thickness:** These options allow you to apply border lines. For more information, see Object properties, Lines tab.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option creates an immediate copy of the selected object.
- **Width, Height**: These options change the size of the object. For more information, see Changing position and size of objects.

Many (but not all) of these functions and some additional options can be found in the dialog box *Object properties*, which you can access, for example, via the context menu. For more information, see the next section.

Changing OLE object properties via the dialog box

The properties of an object can be edited centrally using the **Object properties** dialog box. An object's properties include, for example, its size, margins and numerous other settings options.

To edit the properties of an OLE object, first click on it to select it. Then open the dialog box by clicking on the group arrow ◢ in the bottom right corner of any command group on the contextual ribbon tab **OLE object**.

Alternatively: Right-click on the OLE object to open the context menu and select the entry **OLE object properties**.

The following settings can be made in this dialog box:

Format, Fill, Lines tabs, etc.

These tabs are available for almost all object types. They allow you to change the following settings:

- **Format:** On this tab, you can change the size and positioning of the OLE object. For more information, see Object properties, Format tab.
- **Fill:** This option allows you to change the fill. However, this function is not supported by all OLE applications. For more information, see Object properties, Fill tab.
- **Lines:** This option allows you to apply border lines. For more information, see Object properties, Lines tab.
- **Shadow:** This option allows you to apply a shadow. For more information, see Object properties, Shadow tab.
- **Effects:** This option allows you to apply different effects. For more information, see Object properties, Effects tab.
- **Properties:** This option allows you to change general settings. For more information, see Object properties, Properties tab.
- **AutoShapes:** Here, you can assign a different shape from AutoShapes to the object. For more information, see Object properties, AutoShapes tab.

In the case of OLE objects, there is an additional tab with further options:

Picture tab

On the **Picture** tab, you can make settings that affect the graphical representation of the OLE object.

Note: Depending on the type of OLE object, some of these options may not have a function.

Available options:

- **Variants**

  This list offers some predefined variations – for example, different color modes (grayscale, black & white, etc.) as well as colored variants of the object.

Simply click on the desired variant. The options in the dialog box are then automatically set to the corresponding settings.

- **Options** section

  Here, you can change brightness, contrast, saturation and gamma (an alternative way to change brightness).

- **More** button

  This button opens a dialog box with additional settings:

  If you enable the option **Mix with color** and select a color below it, the object will be colored accordingly.

  The options in the **Cropping** section can be used to crop the object. This is useful if you only need a section of the object. If, for example, you want the upper quarter of the object to be truncated, first select **Use relative values** and enter "25" (percent) for **Top**.

  If you select **Use absolute values**, the entries will be truncated in absolute units of length rather than as a percentage. If you enter negative values, the areas of the object will not be truncated, but will be extended with a frame instead.

Using the SoftMaker Equation Editor

Note: The *SoftMaker Equation Editor* is available only for **Windows** and is not included in all versions of PlanMaker.

The *SoftMaker Equation Editor* allows you to graphically display mathematical and technical equations.

PlanMaker communicates with the Equation Editor via *OLE*, as described in the previous sections. In order to add a graphical representation of an equation to your document, insert an OLE object that uses the Equation Editor as the source application.

To do so, proceed as follows:

1. In PlanMaker, choose the ribbon command **Insert | OLE object frame**. To do so, click directly on the icon itself.
2. In the list **Object Type** of the dialog box, select **SoftMaker Equation Editor**.
3. Confirm with **OK** to start the Equation Editor.

   Tip: You can insert an equation even faster via the ribbon command **Insert | Equation Editor object** .

4. Create the desired equation.
5. When the equation is complete, simply exit the Equation Editor.

The equation is now embedded in the PlanMaker document as an OLE object.

You can subsequently edit objects created with the Equation Editor at any time by double-clicking on them. The Equation Editor is then restarted and opens the equation. Make the desired changes and then simply exit the Equation Editor again.

You can also change the properties of the Equation Editor object by right-clicking on the Equation Editor object, opening the context menu, and choosing the command **OLE object properties**.

Information from the creators of the Equation Editor

The SoftMaker Equation Editor is a special version of the *MathType* program from Maths for More (WIRIS). If you frequently use equations in your documents, you might find that MathType itself is even better suited to your needs than the SoftMaker Equation Editor.
While just as easy to use as the Equation Editor, MathType has a number of additional functions that can help you to become more productive and create more complex equations.

MathType is available in English, German and other languages. For information about how to purchase MathType, contact your software vendor or WIRIS directly.

Maths for More, S.L.
Calle Roger de Flor, 223
08025 Barcelona
Spain

Phone WIRIS Europe:	+34 932 804 805
Phone WIRIS America:	+1 562 432 2920
E-Mail:	sales@wiris.com
Website:	www.wiris.com

Drawings

PlanMaker provides several tools for creating drawings. You can insert the following types of drawing objects into your documents:

- Lines and curves (lines, arrows, connectors, curves)
- AutoShapes (various types of predefined shapes)
- TextArt objects (lettering with effects)

In this section, you will learn everything you need to know about working with drawings. The information covers the following topics:

- Inserting drawings
- Adding text to AutoShapes
- Changing drawing properties via the ribbon tab
- Changing drawing properties via the dialog box

See the following pages for more information.

Inserting drawings

To insert a drawing, for example, a rectangle, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects** | **AutoShape** . Select the desired AutoShape – a **rectangle** in this case.
2. Hold down the left mouse button and draw a frame of the desired size in the document.
3. The rectangle is now inserted.

   You can then move or resize the rectangle:

   To move the rectangle, click on it and (while still holding the mouse button down) drag it to the desired position.

   To adjust its size, drag one of the round handles surrounding the rectangle.

The application of drawings differs somewhat depending on the drawing tool used. The following is a list of all available drawing objects and information on how to handle them:

Note: You can adjust the shape of drawings retroactively at any time. For more information, see Editing shapes retroactively.

AutoShape

You can insert *AutoShapes* using the **AutoShape** tool. These are predefined shapes for various purposes – standard shapes such as rectangles, but also symbols for flowcharts, stars, speech bubbles and many more.

As soon as you have selected an AutoShape, simply drag a frame of the desired size in the document while holding down the left mouse button. If you want, you can then move the AutoShape with the mouse or resize it by dragging one of the round handles at its corners.

Note: Additional handles are displayed in some types of AutoShapes when they are selected. If you drag one of these handles, a parameter controlling the shape of the object is changed. For stars, for example, the length of the points can be changed, and for rounded rectangles, the rounding of the corners can be changed.

Tip: You can even enter text in an AutoShape – just like you can in a text frame. For more information, see Adding text to AutoShapes.

Lines and curves

Use the **Lines** tool on the ribbon tab **Insert** to draw lines and curves.

When you click on this icon, a list of available line and curve tools opens. Select the desired tool and then use it as follows:

- **Lines**

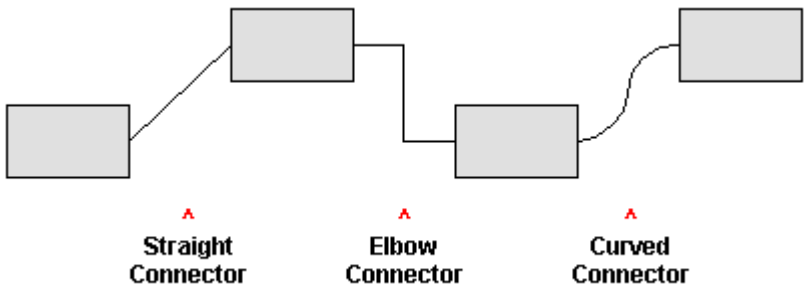
  You can use the tools at the top of the list to draw simple lines and arrows. After clicking on the desired tool, simply draw a line in the document while holding down the mouse button.

  Tip: If you hold down the **Shift key**⇧, the line is limited to 45° angles.

  Note: Arrows are nothing more than ordinary lines for which an arrowhead was selected in the object properties (**Lines** tab) by selecting the option **Start point** and/or **End point**.

- **Connectors (straight, elbow and curved)**

  You can use these tools to drag connectors between two objects. The difference between the three types of connectors can be seen in the following figure:



  To do so, select one of these tools, and then drag the mouse cursor from one object to another while holding down the mouse button. When you move the mouse cursor over an object, small blue rectangles will appear at various points on the object. These show where the line can "dock" to the object. You can influence the course of curved and angled connectors by dragging the diamond handle via the mouse.

Tip: You can change the type of connectors retroactively at any time. Right-click on the line to open the context menu and choose between a *straight*, *elbow* or *curved* connector. Also available in the context menu: The option *Create new connector* causes the shortest connection between two objects.

- **Curves**

  The tools at the bottom of the list are used to draw curves:

  You can use the **Curve** tool to draw Bezier curves.

  To do so, click on the starting point and then on any number of other positions. The curve "follows" your mouse clicks automatically.

  If you let the curve end at its starting point, the drawing automatically becomes a *closed* curve. If, on the other hand, you want to obtain an *open* curve, you must end the drawing by double-clicking on it.

  You can use the **Freehand form** tool to draw curves in the same manner as if you were using a pencil.

  To draw freehand, move the mouse to the desired starting point, press and hold the left mouse button and then start drawing as if using a pencil. Alternatively, you can release the mouse button and click on another location where a straight line will be drawn.

  If you let the curve end at its starting point, the drawing automatically becomes a *closed* curve. If, on the other hand, you want to obtain an *open* curve, you must end the drawing by double-clicking on it.

Note: The small black points on the contour line of curves enable you to further adjust the shape. Use the mouse to drag such a point to another position. Further options can be found in the context menu by right-clicking on the curve. For more information, see Editing shapes retroactively.

TextArt

You can insert *TextArt* objects using the **TextArt object** tool on the ribbon tab **Insert**. TextArt objects are used to create text effects.

If you enable this tool, a dialog box appears. Enter your text in the dialog box and then click on **Variants** to select one of the predefined effects to be applied to the text. Then confirm with **OK**.

For information on the other options for TextArt objects, see Text tab.

Adding text to AutoShapes

You can add text to an AutoShape if desired. This text is then displayed within the AutoShape – exactly as it would be in a text frame.

To do so, proceed as follows:

1. Select the desired AutoShape.
2. Right-click on it to open its context menu.
3. Choose the command **Add text**.

The cursor flashes in the AutoShape. Thus, you can now type text inside the AutoShape.

Tip: There is another faster way to add text to an AutoShape: Select the AutoShape and just start typing.

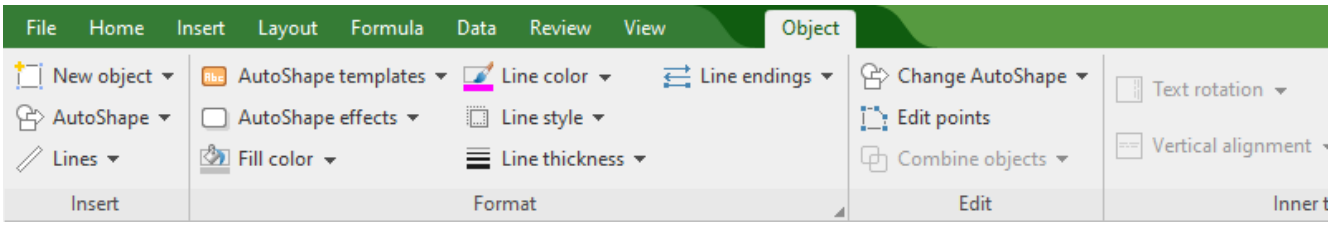
You can use the commands of the ribbon tab **Home** | group **Character** and group **Alignment** to format the text according to your needs..

When you have finished adding text, click anywhere outside the object. If you want to edit the text again later, select the AutoShape again and choose the command **Edit text** from the context menu.

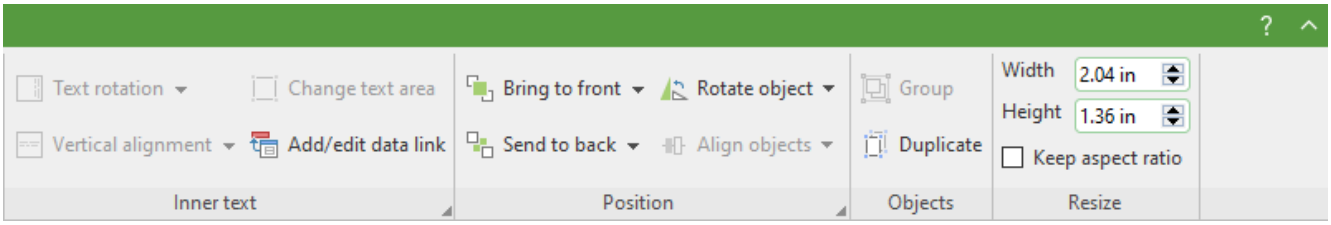
Note: By the way, the *text frames* presented at the beginning of this chapter are nothing more than AutoShapes (simple rectangles in this case) to which text has been added. Thus, all information in Text frames applies equally to AutoShapes to which text has been added.

Changing drawing properties via the ribbon tab

Numerous properties of drawings can be changed directly via the buttons of the contextual ribbon tab **Object**.



The contextual ribbon tab "Object", left section



The contextual ribbon tab "Object", right section

This ribbon tab appears automatically when you select a drawing. It contains the following buttons (from left to right):

- **New object:** This option allows you to insert a new object into the document and different object types are available.
- **AutoShape:** This option allows you to insert a new AutoShape into the document.
- **Lines:** This option allows you to insert simple lines, curves and also connectors (which you can "dock" to the text frame) into the document.
- **AutoShape templates:** This option allows you to apply various predefined color formats to the drawing.

- **AutoShape effects:** Here, you can apply effects such as shadows, reflections, etc. This option largely corresponds to the options of the tabs *Shadow*, *3D* and *Effects* of the dialog box "Object properties". For more information, see next section.
- **Fill color:** The fill can be changed here. For more information, see Object properties, Fill tab.
- **Line color, Line style, Line thickness, Line endings:** These options allow you to edit border lines for the drawing or the inserted lines. For more information, see Object properties, Lines tab.
- **Change AutoShape:** This option allows you to assign a different AutoShape to the drawing (formatting is retained). For more information, see Changing the AutoShape of objects.
- **Edit points:** This option allows you to give the drawing an individual shape with the "Bezier tools". For more information, see Editing shapes retroactively.
- **Combine objects:** This option allows you to combine two or more shapes from different variants into one overall shape. For more information, see Combining shapes.
- Command group **Inner text:** The commands of this group are only possible for shapes to which text has been added. For more information, see Adding text to AutoShapes.

  Text rotation: This option allows you to rotate the inner text by the specified angle.

  Vertical alignment: This option enables you to determine how the inner text should be aligned between the top and bottom edges of the frame.

  Change text area: This option allows you to adjust the size of the inner text area. For more information, see Changing the text area.

  Add/edit link: This option allows you to show the content of the selected cell in an AutoShape. For more information, see Showing the content of a cell in objects.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Rotate object:** This option allows you to rotate a drawing in 90° increments or flip it vertically/horizontally. For more information, see Rotating and flipping objects.
- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option creates an immediate copy of the selected object.
- **Width, Height**: These options change the size of the object. For more information, see Changing position and size of objects.

Many (but not all) of these functions and some additional options can be found in the dialog box *Object properties*, which you can access, for example, via the context menu. For more information, see the next section.

Changing drawing properties via the dialog box

To edit the properties of a drawing via the dialog box, first select it by clicking on it. Then right-click to open the context menu and select the last entry **Properties**, whereupon a corresponding dialog box appears.

Also possible: Click on the group arrow ◢ in the bottom right corner of any command group of the contextual ribbon tab **Object**.

Tip: This dialog box can also be opened by double-clicking on the drawing.

The following settings can be made in the dialog box:

Format, Fill, Lines tabs, etc.

These tabs are available for almost all object types. They allow you to change the following settings:

- **Format:** On this tab, you can change the size and positioning of the drawing. For more information, see Object properties, Format tab.
- **Fill:** This option allows you to change the fill. For more information, see Object properties, Fill tab.
- **Lines:** Here, you can change the line style of the lines used to draw the object. For more information, see Object properties, Lines tab.
- **Shadow:** This option allows you to apply a shadow. For more information, see Object properties, Shadow tab.
- **3D:** This option allows you to apply a 3D effect. For more information, see Object properties, 3D tab.
- **Effects:** This option allows you to apply different effects. For more information, see Object properties, Effects tab.
- **Properties:** This option allows you to change general settings. For more information, see Object properties, Properties tab.

Furthermore, some types of drawings have additional tabs with additional options. (For more information, see the following pages.)

AutoShapes tab (available only for AutoShapes)

This tab appears for drawing objects only for AutoShapes.

On the **AutoShapes** tab, you can assign a different AutoShape to the object. Thus you can turn a rectangle into a speech balloon or any other shape you like, for example.

To change the shape, simply click on the desired AutoShape in the list.

Tip: You will also find this option as the command **Change AutoShape** directly on the contextual ribbon tab **Object**. For more information, see Changing the AutoShape of objects.

For more information on AutoShapes, see Inserting drawings.

Inner text tab (available only for AutoShapes with text)

This tab appears only for AutoShapes to which text has been added (see Add text to AutoShapes) and for Text frames.

You can use this tab to change settings for the text in the text frame.

Tip: Some of these options can also be found directly as commands on the contextual ribbon tab **Object** | group **Inner text**.

Available options:

- **Inner margins** section

  Here, you can change the inner margins for the text. If you enable the option **Automatic**, PlanMaker will automatically determine suitable margins.

- **Overlapping objects** section

  This setting determines what should happen if the object overlaps with another object.

  If the default setting **Ignore object** is enabled, PlanMaker will do nothing. In the area of overlap, the contents of both objects will thus be printed on top of each other.

  If, on the other hand, you enable the option **Wrap text**, PlanMaker will automatically wrap the text in this object so that it flows around the area of overlap.

- **Rotate by...** section

  Here, you can rotate the text by the specified angle.

- **Vertical alignment** section

  This setting determines how the inner text should be aligned between the top and bottom edges of the drawing:

Option	Explanation
Top	The text is aligned to the top edge of the object. (This is the default setting.)
Centered	The text is centered between the top and bottom edges.
Bottom	The text is aligned to the bottom edge of the object.
Justified	The lines of text are vertically justified and are thus evenly distributed so that the text starts exactly at the top edge of the object and ends exactly at the bottom edge.

- **Data link** section

For AutoShapes, you can create a link to a cell. This will display the contents of this cell in the object. For more information, see also Showing the contents of a cell in objects.

Note: The option is available in the dialog box only when you add text to an AutoShape (so that the "Inner text" tab is displayed) and then remove the text from the AutoShape (so that a link can be added).

Text tab (available only for TextArt objects)

This tab appears only for TextArt objects. You can use this tab to select the desired TextArt effect as well as to enter and format the text to be displayed.

Available options:

- **Text**

  Here, you can type the text to be displayed.

- **Variants**

  In this list, you can select the effect with which the text should be displayed.

- **Font** section

  Here you can change the font and enable/disable the text styles **Bold** and **Italic**.

  If you enable the option **Same height**, all letters (including lowercase letters) will be stretched to a uniform height.

- **Spacing** section

  The **Characters** option changes the spacing between characters. If values are less than 100%, the characters are closer together. For values greater than 100%, the characters are further apart.

  The option **Lines** changes the line spacing. This setting only affects text that consists of multiple lines.

- **Alignment** section

  Here, you can change the alignment of the text. This setting only affects text that consists of multiple lines.

- **Vertical text**

  If you enable this option, the letters of the text will be rotated by 90 degrees.

Transformation tab (available only for TextArt objects)

This tab appears only for TextArt objects.

You can use this tab to specify the type of shaping effect to be applied to the text. To do so, click on the desired type of transformation.

QR codes/Barcodes

This feature is only included in **SoftMaker Office Professional** and **NX Universal**.
In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

You can use the ribbon command **Insert | QR code/barcode frame** in PlanMaker to generate your own QR barcodes or some common types of linear barcodes and insert them into the current document. The resulting object corresponds to a picture frame.

Linear barcodes contain a series of lines with different widths, which you know from everyday items (food, books, tickets, etc.).

QR codes, on the other hand, are primarily used to encode URLs that can be accessed by scanning them. In practice, you can use this format, for example, to issue invoices including a QR code to your customers. After scanning the code, the customer will be taken directly to the web page you have defined.

All types of codes that you create in PlanMaker can be scanned with a standard scanner app directly from the screen or from the printout of the document. Use the scanner that is integrated into your smartphone, for example.

To create and insert a QR code/barcode, proceed as follows:

1. Choose the ribbon command **Insert** | group **Objects | QR code/Barcode frame**.
2. This will then open a dialog box in which you can select under **Type** whether you want to generate a QR code or one of the available linear barcodes. For explanations of the different types, see "Options of the dialog box" below.
3. Enter the desired URL, text, or number in the field below.

   Alternatively, you can also tap with the mouse directly on a cell, when it contains the desired data for creating the code.

   Note: For the types *Barcode EAN-13* and *Barcode EAN-8*, you can only enter numbers. They must have exactly 12 or 7 digits.

4. Click on **Insert**. The QR code or barcode is now inserted as a picture frame.

   You can then move or resize the picture frame:

   To move it, click on the picture frame with the mouse, and then drag it (while still holding down the mouse button) to the desired position.

   To adjust its size, drag one of the round handles surrounding the picture frame.

Options of the dialog box

After choosing the ribbon command **QR code/barcode frame**, you can select from the following types of codes in the dialog box:

- **QR code**

  A QR code is a two-dimensional code that can accommodate large amounts of data. You can enter all numbers and characters here, and there is no limit on the length of the entry.

- **Barcode128**

  Code128 is an alphanumeric bar code with high data density. You can use numbers and characters of the ASCII code here, and there is no limit on the length of the entry.

  The option **With text** can only be selected for this type. If it is enabled, the entered text is displayed below the barcode. If the option is disabled, the display of the text is suppressed.

- **Barcode EAN-13**

  An EAN-13 code is a numeric barcode. 12 digits must be entered, while the 13th digit is a check digit that is automatically generated and added at the end.

- **Barcode EAN-8**

  An EAN-8 code is a numeric barcode. 7 digits must be entered, while the 8th digit is a check digit that is automatically generated and added at the end.

Changing properties of QR code/barcode objects

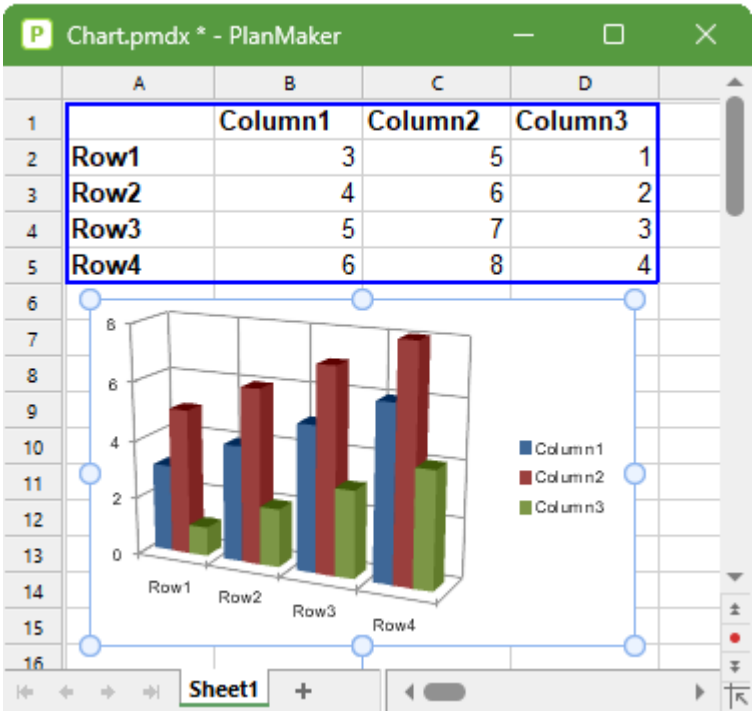
An inserted QR code/barcode is actually nothing more than a picture frame. To change the properties, right-click on the object and select **Picture:Properties** in the context menu. In the dialog box that appears, switch to the **QR code/Barcode** tab, where you can change the type of barcode and the data for a URL, text or number.

You can use the **Export** button to save a copy of the barcode picture in PNG format on the hard disk. The dialog box "Save as" appears for this purpose.

All other options of the dialog box (or on the contextual ribbon tab **Picture**) correspond to a large extent to the properties of picture frames as described in the Pictures section.

Charts

You can use *charts* to graphically display the numeric values of a worksheet and thus present the "raw numbers" more clearly.



In this chapter, you will learn everything you need to know about charts. The following topics are covered:

- **Inserting charts**

  First, you will learn how to create a chart: Select the cells containing the data to be displayed, choose the ribbon command **Insert | Chart frame** and select a chart type.

- **Editing charts**

  This section contains all the information you need to edit charts. You will learn how to change the chart type and the arrangement of the data series, how to work with the individual chart elements (data series, axes, legend, etc.) and how to change the general chart properties.

- **Updating charts**

  A chart always retrieves the values to be displayed from the cells that were selected when the chart was created. If values in these cells change, the chart will be updated immediately. If desired, however, you can disable this automatic updating.

- **Moving charts to another worksheet**

  The ribbon command **Chart | Chart location** enables you to move a chart to another worksheet or even create a separate *chart worksheet* for it. This chart worksheet contains only the full-sized chart without any cells.

- **Saving charts as pictures**

  The ribbon command **Chart | Save chart as a picture** enables you to save an image of a chart in a picture file.

Inserting charts

The ribbon command **Insert** | group **Objects** | **Chart frame** enables you to insert new charts into the document.

To do so, proceed as follows:

1. Select the cells containing the values to be displayed in a chart.

   Tip: If you select row and/or column headings, they will be used automatically to label the axes and the legend.

2. Choose the ribbon command **Insert** | group **Objects** | **Chart frame** to open the dialog box. (Click on the icon itself.)

   Alternatively, you can click on the small arrow of the icon to list a dropdown menu with various chart types that you can insert directly by clicking on them. If you choose **More** here, the dialog box will be shown again.

3. In the dialog box, you can specify which type of chart you want to insert. To do so, select the required **Chart type** and **Subtype**.

   For more information about chart types, see Changing the chart type.

4. Confirm with **OK**.

5. The chart is now inserted.

   You can then move or resize the chart:

   To move the chart, click on it and (with the left mouse button still held down) drag it to the desired position.

   To resize it, drag one of the round handles surrounding the chart.

Editing charts

You will learn how to edit charts in the next sections. The information covers the following topics:

- **Changing the chart type**

  Use the ribbon command **Chart | Chart type** to determine the form in which the data will be represented in the chart – for example, as bars, lines or in the form of a pie chart, etc.

- **Changing the arrangement of data series**

  On the ribbon tab **Chart**, you can specify whether you want to evaluate the data series to be displayed as **Series in columns** or **Series in rows**.

- **Showing/hiding chart elements**

  A chart contains various elements, such as data series, axes, a legend, etc. You can use the ribbon command **Chart | Add chart element** to show or hide certain elements of the chart if necessary.

- **Editing chart elements**

  You can select the elements of a chart by clicking on them and then edit them. You will learn how to do this in this section.

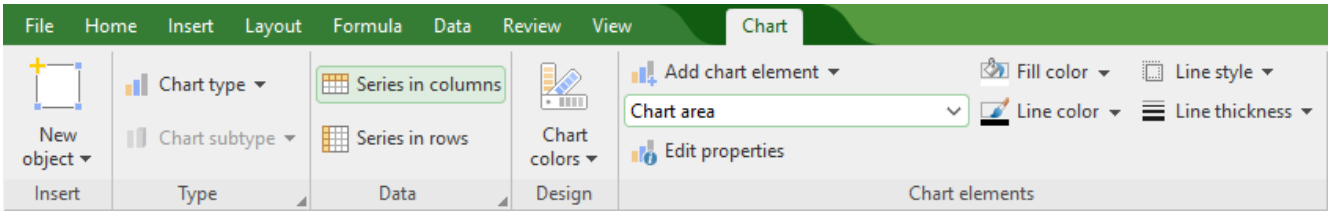
- **Changing chart properties via the dialog box**

  In addition to the properties of individual chart elements, there are also *common* chart properties that you can change. These include various layout options, the chart type, settings for the data series, etc. You can change these options on the contextual ribbon tab **Chart** (see below) or in the **Properties** dialog box (accessible via the context menu).

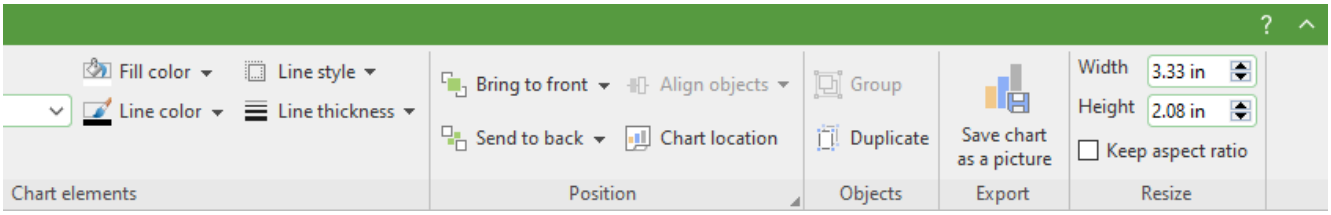
For basic information on working with objects, see Objects.

Using the ribbon tab "Chart"

When you select a chart, the contextual ribbon tab **Chart** appears automatically.



The contextual ribbon tab "Chart", left section



The contextual ribbon tab "Chart", right section

This ribbon tab contains icons for the most important functions for editing charts:

- **New object:** This option allows you to insert a new object into the document, and different object types are available.
- **Chart type/Chart subtype:** This option allows you to select a chart type. (A subtype is only available for some chart types.) For more information, see Changing the chart type.
- **Series in columns/rows:** This option allows you to specify whether the data to be displayed should be evaluated column by column or row by row. For more information, see Changing the arrangement of data series.

- **Chart colors:** This option allows you to change the color scheme of the chart.
- **Add element:** This option allows you to show or hide chart elements such as axis titles, gridlines, legends, etc. For more information, see Showing/hiding chart elements.
- **List of all chart elements** contained in the chart. If you click on an element in the list, it will be selected in the chart – if you select an element in the chart, it will be displayed in the list.
- **Edit properties:** You click on this option to edit the selected chart element and the corresponding dialog box opens. For more information, see Editing chart elements.
- **Fill color, Line color, Line style, Line thickness:** You can use these options to change the fill and border lines of the selected chart elements. For more information, see Editing chart elements.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Chart location:** For more information, see Moving charts to another worksheet.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option allows you to create an immediate copy of the selected chart.
- **Save chart as a picture:** This option allows you to save an image of a chart as a picture file. For more information, see Saving charts as pictures.
- **Width, Height**: These options change the size of the chart. For more information, see Changing position and size of objects.

Using the dialog box

Many (but not all) of these functions and some additional options can be found in the *Object properties* dialog box that you open via the context menu (entry **Chart: Properties**).

Alternatively, you can open the dialog box by clicking on the group arrow ◢ in the bottom right corner of any command group of the ribbon tab **Chart**.

All options of the dialog box are explained in Changing chart properties via the dialog box.

Changing the chart type

The *chart type* determines how data will be displayed in a chart – for example, as columns, bars, lines, etc.

To change the chart type, proceed as follows:

Directly via the icons of the command group "Type"

Many chart types can be used directly via the icons of the command group **Type** on the contextual ribbon tab **Chart**:

1. Select the chart by clicking on it.
2. Click on the **Chart type** icon on the ribbon tab **Chart** | group **Type**.
3. Select the desired **Chart type** from the dropdown menu (see below for explanations of the different chart types). You may use the icon below it to select a **Chart subtype** (only available for certain chart types).

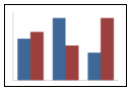
Additional options via the dialog box of the command group "Type"

You will find some more options via the dialog box of the command group **Type** on the contextual ribbon tab **Chart**. Here, you can also survey the possible appearance of the chart in a preview.

1. Select the chart by clicking on it.
2. On the ribbon tab **Chart** | group **Type**, click on the group arrow in the bottom right corner.
3. In the dialog box, switch to the ribbon tab **Chart type**.
4. Select the desired **Chart type** and **Subtype**.

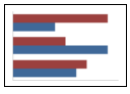
   Tip: A small preview of the currently selected chart will be shown in the right half of the dialog box. You can enable or disable this preview at any time using the >> button or the << button.

The following chart types are available:



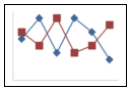
Column chart

Column charts are a good way to compare values. Each value is represented by a correspondingly high column.



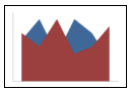
Bar chart

Bar charts correspond to column charts; however, the values are displayed horizontally rather than vertically.



Line chart

Line charts display values as points and/or connecting lines.

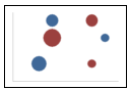


Area chart

Area charts are line charts where the space between the X axis and the individual lines is filled with color.

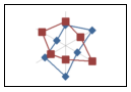
XY scatter chart

XY scatter charts display values as points and/or lines. Unlike line charts, the data area must contain both the X and Y coordinates of the data points.



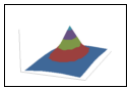
Bubble chart

Bubble charts display values as filled circles ("bubbles"). Bubble charts are usually based on data series containing three values: X coordinate, Y coordinate, and bubble size.



Radar chart

Radar charts display values in a web. The zero point is in the center of the web.



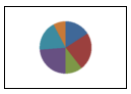
Surface chart

Surface charts display values as a three-dimensional shape. Areas plotted in the same color indicate that they contain similar values.

The values to be displayed should have the following structure:

	y_1	y_2	y_3	...
x_1	z_{11}	z_{12}	z_{13}	...
x_2	z_{21}	z_{22}	z_{23}	...
x_3	z_{31}	z_{32}	z_{33}	...
...	...	...	...	...

(Z values determine the height.)



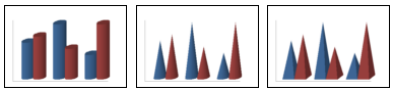
Pie chart

Pie charts are well suited to displaying the percentage of individual values of the total value. The size of each "pie slice" shows the ratio of the value to the total and to the other values.

Tip: You can highlight values by extracting the related slice of the pie from the pie. Select the slice of the pie and extract it via the mouse.

Doughnut chart

Doughnut charts, like pie charts, show the percentage of individual values of the total value. Unlike pie charts, however, they can display any number of data series (rings) and not just a single series.



Cylinder, Cone and Pyramid charts

Cylinder, **Cone** and **Pyramid charts** correspond to column and bar charts. However, they use cylinders, cones or pyramids instead of columns/bars to display the values.

Stock chart

Stock charts (also known as general **high-low charts**) are suitable for displaying the development of stock prices, among other things. They enable the highest price, lowest price and closing price to be seen at a glance, for example.

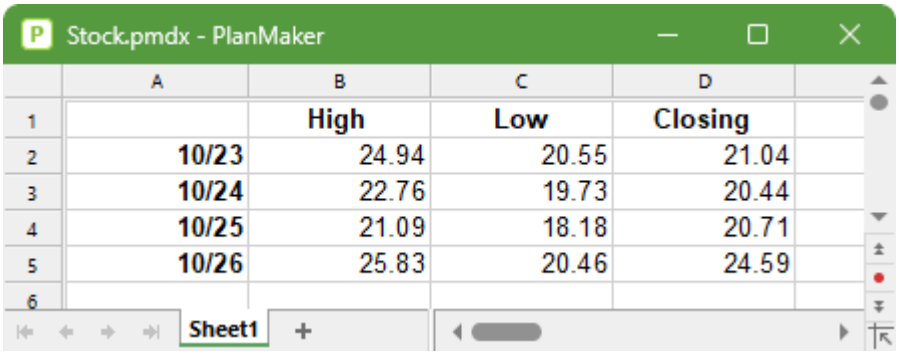
The following subtypes are available:

- **Stock chart (High, Low, Close)**

  This type of chart displays the following values: *highest price, lowest price, closing price*.

  It requires 3 data series in the above order.

  An example of the possible structure of the data (with the date added in column A):



- **Stock chart (Open, High, Low, Close)**

  This type of chart displays the following values: *opening price, highest price, lowest price, closing price*.

  It requires 4 data series in the above order.

- **Stock chart (Volume, High, Low, Close)**

  This type of chart displays the following values: *trading volume, highest price, lowest price, closing price*.

  It requires 4 data series in the above order.

The *trading volume* is displayed as an additional column with its own value axis.

- **Stock chart (Volume, Open, High, Low, Close)**

  This type of chart displays the following values: *trading volume, opening price, highest price, lowest price, closing price*.

  It requires 5 data series in the above order.

  The *trading volume* is displayed as an additional column with its own value axis.

Box plot chart

Box plots (or **box and whisker plots**) are useful for data analysis in statistics. Detailed explanations relating to the application of box plots would go beyond the scope of this manual. You can find further information on them via your preferred Internet search engine.

There are two subtypes: **Horizontal** and **Vertical**.

Depending on the number of data series passed to the chart, different variants of box plots will be rendered:

Note: You can display any type of value in a box plot. The following lists show examples only.

- **3-point box plot**

  This box plot displays, for example, the following values:

  Lower quartile (lower border of the box)
  Median (position of the line within the box)
  Upper quartile (upper border of the box)

  This box plot requires exactly **3** data series in the above order.

- **5-point box plot**

  This box plot displays, for example, the following values:

  Lower whisker (line below the box)
  Lower quartile (lower border of the box)
  Median (position of the line within the box)
  Upper quartile (upper border of the box)
  Upper whisker (line above the box)

  This box plot requires exactly **5** data series in the above order.

- **7-point box plot**

  This box plot displays, for example, the following values:

  Minimum (as a single marker)
  Lower whisker (line below the box)
  Lower quartile (lower border of the box)
  Median (position of the line within the box)
  Upper quartile (upper border of the box)
  Upper whisker (line above the box)
  Maximum (as a single marker)

  This box plot requires exactly **7** data series in the above order.

Changing the arrangement of data series

You can use the two icons **Series in columns** and **Series in rows** on the contextual ribbon tab **Chart |** group **Data** to determine whether the data to be displayed should be evaluated *column by column* or *row by row*:

- **By columns**

  The first icon is selected by default, thus the data will be evaluated column by column. The values in the first column will be the first data series, the values in the second column will be the second data series, etc.

- **By rows**

  If you click on the second icon, the data will be evaluated row by row. The assignment of the rows/columns in the worksheet to the X axis/Y axis in the chart is thus reversed.

Alternatively, you will also find this setting in the dialog box of the ribbon tab **Chart |** group **Data** (click on the group arrow ◢) and switch to the **Data source** tab.

Showing/hiding chart elements

Certain elements of the chart can be shown or hidden if necessary. Some of them are shown automatically when you create the chart, for example, the axes and the legend.

Tip: See Editing chart elements for an illustration of the most important chart elements.

The following elements can be shown/hidden:

- Axes and axis titles
- Chart title
- Data_labels
- Error bars
- Gridlines
- Legend
- Lines
- Trendline
- Up/down bars

For more information, see the following sections.

Showing/hiding axes and axis titles

The horizontal X axis of a chart is also known as the *category axis*.

The vertical Y axis of a chart is also known as the *value axis*.

The Z axis of a chart is also known as the *series axis*. It is only available for certain three-dimensional chart types.

Note: The X axis and Y axis are shown automatically when the chart is created.

To show or hide an axis, proceed as follows:

Via the "Add chart element" icon

You can use the **Add chart element** icon on the contextual ribbon tab **Chart** to show or hide the X axis and/or Y axis (the Z axis only in the dialog box, see below).

To do so, select the chart and use the ribbon command **Add chart element** to select the entry **Axis**. In the following submenu, you can use the entry *Primary horizontal* to show or hide the X axis and *Primary vertical* to show or hide the Y axis.

The entry **More** takes you to a dialog box with additional options (see below).

Axis title: To add, change or remove axis titles, proceed as follows:

- **Adding axis title**

  If you want to add an axis title to the axis, choose the ribbon command **Add chart element** and then select the entry **Axis title**. *Primary horizontal* shows an axis title for the X axis, while *Primary vertical* shows it for the Y axis.

  The entry **More** also takes you to the dialog box here (see below).

- **Changing an axis title**

  To rename an axis, double-click on the axis title and switch to the **Text** tab in the following dialog box. Enter the desired text here.

- **Removing an axis title**

  To remove an axis title, hide it again via the command **Add chart element** or select it directly in the chart and press the **Del** key.

Via the dialog box

For additional axis options (for example, the Z axis or secondary axes), choose the path via the dialog box:

1. Select the chart by clicking on it.

2. Use the ribbon command **Chart | Add chart element** to choose the entry **Axis** (or **Axis title**) and select **More** in the following submenu to open the dialog box.

   Alternatively: Right-click to open the context menu and select the entry **Chart: Properties**.

3. Switch to the **Elements** tab.

   Axes: The checkbox in front of each axis can be used to enable/disable the display of the relevant axis.

   Axis title: To the right of the axis, you can give it a name. This will then be displayed below/next to the axis.

   For more information on the options of the "Elements" tab, see Chart properties, Elements tab.

Editing axes and axis titles

For more information on editing the properties of *axes*, see Category axis (X axis), Value axis (Y axis) and Series axis (Z axis).

For information on editing the properties of the *axis title*, see Axis title.

Showing/hiding chart titles

You can show or hide a *chart title* for the chart. The title will then be displayed above the chart.

To do so, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Chart title**. In the following submenu, select **Above chart** to add a chart title. If you select **None**, the chart title will be removed.

If you select **More**, you will be taken to the dialog box (see below) where you can rename the chart title.

Alternatively: Double-click on the chart title and you can then rename the title on the **Text** tab.

To remove a chart title, select it and press the **Del** key.

Via the dialog box

To show or hide the chart title via the dialog box, proceed as follows:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Chart title** and select **More** in the following submenu to open the dialog box.

   Alternatively: Right-click to open the context menu and select the entry **Chart: Properties**.

3. On the **Elements** tab, you will find an input field for the *Chart title* at the top. Type the desired title here.

To delete the chart title, simply remove the text from this input field.

For more information on the options of the "Elements" tab, see Chart properties, Elements tab.

For information on editing the properties of the chart title, see Chart title.

Showing/hiding data labels

The data points of a chart can be labeled. This allows you to display the exact value of the data points in the chart.

To show or hide data labels, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Data labels**. In the following submenu, you can choose between different options for positioning the data label (left, centered, above, etc.). A check mark in front of it indicates the setting which is currently enabled.

If you choose the entry **None**, the data labels will be removed.

Tip: To remove a single label, select it and press the **Del** key.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

For additional options for data labeling, choose the path via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Data labels** and select **More** in the following submenu to open the dialog box.

   Alternatively: Select the desired data series and right-click to open the context menu and select the entry **Series: Properties**.
3. Switch to the **Data labels** tab and make the desired settings here.

For more information on the options of the dialog box, see Data series and data points, "Data labels" tab.

Showing/hiding error bars

Note: This chart element is only available for certain two-dimensional chart types, such as column, bar and line charts.

You can draw *error bars* for the data points of a data series. Error bars display the possible/tolerable error of the values in the form of a line with a short dash at the end.

To show or hide error bars, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Error bars**. In the following submenu, you can choose from a predefined selection for displaying the values. A check mark in front of it indicates the setting which is currently enabled.

If you choose the entry **None**, the data labels will be removed.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

Additional options for error bars can be found via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Error bars** and select **More** in the following submenu to open the dialog box.

   Alternatively: Select the desired data series and right-click to open the context menu and select the entry **Series: Properties**.
3. Switch to the **Error bar** tab and select the desired settings here.

For more information on the options of the dialog box, see Data series and data points, "Error bar" tab.

Showing/hiding gridlines

Showing *gridlines* in a chart can improve the legibility of its values. Gridlines are a network of lines that extend across the plot area, starting from one of the axes of the chart.

The following types of gridlines are available:

- *Major gridlines* are indicated at every major tick mark on the axis.
- *Minor gridlines* subdivide the major grid into an even finer grid if necessary.

You can enable gridlines separately for each axis.

To show or hide gridlines, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Gridlines**. In the following submenu, you can enable or disable horizontal (for the Y axis) and vertical (for the X axis) major gridlines and minor gridlines for the primary axis. A check mark in front of them indicates the gridlines which are currently enabled.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

For additional options (for example, gridlines for secondary axes), choose the path via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Gridlines** and select **More** in the following submenu to open the dialog box.

   Alternatively: Right-click to open the context menu and select the entry **Chart: Properties**.
3. Switch to the **Elements** tab and enable the option **Major grid** and/or **Minor grid** for the desired axes.

For more information on editing gridlines, see Gridlines.

Showing/hiding the legend

If desired, you can show (or hide) a *legend* in the chart. A legend is a small box that shows which data series are displayed in which color/pattern.

To do so, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Legend**. In the following submenu, you can specify where to position the legend. A check mark in front of it indicates the option which is currently enabled.

You can hide the legend by selecting the entry **None**.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

For additional options, choose the path via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Legend** and select **More** in the following submenu to open the dialog box.

   Alternatively: Select the legend and right-click to open the context menu and select the entry **Legend: Properties**.
3. On the **Legend** tab, specify where the legend should be positioned. If you select the option **Custom**, the legend can be moved to any position (using the mouse). If you select the option **None**, the legend will not be displayed.

> **Tip:** If you choose the entry **Chart: Properties** via the context menu, the main dialog box for charts opens. You will find the option **Legend** here also on the **Elements** tab.

For more information on editing the properties of a legend, see Legend.

Showing/hiding lines

> **Note:** This option can only be used for line charts and area charts.

You can use the options for *lines*, for example, to illustrate the spacing between the individual data series in a line chart.

The following types of lines are available:

- *Drop lines* are lines displayed from the data points to the horizontal axis.
- *High-low lines* are lines displayed between the highest and lowest data points (for two or more data series).

To show/hide drop lines or high-low lines, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Lines**. In the following submenu, you can choose between *Drop lines* and *High-low lines*. A check mark in front of it indicates the option which is currently enabled.

You can hide the lines by selecting the entry **None**.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

For additional options, choose the path via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Lines** and select **More** in the following submenu to open the dialog box.

Alternatively: Select the desired data series and right-click to open the context menu and select the entry **Series: Properties**.

3. Switch to the **Options** tab and enable the desired settings.

For more information on the options of the dialog box, see Data series and data points, "Options" tab.

Showing/hiding the trendline

Note: Trendlines are available only for certain chart types (including two-dimensional column, bar, and line charts). Other limitation: They are not available for stacked chart types.

If desired, you can add *trendlines* to the data series of a chart.

Trendlines can be used to graphically display the *trend* of a data series (i.e., to what extent the data series rises/falls on average). Trendlines can also be extended beyond the given data points, so that they display a forecast for future values.

A statistical technique called *regression analysis* is used to calculate trendlines.

To show/hide trendlines, proceed as follows:

Via the "Add chart element" icon

Select the desired data series to which you want to add a trendline.

Use the ribbon command **Chart | Add chart element** to choose the entry **Add trendline**. In the following submenu, select one of the predefined trendline types.

The entry **Remove all** removes all trendlines (select the chart for this purpose). To remove individual trendlines, select the desired trendline and then choose **Remove all** or simply press the **Del** key.

The entry **More** takes you to the dialog box with additional options (see below).

Via the dialog box

You will find more trendline types and additional options via the dialog box:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Add trendline** and select **More** in the following submenu to open the dialog box.

   Alternatively: Select the desired data series and right-click to open the context menu and select the entry **Add trendline**.
3. Switch to the **Trend** tab and enable the desired settings.

For more information on the options of the dialog box, see Trendlines.

Showing/hiding up/down bars

Note: This option can only be used for line charts.

You can use the option *Up/down bars* to display bars between the first and last data series, for example, in a line chart with two or more data series.

To do so, proceed as follows:

Via the "Add chart element" icon

Select the chart and use the ribbon command **Chart | Add chart element** to choose the entry **Up/down bars**. In the following submenu, select the same entry.

You can use the entry **None** to remove the up/down bars.

The entry **More** takes you to the dialog box (see below) where you can also set the spacing between the bars.

Via the dialog box

Choose the path via the dialog box to make further settings:

1. Select the chart by clicking on it.
2. Use the ribbon command **Chart | Add chart element** to choose the entry **Up/down bars** and select **More** in the following submenu to open the dialog box.

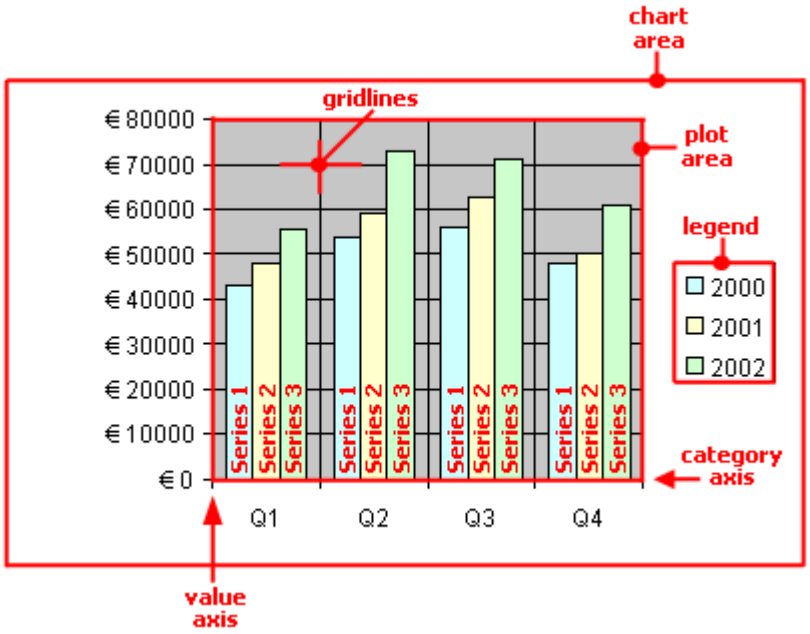
   Alternatively: Select any data series and right-click to open the context menu and select the entry **Series: Properties**.
3. Switch to the **Options** tab and enable the checkbox in front of **Show up/down bars with gap**. In the input field below, you can set the gap between the bars.

For more information on the options of the dialog box, see Data series and data points, "Options" tab.

Editing chart elements

Charts consist of several components. In addition to the data series themselves, for example, there are axes, a legend, etc. These individual components of a chart are called the *chart elements*.

The following figure shows the most important chart elements:



Series 1, Series 2, etc. represent the chart's *data series*.

Three-dimensional chart types contain some additional chart elements (floor, walls, etc.), which will also be described on the following pages.

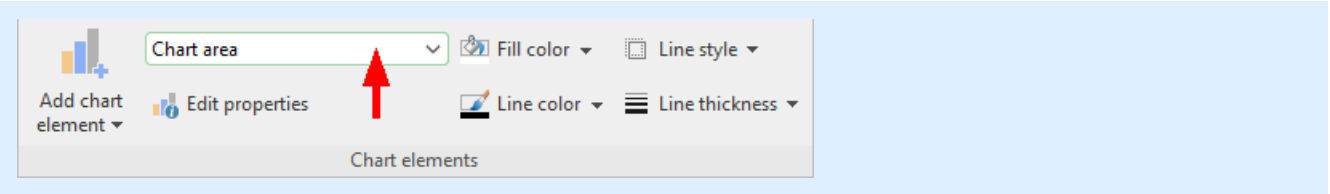
Each of these chart elements can be selected at the click of a mouse and then edited.

Selecting chart elements

To select a chart element, first click on the chart itself to select it and then click on the desired chart element. To select a data series, for example, click on any value within this series.

Note: To select a *single* value within a data series, first click on the data series and then click on the desired single value.

Tip: Alternatively, you can select chart elements by expanding the list of elements on the ribbon tab **Chart** | group **Chart elements** and selecting the desired element from it. Here, you can always see which chart element is currently selected.

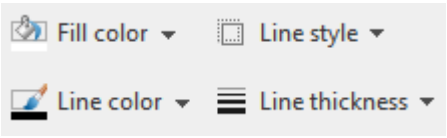


Changing the position and size of chart elements

The position and size of some chart elements (for example, the legend) can be changed. To move an element, select it and then drag it to the desired position. To change its size, drag one of the handles surrounding the element.

Changing the fill and borders of chart elements

The fill color and border lines of chart elements can be applied directly with the buttons of the command group **Chart elements** on the ribbon tab **Chart**. To do so, select a chart element and make the desired adjustments using the buttons **Fill color**, **Line color**, **Line style** and **Line thickness**.



Note: In the case of the buttons *Fill color* and *Line color*, the last color that you used will be applied again when you click directly on the icon itself. By clicking on the icon's arrow, you open the color palette.

Only for the *Fill color* button: If you click on the icon's arrow, you can use the entry **More Fills** to also apply more complex fills. For more information, see Object properties, Fill tab. By clicking on the entry **More colors**, you reach a dialog box with color definitions, which you use according to the descriptions of the section Document properties, Colors tab.

You will also find fill and border settings via the dialog box of the chart elements (see next paragraph). You can also change other properties here, depending on the type of chart element.

Changing properties of chart elements

In addition to changing the properties of the chart itself, as described later in Changing chart properties via the dialog box, you can also edit the properties of individual chart elements.

To change the properties of a chart's **legend**, for example, proceed as follows:

1. Select the chart by clicking on it.
2. Right-click on the legend to open its context menu.

   Android/iOS: In these versions, you can also open the context menu with your finger: Just tap on the screen and hold your finger there for about a second.
3. In the context menu, choose the command **Legend: Properties** (*not* the command **Chart: Properties**).

A dialog box will now appear in which you can make the desired settings.

Tip: Alternatively, you open the dialog box via the ribbon command **Edit properties** .
Even faster: Open the dialog box by double-clicking on the desired element.

The properties that you can change in this dialog box depend on the type of chart element for which you opened it. The following sections provide more information about all available chart elements and their respective properties:

- Chart area
- Plot area
- Walls (three-dimensional charts only)
- Floor (three-dimensional charts only)
- Corners (three-dimensional charts only)
- Data series and data points
- Trendlines
- Category axis (X axis)
- Value axis (Y axis)
- Series axis (Z axis)
- Axis title
- Chart title
- Gridlines
- Legend

Chart area

The *chart area* is the entire area of a chart frame.

Edit properties: To change the properties of the chart area, select it, right-click it to open the context menu and choose the command **Chart area: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to simply double-click on the chart area.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can add a border to the entire chart. **Auto** gives the chart a thin gray standard line as a border. **None** removes the border. The option **Custom** lets you customize the line style.

  If you enable the option **Shadow**, the chart frame will also be highlighted with a shadow. The **Adjust** button opens additional settings for the appearance of the shadow effect.

- **Fill** section

  Here, you can change the fill of the chart area. **Auto** gives the chart area a white fill area. The fill will be transparent if you use the option **None**. You can use the option **Color** to customize the color for the fill.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

Font tab

Here, you can change the text formatting for *all* chart elements.

To do so, select the desired font, font size, alignment, etc.

Important: Changes will affect *all* chart elements, even if they were previously formatted in a different font.

Plot area

The *plot area* of a chart is the area where the data series, axes and gridlines are drawn.

Edit properties: To change the properties of the plot area, select it, right-click it to open the context menu and choose the command **Plot area: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the plot area.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can add a border to the plot area. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

- **Fill** section

  Here, you can change the fill of the plot area. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

Walls (three-dimensional charts only)

Walls are the back walls of charts that use a three-dimensional chart type.

Edit properties: To change the properties of the walls, select one of the walls, right-click it to open the context menu and choose the command **Walls: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the walls.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can add a border to the walls. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

- **Fill** section

  Here, you can change the fill of the walls. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

3D view tab

Here, you can set the 3D effect for the entire chart. For more information, see Chart properties, 3D view tab.

Floor (three-dimensional charts only)

Floors are only present in three-dimensional chart types.

Edit properties: To change the properties of the floor, select it, right-click it to open the context menu and choose the command **Floor: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the floor.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can add a border to the floor. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

- **Fill** section

  Here, you can change the fill of the floor. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

3D view tab

Here, you can set the 3D effect for the entire chart. For more information, see Chart properties, 3D view tab.

Corners (three-dimensional charts only)

Corners are only present in three-dimensional chart types.

Edit properties: To change the properties of the corners, select one of them, right-click it to open the context menu and choose the command **Corners: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**.

You can make the following settings:

3D view tab

Here, you can set the 3D effect for the entire chart. For more information, see Chart properties, 3D view tab.

Data series and data points

Data series are the most important chart elements. They represent the data to be evaluated – in the form of columns, bars, lines, pie segments, etc., depending on the chart type selected.

A **data series** is the graphical representation of a *column* or *row* of cell values on which the chart is based. Each data series, in turn, consists of multiple **data points**, thus the individual cell values.

Edit properties: To change the properties of data series or single data points, proceed as follows:

- If you click on any of the data points of a data series, the *entire* data series will be selected. You can now edit the properties of this data series by right-clicking on it to open the context menu and then choosing the command **Series: Properties**.
- If you click on the same data point once again, only this *single* data point will be selected. The command in the context menu is then called **Point: Properties**.

  Tip: Alternatively, you can choose these commands via the **Edit properties** icon on the ribbon tab **Chart |** group **Chart elements** after you have selected the data series (or the data point).

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart |** group **Chart elements**. For more information, see Editing chart elements.

- **Border** section (or **Line** section)

  Here, you can change the line style used to draw the graph. To do so, use the option **Custom** to select the desired line style. The option **None** removes the line.

  If you enable the option **Shadow**, the graph will also be highlighted with a shadow. You can use the **Adjust** button to make other settings for the appearance of the shadow effect. (For some chart types, the option **Smooth line** appears here instead, which enables you to give the graph a smoother gradient.)

- **Fill** section

  Here, you can change the fill of the data series/data points. **Auto** gives the fill area a standard color. The fill will be transparent if you use the option **None**. You can use the option **Color** to customize the color for the fill. The **color for negative values** can be assigned a different color for certain chart types.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

- **Marker** section (available only for certain chart types)

  Some chart types (for example, certain types of line charts) draw *markers* for each data point. Here, you can customize the appearance of such markers.

3D format tab

Note: The options on this tab are available only for certain chart types.

Use this tab to add a light effect or bevels to the data points to be displayed.

- **Material**

  Here, you can select the material that should be simulated to display the data points.

- **Light**

  Here, you can select the type of light that should be simulated.

- **Angle**

  Here, you can change the angle of the light source.

> ***Tip:*** The above effects usually look better if you also use the following option to add bevels to the data points.

- **Bevels** section

  Here, you can add a bevel effect to the graphical representation of the data points. If you apply these options to a bar chart, for example, the edges of the bars will be rounded. The larger the value, the rounder the edges will be displayed.

Data labels tab

Use this tab to add labels to the selected data points. This is useful, for example, if you want to have the exact values displayed for each data point.

- **Display** section

  Here, you can specify the type of label to be displayed. The default option is **Don't show**, thus no label is displayed. If, for example, you select the option **Show value**, the value of the relevant data point will be displayed, while the option **Show data point** will display its name, etc.

- **Display legend key in label**

  If you enable this option, the legend keys of the relevant data points will be displayed before the labels. The legend key is a small square in the same color as the data point.

- **Placement** section

  Here, you specify where the label should be placed (relative to the respective data point).

Note: If you label data points, you can also click on these labels in the chart. Thus, you can change their properties by selecting a different font or changing the text you want to display, for example.

The following program response also applies here: Clicking once selects *all* labels of a data series, while clicking again selects a *single* label.

Error bar Y tab

> ***Note:*** This tab is available only for certain two-dimensional chart types (including column, bar and line charts).

You can use this tab to add *error bars* to the data points of the selected data series. Error bars display the possible/tolerable error of the values in the form of a line with a short dash at the end.

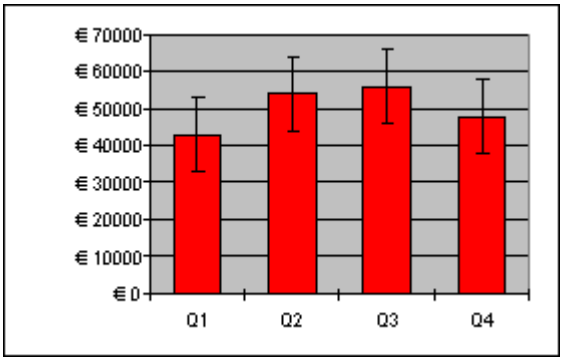


Chart with "Plus" and "Minus" error bars

To add error bars, select the desired type of error bar in the **Display** section: **Plus only** displays a positive error bar, **Minus only** displays a negative error bar, **Plus and minus** displays both error bars.

The option **Show as** lets you determine the appearance of the error bars: with or without short dashes at the ends.

In the **Value** section, you can change the length of the indicator line. If, for example, you select the option **Fixed**, the line will always have the specified length, while if you select the option **Percent**, the length will correspond to the specified percentage of the respective Y value, etc.

Error bar X tab

> ***Note:*** This tab is only available for XY scatter charts and bubble charts.

This tab is identical to the Error bar Y tab (see above), except that it displays error bars for the X values instead of the Y values.

Helper lines tab

> ***Note:*** This tab is available only for certain two-dimensional chart types (including column, bar and line charts).

You can use this tab to display various guides in the chart. If, for example, you enable the option **Average**, a dashed line indicating the average value of the selected data series will be displayed in the chart.

You can enable guides for **Average**, **Minimum/maximum**, **Standard deviation**, and **Linear approximation** (linear trend).

Tip: You can use the ribbon command **Chart | Add chart element** | **Add trendline** to show other types of trendlines in the chart if necessary. For more information, see Trendlines.

Options tab

This tab shows different settings for displaying the data series/data points, depending on the chart type, for greater clarification of the results.

For a line chart, for example, you will find the following settings:

Note: For other chart types, you will find different settings on the **Options** tab.

- **Displaying drop lines**

  If you enable this option, lines will be drawn from the data points to the horizontal axis.

- **Displaying high-low lines**

  If you enable this option, lines will be displayed between the highest and lowest data points (for two or more data series).

- **Up/down bars with gap**

  You can use this option to display bars between the first and last data series, for example, in a line chart with two or more data series. In the input field below, you can also set the gap between the bars.

For the above options, you can choose between different settings of the display in the section **Empty cells are ...** in case individual data points are missing.

If you select the option **Vary color for each data point**, the data points will be automatically colored with different colors (not available if there are multiple data series).

Tip: You can also format these chart elements, if enabled, by selecting them and right-clicking to open the context menu where you choose the relevant entry (**Drop lines**, **High-low lines**, **Up/down bars**): **Properties**.

Trendlines

Note: Trendlines are available only for certain chart types (including two-dimensional column, bar, and line charts). Furthermore, they should not be stacked.

If desired, you can have a *trendline* shown for each data series of the chart. For more information, see Showing/hiding the trendline.

Trendlines can be used to graphically display the *trend* of a data series (i.e., to what extent the data series rises/falls on average). Trendlines can also be extended beyond the given data points, so that they display a forecast for future values.

A statistical technique called *regression analysis* is used to calculate trendlines.

Edit properties: To change the properties of a trendline, select it, right-click it to open the context menu and choose the command **Trendline: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to simply double-click on the trendline.

You can make the following settings:

Format tab

- **Line** section

  Here, you can change the appearance of the trendline. **Auto** uses a thick black standard line. **None** removes the line. The option **Custom** lets you customize the line style.

Trendline tab

- **Type** section

  Here, you can select the type of trendline to be displayed. In addition to linear trendlines (i.e. simple best-fit lines), trendlines such as exponential or polynomial trendlines can also be displayed.

  The available types of trendlines are based on the following formulas:

Linear:	$y = ax + b$
Polynomial:	$y = b + c_1x + c_2x^2 + c_3x^3 + ... + c_6x^6$
Exponential:	$y = ce^{bx}$ (e = Euler's number)
Logarithmic:	$y = c \ln(x) + b$
Potential:	$y = cx^b$
Moving average:	$f(x_i) = (x_i + x_{i-1} + x_{i-2} + ... + x_{i-n+1}) / n$

- **Based on series**

  Here, you can specify the data series for which the trendline is to be displayed.

- **Forecast** section

  Here, you can extend the trendline beyond the existing data points.

  If, for example, there are 3 data points (for example, the sales results for the 1st, 2nd and 3rd year), you can enter a 1 for **Forward**. The line is then extended by an additional period (1 data point), thus displaying a forecast for sales in the 4th year.

- **Crossing point**

  Here, you can force the trendline to intersect the Y axis at a specific Y coordinate. This option is available only for certain types of trendlines.

- **Show formula in chart**

  If this option is enabled, the formula used to calculate the trendline will be displayed in the chart.

- **Show correlation in chart**

  If this option is enabled, the correlation coefficient R^2 of the data series will be displayed in the chart.

- **Name** section

  Here, you can change the name of the trendline if desired. This name is displayed in the legend. To do so, select the option **Custom** and enter the desired name.

Category axis (X axis)

The X axis of a chart is also called the *category axis*.

Tip: For more information on showing or hiding axes, see Showing/hiding axes and axis titles.

Edit properties: To change the properties of the category axis, select it, right-click it to open the context menu and choose the command **Category axis: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to simply double-click on the axis.

You can make the following settings:

Format tab

Tip: Lines can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Line** section

  Here, you can change the line style used to draw the axis. **Auto** uses a thin gray standard line. **None** removes the line. The option **Custom** lets you customize the line style.

- **Major ticks** and **Minor ticks** sections

  Here, you can determine whether small tick marks should be displayed to divide the axis and specify their appearance.

- **Tick labels** section

  Here, you can determine whether labels should be displayed for the tick marks and where to place them.

Scale tab

You can use this tab to change the scale of the axis.

Note: If the category axis contains date values, other settings appear here that are similar to the scaling options of the value axis (Y axis). For more information, see the descriptions in the next section Value axis (Y axis), "Scale tab".

- **Axis ends...** section

  This option allows you to determine where to end the visible part of the axis: **In categories** or **Between categories**.

Explanation: The first, second, third, etc., data point of all data series is called the category of a chart.

- **Number of categories between...** section

  By default, PlanMaker makes a tick mark on the axis for *each* category and labels it. You can change these intervals via the following options:

  The option **Tick marks** determines which categories are marked with a tick mark. Enter 1, and a tick mark is set for every category. Enter 2, and a tick mark is set for every second category, etc.

  The option **Tick labels** determines which tick marks to label. Enter 1, and each tick mark is labeled. Enter 2, and every second tick mark is labeled, etc.

- **Intersection between X and Y axis** section

  You can use this option to determine where the X axis (category axis) should intersect the Y axis (value axis): **At low end** (where the category axis begins), **At high end** (where the category axis ends) or at the specified category number.

- **Invert axis direction**

  If this option is enabled, the direction of the axis is reversed. The order of the categories is also reversed accordingly; the chart thus starts with the last category and ends with the first category.

Font tab

Use this tab to change the formatting of the axis labels.

In addition to font, font size, and text styles, you can also set the alignment, rotate the labels and change the numeric format.

Value axis (Y axis)

The Y axis of a chart is also called the *value axis*.

Tip: For more information on showing or hiding axes, see Showing/hiding axes and axis titles.

Edit properties: To change the properties of the value axis, select it, right-click it to open the context menu and choose the command **Value axis: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the axis.

You can make the following settings:

Format tab

Tip: Lines can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Line** section

  Here, you can change the line style used to draw the axis. **Auto** uses a thin gray standard line. **None** removes the line. The option **Custom** lets you customize the line style.

- **Major ticks** and **Minor ticks** sections

  Here, you can determine whether small tick marks should be displayed to divide the axis and specify their appearance.

- **Tick labels** section

  Here, you can determine whether axis labels should be displayed and where to place them.

Scale tab

You can use this tab to change the scale of the axis.

- **Lowest value** and **Highest value** sections

  These options determine where the visible part of the axis should begin and where it should end. Select either the option **Auto**, which results in PlanMaker automatically determining the most appropriate values, or select **Custom** and enter the desired value yourself.

- **Intersection between X and Y axis** section

  You can use this option to determine where the X axis (category axis) should intersect the Y axis (value axis):

  If **Auto** is selected, PlanMaker automatically determines the most appropriate value, and if **Maximum value** is selected, the X axis is positioned at the end of the Y axis. You specify your own Y value if you select **Custom value**.

- **Major step value** section

  Here, you can set the interval for the tick marks and labels on the axis. If you select the option **Auto**, PlanMaker automatically determines the most appropriate values.

- **Minor step value** section

  Here, you can set the interval between minor tick marks on the axis. If you select the option **Auto**, PlanMaker automatically determines the most appropriate values.

- **Invert axis direction**

  If this option is enabled, the direction of the axis is reversed. In a column chart, for example, the columns are no longer drawn from bottom to top, but from top to bottom.

- **Logarithmic scale**

  If this option is enabled, the axis will use a logarithmic scale (for example, 10, 100, 1000, etc.) rather than a linear scale.

Font tab

You can use this tab to change the formatting of the axis labels.

In addition to font, font size, and text styles, you can also set the alignment, rotate the labels and change the numeric format.

Series axis (Z axis)

The Z axis of a chart is also called the *series axis*. It is only available for certain three-dimensional chart types.

Tip: For more information on showing or hiding axes, see Showing/hiding axes and axis titles.

Edit properties: To change the properties of the series axis, select it, right-click it to open the context menu and choose the command **Series Axis: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the axis.

You can make the following settings:

Format tab

Tip: Lines can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Line** section

  Here, you can change the line style used to draw the axis. **Auto** uses a thin gray standard line. **None** removes the line. The option **Custom** lets you customize the line style.

- **Major ticks** and **Minor ticks** sections

  Here, you can determine whether small tick marks should be displayed to divide the axis and specify their appearance.

- **Tick labels** section

  Here, you can determine whether labels should be displayed for the tick marks and where to place them.

Scale tab

You can use this tab to change the scale of the axis.

- **Number of series between...** section

  By default, PlanMaker makes a tick mark on the axis for *each* data series and labels it. You can change these intervals via the following options:

  The option **Tick marks** determines which data series are marked with a tick mark. Enter 1, and a tick mark will be set for every data series. Enter 2, and a tick mark will be set for every second data series, etc.

  The option **Tick labels** determines which tick marks to label. Enter 1, and each tick mark will be labeled. Enter 2, and every second tick mark will be labeled, etc.

- **Invert axis direction**

  If this option is enabled, the direction of the axis will be reversed. The order of the data series will also be reversed accordingly; the chart will thus start with the last series and end with the first series.

Font tab

You can use this tab to change the formatting of the axis labels.

In addition to font, font size, and text styles, you can also set the alignment, rotate the labels and change the numeric format.

Axis title

You can name each axis. This will then be displayed below/next to the axis. For more information, see Showing/hiding axes and axis titles.

Edit properties: To change the properties of the axis title, select it, right-click it to open the context menu and choose the command **(Category/Value) axis title: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the axis title.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can change the line style for the border of the axis title. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

  If you enable the option **Shadow**, the axis title will also be highlighted with a shadow.

- **Fill** section

  Here, you can change the fill of the axis title. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

Text tab

Here, enter the desired axis title. If you delete the text completely from the input field, the axis title, as a chart element, will be removed. Alternatively, you can also use a link to cell content as text for the axis title.

Font tab

On this tab, you can change the formatting of the axis title.

In addition to font, font size, and text styles, you can also set the alignment, rotate the labels and change the numeric format.

Chart title

Here, you can enter a title for the chart. This is then displayed above the chart. For more information, see Showing/hiding chart titles.

Edit properties: To change the properties of the chart title, select it, right-click it to open the context menu and choose the command **Chart title: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the chart title.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can change the line style for the border of the chart title. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

  If you enable the option **Shadow**, you can also highlight the chart title with a shadow.

- **Fill** section

  Here, you can change the fill of the chart title. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

Text tab

Here, enter the desired title of the chart. If you delete the text completely from the input field, the chart title, as a chart element, will be removed. Alternatively, you can also use a link to cell content as text for the chart title.

Font tab

On this tab, you can change the formatting of the chart title.

In addition to font, font size, and text styles, you can also set the alignment, rotate the labels and change the numeric format.

Gridlines

You can show *gridlines* to make the values in a chart easier to read. (For more information, see Showing/hiding gridlines.)

Gridlines are a network of lines that extend across the plot area, starting from one of the axes of the chart.

The following types of gridlines are available:

- *Major gridlines* are indicated at every major tick mark on the axis.
- *Minor gridlines* subdivide the major grid into an even finer grid if necessary.

Edit properties: To change the properties of the major grid of the value axis, for example, select one of its gridlines, right-click it to open the context menu and choose the command **Value axis major gridlines: Properties**.

Tip: Alternatively, you can chose this command via the **Edit properties** icon on the ribbon tab **Chart**.

You can make the following settings:

Format tab

Tip: The appearance of the lines can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Line** section

  Here, you can change the line style used to draw the gridlines. **Auto** uses thin gray standard lines. **None** removes the lines. The option **Custom** lets you customize the line style.

Changing the scale of gridlines

To change the distances between the gridlines, select the corresponding axis, choose its properties and make the desired changes on the **Scale** tab.

Legend

If desired, you can show a *legend* in the chart. For more information, see Showing/hiding the legend.

A legend is a small box that shows which data series are displayed in which color/pattern.

Edit properties: To change the properties of the legend, select it, right-click it to open the context menu and choose the command **Legend: Properties**.

Tip: Alternatively, you can choose this command via the **Edit properties** icon on the ribbon tab **Chart** | group **Chart elements**. The other alternative is to double-click on the legend.

Note: If you only want to edit a single *legend entry* (you can also apply a formatting to it that differs from that of the legend as a whole), then click again in the area of the desired legend entry after selecting the legend. You can now choose the command **Legend entry: Properties** via the context menu.

You can make the following settings:

Format tab

Tip: Borders and fill can also be conveniently adjusted directly via the buttons of the ribbon tab **Chart** | group **Chart elements**. For more information, see Editing chart elements.

- **Border** section

  Here, you can change the line style of the legend's border. To do so, use the option **Custom** to select the desired line style. The option **None** removes the border.

  If you enable the option **Shadow**, the legend will also be highlighted with a shadow.

- **Fill** section

  Here, you can change the fill of the legend. To do so, use the option **Color** to select the desired fill color. The fill will be transparent if you use the option **None**.

  If you select the fill type **Effects** and click on the **Customize** button, the program will display a dialog box that will allow you to also apply more complex fills (such as a color gradient). The use of this dialog box is described in Object properties, Fill tab.

Font tab

You can use this tab to change the character format (font, font size, text styles, etc.) of the legend.

Legend tab

Here, you can specify where the legend should be displayed.

If you select the option **Custom**, the legend can be moved to any position (using the mouse).

If you select **None**, the legend will be hidden.

Changing chart properties via the dialog box

The previous sections discussed how to edit the properties of specific chart *elements*. This section discusses the properties of the chart *itself*.

To change the properties of a chart, click to select the chart and right-click to open the context menu. Select the entry **Chart: Properties**.

Tip: Alternatively, you can open the dialog box by clicking on the group arrow ◢ in the bottom right corner of any command group of the ribbon tab **Chart**.

The settings that can be made here are described on the following pages.

Chart properties, Format tab

On the **Format** tab, you can change the size and positioning of the chart.

For more information, see Object properties, Format tab.

Chart properties, Properties tab

On the **Properties** tab, you can change the general settings of the chart.

For more information, see Object properties, Properties tab.

Chart properties, Chart type tab

On the **Chart type** tab, you can change the chart type:

First, select the desired **Chart type** on the left and then the desired **Subtype** on the right. For more information on the available chart types, see Changing the chart type.

Color palette: You can change the color scheme of the chart here.

Reset all settings in chart: This option restores any changes you made to the properties of the chart back to their original state.

On the right side of the dialog box, a **Preview** shows the result of your customizations. If necessary, use the button >> to open the preview.

Chart properties, Data source tab

On the **Data source** tab, you can make settings for the data source area on which the chart is based:

- **Data source area**

  Here, you can specify the cell range that contains the data on which the chart is based. By default, the cell range you selected when creating the chart is used. You usually don't have to change anything here.

- **Data series are in** section

  Here, you can specify whether the selected data source area should be displayed by columns or by rows in the chart. For more information, see Changing the arrangement of data series.

- **Data source area contains** section

  Here, you can specify the types of data which the data source area contains.

  Tip: If the top row of the data source area contains headings for the data listed below, enable the option **Names**. PlanMaker will automatically label the data series in the chart with the corresponding headings.

- **Include hidden cells**

  If the data source area contains cells that have been hidden (see Showing and hiding rows/columns)), they are usually *not* displayed in the chart. Enable this option if you want hidden cells to be displayed as well.

Chart properties, Series tab

Data series are the most important chart elements. They represent the data to be evaluated – in the form of columns, bars, lines, etc., depending on the chart type selected.

The **Series** tab allows you to make settings for each data series of a chart.

In the **Series** list, first select the series whose settings you want to change. Then make the desired settings:

- **Series**

  The **Series** list contains a list of all data series currently included in the chart.

  You can use the arrow buttons to the right of the list to change the order of the data series.

  Use the **Add** and **Delete** buttons to add or remove data series.

- **Data sources** section

  Here, you can specify which data should represent the selected data series.

  The fields in this section are filled automatically (using the cell range you selected before inserting the chart). You usually don't have to change them.

  The following fields can be found here:

Name: Here, you can specify the name of the data series. This may be a cell reference to a cell that contains the name or simply a fixed text. This field can also be left empty, whereupon the data series receives an automatically generated name ("Series 1", "Series 2", etc.).

Y values: Here, you can determine the cell range from which the data series should obtain the Y values for the data points.

Explanation: For most chart types, the Y values are the values that should be displayed in the chart. For example, the Y values determine the height of the columns in a column chart.

X values: This is similar to to the option **Y values**, whereby you can determine from which cell range the X values for the data series should originate.

Explanation: For most chart types, the X values are irrelevant and are set to "Auto" (i.e. automatic) because they are only used to label the category axis (X axis). XY scatter charts and bubble charts are the exception, however, as the X and Y values determine the coordinates of the data points in these charts.

Bubble sizes: For bubble charts, there is a third parameter in addition to X and Y values: the size of the bubbles for each data point. To do so, enter a cell reference to the cells that contain these values. If this field is left empty, all bubbles will have the same size.

- **Display this series as** section

  Here, you can specify how the selected data series is displayed in the chart. If, for example, you have selected a column chart as the chart type, but want a particular series to appear as a line chart, you select the option **Lines** for this data series.

- **Use secondary axis for this series**

  If you enable this option, a secondary value axis (Y axis) will be added to the chart for this data series. Secondary value axes can have a different scale to primary value axes.

  Explanation: It is useful to represent specific data series on a secondary axis if these data series require an axis with a completely different scale because they contain significantly larger values than the other data series.

Chart properties, Elements tab

On the **Elements** tab, you can show/hide and label the various chart elements:

- **Chart title**

  Here, you can enter a title for the chart. The title will then be displayed above the chart.

  Tip: The icon means that you can also use a link to cell content as text. Click in the corresponding cell or type the cell address in the input field.

- **Primary axes** section

  Here, you can make settings for the primary axes:

  You can use the checkbox in front of each axis to enable/disable the display of this axis.

  To the right of the checkbox, you can give the axis a name. This will then be displayed below/next to the axis as an *axis title*.

  The **Major grid** and **Minor grid** options determine whether gridlines should be displayed in the background of the chart. These grids make it easier to read the values. For more information, see Gridlines.

- **Secondary axes** section

  If the chart has secondary axes, you can configure them here, similar to the primary axes (see above).

- **Legend**

  Here, you can specify the position of the legend. A legend is a small box that shows which data series are displayed in which color/pattern. For more information, see Legend.

Chart properties, 3D view tab

Note: This tab is only available for three-dimensional chart types.

On the **3D view** tab, you can change the 3D effect for the chart (for three-dimensional chart types).

- **Rotation angle** and **Elevation angle**

  Here, you can set the viewing angle (in degrees) from which the viewer sees the chart. **Rotation angle** rotates the chart around the vertical axis, while **Elevation angle** rotates the chart around the horizontal axis.

- **Perspective**

  If this option is enabled, the chart will be distorted in perspective. You can also specify the amount of distortion (from 0 to 100%).

- **Height** and **Depth**

  Here, you can change the height and depth of the chart (as a percentage of its original size).

Chart properties, Radar tab

Note: This tab is only available for radar charts.

On the **Radar** tab, you can make additional settings for displaying the chart (for radar charts).

- **Starting angle**

  This option allows you to rotate the chart by the specified angle.

- **Orientation**

  This option specifies whether the values should be arranged clockwise or counterclockwise.

- **Round radar chart**

  If this option is enabled, pie segments are drawn between the axes instead of lines.

- **Polar coordinates**

  If this option is enabled, polar coordinates are used instead of Cartesian coordinates. The polar coordinates are only available if **Round radar chart** is enabled.

  If **Angle between axes** is set to *x*, an axis will be plotted every *x* degrees.

  If **Angle between axis descriptions** is set to *x*, an axis label will be plotted every *x* degrees.

- **Limit plot area to radar**

  If this option is enabled, only the area *inside* the radar will be filled. If it is disabled, the entire rectangle surrounding the radar will also be filled.

Updating charts

A chart always retrieves the values to be displayed from the cells that were selected when the chart was created. If values in these cells change, the chart is *automatically* updated and immediately shows the changed values.

Disabling automatic update

If desired, you can disable this automatic updating. To do so, choose the ribbon command **File** | group **File management** | **Properties**, switch to the **Calculate** tab and disable the option **Update automatically** in the **Charts** section.

If this option is disabled, the charts in the current document will only be updated when you initiate this manually via the ribbon command **Formula** | group **Update** | **Update data** | **Update charts**.

Moving charts to another worksheet

When you create a new chart, it will always be inserted as an object in the current worksheet. However, you can change the position of a chart at any time:

To move a chart within the current worksheet, simply click on it and drag it to the desired position with the mouse:

If, on the other hand, you want to move the chart to a *different* worksheet, proceed as follows:

1. Click to select the relevant chart.
2. Choose the command **Chart location** on the contextual ribbon tab **Chart** | group **Position**.
3. In the dialog box that opens, make the desired choice (see below) and confirm with **OK**.

The chart is moved accordingly.

Available options:

- **As new sheet**

  If you select this option, the chart will be moved to its own *chart sheet*. To do so, PlanMaker removes the chart from the current sheet, creates a new chart sheet for it and inserts the chart into it.

  A "chart sheet" is a special type of worksheet that only contains the full-sized chart and does not contain any worksheet cells. It also only provides commands for editing charts.

- **As object in ...**

  If, on the other hand, you select this option, the chart will appear as usual as an object in the selected worksheet.

Saving charts as pictures

You can save an image of a chart as a picture file at any time. This makes it easier to share the chart with others or to insert it into a web page or any other document.

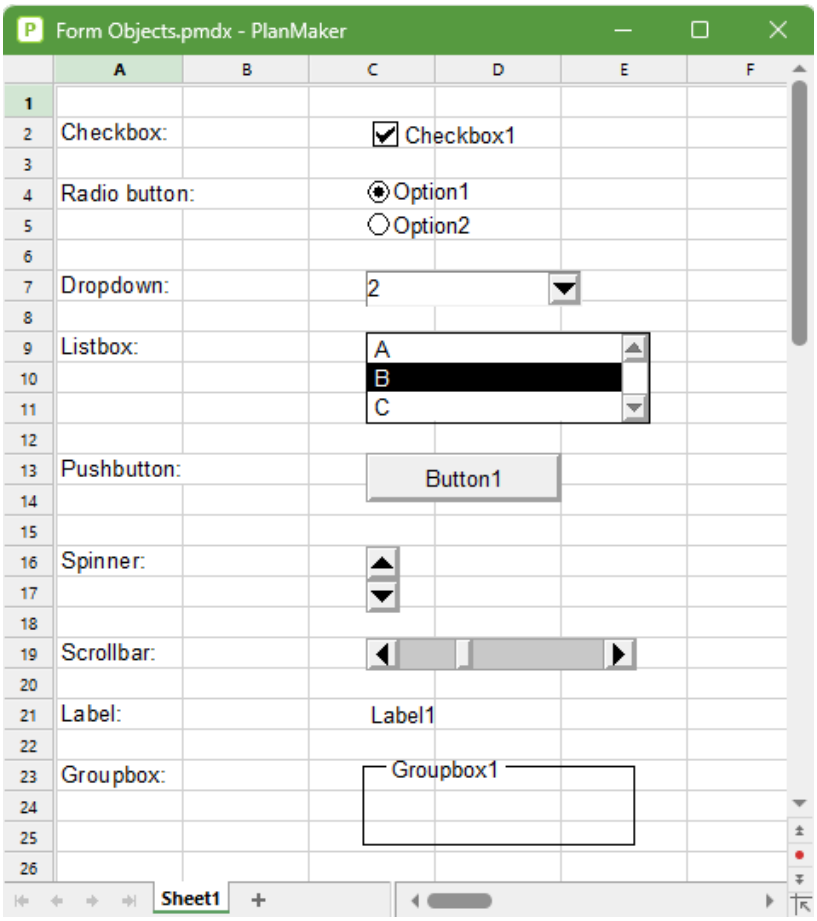
To do so, proceed as follows:

1. Click to select the desired chart.
2. Choose the ribbon command **Save chart as a picture** on the contextual ribbon tab **Chart** | group **Export**.
3. In the **Save as type** list, select the desired picture format.
   Tip: The *PNG format* is always a good choice here since it offers good and completely lossless compression.
4. Enter the file name under which you want to save the picture and confirm with **OK**.
5. Another dialog box appears. Select the desired resolution for the picture or manually enter your own values for the width and height. Then confirm with **OK**.

PlanMaker now creates a corresponding picture file with an image of the chart.

Forms

You can insert *form objects* into your tables to create forms.



The following types of form objects are available:

- **Checkboxes** for enabling
- **Radio buttons** for selecting from among multiple alternatives
- **Dropdowns** for selecting from an expandable list
- **Listboxes** for selecting from a list
- **Pushbuttons** for clicking on
- **Spinners** for increasing/decreasing values via a mouse click
- **Scrollbars** for increasing/decreasing values via a mouse click
- **Labels** for static descriptions
- **Groupboxes** for visually grouping related items

Form objects always have a **Result cell**. This is the cell which holds the value that the form object returns. You can specify which cell this is for each individual form object.

If, for example, you insert a listbox with several entries in it, 1 is displayed in the result cell when you click on the first entry, 2 when you click on the second entry, etc.

This chapter contains detailed information on working with forms. It consists of the following sections:

- **Working with form objects**

  The first section covers general information on working with form objects. You will learn how to insert, edit and evaluate form objects.

- **Form objects in detail**

  The second section contains detailed information on each individual type of form object.

- **Options of the contextual ribbon tab "Object"**

  The last section explains the commands of the contextual ribbon tab **Objects**. This always appears automatically when you select a form object.

Working with form objects

In the following sections, you will find general information on working with form objects:

- Inserting form objects
- Editing form objects
- Using and evaluating form objects
- Form objects and Excel macros and scripts

In Form objects in detail, we will then cover the individual types of form objects in detail.

Inserting form objects

Inserting form objects is not much different from inserting other types of objects. For more information, see Objects.

To insert a form object, proceed as follows:

1. Choose the ribbon command **Insert | Form object** (in the group **Objects**).
2. Select the desired type of object from the dropdown menu.

The object is now inserted.

At the same time, the program automatically displays a contextual ribbon tab **Object** where you can apply general settings for objects (see Options of the contextual ribbon tab "Object").

For more information on the individual types of form objects, see Form objects in detail.

Editing form objects

Editing form objects is not much different from working with other types of objects. For more information, see Objects.

There is, however, one important difference:

> ***Important:*** Form objects *cannot* be selected by left-clicking on them with the mouse. To select a form object, *right-click* on it.
>
> Alternatively, you can also use the ribbon command **View | Object mode** to switch to *Object mode* in which form objects can also be selected by left-clicking on them.

Once you have selected a form object, you can edit it like any other type of object – for example, move it with the mouse, resize it, etc.

By right-clicking with the mouse on the form object, you can modify its properties with the entry **Form object: Properties** in the context menu.

For more information on the properties of form objects, see Form objects in detail.

Using and evaluating form objects

Using form objects is very much like using control elements in dialog boxes. For example, check marks can by set or removed by clicking on them. A listbox entry can be selected by clicking on it, etc.

Evaluating form objects

Form objects are evaluated via the *result cell* of the object. This is the cell which holds the value that the form object returns.

The location of the result cell can be determined individually for each form object: To do so, select the form object, right-click to open the context menu and select **Form object: Properties**. In the dialog box, switch to the **Form object** tab and enter the desired cell address in the **Result cell** field.

For example: You have assigned the result cell D4 to a list. If you click on the first entry in the list, 1 will be displayed in D4. If you click on the fifth entry, 5 will be displayed, etc.

The relationship between a form object and its result cell is also two-way: If you manually enter 3 in the result cell, the third entry will be automatically selected in the list.

Form objects and Excel macros and scripts

Microsoft Excel files can contain macros and VBA scripts that can be applied to form objects. This is not possible in PlanMaker, but:

Important: If you open an Excel file in PlanMaker and it has macros or scripts, you won't be able to run them; however, they will remain in the file. Thus, if you open and edit such an Excel file in PlanMaker and then save it, the macros and scripts will *not* be lost.

Form objects in detail

In this section, the individual types of form objects are described in detail.

The following objects are covered:

- Checkboxes
- Radio buttons
- Dropdown lists
- Listboxes
- Pushbuttons
- Spinners
- Scrollbars
- Labels and groupboxes

Checkboxes

You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Checkbox** to insert a *checkbox*.

Checkboxes can be used in Forms for Yes/No entries. If the box is checked, it stands for **Yes**; if it is unchecked, it means **No**.

Using checkboxes

Simply click on the box to give it a check mark – in other words: to check it. If you click on it again, the check mark is removed.

Changing the properties of checkboxes

To change the properties of a checkbox, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the checkbox itself:

- **Text**

  Here, you can enter the text that is to be displayed to the right of the checkbox.

- **Value** section

  Here, you can specify whether or not the checkbox should be checked. This corresponds to checking the checkbox directly in the document.

- **Result cell**

  Here, you specify to which cell the result of the checkbox should be returned.

  One of the following values will appear in this cell:

  TRUE if the checkbox is enabled.

  FALSE if the checkbox is disabled.

  The error value #N/A if the state of the checkbox is undetermined.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

- **3D effect**

  If you enable this option, the checkbox is drawn with a 3D effect.

Radio buttons

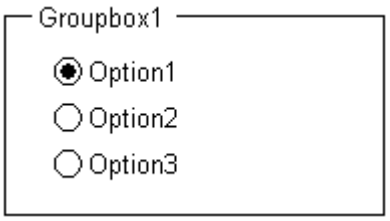
You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Radio button** to insert a *radio button*.

Radio buttons can be used in forms to select one of several options.

Using groupboxes to combine radio buttons

Important: Radio buttons must always be used in groups of at least two radio buttons.

To ensure that PlanMaker knows which radio buttons belong to a group, enclose them within a groupbox.



First, insert the related radio buttons into the document (for example, one below the other) and then use the ribbon command **Insert** | group **Objects** | **Form object** | **Groupbox** to drag a groupbox around them.

Using radio buttons

Click on one of the related radio buttons to select it. Only one radio button within a group can be selected at a time.

Changing the properties of radio buttons

To change the properties of a radio button, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change some general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the checkbox itself:

- **Text**

  Here, you can enter the text that is to be displayed in the radio button.

- **Value** section

  Here, you can specify whether or not the radio button should be checked. This corresponds to checking radio button directly in the document.

- **Result cell**

  Here, you specify to which cell the result of the radio button should be returned.

  As mentioned previously, radio buttons must always be used in groups of at least two. The result cell then displays which of the radio buttons in the group is selected. When the first is selected, 1 is displayed, when the second is selected, 2 is displayed, etc.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

- **3D effect**

  If you enable this option, the radio button is drawn with a 3D effect.

Dropdown lists

You can use the ribbon command **Insert |** group **Objects | Form object | Dropdown** to insert a *dropdown list*.

You encounter dropdown lists in many dialog boxes. When you expand it, a list of available options is shown. Such lists are ideal for forms because they reduce the required amount of typing when filling in forms and prevent typing errors.

Using dropdown lists

Click on the arrow to the right of the list to open the list. You can now select an entry by clicking on it.

Changing the properties of dropdown lists

To change the properties of a dropdown list, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the dropdown list itself:

- **Lines (max.)**

  Here, you can specify the maximum number of entries that the list should display when it is expanded.

- **List area**

  Here, you specify the cell range that contains the entries to be displayed in the dropdown list.

  If, for example, you fill the cells F5 to F7 with the values "dog", "cat" and "mouse" and enter F5:F7 in the list area, those three values will appear in the list.

- **Result cell**

  Here, you specify to which cell the result of the dropdown list should be returned.

  The result cell displays which of the list entries is selected. When the first entry is selected, 1 appears. When the second entry is selected, 2 appears, etc.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

- **3D effect**

  If you enable this option, the dropdown list is drawn with a 3D effect.

Listboxes

You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Listbox** to insert a *listbox*.

You encounter listboxes in many dialog boxes. They display several entries from which the user can choose by clicking on them. Such lists are ideal for forms because they reduce the required amount of typing when filling in forms and prevent typing errors.

Changing the properties of listboxes

To change the properties of a listbox, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change some general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the list itself:

- **Selection type**

  You should always use the option **Single** here.

  The other options **Multi** and **Extended** allow multiple items to be selected in the list, however, no result is then output in the result cell. They are only provided for compatibility with Excel.

- **List area**

  Here, you specify the cell range that contains the entries to be displayed in the list.

  If, for example, you fill the cells F5 to F7 with the values "dog", "cat" and "mouse" and enter F5:F7 in the list area, those three values will appear in the list.

- **Result cell**

  Here, you specify to which cell the result of the list should be returned.

  The result cell displays which of the list entries is selected. When the first entry is selected, 1 appears. When the second entry is selected, 2 appears, etc.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

- **3D effect**

  If you enable this option, the list will be drawn with a 3D effect.

Pushbuttons

You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Pushbutton** to insert a *pushbutton*.

Note: In PlanMaker, pushbuttons *cannot* be used to run macros or VBA scripts; they are merely provided for compatibility with Excel.

Changing the properties of pushbuttons

To change the properties of a pushbutton, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change some general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the pushbutton itself:

- **Text**

  Here, you can specify the text that should appear on the button.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

Spinners

You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Spinner** to insert a *spinner*.

Spinners allow you to increment or decrement a value via a mouse click. Clicking on the up arrow increments the value in the result cell, while clicking on the down arrow decrements it.

Changing the properties of spinners

To change the properties of a spinner, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change some general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the spinner itself:

- **Parameters** section

  Here, you can specify the following parameters:

  Current value: The current value (corresponding to the value in the result cell)

  Minimum value: The lowest value allowed

  Maximum value: The highest value allowed

  Incremental change: The value that will be added/subtracted to the current value when you click on one of the two arrows

- **Result cell**

  Here, you specify to which cell the resulting value should be returned.

- **3D effect**

  If you enable this option, the spinner is drawn with a 3D effect.

Scrollbars

You can use the ribbon command **Insert** | group **Objects** | **Form object** | **Scrollbar** to insert a *scrollbar*.

Scrollbars can be used to increase or decrease values via a mouse click. Clicking on the upward pointing arrow increments the value by the specified incremental value, while clicking on the downward pointing arrow decrements it.

Furthermore, you can make larger changes by moving the slider on the scrollbar with the mouse.

Changing the properties of scrollbars

To change the properties of a scrollbar, select it (for example, by *right-clicking* on it) and select **Form object: Properties** from the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change some general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the scrollbar itself:

- **Parameters** section

  Here, you can specify the following parameters:

  Current value: The current value (corresponding to the value in the result cell)

  Minimum value: The lowest value allowed

  Maximum value: The highest value allowed

  Incremental change: The value that will be added/subtracted to the current value when you click on one of the two arrows in the scrollbar

  Page change: The value that will be added/subtracted when you click on somewhere between the scrollbar's slider and one of the arrow buttons

- **Result cell**

  Here, you specify to which cell the resulting value should be returned.

- **3D effect**

  If you enable this option, the scrollbar is drawn with a 3D effect.

Labels and groupboxes

In addition to the form objects described above, there are two types of form objects that *cannot* be filled in and are only intended for applying labels:

- **Labels**

  Labels: Here, you can enter any labels that should be displayed in the form.

- **Groupboxes**

  Groupboxes are rectangles that can be labeled in the top left corner. They can be used for the visual grouping of related parts of a form.

To insert a label or a groupbox, choose the ribbon command **Insert** | group **Objects** | **Form object** | **Label** or **Group box**.

The corresponding form object is then inserted. Now move it with the mouse to the desired position. You can also resize it by dragging one of the round handles that surround the object.

Changing the properties of labels and groupboxes

To change the properties of a label or group field, select it (for example, by *right-clicking* on it) and choose **Form object: Properties** in the context menu.

Alternatively: On the contextual ribbon tab **Object**, click on the group arrow ◢ in the bottom right corner of any command group.

The program displays a dialog box with the following options:

Format tab

On this tab, you can change the size and positioning of the object. For more information, see Object properties, Format tab.

Properties tab

On this tab, you can change general settings. For more information, see Object properties, Properties tab.

Form object tab

On this tab, you can make settings for the text that you want the label or group field to display:

- **Text** section

  Here, you can enter the text to be displayed.

- **Appearance** section

  Here, you can specify the character format (font, size, color, etc.) for the text.

- **3D effect**

  This option is only available for groupboxes. If you enable this option, the groupbox is drawn with a 3D effect.

Options of the contextual ribbon tab "Object"

If you click to select a form object, the contextual ribbon tab **Object** appears automatically.

However, you can only apply some general options for objects here:

- **New object:** This option allows you to insert a new object into the document and different object types are available.
- **AutoShape:** This option allows you to insert a new AutoShape into the document.
- **Lines:** This option allows you to insert simple lines, curves and also connectors (which you can "dock" to the text frame) into the document.
- **Bring to front, Send to back:** These options allow you to arrange the order of overlapping object frames. For more information, see Changing the order of objects.
- **Align objects:** This option allows you to align or evenly distribute two or more object frames. For more information, see Aligning and distributing objects.
- **Group:** This option allows you to combine several selected objects into one unit, which can then be treated like a single object. For more information, see Grouping objects.
- **Duplicate:** This option creates an immediate copy of the selected object.
- **Width, Height**: These options change the size of the object. For more information, see Changing position and size of objects.

Language tools

PlanMaker has a powerful spell check. This enables you to have the spelling of text checked and corrected. Automatic hyphenation is also available.

This chapter provides information on working with this tool. It consists of the following sections:

- **Setting the language**

  You can choose the ribbon command **File | Options** (**Language** tab) to set the *language* for the spell check and hyphenation.

- **Spell check**

  The *spell check* checks a document's text for typos and suggests corrections in the event of errors.

- **Hyphenation**

  Automatic *hyphenation* hyphenates long words at the end of lines. By default, it is active only in text frames, but you can also activate hyphenation for multi-line text in worksheet cells.

- **Auto correction**

  With *Auto correction*, you can have PlanMaker automatically correct typical careless mistakes when entering text – for example, if you often type "yuo" instead of "you". In addition, you will find many other useful auto correct functions here.

- **SmartText**

  You can use *SmartText* to define abbreviations for frequently used phrases – for example, "sd" for "sales department".

- **Research**

  Only available in **SoftMaker Office Professional** and **NX Universal**: Look up terms on online research platforms (Wikipedia, Oxford dictionary, Cambridge dictionary, etc.) directly from the document.

- **Read aloud**

  Only included in **SoftMaker Office Professional** and **NX Universal**: Have the content of your PlanMaker document read aloud to you.

Setting the language

If you have installed more than one language, you can change the language for the spell check and hyphenation at any time.

To do so, proceed as follows:

1. Choose the ribbon command **File | Options** .

   Alternatively: Choose the ribbon command **Review | Spell check** | **Settings**.

2. Switch to the **Language** tab.
3. Select the desired language from the list "Language".

Note: Please note that there are *two* entries in this list for German:

- **"German (Germany)"** stands for the *new* spelling (according to the 1996 German spelling reform in its last revision in 2006).
- **"German (Germany, old)"**, on the other hand, stands for the *old* spelling.

The same applies to "German (Austria)" and "German (Switzerland)".

Spell check

The *spell check* checks a document's text for typos and suggests corrections in the event of errors.

This section describes the different spell check tools in detail. The information covers the following topics:

- **Manual spell check**

  The *manual spell check* lets you check and correct possible spelling errors of the current worksheet.

- **Checking the spelling as you type**

  The *Check spelling as you type* function immediately checks each word you type. When a typing error is detected, the program displays a dialog box that lets you correct the word.

- **Editing user dictionaries**

  When you teach the spell check new words, they are added to your *user dictionary*. You can edit this dictionary at any time, for example, to delete incorrect entries.

For more information on these topics in the above order, see the following pages.

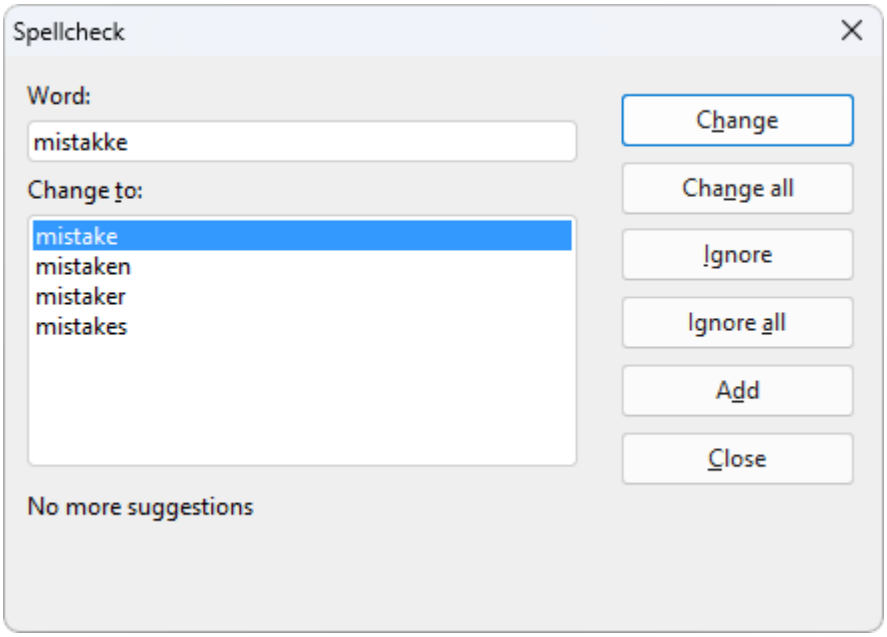
Note: The spell check relies on the dictionaries having been installed and enabled. For more information, see Installing dictionaries.

Manual spell check

Note: Cells containing calculations are skipped by the spell check.

You choose the ribbon command **Review | Spell check** to start the subsequent spell check. (Tip: You can also choose this command with the **F7** key.)

This checks the current worksheet word for word for spelling errors. If the spell check encounters an unfamiliar word, it stops and displays the word in a dialog box.



You can then choose to correct the word, add it to the dictionary or simply ignore the supposed error. The list under **Change to** will also contain some suggestions for the correct spelling of the word (if any are found).

Use the buttons in the dialog box to specify what to do with the unfamiliar word:

Button	Explanation
Change	This option lets you correct the misspelled word. Before you use this button, type the correct spelling in the **Change to** input field or select one of the suggested words from the list.
Change all	This option works like the **Change** button, but it changes *all* further instances of the word in the entire document.
Ignore	This option instructs PlanMaker to ignore this spelling error and continue with the spell check.
Ignore all	This option instructs PlanMaker to ignore *all* further instances of this word. Note: PlanMaker only remembers the list of ignored words *temporarily*. When you exit PlanMaker, it forgets it. If you want PlanMaker to learn words *permanently*, use the command **Add** instead.
Add	This option instructs PlanMaker to add the word to the user dictionary and thus increase its vocabulary. Use this option for correctly spelled words that PlanMaker does not yet know. PlanMaker remembers these words permanently, even after a restart.

If you want to exit the spell check before it reaches the end of the worksheet, click on **Close**.

Checking the spelling as you type

If the option **Check spelling as you type** is enabled, the spelling is checked as the text is typed, and the spell check dialog box will open automatically when an unknown word is typed.

Note: If you enter a calculation into a cell, its spelling will *not* be checked. This would not make sense with calculation formulas.

To enable this on-the-fly spell checking, choose the command **File | Options** , switch to the **Language** tab and enable the option **Check spelling as you type**.

Every time you type a word, PlanMaker now quickly looks up that word in its dictionaries. As long as you type words that the spell check knows, nothing visible happens. If you type an unknown word, however, the spell check displays a dialog box.

This dialog box is similar to the dialog box for the manual spell check presented in the previous section. You can read there about how to use this dialog box.

You will notice that there is also a **SmartText** button in the dialog box, which can be used to create a SmartText entry from the word. You will learn more about this feature in the SmartText section.

Editing user dictionaries

When you teach the spell check new words, they are added to your *user dictionary*. You can edit this dictionary at any time, for example, to delete incorrect entries.

To do so, click on the small arrow of the icon on the ribbon tab **Review | Spell check** and select **Edit user dictionaries** from the dropdown menu. The program displays a dialog box with the following buttons and options:

- **Language**

  There is a separate user dictionary for each language. You can select the user dictionary to be edited from the dropdown list **Language**.

  Note: Note that there are *two* entries for each German in this list: "German (Germany)" stands for the *new* spelling and "German (Germany, old)" for the *old* spelling. The same applies to "German (Austria)" and "German (Switzerland)".

- **Entries**

  This list contains all words that have been added to your user dictionary (for the selected language).

- **"Close" button**

  This option closes the dialog box.

- **"Add" button**

  This button enables you to manually add a word to your user dictionary. To do so, type the desired word into the dialog box that appears after you press this button.

  Words added in this way are treated exactly the same as words that were added using the **Add** button in the spell check.

- **"Change" button**

  Use this button to edit the word that is currently selected in the list. This is useful for misspelled words that you have added to your user dictionary accidentally. By clicking on the button, you open a dialog box where you can correct the spelling of the word.

- **"Delete" button**

  This button deletes the word currently selected in the list from your user dictionary.

  The spell check will therefore regard this word as misspelled again in the future.

Hyphenation

Automatic *hyphenation* hyphenates long words at the end of lines. This works fully automatically as you type.

However, hyphenation is only available in the following cases:

- **Hyphenation in text frames**

  In text frames, automatic hyphenation is enabled by default.

- **Hyphenation in table cells**

  In table cells, automatic hyphenation is *not* enabled by default. For cells that contain multi-line text, however, you can enable this if desired.

For more information, see the following pages.

Important: Of course, hyphenation only provides correct results if you have set the correct language in the program settings. To set the language, choose the command **File | Options**, switch to the **Language** tab and select the language from the dropdown list "Language".

Hyphenation in text frames

Whenever you enter text in a text frame, PlanMaker will automatically hyphenate it. This is performed fully automatically in the background as you type.

All you have to do is to make sure that the correct *language* is set in PlanMaker's settings. For more information, see Setting the language.

Setting the hyphenation frequency

If desired, you can specify how often you want the hyphenation function to separate words – or disable hyphenation completely. This setting can be set separately for each paragraph in a text frame.

To do so, select the relevant paragraphs in the text frame, click on the group arrow on the ribbon tab **Home** | group **Alignment** and select the desired setting for **Hyphenation** in the dialog box.

Alternatively, you can open the dialog box by right-clicking in the text frame and selecting the entry **Paragraph** from the context menu.

The following options are available:

Setting	Result
None	No hyphenation is performed.
Always	Any amount of consecutive hyphenation is attempted. This is the default setting.

Every 2 lines	Hyphenation is attempted only in every second line.
Every 3 lines	Hyphenation is attempted only in every third line.

If you select **None**, you completely disable hyphenation for the selected paragraphs, whereas if you select **Always**, the program tries to hyphenate any line of a paragraph.

When do you use the other two options? It is not always a good idea to hyphenate at every point possible. For text in narrow columns, for example, this may lead to words being split at the end of almost every line and thus to a reduction in the document's legibility. Thus, you may want to use the option **Every 2 lines** or **Every 3 lines** to instruct PlanMaker to attempt hyphenation only in every second or third line.

Hyphenation in table cells

Automatic hyphenation is not enabled by default in worksheet cells, but it can be enabled by the user. Of course, hyphenation is then only performed in cells that contain multi-line text.

In order to obtain hyphenation in cells, two steps are required:

Step 1: Enabling hyphenation in cells

In order for hyphenation to be performed at all in cells, you must first enable hyphenation for cells, as it is disabled by default. To enable it, proceed as follows:

1. Choose the ribbon command **File** | group **File management** | **Properties** .
2. Switch to the **Options** tab.
3. Enable the option **Hyphenation**.

From now on, automatic hyphenation will also be performed in worksheet cells.

Note: This is a *document* option, thus you can enable or disable it individually for each document.

Step 2: Enabling the option Wrap text for cells

Hyphenations will only be performed in worksheet cells that have the option **Wrap text** enabled. This ensures that long text will be automatically wrapped to multiple lines if it doesn't fit into a single line.

To enable the option **Wrap text**, proceed as follows:

1. Select the relevant cells.
2. Click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Number**.

   Alternatively: Right-click to open the context menu and select the entry **Cell**.
3. Switch to the **Alignment** tab.
4. Enable the option **Wrap text**.

Automatic hyphenation will now be performed in these cells (if they contain multi-line text) where required.

Auto correction

You're bound to be familiar with making the same typos over and over again when writing. For example, you want to write "you", but the text ends up as "yuo", which was not your intention. In such cases, PlanMaker intervenes via **Auto correction** and corrects such common typos automatically.

The ribbon command **Review** | group **Spelling** | **Auto correction** opens a dialog box in which you can specify the settings for automatic correction.

Apart from the ability to avoid these accidental typos as in the example above, there are other useful auto correction functions in the dialog box:

- Capitalize first letter of sentences
- Correct capitalization of first TWo LEtters
- Special replacements: For example, automatically create a smiley from the combination :-)
- Automatically convert quotation marks to the typographically correct characters when typing
- Format URLs as hyperlinks
- Add superscript to ordinal numbers

Tip 1: Automatic correction is triggered as soon as a term is terminated with the space bar, the Enter key ↵ or with a punctuation mark.

Tip 2: If you want to undo an unwanted auto correction in your document, use the **Undo** command or press the key combination **Ctrl+Z** immediately after the auto correction.

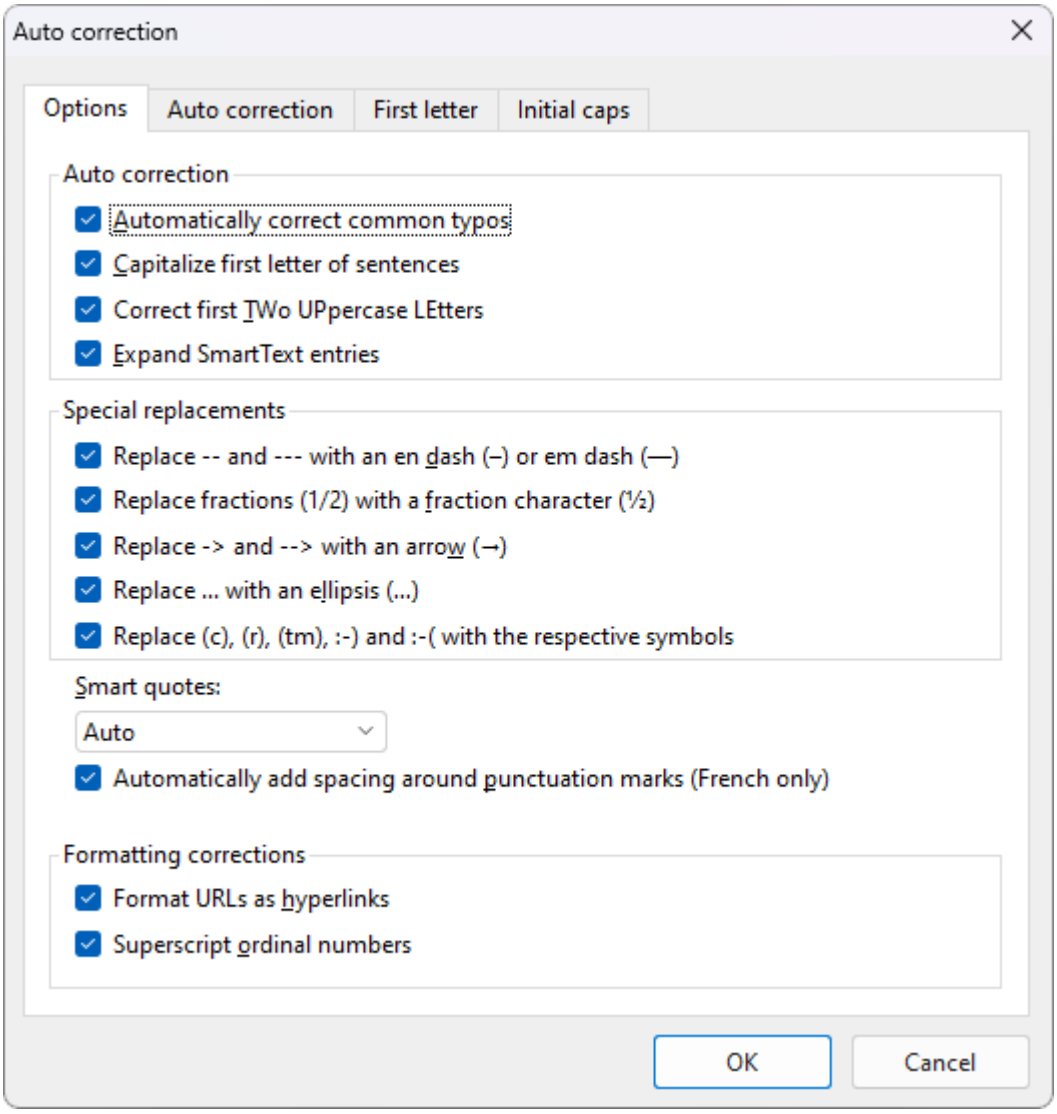
The following pages contain detailed explanations about all auto correction functions.

Options tab

You can adjust the auto correction settings according to your needs.

Note: Auto correction is a global setting. Changes you make here will thus be applied to every document.

To do so, choose the ribbon command **Review** | group **Spelling** | **Auto correction**, and the following dialog box will appear:



On the **Options** tab, you will find the following auto correction functions:

"Auto correction" section

- **Automatically correct common typos**

  This option ensures that typical typing errors are corrected. If, for example, you type "thier", PlanMaker automatically corrects this word to "their".

  A list containing common typos is pre-installed for some languages. You will find this list on the **Auto correction** tab, where you can delete existing pairs of terms or add new ones. For more information, see Auto correction tab.

- **Capitalize first letter of sentences**

  Note: Enabling this option will only affect *text frames* and not *table cells*.

If this option is enabled, PlanMaker automatically capitalizes the first letter of every sentence you type – even if you accidentally enter a lowercase letter.

If, for example, you typed the sentence "this is a new sentence.", it would be corrected automatically to "This is a new sentence.".

Exception list: For certain terms, however, it may be undesirable for this automatic correction to occur (for example, after an abbreviation such as "max."). Switch to the **First letter** tab, where you can add terms that should be excluded from this correction. For more information, see First letter tab.

- **Correct first TWo UPpercase LEtters**

  If this option is enabled, PlanMaker automatically corrects the case of words whose first two letters have been capitalized inadvertently.

  If, for example, you typed the word "HEnry", it would be corrected automatically to "Henry".

  Exception list: If, however, you want to exclude certain expressions from this correction (for example, "BSc"), then switch to the **Initial caps** tab. Here, you can add terms where two capital letters at the beginning should not be corrected. For more information, see Initial Caps tab.

- **Expand SmartText entries**

  As described in SmartText, you can create SmartText entries with PlanMaker (for example, "sd" for "sales department").

  If this option is enabled, your SmartText entries can be directly expanded as the text is entered. To do so, simply type the abbreviation for the SmartText (for example, "sd") and then a space, Enter↵ or a punctuation mark, and PlanMaker will automatically replace the abbreviation with the SmartText content.

  If this option is disabled, however, SmartText entries can only be inserted via the dialog box of the ribbon command **Insert** | group **Text** | **SmartText**.

  Tip: The option "Expand SmartText entries" can also be enabled or disabled in this dialog box.

"Special replacements" section

PlanMaker can trigger some special characters automatically as soon as you type a certain combination of characters in your text. For example, three dots in a row ("...") are converted to ellipsis points. Enable or disable the desired options in this section.

Tip: On the ribbon tab **Insert** | group **Text** you can use the commands **Character** and **Symbol** to find other common characters and symbols that you can insert manually into your text.

Smart quotes

Note: Enabling this option will only affect *text frames* and not *table cells*.

PlanMaker can automatically convert double quotes (") and single quotes (') as they are typed to ensure that they are typographically correct for the selected language. Here, set the desired language or select "None" if you want PlanMaker to leave your quotes marks unchanged.

If, for example, you type "Text" or 'Text', the relevant setting will produce the following results:

Setting	"Text"	'Text'
Auto*		

None (no change)	"Text"	'Text'
German	„Text“	‚Text‘
Swiss German	»Text«	›Text‹
English	“Text”	‘Text’
French**	«Text»	‹Text›
Swedish	”Text”	’Text’
Polish	„Text”	‚Text’

* A special position is assumed by the option **Auto**: If you select this option, PlanMaker automatically determines the language and converts the quotes accordingly.

** For the *single quote* in the *French* setting, please note the following: the closing quote is first executed as an apostrophe ('). If you press the single quote on the keyboard once again, it will be converted into the single guillemet (›).

Tip: You can use the command **Undo** (or the key combination **Ctrl+Z**) to convert smart quotes into "neutral" quotes immediately after typing. The simple smart quote then becomes an apostrophe, while the double smart quote becomes a normal double quote.

Automatically add spacing around punctuation marks (French only)

Note: Enabling this option will only affect *text frames* and not *table cells*.

Enable the option to automatically add spaces both before punctuation marks : ; ! ? as well as between quotes « text » ‹ text ›. This option is only effective if *French* is the assigned language in the following setting:

- In the dialog box of the command **File | Options**, **Language** tab: *French*

"Formatting corrections" section

- **Format URLs as hyperlinks**

  If this option is enabled, URLs (web page addresses) are automatically formatted as hyperlinks immediately after entering them.

  If, for example, you type "www.softmaker.de" in the text followed by a space, it will be converted automatically into a clickable hyperlink and displayed accordingly (usually blue and underlined).

- **Superscript ordinal numbers**

  This option only supports the following languages: English, French, Italian, Portuguese and Spanish.

  In these languages, superscripts are usually added to abbreviated *ordinal numbers* – for example 1st, 2nd, 3rd in the English language.

  If this option is enabled, superscripts are added automatically whenever you enter such a number. If, for example, you type "2nd" in an English text and then press the space bar, it will be automatically reformatted as "2nd" – the "nd" will be converted into a superscript.

Note: This only works if one of the supported languages is set (thus English in our example). For information on how to change this setting, see Setting the language.

Auto correction tab

If the **Automatically correct common typos** function is enabled on the Options tab, common typos can be corrected automatically by PlanMaker.

On the **Auto correction** tab, you will find a list where you can enter typical typos and their corresponding correct spelling. An extensive list is already provided for many languages.

You can add new word pairs to this list, or edit or delete existing word pairs.

Note: The list is automatically linked to the language that is set as the default language (see Setting the language). However, you can also edit the lists for other languages independently at the top of the **Auto correction** tab.

Add new word pairs to auto correction

1. On the **Auto correction** tab, click in the **Replace:** field and enter the term here as you typically misspell it.
2. Then click in the **With:** field and enter the correct spelling for this term here.
3. Click on the **Add** button.
   Add further word pairs to this list in this way.

The newly added terms are now included in the auto correction list and the incorrect spelling will be automatically corrected with the right spelling in future.

Edit existing word pairs

1. On the **Auto correction** tab, click on the word pair that you want to change in the lower list.
2. In the **Replace:** and **With:** fields, adjust the incorrect and correct spelling to your liking:

   If you change something in the **Replace:** field (left side, for the wrong spelling), pressing the **Add** button will add the new word pair to the auto correction list. The original word pair also remains in the list.

   If you change something in the **With:** field (right side, for the correct spelling), pressing the **Replace** button will overwrite the original word pair in the auto correction list.

Delete word pairs

1. In the lower list on the **Auto correction** tab, click on the term that should no longer trigger an automatic correction.

2. Click on the **Delete** button.

The term is deleted and will not be detected by auto correction in future.

First letter tab

If the Auto correction function **Capitalize first letter of sentences** is enabled on the Options tab, PlanMaker automatically capitalizes the first letter at the beginning of a sentence – even if you have accidentally typed a lowercase letter.

Here, on the **First letter** tab, you can set exceptions to this rule:

This applies, for example, to abbreviations such as "amt.", "fig.", "p.m.", which end with a period and thus indicate the end of a sentence without actually being one. Thus, there is no desire here for PlanMaker to automatically capitalize the next word.

Enter the appropriate abbreviations in the exception list on the **First letter** tab. An extensive list is already provided for many languages.

You can add new entries to this list or delete existing entries.

Note: The list is automatically linked to the language that is set as the default language (see Setting the language). However, you can also edit the lists for other languages independently at the top of the **Auto correction** tab.

Initial Caps tab

If the auto correction function **Correct first Two UPpercase LEtters** is enabled on the Options tab, PlanMaker automatically corrects the case of words whose first two letters have been capitalized inadvertently.

Here, on the **Initial caps** tab, you can set exceptions to this rule:

This applies, for example, to terms such as "BSc", "DAdmin", "CDs", where it is not at all desirable for the combination of two capital letters at the beginning of the word to be corrected.

Enter the appropriate terms in the exception list on the **Initial caps** tab. An extensive list is already provided for many languages.

You can add new entries to this list or delete existing entries.

Note: The list is automatically linked to the language that is set as the default language (see Setting the language). However, you can also edit the lists for other languages independently at the top of the **Auto correction** tab.

SmartText

PlanMaker's *SmartText* function can save you a lot of work: You can create SmartText entries for frequently used phrases and then insert them instantly in the text.

For example, create a SmartText entry called "sd" with the content "sales department". You can now retrieve this SmartText entry at any time: To do so, simply type "sd" in the document and then press the space bar, the **Enter key**↵ or a punctuation mark. The abbreviation "sd" will be replaced immediately "sales department".

In this way, you can create your personal "computer steno" with PlanMaker and save a lot of time when typing.

In this section, you will learn about working with SmartText entries:

- **Creating SmartText entries**

  New SmartText entries can be created in a dialog-box-controlled manner with the ribbon command **Insert** | group **Text** | **SmartText** or via the spell check. Each SmartText entry must have a name (for example, "sd") and the desired content (for example, "sales department").

- **Inserting SmartText entries**

  If the option **Expand SmartText entries** is enabled in the program settings, SmartText entries can be inserted very easily: To do so, type the name of the SmartText entry and then press the space bar, **Enter key**↵ or a punctuation mark. The name will be immediately replaced by the content of the SmartText entry.

  Alternatively, entries can be inserted manually with the ribbon command **Insert** | group **Text** | **SmartText**.

- **Editing SmartText entries**

  Not only can you create new SmartText entries with the ribbon command **Insert** | group **Text** | **SmartText**, but you can also edit, rename or delete existing ones.

For more information, see the following pages.

Creating SmartText entries

You can choose to create SmartText entries a) in a dialog-box-controlled manner or b) using *Check spelling as you type*:

A) With the command Insert | SmartText

To create a SmartText entry with the name "sd" and the content "sales department", for example, proceed as follows:

1. Choose the ribbon command **Insert** | group **Text** | **SmartText** .
2. Click on the **New** button to create a new SmartText entry.
3. Give the SmartText entry a name ("sd" in our example).

   The SmartText entry can be retrieved later by its name.

4. Confirm with **OK**.
5. Now, type the text for the SmartText entry in the large input field on the right (in our example: "sales department").
6. Click on **Save** to save your new SmartText entry.
7. Exit the dialog box with **Close**.

The SmartText entry has now been created. You will find out how to insert it in the next section (Inserting SmartText entries).

B) Via the option "Check spelling as you type"

If you have enabled *Check spelling as you type*, you can also create SmartText entries as follows:

1. In the dialog box of the ribbon command **File | Options** (on the **Language** tab), make sure that the option **Check spelling as you type** is enabled.
2. Type the two letters "sd" and press the space bar.
3. The on-the-fly spell check displays a dialog box indicating that the word "sd" is unknown.
4. Click on the **SmartText** button.
5. In the open dialog box, type "sales department" into the large input field on the right.
6. Confirm with **OK**.

The result is the same: The SmartText entry "sd" has now been created.

For more information on the option *Check spelling as you type*, see Checking the spelling as you type.

Inserting SmartText entries

You can now insert the SmartText entry created in the previous section at any time.

This is very simple: Type the name of the SmartText entry in a cell or in a text frame – "sd" in our example – and then a space, a punctuation mark or the **Enter key**↵. PlanMaker will immediately replace "sd" with the content of the SmartText entry: "sales department".

Note: If this does not work, the option **Expand SmartText entries** is disabled. Choose the ribbon command **Review | Auto correction** and enable this option again in the dialog box (or in the dialog box of the command **Insert** | group **Text | SmartText**).

Alternatively, you can access the defined SmartText entries via the dialog box by choosing the ribbon command **Insert** | group **Text | SmartText**, selecting the desired SmartText entry on the left side of the dialog box, and then clicking on the **Insert** button.

Tip: Do not expand SmartText entries

If you do not want a defined SmartText entry to be inserted when you type in the text, then use the key combination **Alt+.** (period) in the place directly behind the SmartText entry. In this case, the specified addition is then suppressed.

Editing SmartText entries

By choosing the ribbon command **Insert** | group **Text** | **SmartText** , you can still edit the already created SmartText entries via the dialog box:

- **Creating a new SmartText entry**

  Click on the **New** button to create a new SmartText entry in the dialog box. For more information, see Creating SmartText entries.

- **Deleting a SmartText entry**

  To delete a SmartText entry, select it in the list **SmartText entries** on the left of the dialog box and then click on the **Delete** button.

- **Renaming a SmartText entry**

  To change the name of a SmartText entry, select it in the list on the left, and click on the **Rename** button. The program displays a dialog box in which you can enter the new name.

- **Editing a SmartText entry**

  To edit an entry, select it in the list on the left and then click in the large input field on the right of the dialog box. You can now change the content of the SmartText entry.

  When you have made the desired changes, click on the **Save** button.

- **Inserting a SmartText entry**

  To insert a SmartText entry into the text, select it in the list on the left of the dialog box and click on the **Insert** button. For more information, see also Inserting SmartText entries).

You can finally exit the dialog box with the **Close** button. If the enabled SmartText entry has been changed but not yet saved, PlanMaker will ask automatically if it should save the changes.

Research

This feature is only included in **SoftMaker Office Professional** and **NX Universal**.
In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

You can use the *Research function* to look up the meaning of the terms you selected in PlanMaker directly in online dictionaries and encyclopedias. PlanMaker suggests various online research platforms (Wikipedia, Oxford dictionary, Cambridge dictionary, etc.) that you can open interactively for your search.

To do so, proceed as follows:

1. Select one or more terms in your document.
2. Choose the ribbon command **Review** | group **Spelling** | **Research** .
3. In the dropdown menu, click on the desired platform on which you want to look up your research term.

The selected online platform will then be opened directly with the results for your selected term.

Available categories

In the open dropdown menu of the **Research** icon , you can choose from the following categories:

- **Recently used**

  The platforms most recently used by the research function are displayed here for quick access again (the last 3 searches).

- **English** (depending on the selected language)

  General reference books are displayed here in the language you selected in the settings (see "More" below).

- **German - English** (depending on the selected languages)

  If you have selected multiple languages in the settings (see "More" below), this category will appear with translation platforms corresponding to the selected languages.

- **More**

  You use the entry **More** to open a dialog box with the settings. Here, you specify the languages for which PlanMaker should suggest research platforms to you. Enable the desired language(s) by clicking on the respective checkbox.

  This results in the menu of the **Research** icon displaying the general reference books and translation platforms for all languages you selected, insofar as they are available in these languages.

Read aloud

This feature is only available in **SoftMaker Office Professional** and **NX Universal** and is only supported for the Windows version. It is not available in the versions for Mac, Linux and Android/iOS.

The ribbon command **Review | Speak** lets all or part of the text in your PlanMaker document be read aloud to you.

Set the desired language in your document beforehand (see Setting the language). This allows PlanMaker to recognize which language should be used for reading aloud.

To have the content read aloud, please proceed as follows:

1. Select the cell range that you want PlanMaker to read aloud.
2. Choose the ribbon command **Review | Speak** , the Text-to-speech module will open.
3. If necessary, click on the **Options** button before playback to adjust the settings for the speech output. For example, you can add the language pack required for a language in the dialog box if this language has not yet been installed in the Windows language settings. For more information on this and the other options in the dialog box, see the paragraph "Text-to-speech options" below.
4. Press the arrow button to start. Adjust the **Volume** and **Speed** as needed by using the sliders.

   The button pauses the reading aloud. Press again to continue.

   The button jumps to the previous cell and jumps to the next cell.

   The button stops the reading aloud. If you now start again with , the reading aloud will resume at the beginning of the selected cell range.
5. Click on **Close** to exit the Text-to-speech module.

Text-to-speech options

If you click on the **Options** button in the Text-to-speech module, the program opens a dialog box in which you can make settings for the speech output.

The following options are available:

- **"Tables" section**

  Here, you can choose between two methods of reading tables.

  Speak cells by row: After the first cell has been read out, PlanMaker jumps to the cell to the right of it for further reading. Once the last cell in the horizontal direction is reached, the program jumps to the next row and continues there with the cell on the far left.

Speak cells by columns: After the first cell of the table is read out, PlanMaker jumps to the cell below it for further reading. Once the last cell in the vertical direction is reached, the program jumps to the next column and continues there with the top cell.

- **"Options" section**

  Spell numbers: With this option enabled, numbers are pronounced distinctly to make it easier to recognize the occurrence of a number. This option is very useful for enumerations with numbers, for example.

  Spell punctuation: Enable this option if you want the punctuation marks of the text (. ; , ? ! - "" etc.) to be expressed as in dictation to make it easier to recognize them.

- **Language/Voices**

  There are different voices available for reading aloud. In the list **Language**, first select the language for which the available voices are to be displayed. Then click on the desired voice for this language in the list **Voices** below.

  If necessary, click on the **Add Installed Voices** button to add more voices to the list.

- **New language packs**

  PlanMaker can only read aloud content in the languages that have been installed in the Windows language settings. If the language required for reading aloud is missing here, you can access the Windows language settings directly via the **New language packs** button. Add the missing language in the "Preferred languages" section.

Document management

PlanMaker has functions that make it easier for you to manage and access your documents faster:

- **Quick paths**

  You can create *quick paths* in PlanMaker in order to quickly switch to frequently used folders when opening or saving files.

- **Document summary**

  Every document can have some additional info that you can display and edit using the **Summary** tab in the dialog box of the ribbon command **File | Properties**. You can enter the title, subject and author of the document, specify keywords for the file manager's search function and write a brief summary of the content.

- **File manager**

  You can use the integrated file manager to easily access your documents. It allows you to see the names and the document infos on the **Summary** tab for all documents at a glance. You can view, open, print or delete documents at the touch of a button.

  The **Search function** of the file manager not only lets you search for your documents by file name, but it also allows you to search by topic, title, author, etc.

- **Backup copies (file versions)**

  If desired, PlanMaker creates a *backup copy* of the document each time the document is saved. You can even instruct the program to automatically take "snapshots" of the document at regular intervals while you're working on it.

The following pages provide detailed information on these functions.

Quick paths

PlanMaker allows you to create up to twenty *quick paths* via which you can quickly switch to a specific folder when opening or saving files.

A quick path is a symbolic specification, such as "Taxes", which represents a specific folder on the hard disk (for example, the folder `C:\Accounting\Taxes`).

If you want to open a file from this folder, you must first choose the ribbon command **File | Open** and then simply click on the **Quick path** button in the dialog box to select the created quick path. The dialog box then switches immediately to the `C:\Accounting\Taxes` folder.

Quick paths are not only available in the dialog box of the command **Open**, but they are also in all other dialog boxes that are used to open or save files.

Creating quick paths

To create a new quick path, proceed as follows:

1. Open any file dialog box, for example, with the ribbon command **File | Open**. (Click directly on the icon itself to open the file dialog box.)
2. Click on the **Quick path** button in the file dialog box.
3. A menu opens below the button. Choose the command **Create new quick path** from it.
4. Enter any name for the quick path in **Name of quick path** – for example, "Taxes".
5. Enter the desired folder under **Path** – for example, "C:\Accounting\Taxes".

   Tip: Alternatively, click on the button with the three dots next to this input field. This opens a file dialog box in which you can easily select the desired folder.

6. Confirm with **OK**.

PlanMaker now knows that the quick path "Taxes" should switch to the folder `C:\Accounting\Taxes`.

Opening quick paths

Quick paths are available in all file dialog boxes – for example in the dialog boxes of the ribbon commands **File | Open** or **Insert | Picture frame**. (Click on the icon itself to open the file dialog boxes.)

To use a quick path, proceed as follows:

1. Click on the **Quick path** button in a file dialog box.
2. A menu of all the previously created quick paths opens. Select the desired quick path.

PlanMaker now switches to the folder specified for the quick path.

Editing and deleting quick paths

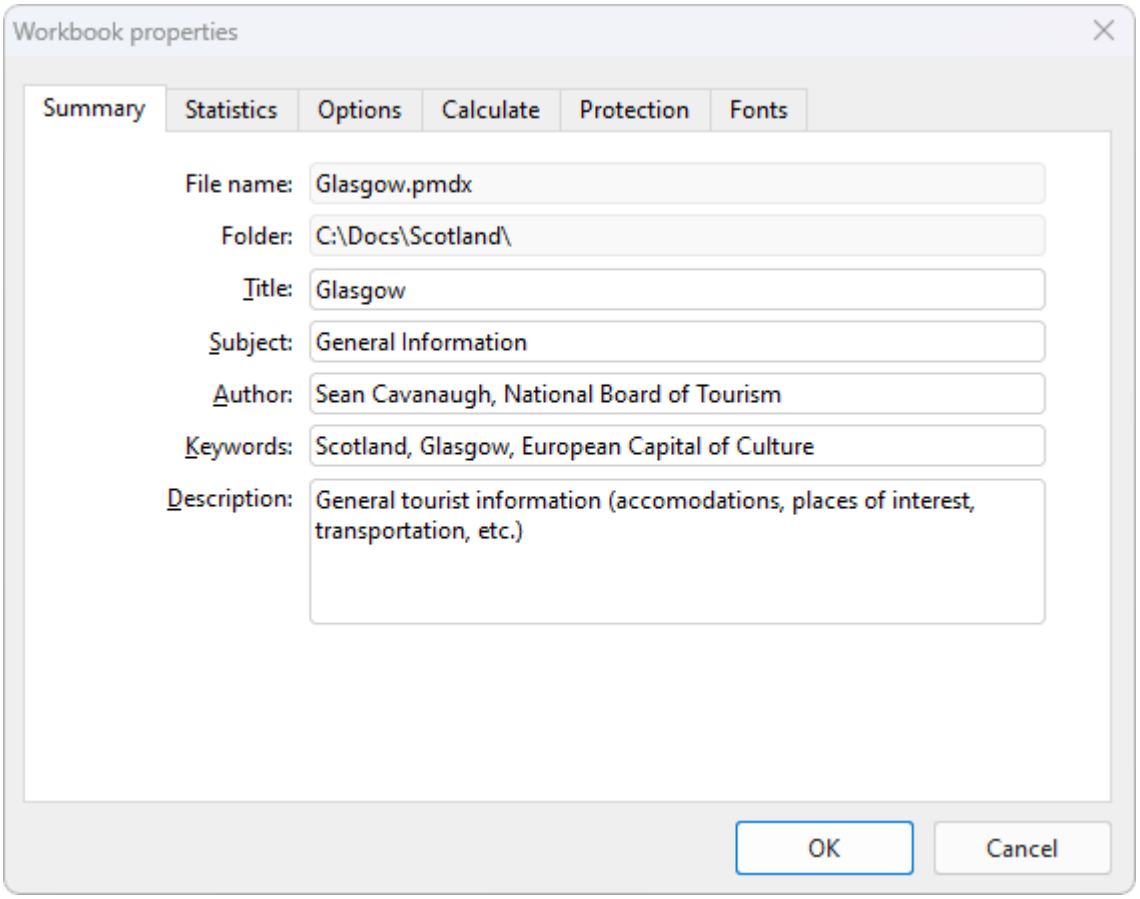
To edit or delete an existing quick path, proceed as follows:

1. Click on the **Quick path** button in a file dialog box.
2. A menu opens. Choose the command **Change quick paths** in it.
3. A dialog box appears. Select the desired quick path in it.
4. Click on **Change** to edit the name or path, or click on **Delete** to remove the quick path.

Document summary

The ribbon command **File** | group **File management** | **Properties** not only enables you to make document-specific settings but also to enter some document info on the *Summary* tab.

You can enter additional information (such as subject, author, keywords, etc.) about a document here. To view or edit such information, choose the command **File** | **Properties** and switch to the **Summary** tab.



Make the desired changes and confirm with **OK**.

Tip: You can use the file manager (see next section) to search for documents by their document info, for example, to find all documents on a specific topic.

Automatic prompt for summary information when saving: You can have PlanMaker prompt you to fill in the document summary whenever you save a new document.

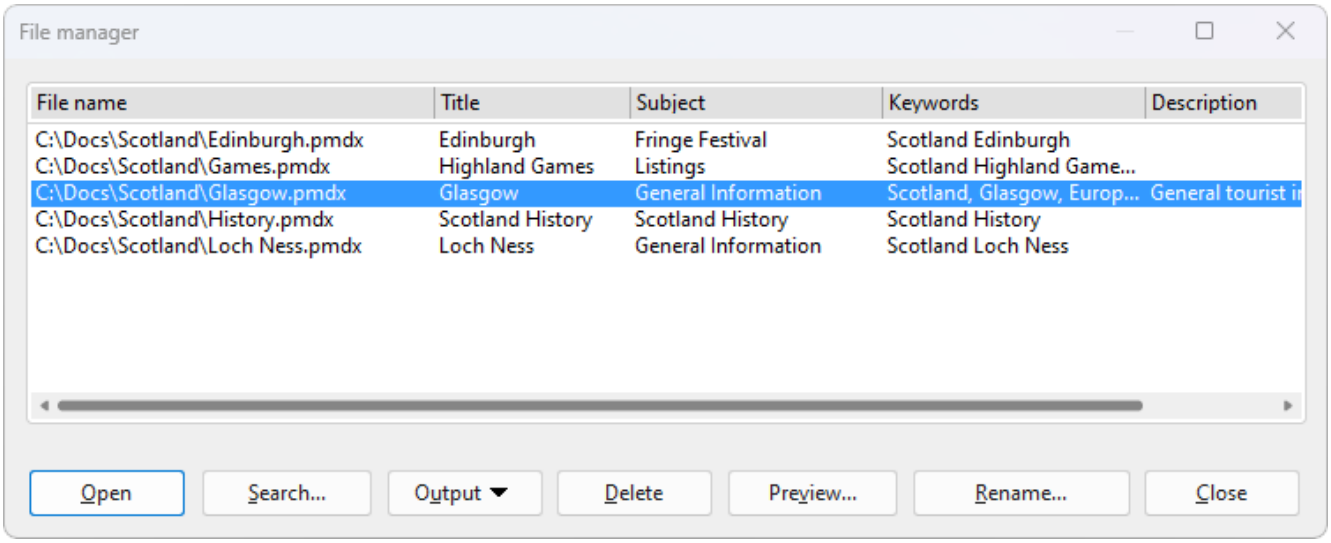
To do so, choose the ribbon command **File** | **Options**, switch to the **Files** tab and enable the option **Prompt for summary information when saving**. The dialog box for entering the document info on the summary tab will now appear every time you save a new document for the first time.

File manager

Android/iOS: The *File manager* is not available in these versions.

The file manager displays a list of documents from one or more folders and lets you open, print, delete or view any document by clicking on it.

You start the file manager with the ribbon command **File | File manager** .



You can use the mouse to move or resize the file manager window at any time. You can also change the column widths by dragging the separator lines between the column headings.

By clicking on one of the column headings, you can specify the column according to which the files are to be sorted.

You can use the **Search** button to select a different folder and also use the Search function to start searching for files. Furthermore, you can search for a specific title, subject, author, keyword, etc., within the Document summary located on the **Summary** tab.

To perform an action with a file, select the file and then click on one of the buttons. The individual buttons have the following functions:

- **Open**

  If you click on this button, PlanMaker opens the selected file.

  Tip: You can also double-click on a file name to open the file.

- **Search**

  Click on this button to search for specific files or simply to select the folder you want to display in the file manager. For more information on Searching with the file manager, see the next section.

- **Output**

  Use this button to output the selected file. When you click on the button, a menu opens with the following functions (depending on the operating system):

  E-mail: Send the document by e-mail

  Print: Print the document

- **Delete**

  If you click on this button, the selected file will be deleted (after confirmation).

- **Preview**

  Click on this button to open a preview window where you can view the selected document without having to open it.

  To actually open this document, click on the **Open** button in the preview. To exit the preview window, click on the **Cancel** button.

- **Rename**

  If you click on this button, you can rename the selected file.

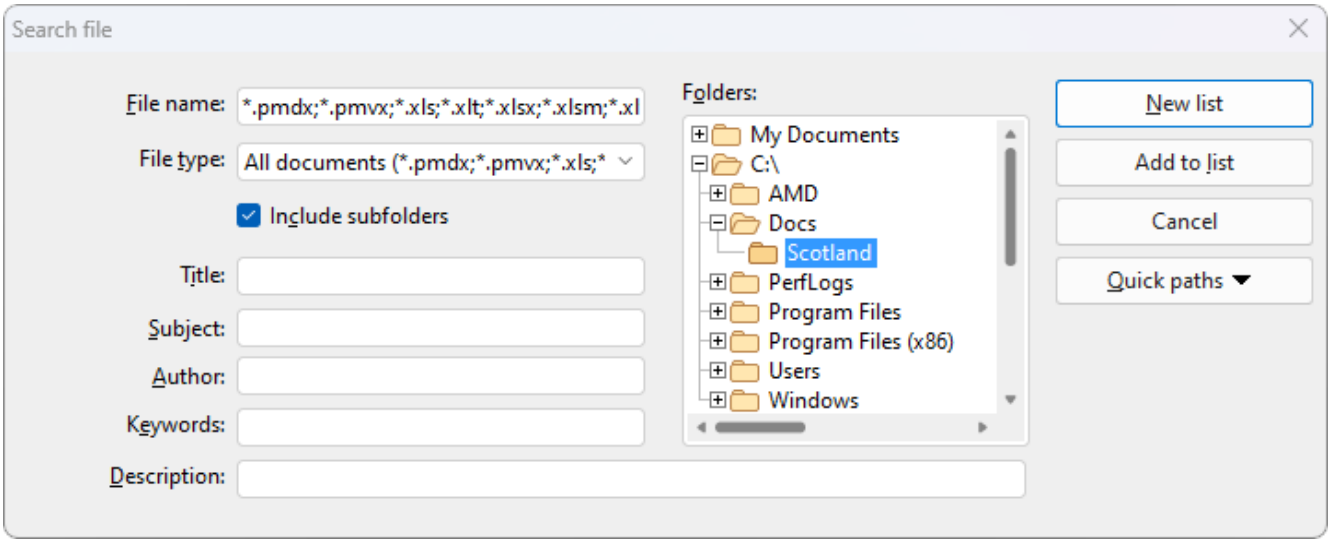
- **Close**

  Clicking on this button will close the file manager.

Searching with the file manager

Android/iOS: The *File manager* is not available in these versions.

You can use the **Search** button in the file manager to have PlanMaker search for specific files or simply choose the folder to be displayed in the file manager.



You can search for files that meet the following criteria: file name, folder, file type and the document info (title, subject, author, keywords) on the Summary tab.

Several search criteria can also be combined. If, for example, you specify a specific folder in the dialog box, PlanMaker will display *all* documents in this folder. If you also enter a search term in the **Title** field, however, PlanMaker will find only those documents that are both in the selected folder and contain the specified search term in the **Title** field on the Summary tab.

With the dialog box shown above, you control the search function:

- **File name**

  You can use this input field to search for specific files.

  If you do not enter anything here (thus accept the default setting), the search function will find *all* documents of the selected file type in the selected folder.

  If you specify a unique filename like `MyReport.pmdx`, only documents with the exact same name will be found.

  If you specify an ambiguous file name like `My*.pmdx`, the search will find all PlanMaker documents whose file name begins with "My".

- **File type**

  You can use this list to select the type of the files to be targeted in the search. If, for example, you want to search for files in Excel Format, select this format from the list **File type**.

- **Title, Subject, Author, etc.**

  By entering content in these fields, the file manager will search the document info of your documents (see Document summary).

  If you fill in several fields, the file manager will search for documents that meet *all* of your specified conditions. If, for example, you enter "court" in the **Keywords** field and "Smith" in the **Author** field, only documents whose document info contain corresponding entries for both fields will be found.

  You can also specify only part of the term for which you are searching. If, for example, you entered "court" as the search term in the **Keywords** field, your search would return all documents that contained the word "court" – thus terms like "courthouse", and "district court" would be found.

  Case is of no significance for search terms. If the search term is entered as "court", the search will also find documents whose keywords contain "Court".

- **Folders**

  Here, you can select the folder in which the file manager should perform the search.

- **Include subfolders**

  If this option is enabled, the file manager not only searches the selected folder but also all folders that are subordinate to it.

- **New list** button

  If you click on this button, you start a new search with the current settings.

- **Add to list** button

  Clicking on this button also starts a new search; however, any previous search results remain in the list instead of being cleared from the list.

Backup copies (file versions)

Android/iOS: The functions are not available in this form in these versions. Only *simple* backups can be made on Android/iOS, and this setting can be found on the Files tab.

If desired, PlanMaker creates a backup copy of the previous version of the document each time the document is saved. You can even instruct the program to automatically take "snapshots" of the document at regular intervals while you're working on it (similar to the common Auto-Recover function). You can also access these snapshot versions later even if you have not saved the document manually using the command **File | Save**.

The settings for this can be found on the **Backup** tab in the dialog box of the command **File | Options** . Select here whether and how file backups are to be created:

- **Keep older versions of the document** (if you save a file manually)

  Several generations of file backups are created for each document. All of these copies are stored in a special **Backup** folder.

  Tip: If this option is enabled, the command **File | Versions** (in the group **File management**) is also available, which allows you to easily return to a previous file version of the currently open document (see below).

  Use the **Number of versions to keep** field to specify the maximum number of versions of backups (generations) to be kept per document.

- **Automatically created file versions** ("snapshots")

  If you want automatic "snapshots" to be created as an additional backup while the document is being edited, enable the checkbox here. In the dropdown field to the right, you specify the interval at which the snapshots are to be taken regularly.

  In the dropdown field below, enter the maximum number of snapshots you want to save.

- **Folder for file versions**

  Here, you can change the path for the **Backup** folder in which all backups are stored.

- **"Clean up" button**

  This button provides two options for deleting file backups from the backup folder:

  Delete orphaned file versions removes all PlanMaker backups for which the corresponding original document no longer exists.

  Delete file versions of all documents removes *all* PlanMaker backups.

How do I use the command "File | Versions"?

If you enable the creation of file backups (see above), all backups end up in a special **Backup** folder. This folder is managed automatically by the program. We do *not* recommend that you manually move or rename the files in this folder.

To return to a previous version of a document, you don't need to be interested in this folder at all – as there is a separate command for it: **File | Versions**.

You can use this command to revert to a previous version of a document as follows:

1. Open the desired document (if it is not already open).
2. Choose the ribbon command **File** | group **File management** | **Versions** . (Click directly on the icon itself to open the version manager.)

   Tip: You can proceed even faster by clicking on the icon's arrow for the dropdown menu instead of on the icon itself. The available backups are listed here. Restore the desired version by clicking on the version entry – this will open the version in a *new* document window.
   However, the icon to the right of it will open the version in the *current* document window. (***Warning:*** This will overwrite the currently open document with an older version.)

   If you click on the small black arrow of the **Versions** icon and select the entry **File versions** in the menu that appears, you will also reach the version manager, which will provide you with additional details for a better overview. For more information, see the following steps 3. and 4.
3. Version manager: The program displays a dialog box that lists all available backups for the current document. Use the dates and times to find and select the version that you want to restore. You can use the **Preview** button to display a preview of the selected version.

   The version manager list shows that a backup for the `Test.pmdx` file might be called `Test.pmdx.135.pmbak`, for example. Thus, a sequential number and the extension `.pmbak` are always appended to the file name (and an "s" is also inserted into the sequential number for snapshots).
4. If you click on the **Restore** button, the selected previous version of the document will be restored in the current window.

   Warning: The currently open document is now replaced by the selected previous version. So if you save it now, you're overwriting the current version with an older version!

   Alternatively, you can also click on the **Open as copy** button. This opens the selected file version in a *new* document window (instead of replacing the document in the current window). This is useful if you want to compare the current version with an older version.

What should I do if I lose the original file?

The aforementioned procedure only works if the original document is still available. What should you do if the original file has been lost, for example, because it was accidentally deleted?

Even in this case, you can still access the backup copies of this document (unless they have been deleted as well). Simply open the backup copy itself.

You can see where the backups are in the dialog box of **File | Options** . On the **Backup** tab, the file path is displayed below in the **Folder for file versions** field. By default, this is a folder called **Backup** within the **SoftMaker** folder.

When you open a backup from this folder in your file browser, PlanMaker immediately starts to open the file. The procedure is as follows:

1. In your file browser (for example, Windows Explorer), navigate to the **Backup** folder.
2. In this folder, search for files whose name starts with the name of the original file. A backup for the `Test.pmdx` file might be called `Test.pmdx.135.prbak`, for example. Thus, a sequential number and the extension `.pmbak` are always appended to the file name (and an "s" is also inserted into the sequential number for snapshots).

   Tip: The higher the sequential number in the file name, the more recent the file version.
3. In the file browser, double-click on the file version that you want to open.
4. PlanMaker detects that you are trying to open a backup and asks you what you would like to do now. It displays a dialog box with the following options:

   Open the backup file: This option opens the selected file version.

   Invoke the version manager: This option opens the version manager like the command **File |** group **File management | Versions**. This then offers *all* available earlier versions of the original document for selection. For more information on using this command, see above.

   Open the original file: This option opens the original file for this file backup. (Of course, this only works if the original file still exists.)
5. Make your choice, and then click on **OK**.

Once again, please note that PlanMaker *automatically* creates and manages the files in the **Backup** folder. Thus, you should not make any manual changes to these files, such as renaming or moving the files.

If you need more space on the disk, you can delete all files in this folder at any time, but then you will lose all file backups as a result.

Outlines

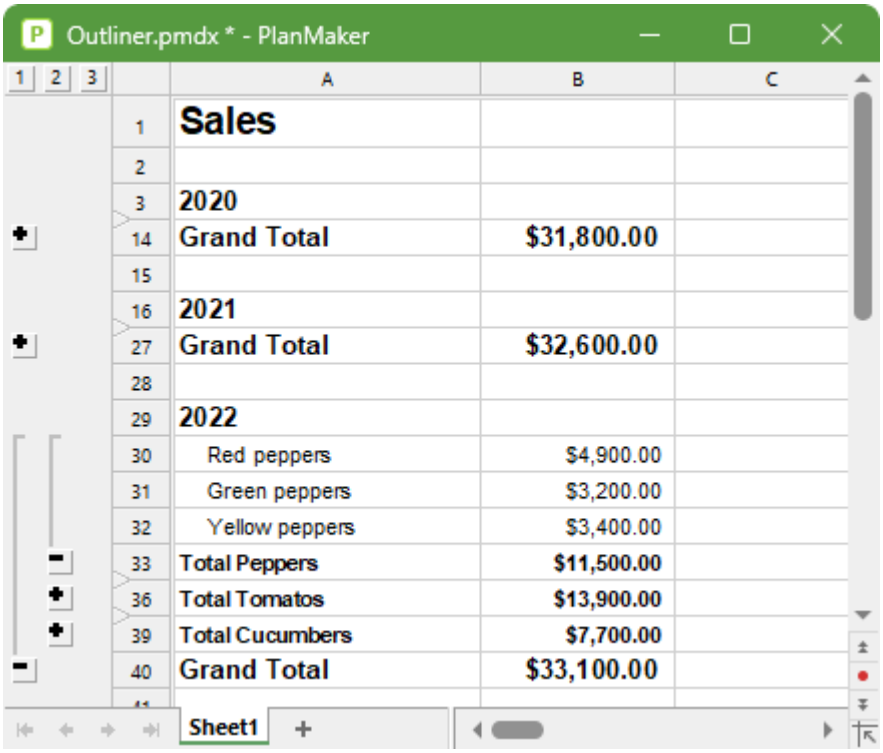
The *Outliner* allows you to create an outline for a worksheet. You can then easily decide via a mouse click whether only the most important data or also detailed data should be displayed.

In order to outline a worksheet, you have to *group* the related detailed data. If, for example, a worksheet contains sales figures for several years, with the monthly sales listed below each year, group each list of monthly figures. From then on, you will be able to show or hide them via a mouse click – depending on whether you only want to view the annual sales or also the monthly sales.

Outlines can also contain several levels – thus you can group data again within a group of data.

Outline pane

As soon as a worksheet contains grouped data, an *Outline pane* is displayed to the left of the document window.



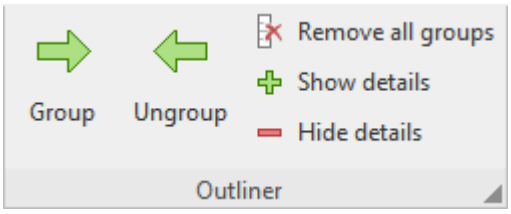
Use this Outline pane to determine which detailed data should be displayed via a mouse click:

- By clicking on the plus ✚ and minus ━ signs in the Outline pane, you can show or hide any groups of detailed data individually.
- Use the 1 2 3 buttons at the top of the Outline pane to specify that only detailed data from a certain level should generally be displayed. If, for example, you click on the "2" button, only rows of the first and second outline levels will be displayed.

Note: Worksheets can be outlined by row or by column. In a column outline, columns are grouped instead of rows. In this case, the Outline pane is displayed above the worksheet.

Using the command group "Outline"

You will find the commands for grouping cells on the ribbon tab **Data** | group **Outliner**.



This command group provides the most important icons for working with outlines:

- Group the selected cells
- Ungroup the selected cells
- Remove all groups
- Show details (equivalent to clicking on the plus sign in the Outline pane)
- Hide details (equivalent to clicking on the minus sign in the Outline pane)

The following pages contain detailed information on working with outlines.

Grouping cells

To create an outline, *group* the rows (or columns) that contain detailed data. You can then show or hide them later via a mouse click as required.

To group cells, proceed as follows:

1. Select the rows (or columns) to be grouped.
2. Choose the ribbon command **Data** | group **Outliner** | **Group** .

The cells are now grouped.

Note: Ungrouped cells have an outline level of 1. When you group cells, their outline level is increased to 2. You can even group a subset of cells within a group of cells, thus increasing their outline level to 3. Outlines can contain up to 8 outline levels.

Ungrouping grouped cells

You can ungroup cells at any time.

Proceed as follows:

1. Select the rows (or columns) to be ungrouped.
2. Choose the ribbon command **Data** | group **Outliner** | **Ungroup** .

The cells are no longer grouped – more precisely, their outline level is reduced by one level. This means that if you ungroup cells with an outline level of 3, they will not be completely ungrouped, but their outline level will be reduced to 2.

Completely removing the outline of cells

If you want to completely remove the outline of cells, proceed as follows:

1. Select the rows (or columns) whose outline you want to remove. ***Note:*** If you do not select any cells, the outline of the *entire* worksheet will be removed.
2. Choose the ribbon command **Data** | group **Outliner** | **Remove all groups** .

The outline of the cells has now been completely removed, thus their outline level is reset to 1.

Showing/hiding grouped cells

If you have created an outline for a worksheet (as described in the previous section), you can show/hide detailed data (thus grouped cells) at any time as required.

To show/hide grouped cells, use the Outline pane as follows:

- A bar with a minus sign is displayed next to grouped cells in the outline pane. To hide the grouped cells, click on the minus sign.
- A plus sign appears next to hidden cells in the Outline pane. To show the grouped cells, click on the plus sign.
- Use the buttons at the top of the Outline pane to specify that only detailed data from a certain level should generally be displayed. If, for example, you click on the "2" button, only rows of the first and second outline levels will be displayed.

For rows, you can also use the commands **Show details** and **Hide details** on the ribbon tab **Data** | group **Outliner** to show or hide the grouping (corresponds to the plus and minus signs in the Outline pane, see above).

Note: When you hide grouped cells, they are not only hidden on the screen but are also no longer considered in charts.

Changing outline settings

You can change outline settings via the dialog box *Outliner options*. To open this dialog box, click on the group arrow ◢ in the bottom right corner of the ribbon tab **Data** | group **Outliner**.

Here, you can make the following settings for the structure of the current worksheet:

- **Title below group rows**

  This option basically only determines whether the plus and minus signs should be displayed above or below the bar for grouped cells in the Outline pane.

  Enable this option if you want the plus and minus signs to be located below the grouped cells. Disable it if you want them to be above the grouped cells.

- **Title to the right of group columns**

  This option corresponds to the above option; however, it refers to worksheets where columns have been grouped rather than rows. You can use it to determine whether the plus and minus signs are located to the left or right of the grouped cells.

- **Automatically show outline pane**

  This option determines whether the Outline pane should be displayed automatically if the worksheet contains grouped cells.

  If this option is enabled, this pane will appear automatically as soon as the worksheet contains at least one group of cells. If disabled, the Outline pane will not be displayed.

- **Protect outline**

  If this option is enabled, the outline will be protected if *sheet protection* is enabled for the worksheet. For more information, see Sheet protection.

  If sheet protection is enabled, the current state of the outline will then be "frozen". Users will not be able to show or hide grouped cells nor group or ungroup cells.

Using hyperlinks

Choose the ribbon command **Insert** | group **Links** | **Hyperlink** to add a *hyperlink* to cell contents or complete cells at any time. Hyperlinks are links to, for example, websites or other documents, which can then be opened simply by clicking on the link.

If, for example, you link a cell to a web page, the web browser will automatically start when you click on this link and will access this page in it.

By the way, the file path of another document is also allowed as a destination for hyperlinks. This is then opened automatically when you click on the link.

Tip: It is also possible to use an e-mail address as a destination by placing "mailto:" in front of it.

Creating a hyperlink

To create a hyperlink, proceed as follows:

1. Select the cell or cell contents to which you want to attach a hyperlink.
2. Choose the ribbon command **Insert** | group **Links** | **Hyperlink** .
3. For **URL or file name**, specify the destination for the hyperlink: Enter the path and name of the Internet resource or file to which you want the selected text to refer.

   For links to web pages, "http://" and the address of the page should be entered – for example, "http://www.example.com".

   For links to an e-mail address, enter "mailto:" and the recipient's address – for example, "mailto:xy@example.com".

   For links to a file, the **Browse** button can be used.
4. Optional: In the **Jump target** field, you have the option of specifying a bookmark or a cell address in the document to be opened (see below).

   Note: If you want to jump to a cell address in the *currently* open document, enter a jump target, for example to cell D42 in Sheet2, in this form: Sheet2!D42
5. Optional: In the **Hyperlink text** field, you can change the text for the cell you have just selected in the document. If the cell is empty, PlanMaker automatically inserts the selected path and file name or Internet address as text.
6. Confirm with **OK**.

   Note: The **Reset** button allows you to undo all changes made *in the open dialog box* provided that you have not yet confirmed them with OK.

The hyperlink is now created. You recognize this by a change in the color of the text, which indicates the presence of a link. (Provided that there is text in the cell.)

Tip: Links to URLs and e-mail addresses can be created faster and easier if the option **Format URLs as hyperlinks** is enabled in the dialog box of the ribbon command **Review | Auto correction**. You can then type the destination address (for example, "www.example.com" or "mailto:xy@example.com") directly into the text and press the space bar. PlanMaker recognizes that you have entered a web address or e-mail address and automatically formats it as a link. This setting is global and not document-specific.

Tip: Alternatively, hyperlinks can also be inserted using the HYPERLINK function, which is much more flexible to use.

Jumping to a specific bookmark or cell address

The aforementioned dialog box also gives you the option of specifying a jump target if necessary in the document to be opened. For **Jump target**, enter the bookmark (for TextMaker or HTML documents) or the cell address (for PlanMaker documents) to which you want to jump.

For example: If you set **URL or file name** to "Sales.pmdx" and enter "Sheet2!D42" under **Jump target**, the link will open the file `Sales.pmdx` and jump to cell D42 in Sheet2.

If you want to jump to a cell address in the *currently* open document, enter a jump target, for example to cell D42 in Sheet2, in this form: Sheet2!D42

Using wildcards in links

Tip: In hyperlinks, you can use an asterisk (*) as a wildcard for the cell content.

Example:

If the cell is still empty, choose the ribbon command **Insert |** group **Links | Hyperlink** and enter http://* in the **URL or file name** box. If you then enter e.g., www.example.com in that cell, the link will point to the Internet address http://www.example.com.

This also works with other protocols – for example, mailto:* can be used to create a link to the e-mail address in a cell.

Opening hyperlinks

To open a hyperlink, simply click on the cell or text containing the hyperlink with the mouse. PlanMaker then opens the linked document.

Editing or removing a hyperlink

Existing hyperlinks can be edited as follows:

- **Changing destination**

  To change the destination of a hyperlink, select the cell(s) or text passage with the link, and then choose the ribbon command **Insert |** group **Links | Hyperlink** again. The program again displays the dialog box in which you can edit the destination for the link.

To remove a hyperlink, proceed as follows:

- **Removing a hyperlink by deleting the link text/cell**

  If you delete a text passage that contains a hyperlink, the link will also be removed. If a cell is deleted completely, this also removes the hyperlink applied to the cell.

- **Removing only the hyperlink**

  If you only want to remove the hyperlink from the cell, but retain the cell itself and its contents, select it and choose the ribbon command **Home |** group **Contents | Delete | Remove link**.

Outputting documents

In the Android/iOS version, the commands listed below are locked. (To unlock the commands, see Welcome.)

This chapter provides information on how to output documents via PlanMaker.

The chapter contains the following sections:

- **Print preview**

  The first section deals with the command **File | Print preview**. This displays the document on the screen exactly as it will appear on the printout, thus saving unnecessary test prints.

- **Printing a document**

  To actually print the current document on the printer, choose the command **File | Print**.

- **Exporting a document as a PDF file**

  You can also export a document to a PDF file by choosing the command **File | PDF export**.

  PDF files can be viewed on virtually any computer, provided that a suitable PDF viewer is installed there. All formatting and objects of the document are retained exactly as in the original file.

- **Sending a document by e-mail**

  Finally, documents can also be sent by e-mail. The command **File | Send** is responsible for this process.

See the following pages for more detailed information on these functions.

Print preview

In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

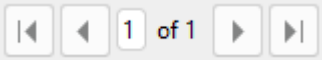
The *print preview* displays a document on the screen exactly as it will look in the printout. This feature enables you to avoid wasting paper on test prints.

To start the print preview, choose the ribbon command **File | Print preview** .

To exit it again, click on the **Close** button in the preview module or press the **Esc** key.

The icons of the print preview module

The print preview appears in a separate window. This print preview module has the following functions:



These buttons are used for scrolling between pages. The two outer buttons jump to the first/last page. The two inner buttons scroll to the previous/next page.

The input field in the middle lets you jump to a specific page. Type in the page number here and press the **Enter key**↵.

The next three buttons have the following functions:

- The left button corresponds to the ribbon command **File | Page setup**, with which you can define the page format (paper size, orientation, headers and footers, etc.). For more information, see also Setting page format.
- The middle button corresponds to the ribbon command **File | Print**, which you can use to print the document. For more information, see also Printing a document.
- The right button corresponds to the ribbon command **File | PDF export** with which you can create a PDF file from the document. For more information, see also Exporting a document to a PDF file.

These controls can be used to change the zoom level. Enter the desired percentage value in the input field and press the **Enter key**↵. Alternatively, use the three buttons to the right of the input field to switch to a predefined zoom level:

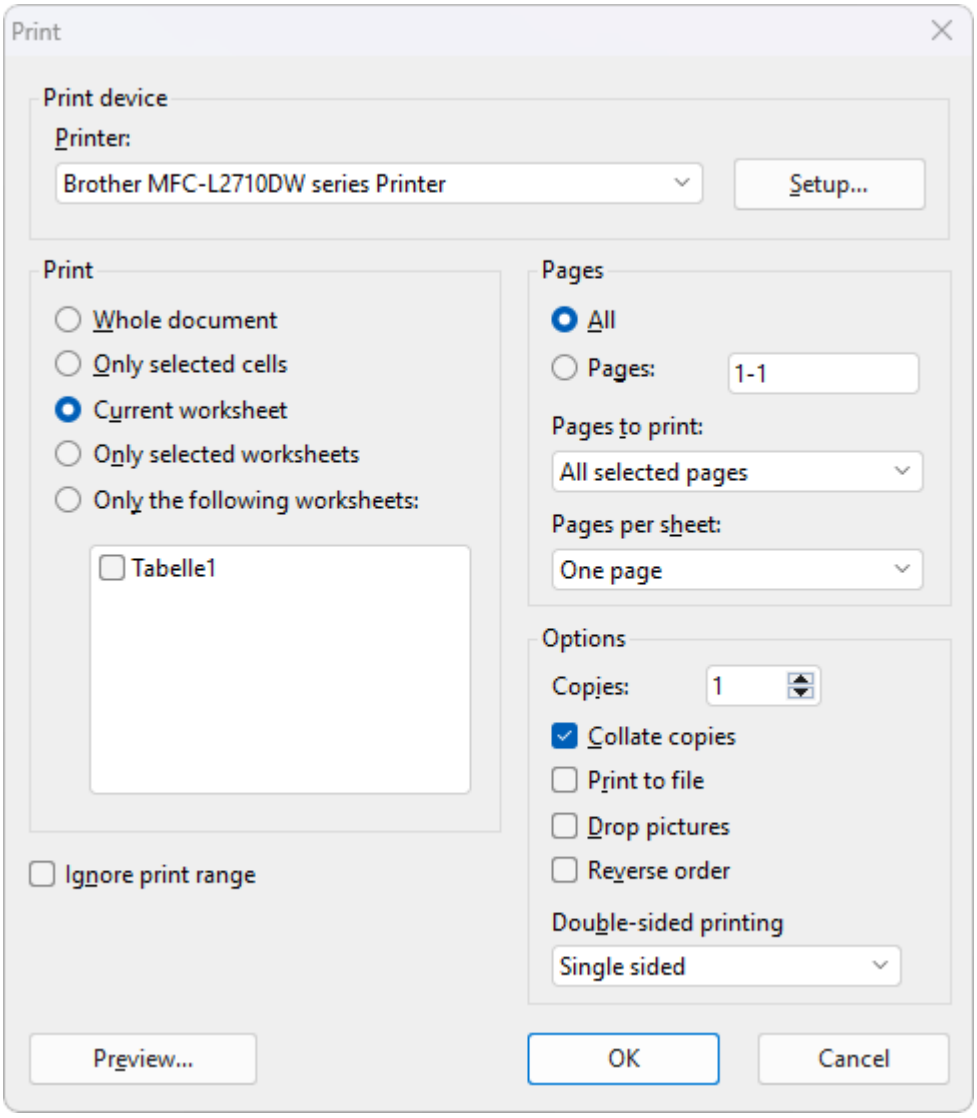
- Actual size of document (zoom level 100%)
- Fit page in window
- Fit margins in window

To close the print preview window, click on the **Close** button.

Printing a document

In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

To print the current document, choose the ribbon command **File | Print** .



The program will display a dialog box in which the following settings can be made:

Printer

Here, select which of the installed printers should print the printout.

The **Setup** button opens a window for setting up and configuring the printer. For more information, see your printer manual.

Linux users can also specify here the command to be used to initiate the printing process.

MacOS and iOS users select the printer including its setup via the **Print** button.

Print

Here, you can choose which parts of the document to print:

- **Whole document**

  Print the entire document (all worksheets).

- **Only selected cells**

  Print only those cells that are currently selected in the current worksheet.

- **Current worksheet**

  Print only the current worksheet.

- **Only selected worksheets**

  Print only those worksheets that are currently selected in the worksheet register.

- **Only the following worksheets**

  Print only the selected worksheets. To select them, use the list below this option. Click to set a check mark in front of each worksheet that you want to print.

Ignore print range

If you enable this option, PlanMaker will ignore all *print ranges* defined in the document when printing.

Background information: You can define a *print range* for each worksheet in a document if required. When a print range is defined, only those cells that are within this cell range will appear in printouts. The rest of the sheet will be omitted.

You can temporarily disable this feature at any time by enabling this option.

For more information on print ranges, see Setting page format options.

Pages

Specify here the pages of the document to be printed:

- **All:** Print all pages of the document.
- **Pages:** Print only the specified pages. To do so, enter the desired page numbers.

  Some examples:

12	Print page 12 only
12-15	Print pages 12 to 15
12-	Print page 12 and all of the following pages
-12	Print pages 1 to 12

  You can also enter more than one such page number by separating them with commas:

2-5, 12	Print pages 2 to 5 and page 12

  etc.

- You can also use the option **Pages to print** to specify whether all selected pages or only the pages with a even or odd page number should be printed.

- For **Pages per sheet**, you can specify how many pages to print on a sheet of paper. PlanMaker usually prints only one page per sheet. However, you can instruct the program to output four pages (scaled down accordingly) per sheet at a time.

Options

This section of the dialog box contains additional printing options:

- **Copies**

  Here, you can specify the number of copies of the document to be printed.

- **Collate copies**

  This option specifies whether to sort the print output by page number when printing multiple copies of a document.

  If you enable this option and print multiple copies of a document with, for example, three pages, the pages will be output in the following order: 1-2-3, 1-2-3, 1-2-3, and so on.

  If it is not enabled, the order will be: 1-1-1..., 2-2-2..., 3-3-3...

  Note: Some printers do not support this function.

- **Print to file**

  This option redirects the print output to a file instead of a physical printer.

- **Drop pictures**

  This option skips all pictures and drawings when printing (useful for fast test printouts).

- **Reverse order**

  This option prints the pages from last to first.

- **Double-sided printing** (Windows and Linux only)

  Instead of **Single-sided** printing only on the front of a page, you can set here for printing to also be performed on the back of a page ("duplex printing"):

  Double sided, long edge means that you can turn the pages like a book.
  Double sided, short edge means that you scroll the pages upwards like a clipboard.

  Note: Your printer must support duplex printing in order to be able to use this feature.

To start printing, click on **OK**.

Before actually printing a document, you can preview the printout via the command **File | Print preview**. This feature enables you to avoid wasting paper on test prints.

Tip: Print comments applied to cells

If the comments applied to table cells via the ribbon command **Insert | Comment** are to appear in the printout, perform the following two steps before printing:

- Open the dialog box "Page setup" by choosing the ribbon command **File** | group **Print** | **Page setup**. Switch to the **Options** tab and enable "Print: Comments" there (see also Setting page format options).

  Note: This option can be set separately for each worksheet.

- It is also necessary for you to *permanently* show the comments inserted in PlanMaker. To do so, use the **Comment visibility** button on the ribbon tab **Review** | group **Comments**. This allows you to permanently show all comments or single comments in your PlanMaker document (see also Using comments).

  Only these displayed comments of a worksheet will be taken into account during PDF export.

Exporting a document as a PDF file

In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

You can also export documents to a PDF file.

PDF files can be viewed on virtually any computer, provided that a suitable PDF viewer is installed there (for example, Adobe Reader). All formatting and objects (pictures, drawings, etc.) of the document are retained exactly as in the original file.

To export a document to a PDF file, proceed as follows:

1. Choose the ribbon command **File | PDF export** .
2. A dialog box appears. Make the desired settings there (see below) and confirm with **OK**.
3. Now another dialog box appears. Enter a file name under which you want to save the PDF file and confirm with **OK**.

The PDF file will then be created. To view it, open the PDF file in a PDF viewer of your choice.

Tip: If no viewer application for PDF documents is installed on your device, simply search for "PDF viewer" using your web browser. You will find numerous suitable programs, many of which are free.

In the dialog box of the above command, you can make the following settings for the PDF export:

General tab

On this tab, you can specify whether to output only parts of the document and make other settings. The options are similar to the command **File | Print** described in Printing a document.

In addition, the following options are also available:

- **Create a bookmark for each worksheet**

  If this option is enabled, PlanMaker automatically creates a bookmark in the PDF file for each worksheet that your document contains.

Your PDF viewer will then display these bookmarks in a pane to the left of the document. You can use this pane like an interactive table of contents: Just click on one of the bookmarks to instantly jump to the corresponding worksheet in the PDF file.

Tip: If your PDF viewer does not display this pane, search for a command for displaying bookmarks in it and enable it.

- **Create tagged PDF**

  If this option is enabled, information about the logical structure of the document will also be saved in the PDF file.

  This information ensures that your PDF files are accessible for the disabled and can also be used, for example, with PDF viewers that reproduce or read aloud the document content in Braille.

- **Reset** (see "Reset" button at the end of this section)

Preferences tab

On the this tab, you can make settings for the output quality:

- **AutoShape options**

  Here, you can set the resolution (i.e., number of pixels) in which AutoShapes and all other types of drawings are exported. The higher the resolution, the larger the generated PDF file will be.

- **Picture options**

  The option **Compress to** sets the upper limit for the resolution of the images (number of pixels) in the current PDF export. The value set here is the maximum value for all images in the document, regardless of whether they have a higher or lower resolution. If the resolution of the images in your document is lower than the set value anyway, their resolution will not change.

  Note: The images in your PlanMaker document itself will not be compressed in the process, but you can achieve this with the command Compress pictures.

  With the options below, you can specify whether to save images with **lossless compression** or with lossy **JPEG compression**. If you opt for JPEG compression, you can set the quality level of the compression.

  Tip: Normally, you should choose lossless compression here because it ensures perfect image quality. If your document contains a large number of images (especially photographs), however, the resulting PDF file may be very large. In this case, we recommend that you use JPEG compression, which provides a much higher compression ratio for photographs.

  The option **Use alpha channel for transparencies** determines whether an alpha channel should be used to represent transparency in pictures and drawings. Usually, they look best when this option is enabled. However, some PDF viewers represent alpha channels incorrectly, thus you can also disable this option.

- **Other options**

  Include PlanMaker document: This option saves a copy of your PlanMaker document inside the PDF file. You can then open this PDF file directly in PlanMaker, whereupon the program will extract the original PlanMaker document from the PDF file and display it.

  Open the file in a PDF viewer after exporting: This option launches your PDF viewer automatically and displays the PDF file after it has been created (provided that you have a PDF viewer installed on your computer).

Use relative paths for file links: This option automatically converts the file path from hyperlinks (links) to files into a relative path (rather than an absolute path). If, for example, you have inserted a link to the file `C:\Documents\Lists\Test.txt` and export the PDF document to the folder `C:\Documents`, this link will be automatically converted into "Lists\Test.txt".

- **Font options**

  The option **Font embedding** allows you to specify whether the fonts used in the document should be embedded in the PDF file.

  If font embedding is enabled, the resulting PDF file can be viewed in its original fonts on *any* computer – even if the relevant computer does not have the fonts installed. If, on the other hand, the option is disabled, the PDF file will be displayed with automatically selected replacement fonts.

  The option **Embed only used characters** determines whether all characters of a font or only the characters actually present in the document should be embedded when embedding fonts. The advantage of enabling this option is that the resulting PDF file is smaller in size.

- **Archiving**

  This option enables you to select from various variants in the standard **PDF/A** to create standard-compliant PDF files. This standardized format allows the long-term archiving of documents and is considered a prerequisite when documents must be stored in a non-editable manner and in an identical format to the original documents.

- **Reset** (see "Reset" button at the end of this section)

View tab

This tab contains advanced options for displaying the PDF file in a PDF viewer:

- **Navigation pane**

  Use this option to specify what should be displayed in the navigation pane of the PDF viewer. (The navigation pane is a bar displayed to the left of the PDF document.)

  Available options:

  Default: Use the PDF viewer's default setting.

  Empty: Do not display anything in the navigation pane.

  Bookmarks: Display the document's bookmarks in the navigation pane.

  Page thumbnails: Display thumbnails of the document's pages in the navigation pane.

- **Zoom**

  This option specifies the zoom level at which the PDF file should be displayed:

  Default: Use the PDF viewer's default setting.

  Full page: Select the zoom level at which the entire page fits in the window.

  Fit width: Select the zoom level at which the page width fits in the window.

  Fit height: Select the zoom level at which the page height fits in the window.

  Custom: Set the zoom level to a custom value (in percent).

- **Page layout**

  This option specifies the layout with which the pages should be displayed:

  Default: Use the PDF viewer's default setting.

  Single pages: Display as single pages.

  Two pages - odd pages left: Display as two facing pages (odd pages on the left).

  Two pages - odd pages right: Display as two facing pages (odd pages on the right).

- **Additional options**

  This section provides the following additional options:

  Display document title: Display the document's title in the title bar of the PDF viewer (instead of its file name). Tip: You can enter the title of a document on the **Summary** tab in the dialog box of the command **File | Properties**.

  Center viewer on screen: This option automatically centers the program window of the PDF viewer on the screen when the PDF file is opened.

  Full-screen mode: This option automatically switches the PDF viewer to full-screen mode when the PDF file is opened.

  Hide menu bar: This option hides the menu bar of the PDF viewer.

  Hide toolbars: This option hides all toolbars of the PDF viewer.

  Hide navigation pane controls: This option hides all controls of the navigation pane.

- **Reset** (see "Reset" button at the end of this section)

Protection tab

This tab provides options that enable you to encrypt the resulting PDF file and to use passwords to protect the file from being opened or changed.

- **Encryption type**

  Here, you can specify whether the PDF file should be encrypted. You can choose between **40 bit encryption** (low level of security) and **128 bit encryption** (high level of security).

  > **Note:** The other options in this dialog box are not available until you have selected an encryption type.

- **Password to open the document**

  If you enter a password here, the PDF viewer will ask for this password when users try to open the PDF file. If an incorrect password is entered, the user cannot open the file.

  If you leave this field empty, any user can open the file without having to enter a password.

- **Password to set permissions**

  You can also restrict the permissions for the PDF file. To do so, enter a password to change the permissions. Then use the options in the **Permissions** section of the dialog box to specify which permissions should be granted to users.

Background information: Some PDF viewers have a command for changing the permissions. If a user chooses this command, they can only continue if they enter the password that you have set here.

- **Permissions**

  Here, you can specify which permissions users should have with respect to the PDF file when they open it in a PDF viewer. (This is only available if you have entered a password above for setting permissions.)

- **Reset** ("Reset" button below)

"Reset" button

All changes to the PDF export options are retained as global settings in PlanMaker and are thus applied to the next export of a PDF document. You can restore the original default settings by clicking on the **Reset** button in the lower left corner of the dialog box.

You can click on this button on any tab of the dialog box *PDF Export*, thereby resetting the options of all tabs at once.

Tip: Ensure comments applied to cells are output in the PDF file

If the comments applied to table cells via the ribbon command **Insert | Comment** are to appear in the PDF export, perform the following two steps before the PDF export:

- Open the dialog box "Page setup" by choosing the ribbon command **File |** group **Print | Page setup**. Switch to the **Options** tab and enable "Print: Comments" there (see also Setting page format options).

  Note: This option can be set separately for each worksheet.

- It is also necessary that you *permanently* show the comments inserted in PlanMaker. To do so, use the **Comment visibility** button on the ribbon tab **Review |** group **Comments**. This allows you to permanently show all comments or single comments in your PlanMaker document (see also Using comments).

  Only these displayed comments of a worksheet will be taken into account during PDF export.

Sending a document by e-mail

In the Android/iOS version, the command is locked. (To unlock the command, see Welcome.)

If you have a suitable e-mail program installed on your device, you can send documents by e-mail directly from PlanMaker.

To do so, proceed as follows:

1. Create or open the document that is to be e-mailed.
2. In PlanMaker, choose the ribbon command **File** group **| Document | Send** .

   Note for users of **Android/iOS** and **Windows 10** or higher: Here you can use the command **Share**, which enables sharing via different channels.

3. Select the format in which the document is to be sent.

   Note: If you select the **PlanMaker document** format here, the recipient will receive an e-mail with the PlanMaker file as an attachment. In order to open this attachment, the recipient must also have PlanMaker installed.

PlanMaker now generates an e-mail with the file as an attachment and transfers it to your e-mail program.

Protecting cells and documents

PlanMaker provides several options for protecting documents or specific document contents. For example, cells can be protected from unintentional changes, or entire documents can be password-protected.

The following options are available:

- **Sheet protection**

  You can use *sheet protection* to protect specific cells in the worksheet. If sheet protection is enabled, these cells can no longer be changed and/or their content is hidden on screen or printouts.

- **Workbook protection**

  Use *workbook protection* to prevent users from adding, deleting or showing/hiding worksheets.

- **Document protection**

  Use *document protection* to password-protect the entire document against unauthorized reading or saving. If read protection is enabled, PlanMaker will additionally encrypt the document.

See the following pages for more detailed information.

Sheet protection

Enable *sheet protection* for a worksheet to protect it from unwanted changes. When sheet protection is enabled, all cells that are marked accordingly are protected in the current worksheet. Furthermore, many commands are grayed out (are thus no longer selectable) and no objects can be inserted.

Before you enable sheet protection, however, you must first define *which* cells are to be protected and *how extensive* the protection is to be.

Protecting cells involves two steps:

- Step 1: Setting up protection settings for cells
- Step 2: Enabling sheet protection

For more information, see the following pages.

Step 1: Setting up protection settings for cells

To protect cells, you first define *which* cells are to be protected and *how extensive* the protection is to be.

To do so, proceed as follows:

1. Select the cell(s) whose protection settings are to be changed.
2. Click on the group arrow ◢ in the bottom right corner of the ribbon tab **Home** | group **Number** to open the dialog box.

   Alternatively: Right-click to open the context menu and select the entry **Cell**.
3. Switch to the **Protection** tab.
4. Select the desired protection settings (see below).
5. Confirm with **OK**.

Tip: The options *Protect cell* and *Hide formula* can also be found directly on the ribbon tab **Review** | group **Protection** (as options *Cell protection* and *Hide formula* here).

The individual protection settings in the dialog box of this command mean the following:

- **Protect cell**

  This option protects cells against changes.

  If sheet protection is enabled, neither the content nor formatting of cells can be changed.

  Important: By default, this option is enabled for *all* cells in the worksheet. If you want certain cells to still be editable if sheet protection is enabled, select those cells and *disable* the option **Protect cell**.

- **Hide formula**

  This option only shows the result and does not display the formula used to calculate the cell content.

  If sheet protection is enabled, the calculation contained in this cell cannot be determined.

- **Hide cell**

  This option hides the cell content on screen.

  If sheet protection is enabled, the cell becomes invisible. However, it still appears in the printout (see next option).

- **Do not print cell**

  This option hides the cell content in printouts.

  If sheet protection is enabled, the cell will not appear when you print the document.

Step 2: Enabling sheet protection

The second step in protecting cells is to *enable* sheet protection for the current worksheet.

This is because if you have changed the protection settings of cells (as described in the first step "Setting up protection settings for cells"), this will have *not* yet affected the document.You must first *enable* **Sheet protection** for these settings to take effect.

Note: Enabling sheet protection only affects the current worksheet.

To enable sheet protection for a worksheet, proceed as follows:

1. Choose the ribbon command **Review** | group **Protection** | **Sheet protection** .
2. If desired, you can now enter a **password** that the user must enter in order to disable sheet protection again.

   If you do not want to assign a password, simply leave the input field blank.
3. Confirm with **OK**.

The sheet protection is now enabled; the protection settings of the cells now take effect.

You notice this, among other things, by the fact that you can only edit cells for which the option **Protect cell** has *not* been enabled. These cells are displayed with a small green triangle in their lower right corner. This allows you to see immediately which cells can still be changed when sheet protection is enabled.

Tip: If sheet protection is enabled, only editable cells can be accessed when moving around in the table with the **Tab** key. Protected cells are skipped.

If sheet protection is enabled, there are also the following restrictions:

- Most commands for editing the worksheet are unavailable.
- Position, size and all other properties of objects can no longer be changed (except for objects for which you have disabled the **Locked** property).
- Text in text frames can no longer be changed (except for objects for which you have disabled the **Text locked** property).
- New objects can no longer be inserted into the worksheet.

Disabling sheet protection

If you enabled sheet protection, you can remove it at any time. (You will recognize active sheet protection by the fact that the icon has a dark color.)

To do so, choose the ribbon command **Review** | group **Protection** | **Sheet protection** again.

The sheet protection is then disabled again.

The protection settings defined in the dialog box of the ribbon tab **Home** | group **Number** ◢ are now ineffective again. However, PlanMaker still remembers them. Thus, if you re-enable sheet protection, the same protection settings will become active again.

Tip: If you save a document with sheet protection enabled, sheet protection will be automatically re-enabled the next time the document is opened. If sheet protection is still secured with a password, it can only be disabled with the correct password. This allows you to protect documents against changes by unauthorized persons.

Workbook protection

If you enable *workbook protection* in a document, you can no longer make any changes to the worksheet register in it. Furthermore, the commands of the **Insert |** group **Tables | Sheet** button are no longer available.

The main effects of this are as follows:

- Users can no longer add worksheets.
- Users can no longer copy, delete or rename worksheets.
- Users can no longer show/hide worksheets.

Choose the command **Review |** group **Protection | Workbook protection** to enable workbook protection. You can also disable it again with the same command.

For more information, see the following pages.

Enabling workbook protection

To enable workbook protection for the current document, proceed as follows:

1. Choose the ribbon command **Review |** group **Protection | Workbook protection** .
2. If desired, you can now enter a **password** that must be entered by the user to disable workbook protection.

   If you do not want to assign a password, simply leave the input field blank.
3. Confirm with **OK**.

Workbook protection is now enabled; it is no longer possible to insert, delete, show/hide worksheets, etc.

Furthermore, most of the options in the dialog box of the command **File | Properties** are no longer available.

Disabling workbook protection

If workbook protection has been enabled in the current document, you can remove it at any time. (You will recognize active workbook protection by the fact that the icon has a dark color.)

To do so, choose the ribbon command **Review |** group **Protection | Workbook protection** again.

This disables workbook protection again. You can then insert, delete, show/hide worksheets, etc., again.

Tip: If you save a document with workbook protection enabled, workbook protection will be automatically re-enabled the next time the document is opened. Furthermore, if the protection is secured with a password, it can only be disabled with the correct password.

Document protection

You can protect your documents from being read or saved by unauthorized persons by applying *document protection* – or more specifically, read and/or write protection. For example, you can require the entry of a password to open or save a protected document.

To protect a document, choose the ribbon command **File | Properties**, switch to the **Protection** tab in the dialog box and select the type of document protection. Also enter the desired read and/or write password.

The following types of protection are available:

- **No protection** (this is the default setting whereby the document is not protected)
- **Write protection** (the document can be opened, but saving is only possible with the password)
- **Read protection** (the document can be opened only with the password)
- **Protection depending on password** (a combination of read and write protection)

For more information on enabling and disabling document protection, see the following pages.

Enabling document protection

You can protect your documents from being read or saved by unauthorized persons by applying *document protection* – or more specifically, read and/or write protection. For example, you can require the entry of a password to open or save a protected document.

Read protection also encrypts the document so that it cannot be viewed with programs other than PlanMaker.

To protect a document, proceed as follows:

1. Choose the ribbon command **File** | group **File management** | **Properties** .
2. Switch to the **Protection** tab.
3. Select the desired type of document protection (see below).
4. Unlike sheet protection and workbook protection, you must *always* assign a password for document protection. Enter the read and/or write password (between 4 and 15 characters).

   Important: If you forget a password, you can no longer open or save the document. It is thus advisable to make a note of the password. Pay close attention to uppercase/lowercase!

5. Confirm with **OK**.
6. The program displays a dialog box in which you are required to verify the read and/or write password by re-entering it.
7. Confirm with **OK**.
8. Exit the dialog box with **OK**.
9. Save the document in order for the changes to take effect.

From now on, anyone who attempts to open this document will be asked automatically for the password. If the correct password is entered, the protection is temporarily overridden.

Types of document protection

You can choose between the following types of document protection in the above dialog box:

- **No protection**

  The document is not protected. This is the default setting.

- **Write protection**

  When the document is opened, the user is prompted to enter the write password. The document can only be saved if the correct write password has been entered here.

  However, it is still possible to open the document as *read-only*, even even if the user does not have the password. However, it cannot be saved – not even under a different name.

- **Read protection**

  If you try to open the document, you are prompted to enter the read password. If you enter an incorrect password, you will not be able to open the document.

  The document is saved in an encrypted form so that it cannot be viewed with programs other than PlanMaker.

- **Protection depending on password**

  Here, two dialog boxes prompting for a password appear in succession when the document is opened. The first dialog box asks for the read password, while the second one asks for the write password.

  If the user enters the *read password*, he is permitted to open the document but not to save it.

  If he also enters the *write password*, he has full access and can thus save the document.

  If he enters an *incorrect password* or nothing at all, access to the document is completely denied.

Encryption method: "Most compatible" or "Most secure"?

For documents with **Read protection** or **Protection depending on password**, you can also select the desired **Encryption method**.

Note: This setting primarily affects newly created documents. When you open an *existing* encrypted document, the encryption method used in it is automatically determined and this option is set accordingly.

Choosing an encryption method is especially important for documents in Microsoft Excel file format (.xls and .xlsx). Some old versions of Microsoft Office may not be able to open documents encrypted with the new "Most secure" encryption method.

Available options:

- **Most compatible**

  When you save the document, an old encryption method is used that can be read by *all* versions of Microsoft Office. However, this method is less secure than the other method.

- **Most secure**

  Here, a more modern and more secure encryption method is used. However, some old versions of Microsoft Office may *not* be able to read documents encrypted with this method.

Detailed information about the currently selected encryption method is displayed at the bottom of the dialog box.

Disabling document protection

When you succeed in opening a read-protected or write-protected document by entering the correct password, protection is disabled – but only *temporarily*. If you close this document, the protection will be enabled again; the next time you open it, the password prompt will appear again.

To *permanently* remove the protection on a document, proceed as follows:

1. Open the document.
2. Choose ribbon command **File** | group **File management** | **Properties** .
3. Switch to the **Protection** tab.
4. Select the option **No protection**.
5. Confirm with **OK**.
6. Save the document in order for the change to take effect.

Document protection is now permanently disabled.

The next time someone opens the document, no password prompt will appear. All users can now open and save it.

Working with other file formats

PlanMaker normally saves documents in its "in-house" file format called *PlanMaker document*. PlanMaker documents can be recognized easily by the name extension `.pmdx`. If, for example, you save a document under the name "Taxes", its complete file name will be `Taxes.pmdx`.

You can also instruct PlanMaker to save files in another file format. This is necessary, for example, if you want to edit a PlanMaker document with another program. Conversely, you can also open documents in PlanMaker, which have been created with other spreadsheet applications.

In this chapter, you will learn everything you need to know about working with other file formats. It is divided into the following sections:

- **Saving and opening files in other file formats**

  The first section explains how to save or open documents in a different file format: Simply choose the ribbon command **File | Save as** or **File | Open** and select the desired format from the dropdown list of file types.

- **Notes on the text format**

  When you open or save text files, a dialog box appears with additional options. This section contains information on these options.

- **Notes on the Excel format**

  This section provides information on working with Microsoft Excel files. In this section, you will learn everything you need to know about data exchange between PlanMaker and Excel.

- **Notes on the HTML format**

  The last section deals with saving documents in HTML format.

See the following pages for more information.

Saving and opening files in other file formats

This section provides information on saving and opening documents in another file format.

Saving a document in another file format

To save the current document in another file format, choose the ribbon command **File | Save as** and open the dropdown list **Save as type**. Select the desired format, enter a different file name if required and confirm saving.

Opening a document in another file format

To open a document in another file format, use the ribbon command **File | Open** (click directly on the icon itself) to open the file dialog box, open the dropdown list of file types and select the format in which the file is available. Then select the desired file, and confirm that you want to open it.

Supported file formats

In the dialog boxes for opening and saving documents, the list of file types includes the following file formats:

- **PlanMaker document**

  The in-house format for PlanMaker documents in which you usually save documents (name extension: .pmdx).

- **PlanMaker template**

  Document template for PlanMaker documents (name extension: .pmvx). For more information, see Document templates.

- **Microsoft Excel document**

  You can also open and save documents in Microsoft Excel file format. The program supports .xls files (Excel 5.0 to 2003) as well as .xlsx files (Excel 2007 or later).

  For more information on working with Excel files, see Notes on the Excel format.

- **Microsoft Excel template**

  Document template for Microsoft Excel documents.

- **PlanMaker 2012 document**

  Documents are saved in the file format of PlanMaker 2012, an older version of PlanMaker (name extension: .pmd).

- **TextMaker document** (save only)

  This option saves the document in a file format that can be read by SoftMaker's word processor TextMaker.

- **HTML** (save only)

  Documents are saved in HTML format. For more information, see Notes on the HTML format.

- **dBASE**

  dBASE database file format with either a DOS, Windows or Unicode character set. Before opening/saving the database, the program displays a dialog box in which you can select the character set.
  Note: Most dBASE databases are stored in **dBASE/*DOS*** file format – not in **dBASE/*Windows*** file format. Even Windows database programs usually use the DOS format.

  Tip: In the preview window of the dialog box, you can click on the column header to specify the data type for each column – that is, the format with which the cells of the column are to be displayed in PlanMaker.

- **SQLite database**

  The SQLite format is a fast, flexible and widely used SQL database that you can access locally (i.e. without the usual database server). Several tables can also be created in a SQLite database.

- **Text file/CSV file**

  This is a plain text file format, for example, with the cell values separated by tabs or commas. For more information, see Notes on the text format.

- **Text file/CSV file with additional settings**

  As above, but before the file is saved, the program opens an additional dialog box in which you can make advanced settings with respect to the file format (for example, specify separators). For more information, see Notes on the text format.

  Note: When *opening* a text file, the advanced settings always appear automatically.

Notes on the text format

As mentioned in the previous section, you can also open and save plain text files (including CSV files) with PlanMaker:

- To open a text file, use the ribbon command **File | Open** (click directly on the icon itself) to open the file dialog box, simply select the desired file and confirm opening.
- To save a document as a text file, choose the ribbon command **File | Save as**, select **Text file** in the **Save as type** list, enter the desired file name and confirm saving.

In both cases, the program displays a dialog box in which you can specify the format of the text file.

The dialog box provides the following options:

Options when saving text files

Note: To be able to make advanced settings for the file format (for example, set separators) via the dialog box, select **Text file/CSV file with additional options** in the dropdown list **Save as type** when saving.

When you save a document as a text file, you can make the following additional settings in the dialog box:

- **Character set**

  Here, you can specify which character set to use when the text file is saved.

  Note: PlanMaker suggests a suitable character set automatically. Usually, there is no need to change this setting.

- **Data format**

  Here, you can set the format in which the cell contents are to be saved in the text file. The following options are available:

  Separators between fields: If you select this option, the cell contents will be separated by a certain character (for example, a semicolon or a tab, see "Separator" below).

Example:

```
Peter;Miller;24;Main Street
Elizabeth;Jones-Nightingdale;1733;University Drive
```

Fixed field widths: If you select this option, the cell contents will be distributed over columns with a fixed width. Gaps will be filled with spaces.

Example:

```
Peter       Miller               24     Main Street
Elizabeth   Jones-Nightingdale   1733   University Drive
```

- **Decimal separator/Thousands separator**

  Here, you can specify which character is used as the decimal separator/thousands separator for numbers for the export of the text file.

  Explanation: The **Decimal separator** is the character that is placed in a number with decimal places (for example, the period in 17.50). The **Thousands separator** is the character that can be used to group long digits (for example, the commas in 1,000,000). However, different characters are used for both separators depending on the region.

  If you would like to use other separators for the exported text file (e.g. 17.50 and 1,000,000), you can specify them here.

  Note: For the thousands separator, please note that you must have previously enabled the display of thousands separators via the Number format so that these characters also appear in the text file after the export.

  Important: The thousand separator can only be applied when the cell is formatted as *Number* or *Currency*.

- **Separator**

  Here, you can specify which character should be used to separate the individual cell contents from each other (for example, semicolons or tabs). Thus, you set the separator here if you have chosen the option "Separators between fields" above.

  Note: Of course, the separator must not occur *within* cell contents, otherwise PlanMaker would make two entries from a corresponding cell.

- **Text marker**

  When *saving*, PlanMaker can enclose all cells that contain text within certain characters (for example, quotation marks) if desired.

  When *opening* a text file, you can specify here whether text components in the file are surrounded by such text markers. These characters will then be removed when importing to PlanMaker.

  If you want to do so, select the desired character from the list. Otherwise, select **None**.

  If you select the option **Auto** (only when saving), a text marker will never be set. unless a line break or a quotation mark already occurs in a cell.Additional quotation marks are then set as a text marker for these cells.

- **Encoding of newlines**

  Here, you can select the control characters to insert at the end of each line (to indicate the beginning of a new line).

  Note: PlanMaker suggests suitable control characters automatically. Usually, there is no need to change this setting.

  Available options:

  CR+LF: This is the recommended setting for **Windows**.

  LF: This is the recommended setting for **Linux** and **Mac**.

  CR: This control character was common on older Apple Mac systems (before macOS).

Options when opening text files

When you *open* a text file, most of the same options are available as when saving. Use these options as described above to specify the format of the text file you want to open.

There are some additional options when opening files:

- **Date format**

  If your text file contains date values, you can decide here whether the components of the date should be positioned differently when importing. If the data in your text file is in the date format DD/MM/YYYY and not in the usual MM/DD/YYYY format, you can specify this here.

  Example:

  You always use the date format that is commonly used in the United States, for example, "12/20/2022". In your text file, however, the date values are in the UK format "20/12/2022". Thus, select the "DMY" option to indicate which part of the date is in which position. After importing, the date will then be displayed in PlanMaker in your region-specific format – i.e. "12/20/2022".

  In this case, however, the data type for this column must be set from "Default" to "Date" in the preview window of the dialog box (by clicking on the column header).

- **Number of header lines**

  If the first few lines of your text file contain headers (or lines with some explanatory text or similar), enter the number of the lines here. PlanMaker will then refrain from distributing these lines over several columns like the rest of the data and will store them unchanged in just one cell per line.

  The default value is 0 (i.e., no header lines).

- **Decimal separator/Thousands separator**

  Here, you can specify which character is used as the decimal separator/thousands separator for numbers in your text file.

  Explanation: The **Decimal separator** is the character that is placed in a number with decimal places (for example, the period in 17.50). The **Thousands separator** is the character that can be used to group long digits (for example, the commas in 1,000,000). However, different characters are used for both separators depending on the region.

  If other separators (e.g., 17,50 and 1.000.000) are used in your text file rather than the usual separators in your region, you can specify them here. After the import, the numbers in PlanMaker will then be displayed with the specific separators of your region (as in the examples above).

Note: Ensure that you do not use the same characters for decimal separator/thousands separator and for the separators mentioned below.

- **Separators** (several can be selected at the same time)

  Here, you specify which character is used in your text file to separate the individual cell contents from each other (for example, semicolons or tabs).

  In order to facilitate the separation of text in more complex text files, you can select different separators at the same time.

- **Change data type in the preview window**

  In the preview window, the data type for each column can be specified by clicking on the column header. Here, you can determine the format in which the cells for the respective column should be displayed in PlanMaker.

  For example, change the data type of a column to text so that numbers with leading zeros (as in phone area codes or zip codes) are retained.

  For numbers surrounded by quotes and thus interpreted as text, set the desired number format.

- **Extended number scanning**

  As described above, you can specify the desired format using the column header in the preview window. If you enable the option "Extended number scanning", such manual adjustment is no longer necessary in most cases for certain number formats that are interpreted as text. PlanMaker then automatically detects numbers or dates and converts them from text to an appropriate number format (right-aligned).

  If the option is disabled, certain number formats may be displayed in text format (left-aligned). In this case, you can manually set the desired number formats in the preview window using the column headers.

Notes on the Excel format

If you were using Microsoft Excel spreadsheets before PlanMaker, you will be pleased to know that PlanMaker can open virtually any Excel file without any problems. Conversely, PlanMaker also allows you to save documents in Excel format.

On the next pages, read what to consider when opening and saving Excel documents.

Opening and saving Excel files

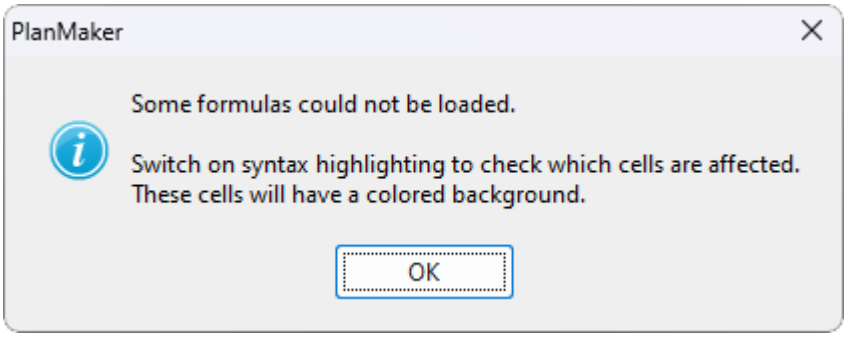
As mentioned in the section Saving and opening files in other file formats, PlanMaker is able to open and save documents in Microsoft Excel format.

This works smoothly in most cases. However, some arithmetic functions are either incompatible or are not supported by PlanMaker or Excel.

This section provides help in resolving these issues:

Warning message issued when opening an Excel file

When you open an Excel file containing arithmetic functions that are not compatible with PlanMaker, the following warning message is displayed:



Then proceed as follows:

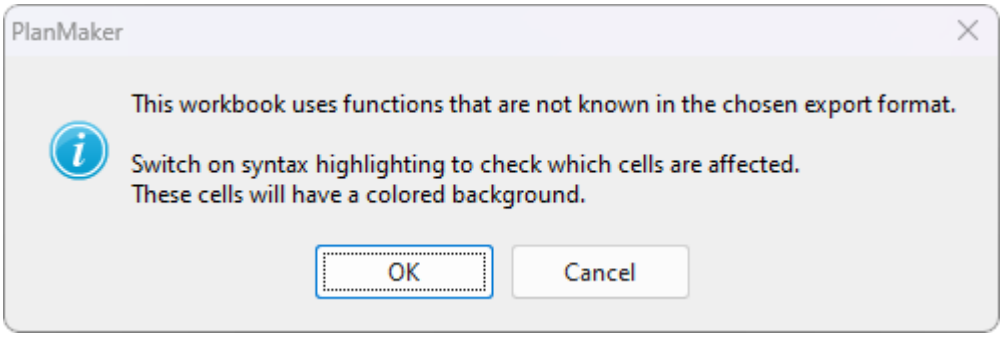
1. Choose the ribbon command **View | Syntax highlighting** to enable syntax highlighting.
2. All cells containing formulas for which the conversion failed will be highlighted with a colored background. These cells must be revised manually.

   Compare its contents with the contents of the corresponding cell in the Excel document. Try to replace the incompatible arithmetic functions with PlanMaker functions. For more information on all arithmetic functions supported by PlanMaker, see Functions from A to Z.
3. When all cells are revised, disable syntax highlighting.

Save the document (in PlanMaker format) to finalize the changes.

Warning message issued when saving a document in Excel format

When you save a PlanMaker document in Excel format and the document contains formulas that are incompatible with Excel, the following warning message will be displayed:



Then proceed as follows:

1. Choose the ribbon command **View | Syntax highlighting** to enable syntax highlighting.
2. All cells containing formulas that are incompatible with Excel will be highlighted with a colored background and must be revised manually. All arithmetic functions that are incompatible with Excel should

be replaced with Excel functions if possible. For more information, see the Help section for the respective arithmetic function.

3. When all cells are revised, disable syntax highlighting.

Save the document in Excel format again. If no further warning messages are displayed, all incompatibilities have been eliminated.

Differences between PlanMaker and Excel

This section provides a list of the most significant differences between PlanMaker and Excel:

General differences

- PlanMaker is unable to execute **macros** and **VBA scripts** stored in Excel documents. If you open an Excel file that contains macros or scripts, they will be ignored.

  However, PlanMaker will not remove them – if the file is saved in Excel format, the macros and scripts will remain functional.

Arithmetic functions

- Some of PlanMaker's **arithmetic functions** are not supported by Excel (and vice versa). If you open or save a document in Excel format, a warning message will be displayed if the file contains incompatible arithmetic functions. For more information, see Opening and saving Excel files.

Notes on the HTML format

PlanMaker documents can also be saved in HTML format.

To do so, proceed as follows:

1. Choose the ribbon command **File | Save as**.
2. Select the **HTML** format from the dropdown list **Save as type**.
3. For **File name**, enter the name under which the file is to be saved and confirm saving.

The document is now saved in HTML format.

All formatting not supported in HTML format will be automatically removed. This means the following:

- Formatting that the HTML format does not recognize will not be saved.
- Furthermore, HTML files are not able to perform calculations on their own. Thus, when saving in HTML format, PlanMaker calculates the entire worksheet and replaces all calculation formulas with their results.

Tip: After saving, open the document with your web browser to check its final appearance.

Working with Arabic text

In this chapter, you will learn everything you need to know about editing *Arabic* text in your PlanMaker documents. It is divided into the following sections:

- Enabling extended support for Arabic text
- Changing the text direction in table cells
- Changing the text direction in text frames
- Changing the direction of worksheets

For more information, see the following pages.

Important: The functions described in this chapter are only available if you have enabled the option **Extended support for Arabic text** in the program settings. For more information, see the next section.

Enabling extended support for Arabic text

PlanMaker has "extended support for Arabic text", which provides some additional functions for editing text in Arabic font.

To enable this option, proceed as follows:

1. Choose the ribbon command **File | Options** .
2. In the dialog box, switch to the **General** tab.
3. Enable the option **Extended support for Arabic text**.

Important: The functions described in this chapter are only available if this option is enabled.

Changing the text direction in table cells

If the option **Extended support for Arabic text** is enabled in Options , you can switch the text direction between left-to-right and right-to-left in a table cell at any time.

Proceed as follows:

1. Right-click on the desired cell to open the context menu and select the entry **Cell**.

2. In the dialog box, switch to the **Alignment** tab.
3. Under **Text direction**, select the desired option: context-sensitive, left-to-right or right-to-left.

Tip: With the default setting **Context-sensitive**, PlanMaker automatically chooses the appropriate text direction: When you enter text in Arabic into a cell, the text direction is set to right-to-left, otherwise, it is set to left-to-right.

Changing the text direction in text frames

In *text frames*, you can switch the text direction for the current paragraph between left-to-right and right-to-left at any time if the option **Extended support for Arabic text** is enabled in Options.

Proceed as follows:

1. Place the text cursor in the desired paragraph within the text frame.
2. Right-click to open the context menu and select the entry **Paragraph**.
3. Switch to the **Paragraph** tab.
4. Under **Text direction**, enable the option **Right to left** to change the direction to right-to-left.

   To set the text direction to left-to-right, disable the option.

Tip: Using the keyboard

You can change the text direction of a paragraph in a text frame much faster with the following shortcut keys:

- If you press **Ctrl** and the *left* **Shift key**⇧ at the same time, the text direction is set to left-to-right.
- If you press **Ctrl** and the *right* **Shift key**⇧ at the same time, the text direction is set to right-to-left.

Mac/Linux: On some systems (including macOS and several Linux distributions), these shortcut keys do not work. Alternatively, you can use the shortcut keys **Ctrl+Shift+A** (for left-to-right) and **Ctrl+Shift+D** (for right-to-left).

Changing the direction of worksheets

If the option **Extended support for Arabic text** is enabled in Options, you can change the text direction of a worksheet at any time.

To do so, proceed as follows:

1. Choose the ribbon command **Insert** | group **Tables** | **Sheet** | **Properties** in the desired worksheet.

   Alternatively: Right-click on the worksheet in the worksheet register, and select **Properties**.

2. Under **Sheet direction**, enable the option **Right to left** to change the direction to right-to-left.

The direction of the worksheet then changes accordingly.

Thus, if you change the direction of a worksheet to right-to-left, the column on the far *right* becomes the first column (instead of the column on the far *left*).

This means that the table is mirrored horizontally, as shown in the following figure:

A1	A2	A3
B1	B2	B3
C1	C2	C3

A3	A2	A1
B3	B2	B1
C3	C2	C1

Left: Table with sheet direction set to left-to-right
Right: Table with sheet direction set to right-to-left

Note: The text direction of each individual table cell can still be defined individually, of course (as described in Changing the text direction in table cells).

Scripts (BasicMaker)

Note: *BasicMaker* is only available for Windows and is not included in all versions of SoftMaker Office.

You can use the *BasicMaker* program to write and run scripts written in the programming language *SoftMaker Basic*. *SoftMaker Basic* is based on *Visual Basic for Applications (VBA)*, the programming language of Microsoft Office.

Scripts can be used to automate workflows in the spreadsheet application PlanMaker and the word processing application TextMaker.

Starting BasicMaker

To start BasicMaker or to run a BasicMaker script, use one of the following procedures:

- **Starting BasicMaker via the Start menu**

  You can start BasicMaker by clicking on the **Start** menu (the Windows logo button) in the lower left corner of the screen and choosing **SoftMaker Office > BasicMaker**.

- **Starting BasicMaker from within TextMaker/PlanMaker**

  You can also start BasicMaker from within PlanMaker or TextMaker.

  To do so, choose the ribbon command **File | Edit script** in PlanMaker/TextMaker.

- **Running a script directly from within TextMaker/PlanMaker**

  In PlanMaker/TextMaker, choose the ribbon command **File | Run script** to run a script. A file dialog appears. Select the script to be run, and confirm with **OK**.

All further information about BasicMaker and how to "program" PlanMaker and TextMaker can be found in the help of BasicMaker, which can be accessed via the **?** menu in BasicMaker.

Working with multiple documents

Android/iOS: This function is not available for *smartphones*. For *tablets*, multiple documents can be opened in the same program window.

You can have multiple documents open at any time – to compare them, edit them at the same time, exchange data between them, etc.

You work with multiple documents as follows:

- **Opening multiple documents in the same program window**

  You can open any number of documents at the same time in the current program window.

  There are many ways to switch between these documents, including a bar that contains *tabs* for each currently open document (displayed above the current document):

- **Opening files in a new program window**

  You can also open documents in a new program window.

  To do so, select the option **New program window** in the dialog box of the command **File | New** or of the command **File | Open** when you create/open the document.

  Note: To open the dialog box of the commands **File | New** or **File | Open**, click directly on their actual icons and not on their small arrow.

The methods can be combined as desired. You can open as many program windows as you like and open any number of documents as tabs in each of these windows. You can also open all documents in just one program window. Alternatively, you can ensure that each document appears in its own program window. It is entirely your choice.

In this chapter, you will learn everything you need to know about working with multiple documents at the same time. For more information, see the following pages.

Creating or opening a document

To create a new document or to open a document, proceed as follows:

- To create a new document, choose the ribbon command **File | New** .

  Note: The option "New program window" (see below) can only be accessed via the dialog box: To do so, click on the icon itself. If you click on the small arrow of the icon, however, the document templates used so far will be displayed for quick access.

- To open an existing document, use the ribbon command **File | Open** .

  Note: The option "New program window" (see below) can only be accessed via the dialog box: To do so, click on the icon itself. If you click on the small arrow of the icon, however, you can choose from the most recently opened files.

These two commands **New** and **Open** are described in detail in Basics.

Option "New program window"

The option **New program window** in the dialog box of these two commands has the following significance: When it is enabled, the document appears in a new program window. If it is disabled, the document will be opened with a new tab in the existing program window.

This setting is also saved and is applied each time a document is opened, both when new documents are created and when existing documents are opened.

(Note: In the Android/iOS version for *smartphones*, a separate program window is always opened, while in the version for *tablets*, a new tab is always opened in the same program window.)

Switching between open documents

Android/iOS: This function is not available for *smartphones*. For *tablets*, you can switch between document tabs within the same program window.

If you have multiple documents open at the same time, you can switch to a specific document and thus make it the active document:

- **Ribbon interface: "Windows" icon**

  In the ribbon, you will find an icon called **Windows** on the ribbon tab **View**. If you click on it, a list of all open documents appears. Click on the desired document.

- **Classic menu interface: "Window" menu item**

  For classic menus, you will find a list of all currently open documents in the **Window** menu. Select the entry for the document to which you want to switch.

- **Toolbars interface (Android/iOS only): "Window" menu item**

  You will find a list of all currently open documents for *tablets* via the hamburger menu in the **Window** menu. Select the entry for the document to which you want to switch.

- **Mouse: Clicking on a document tab**

  Above the document, you will find a bar with *document tabs* for all currently open documents. Click on the tab for the document to which you want to switch.

- **Mouse: Clicking into the program window containing the document**

  Of course, if you have documents open in multiple program windows (see previous section), you can also simply click into the program window with the desired document to make this document the active document.

- **Keyboard: Ctrl+F6**

  You can also use the keyboard to switch between the currently open documents: **Ctrl+F6** switches to the next document and **Ctrl+Shift+F6** switches to the previous one.

  Mac: On a Mac, you have to use the key combinations **Cmd+F6** or **Cmd+Shift+F6** to switch between documents.

Arranging documents on the screen

Android/iOS: This function is not available for *smartphones*. For *tablets*, document tabs can be moved within the program window.

If you have multiple documents open, you can arrange them as follows:

- The document tabs (the *tabs* in the bar above the document) can be moved within the bar and thus arranged in any order.

  You can even drag a document tab out of the program window with the mouse in order to have a separate program window for this document.

  Alternatively, you can drag a document tab from one program window to another. The document will then be integrated into the tab bar of the other program window.

- If documents are open in different program windows, you can move the individual program windows manually and resize them, of course – just as you can with all types of program windows.

- If you choose the ribbon command **View | Windows | View side by side** (in the classic menus interface: **Window > View side by side**), you can display the current document side by side with another open document on the screen. If more than two documents are currently open, the program first asks which of them is to be used in the other window.

Closing a document

Android/iOS: Not all of the listed methods are available in these versions.

If you want to stop working on a document, you can close it at any time. If the document has been modified since it was last saved, the program automatically asks you if you want to save the document first.

To close a file, you have the following options:

In the ribbon interface

The ribbon tab **File** contains a **Close** icon which closes the current document.

Tip: Use the ribbon command **View | Windows** | **Close all** to close all currently open documents at once.

In the classic menu interface

Choose the menu command **File > Close** to close the current document.

Tip: Use the menu command **Window > Close all** to close all currently open documents at once.

In the toolbars interface (Android/iOS only)

In the group **File**, choose the command **Close**.

Tip: For *tablets*, you can close all currently open documents at once via the hamburger menu by choosing the menu command **Window > Close all**.

Tabs for documents

Above the document, you will find a bar with *document tabs* for all currently open documents. If you click on one of these tabs with the *middle* mouse button, the corresponding document is closed.

Alternatively, left-click on the **x** icon displayed on the far right of each tab to close it.

Furthermore: If you *right-click* on a tab, a small menu appears with commands for closing the current document **(Close tab)** or all other open documents **(Close all other tabs)**.

Via the keyboard

You can also use the shortcut key **Ctrl+F4** or alternatively **Ctrl+W** to close the current document.

Mac: On a Mac, you have to use the key combination **Cmd+F4** or **Cmd+W** for this purpose.

Customizing PlanMaker

PlanMaker lets you determine a variety of program settings so that you can customize the program to suit your personal work style.

In this chapter, you will learn everything you need to know about it. It is divided into the following sections:

- **Changing PlanMaker settings**

  You can use the ribbon command **File | Options** to change the *program settings*. These settings apply to the program as a whole and thus to *all* documents.

- **Changing document properties**

  The *document properties*, on the other hand, are settings that *only* affect the current document and are also saved in this document. They can be changed with the command **File | Properties**.

- **Changing worksheet properties**

  Ultimately, the *worksheet properties* are settings that can be set differently for each worksheet in a document. They can be changed with the ribbon command **Insert** | group **Tables | Sheet | Properties**.

- **Customizing the document display**

  This section describes how to customize the appearance of the document on the screen. Most of the required commands can be found on the ribbon tab **View**.

- **Customizing the ribbon**

  Only for "Ribbon": This section is relevant only for users who have selected in the settings that they want to operate the program with the "ribbon".

  Here, you will learn how to configure and customize the *ribbon*.

- **Customizing toolbars**

  Only for "Classic menus with toolbars": This section is relevant only for users who have selected in the settings that they would prefer to operate the program with "Classic menus with toolbars".

  Here, you will learn how to configure the *toolbars* (Standard toolbar, Formatting toolbar, etc.) with the menu command **View > Toolbars** and how to customize them to your liking with **Tools > Customize**.

- **Creating user-defined icons**

  You can create *user-defined icons* at any time and add them to the ribbon or to a toolbar. Such icons can be used to start any program.

- **Customizing shortcut keys**

  You can also customize the shortcut keys for choosing frequently used commands. The ribbon command **File | Customize** is responsible for this purpose.

- **Editing lists for automatic filling**

  In the dialog box of the ribbon command **Home** | group **Contents | Fill | Series**, you can create your own lists for the **Fill** function.

- **Installing dictionaries**

  In this section, you will learn how to install dictionaries for the spell check.

For more information on these topics, see the following pages.

Changing PlanMaker settings

General settings can be changed with the ribbon command **File | Options** . These settings apply to *all* documents.

The options in the dialog box of this command are distributed by topic across multiple tabs. If you click on the **Files** tab header, for example, you can make settings for opening and saving files.

The following pages contain detailed information on all available settings.

Settings, View tab

On the **View** tab in the dialog box of the ribbon command **File | Options** , you can make some settings for the appearance of spreadsheets on the screen:

Markers

Here, you can change the appearance of the small triangles that are displayed in cells that contain a comment or are protected.

- **Color of comment markers**

  Here, you can change the color of the small triangle that is displayed inside cells that contain a comment.

- **Color of protection markers**

  Here, you can change the color of the small triangle that is displayed in all cells that are not protected when sheet protection is enabled.

- **Size**

  This option allows you to change the size of these markers.

Snapping

Here, you can configure the available aids for positioning objects (text frames, picture frames, etc.) with your mouse:

- **Snap to cells**

  This option is disabled by default. Thus, you can move objects freely with the mouse.

  If you enable this option and try to change the position of an object with your mouse, however, the object jumps from one table cell to the next.

Enabling or disabling this function temporarily: If the option **Snap to cells** is enabled, you can temporarily disable it at any time by pressing and holding down the **Alt** key while moving an object with the mouse. The reverse is also possible: If the option is disabled, it can be temporarily enabled with the **Alt** key while moving.

- **Snap to other objects**

  When this option is enabled, the software automatically creates "magic" guides for the edges of *each* object in the current worksheet. These guides make it a lot easier to position objects perfectly aligned and evenly distributed.

  These "magic" guides are usually invisible. They only become visible when you drag an object near such a guide with the mouse. As soon as the guide appears, the object automatically snaps to it. Something similar happens when you resize an object with your mouse.

  Some examples of such guides:

  If you drag an object to a position where it is *roughly* left-aligned with another object, a guide will automatically appear representing the left edge of the other object. If you let your object snap to it, it will be *exactly* left-aligned with the existing object.

  The same applies to the right edge of objects.

  If you drag an object to a position where it is *roughly* centered with another object, a guide will automatically appear representing the center of the object. If you let your object snap to it, it will be aligned *exactly* centered with the other object.

  If you drag an object to a position below two other objects, you will also see a horizontal guide at the position where this third object would have exactly the same relative distance as the other two objects. (The three objects would then be distributed evenly.)

  All of this works in both directions (horizontally and vertically) – for example, a magic guide is generated for the top and bottom edges of each object.

  Tip: If you hold down the **Alt** key while moving the object, you can temporarily disable the option **Snap to other objects** that is generally enabled.

- **Snap tolerance**

  Here, you can change the tolerance for the option **Snap to other objects**. The higher the set value, the quicker a moving object will snap to one of the "magical" guides for existing objects.

  The default value is a distance of 8 pixels.

Paint grid behind cells

When you enable this option, the gray gridlines displayed between the table cells are painted in the background of the cells (and no longer in the foreground). As a result, the gridlines will no longer be visible for cell ranges that have a colored background.

Cell marker

Here, you can change the color and line width of the *cell frame* (the dark frame that indicates the currently active cell in a worksheet).

Worksheet tabs

The *worksheet register* at the bottom of the document window displays tabs for each worksheet in the current document.

This option allows you to change the font size used to display the worksheet name on these tabs.

Highlight (only available in SoftMaker Office NX Home and NX Universal)

Highlight current column and row: If you enable this option, the column and row for the currently selected cell are highlighted in color. This gives you a better idea of where the selected cell is currently located on the worksheet.

Note: The highlighting only appears if a single cell is selected and not if several cells are selected.

You can change the color of the highlighting by using the dropdown below.

Settings, General tab

On the **General** tab in the dialog box of the ribbon command **File | Options**, you can make general settings:

Maximum number of undo steps

Here, you can set how many actions can be undone with the command **Undo** (in the Quick access toolbar). You can increase this number up to a maximum of 999 undo operations.

> **Note:** On devices with little main memory, it is not recommended to increase this value.

For information on the command **Undo**, see Undoing changes.

"User info" button

Here, you can enter your name, address and other contact information. To insert this information in documents, use the USERFIELD function.

Furthermore, the data typed in the "Name" field will appear when comments are added.

Enable dictation support

Mac only: If you enable this option, you can make better use of the macOS dictation feature in PlanMaker. However, you must expect the program to run more slowly as a result.

Extended support for Asian fonts

Note: Enabling this option only affects the font selection in *text frames*. In *table cells*, only one font can be selected at a time.

If this option is enabled – for text in *text frames* only – *three* lists for selecting font, font size, font style and language appear on the **Font** tab in the dialog box of the command group **Home | Character**:

- For Latin scripts (for example, German, English)
- For East Asian scripts (Chinese, Japanese, Korean)
- For complex scripts (for example, Arabic)

Thus, you can make the settings for each type of script separately.

If, for example, you set the font to "Arial" for Latin scripts and the font to "SimHei" for Asian scripts, all Latin scripts that you type will automatically appear in Arial, and all Asian scripts will appear in SimHei.

PlanMaker recognizes automatically whether the characters entered are Latin, Asian or complex (for example, Arabic) scripts.

Extended support for Arabic text

If you enable this option, some additional features will become available for editing text in *Arabic* script. For more information, see Working with Arabic text.

Select whole words when selecting

Windows, Mac and Linux only: If this option is enabled, PlanMaker automatically extends any selection that you make with the mouse to whole words. If it is not enabled, the selection is extended on a character-by-character basis.

Add or remove spaces when pasting

If this option is enabled, spaces before and after words are automatically added or removed when you delete, cut, or paste words from the clipboard.

If, for example, you select the word "is" in the sentence "This is a test" and cut it with the command **Cut**, PlanMaker automatically deletes the space behind "is" that is now superfluous.

If you now insert the word "is" before another word using the command **Paste**, PlanMaker automatically inserts a space after "is".

Middle mouse button

Mac and Linux only: Here, you can specify what should happen when you click somewhere using the *middle* mouse button. Available options:

- **No function:** The middle mouse button doesn't have any function.

- **Paste selected content:** If you click in the document with the middle mouse button, a copy of the currently selected content is inserted at this point (for example, the currently selected text).

Apply character formatting to entire words

Enabling this option has the following effect:

If the text cursor is inside a word and you change the character formatting (for example, select a different font or enable bold), this will affect the entire word.

This allows you to quickly change the formatting of a single word without first having to select it.

Use the Insert key to toggle overwrite mode

This option is enabled by default and allows you to use the **Ins** key on your keyboard to switch between insert mode and overwrite mode (see also Status bar).

If you remove the check mark in front of the option, this typical function of the **Ins** key will be disabled when working in PlanMaker. This prevents accidental switching to overwrite mode.

Insert frames and drawings immediately

This option determines what should happen if you insert a frame or a drawing into a document (text frame, picture frame, AutoShape, etc.) – for example, with the ribbon command **Insert | Text frame**:

- **Always:** The corresponding object is inserted immediately. Its position and size are fixed without further action.

  (Of course, you can always change the object's position after insertion by simply dragging it to the desired location with the mouse. You can also resize it by dragging one of the round handles that surround the object.)

- **Never:** Before the object is actually inserted, the mouse cursor changes to a crosshair. In order to proceed, drag a rectangle in the document to specify the desired position and size. The object will then be inserted exactly according to your specifications.

- **Auto:** This is a smart combination of the two aforementioned options that is already preset in PlanMaker for each specific object type. Objects such as charts or picture frames are inserted directly (as with **Always**), for example, while objects such as AutoShapes or text frames are inserted via the mouse cursor (as with **Never**).

"Auto correction" button

PlanMaker can automatically correct common typos and also provides other useful auto correction functions. If you click on the **Auto correction** button, the program opens an additional dialog box in which you can make the appropriate settings.

Tip: The ribbon command **Review** | group **Spelling** | **Auto correction** can also be used to open the auto correction dialog box.

For more information on this topic, see Auto correction.

Use left Alt for shortcut keys

Mac only: This option determines which function the left **Alt** key on your keyboard should have:

- **Off:** The left **Alt** key does the same as the right **Alt** key: It inserts icons into the text.

  Alt+E, for example, returns a euro sign €.

- **On:** The left **Alt** key can be used to open menus in the classic menu.
  Note: This only works if you use the program with the classic menu interface instead of the ribbon interface.

  Here, **Alt+E** no longer returns a euro sign, but opens the **Edit** menu instead.

Show warning when loading OLE objects

For Mac, Linux and Android/iOS: If this option is enabled, the program displays a warning that OLE objects cannot be displayed when you open a document containing such objects. (OLE objects can only be displayed on *Windows*.)

Settings, Edit tab

On the **Edit** tab in the dialog box of the ribbon command **File | Options** , you can change settings for editing worksheet cells:

After input, move selection ...

This option determines the direction in which the *cell frame* (the dark frame that indicates the current cell on the screen) moves when you type something in the cell and press the **Enter key**↵.

Height of the input field

Here, you can set whether the single-line input field for formulas in the Edit toolbar should expand so that you can see the complete content even for long formulas. For more information, see Edit toolbar.

In-cell editing

This option is enabled by default. This means that you can edit the contents of a cell directly in the worksheet after pressing the **F2** key. If you disable this option, the cell contents can only be edited in the Edit toolbar (displayed above the spreadsheet) after pressing **F2**.

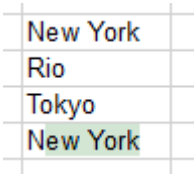
Automatic percent input

This option determines how PlanMaker reacts when you type numbers into cells that contain percentages or are formatted with the "Percent" number format. If enabled, a percent sign is automatically added to your input.

Autocomplete cells

While you type text in a cell with this option enabled, PlanMaker automatically makes suggestions from the existing text entries found in the cells above and below this cell.

Example: You have entered the values "New York", "Rio", and "Tokyo" in a column. If you now go to the cell below these entries and type the letter "N", the program will automatically suggest "New York". If you type the letter "T", "Tokyo" will be suggested, etc.



To accept the suggestion, press the **Enter key**↵. To reject it, simply continue typing or press the **Del** key.

If you do not want PlanMaker to make suggestions while you are typing, you can always disable this option.

Left arrow key never exits cell editing

Use this option to specify what should happen when you navigate to a cell, type in a value and press the left arrow key ←:

- **On:** The text cursor is moved to the left (within your input).
- **Off:** The cell frame is moved to the left. This means that the value you enter is accepted, and the cell to the left of the current cell is made the active cell.

Warning if a formula contains errors

If you enter a formula that contains an error (for example, a closing parenthesis is missing), PlanMaker issues an error message when you press **Enter**↵. If you do not want that to happen, disable this option.

Note: Regardless of this option, incorrect formulas are generally replaced by the text #NULL! when you save the document.

Formula tooltip

If this option is enabled and you enter an arithmetic function in a cell, a tooltip providing information about the required function parameters appears below the cell.

Expand tables automatically

As described in the section Tables in worksheets, the ribbon command **Insert | Table** enables you to create "Tables in worksheets".

This option has the following effect on such tables:

- **On:** When you enter something in one of the cells directly to the right of the table, the table is automatically expanded by another column. The same applies to rows: When you type something in a cell directly below the table, another row is added automatically.
- **Off:** The table is not expanded automatically.

Number separators

Here, you can set which characters to use as the decimal separator and thousands separator for numbers.

Explanation: The *decimal separator* is the character that is placed in a number with decimal places (for example, the period in 17.50). The *thousands separator* is the character that can be used to group long digits (for example, the commas in 1,000,000).

- If you select **System default**, PlanMaker automatically determines the decimal separator and thousands separator from your current system settings. (This is the default setting.)
- If you switch to **Custom**, PlanMaker lets you specify a custom decimal separator and thousands separator.

> **Warning:** If you specify custom settings that are incorrect, calculations may return invalid or wrong results!

Calculation in status bar

If you select several cells, their sum is automatically displayed in the status bar per "Online calculation". However, you can also specify other calculation methods here. You can even select several calculation options, the results of which are then displayed next to each other in the status bar.

Tip: You can also set the options for "Calculation in status bar" directly in the status bar itself. To do so, click in the status bar – at the point where the result of the calculation is displayed.

Settings, Appearance tab

On the **Appearance** tab in the dialog box of the ribbon command **File | Options** , you can make settings that affect the appearance of the PlanMaker user interface:

Dialog language

Here, you can select the language in which the user interface (ribbon, menus, dialog boxes, etc.) is to be displayed.

"User interface" button

If you click on the **User interface** button, the program displays a dialog box in which you can select the type of user interface that the SoftMaker Office applications should use:

- **Ribbon** (top row)

  If you select one of the styles from the top row, the programs use a "ribbon" as the user interface. The only difference between the individual styles is in the color scheme used: colorful, light grey or dark.

- **Classic menus with toolbars** (bottom row)

  If you select one of the styles from the bottom row, the programs will use classic menus with toolbars. You can also choose between different color schemes here.

Android/iOS: If you are using these versions, select the **user interface** via the dropdown list. Here, you have an additional option: **Toolbars**. For more information, see User interface on Android/iOS. The options *Quick access toolbar* and *Touch mode* can be found separately elsewhere on this **Appearance** tab and work as described below.

Furthermore, the following settings can also be made for the user interface:

- **Quick access toolbar** (only in the ribbon interface)

  This option determines where the *Quick access toolbar*, which contains icons for some of the most frequently used commands, should be displayed: to the left of the document tabs – or in a separate toolbar directly below the ribbon.

- **Touch mode**

  If you enable this option, all icons in the ribbon user interface or menu interface and toolbars will be slightly enlarged. This is useful when operating the software with your finger (for example, on a tablet).

  Tip: Alternatively, you can also enable/disable touch mode with the following commands:

  Ribbon user interface: Here, you choose the command **Touch mode** in the Quick access toolbar.

  Classic menu interface: Here, you choose the command **View > Touch mode**.

- **Update dark and light design automatically from the system**

  If you enable this option, the style of the user interface (light or dark) will be based on the color settings of your operating system. Thus, if you operate your device in dark mode, the SoftMaker Office applications will automatically switch to this mode also.

  (This option is not available in the Linux version.)

"User interface size" button

This button opens a dialog box that allows you to reduce or enlarge the user interface (menu, toolbars, dialog boxes, etc.).

If you select a lower setting here, all user interface elements will be reduced, which will allow you to see more elements at a time. A higher setting, on the other hand, increases legibility.

Use the mouse (or your finger) to drag the **Size of dialog elements** slider to gradually change the size of the user interface.

The **Sample** field acts as a small tool to help you find a suitable setting: It gives you a preview of how a checkbox in a dialog box or even in the Quick access bar will look after you have applied your changes.

Note: Inappropriate settings may cause the user interface to be far too small or too large, especially on devices with a relatively low resolution. If this happens, you can always click on the **Reset to default** button in the lower left of this dialog box to return to the default settings.

The option **Touch mode** only enlarges the icons (see also description above). Incidentally, you will also find this function directly in the Quick access bar.

Document scaling (DPI)

You can scale the document higher or lower for a larger/smaller view. To do so, select *Custom* and enter the desired value.

Show full path in title bar

Enable this option if you want the title bar to display not only the name of the document but also its file path. (Not available in the Android/iOS version.)

Show status bar in ribbon mode

This applies only to the *ribbon interface*: You can enable/disable the display of the status bar here. To change this for the *classic menu interface*, see Showing and hiding toolbars.

Show fonts in font list

If this option is enabled, PlanMaker displays all font names in the respective font in font lists (for example, in the font list of the command group **Home | Character** in the ribbon or in the Formatting toolbar in the classic menu interface). This lets you see the actual appearance of each font at a glance.

Show tooltips

This option lets you specify whether or not *tooltips* should be displayed. These are short info texts that are displayed next to the mouse cursor when you hover over a screen element (for example, a button in the ribbon or in the Standard toolbar for the classic menu interface).

Beep on errors

If this option is enabled, PlanMaker emits a beep for error or warning messages.

Use system file dialogs

This option controls the type of dialog boxes that appear when you use commands for opening and saving files. The options are:

- **Disabled:** PlanMaker uses its own file dialogs.
- **Enabled:** The default file dialogs of the operating system appear, which you know from the majority of applications.

The former are easier to use; the latter offer more options.

Note for users of the **iOS** version: The system file dialogs are always used automatically here.

Note for users of **Linux**: You must be running the **Gnome** desktop environment so that this function is available. (This automatically means you have the Gtk libraries installed).

Live preview

Here, you can enable/disable a live preview for the dropdown lists that are located, for example, in the command group **Home | Character**.

If this is enabled, for example, you can select text, open the font list in the command group **Home | Character** and then hover over the fonts in it. The document immediately shows how the selected text would look "live" in the corresponding font.

To actually apply the font, simply click on it. To discard the change, click anywhere outside the font list or press the **Esc** key.

Smooth edges of screen fonts

If this option is enabled, PlanMaker uses a technology called "anti-aliasing" to smooth the edges of fonts and improve their appearance on the screen.

Different options are available depending on the operating system.

Workspace color

This option allows you to change the background color of document windows (this option is especially useful for TextMaker and Presentations, but also suitable for PlanMaker, for example, for print previews).

Measurement

Mac, Linux, Android/iOS only: Here, you can specify the units of measure which PlanMaker should use by default: **Metric** (centimeter) or **U.S.** (inch).

(On Windows, this option is not required because the program automatically uses the setting that was selected in the options "Region and Language" of the Windows Control Panel.)

Tip: You can always override this setting and enter a value into dialog boxes in another unit of measure by typing one of the following unit of measures after the value:

Unit	Explanation
cm	Centimeter
in	Inch – 1 in equals 2.54 cm.
pt	Point – 72 pt equal 1 in.
pi	Pica (character) – 1 pi equals 12 pt

If, for example, you type "5.08 cm" for the left margin, PlanMaker sets it to 2 inches (5.08 cm).

Desktop mode

Android/iOS only: If you are using a tablet, you should enable this option so that PlanMaker is displayed in an optimized manner on tablet devices. If you are using the application on your smartphone, leave this option disabled.

Settings, Language tab

On the **Language** tab in the dialog box of the ribbon command **File | Options** , you can make settings for PlanMaker's spell check and hyphenation:

Language

Here, you can select the language to be used for the spell check and hyphenation. To do so, open the dropdown list and select the desired language from the available options.

Note: Please note that there are *two* entries in this list for German:

- **"German (Germany)"** stands for the *new* spelling (according to the 1996 German spelling reform in its last revision in 2006).
- **"German (Germany, old)"**, on the other hand, stands for the old spelling.

The same applies to "German (Austria)" and "German (Switzerland)".

Spell check

Here, you can make the following settings for the spell-check:

- **Check spelling as you type**

  If you enable this option, PlanMaker will check the spelling of text as you type. As soon as you make a mistake, a dialog box will appear to facilitate a correction. For more information, see Checking the spelling as you type.

  Note: If this option is disabled, you don't have to do without a spell check – you can always start it manually with the ribbon command **Review | Spell check**. For more information, see Manual spell check.

- **Beep on errors**

  This option determines whether PlanMaker should beep when **Check spelling as you type** encounters an unknown word.

- **Ignore words that start with a number**

  If you enable this option, the spell check ignores all words that start with a number (for example, "1980s").

- **Manage dictionaries**

  As an alternative to its own dictionaries, SoftMaker Office also supports the use of *Hunspell* dictionaries. They are available free of charge for installation. Most of them were created by private users or non-profit organizations.

  The **Manage dictionaries** button allows you to install dictionaries and also specify which of these dictionaries should be used for a language. For more information, see Installing dictionaries.

Settings, Files tab

On the **Files** tab in the dialog box of the ribbon command **File | Options** , you can make settings for opening and saving files:

Template folder

Here, you can specify the folder in which the templates for PlanMaker documents are stored.

The dialog box that appears when you choose the ribbon command **File | New** to let you select a document template displays all templates that are in this folder.

Note: Normally, this setting should not be changed. Change it only in the event that you have actually moved the template folder to another place.

Default file format

Here, you can select the file format in which PlanMaker should save newly created documents by default.

If you create a new document with the ribbon command **File | New** and then save it for the first time, the Save dialog box always suggests the file format set here as default.

You can choose between several versions of both the PlanMaker format and Microsoft Excel format.

Important: Note that the Excel format does not know *all* of PlanMaker's features. If you save a PlanMaker document in Excel format, certain formatting options and other features may be lost. We thus recommend that you use the default setting "PlanMaker" here.

Saving section

In the **Saving** section, you can make settings for saving documents:

- **Prompt for summary information when saving**

  If you enable this option, a dialog box will appear automatically the first time you save a new document to enable you to enter some additional information about it. For more information, see Document summary.

Opening section

In the **Opening** section, you can make settings for opening documents:

- **Recently used files in File menu**

  If you click on the small arrow on the ribbon command **File | Open** , PlanMaker displays a list of the most recently opened files. If you select one of these entries, the corresponding file will be opened immediately. Here, you can specify the number of files to be displayed in the list.

Android/iOS only: Backup section

The Android/iOS-Version does not contain a "Backup" tab and its settings, which are described in the next section. Instead, you have to choose either *No backup* or *Simple backup* in the dropdown list on the "Files" tab.

- **No backup**

  If you select this option, *no* backups will be created when documents are saved. (This option is not recommended.)

- **Simple backup**

  This makes *one* backup per document. It contains the previous version of the document. It has the name extension `.bak` and is in the same folder as the document itself.

The Android/iOS version has also the following option for the **Opening** section above.

- **Auto recovery: Save state every ... minutes**

  If this option is enabled, PlanMaker automatically makes a temporary backup copy of every opendocument on a periodic basis. You can enter a period of 1 to 100 minutes.

  Note: These temporary backup copies work completely independently of the *Simple backup/No backup* option described above.

  When you exit PlanMaker in the normal manner, these temporary backup copies are automatically deleted. However, if it is not terminated correctly (for example, due to a crash), PlanMaker will detect this the next time the program is started. It then opens (after prompting) the temporary backup copies of all documents that were open when the crash occurred and contained unsaved changes.

  You can then check each of the restored documents to determine if any of the most recently made changeshave actually been lost, and then save them.

Settings, Backup tab

Android/iOS: This tab is not available in the Android and iOS versions. However, *simple* backups can be made here. This setting can be found on the Files tab.

On the **Backup** tab in the dialog box of the ribbon command **File | Options** , you can make settings for creating backup copies:

"When you save a file manually..." section

- **Keep older versions of the document** (if you save a file manually)

  Several generations of file backups are created for each document. All of these copies are stored in a special **Backup** folder.

Tip: If this option is enabled, the command **File | Versions** (in the group **File management**) is also available, which allows you to easily return to a previous file version of the currently open document.

- **Number of versions to keep**

  Here, you specify the maximum number of versions of backups (generations) to be kept per document.

"Automatically created file versions (snapshots)" section

- **Save unsaved changes every ... minutes**

  If you want automatic "snapshots" to be created as an additional backup while the document is being edited, enable the checkbox here. In the dropdown field to the right, you specify the interval at which the snapshots are to be taken regularly.

- **Number of snapshots to keep**

  In this dropdown field, you specify the maximum number of snapshots you want to keep.

Folder for file versions

Here, you can change the path for the **Backup** folder in which all backups are stored.

"Clean up" button

This button provides two options for deleting file backups from the backup folder:

- **Delete orphaned file versions**

  This option removes all PlanMaker backups for which the corresponding original document no longer exists.

- **Delete file versions of all documents**

  This option removes *all* backups from PlanMaker.

For more information on using file backups, see Backup copies (file versions).

Settings, System tab

On the **System** tab in the dialog box of the ribbon command **File | Options** , you can make the following system settings:

Smooth edges of pictures

If this option is enabled, PlanMaker uses anti-aliasing to smooth edges and lines in pictures. Advantage: The display quality of scaled pictures is improved. Disadvantage: This procedure consumes some CPU time when displaying pictures, of course.

Enable OpenGL AutoShapes

If this option is enabled, PlanMaker uses the OpenGL graphics engine to display AutoShapes optimally. If you disable this option, 3D effects used for AutoShapes may no longer be displayed correctly. Thus, you should usually leave this option enabled.

Check for updates

Windows, Mac and Linux only: Here, you can specify how often you want the software to check whether there are updates for SoftMaker Office. Updates provide bug fixes and feature improvements.

Note: Checking for updates only works if your device can access the Internet.

You can also disable the automatic update check (not recommended) by selecting **Never** from the dropdown list. Checking for updates can then only be done manually – by clicking on the **Check now** button.

Tip: Subscribe to our free **newsletter** at www.softmaker.com to automatically receive an e-mail whenever updates are available. The newsletter will also provide you with other useful information and special offers with respect to SoftMaker Office on a regular basis.

Android/iOS: For applications installed from the Google Play/Apple App Store, your device will automatically notify you of updates (unless disabled in the settings of the respective store).

Associate file types

Windows only: This button opens a dialog box where you can associate certain file types with PlanMaker. This will make PlanMaker the *default application* for the corresponding file types.

If, for example, you link the file type "Microsoft Excel documents" with PlanMaker , files of this type will always be opened in PlanMaker when you double-click on such a file in Windows Explorer.

Show a message during program launch when PlanMaker is not the default program

Windows only: The aforementioned button enables you to set PlanMaker as the default program for opening certain file types. Over time, another application may assume the role as the default program for opening such file types.

If this option is enabled, a message will then appear the next time you open this file type in PlanMaker asking whether you want to set PlanMaker as the default program again. If this option is disabled, the message will no longer be displayed.

Show/hide keyboard automatically

Android/iOS only: If this option is enabled, the on-screen keyboard opens automatically when you can type something (for example, if you double-tap on a cell or tap into an input field in a dialog box).

Change keyboard layout automatically

Android/iOS only: If this option is enabled, the layout of the on-screen keyboard will automatically adapt to the current situation. If, for example, you type into an input field in a dialog box which only accepts numbers, the keyboard will automatically be switched to numeric mode.

Furthermore, when you enter values into table cells, PlanMaker will note the current mode of the keyboard if this option is enabled. If it is disabled, the keyboard will be reset to its standard mode (alpha-numeric) each time you enter data in a new cell.

Enable OpenGL engine

Linux only: If this option is enabled, the program uses the OpenGL graphics engine. This speeds up the screen display. Thus, this option should usually be enabled.

Note: On systems with a very outdated or faulty OpenGL installation, display errors may occur within the application. This depends on your individual system configuration and especially on the graphics driver you are using. In this case, disable this option and restart the application.

Furthermore: If the program detects problems with the installed OpenGL engine at startup, it will automatically disable this option.

Use SHM extensions

Linux only: This option is only available if the option **Use OpenGL engine** is *dis*abled (not recommended).

If the SHM extensions are enabled, the program uses shared memory (if available), which can speed up the responsiveness of the display.

Use XIM

Linux only: If this option is enabled, PlanMaker supports the composition of accented characters (à, ô, é etc.) using "XIM".

On a German keyboard, for example, the character "ô" can be created by first pressing the ^ key and then the **O** key.

If you disable this option, you can no longer compose characters in this way.

Note: On some systems, disabling this option improves the responsiveness of the display when you are typing text quickly.

Show hidden files and folders

Mac, Linux only: If this option is enabled, hidden files and folders are also displayed in file dialogs (for example, those of the ribbon command **File | Open**).

Lock files for exclusive write access

Mac, Linux only: If this option is enabled, some distributions of macOS and Linux can prevent a PlanMaker file from being changed by different programs at the same time. If the option is disabled, different programs can access a file at the same time.

User-defined font paths

Mac, Linux, Android/iOS only: Here, you can specify paths for additional fonts that PlanMaker should use. To do so, enter the subdirectory in which the font files are located. If you want to enter multiple directories, separate them with a colon.

PlanMaker can then use these fonts in addition to the fonts available in your system.

Note: Changes to this setting only become effective after the program is restarted.

Mouse wheel

Mac, Linux only: If your mouse has a mouse wheel, you can set the scrolling increment for a turn of the wheel either by page (one complete page) or by a certain number of lines.

External applications

Mac, Linux only: This button opens a dialog box where you can specify the external programs that start automatically for specific events. When a PDF file is created, for example, the application specified in the **PDF viewer** field will be launched to display the result.

Note: You don't *have to* fill in these fields. For all fields that you leave empty, your system's default application will be automatically used for this action. Make changes to this dialog box only if you want an application *other* than the default application to be used. To do so, simply enter the file name of the corresponding executable file (including the entire file path, if necessary).

Settings, Fonts tab

On the **Fonts** tab in the dialog box of the ribbon command **File | Options** , you can specify which fonts are to be displayed in font lists (for example, the font list on the ribbon command **Home |** group **Character**).

The tab presents a list of all fonts installed on your system. To enable or disable a font in the list, click on the checkbox in front of its name. All fonts for which you have removed the check mark are no longer displayed in font lists.

Background information: The fonts installed on your system by default include several fonts that you will probably never need – for example, fonts that only contain scripts for foreign languages that you don't use. If you like, you can hide such fonts as described above.

Windows: Use font filtering of the operating system

Windows users will also find the option **Use font filtering of the operating system** on this tab. If this option is enabled, Windows will automatically hide in font lists *all* fonts that only contain scripts for foreign languages (for example, Asian/Arabic fonts).

Mac: Load only fonts relevant for installed keyboard layouts

MacOS users will also find an option on this tab called **Load only fonts relevant for installed keyboard layouts**.

Explanation:

On macOS, there are numerous fonts installed on the system with foreign language characters that you usually do not need (for example, Asian/Arabic fonts).

If this option is enabled, such fonts are automatically hidden in the font lists. These fonts will continue to appear in the font lists only if you use appropriate keyboards for Asian/Arabic fonts or enable font support. For more information, see Settings, General tab.

If you disable this option, all system fonts are always available in the font lists.

"Manage" button

In the dialog box of the ribbon command **File | Options** , you will find the **Manage** button on each tab. You can use it to save the settings of the program and, for example, transfer them to a new version.

For more information, see the following pages.

Exporting/importing settings

You can save your individual settings that you have configured in the SoftMaker Office applications. In the first step, export the settings data – for example, before you install a new version. Then, in the second step, import this data into the newly installed version.

For both steps, use the **Manage** button in the dialog box of the command **File | Options** .

For information on which settings you can export/import exactly, see below.

Note: Export and import of settings data is only possible as of version 2021.

What reasons could there be to export/import settings?

The following reasons could be considered for exporting/importing your settings:

- You want to apply the settings of the old version to a new version of SoftMaker Office that you are installing on the same computer.
- You have also installed the current version of SoftMaker Office on another computer and want to use the same settings there.
- You want to apply the settings of an old version of SoftMaker Office to another computer on which you have installed the new version of SoftMaker Office.

Which files are backed up?

For the export, PlanMaker accesses the settings files (...config.ini, ...tools.dat, etc.) that are stored on your device in the `SoftMaker > Settings` folder. Depending on the operating system, these files are located in the following directory:

- **Windows:** In the `SoftMaker\Settings` folder in your `Documents` folder.
- **macOS:** In the `SoftMaker/Settings` folder in your `Documents` folder.
- **Linux:** In the `SoftMaker/Settings` folder in your `Home` folder.
- **Android:** In the `SoftMaker/Settings` folder on your SD card.
- **iOS:** In the `SoftMaker/Settings` folder in your `App` folder.

Note: The above information is for illustrative purposes only and you usually do not need to do anything here. PlanMaker will automatically replace these files for you if you perform the steps described below for exporting/importing the settings.

Step 1: Export the settings

To export settings, choose the command **File | Options** , and the "Options" dialog box will open. Here, click on the **Manage** button located at the bottom left of each tab of this dialog box.

This will open another dialog box "Manage settings for Office". Here, select which settings you want to save from the following options:

Tip: You can also just export all of them, but you will have to ensure when subsequently importing that you only select the options you really want.

- **Configuration files**

  This option saves all the settings you have made in the dialog box of the command **File | Options**. Some of these settings are also located directly on the ribbon tabs (or in the toolbars).

- **Customized ribbon/toolbars**

  If you have changed the arrangement on the ribbon, in the Quick access toolbar or in toolbars (see section Customizing the ribbon), you can save your individual arrangement with this option.

  Please note for the import of the settings from an old to a new version of SoftMaker Office: If this option is enabled, command icons that have been newly added in the more recent version will not be displayed there. Thus, if you would prefer the new icons to be displayed instead of keeping your old arrangement, you should disable this option. Alternatively, you can import your old arrangement anyway and look for the new symbols relevant to you afterwards using the command **File | Customize** (see section Customizing icons and groups in the ribbon).

- **Keyboard shortcuts**

  This option allows you to apply shortcut keys that you have assigned yourself (see Customizing shortcut keys).

- **User dictionaries**

  This option allows you to save the words that you have added to your user dictionaries (see section Editing user dictionaries).

- **Other (SmartText, Labels, ...)**

  Enable this option if you want to apply all other settings that can be saved in SoftMaker Office (for example, SmartText).

When you finally click on the **Export** button, a zip file will be created with the options selected above. Save this file to any location that you can easily access for subsequent import.

Note: The settings of all SoftMaker Office applications are exported (TextMaker, PlanMaker, Presentations, BasicMaker). The same applies to the import in the following step. In the **iOS** version, the export/import must be performed separately for each application.

Step 2: Import the settings

To transfer the settings to another installation of the program, choose the command **File | Options** there. In the dialog box "Options", click on the **Manage** button.

In the following dialog box "Manage settings for Office", insert a check mark in front of the settings you want to import. (Details of the options are described above in step 1.)

Now click on the **Import** button, and select the zip file that you created in step 1. The imported settings will take effect in the current PlanMaker application when you restart the program.

"Reset" button

The **Reset** button resets all program settings back to their delivery state.

Note: This action resets the settings for all SoftMaker Office applications (TextMaker, PlanMaker, Presentations, BasicMaker) of this version. For **iOS** users, the reset applies to each application separately.

Changing document properties

You can use the ribbon command **File** | group **File management** | **Properties** to change settings that only affect the current document and are saved with it. These settings are called *document properties*.

The available document properties are distributed by topic across multiple tabs. If you click on the **Summary** tab, for example, you can edit the document info.

The following pages provide detailed explanations of all available document properties.

Document properties, Summary tab

On the **Summary** tab in the dialog box of the ribbon command **File** | **Properties**, you can enter some additional info for the current document.

This can include explanatory information about the current document (subject, author, keywords, etc.) that the file manager can use in searches.

For more information, see Document summary.

Document properties, Colors tab

On the **Colors** tab in the dialog box of the ribbon command **File** | **Properties**, you can edit the color palette of the current document.

When you open a color list – for example, the color list for the font color – you see only a small selection of the up to 16 million colors available. This selection is called the *color palette* of the document.

You can modify this color palette at any time. You can add new colors and change colors that you have added. However, the first 24 colors in the palette are default colors that cannot be changed.

Important: Changes to the color palette are saved *in the document*. This means that you can create a separate color palette for each document.

Adding a color

To add a color to the palette of the current document, proceed as follows:

1. Choose the command **File** | **Properties** and switch to the **Colors** tab.

> ***Tip:*** Alternatively, this color palette can be accessed via any button that contains a color list (for example, font color). To do so, select the entry **Define color...** in the color list (the last entry).

2. Set the desired color. For more information, see "Working with the color controls" section below.
3. Click on the **New** button.
4. Give the new color whatever name you like and confirm with **OK**.
5. Exit the dialog box with **OK**.

PlanMaker now adds the new color to the document's color palette. From now on, the color will be available in all dialog boxes where a color can be selected.

Changing a color

> ***Note:*** You can only change colors that you have added. The first 24 colors in the color palette are default colors that cannot be changed.

To change a color, proceed as follows:

1. Choose the command **File | Properties** and switch to the **Colors** tab.
   Alternatively: You can also select the entry **Define color...** in any color list (for example, font color) to open this dialog box.
2. Select the color to be changed from the **Color palette** list.
3. Make the desired changes. For more information, see "Working with the color controls" section below.
4. Click on **Change**.

The color has now been changed for the current document.

Renaming a color

You can change the name of self-defined colors at any time. To do so, open the color dialog, select a color from the dropdown list **Color palette** and click on the **Rename** button.

Working with the color controls

The controls in the above dialog box let you set colors in many ways. All these ways lead to the same result – just select your preferred method:

- **Using the color field and luminance slider**

  The easiest way to set colors is to use the large color field and the luminance slider to its right. The color field presents all available hues in all available saturations, while the luminance slider next to it provides a means of varying the luminance.

  To set a color, first click on the desired color in the large color field. Then click on the desired luminance in the luminance slider.

- **Using the hue, saturation and luminance controls**

  Alternatively, you can set colors with the controls labeled **Hue**, **Sat** (saturation) and **Lum** (luminance). Values between 0 and 240 are allowed.

- **Using the Red, Green and Blue controls**

  Colors can also be set via their red, green and blue components. The controls labeled **Red**, **Green** and **Blue** are provided for this purpose. Values between 0 and 255 are allowed.

- **Typing the hex code for a color**

  You can also specify a color by typing its *hex code* in the input field with the hash (pound) sign #.

  Such hex codes are common in the HTML code of web pages, for example. The hex code consists of 3 two-digit hexadecimal values for the colors red, green and blue (RRGGBB). For example, 00FF00 represents pure green.

Document properties, Statistics tab

On the **Statistics** tab in the dialog box of the ribbon command **File | Properties** , you can retrieve statistical information about the current document:

- The **Cells** section displays how many cells are filled and how many of them contain text, numbers, formulas or comments.
- The **General** section displays the number of worksheets and the number of pages.
- The **Objects** section displays the number of objects (charts, pictures, etc.) in the current documents.
- The **Actions** section displays when the document was created, last saved and last printed.

Document properties, Options tab

On the **Options** tab in the dialog box of the ribbon command **File | Properties** , you can make general settings for the current document:

"Text frames" section

- **Tab width**

  This option determines the tab width for text entered in text frames.

"Cell contents" section

- **Syntax highlighting**

  If this option is enabled, cells will be colored according to their content. This corresponds to the ribbon command **View | Syntax highlighting**. For more information, see Syntax highlighting.

- **Show formulas**

  If this option is enabled, cells containing a calculation will display the formula instead of the result. This corresponds to the ribbon command **View | Show formulas**. For more information, see Showing formulas.

- **Protection indicator**

  If this option is enabled, a small green triangle will be displayed in all *unprotected* cells when sheet protection is enabled. This makes it easier to see which cells may still be edited. For more information, see also Sheet protection.

- **Hyphenation**

  If this option is enabled, automatic hyphenation is performed in all cells where the option **Wrap text** has been enabled in the dialog box of the command group **Home | Alignment** ◢. For more information, see also Hyphenation.

"Comments" section

- **Show indicators**

  If this option is enabled, a small red triangle is displayed in all cells that contain a comment (inserted with the command **Insert | Comment**). For more information, see also Using comments.

- **Always show comments**

  Normally, comments are displayed only when you hover with your mouse over a cell that contains a comment. If you enable this option, all comments in the current document will be displayed permanently.

"Decimal point after input" section

- **Move by ... decimals**

  If this option is enabled, numbers entered in cells will be shifted by the specified number of decimal places.

  This is useful if, for example, many numbers with two decimal places have to be entered. If you set the number of digits to the value 2 and enter the number 42, this will automatically be converted to 0.42; the input 234 will be converted to 2.34, and so on.

"Window" section

- **Vertical scrollbar**

  If this option is enabled, a vertical scrollbar is displayed in the document window.

- **Horizontal scrollbar**

  If this option is enabled, a horizontal scrollbar is displayed in the document window.

- **Sheet tabs**

  If this option is enabled, the worksheet register with tabs for each worksheet is displayed below the document. These tabs can be used for switching between worksheets and creating/managing worksheets.

"Objects" section

- **Guidelines for text frames**

  If this option is enabled, non-printing gray lines are displayed around text frames. These lines make it easier to determine and change the position/size of text frames.

- **Show hidden objects**

  As described in the section Hiding objects, you can hide inserted objects on the screen, that is, make them invisible.

  If you enable the option **Show hidden objects**, however, hidden objects that have the **Visible** property disabled will also become visible on the screen.

Default currency

Use this option to change the default currency of the current document.

Note: This option should be set to **System default** by default. This will ensure that PlanMaker uses the default currency from your system's country settings.

If you select a different currency, this will have the following effect:

Whenever PlanMaker *automatically* applies the **Currency** number format to a cell (for example, because it uses an arithmetic function that returns a monetary value), the currency set here will be used instead of the system's default currency.

Compatibility button

Here, you can make internal settings for compatibility with documents created in older versions of PlanMaker or in Microsoft Excel. These options are set automatically when the document is opened and should not normally be changed.

Document properties, Calculate tab

On the **Calculate** tab in the dialog box of the ribbon command **File | Properties** , you can change settings for the calculations in the current document:

"Recalculation" section

Here, you can determine whether the calculations in the current worksheet should be updated automatically when you make changes to the worksheet contents:

- **Recalc automatically**

  If this option is enabled, all calculations are updated automatically when the content of a cell is changed.

  This is the default setting. It ensures that all calculations always provide up-to-date results.

- **Recalc only before ...**

  If this option is enabled, calculations will be automatically updated only when **saving** or **printing** the document, or when **copying** (or cutting) cells. If all three options are disabled, calculations will no longer be updated automatically.

  However, you can have the calculations updated manually at any time: To do so, choose the ribbon command **Formula** | group **Update**| **Recalculate** or press the **F9** key.

  For more information, see also Updating sheets.

"External references" section

Here, you can make settings for *external* cell references. These are references that refer to cells in other workbooks. For more information, see External cell references.

Available options:

- **Update data on opening**

  Here, you can specify whether all external cell references in a document should be updated (once) when the document is opened:

  Always: When the document is opened, update its external references automatically.

  Never: When the document is opened, *do not* update its external references.

  Ask user: When the document is opened, ask the user if external references should be updated.

- **Save external data in document**

  If this option is enabled, PlanMaker saves a copy of all values returned by external cell references in the document. If the file for an external reference is missing when the external reference is updated, the value last stored will still be displayed.

  If you disable this option, an error value will be displayed instead if you update an external reference and the associated file cannot be accessed.

"Rounding" section

Enabling these options increases the accuracy of calculations. Since computers use a different numeral system than humans, some calculations may lead to very small rounding errors in very rare cases. This issue is not PlanMaker-specific but affects practically all computer applications.

However, PlanMaker has two options for eliminating these rounding errors as much as possible:

- If the option **Round final result** is enabled, PlanMaker automatically rounds the final result of each calculation to 15 decimal places.
- If **Round intermediate results** is enabled, PlanMaker also rounds any intermediate result within a calculation to 15 places.

The first option already significantly reduces the probability of obtaining a rounding error, while the second option makes this even less likely. The disadvantage of both options, however, is that the recalculation of worksheets takes a little longer.

A classic example of this rounding problem:

If both options are disabled, the formula (0.1+0.2-0.3)=0 returns FALSE, although TRUE would be the correct result. If both options are enabled, the calculation returns the correct result.

- A third option is **Precision as displayed**: If you enable this option, all numbers are applied exactly as they are displayed after rounding to decimal places. This means that the values that are entered are truncated to the decimal places set in the cell options for the number format.

  An example for illustrative purposes:

  You have the sum of 2.3 + 2.4. The exact result is 4.7. However, you choose not to display the decimal places after the decimal point. You thus set the decimal places in the cell options to 0. The table now shows 2 + 2.

  Without enabling the option *Precision as displayed*: You obtain the result 5 (because the values 2.3 + 2.4 still exist in the background and are rounded to 5). You can also continue to calculate with the exact result 4.7.

  By enabling the option *Precision as displayed*: You obtain the result 4 (because only the values 2 + 2 exist, exactly as they are displayed in the cell). However, the exact result 4.7 has been lost for further calculation.

Note on the option *Precision as displayed*: If decimal numbers are longer than the decimal places that you set for the number format in the cell options, the original, more accurate values are permanently lost. You should thus be sure that you want to use this option in your calculation.

"Charts" section

Like calculations, charts are automatically updated by default when the content of any cell is changed.

If you don't want this, disable the option **Update automatically** in the **Charts** section.

Charts must then be updated manually if necessary. To do so, choose the ribbon command **Formula |** group **Update | Update data | Update charts**.

"Iterations" section

This option only affects cells that contain a circular reference. If, for example, cell A1 contains the formula =A1+A2, this is called a circular reference because the calculation in cell A1 refers to cell A1 itself.

If the option **Use iterations** is enabled, such calculations are repeated each time the worksheet is recalculated until either the number of **Maximum iterations** is reached or the result of the calculation differs from the previous result by less than the **Maximum change**.

Normally, tables should not contain circular references, thus this option is disabled by default.

Note: Unless you actually use circular references in calculations on purpose (for example, to determine values using an iteration method), this option should not be enabled, since it significantly increases the time required to recalculate the worksheet.

Document properties, Protection tab

On the **Protection** tab in the dialog box of the ribbon command **File | Properties** , you can provide *document protection* to the current document.

You can ensure that opening or saving a protected document is only possible after entering the correct password. Documents with read protection are also encrypted, so that they cannot be viewed with programs other than PlanMaker.

For information on this topic, see Document protection.

Document properties, Fonts tab

A list of all fonts used in the current document is displayed on the **Fonts** tab in the dialog box of the ribbon command **File | Properties** .

The list is divided into fonts that are available on the current device and fonts that are not available (i.e., not installed).

Changing worksheet properties

You can use the ribbon command **Insert** | group **Tables** | **Sheet** | **Properties** to define settings that will only affect the current *worksheet*.

If a document consists of several worksheets, you can change these settings separately for each sheet. To do so, switch to the desired worksheet and then choose the above command.

Tip: You will also find the command in the context menu by right-clicking on the worksheet register.

The program displays a dialog box with the following options:

- **Row header**

  If this option is enabled, row headers (buttons labeled 1, 2, 3, etc., to the left of the table) are displayed in the worksheet.

- **Column header**

  If this option is enabled, column headers (buttons labeled A, B, C, etc., above the table) are displayed in the worksheet.

  Tip: You can also use the ribbon command **View** | **Row and column headers** to enable/disable the row and column headers together.

- **Page breaks**

  This option determines whether dark gray lines indicating the location of page breaks for the printout are displayed in the worksheet. (For more information, also see Adjusting page breaks manually.)

  Tip: This option can also be found directly on the ribbon tab **View** as **Show page breaks**.

- **Show zero values**

  A cell contains a *zero value* when it contains either the number 0 (zero) or a calculation that returns zero.

  If this option is enabled, the number "0" will be displayed in such cells as usual. If you disable it, nothing will be displayed there.

  Tip: This option can also be found directly on the ribbon tab **View** as **Show zero values**.

- **Gridlines**

  This option determines whether gridlines are displayed between cells on-screen. You can also change the **color** of these lines as desired.

  Tip: This option can also be found directly on the ribbon tab **View** as **Show sheet grid**.

  Gridlines do not appear in printouts by default. To include gridlines in a printout, choose the ribbon command **File** | group **Print** | **Page setup**, switch to the **Options** tab and enable the option **Grid**.

- **Sheet tab**

  The *worksheet register* at the bottom of the document window displays tabs for each worksheet in the current document.

  With this option, you can change the **color** of the tab for the current worksheet.

- **Background**

  If you enable the option **Alternating shade**, alternating shading is applied to the rows of the worksheet: Odd lines are shaded as specified with the **Shading 1** button, while even lines are shaded as specified with the **Shading 2** button.

- **Sheet direction**

  For worksheets in *Arabic* script, you can change the direction of the worksheet to right-to-left here.

  Doing this will reverse the horizontal direction of the worksheet – thus cell A1 will now be in the top right corner rather than in the top left corner.

  For more information, see also Working with Arabic text.

Customizing the document display

In this section, you will learn how to customize the appearance of the worksheet on the screen. The required commands can be found on the ribbon tab **View**.

This section covers the following topics:

- **Zoom level**

  The ribbon commands **View | Actual size** and **View | Set zoom** in the command group **Zoom** allow you to set the zoom level of the worksheet.

- **Full screen view** (not included in SoftMaker FreeOffice)

  The ribbon command **View | Full screen** in the command group **Windows** maximizes the program window and hides most parts of the user interface so that as much of the document as possible can be seen.

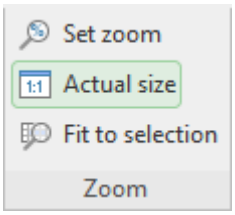
- **Dark document** (only available in SoftMaker NX Home and NX Universal)

  Choose the command **Dark document** in the Quick access toolbar for a dark display of the document.

For more information, see the following pages.

Zoom level

With the commands of the ribbon tab **View** in the group **Zoom**, you can select the zoom level for the current document, among other things.



The command group "Zoom" on the ribbon tab **View**

- The command **Actual size** always sets the document to a zoom level of 100%.
- The command **Fit to selection** automatically selects the zoom level at which the currently selected cells fit exactly into the main window.
- For an exact selection of the desired zoom, choose the command **Set zoom**. The program displays a dialog box with the following options:

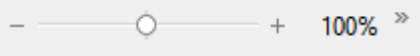
Zoom level	Explanation
Fit to selection	This option automatically selects the appropriate zoom level, see above.

Previous zoom level	This option restores the previous zoom level.
50%, **75%**, etc.	This option sets the zoom level to the desired level.

Alternatively, you can enter any zoom level in the input field above the list (50% to 400%).

Using the zoom slider

You can also use the zoom slider in the bottom right corner of the screen to change the zoom level:



To change the zoom level, either drag the slider with your mouse or click on the plus or minus icon.

If you click on the percentage to the right of the zoom slider, you open the dialog box of the aforementioned command **View** | group **Zoom** | **Set zoom**.

Tip: Also note the key combination **Ctrl+middle mouse wheel** for continuous zooming.

Full screen view

With the ribbon command **View** | group **Windows** | **Full screen**, you can switch to the *full screen view* in which the program uses the entire screen to display the document. PlanMaker enlarges its program window to the full screen size. (In the *classic menu interface*, all other toolbars are then hidden.)

To exit this view mode, press the **Esc** key or choose the ribbon command **View** | **Full screen** again.

Using the toolbar of the full screen view

When you enable full screen mode, an additional mini toolbar appears on the screen.

The mini toolbar "Full screen mode"

The icons in this toolbar have the following functions (from left to right):

- Show/hide ribbon
- Show/hide pivot table sidebar
- Enable/disable "Drag sheet" mode

  If you enable "Drag sheet" mode, you can scroll through your document using the mouse. Simply click into the document and drag the mouse cursor in the desired direction (with the mouse button still held down).

  Note: Clicking on cells and selecting text or objects is no longer possible in this mode.

  To exit "Drag sheet" mode, click on this icon again.
- Clicking on the **Close** button exits full screen mode.

Dark document mode

This feature is only available in **SoftMaker Office NX Home** and **NX Universal**.

You can use the ribbon command **Dark document** in the Quick access toolbar to switch the worksheets from their usual white to a dark display.

Note: If this button is not available, click on the » icon on the far right of the Quick access toolbar. Select **Add/remove buttons** and enable **Dark document**.

If you click on this command once, the worksheets will be displayed in dark mode. If you click on this command again, the worksheets will be displayed in white again.

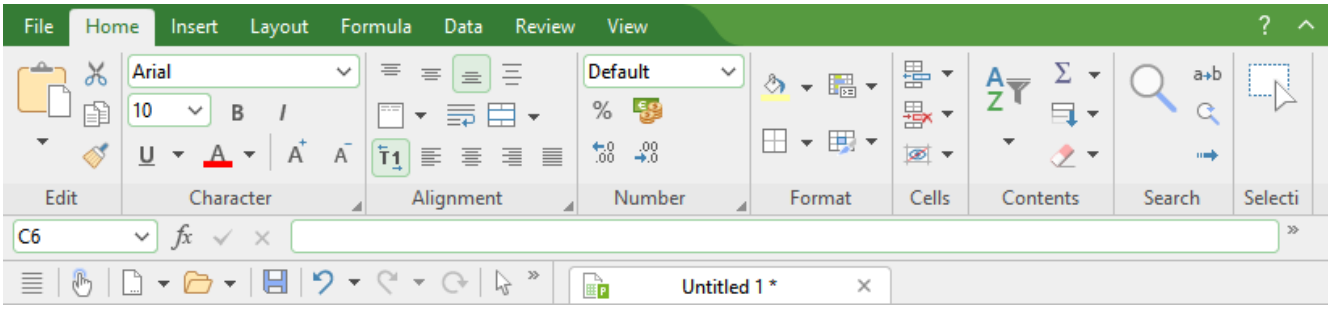
Switching this display has no effect on the printout of the document.

Customizing the ribbon

"Ribbon" only: This section is relevant only for users who have configured the program to use the **ribbon** interface. (For information on customizing the **classic menu interface**, see Customizing toolbars.)

Tip: You can switch the user interface between *ribbon* and *classic menus* at any time. To do so, choose the command **File | Options** in the ribbon (or choose **Tools > Options** in the classic menu interface). In the dialog box, switch to the **Appearance** tab and click on the **User interface** button. The program will display a dialog box in which you can choose the desired user interface.

In this section you will learn how to configure the *ribbon* and add additional icons.



The "Ribbon" and the Quick access toolbar (below)

The information covers the following topics:

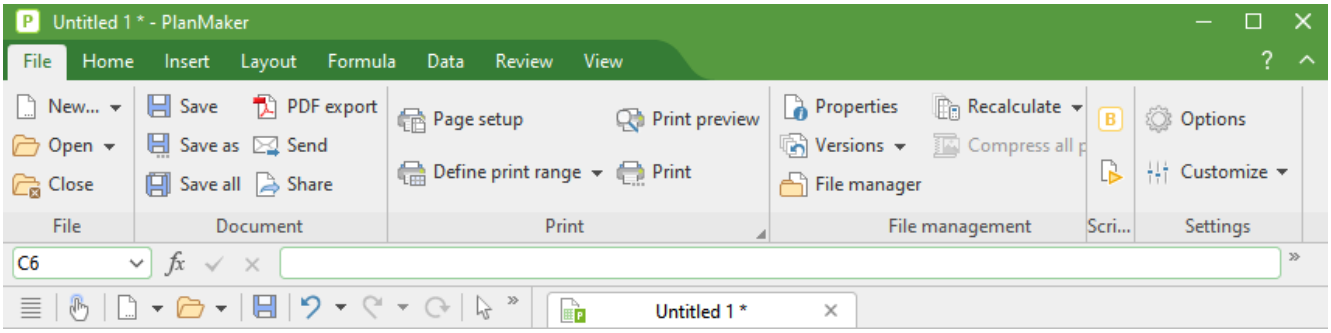
- Showing and hiding the ribbon
- Customizing icons and groups in the ribbon
- Changing the position of the Quick access toolbar
- Customizing icons on the Quick access toolbar

For more information, see the following pages.

Showing and hiding the ribbon

"Ribbon" only: This section is relevant only for users who have configured the program to use the **ribbon** interface. (For information on customizing the **classic menu interface**, see Customizing toolbars.)

The ribbon is usually fully visible:



The "Ribbon" and the Quick access toolbar (below)

However, you can hide the ribbon at any time by *minimizing* it. This is useful if you want to have as much screen space as possible to display the document.

Proceed as follows:

1. Choose the ribbon command **File** | group **Settings** | **Customize** .
   Also possible: Right-click anywhere on the ribbon.
2. A menu opens. There, you will find the option **Minimized ribbon**, which can be enabled or disabled by clicking on it (a check mark appears in front of the option if it is currently enabled).

After you enable this option, the ribbon will be minimized. Only the ribbon tabs and the quick access toolbar can then be seen.

Tip: Alternatively, you can use the icon in the upper right corner of the program window next to the **?** icon to minimize the ribbon. If you click on the icon again, the ribbon is restored to its full size.

In addition, you can use the key combination **Ctrl+F1** (Mac: **Cmd+F1**) to enable/disable minimization of the ribbon.

Working with a minimized ribbon

You work with a minimized ribbon as follows:

- When you click on one of the ribbon tabs of the minimized ribbon, the ribbon becomes completely visible again *temporarily* and displays the corresponding ribbon tab. As soon as you trigger a command (for example, click on the icon for a command), however, the ribbon will be automatically minimized again.
- If you *double*-click on one of the ribbon tabs, the ribbon will be *permanently* visible again, thus removing the minimization.

- As mentioned above as a tip, you can also use the icon in the upper right corner of the program window to enable/disable the minimization of the ribbon.

Customizing icons and groups in the ribbon

"Ribbon" only: This section is relevant only for users who have configured the program to use the **ribbon** interface. (For information on customizing the **classic menu interface**, see Customizing toolbars.)

The command **Customize ribbon** allows you to customize the ribbon to a certain extent. The main purpose of this command is to add *additional* icons, command groups or even complete ribbon tabs to the ribbon.

Important: The icons in the predefined command groups on the ribbon cannot be changed. This means that you cannot add, remove, or change the order of icons in the existing command groups. Additional icons can only be inserted in new command groups at the end of a ribbon tab (or optionally in a completely new ribbon tab).

To customize the ribbon, proceed as follows:

1. Choose the command **Customize ribbon** under **File** | group **Settings** | **Customize** to open the dialog box. Alternatively: Right-click anywhere on the ribbon with the mouse button.

   Tip: The dialog box can be opened even faster by double-clicking on any empty area of the ribbon.

2. Use one of the procedures described below to edit the ribbon tabs.

Basic structure of the dialog box

The dialog box of the command **Customize ribbon** contains two lists:

- **Left list: all available commands**

  The *left* list shows all commands available in the program.

  Tip: You can use the option **Categories** (above the left list) to filter the list by a specific category (for example, all commands of the "File" category).

  You can also search the list by typing something in the **Quick search** field.

- **Right list: icons of the currently selected ribbon tab**

  The *right* list displays the icons contained in the currently selected ribbon tab.

  You define the ribbon tab to be displayed here via the dropdown list **Customize** (top right).

In this dialog box, you can perform the following actions:

Adding an icon

Note: You can only add icons to command groups that you have added yourself (or to completely new ribbon tabs). You cannot insert an icon into the predefined groups.

To add an icon to a specific ribbon tab, perform the following steps in the dialog box of the command **File | group Settings | Customize | Customize ribbon**:

1. In the dropdown list **Customize** (top right), select the ribbon tab to which you want to add an icon.

   The right list now displays all icons that are currently on the selected ribbon tab.

2. In the *left* list, select the icon that you want to add.

3. Click on the button >>.

   Even faster: Simply double-click on the icon in the left list.

The icon is now added to the ribbon tab. If you have not yet created a new group here, the program automatically creates a new command group for the icon at the end of the ribbon tab.

Removing an icon

Note: You can only remove icons that you added to the ribbon yourself. You cannot delete any of the predefined icons.

To remove one of the icons you added from a ribbon tab, perform the following steps in the dialog box:

1. In the dropdown list **Customize**, select the ribbon tab on which the icon is located.

2. In the right list below it, select the icon to be removed.

3. Click on the button <<.

   Even faster: Simply double-click on the icon in the right list.

The icon is now removed from the ribbon tab.

Moving an icon

Note: You can only move icons that you added to the ribbon yourself. You cannot change the position of any of the predefined icons.

To move an icon in a ribbon tab, perform the following steps in the dialog box:

1. In the dropdown list **Customize**, select the ribbon tab on which the icon is located.

2. In the right list below it, select the icon to be moved.

3. Use the up and down arrow buttons to move the icon within the ribbon tab.

   Note: If you want to move an icon to a *different* ribbon tab, you must first remove it from its current ribbon tab and then add it to the other ribbon tab.

Adding a new command group

Note: New command groups can only be added *after* the default command groups. You cannot insert a user-defined group between the predefined command groups.

To add a new command group (a new section) to a ribbon tab, perform the following steps in the dialog box:

1. Select the desired ribbon tab from the dropdown list **Customize**.
2. Click on the **New group** button (displayed below the right list).

The new command group is now created (at the end of the ribbon tab).

Renaming a command group

Note: Only user-defined command groups can be renamed. You cannot rename any of the predefined command groups.

To rename a user-defined command group in a ribbon tab, perform the following steps in the dialog box:

1. Select the desired ribbon tab from the dropdown list **Customize**.
2. Select the desired command group from the right list below it.
3. Click on the **Rename** button (displayed below the right list).
4. Type in the new name and confirm it.

The command group will now be renamed accordingly.

Removing a command group

Note: Only user-defined command groups can be removed. You cannot delete any of the predefined command groups.

To remove a user-defined command group from a ribbon tab, follow these steps in the dialog box:

1. Select the desired ribbon tab from the dropdown list **Customize**.
2. Select the desired command group from the right list below it.
3. Click on the button . (Any icons must have been removed from the group.)
   Even faster: Simply double-click on the command group in the right list.

The command group is now removed from the ribbon tab.

Assigning shortcut keys

If you click on the **Shortcut keys** button in the dialog box, the dialog box will expand to include an additional area called **Shortcut keys**, which can be used to edit shortcut keys for commands.

Summary: You can assign a shortcut key of your choice to any command here: Select the command in either of the two lists, click on the **Press new shortcut key** field and then press the key combination with which you want to choose this command in future.

For more information, see Customizing shortcut keys.

"Edit" button

The **Edit** button is only enabled if you have selected a user-defined icon in the list on the left side of the dialog box. For more information see Creating user-defined icons.

Buttons for managing the ribbon tabs (right side)

Above and below the list on the right side of the dialog box, there are buttons for managing the ribbon tabs. Some of these buttons have already been mentioned in this section. To ensure that users understand them better, all buttons and their functions are clearly listed again below.

Above the list on the right:

- **New**

  This button creates a new, empty ribbon tab. You can then customize this ribbon tab by adding icons and command groups.

- **Rename**

  Use this button to rename the ribbon tab selected in the dropdown list **Customize** (only available for user-defined ribbon tabs).

- **Reset**

  This button resets the ribbon tab selected in the dropdown list **Customize** back to its delivery state.

- **Delete**

  This button deletes the ribbon tab selected in the dropdown list **Customize** (only available for user-defined ribbon tabs).

Below the list on the right:

- **New group**

  This button adds a new command group to the ribbon tab selected in the dropdown list **Customize**.

- **Rename**

  Use this button to rename a command group (only available for user-defined command groups). First, select the desired command group in the list on the right of the dialog box.

- **Reset all**

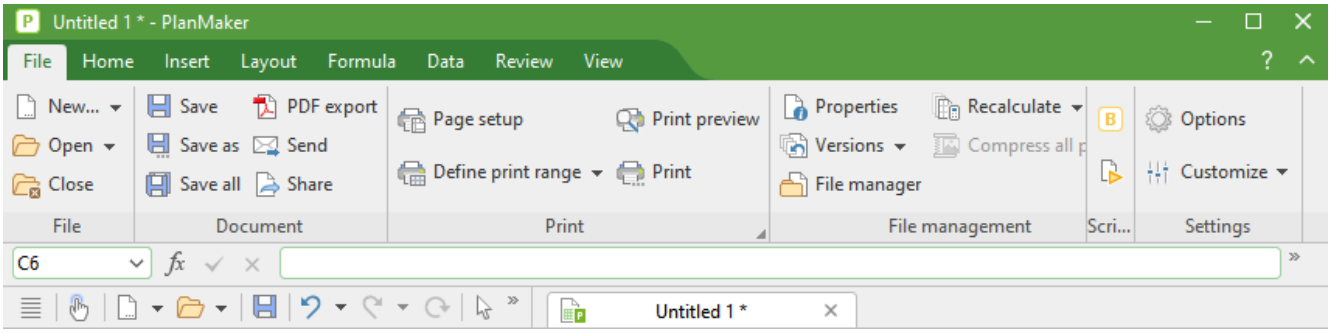
  This button resets all ribbon tabs back to their delivery state.

The **Close** button closes the dialog box and all changes are applied.

Changing the position of the Quick access toolbar

"Ribbon" only: This section is relevant only for users who have configured the program to use the **ribbon** interface. (For information on customizing the **classic menu interface**, see Customizing toolbars.)

The *Quick access toolbar* is displayed below the ribbon. It provides a selection of frequently used commands.



The "Ribbon" and the Quick access toolbar (below)

The Quick access toolbar can either share the space with the document tabs – or it is located in its own toolbar. To change this, proceed as follows:

1. Choose the ribbon command **File** | group **Settings** | **Customize** .

   Alternatively, right-click anywhere on the ribbon.

2. The program opens a menu that contains one of the following two commands:

   Show Quick access toolbar as separate toolbar: By default, the Quick access toolbar is displayed in the toolbar that also contains the document tabs. When you choose this command, the Quick access toolbar moves to its own toolbar instead.

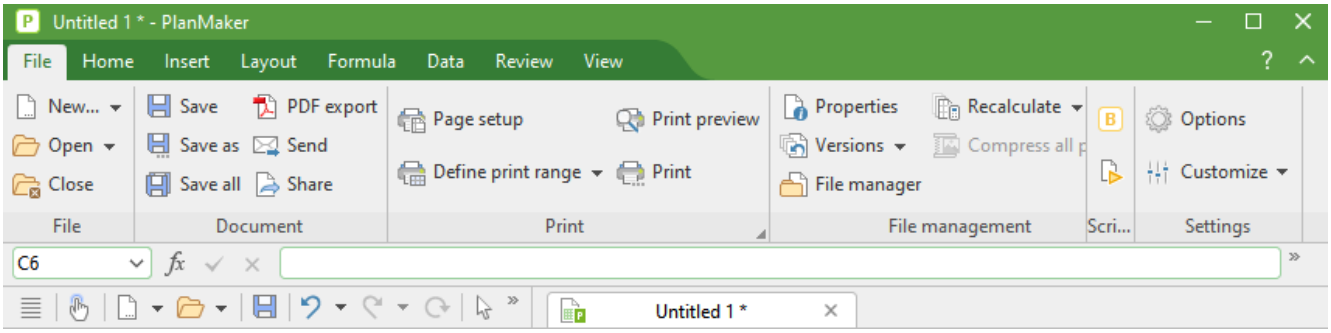
   Show Quick access toolbar beside the document tabs: This command does exactly the opposite: If the Quick access toolbar is currently displayed as a separate toolbar, choosing this command will cause the Quick access toolbar to be moved back to the toolbar that also contains the document tabs.

Tip: You can also click on the double arrow button » directly in the Quick access toolbar itself. There, you can use the entry **Position** to find the corresponding options for displaying the Quick access toolbar: *Beside the document tabs* or *Below the ribbon*.

Customizing icons on the Quick access toolbar

"Ribbon" only: This section is relevant only for users who have configured the program to use the **ribbon** interface. (For information on customizing the **classic menu interface**, see Customizing toolbars.)

The *Quick access toolbar* is displayed below the ribbon. It provides a selection of frequently used commands.



The "Ribbon" and the Quick access toolbar (below)

If desired, you can customize the icons on the Quick access toolbar at any time, for example, you can add, remove and reposition icons.

To do so, choose **File** | group **Settings** | **Customize** | **Customize Quick access toolbar** (or click on the icon » in the Quick access toolbar and select *Customize* there). In the following dialog box, you can edit the Quick access toolbar according to the descriptions in Customizing toolbar icons – except for the passage "Managing toolbars".

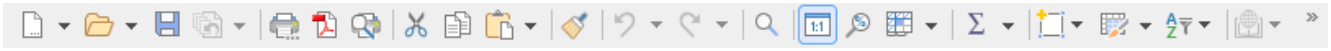
Tip: Alternatively, you can simply drag a command from the dialog box into the Quick access toolbar with the mouse. Even quicker (without opening the dialog box): Hold down the **Alt** key and drag a command icon directly from the ribbon to the Quick access toolbar. To remove it, drag the icon back into the ribbon while holding down the **Alt** key.

Customizing toolbars

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

Tip: You can switch the user interface between *Ribbon* and *Classic menus* at any time. To do so, choose the command **File | Options** in the ribbon (or choose **Tools > Options** in the classic menu interface). In the dialog box, switch to the **Appearance** tab and click on the **User interface** button. The program will display a dialog box in which you can choose the desired user interface.

In this section, you will learn how to configure the *toolbars* (Standard toolbar, Formatting toolbar, etc.) with the menu command **View > Toolbars** and how to customize them with **Tools > Customize**.



A toolbar (here: the Standard toolbar)

The following topics are covered:

- Showing and hiding toolbars
- Positioning toolbars on the screen
- Managing toolbars (creating, deleting, renaming, resetting toolbars)
- Customizing toolbar icons (adding, removing, moving icons)
- Using the toolbar menu (for extra quick access to the above functions)

For more information, see the following pages.

Showing and hiding toolbars

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

You can enable or disable the available toolbars at any time to make them visible or to hide them.

To do so, proceed as follows:

1. Choose the menu command **View > Toolbars**.

   Alternatively, click on the double arrow » at the end of a toolbar and select **Toolbars**.

2. Click on the checkbox in front of the desired toolbar to enable or disable it. When the toolbar is enabled, a check mark appears in the box.

3. Exit the dialog box with **Close**.

The corresponding toolbar has now been enabled or disabled.

Tip: There is a faster way to do this: If you *right-click* on any toolbar, a list of all available toolbars is displayed. If you click on one of these menu items, the corresponding bar will be enabled or disabled.

Visible in mode...

Some toolbars are visible only in particular situations. The Picture toolbar, for example, appears only when you select a picture.

This is due to the option **Visible in mode**, which determines when a toolbar should be visible. Some examples:

Option	Explanation
All modes	The selected toolbar is *always* visible (unless it has been disabled). Example: the Standard toolbar
Edit mode	The bar is only visible in normal *editing mode* but not in object mode. For more information, see also Object mode. Example: the Formatting toolbar
Object	The bar is only visible in *object mode* but not in normal editing mode. For more information, see also Object mode. Example: the Object toolbar
Chart	The bar only appears if a chart is selected in the document. Example: the Chart toolbar
Outline	The bar only appears if the current worksheet contains an outline (see also Outlines). Example: the Outline toolbar
Picture	The bar only appears if a picture is selected in the document. Example: the Picture toolbar
Table	The bar only appears if you are in a cell of a newly inserted table (see also Tables in worksheets).
AutoShape	The bar only appears if a drawing or text frame is selected in the document.

To change this setting for one of the toolbars, choose the menu command **View > Toolbars**, select the toolbar in the list **Toolbars** and select the desired option for **Visible in mode**.

Note: Changing this setting in the default toolbars (Standard toolbar, Formatting toolbar, etc.) is usually not recommended. This option is primarily intended for user-defined toolbars.

Positioning toolbars on the screen

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

You can position each toolbar as desired on the screen. To do so, use one of the following procedures:

Changing the position with the command View > Toolbars

You can change the position of a toolbar at any time with the menu command **View > Toolbars**. To do so, proceed as follows:

1. Choose the menu command **View > Toolbars**.
2. Select the desired toolbar by clicking on it in the list.
3. Select the desired position for the toolbar in the list **Position**.

Note: If you select the option **Floating**, the toolbar icons are displayed in a window that can be freely moved or resized just like an application window.

Changing the position with the toolbar menu

Each toolbar has a button with a double arrow » at its right edge. Select the entry **Position** for the desired toolbar. The following submenu provides a dropdown list of possible positions for the toolbar.

Note: If you have selected the option **Floating**, you can also use this button » to integrate the toolbar back into a desired fixed position via the entry **Position** entry.

For more information on the toolbar menu options, see Using the toolbar menu.

Changing the position with the mouse

Some toolbars can also be repositioned with the mouse. To do so, proceed as follows:

1. Position the mouse cursor over an *empty* area of the toolbar.
2. Press the left mouse button and hold it down.
3. While still holding the mouse button down, drag the toolbar to the desired position.
4. Release the mouse button to place the toolbar in the new position.

Note: By default, most toolbars are *locked* to prevent unintentional movement with the mouse. However, you can release this lock at any time: To do so, choose the menu command **View > Toolbars**, select the toolbar and disable the option **Locked**. From now on, this bar can also be moved with the mouse.

By the way, a positioning aid is enabled while dragging a toolbar: If you drag the toolbar near one of the edges of the program window, it automatically jumps to the corresponding edge. If, for example, you release the bar at the top edge, this corresponds to selecting the **Top** position in the dialog box of the menu command **View > Toolbars**.

Managing toolbars

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

You can create new toolbars at any time and delete or rename existing toolbars, etc. To do so, use the buttons in the dialog box of the menu command **View > Toolbars**.

Creating new toolbars

The program lets you create your own toolbars at any time.

If, for example, you frequently need to use commands of the **Window** menu that are not present in the Standard toolbar, it might be worth your while to create your own toolbar for them.

You create a new toolbar as follows:

1. Click on the **New** button (in the dialog box of the menu command **View > Toolbars**).
2. The program displays a dialog box in which you enter a name for the new toolbar – for example, "My toolbar". Then confirm with **OK**.
3. The new toolbar has been created. Close the dialog box or click on the **Customize** button to add icons to the toolbar. For more information, see Customizing toolbar icons.

Customizing toolbar icons

You can edit the icons on toolbars with the **Customize** button. For more information, see the next section.

Deleting toolbars

The **Delete** button allows you to delete a user-defined toolbar as follows:

1. Select the toolbar to be deleted from the list.
2. Click on the **Delete** button.

Note: You can only delete user-defined toolbars. The default toolbars (Standard toolbar, Formatting toolbar, etc.) cannot be removed.

However, you can disable a default toolbar at any time if you don't need it by removing the check mark in front of it. For more information, see Showing and hiding toolbars.

Renaming toolbars

The **Rename** button can be used to rename a user-defined toolbar as follows:

1. Select the desired toolbar from the list.
2. Click on the **Rename** button.
3. Type in the new name and confirm it with **OK**.

Note: You can only rename user-defined toolbars. The default toolbars (Standard toolbar, Formatting toolbar, etc.) cannot be renamed.

Resetting toolbars

You can undo any changes you have made to one of the default toolbars by clicking on the **Reset** button:

1. Select the desired toolbar from the list.
2. Click on the **Reset** button.

The toolbar now contains the default icons again. In addition, the default settings for the options **Visible in mode** and **Position** are restored.

Note: This command only applies to the default toolbars (Standard toolbar, Formatting toolbar, etc.) but not to user-defined toolbars.

Resetting all toolbars

To reset *all* toolbars at once, click on the **Reset all** button.

Note: This command only resets the default toolbars. User-defined toolbars remain unchanged.

Customizing toolbar icons

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

The menu command **Tools > Customize** allows you to customize the icons as desired on a toolbar. You can add icons and remove or reposition existing icons, among other things.

Proceed as follows:

1. Choose the menu command **Tools > Customize** to open the dialog box. Alternatively: Click on the button » at the end of a toolbar and select the entry **Customize** here.

   Even faster: Double-click on an empty area in any toolbar.

2. Use one of the procedures described below to add, remove or reposition icons.

Basic structure of the dialog box

The dialog box of the menu command **Tools > Customize** contains two lists:

- **Left list: all available commands**

  The *left* list shows all commands available in the program.

  Tip: You can use the option **Categories** (above the left list) to filter the list by a specific category (for example, all commands of the "File" category).

  You can also search the list by typing something in the **Quick search** field.

- **Right list: icons of the currently selected toolbar**

  The *right* list displays the icons contained in the currently selected toolbar.

  You define the toolbar to be displayed here via the dropdown list **Customize** (top right).

In this dialog box, you can perform the following actions:

Adding an icon

To add an icon to a specific toolbar, perform the following steps in the dialog box of the command **Tools > Customize**:

1. In the dropdown list **Customize** (top right), select the toolbar to which you want to add an icon.

   The right list now displays all icons that are currently in the selected toolbar.

   Optional: In the *right* list, select the icon behind which the new icon is to be inserted.

2. In the *left* list, select the icon that you want to add.

3. Click on the button >>.

   Even faster: Simply double-click on the icon in the left list.

The icon is now added to the toolbar.

Tip: Alternatively, you can add an icon to any toolbar as follows: Choose the command **Tools > Customize**. Then simply drag the icon out of the dialog box with the mouse and drop it at the desired position in the toolbar. (Do this in the toolbar itself and not in the dialog box!) Of course, this procedure only works when the toolbar is currently visible on the screen.

Removing an icon

To remove an icon from a toolbar, perform the following steps in the dialog box:

1. Select the desired toolbar from the dropdown list **Customize** (top right).

2. In the right list below it, select the icon to be removed.

3. Click on the button <<.

   Even faster: Simply double-click on the icon in the right list.

The icon is now removed from the toolbar.

Tip: Alternatively, you can remove an icon from a toolbar as follows: Choose the command **Tools > Customize**. Then simply drag the icon out of the toolbar with your mouse and drop it somewhere outside the toolbar, for example, in the document. (Do this in the toolbar itself and not in the dialog box!)

Moving an icon

To move a toolbar icon to a different position, perform the following steps in the dialog box:

1. Select the desired toolbar from the dropdown list **Customize**.
2. In the right list below it, select the icon to be moved.
3. Use the up and down arrow buttons to move the icon within the toolbar.

   Note: If you want to move an icon to a *different* toolbar, you must first remove it from its current toolbar and then add it to the other toolbar.

Tip 1: Alternatively, you can also move an icon as follows: Choose the command **Tools > Customize**. Then drag the icon directly in the toolbar to the desired position (in the toolbar itself and not in the dialog box!). This method also allows you to move an icon directly from one toolbar to another.

Tip 2: Moving icons works even if this dialog box is not open: When you hold down the **Alt** key, you can move a toolbar icon by clicking on it and then dragging it to the desired position (while still holding down the mouse button).

Inserting a separator

To insert a separator in front of an icon in a toolbar, perform the following steps in the dialog box:

1. Select the desired toolbar from the dropdown list **Customize**.
2. In the *right* list, select the icon in front of which the separator is to be inserted.
3. Click on the **New separator** button (displayed below the right list).

The separator is now inserted.

Tip: Alternatively, you can proceed as follows: Choose the command **Tools > Customize**. To insert a separator to the left of an icon in any toolbar, simply drag the icon slightly to the right (in the toolbar itself and not in the dialog box!).

Removing a separator

To remove a separator from a toolbar, perform the following steps in the dialog box:

1. Select the desired toolbar in the dropdown list **Customize**.
2. In the right list below it, select the separator to be removed.
3. Click on the button <<.

Even faster: Simply double-click on the separator.

The separator is now removed from the toolbar.

Tip: Alternatively, you can also use the following procedure: Choose the command **Tools > Customize**. If you now drag the icon that is to the immediate right of the separator a little bit to the left, the separator will be removed. (Do this in the toolbar itself and not in the dialog box!)

Assigning shortcut keys

If you click on the **Shortcut keys** button in the dialog box, the dialog box will expand to include an additional area called **Shortcut keys**, which can be used to edit shortcut keys for commands.

Summary: You can assign a shortcut key of your choice to any command here: Select the command in either of the two lists, click on the **Press new shortcut key** field and then press the key combination with which you want to choose this command in future.

For more information, see Customizing shortcut keys.

"Edit" button

The **Edit** button is only enabled if you have selected a user-defined icon in the list on the left side of the dialog box. For more information see Creating user-defined icons.

Buttons for managing the toolbars

The dialog box of the command **Tools > Customize** also has buttons for managing toolbars:

- **New**

  This button creates a new, empty toolbar.

- **Rename**

  Use this button to rename the toolbar selected in the dropdown list **Customize** (only available for user-defined toolbars).

- **Reset**

  This button resets the toolbar selected in the dropdown list **Customize** back to its delivery state.

- **Delete**

  This button deletes the toolbar selected in the dropdown list **Customize** (only available for user-defined toolbars).

- **Reset all** (displayed below the right list)

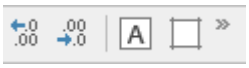
  This button resets all default toolbars back to their delivery state.

Further options for managing and configuring toolbars are available in the dialog box of the menu command **View > Toolbars**. For more information, see Managing toolbars.

Using the toolbar menu

Only for "Classic menus with toolbars": This section is relevant only for users who have configured the program to use the **classic menus interface**. For information on customizing the **ribbon**, see Customizing the ribbon.

Some toolbars end on the right with a double arrow », as shown in the following figure:



If you click on this double arrow, the *toolbar menu* will open for this toolbar.

This menu gives you quick access to all toolbar-related commands described in the previous sections. The toolbar menu contains the following commands:

- **Add/remove buttons**

  This command displays a list of all icons available for a toolbar. If you click on one of the entries, the corresponding icon in the bar will immediately be enabled or disabled.

  (Note: This command is only meant for quickly enabling or disabling icons. For information on customizing the icons in a toolbar, see Customizing toolbar icons.)

- **Position**

  You can use this command to change the position of a toolbar. The available options include top, bottom, left, right or floating (in a little window that can be moved and resized with the mouse).

- **Toolbars**

  This option chooses the menu command **View > Toolbars** that allows you to enable or disable, configure and manage toolbars.

  For more information, see Customizing toolbars.

- **Customize**

  This option chooses the menu command **Tools > Customize** that allows you to edit the icons on a toolbar (add, remove, reposition icons).

  For more information, see Customizing toolbar icons.

- **Reset this toolbar**

  This option resets the toolbar. The bar then displays its default icons again. In addition, all settings made for the bar are discarded.

Creating user-defined icons

You can create *user-defined icons* at any time and then add them to the ribbon or a toolbar. Such icons can be used to start other desired programs.

To create a user-defined icon – for example, an icon to start the Windows Editor – proceed as follows:

1. Choose the following command in the ribbon or menu:

   Ribbon: **File** | group **Settings** | **Customize** | **Customize ribbon**

   Menu: **Tools > Customize**

2. In the dropdown list **Categories**, select the **User** category.

3. Select one of the icons from the list **Commands**.

   The first three icons are pre-assigned (as examples) to the Windows Calculator, the Windows Character Map and the Windows Control Panel. Select the fourth icon, for example.

4. Click on the **Edit** button displayed below the list. Another dialog box appears.

5. Enter a short description of the program to be started in the **Description** field – for example, "Notepad".

6. Enter the complete path and file name for the program to be started in the **Command line** field, for example, `C:\Windows\notepad.exe`.

7. *Optional:* If you want the program to display an icon other than its default icon, enter the path and file name of the desired icon file in the **Icon file** field.

   Note that icons in toolbars are slightly smaller than the usual Windows icons.

As soon as you confirm with **OK**, the user-defined icon is created.

If you now want to add the user-defined icon to the ribbon or toolbar, proceed as described in the following sections:

- Ribbon: Customizing icons and groups in the ribbon
- Toolbars: Customizing toolbar icons

Customizing shortcut keys

Commands that are frequently used in a program can also be chosen via *shortcut keys*. For example, the command for saving the current document can be chosen very quickly by pressing the key combination **Ctrl+S**.

The predefined shortcut keys can be customized at any time in the dialog box of the ribbon command **File | Customize | Customize ribbon**. You can assign a shortcut key to any command and change or remove existing shortcut keys.

To do so, proceed as follows:

Adding shortcut keys

To assign a shortcut key to a command, perform the following steps:

1. Choose the following command in the ribbon or in the classic menu interface:

   Ribbon: **File** | group **Settings** | **Customize** | **Customize ribbon**

   Classic menus: **Tools > Customize**

2. In the dialog box that appears, click on the **Shortcut keys** button.

   The dialog box now expands to include an additional section called **Shortcut keys**, which can be used to edit the shortcut keys for commands.

3. In the left or right list, select the command to which you want to assign a shortcut key.

   The *left* list shows *all* commands available in the program. (You can use the option **Categories** to filter this list by a specific category and perform a search using the **Quick search** field.)

   The *right* list displays only the icons that are in the currently selected ribbon tab or toolbar.

4. Once you have selected the desired command, click into the **Press new shortcut key** field and press the intended key combination (starting with **Ctrl** or **Alt**).

   Tip: If you make a typing mistake, you can always press the **Backspace key**⌫ to delete the shortcut key you entered.

5. ***Do not forget:*** Click on **Add** to assign this shortcut to the command now.

6. Exit the dialog box with **Close**.

From now on, you can execute the selected command with the specified shortcut key.

Editing shortcut keys (removing, changing, etc.)

You can also edit existing shortcut keys for commands (for example, delete or change them) at any time. Proceed as follows:

1. Choose the following command in the ribbon or in the classic menu interface:

   Ribbon: **File** | group **Settings** | **Customize** | **Customize ribbon**

   Classic menus: **Tools > Customize**

2. In the dialog box that appears, click on the **Shortcut keys** button.

   The dialog box now expands to include an additional section called **Shortcut keys**, which can be used to edit the shortcut keys for commands.

3. In the left or right list, select the command whose shortcut keys you want to edit.

4. In the **Shortcut keys** section, perform one of the following actions:

   Remove shortcut keys: The list **Current shortcut keys** displays all shortcut keys currently assigned to the selected command. To remove a shortcut key, select it from the list and click on **Remove**.

Change shortcut keys: To change the shortcut key for a command, first remove it (as described above) and then assign it a new shortcut key (as described at the beginning of this section).

Reset shortcut keys: To reset *all* shortcut keys for the entire program to their default values, click on the **Reset all shortcut keys** button and confirm the confirmation prompt. Any changes you have made to shortcut keys will then be discarded.

Display a list of all shortcut keys: When you click on the **Export** button, a table listing all currently assigned shortcut keys is displayed. The buttons next to the table allow you to copy it to the clipboard, for example.

5. Exit the dialog box with **Close**.

Available shortcut keys

Note that some of the key combinations that are possible on your keyboard are not allowed for shortcut keys.

As a rule, you should use **alphabetic keys**, **numeric keys** or **function keys** for shortcut keys. You can combine them with **Ctrl**, **Alt** and/or the **Shift key**⇧.

You can easily check whether the desired key combination is allowed. Click in the **Press new shortcut key** field in the dialog box described above, and then enter the key combination. If it does not appear in the field, it is not allowed.

Some examples of valid shortcut keys include:

- **Ctrl+A**
- **Alt+A** (However, key combinations with the Alt key are *not* recommended – **Alt+A**, for example, is responsible for opening the **View** menu!)
- **Ctrl+Alt+A**
- **Ctrl+Shift+A**
- **Ctrl+Alt+Shift+A**
- **Ctrl+F2**
- etc.

Note: Letters on their own are not allowed, of course. Thus, you can't use **A** or **Shift+A** as shortcut keys.

Shortcut key already assigned: If you press a shortcut key that is already assigned, the current assignment will be shown just below the input field. You should then press the **Backspace key**⌫ to delete your shortcut key and try a different shortcut key. Otherwise, you will overwrite the existing assignment of this shortcut key.

Two-part shortcut keys: You can also use two-part shortcut keys (according to the WordStar standard) – for example, **Ctrl+K+X**. However, only shortcut keys according to the scheme "Ctrl + letter + letter" are allowed here.

Editing lists for automatic filling

As described in Filling cells automatically, you can use the command **Home | Fill** to fill a cell range with a fixed value or a sequence of values.

If you choose the entry **Series** from the ribbon command **Home** | group **Contents** | **Fill** , a dialog box opens. Here, you can use the **Edit lists** button to also create your own lists for the *Fill* function.

If, for example, you create a list with the content "Red", "Green" and "Blue", and then subsequently start a fill operation from a cell that contains the text "Red", PlanMaker will continue the series and fill with Green, Blue, Red, Green, Blue, etc.

Note: You can also add your own **Edit lists** button to a ribbon tab if required. To do so, choose the ribbon command **File | Customize | Customize ribbon** to open the dialog box for customizing the user interface, and search there for the command "Edit lists". Then proceed as described in Customizing icons and groups in the ribbon.

You will also find the command directly via the "Hamburger menu" ☰: **Tools > Edit lists**

The dialog box of the command **Edit lists** provides buttons with the following functions:

- **"New" button:** create a new list
- **"Edit" button:** edit the list created
- **"Delete" button:** delete the list created
- **"Import" button:** create a list from the currently selected cells

See the following paragraphs for more information.

Creating a new AutoFill list ("New" button)

To create a new AutoFill list for the *Fill* function, proceed as follows:

1. Choose the command **Edit lists**.
2. Click on the **New** button in the following dialog box.
3. Another dialog box appears. Enter the desired list entries in it. Please note that each line may only contain *one* entry. Press the **Enter key**↵ after each entry to start a new line.
4. Then click on the **OK** button.

The new list has now been created.

Editing an AutoFill list ("Edit" button)

To edit a list you have created for the *Fill* function, proceed as follows:

1. Choose the command **Edit lists**.

2. In the following dialog box, select the list to be edited by clicking on it with the mouse.
   Note: The predefined sample lists cannot be edited.
3. Click on the **Edit** button.
4. Another dialog box appears with the list entries. Edit them as desired. Please note that each line may only contain *one* entry. Press the **Enter key**↵ after each entry to start a new line.
5. Then click on the **OK** button.

The list has now been changed accordingly.

Deleting an AutoFill list ("Delete" button)

To delete a list you have created for the *Fill* function, proceed as follows:

1. Choose the command **Edit lists**.
2. In the following dialog box, select the list to be deleted by clicking on it with the mouse.
   Note: The predefined sample lists cannot be deleted.
3. Click on the **Delete** button.

The list has now been removed.

Importing an AutoFill list from cells ("Import" button)

If a PlanMaker document contains a cell range whose contents you want to save as an AutoFill list, proceed as follows:

1. Select the desired cells.
2. Choose the command **Edit lists**.
3. In the following dialog box, click on the **Import** button.

PlanMaker reads the contents of the selected cells and creates a new AutoFill list from them. Each selected cell becomes a list entry.

If you have selected multiple rows and columns, PlanMaker creates multiple lists accordingly. Before this, a query appears as to whether you want to import the data row by row or column by column. Choose **Row by row** if you want each *row* to become a list, or **Column by column** if you want each *column* to become a list.

Installing dictionaries

As an alternative to its own dictionaries, SoftMaker Office also supports the use of *Hunspell* dictionaries. They are available free of charge for installation.

You can install dictionaries for the spell check and also specify which of these dictionaries should be used for a language. For information on how to install these dictionaries, see the following sections:

- **Installing additional dictionaries**

  In this section, you will learn how to add more dictionaries that are not currently installed.

  Note: This is particularly important for the **Android** and **iOS** versions, since most dictionaries are already pre-installed in SoftMaker Office for **Windows**, **Mac** and **Linux**.

- **Choosing a different dictionary for a language**

  If you have multiple dictionaries installed for a language (for example, one from SoftMaker *and* one from Hunspell), you can also specify which of these dictionaries to use. Read this section for details.

For more information on these topics, see the following pages.

Installing additional dictionaries

SoftMaker Office comes with premium dictionaries for numerous languages. In addition to SoftMaker dictionaries, SoftMaker Office also supports the use of *Hunspell* dictionaries. These dictionaries are available as free downloads. Most of them were created by private users or non-profit organizations.

Hunspell dictionaries are available for numerous languages. If there is no SoftMaker dictionary for a language, you will certainly find a Hunspell dictionary for it.

In this section, you will learn how to install dictionaries that are not currently installed:

Windows, Mac and Linux

When you install SoftMaker Office for **Windows**, **Mac** or **Linux**, all relevant dictionaries are already integrated automatically – so there is usually no need to subsequently install additional dictionaries.

Note, however, that you can choose between different dictionaries here (Hunspell, SoftMaker) by clicking on **Switch** for the respective dictionary. For more information, see the next section Choosing a different dictionary for a language.

Android and iOS

The situation is different with SoftMaker Office for **Android** and **iOS:** Here, dictionaries are not automatically integrated and must be subsequently installed for a language as needed.

Proceed as follows:

1. Launch any of the apps (TextMaker, PlanMaker or Presentations) on your Android/iOS device.
2. Choose the ribbon command **File | Options** for the dialog box and switch to the **Language** tab.
3. Click on the **Manage dictionaries** button.
4. The program displays the dialog box "Dictionaries" with a list of all available languages. To download a dictionary for a language, select it from the list and press the **Install** button.

The dictionary is now installed and will be available from now on in any SoftMaker Office application.

Note: In addition to Hunspell dictionaries (with *Hunspell* added at the end), the above list also includes in-house SoftMaker dictionaries *(nothing added* to the end) for many languages. If both dictionaries are installed for a language, you can select which dictionary to use by clicking on the **Switch** button. For more information, see the next section.

Ask to install dictionaries when necessary

This option in the dialog box "Dictionaries" is also particularly important for the **Android** and **iOS** versions, as dictionaries are not pre-installed in these versions.

When enabled, the option has the following effect: If you enter text in your document in a language for which a dictionary is not yet installed, the program will automatically ask you whether you want to install the dictionary now.

Note: Of course, the automatic prompting by the program only occurs if the Spell check is enabled.

Choosing a different dictionary for a language

If you have *multiple* dictionaries installed for a language (for example, a SoftMaker dictionary and a Hunspell dictionary), you can always choose which of these dictionaries to use for the language. Proceed as follows:

1. Choose the ribbon command **File | Options** for the dialog box and switch to the **Language** tab.
2. Click on the **Manage dictionaries** button.

The program displays the dialog box "Dictionaries" with a list of all installed languages/dictionaries.

3. Navigate to the desired **Language** in this list. The dictionaries of a language can appear in the list with the following additions:

   No addition: SoftMaker Office's in-house dictionary for this language.

   Hunspell: Free Hunspell dictionary for this language.

4. Now specify which of these dictionaries should be used for the desired language. Here, the icons to the left of a dictionary have the following meaning:

   Blue icon: The dictionary is already installed and switched on.

   Gray icon: The dictionary is already installed and is *not* switched on.

No icon: The dictionary is not yet installed.

A blue icon thus indicates the currently active dictionary for the respective language. If you want to change the active dictionary, select another dictionary for this language from the list and click on the **Switch** button. (If the dictionary is not installed yet, click on the **Install** button beforehand.)

5. Confirm your changes by pressing **OK**.

From now on, the spell check will use the selected dictionary for the specified language.

Show only installed dictionaries

Enable this option if only the installed dictionaries should be displayed in the list.

Ask to install dictionaries when necessary

For more information on this option, see the descriptions in the previous section Installing additional dictionaries.

Formulas and functions

This chapter provides instructions on working with formulas, including comprehensive descriptions of PlanMaker's arithmetic functions:

The chapter consists of the following sections:

- **Formula basics**

  The first section provides basic information on formulas.

- **Entering formulas**

  This section provides information on entering formulas in cells.

- **Relative versus absolute cell references**

  This section explains the difference between *relative* cell references (like =A1) and *absolute* cell references (like =$A$1) in formulas.

- **External cell references**

  The next section is about *external* cell references in formulas. These are cell references that refer to cells in other workbooks.

- **Error values**

  If a formula results in an error, an *error value* is displayed instead of the result. This section provides information on error values.

- **Working with arrays**

  This section provides instructions on working with arrays and array formulas.

- **Working with database functions**

  This section provides information on database functions such as DCount, DSum, etc.

- **Functions from A to Z**

  This section contains comprehensive descriptions of PlanMaker's arithmetic functions.

Tip: A short introduction to working with formulas can also be found in The PlanMaker tour.

Formula basics

If you want to perform a calculation in PlanMaker, first place the cell frame on the cell in which you want the result to appear. Then type the desired calculation into this cell.

Important When you enter a formula, you must always type an *equal sign* = first, as PlanMaker will consider your input as text or a numerical value otherwise.

Formulas can be composed of the following components:

- Fixed values (for example, a number or text)
- Cell references (for example, A1 or D2:D3)
- Arrays (see also Working with arrays)
- Arithmetic operators (for example, + or -, see also Operators in formulas)
- Arithmetic functions (for example, SUM)

The result of a calculation can represent both a final value and an intermediate result because any cell that contains the result of one calculation can be reused in another cell.

For example, a valid formula can look as follows:

```
= 67 + (A3 * sum(B1:C5))
```

Operators in formulas

Formulas can contain any of the operators listed below.

Please note that operators follow a strict *order of precedence*, which defines the evaluation order of expressions containing operators: Operators with higher precedence are evaluated before operators with lower precedence.

Parentheses can be used to override the order of precedence and force parts of a formula to be evaluated before other parts.

For example, 1+2*2 returns 5, since multiplication has precedence over addition and is thus calculated first. If the formula (1+2)*2 is used instead, the result will be 6, since the parentheses override the order of precedence.

Operators listed in order of precedence (highest first):

1st	Space	Intersection (see also INTERSECTION function)
2nd	%	Percentage
3rd	+	Positive sign (not addition!)
	–	Negative sign (negation)** (not subtraction!)
4th	^	Exponentiation**
5th	*	Multiplication

	/	Division
6th	+	Addition
	–	Subtraction
7th	&	Concatenation (For example, "Plan" & "Maker" returns "PlanMaker")
8th	=	Is equal to
	<>	Is not equal to
	<	Is less than
	>	Is greater than
	<=	Is less than or equal to
	>=	Is greater than or equal to

** **Note:** To ensure compatibility with Microsoft Excel, the order of precedence for the **exponentiation operator** (^) and the **negation operator** (-) is reversed but mathematically incorrect. For example, the correct mathematical order (first exponentiation, then sign) for the formula -2^2 would lead to the mathematically correct result -4. However, the actual result in PlanMaker (and in Excel) is 4.

Entering formulas

There are several ways to enter a calculation formula in a cell. These can be combined as desired.

Tip: A short introduction to entering formulas is also provided in The PlanMaker tour.

Entering formulas manually

You can easily enter formulas manually in the desired worksheet cell.

Important: When you enter formulas directly in a worksheet cell, you must always type the equal sign (=) first, so PlanMaker knows that this will be followed by a formula and not text.

When you have entered the formula, press the **Enter key**↵ to confirm your entry – or **Esc** to cancel it.

Alternatively, you can enter data via the following buttons of the Edit toolbar:

- To accept the input, click on the ✓ button.
- To cancel the input, click on the ✗ button.

To edit a formula later, navigate to the cell containing the formula and press the **F2** key. Alternatively, double-click on the cell.

Using the mouse to enter formulas

When entering a formula, you can use the mouse to paste cell ranges or single cells into the formula. Simply drag a rectangle in the worksheet or click on the desired cell.

An example:

You want to enter the formula =SUM(B2:D4) in a cell. You can do so as follows:

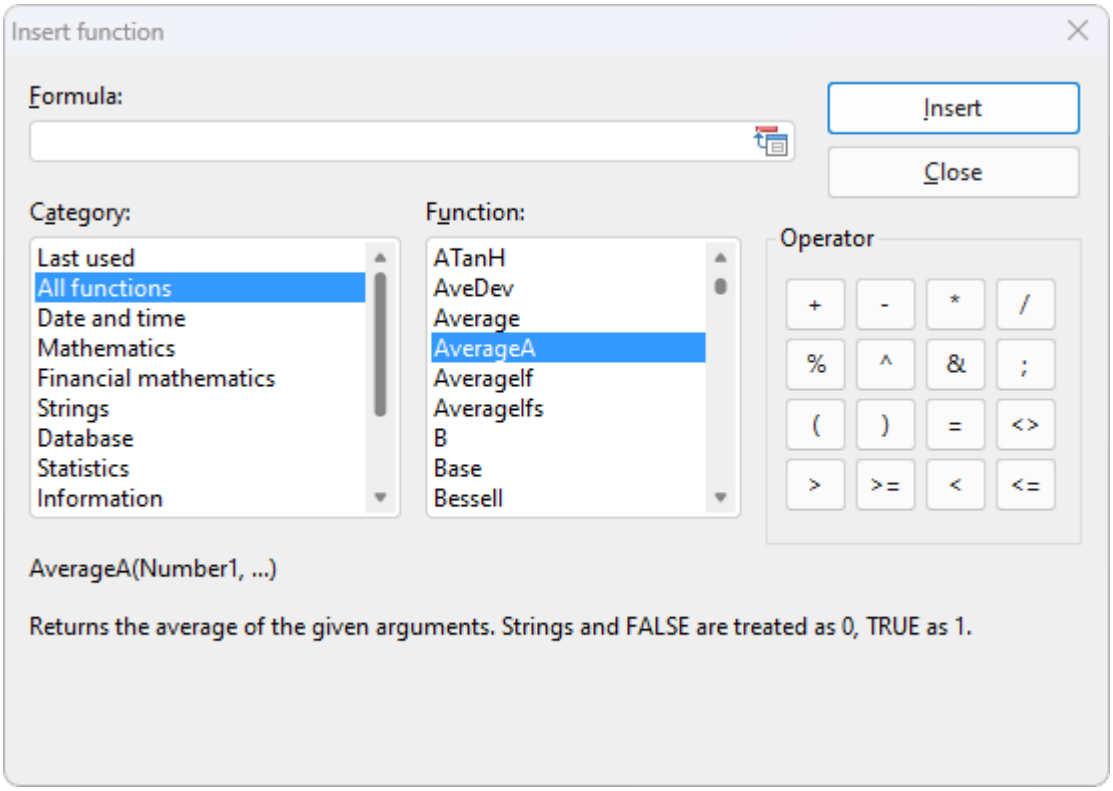
1. Enter =SUM(
2. Select cells B2 to D4 with the mouse. To do so, hold down the left mouse button and drag a rectangle that starts in cell B2 and ends in cell D4.
3. When you release the mouse button, the cell address is automatically pasted into the formula. The formula now displays: =SUM(B2:D4
4. Enter the closing parenthesis) and press the **Enter key**↵ to complete the formula.

The cell will now contain the formula =SUM(B2:D4).

The command Formula | Function

If you want to use arithmetic functions in formulas, you can simply enter them.

Alternatively, you can choose the ribbon command **Formula | Function** fx. This allows you to conveniently select formulas with the mouse.



The dialog box for this command has the following components:

- **Formula** field

  Here, you can edit the formula that is entered in the cell.

- **Category**

  In this list, you can select the types of calculation functions to be displayed in the **Function** list.

  In the *Named ranges* category at the end of the list, you can access your existing named cell ranges to insert them into formulas. For more information, see Using named ranges.

- **Function**

  This list displays all functions of the selected category. To copy a function to the **Formula** input field, double-click on it.

  Tip: If you select a function, a short description of it is automatically displayed in the lower part of the dialog box. You can also open a help page for each function by clicking on it and then pressing the **F1** key.

- **Operator**

  The most important arithmetic operators are displayed here. If you click on an operator, it will be inserted into the **Formula** input field.

- **Insert** button

  Click on this button to insert the contents of the **Formula** input field into the current cell and close the dialog box.

- **Close** button

  This button closes the dialog box without inserting the formula.

Relative versus absolute cell references

Cell references in calculations can be entered either as *relative* cell references (like =A1) or as *absolute* cell references (like =$A$1).

This section explains the difference between relative and absolute cell addressing:

Relative cell references

When you type the address of another cell in a formula, PlanMaker always interprets that cell reference as a *relative* cell reference.

An example: You enter the following formula in cell B3:

```
=B1+B2
```

PlanMaker *does not* interpret this as "Add B1 and B2", but rather as "Add the values in the two cells that are directly above cell B3".

You will notice this when you copy or move cell B3 to C3. PlanMaker then adapts the formula immediately:

```
=C1+C2
```

Absolute cell references

If you want to prevent the formula from being adapted automatically, you must type the cell reference as *absolute* by placing a dollar sign ($) in front of the column number and/or row number. You have the option of classifying complete cell addresses as absolute (for example, $B$1) or just one component of it (for example, $B1 or B$1).

An absolute cell reference always refers to the original cell address, regardless of where you subsequently move the cell with the formula.

For example: `=$B$1 + $B$2`

If you enter this formula in cell B3 and move this cell to C3, PlanMaker will still refer to cells B1 and B2.

Tip: Using the F4 key to change cell addressing

When you enter a cell reference in a formula, you can then use the **F4** key to switch between relative and absolute addressing.

If, for example, you have typed =A4 in a cell,

=$A$4 will be displayed if you press **F4** once,

=$A4 will be displayed if you press **F4** again,

=A$4 will be displayed if you press **F4** again.

If you press **F4** once again, the old entry =A4 will be displayed again.

External cell references

In calculations, you can also use cell references that refer to cells located in *different* PlanMaker or Excel documents. Such references are called *external cell references*.

In this section, you will learn everything you need to know about them. The following topics are covered:

- Entering external cell references
- Updating and managing external cell references

For more information, see the following pages.

Entering external cell references

Cell references that refer to cells in a different document have to be entered in calculations as follows:

='C:\My Folder\[My Workbook.pmdx]Sheet1'!A1

This example returns a reference to cell A1 in the worksheet "Sheet1" in the PlanMaker document "My Workbook.pmdx" in the folder "C:\My Folder".

As you can see, external cell references always consist of the following components:

- **Quotation mark**
  External cell references should always start with a quotation mark. You can use either single quotation marks (') or double quotation marks (").
- **Folder**
  Next, enter the entire path to the folder where the external file is located. For example: C:\My folder\
  Tip: If the external file is in the same folder as your document, you can omit the path.
- **File name in square brackets**
  You then have the file name enclosed within square brackets. Only PlanMaker and Excel files are allowed.
- **Sheet name**
  The next component is the name of the worksheet in which the desired cell is located.
- **Quotation mark**
  You then have to enter another quotation mark.
- **Exclamation mark**
  Next, you type an exclamation mark.
- **Cell reference**
  The last component is the address of the cell(s) to which you want to refer. Of course, cell ranges are also permitted.

To complete your input, press the **Enter key**↵. PlanMaker will then read the content of the cell from the specified file.

For information on updating and managing external cell references, see the next section.

Notes

- If PlanMaker is unable to find the specified file, it will display a dialog box in which you can select the file after you press the **Enter key**.

Tip: If you don't want to type in the entire path and file name manually, you can use the following trick: In the cell reference, simply enter [x.pmdx] as the path and file name – for example, **=[x.pmdx] Sheet1!A1**. Thus, you can intentionally trigger the appearance of the file dialog box after pressing **Enter** and conveniently select the file from the dialog box.

- If PlanMaker is unable to find the specified worksheet, it will display a dialog box in which you can select the worksheet after you press the **Enter key**.
- The path to the external file is usually saved absolutely. If the external document is in the same folder as your workbook (or a subfolder of it), however, PlanMaker stores only the relative path.

  In practice, this means that if your document and the external file are located in the *same* folder, references to this external file will stay intact even if you move both files to a different folder.

Tip: Quickly insert a reference via "Paste special"

If you want to avail of an alternative way to create external references without having to type in anything manually, proceed as follows:

1. Open the document that contains the cell to which you want to refer.
2. Copy this cell to the clipboard.
3. Switch back to the current document.
4. Navigate to the cell where you want the reference to be inserted.
5. Click on the arrow of the **Paste** icon (on the ribbon tab **Home** | group **Edit**) and choose the command **Paste special** from the dropdown menu.

   Alternatively: Right-click to open the context menu and select **Paste special** or simply use **Ctrl+Alt+V**.
6. In the following dialog box, click on the **Paste reference** button.

PlanMaker now creates an external reference to this cell.

Updating and managing external cell references

Choose the ribbon command **Data** | group **Edit** | **External references** to update and manage the external references in a document.

The command opens a dialog box that lists all the files to which there are external references.

The buttons in this dialog box enable you to perform the following actions:

- **Update**

  This option updates only those external cell references that refer to the file currently selected in the list. For more information, see "Notes on updating external cell references" below.

- **Update all**

  This option updates all external references in the entire document.

  Tip: Alternatively, you can trigger **Update all** via the ribbon command **Formula** | group **Update** | **Update data** | **Update external references**.

- **Open**

  This option opens the external file currently selected in the list.

- **Change**

  This option lets you choose a different file for the external file currently selected in the list. All references to this file will then be adapted accordingly in your document.

- **Previous**

  This option jumps to the previous cell that contains a reference to the file selected in the list.

- **Next**

  This option jumps to the next cell that contains a reference to the file selected in the list.

Click on the **Close** button to exit the dialog box.

Notes on updating external cell references

Important: The values returned by external cell references are not updated automatically each time the worksheet is recalculated.

External cell references are updated only in the following cases:

- When you open a document, all external cell references it contains are updated automatically (once).

  You can disable this automatic update – see the options described below.

- You can then manually update the external references at any time:

  To do so, choose the ribbon command **Data** | group **Edit** | **External references** and use the **Update** or **Update all** button in the dialog box as described above.

  Alternatively: Use the ribbon command **Formula** | group **Update** | **Update data** | **Update external references**, which does the same as the **Update all** button.

Options for updating external cell references

For updating external cell references, the document properties offer the following options.

To change these options, choose the ribbon command **File** | group **File management** | **Properties** and switch to the **Calculate** tab.

In the section **External references**, you will find the following options:

- **Update data on opening**

  Here, you can specify whether all external cell references in a document should be updated (once) when the document is opened:

  Always: When the document is opened, update its external references automatically.

  Never: When the document is opened, do not update its external references.

  Ask user: When the document is opened, ask the user if external references should be updated.

- **Save external data in document**

  If this option is enabled, PlanMaker saves a copy of all values returned by external cell references in the document. If the file for an external reference is missing when the external reference is updated, the value last saved will still be displayed.

  If you disable this option, an error value will be displayed instead if you update an external reference and the associated file cannot be accessed.

Note: The above options are document properties rather than global settings. This means that you can set them individually for each document.

Error values

If an error occurs in a calculation, it returns an *error value* as the result.

An example: A division is to be performed in a cell by the content of another cell. This other cell now takes a value of 0. PlanMaker cannot perform this division by zero, which is not allowed in mathematics. The *error value* #DIV/0! will now be displayed in the cell, which indicates that a division by zero is being performed.

The following error values exist:

Error value	Explanation
#REF!	Invalid cell reference. This is displayed when the formula contains a cell reference that is invalid or nonexistent after deleting cells (e.g., with the ribbon command **Home** | group **Cells** | **Delete**).
#DIV/0!	Division by zero. This is displayed when a formula contains a division by zero (which is never permitted in mathematics).
#NULL!	Invalid formula. This is displayed for misspelled or unknown function names, among other things, if open parentheses are not closed, or if a function is called with too many or too few arguments.
#NAME?	Reference with an invalid name. This displayed when a cell reference refers to a sheet name or range name that does not exist.
#N/A	"Not available". This is displayed when a cell contains a #N/A error value or a reference to a cell with a #N/A error value. For more information, see NA (error value #N/A).

#VALUE!	Invalid value. This is displayed when a function is used with an inappropriate parameter type (for example, if a number is passed to a function that is expecting a text string).
#NUM!	Invalid number. This is displayed when a function is used with a value that is outside the range of values permitted for the function (for example, SQRT(-2)).

Tip: To test if a cell returns an error value, use the functions ISERROR and ISERR.

Tip: You can also use the formula auditing from the command group **Traces** on the ribbon tab **Formula** to find and analyze cells that contain an error value. For more information, see Analyzing sheets.

Working with arrays

PlanMaker lets you enter *arrays* (also known as *matrices*) in spreadsheets and perform calculations with them. An array **A** is a rectangular number scheme in the following form:

$$\mathbf{A} = \begin{pmatrix} a_{11} & a_{12} & \dots & a_{1n} \\ a_{21} & a_{22} & \dots & a_{2n} \\ \vdots & & & \\ a_{m1} & a_{m2} & \dots & a_{mn} \end{pmatrix}$$

The entries a_{11} to a_{mn} are called the *elements* of the array. They are divided into *m* rows and *n* columns. This is why it is called an *m* x *n* array.

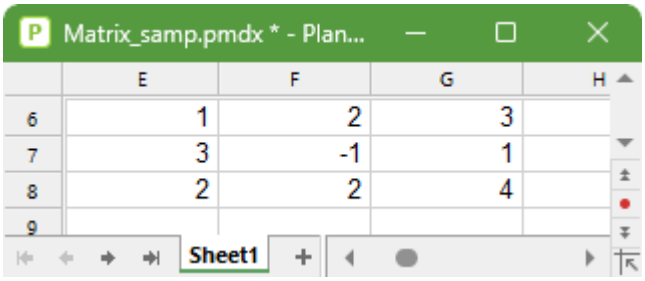
Entering arrays into corresponding cell ranges

To enter an array in PlanMaker, simply distribute the array's rows and columns over the worksheet's rows and columns.

For example, the following array ...

$$\begin{pmatrix} 1 & 2 & 3 \\ 3 & -1 & 1 \\ 2 & 2 & 4 \end{pmatrix}$$

... has to be entered as follows:



As you can see, each (rectangular) cell range can also be considered an array – and vice versa. Thus, for arithmetic functions that expect an array as an argument, you can always specify a cell range that contains the elements of the array.

Entering array formulas

PlanMaker provides *array functions* with which you can perform calculations with arrays – for example, find the inverse of an array. A formula containing an array function is called an *array formula*.

In contrast to "ordinary" formulas, array formulas only return an entire array of values rather than a single value. For this reason, such array formulas have to be entered in a different manner. Let's take a look at an example of this:

You want to determine the inverse of the 3x3 matrix shown above. To do so, proceed as follows:

1. **Select a cell range for the resulting array**

   Before entering an array formula, you have to select the cell range where the resulting array should be placed. The inverse of an array always has exactly as many rows and columns as the initial array. Thus, you have to select a range of 3x3 cells – for example, cells E10:G12.

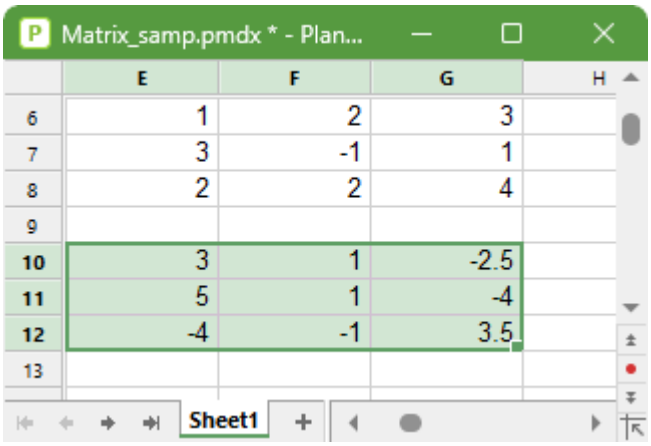
2. **Enter the array formula**

   Now enter the desired array formula. In our example, this would be the formula =MINVERSE(E6:G8).

3. **Ctrl+Shift+Enter**

> **Important:** To complete the formula, press the key combination **Ctrl+Shift+↵** rather than the **Enter key↵**.

The cells E10:G12 now contain the resulting array, i.e., the inverse of the array in E6:G8:



Notes:

- If you have selected a cell range larger than the resulting array, the error value #N/A ("not available") will be displayed in the superfluous cells. ***Important:*** If the selected cell range is too small, parts of the array will not be displayed.
- To edit an existing array formula: Select all cells of the resulting array, edit the formula and press **Ctrl+Shift+↵**. If you just press the **Enter key↵** instead, PlanMaker will issue a warning and ask you if you want to overwrite the array.
- **Tip:** To select all cells covered by an array formula, click on one of these cells and then press **Ctrl+7**.

Entering arrays with constants

If desired, arrays can be entered as *constants* instead of cell references. For this purpose, enclose the array within curly braces { }. Separate columns via commas and rows via semicolons.

For the array already used as an example above...

$$\begin{pmatrix} 1 & 2 & 3 \\ 3 & -1 & 1 \\ 2 & 2 & 4 \end{pmatrix}$$

... you could also enter the following in PlanMaker:

={1,2,3;3,-1,1;2,2,4}

Notes:

- The notation described above is only permitted for *fixed* values; formulas and cell references are *not* allowed.
- Of course, you can also enter *vectors* in the above notation. For a row vector such as **a** = (1, 2, 3), enter {1.2.3}; for a corresponding column vector, enter {1;2;3}.

Working with database functions

PlanMaker provides *database functions* that allow you to query and evaluate a cell range that is structured like a database.

All database functions expect the following syntax:

FUNCTION_NAME(DatabaseArea, Column, SearchCriteriaArea)

For example, the DSUM function returns the sum of all cells in the **DatabaseArea** cell range that are in the specified **Column** and match the conditions defined in the **SearchCriteriaArea** cell range.

To use the database functions, proceed as follows:

Creating a database area

A *database area* is any cell range containing the data to be evaluated. It has to be structured as follows:

- Its first row must contain labels for the database columns (e.g., Amount, Price, etc.).
- The data has to be arranged row by row (one row for each record).

The rows of the database area represent what is called a *record* in database applications, with the cells being the *fields* of the database.

Creating a search criteria area

The *search criteria area* is a cell range containing conditions that the database records have to match.

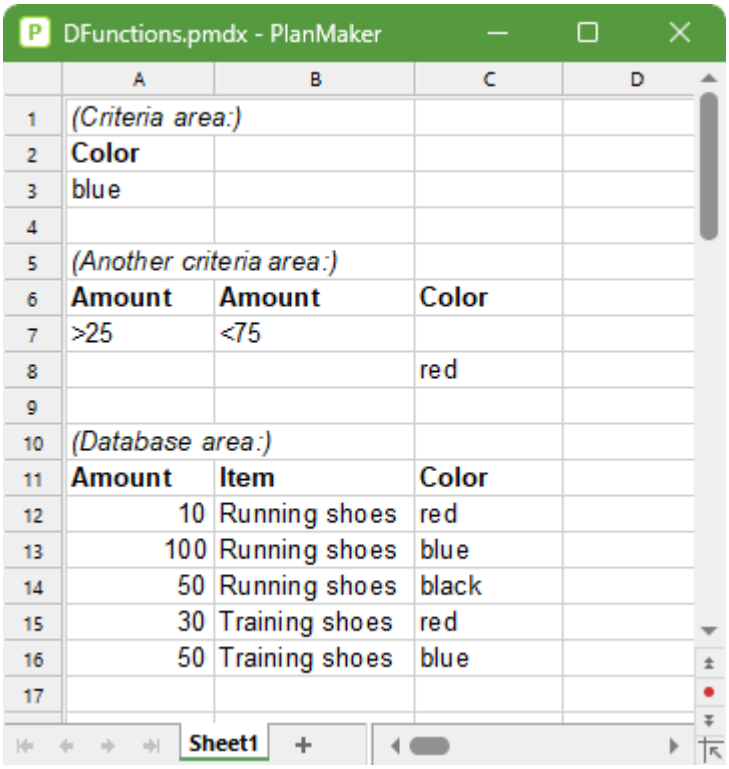
The search criteria area has to be structured as follows:

- ***Important:*** The search criteria area has to be located *above* the database area. The criteria area must not overlap the database area.
- The first row must also contain column labels. If, for example, you want to define criteria for the columns that you have assigned the labels Amount and Price in the database area, you must also enter Amount and Price here.
- The conditions to be met must be entered below the labels. You can enter either single values or comparisons like ">42" or "<10".
- If multiple conditions have been entered in the same row, the conditions in column 1 *and* column 2 *and* column 3, etc., must be met.
- If multiple rows with conditions have been entered, the conditions in row 1 *or* row 2 *or* row 3, etc., must be met.

Using database functions

After you have filled in the database area and search criteria area, you can use the database functions.

Here are some examples. Let's say you've created the following worksheet:



	A	B	C	D
1	*(Criteria area:)*			
2	**Color**			
3	blue			
4				
5	*(Another criteria area:)*			
6	**Amount**	**Amount**	**Color**	
7	>25	<75		
8			red	
9				
10	*(Database area:)*			
11	**Amount**	**Item**	**Color**	
12	10	Running shoes	red	
13	100	Running shoes	blue	
14	50	Running shoes	black	
15	30	Training shoes	red	
16	50	Training shoes	blue	
17				

For example, the following calculations can be performed here:

DCOUNT(A11:C16, , A2:A3) returns 2. This formula counts the number of records matching the condition Color = blue.

DCOUNT(A11:C16, , A6:B7) returns 3. This formula counts the number of records matching the condition (Amount > 25 and Amount < 75).

DCOUNT(A11:C16, , A6:C8) returns 4. This formula counts the number of records matching the condition (Amount > 25 and Amount < 75) *or* (Color = red).

DSUM(A11:C16, "Amount", A2:A3) returns 150. This formula calculates the sum of the values in the "Amount" column for all records matching the condition Color = blue.

DSUM(A11:C16, 1, A2:A3) returns 150. This is the same as above, except that the column was specified by its relative number instead of its label.

Functions from A to Z

The following pages contain an alphabetical list of PlanMaker's arithmetic functions, including comprehensive descriptions of each function.

ABS (absolute value)

Syntax:

ABS(Number)

Description:

This returns the absolute value of a number, i.e., removes its sign.

Example:

ABS(42) returns 42

ABS(0) returns 0

ABS(-42) returns 42

See also:

SIGN, NEG

ACOS (arccosine)

Syntax:

ACOS(Number)

Description:

The **ACOS** function (arccosine) is the inverse function of the COS function: It returns the inverse cosine of a number – that is, the angle whose cosine is the **Number**.

Number must be within the range from -1 to 1 (including -1 and 1).

The result is expressed in radians.

Example:

ACOS(1) returns 0

ACOS(COS(0.1234)) returns 0.1234

Tip:

You can use the DEGREES function to convert the result from radians to degrees.

See also:

COS, ASIN, ATAN

ACOSH (inverse hyperbolic cosine)

Syntax:

ACOSH(Number)

Description:

The **ACOSH** function (inverse hyperbolic cosine) is the inverse function of the COSH function: It returns the inverse hyperbolic cosine of a number – that is, the value whose hyperbolic cosine is the **Number**.

Example:

ACOSH(1.1) returns 0.44357

ACOSH(COSH(0.1234)) returns 0.1234

See also:

COSH

ACOT (arccotangent)

Syntax:

ACOT(Number)

Description:

The **ACOT** function (arccotangent) is the inverse function of the COT function: It returns the inverse cotangent of a number – that is, the angle whose cotangent is the **Number**.

The result is expressed in radians.

Example:

ACOT(1) returns 0.7854 (45°)

ACOT(COT(0.1234)) returns 0.1234

Tip:

You can use the DEGREES function to convert the result from radians to degrees.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

COT

ACOTH (inverse hyperbolic cotangent)

Syntax:

ACOTH(Number)

Description:

The **ACOTH** function (inverse hyperbolic cotangent) is the inverse function of the COTH function: It returns the inverse hyperbolic cotangent of a number – that is, the value whose COTH is the **Number**.

Example:

ACOTH(2) returns 0.54931

ACOTH(COTH(0.1234)) returns 0.1234

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

COTH

ADDRESS (cell address as text)

Syntax:

ADDRESS(Row, Column [, Abs] [, Mode] [, SheetName])

Description:

This returns a text string containing the address of a cell.

For **Row** and **Column**, the row number and column number of the cell must be specified.

The optional argument **Abs** specifies whether to deliver an absolute or relative address:

1 or omitted: Absolute row and column

2: Absolute row, relative column

3: Relative row, absolute column

4: Relative row and column

The optional argument **Mode** specifies the format in which the cell address is to be output:

TRUE or omitted: Normal cell address (for example, A1)

FALSE: Cell address in R1C1 format (an alternative type of cell addressing, supported by Microsoft Excel only)

The optional argument **SheetName** lets you specify a worksheet name. If it is specified, it is placed in front of the resulting cell reference. In other words:

If it is not specified, only the cell address will be returned.

If it is specified, *SheetName!Address* will be returned (see example).

Example:

ADDRESS(1, 5) returns $E$1

ADDRESS(1, 5, 4) returns E1

ADDRESS(1, 5, 4, , "MySheet1") returns MySheet1!E1

See also:

INDIRECT, OFFSET, COLUMN, ROW

AND (logical AND function)

Syntax:

AND(Value1 [, Value2, Value3 ...])

Description:

This returns the logical value TRUE if all of the given arguments are TRUE, otherwise it returns FALSE.

You can use it to check, for example, whether several conditions *all* apply.

Example:

AND(TRUE, TRUE) returns TRUE

AND(TRUE, FALSE) returns FALSE

AND(FALSE, FALSE) returns FALSE

AND(A1>0, A1<1) returns TRUE if A1 is greater than zero *and* less than one.

See also:

OR, XOR, NOT

AREAS (number of areas)

Syntax:

AREAS(reference)

Description:

This determines how many cell ranges the specified cell reference contains.

Example:

AREAS(A1:B2) returns 1

AREAS((A1:B2, E5:G7)) returns 2

See also:

INDEX, COLUMNS, ROWS

ASIN (arcsine)

Syntax:

ASIN(Number)

Description:

The **ASIN** function (arcsine) is the inverse function of the SIN function: It returns the inverse sine of a number – that is, the angle whose sine is the **Number**.

Number must be within the range from -1 to 1 (including -1 and 1).

The result is expressed in radians.

Example:

ASIN(1) returns 1.5708... (corresponds to PI()/2, thus 90°)

ASIN(SIN(0.1234)) returns 0.1234

Tip:

You can use the DEGREES function to convert the result from radians to degrees.

See also:

SIN, ACOS, ATAN

ASINH (inverse hyperbolic sine)

Syntax:

ASINH(Number)

Description:

The **ASINH** function (inverse hyperbolic sine) is the inverse function of the SINH function: It returns the inverse hyperbolic sine of a number – that is, the value whose SINH is the **Number**.

Example:

ASINH(1.1) returns 0.9503

ASINH(SINH(0.1234)) returns 0.1234

See also:

SINH

ATAN (arctangent)

Syntax:

ATAN(number)

Description:

The **ATAN** function (arctangent) is the inverse function of the TAN function: It returns the inverse tangent of a number – that is, the angle whose tangent is the **Number**.

The result is expressed in radians.

Example:

ATAN(1) returns 0.7854 (45°)

ATAN(TAN(0.1234)) returns 0.1234

Tip:

You can use the DEGREES function to convert the result from radians to degrees.

See also:

ATAN2, TAN, ACOS, ASIN

ATAN2 (arctangent 2)

Syntax:

ATAN2(x, y)

Description:

This returns the arctangent of the given coordinates, i.e., the angle between the x axis and a line from the origin (0, 0) to a point with the coordinates (**x**, **y**).

The **x** and **y** coordinates of this point must be given as arguments.

The result is expressed in radians.

Example:

ATAN2(1, 1) returns 0.7854 (45°)

Tip:

You can use the DEGREES function to convert the result from radians to degrees.

See also:

ATAN, TAN

ATANH (inverse hyperbolic tangent)

Syntax:

ATANH(Number)

Description:

The **ATANH** function (inverse hyperbolic tangent) is the inverse function of the TANH function: It returns the inverse hyperbolic tangent of a number – that is, the value whose TANH is the **Number**.

Number must be within the range from -1 to 1 (excluding -1 and 1).

Example:

ATANH(0.5) returns 0.54931

ATANH(TANH(0.1234)) returns 0.1234

See also:

TANH

AVEDEV (average deviation)

Syntax:

AVEDEV(Number1 [, Number2, Number3 ...])

Description:

This returns the average deviation of the given numbers from their mean.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

AVEDEV(2, 2, 2, 2) returns 0

AVEDEV(2, 2, 2, 3) returns 0.375

See also:

DEVSQ, STDEV.S/STDEV, VAR.S/VAR

AVERAGE (arithmetic mean)

Syntax:

AVERAGE(Number1 [, Number2, Number3 ...])

Description:

This returns the arithmetic mean of the given numbers.

This is calculated by dividing their sum by their number.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

AVERAGE(1, 2, 6) equals (1+2+6) / 3, which is 3.

If the cells A1:A4 contain the values 0, 1, 2 and the logical value TRUE, the following applies:

AVERAGE(A1:A4) equals (0+1+2) / 3, which is 1. The cell with the logical value is not considered here – not even when determining the number of values.

AVERAGE(A1:A4, TRUE) equals (0+1+2+1) / 4, which is 1. Since the entered value TRUE is evaluated as 1, FALSE is evaluated as 0 accordingly.

See also:

AVERAGEA, GEOMEAN, HARMEAN, TRIMMEAN, MEDIAN

AVERAGEA (arithmetic mean)

Syntax:

AVERAGEA(Value1 [, Value2, Value3 ...])

Description:

The **AVERAGEA** function is almost identical to the AVERAGE function (see note). It returns the arithmetic mean of the given values.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

Note:

Unlike the AVERAGE function, AVERAGE**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Example:

AVERAGEA(1, 2, 6) equals (1+2+6) / 3, which is 3.

If the cells A1:A4 contain the values 0, 1, 4 and the logical value TRUE, the following applies:

AVERAGEA(A1:A4) equals (0+1+4+1) / 4, which is 1.5. The logical value TRUE is interpreted as 1 here.

For comparison:

AVERAGE(A1:A4) equals (0+1+4) / 3, which is 1.66667. The logical value is ignored here.

See also:

AVERAGE

AVERAGEIF (average if condition is true)

Syntax:

AVERAGEIF(Range, Criterion [, AverageRange])

Description:

This returns the arithmetic mean of those values in a cell range that fulfill the specified criterion.

Range is the cell range to be evaluated.

Criterion is the condition that the values in **Range** have to fulfill to be included:

Use numbers or text (like "42" or "screws") to obtain the average of all cells that contain that value.

Use conditions (like ">10" or "<=5") to obtain the average of all cells that match the specified condition.

Note: Criteria always have to be enclosed within double quotation marks (").

AverageRange (optional) lets you specify the cell range that contains the values to be used for calculating the average. If **AverageRange** is omitted, the values in **Range** will be used.

Compatibility notes:

This function is not supported by the **.xls** file format (used in Microsoft Excel 2003 and earlier). If you save a document in this format, all calculations using this function will be replaced with their current result as a *fixed value*.

If you want to use this function, you should not save your document in the "Microsoft Excel 97-2003 (.xls)" file format, but choose one of the following formats instead:

- "PlanMaker document (.pmdx or .pmd)"
- or: "Microsoft Excel 2007-2021 (.xlsx)"

Example:

The cells A1:A5 contain the values 1, 2, 3, 2, 1.

Then the following applies:

AVERAGEIF(A1:A5, "2") returns 2

AVERAGEIF(A1:A5, ">=2") returns (2+3+2)/3, which is 2.33333

See also:

AVERAGE, AVERAGEIFS, SUMIF, SUMIFS, COUNTIF, COUNTIFS

AVERAGEIFS (average if conditions are true)

Syntax:

AVERAGEIFS(AverageRange, Range1, Criterion1 [, Range2, Criterion2 ...])

Description:

This returns the arithmetic mean of those values in a cell range that fulfill *all* of the specified criteria.

This function is similar to the AVERAGEIF function, but it allows you to specify multiple search ranges (including one criterion for each range).

Note: The order of the parameters is different here: In AVERAGEIF, **AverageRange** is the last parameter – while it is the first parameter in AVERAGEIFS.

AverageRange is the cell range that contains the values whose average is to be calculated.

Range1 is the first cell range to be checked for the specified criterion. This range must have the same dimensions as **AverageRange**.

Criterion1 is the condition that the values in **Range1** have to fulfill to be included:

Use numbers or text (like "42" or "screws") to obtain the average of all cells that contain that value.

Use conditions (like ">10" or "<=5") to obtain the average of all cells that match the specified condition.

Note: Criteria always have to be enclosed within double quotation marks (").

Unlike the AVERAGEIF function, AVERAGEIFS allows you to specify multiple search ranges (including one criterion for each range). If you do so, only occurrences where *all* criteria are fulfilled are included in the calculation of the average.

Compatibility notes:

This function is not supported by the **.xls** file format (used in Microsoft Excel 2003 and earlier). If you save a document in this format, all calculations using this function will be replaced with their current result as a *fixed value*.

If you want to use this function, you should not save your document in the "Microsoft Excel 97-2003 (.xls)" file format, but choose one of the following formats instead:

- "PlanMaker document (.pmdx or .pmd)"
- or: "Microsoft Excel 2007-2021 (.xlsx)"

Example:

AVERAGEIFS(A1:A10, C1:C10, ">1", C1:C10, "<2")

This formula returns the average of only those cells in the cell range A1:A10 where the corresponding cell in the range C1:C10 is greater than 1 and less than 2.

See also:

AVERAGE, AVERAGEIF, SUMIF, SUMIFS, COUNTIF, COUNTIFS

B (compatibility function)

Syntax:

B(n, p, k1 [, k2])

where:

n is the number of independent trial runs.

p is the probability of success for each trial run (0 to 100%). The value must be > 0 and < 1.

k1 is the lower limit for the number of successes.

k2 (optional) is the upper limit for the number of successes.

Description:

This returns the probability of obtaining exactly **k1** successes or between **k1** and **k2** successes in a binomial experiment with **n** independent trials.

Note:

This function was retained only for compatibility with PlanMaker 97. We recommend that you use the following Excel-compatible function instead:

B(n, p, k1) equals BINOMDIST(k1, n, p, FALSE) or BINOM.DIST(k1, n, p, FALSE)

B(n, p, k1, k2) equals BINOM.DIST.RANGE(n, p, k1, k2)

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

In Excel 2013, a function identical to the B function was introduced: BINOM.DIST.RANGE. PlanMaker also supports this function. However, please bear in mind that older versions of Excel (and PlanMaker) do not know this function.

See also:

BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST

BASE (convert decimal number into another base)

Syntax:

BASE(Number, Base [, Digits])

Description:

This converts the given decimal number (base 10) to a number with a different base – for example, a binary number (base 2) or a hexadecimal number (base 16).

Note: Please note that the result of this function is a text string and not a number.

Number is the number to be converted. Only positive numbers are allowed.

Base is the base into which the number is to be converted. Only integers between 2 and 36 are allowed.

The optional argument **Digits** lets you specify the *minimum* number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places. **Digits** must be a positive integer.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

BASE(42, 2) returns 101010 (decimal number 42 displayed as a binary number)

BASE(42, 2, 8) returns 00101010 (ditto, with the minimum places set to 8)

See also:

DECIMAL, DEC2BIN, DEC2HEX, DEC2OCT

BESSELI (modified Bessel function In(x))

Syntax:

BESSELI(x, n)

Description:

This returns the modified Bessel function $I_n(x)$.

x is the value to be evaluated.

n is the order of the Bessel function. **n** must be ≥ 0. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

BESSELJ, BESSELK, BESSELY

BESSELJ (Bessel function Jn(x))

Syntax:

BESSELJ(x, n)

Description:

This returns the Bessel function $J_n(x)$.

x is the value to be evaluated.

n is the order of the Bessel function. **n** must be ≥ 0. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

BESSELI, BESSELK, BESSELY

BESSELK (modified Bessel function Kn(x))

Syntax:

BESSELK(x, n)

Description:

This returns the modified Bessel function $K_n(x)$.

x is the value to be evaluated.

n is the order of the Bessel function. **n** must be ≥ 0. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

BESSELI, BESSELJ, BESSELY

BESSELY (Bessel function Yn(x))

Syntax:

BESSELY(x, n)

Description:

This returns the Bessel function $Y_n(x)$ (also known as the Weber function or the Neumann function).

x is the value to be evaluated.

n is the order of the Bessel function. **n** must be ≥ 0. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

BESSELI, BESSELJ, BESSELK

BETADIST (beta distribution)

Note: BETADIST is supplemented by the new function BETA.DIST with additional functionality, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

BETADIST(X, Alpha, Beta [, A, B])

Description:

This returns the beta cumulative distribution function.

X is the value whose probability is to be calculated.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

A and **B** (optional) determine the limits of the interval for X. If they are omitted, **A** is set to 0 and **B** is set to 1 (thus leading to a normal distribution).

X must be in the range **A** to **B**.

See also:

BETA.DIST, BETA.INV/BETAINV

BETA.DIST (beta distribution)

Syntax:

BETA.DIST(X, Alpha, Beta, Cumulative [, A, B])

Description:

This returns the (cumulative) beta distribution function.

X is the value whose probability is to be calculated.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative distribution function of the beta distribution is calculated. By entering the value FALSE, the probability density function of the beta distribution is calculated.

A and **B** (optional) determine the limits of the interval for X. If they are omitted, **A** is set to 0 and **B** is set to 1 (thus leading to a normal distribution).

X must be in the range **A** to **B**.

Note:

The BETA.DIST function supplements the previous BETADIST function with the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

The inverse function is BETA.INV

See also:

BETADIST, BETA.INV/BETAINV

BETAINV (percentiles of the beta distribution)

Note: BETAINV is supplemented by the new identical function BETA.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

BETAINV(Probability, Alpha, Beta [, A, B])

Description:

This returns percentiles of a beta distribution.

Probability is the given probability from the beta distribution. It must be > 0 and ≤ 1.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

A and **B** (optional) determine the limits of the interval. If they are omitted, **A** is set to 0 and **B** is set to 1 (thus leading to a normal distribution).

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

BETA.INV, BETA.DIST/BETADIST

BETA.INV (percentiles of the beta distribution)

Syntax:

BETA.INV(Probability, Alpha, Beta [, A, B])

Description:

This returns percentiles of a beta distribution.

Probability is the given probability from the beta distribution. It must be > 0 and ≤ 1.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

A and **B** (optional) determine the limits of the interval. If they are omitted, **A** is set to 0 and **B** is set to 1 (thus leading to a normal distribution).

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

BETA.INV is the inverse function of BETA.DIST

See also:

BETAINV, BETA.DIST/BETADIST

BIN2DEC (binary number to decimal number)

Syntax:

BIN2DEC(Number)

Description:

This converts the given binary number (base 2) to a decimal number (base 10).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

Number must not contain more than 32 places, thus the permissible value range is:

-(2^31) ≤ *Number* ≤ (2^31)-1

Example:

BIN2DEC("101010") returns 42

BIN2DEC("00101010") also returns 42

BIN2DEC("11111111111111111111111111010110") returns -42

See also:

DEC2BIN, BIN2HEX, BIN2OCT, DECIMAL

BIN2HEX (binary number to hexadecimal number)

Syntax:

BIN2HEX(Number [, Digits])

Description:

This converts the given binary number (base 2) to a hexadecimal number (base 16).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

Number must not contain more than 32 places, thus the permissible value range is:

-(2^31) ≤ *Number* ≤ (2^31)-1

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

BIN2HEX("101010") returns 2A

BIN2HEX("00101010") also returns 2A

BIN2HEX("101010", 1) returns the #NUM! error value, as this number can only be represented with at least 2 digits

BIN2HEX("11111111111111111111111111010110") returns FFFFFFD6 (negative value)

See also:

HEX2BIN, BIN2DEC, BIN2OCT

BIN2OCT (binary number to octal number)

Syntax:

BIN2OCT(Number [, Digits])

Description:

This converts the given binary number (base 2) to an octal number (base 8).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

Number must not contain more than 32 places, thus the permissible value range is:

-(2^31) ≤ *Number* ≤ (2^31)-1

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

BIN2OCT("101010") returns 52

BIN2OCT("00101010") also returns 52

BIN2OCT("101010", 1) returns the #NUM! error value, as this number can only be represented with at least 2 digits

BIN2OCT("11111111111111111111111111010110") returns 37777777726 (negative value)

See also:

OCT2BIN, BIN2DEC, BIN2HEX

BINOMDIST (binomial distribution)

Note: BINOMDIST is supplemented by the new identical function BINOM.DIST and the completely new function BINOM.DIST.RANGE, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

BINOMDIST(k, n, p, Cumulative)

Description:

This returns the binomial distribution function.

The binomial distribution can be used to determine the probability of obtaining **k** successes in an experiment with **n** independent trials with only two possible outcomes (success or failure) with a fixed probability of **p** for each trial.

For example, you can use this function to determine the probability of obtaining "heads" exactly 4 times if you toss a coin 10 times.

k is the number of successes, thus 4 here. It must be within the range 0 to **n**.

n is the number of independent trial runs, thus 10 here.

n and **k** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

p is the probability of the success of each trial run, for example, the probability of obtaining "heads" here, i.e., 50%.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned. BINOMDIST thus determines the probability of having *exactly* **k** successes.

TRUE: The cumulative distribution function is returned. BINOMDIST thus determines the probability that there are *at most* **k** successes.

Example:

Let's take the above example:

When you toss a coin 10 times (n=10), what is the probability of it landing on "heads" (p=50%) *exactly* 4 times?

BINOMDIST(4, 10, 50%, FALSE) returns 0.20508 =20.5%

What is the probability of it landing on "heads" 4 times *at most*?

Instead of the density function, you use the distribution function (specify the **Cumulative** argument with TRUE):

BINOMDIST(4, 10, 50%, TRUE) returns 0.37695 =37.7%

See also:

BINOM.DIST/BINOM.DIST.RANGE, BINOM.INV/CRITBINOM, B, COMBIN, NEGBINOM.DIST/NEGBINOMDIST, POISSON.DIST/POISSON

BINOM.DIST (binomial distribution)

Syntax:

BINOM.DIST(k, n, p, Cumulative)

Description:

This returns the probabilities of a binomially distributed random variable.

The binomial distribution can be used to determine the probability of obtaining **k** successes in an experiment with **n** independent trials with only two possible outcomes (success or failure) with a fixed probability of **p** for each trial.

For example, you can use this function to determine the probability of obtaining "heads" exactly 4 times if you toss a coin 10 times.

k is the number of successes, thus 4 here. It must be within the range 0 to **n**.

n is the number of independent trial runs, thus 10 here.

n and **k** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

p is the probability of a success for each trial run, for example, the probability of obtaining "heads" here, i.e., 50%.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned. Thus, BINOM.DIST determines the probability that there are *exactly* **k** successes.

TRUE: The cumulative distribution function is returned. BINOM.DIST thus determines the probability that there are *at most* **k** successes.

Example:

Let's take the above example:

When you toss a coin 10 times (n=10), what is the probability of it landing on "heads" (p=50%) *exactly* 4 times?

BINOM.DIST(4, 10, 50%, FALSE) returns 0.20508 =20.5%

What is the probability of it landing on "heads" 4 times *at most*?

Instead of the density function, you use the distribution function (specify the **Cumulative** argument with TRUE):

BINOM.DIST(4, 10, 50%, TRUE) returns 0.37695 =37.7%

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

BINOMDIST/BINOM.DIST.RANGE, BINOM.INV/CRITBINOM, B, COMBIN, NEGBINOM.DIST/NEGBINOMDIST, POISSON.DIST/POISSON

BINOM.DIST.RANGE (binomial distribution)

Syntax:

BINOM.DIST.RANGE(n, p, k1 [,k2])

Description:

This returns the probability of success of a number of trials using a binomial distribution.

n is the number of independent trial runs. Must be ≥ 0.

p is the probability of the success of each trial run. It must be within the range 0 to 1. (0 corresponds to 0% ; 1 corresponds to 100%)

k1 is the number of successes. It must be within the range 0 to **n**.

k2 is optional and returns the probability that the number of successes will fall between k1 and k2. k2 must be within the range k1 to n.

n and **k1,k2** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

Additional info:

If the optional argument **k2** is omitted, the probability of the *exact* number of successes is calculated.

Example:

When you toss a coin 10 times (n=10), what is the probability of it landing on "heads" (p=50%) *exactly* 4 times?

BINOM.DIST.RANGE(10, 50%, 4) returns 0.20508 =20.5%

What is the probability of it landing on "heads" *between 4 and 8 times*?

BINOM.DIST.RANGE(10, 50%, 4, 8) returns 0.81738 =81.7%

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

BINOMDIST/BINOM.DIST, BINOM.INV/CRITBINOM, B, COMBIN, NEGBINOM.DIST/NEGBINOMDIST, POISSON.DIST/POISSON

BINOM.INV (binomial distribution)

Syntax:

BINOM.INV(n, p, Alpha)

Description:

This returns the smallest value for which the cumulative binomial distribution is greater than or equal to the criterion value **Alpha**.

You thus calculate the maximum number of times a certain event with probability **p** may occur when performing a sample with **n** repetitions before its cumulative probabilities assume a value greater than or equal to the error probability **Alpha**.

This function is applicable for *binomial distributions* only (see also BINOM.DIST/BINOMDIST function); thus, they must be independent draws where only two outcomes are possible: success (event occurs) or failure (event does not occur).

n is the number of independent trial runs.

p is the probability of the success of each trial run. The values must be within the range 0 to 1.

Alpha is the probability of error that must not be exceeded. The values must be within the range 0 to 1.

Example:

You know from previous deliveries that 7% of a certain electronic component is usually defective. To check this statement, you carry out a sample of 100 components for a new delivery. What is the maximum number of them that can be defective if the probability of error is not to exceed 1%?

BINOM.INV(100, 7%, 1%) returns 2

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

CRITBINOM, BINOM.DIST/BINOM.DIST.RANGE/BINOMDIST

CEILING (round up to a multiple of base)

Syntax:

CEILING(Number, Base)

Description:

This rounds **Number** up (away from zero) to the nearest multiple of **Base**.

Positive numbers are rounded to the nearest multiple of **Base** that is larger than **Number**.

Negative numbers are rounded to the nearest multiple of **Base** that is smaller than **Number**.

Number and **Base** must have the same sign, otherwise the function returns a #NUM! error value.

Example:

CEILING(42.5, 1) returns 43

CEILING(-42.5, -1) returns -43

CEILING(21, 5) returns 25

CEILING(12.3456, 0.01) returns 12.35

See also:

FLOOR, MROUND, ROUND, TRUNC

CELL (information about a cell)

Syntax:

CELL(Type [, Reference])

Description:

This returns various types of information about a cell.

Type specifies the type of information to return. **Type** can be one of the following text strings:

Type	Explanation
"address"	This returns the address of the cell (as a text string).
"width"	(This is not supported and thus returns a #N/A error value.)
"filename"	This returns the full path and filename of the document that contains the cell.
"color"	This returns 1 if the cell uses the number format option **Negative numbers in red**, or 0 if not.
"format"	(This is not supported and thus returns a #N/A error value.)
"contents"	This returns the value stored in the cell.
"parentheses"	(This is not supported and thus returns a #N/A error value.)
"prefix"	This returns one of the following text strings: ' if the cell contains left-aligned text ^ if the cell contains centered text " if the cell contains right-aligned text \ if the cell contains fill-aligned text "" if the cell contains anything else
"protect"	This returns 1 if the cell is protected, or 0 if not.
"col"	This returns the column number of the cell (as a number).
"type"	This returns one of the following text strings: b (for blank) if the cell is empty l (for label) if the cell contains text v (for value) if the cell contains anything else

"row"	This returns the row number of the cell.

Reference (optional) is the address of the cell to be evaluated.

If **Reference** is a range of cells, only the top left cell will be evaluated.

If **Reference** is omitted, the current cell will be evaluated. (Excel behaves differently, see the note below.)

Compatibility notes:

This function is not fully identical to Microsoft Excel's CELL function. The following restrictions apply:

1. Some types of information are not supported by PlanMaker (see the table above).

2. If the parameter **Reference** is omitted, PlanMaker evaluates the current cell, whereas Excel evaluates the cell that was last changed.

Example:

=CELL("row", B5) returns the row number of cell B5, i.e., 5.

See also:

FILENAME, COLUMN, ROW

CHAR (character from ANSI code)

Syntax:

CHAR(Number)

Description:

This returns a single character with the specified ANSI code.

Number must be at least 10.

Example:

CHAR(65) returns A

CHAR(32) returns a blank

CHAR(CODE("X") returns X

See also:

CODE

CHIDIST (chi-square distribution)

Note: CHIDIST is supplemented by the new identical function CHISQ.DIST.RT, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

CHIDIST(X, DegreesFreedom)

Description:

This returns the **right-tailed** probability of the chi-square distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1 and < 10^10. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

CHISQ.DIST.RT, CHISQ.INV.RT/CHIINV, CHISQ.TEST/CHITEST

CHIINV (percentiles of the chi-square distribution)

Note: CHIINV is supplemented by the new identical function CHISQ.INV.RT, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

CHIINV(Probability, DegreesFreedom)

Description:

This returns the inverse of the **right-tailed** probability of the chi-square distribution function (i.e., percentiles of this distribution).

Probability is the probability to be evaluated. The values must be within the range 0 to 1.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1 and < 10^10. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

CHISQ.INV.RT, CHISQ.TEST/CHITEST, CHISQ.DIST.RT/CHIDIST

CHISQ.DIST.RT (chi-square distribution)

Syntax:

CHISQ.DIST.RT(X, DegreesFreedom)

Description:

This returns the **right-tailed** probability of the chi-square distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1 and < 10^10. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

CHIDIST, CHISQ.INV.RT/CHIINV, CHISQ.TEST/CHITEST

CHISQ.INV.RT (percentiles of the chi-square distribution)

Syntax:

CHISQ.INV.RT(Probability, DegreesFreedom)

Description:

This returns the inverse of the **right-tailed** probability of the chi-square distribution function (i.e., percentiles of this distribution).

Probability is the probability to be evaluated. The values must be within the range 0 to 1.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1 and < 10^10. An integer is expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

CHIINV, CHISQ.TEST/CHITEST, CHISQ.DIST.RT/CHIDIST

CHISQ.TEST (chi-square test for independence)

Syntax:

CHISQ.TEST(ObservedValues, ExpectedValues)

Description:

This returns the chi-square test for independence.

A chi-square test provides a general criterion for the conformity of a population with a sample.

ObservedValues are the observed values to be tested against the expected values. **ExpectedValues** are the expected values.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

CHITEST, CHISQ.INV.RT/CHIINV, CHISQ.DIST.RT/CHIDIST

CHITEST (chi-square test for independence)

Note: CHITEST is supplemented by the new identical function CHISQ.TEST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

CHITEST(ObservedValues, ExpectedValues)

Description:

This returns the test statistics of a chi-square test for independence.

A chi-square test provides a general criterion for the conformity of a population with a sample.

ObservedValues are the observed values to be tested against the expected values. **ExpectedValues** are the expected values.

See also:

CHISQ.TEST, CHISQ.INV.RT/CHIINV, CHISQ.DIST.RT/CHIDIST

CHOICE (x>0, x=0, x<0?)

Syntax:

CHOICE(Number, IfGreater [, IfZero, IfLess])

Description:

This returns one of the following values:

IfGreater, if **Number** is greater than zero.

IfZero, if **Number** equals zero.

IfLess, if **Number** is less than zero.

The last two arguments are optional.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

CHOICE(42, "Greater", "Zero", "Less") returns Greater

CHOICE(0, "Greater", "Zero", "Less") returns Zero

CHOICE(-42, "Greater", "Zero", "Less") returns Less

See also:

IF

CHOOSE (choose value from list)

Syntax:

CHOOSE(Index, Value1 [, Value2, Value3 ...])

Description:

This returns the value with the specified **Index** from a list of values.

Index specifies the position in the list of arguments. If **Index** is 1, the first value will be returned, if **Index** is 2, the second value will be returned, etc.

If **Index** is smaller than 1 or greater than the total number of values, the function returns a #VALUE! error value.

Example:

CHOOSE(2, "One", "Two", "Three") returns Two

See also:

SWITCH, INDEX, VLOOKUP, HLOOKUP

CLEAN (remove non-printable characters)

Syntax:

CLEAN(Text)

Description:

This removes all non-printable characters (characters with an ANSI code below 32) from the specified text string.

See also:

TRIM

CODE (ANSI code of a character)

Syntax:

CODE(Text)

Description:

This returns the ANSI code of the first character in the specified text string.

Example:

CODE("A") returns 65

CODE("Arthur") also returns 65

CODE(CHAR(123)) returns 123

CODE("") returns 0

See also:

CHAR

COLUMN (column number of cells)

Syntax:

COLUMN([Reference])

Description:

This returns the column number(s) of a cell or cell range:

If **Reference** is a single cell, the column number of this cell is returned.

If **Reference** is a range of cells, their column numbers are returned as a horizontal array. Note: In this case, the formula has to be entered as an *array formula* (see Working with arrays).

If **Reference** is omitted, the column number of the cell that contains this function is returned.

Example:

COLUMN(D2) returns 4

COLUMN(D2:F4) returns {4,5,6}

COLUMN() returns 3 when you use this function in cell C5

See also:

COLUMNS, ROW, CELL, ADDRESS, OFFSET, INDIRECT

COLUMNS (number of columns in a range)

Syntax:

COLUMNS(Range)

Description:

This returns the number of columns in the specified cell range.

Example:

COLUMNS(A1:D5) returns 4

See also:

ROWS, COLUMN, AREAS

COMBIN (combinations)

Syntax:

COMBIN(n, k)

Description:

This returns the number of combinations of size **k** in a population of size **n**.

In plain language: **n** items are available in total. From these, a subset of exactly **k** items is to be drawn. This function calculates how many different subsets there can be.

n is the total number of items. **n** must be > **k**.

k is the number of items in each combination. **k** must be > 0 and < **n**.

n and **k** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

Let's consider a lottery in which 6 balls are drawn from 49 different numbered balls.

How many combinations are possible?

COMBIN(49, 6) returns 13983816

Thus, there are 13,983,816 possible combinations.

See also:

PERMUT, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST

COMPLEX (create complex number)

Syntax:

COMPLEX(Real, Imag [, Suffix])

Description:

This creates a complex number in the form x+yi or x+yj from the specified **Real** and **Imag**.

With **Suffix** (optional), you can determine which letter should be used for the imaginary unit: either "i" or "j" (lowercase!). If **Suffix** is not specified, "i" is automatically used.

Tip:

To enter complex numbers in cells, you do not actually need the COMPLEX function, but can also simply type in the value. For example, the complex number 2+4i can be entered either as =COMPLEX(2, 4) or simply as 2+4i.

The same applies for formulas – **except** for the fact that complex numbers have to be enclosed within double quotation marks ("). For example, instead of typing =IMPOWER(COMPLEX(2, 4)), you could also type =IMPOWER("2+4i").

Example:

COMPLEX(2, 4) returns 2+4i

COMPLEX(2, 4, "i") also returns 2+4i

COMPLEX(2, 4, "j") returns 2+4j

See also:

IMAGINARY, IMREAL

CONCAT (concatenate strings from list)

Syntax:

CONCAT(String1 [, String2, String3 ...])

Description:

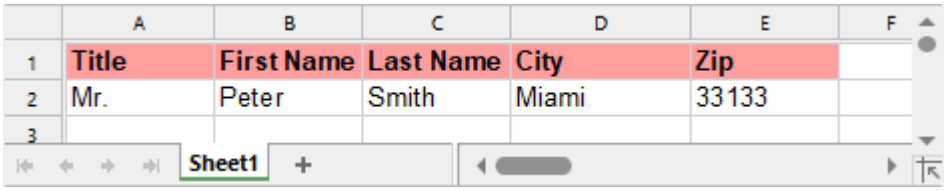
This combines two or more text strings into one text string. Unlike the CONCATENATE function, you can use CONCAT to select complete cell ranges and do not have to specify individual cells.

Example:

CONCAT("Good ", "Morning") returns Good Morning

CONCAT(2, " slices", " of ", "bread") returns 2 slices of bread

There is no difference so far to the CONCATENATE function. The advantage of CONCAT can be explained using the following example:



The function CONCATENATE(A2, B2, C2, D2, E2) returns MrPeterSmithMiami33133

The function CONCAT(A2:E2) also returns MrPeterSmithMiami33133

Conclusion: The new CONCAT function saves you from having to enter each cell individually.

For separators between the strings, however, you would have to enter the cells individually again and separate them with " ": CONCAT(A2, " ", B2, " ", C2, " ", D2, " ", E2).

In this case, the new function TEXTJOIN does a better job.

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

See also:

TEXTJOIN, CONCATENATE

CONCATENATE (concatenate text strings)

Note: CONCATENATE is supplemented by the new function CONCAT with additional functionality, which is available in newer versions of Microsoft Excel (2019 or later).

Syntax:

CONCATENATE(String1 [, String2, String3 ...])

Description:

This combines two or more text strings into one text string.

Additional info:

The **&** operator does exactly the same: CONCATENATE("Text1", "Text2") is equivalent to "Text1" & "Text2".

Example:

CONCATENATE("Good ", "Morning") returns Good Morning

CONCATENATE(2, " slices", " of ", "bread") returns 2 slices of bread

See also:

CONCAT, TEXTJOIN

CONFIDENCE (confidence interval)

Note: CONFIDENCE is supplemented by the new identical function CONFIDENCE.NORM, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

CONFIDENCE(Alpha, Sigma, n)

Description:

This returns a value that allows you to determine the 1-alpha confidence interval for the population mean of a distribution.

The 1-alpha confidence interval is a range of values around the sample mean x covering the interval from (x - CONFIDENCE) to (x + CONFIDENCE). The probability that this interval contains the population mean is 1-**Alpha**.

Alpha is the significance level for computing the confidence level. The values must be within the range 0 to 1.

If, for example, **Alpha** is 5%, the probability that the population mean is outside the confidence interval is 5%. The probability that it is inside is 1-**Alpha**, i.e., 95%. Thus, 1-alpha is also called the "confidence level".

Sigma is the population standard deviation. It must be > 0.

n is the size of the sample. It must be > 0.

Example:

The average weight of 1000 piglets is to be determined in a pig farm. For this purpose, a sample (with replacement) of the size **n**=65 is taken. The average weight determined is 30 kg. The standard deviation is 1.6 kg.

The 95% confidence interval (alpha is thus 0.05 here) is now to be determined. Thus, you would calculate as follows:

CONFIDENCE(0.05, 1.6, 65) returns 0.38897 (kg)

The average weight of the population (of all 1000 piglets) is thus with a probability of 95% between 30-0.38897 kg and 30+0.38897 kg.

See also:

CONFIDENCE.NORM

CONFIDENCE.NORM (confidence interval)

Syntax:

CONFIDENCE.NORM(Alpha, Sigma, n)

Description:

This returns a value that allows you to determine the 1-alpha confidence interval for the population mean of a distribution.

The 1-alpha confidence interval is a range of values around the sample mean x covering the interval from (x - CONFIDENCE) to (x + CONFIDENCE). The probability that this interval contains the population mean is 1-**Alpha**.

Alpha is the significance level for computing the confidence level. The values must be within the range 0 to 1.

If, for example, **Alpha** is 5%, the probability that the population mean is outside the confidence interval is 5%. The probability that it is inside is 1-**Alpha**, i.e., 95%. Thus, 1-alpha is also called the "confidence level".

Sigma is the population standard deviation. It must be > 0.

n is the size of the sample. It must be > 0.

Example:

The average weight of 1000 piglets is to be determined in a pig farm. For this purpose, a sample (with replacement) of the size **n**=65 is taken. The average weight determined is 30 kg. The standard deviation is 1.6 kg.

The 95% confidence interval (alpha is thus 0.05 here) is now to be determined. Thus, you would calculate as follows:

CONFIDENCE.NORM(0.05, 1.6, 65) returns 0.38897 (kg)

The average weight of the population (of all 1000 piglets) is thus with a probability of 95% between 30-0.38897 kg and 30+0.38897 kg.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

CONFIDENCE

CONVERT (unit conversion)

Syntax:

CONVERT(Number, From, To)

Description:

This converts a **Number** from one unit to another.

From and **To** are codes that represent the source unit and target unit (see table below).

Both units have to be in the same category. For example, you can convert meters into miles but not into minutes, of course.

An error value #N/A is displayed if a) **From** and **To** are not from the same category or b) one of the two unit codes was specified incorrectly.

Note: Unit codes are **case-sensitive**!

The following unit codes are available:

Mass	Code
Gram	g
Slug (unit of mass)	sg
Pound* (avoirdupois)	lbm
u (atom mass)	u
Ounce (avoirdupois)	ozm

* Does not correspond to the pound still partly used in German-speaking countries (½ kilo)

Distance	Code
Meter	m
Statute mile	mi
Nautical mile	Nmi
Inch	in
Foot	ft
Yard	yd
Ångström	ang
Pica	Pica
Light year	ly
Parsec (parallax second)	parsec

Time	Code
Second	sec
Minute	mn
Hour	hr
Day	day
Year*	yr
Fortnight**	ftn

* Based on a year with 365.25 days

** This unit is not supported by Microsoft Excel.

Liquid measure	Code
Liter	l (lowercase "L")
Teaspoon	tsp
Tablespoon	tbs
Fluid ounce	oz
Cup	cup
Pint (U.S.)	pt
Pint (U.K.)	uk_pt
Quart	qt

Gallon	gal

Force	Code
Newton	N
Dyne	dyn
Pound-force	lbf

Power	Code
Watt	W
HP (Horsepower)	HP

Magnetism	Code
Tesla	D
Gauss	ga

Energy	Code
Joule	J
Erg	e
Thermodynamic calorie	c
IT calorie	cal
Electron volt	eV
Horsepower-hour	HPh
Watt-hour	Wh
Foot-pound force	flb
BTU (British Thermal Unit)	BTU

Pressure	Code
Pascal	Pa
Atmosphere	atm
mm of Mercury	mmHg

Temperature	Code
Degree Fahrenheit	F

Degree Celsius	C
Degree Kelvin	K

Furthermore, prefixes can be added to metric units, "mg", for instance, stands for milligrams.

Please pay close attention to **capitalization** here also!

The following prefixes are available:

Prefix (and factor)	Code
Exa (10^18)	E
Peta (10^15)	P
Tera (10^12)	D
Giga (10^9)	G
Mega (10^6)	M
Kilo (1000)	k
Hecto (100)	h
Deka (10)	e
Deci (0.1)	d
Centi (0.01)	c
Milli (0.001)	m
Micro (10^-6)	u
Nano (10^-9)	n
Pico (10^-12)	p
Femto (10^-15)	f
Atto (10^-18)	a

Example:

CONVERT(1, "day", "sec") returns 86400 (days -> seconds)

CONVERT(1, "mi", "km") returns 1.60934 (miles -> kilometers)

CONVERT(100, "kW", "HP") returns 134.10201 (100 kW -> HP)

CONVERT(1, "aparsec", "cm") returns 3.08568 (Attoparsec -> cm)

CORREL (correlation coefficient)

Syntax:

CORREL(Range1, Range2)

Description:

This returns the correlation coefficient of two data sets.

The correlation coefficient is a measure of the relationship between two data sets.

Range1 and **Range2** are cell references to the two data sets. Empty cells, text and logical values are ignored.

The data sets must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 2, 5, 3 and B1:B3 contains 2, 7, 4, the following applies:

CORREL(A1:A3, B1:B3) returns 0.99718

See also:

FISHER, FISHERINV, COVARIANCE.P/COVARIANCE.S/COVAR

COS (cosine)

Syntax:

COS(Number)

Description:

This returns the cosine of an angle.

The angle must be specified in radians.

Tip:

You can use the RADIANS function to convert an angle given in degrees into radians.

Example:

COS(PI()) returns -1

COS(RADIANS(180)) returns -1

See also:

ACOS, SIN, TAN

COSH (hyperbolic cosine)

Syntax:

COSH(Number)

Description:

The function **COSH** returns the hyperbolic cosine of a number.

Example:

COSH(0.1) returns 1.005

See also:

ACOSH, SINH, TANH

COT (cotangent)

Syntax:

COT(Number)

Description:

This returns the cotangent of an angle.

The angle must be specified in radians.

Tip:

You can use the RADIANS function to convert an angle given in degrees into radians.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

COT(PI()/4) returns 1

COT(RADIANS(45)) returns 1

See also:

TAN, ACOT

COTH (hyperbolic cotangent)

Syntax:

COTH(Number)

Description:

The **COTH** function returns the hyperbolic cotangent of a number.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

COTH(0.45) returns 2.37024

See also:

ACOTH, TANH

COUNT (number of cells filled with numbers)

Syntax:

COUNT(Range1 [, Range2, Range3 ...])

Description:

This determines how many cells in one or more ranges are filled with *numbers*. This function only counts cells with numbers (including dates) and text representing a number.

Example:

If the cells A1:A3 contain 42, "Test" and TRUE and cell A4 is empty. The following applies:

COUNT(A1:A4) returns 1

For comparison: COUNT**A**(A1:A4) returns 3

Additional info:

The function COUNTA works in a similar manner, but it counts cells with *any* content

See also:

COUNTA, COUNTBLANK, COUNTP, FREQUENCY, ISBLANK, ISNUMBER, COUNTIF, COUNTIFS

COUNTA (number of cells filled)

Syntax:

COUNTA(Range1 [, Range2, Range3 ...])

Description:

This determines the number of cells containing any kind of value (thus, cells that are not empty).

The cells may contain any numbers, text strings, logical values or error values.

Example:

If cells A1:A3 contain 42, "Test" and TRUE and cell A4 is empty. The following applies:

COUNTA(A1:A4) returns 3

For comparison: COUNT(A1:A4) returns 1

Additional info:

The COUNT function works in a similar manner, but it only counts cells with *numbers*.

See also:

COUNT, ISBLANK

COUNTBLANK (number of empty cells)

Syntax:

COUNTBLANK(range)

Description:

This determines how many cells in a range are empty.

Example:

If the cells A1:A3 contain 42, "Test" and TRUE and cell A4 is empty. The following applies:

COUNTBLANK(A1:A4) returns 1

See also:

ISBLANK, COUNT

COUNTIF (count if condition is true)

Syntax:

COUNTIF(Range, Criterion)

Description:

This counts those cells in a cell range that fulfill the specified criterion.

Range is the cell range to be evaluated.

Criterion is the condition that the values in **Range** have to fulfill to be included:

On the one hand, you can simply specify a value here (for example, "42" or "screws"). In this case, all items that correspond to this value are counted.

Tip: To count all non-empty cells, specify "*" as the criterion. For example, COUNTIF(D2:D4, "*") counts all cells in the specified range that are not empty.

Furthermore, you can also use conditions like ">10" or "<=5". All items that match the specified condition are counted here.

Note: Criteria always have to be enclosed within double quotation marks (").

Example:

The cells A1:A5 contain the values 1, 2, 3, 2, 1.

Then the following applies:

COUNTIF(A1:A5, "2") returns 2

COUNTIF(A1:A5, ">=2") returns 3

Tip: A modification of the formula is required if a cell reference is entered for the search criterion instead of a constant. Here's an example:

Cells A1 to A5 contain the above example values. In addition, cell C1 contains the value 2.
You now want to find out how many values in the range A1:A5 are greater than the value in cell C1.

Then the formula is COUNTIF(A1:A5, ">"&C1) and returns 1

See also:

COUNT, AVERAGEIF, AVERAGEIFS, SUMIF, SUMIFS, COUNTIFS

COUNTIFS (count if conditions are true)

Syntax:

COUNTIFS(Range1, Criterion1 [, Range2, Criterion2 ...])

Description:

This applies criteria to the cells in the specified cell ranges and counts how often *all* of these criteria are fulfilled.

This function is similar to the COUNTIF function, but it allows you to specify multiple search ranges (including one criterion for each range).

Range1 is the first cell range to be checked for the specified criterion. Please note that all of the cell ranges specified must have the same dimensions.

Criterion1 is the condition that the values in **Range1** have to fulfill to be included:

On the one hand, you can simply specify a value here (for example, "42" or "screws"). In this case, the condition is considered to be fulfilled if the cell contains this value.

Furthermore, you can also use conditions like ">10" or "<=5". Here, the condition is considered to be fulfilled if the cell content matches this condition.

Note: Criteria always have to be enclosed within double quotation marks (").

Unlike the COUNTIF function, COUNTIFS allows you to specify multiple search ranges (including one criterion for each range). If you do so, the function counts only occurrences where *all* criteria are fulfilled. See examples below.

Compatibility notes:

This function is not supported by the **.xls** file format (used in Microsoft Excel 2003 and earlier). If you save a document in this format, all calculations using this function will be replaced with their current result as a *fixed value*.

If you want to use this function, you should not save your document in the "Microsoft Excel 97-2003 (.xls)" file format, but choose one of the following formats instead:

- "PlanMaker document (.pmdx or .pmd)"
- or: "Microsoft Excel 2007-2021 (.xlsx)"

Example:

COUNTIFS(C1:C10, ">1", C1:C10, "<2")

This formula counts how many cells in the cell range C1:C10 are greater than 1 and less than 2.

COUNTIFS(A1:D4, "tuna", G10:K14, ">500")

This formula counts how often the following conditions are met: The cell in A1:D4 contains "tuna" *and* the corresponding cell in G10:K14 contains a value greater than 500.

Tip: A modification of the formula is required if a cell reference is entered for the search criterion instead of a constant. Here's an example:

The cells A1:A5 contain the values 1, 2, 3, 4, 5. In addition, cell C1 contains the value 2.
You now want to find out how many values in the range A1:A5 are greater than the value in cell C1.

Then the formula is COUNTIF(A1:A5, ">"&C1) and returns 3

See also:

COUNT, AVERAGEIF, AVERAGEIFS, SUMIF, SUMIFS, COUNTIF

COUNTP (PlanMaker 97 compatibility function)

Syntax:

COUNTP(Range1 [, Range2, Range3 ...])

Description:

This determines how many cells in one or more ranges are filled with numbers or logical values.

Note:

This function was retained only for compatibility with PlanMaker 97. If possible, use the Excel-compatible functions COUNT and COUNTA instead of this function.

The difference between these three functions is as follows:

COUNTA counts all cells that are *not empty* (regardless of their content).

COUNT counts all cells that contain *numbers*.

COUNTP counts all cells that contain *numbers* or *logical values*.

Example:

Let's assume that cells A1:A3 contain 42, "Test" and TRUE and cell A4 is empty. The following applies:

COUNTA(A1:A4) returns 3

COUNT(A1:A4) returns 1

COUNTP(A1:A4) returns 2

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

COUNT, COUNTA

COVAR (population covariance)

Note: COVAR is supplemented by the new identical function COVARIANCE.P (population covariance) and the completely new function COVARIANCE.S (sample covariance), which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

COVAR(Range1, Range2)

Description:

This returns the **population** covariance of two data sets.

The covariance is the average of the products of the deviations for each pair of values. It can be used to determine the extent to which two data sets vary ("co-vary").

Range1 and **Range2** are cell references to the two data sets. These must contain integers. Empty cells, text and logical values are ignored.

The data sets must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 2, 5, 3 and B1:B3 contains 2, 7, 4, the following applies:

COVAR(A1:A3, B1:B3) returns 2.55556

See also:

COVARIANCE.P/COVARIANCE.S, FISHER, FISHERINV, CORREL

COVARIANCE.P (population covariance)

Syntax:

COVARIANCE.P(Range1, Range2)

Description:

This returns the **population** covariance of two data sets.

The covariance is the average of the products of the deviations for each pair of values. It can be used to determine the extent to which two data sets vary ("co-vary").

Range1 and **Range2** are cell references to the two data sets. These must contain integers. Empty cells, text and logical values are ignored.

The data sets must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 2, 5, 3 and B1:B3 contains 2, 7, 4, the following applies:

COVARIANCE.P(A1:A3, B1:B3) returns 2.55556

Note:

The COVARIANCE.P and COVARIANCE.S functions supplement the previous COVAR function, whereby COVARIANCE.P returns the **population** covariance just like COVAR and COVARIANCE.S returns the **sample** covariance.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

COVARIANCE.S/COVAR, FISHER, FISHERINV, CORREL

COVARIANCE.S (sample covariance)

Syntax:

COVARIANCE.S(Range1, Range2)

Description:

This returns the **sample** covariance of two data sets.

The covariance is the average of the products of the deviations for each pair of values. It can be used to determine the extent to which two data sets vary ("co-vary").

Range1 and **Range2** are cell references to the two data sets. These must contain integers. Empty cells, text and logical values are ignored.

The data sets must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 2, 5, 3 and B1:B3 contains 2, 7, 4, the following applies:

COVARIANCE.S(A1:A3, B1:B3) returns 3.83333

Note:

The COVARIANCE.S and COVARIANCE.P functions supplement the previous COVAR function, whereby COVARIANCE.S returns the **sample** covariance and COVARIANCE.P returns the **population** covariance just like COVAR.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

COVARIANCE.P/COVAR, FISHER, FISHERINV, CORREL

CREATEDATE (date the document was created)

Syntax:

CREATEDATE()

Description:

This returns the date/time the current document was created.

Example:

If a document was created on 31st of July, 2018 at 4:03 PM, the following applies:

CREATEDATE() returns 07/31/18 4:03 PM

Tip: By selecting a suitable date format in the dialog box of the ribbon tab **Home** | group **Number** ◢, you can determine whether only the date, only the time or both should be displayed.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

NOW, TODAY

CRITBINOM (binomial distribution)

Note: CRITBINOM is supplemented by the new identical function BINOM.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

CRITBINOM(n, p, Alpha)

Description:

This returns the smallest value for which the cumulative binomial distribution is greater than or equal to the criterion value **Alpha**.

You thus calculate the maximum number of times a certain event with probability **p** may occur when performing a sample with **n** repetitions before its cumulative probabilities assume a value greater than or equal to the error probability **Alpha**.

This function is applicable for *binomial distributions* only (see also BINOM.DIST/BINOMDIST function); thus, they must be independent draws where only two outcomes are possible: success (event occurs) or failure (event does not occur).

n is the number of independent trial runs.

p is the probability of the success of each trial run. The values must be within the range 0 to 1.

Alpha is the probability of error that must not be exceeded. The values must be within the range 0 to 1.

Example:

You know from previous deliveries that 7% of a certain electronic component is usually defective. To check this statement, you carry out a sample of 100 components for a new delivery. What is the maximum number of them that can be defective if the probability of error is not to exceed 1%?

CRITBINOM(100, 7%, 1%) returns 2

See also:

BINOM.INV, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST

CUMIPMT (cumulative interest)

Syntax:

CUMIPMT(Rate, NPer, PV, P1, P2, Type)

Description:

This returns the cumulative interest payments between period **P1** and period **P2**. This function is applicable to loans with periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how large the interest portion of the annuities is in the specified periods for a loan.

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

NPer is the total number of payment periods.

PV is the present value (initial amount)

P1 and **P2** define the time interval to be evaluated. **P1** is the first period, while **P2** is the last period. (The periods are numbered consecutively, starting with 1.)

Type is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

A loan of $100,000 is taken out under the following terms:

Annual interest rate: 10%

Payments: monthly, at the end of each month

Duration: 6 years

What is the interest portion of payments in periods (months) 24 to 36 in total?

CUMIPMT(10%/12, 6*12, 100000, 24, 36, 0) returns -7219.88.

Note that all values have to use the same time unit, *months* in this case, since the payments are made monthly.

Thus, in the above formula, the annual interest rate was divided by 12 to get the *monthly* interest rate and the number of payment periods was multiplied by 12.

Additional info:

The CUMPRINC function is the counterpart of CUMIPMT: It returns the total of the principal components.

See also:

CUMPRINC, PPMT, IPMT, PMT

CUMPRINC (cumulative principal)

Syntax:

CUMPRINC(Rate, NPer, PV, P1, P2, Type)

Description:

This returns the cumulative principal payments between period **P1** and period **P2**. This function is applicable to loans with periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how large the principal portion of the annuity is in the specified periods for a loan.

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

NPer is the total number of payment periods.

PV is the present value (initial amount)

P1 and **P2** define the time interval to be evaluated. **P1** is the first period, while **P2** is the last period. (The periods are numbered consecutively, starting with 1.)

Type is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

A loan of $100,000 is taken out under the following terms:

Annual interest rate: 10%

Payments: monthly, at the end of each month

Duration: 6 years

What is the principal portion of payments in periods (months) 24 to 36 in total?

CUMPRINC(10%/12, 6*12, 100000, 24, 36, 0) returns -16863.71.

Note that all values have to use the same time unit, *months* in this case, since the payments are made monthly.

Thus, in the above formula, the annual interest rate was divided by 12 to get the *monthly* interest rate and the number of payment periods was multiplied by 12.

Additional info:

The CUMIPMT function is the counterpart of CUMPRINC: It returns the total of the interest components.

See also:

CUMIPMT, PPMT, IPMT, PMT

CURRENCY (format number as currency)

Syntax:

CURRENCY(Number [, n])

Description:

This converts a number into a text string. The resulting text string contains the number rounded to **n** decimal places and formatted as a currency.

PlanMaker determines the currency to be displayed from your computer's country settings. For example, a dollar sign $ is displayed for users in the U.S., while a euro sign € is displayed for users in Germany, etc.

Number is the number to be converted.

The optional argument **n** determines the desired number of decimal places. If **n** is omitted, **Number** is rounded to 2 decimal places.

If **n** is a negative value, **Number** is rounded to the left of the decimal point. For example, if **n** is -2, **Number** is rounded to the nearest multiple of a hundred (see last example).

Note:

Note that the result of this function is a *text string*. To round a number without converting it into a text string, use one of the rounding functions (for example, ROUND) instead.

To *display* a number only rounded and formatted as currency, you can also change its number format using the commands of the ribbon tab **Home** | group **Number**.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

CURRENCY(123.456) returns the text string $123.46

CURRENCY(123.456, 2) returns the text string $123.46

CURRENCY(123.456, -2) returns the text string $100

DATE (create a date value)

Syntax:

DATE(Year, Month, Day)

Description:

This returns a date created from the specified **Year**, **Month** and **Day**.

Example:

DATE(2018, 9, 25) returns 43368

However, PlanMaker automatically formats this date value in the number format "Date/Time", thus the cell displays: 09/25/2018.

See also:

DATEVALUE, TIME, TIMEVALUE

DATEDIF (date difference)

Syntax:

DATEDIF(StartDate, EndDate, Mode)

Description:

This returns the number of days, months or years between two dates.

StartDate and **EndDate** are the two dates to be evaluated. StartDate must be less (i.e., earlier) than EndDate.

Mode determines how and in which unit the difference between the StartDate and the EndDate should be calculated:

"y"	Number of years between StartDate and EndDate
"m"	Number of months between StartDate and EndDate
"d"	Number of days between StartDate and EndDate
"ym"	Number of months, excluding years (as if StartDate and EndDate were in the same year)
"yd"	Number of days, excluding years (as if StartDate and EndDate were in the same year)
"md"	Number of days, excluding month and years (as if StartDate and EndDate were in the same month and year)

Note:

This function is an undocumented Excel function. Although it still exists in Microsoft Excel, it has been removed from Excel's help file and thus seems to be no longer officially supported.

Example:

To calculate the age of a person, enter their date of birth in cell A1 (e.g., 4/3/1970). In A2, enter the current date (e.g., 6/8/2012).

Calculate the person's age as follows:

Age in years:
DATEDIF(A1, A2, "y") returns 42 (years)

Age in months:
DATEDIF(A1, A2, "m") returns 506 (months)

Age in days:
DATEDIF(A1, A2, "d") returns 15407 (days)

Age in years, months and days:
DATEDIF(A1, A2, "y") returns 42 (years)
DATEDIF(A1, A2, "ym") returns 2 (months)
DATEDIF(A1, A2, "md") returns 5 (days)
(The person's age is thus 42 years, 2 months and 5 days.)

See also:

NOW, NETWORKDAYS, DAYS/DAYSP

DATEVALUE (convert text into date value)

Syntax:

DATEVALUE(Text)

Description:

This converts a text string representing a date to a serial date value.

Note:

This function was retained only for compatibility with older spreadsheet applications. It is **no** longer commonly used, as current spreadsheet applications (including PlanMaker) automatically convert text representing a date to a serial date value if necessary.

Example:

DATEVALUE("09/25/2018") returns 43368.

If you now use the options of the ribbon tab **Home** | group **Number** to format this in one of the date/time formats (for example M/D/YYYY), the cell will display the date behind this value: the 9/25/2018.

See also:

TIMEVALUE, DATE, TIME

DAVERAGE (database function)

Syntax:

DAVERAGE(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *average* of those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

AVERAGE, DPRODUCT, DSUM

DAY (day of a date)

Syntax:

DAY(Date)

Description:

This returns the day of a date.

Example:

DAY("09/25/2018") returns 25

DAY("17th August") returns 17

If cell A1 contains the date value 9/25/2018: DAY(A1) returns 25

See also:

WEEKDAY, ISOWEEKNUM, ISOWEEEK, WEEKNUM, MONTH, YEAR

DAYS (days between two dates)

Syntax:

DAYS(EndDate, StartDate)

Description:

This returns the number of days between two dates.

Please note the order of the parameters: The *end* date has to be specified first and then the start date.

Note:

This function was revised in PlanMaker 2016: The order of its two parameters was reversed.

Background info: The DAYS function has been available in PlanMaker for a long time. Excel did not have this function until Excel 2013 when a function with the same name was added. This also works exactly like the DAYS function in PlanMaker – but the order of the parameters in Excel is the other way around: first the end date and then the start date. In order to be compatible with Excel, this order was adjusted accordingly in PlanMaker 2016.

The previous version of the function was renamed to DAYSP. Thus, the functions DAYS and DAYSP are identical, and only the order of their parameters differs:

DAYS(EndDate, StartDate) equals

DAYSP(StartDate, EndDate)

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

DAYS("01/14/2018", "01/01/2018") returns 13

If cell A1 contains the date value 01/14/2018 and cell A2 contains the date value 01/01/2018:
DAYS(A1, A2) returns 13

See also:

DAYSP, DATEDIF, NETWORKDAYS, DAYS360, TIMEDIFF

DAYS360 (days between two dates)

Syntax:

DAYS360(StartDate, EndDate [, Method])

Description:

This returns the number of days between two dates – based on a year with 360 days (12 months with 30 days each) and ignoring leap years.

StartDate and **EndDate** are the two dates to be evaluated.

Method (optional) is a logical value that lets you specify how dates occurring on the 31st of a month should be evaluated:

FALSE or omitted: The *U.S. (NASD) method* is used: If **StartDate** is the 31st of a month, the 30th of the same month will be used instead. If **EndDate** is the 31st of a month, the 1st of the following month will be used instead. Exception: If StartDate *and* EndDate are the 31st of a month, the 30th of the same month will be used instead (for both dates).

TRUE: The *European method* is used: If **StartDate** or **EndDate** is the 31st of a month, the 30th of the same month will be used.

Note:

Thus, if you do not want to calculate according to the U.S. (NASD) method, the **Method** argument must be specified as TRUE.

Example:

DAYS360("01/01/2018", "02/01/2018", TRUE) returns 30

DAYS360("02/01/2018", "03/01/2018", TRUE) returns 30

DAYS360("01/01/2012", "01/01/2013", TRUE) returns 360

See also:

DATEDIF, NETWORKDAYS, DAYS/DAYSP, TIMEDIFF

DAYSP (compatibility function)

Syntax:

DAYSP(StartDate, EndDate)

Description:

Compatibility function for older PlanMaker documents.

This function has been replaced by the DAYS function. (Please note that the order of the parameters is reversed.)

See the notes below.

Note:

DAYSP is merely a compatibility function. When you open a document created with PlanMaker 2012 or earlier, all occurrences of the DAYS function are automatically converted into DAYSP. Reason: In PlanMaker 2016, the order of the parameters for the DAYS function was reversed.

For new documents, we recommend that you use the DAYS function (compatible with Excel 2013), which is identical – except for the order of its parameters:

DAYS(EndDate, StartDate) equals

DAYSP(StartDate, EndDate)

For more information, see DAYS.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

DAYSP("01/01/2018", "01/14/2018") returns 13

If cell A1 contains the date value 01/01/2018 and cell A2 contains the date value 01/14/2018:
DAYSP(A1, A2) returns 13

See also:

DAYS, DATEDIF, NETWORKDAYS, DAYS360, TIMEDIFF

DAYSPERMONTH (days per month)

Syntax:

DAYSPERMONTH(Date)

Description:

This returns the number of days in the specified month.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

DAYSPERMONTH("09/25/2018") returns 30 (days)

DAYSPERMONTH("10/25/2018") returns 31 (days)

If cell A1 contains the date value 09/25/2018: DAYSPERMONTH(A1) returns 30 (days)

See also:

DAYSPERYEAR, YEAR, DAYS/DAYSP, DAYS360

DAYSPERYEAR (days per year)

Syntax:

DAYSPERYEAR(Date)

Description:

This returns the number of days in the specified year.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

DAYSPERYEAR("01/01/2012") returns 366 (2012 was a leap year)

DAYSPERYEAR("01/01/2013") returns 365 (2013 was not a leap year)

If cell A1 contains the date value 1/1/2012, then DAYSPERYEAR(A1) returns 366

See also:

YEAR, DAYSPERMONTH, DAYS/DAYSP, DAYS360

DB (fixed-declining balance depreciation)

Syntax:

DB(Cost, Salvage, Life, Per [, Months])

Description:

This returns the depreciation of an asset for the specified period, using the fixed-declining balance method.

Cost = Initial cost of asset

Salvage = Salvage value (value at the end of the depreciation)

Life = Life of asset (in number of periods)

Per = Period to evaluate (in the same time unit as **Life**)

Months (optional) = Number of months in the first year (12 if omitted)

Example:

In the case of a machine purchased for €200,000 that is used for 6 years and then has a salvage value of €10,000, the following annual depreciation amount must be applied for the first year, broken down into 7 months:

DB(200000, 10000, 6, 1, 7) returns €45,850

See also:

DDB, SYD, SLN

DCOUNT (database function)

Syntax:

DCOUNT(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *number* of cells in a database area that match the conditions specified in the search criteria area.

Only cells containing numbers (including dates) are counted, while cells with text or logical values are ignored.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

If the argument **Column** is not empty, the function only considers cells from the corresponding column. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

DCOUNTA

DCOUNTA (database function)

Syntax:

DCOUNTA(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *number* of cells in a database area that match the conditions specified in the search criteria area.

Unlike the DCOUNT function, DCOUNT**A** not only counts cells containing numbers but also counts cells with text or logical values.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

If the argument **Column** is not empty, the function only considers cells from the corresponding column. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

DCOUNT

DDB (double-declining balance depreciation)

Syntax:

DDB(Cost, Salvage, Life, Per [, Factor])

Description:

This returns the depreciation of an asset for the specified period, using the double-declining balance method (or a variable-declining balance method if **Factor** is specified).

Cost = Initial cost of asset

Salvage = Salvage value (value at the end of the depreciation)

Life = Life of asset (in number of periods)

Per = Period to evaluate (in the same time unit as **Life**)

Factor (optional) = Rate at which the balance declines (2 if omitted)

Example:

In the case of a machine purchased for €200,000 that is used for 10 years and then has a salvage value of €10,000, the following annual depreciation amount must be applied for the first year:

DDB(200000, 10000, 10, 1) returns €40,000

For the second year:

DDB(200000, 10000, 10, 2) returns €32,000,

etc.

See also:

DB, SYD, SLN

DEC2BIN (decimal number to binary number)

Syntax:

DEC2BIN(Number [, Digits])

Description:

This converts the given decimal number (base 10) to a binary number (base 2).

> ***Note:*** Please note that the result of this function is a text string and not a number.

Number is the number to be converted.

The permissible value range for **Number** is:

-(2^31) ≤ *Number* ≤ (2^31)-1

That is equivalent to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case (see last example).

Example:

DEC2BIN(42) returns 101010

DEC2BIN(42, 8) returns 00101010

DEC2BIN(42, 4) returns the #NUM! error value because the number 42 cannot be represented with fewer than 6 binary digits

DEC2BIN(-42) returns 11111111111111111111111111010110

See also:

BIN2DEC, DEC2HEX, DEC2OCT, BASE

DEC2HEX (decimal number to hexadecimal number)

Syntax:

DEC2HEX(Number [, Digits])

Description:

This converts the given decimal number (base 10) to a hexadecimal number (base 16).

Note: Please note that the result of this function is a text string and not a number.

Number is the number to be converted.

The permissible value range for **Number** is:

-(2^31) ≤ *Number* ≤ (2^31)-1

That is equivalent to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case (see last example).

Example:

DEC2HEX(42) returns 2A

DEC2HEX(42, 8) returns 0000002A

DEC2HEX(42, 1) returns the #NUM! error value because the number 42 cannot be represented with fewer than 2 hexadecimal digits

DEC2HEX(-42) returns FFFFFFD6

See also:

HEX2DEC, DEC2BIN, DEC2OCT, BASE

DEC2OCT (decimal number to octal number)

Syntax:

DEC2OCT(Number [, Digits])

Description:

This converts the given decimal number (base 10) to an octal number (base 8).

Note: Please note that the result of this function is a text string and not a number.

Number is the number to be converted.

The permissible value range for **Number** is:

-(2^31) ≤ *Number* ≤ (2^31)-1

That is equivalent to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case (see last example).

Example:

DEC2OCT(42) returns 52

DEC2OCT(42, 8) returns 00000052

DEC2OCT(42, 1) returns the #NUM! error value because the number 42 cannot be represented with fewer than 2 octal digits

DEC2OCT(-42) returns 37777777726

See also:

OCT2DEC, DEC2BIN, DEC2HEX, BASE

DECIBEL (decibel value of two quantities)

Syntax:

DECIBEL(P2 [, P1])

Description:

This returns the decibel value (dB) for the ratio of two quantities.

The calculation is based on the following formula:

DECIBEL(P2, P1) = 10 * LOG10(P2/P1)

Note the order of the two parameters: The dividend **P2** of the quotient has to be specified first and then the divisor **P1** – not vice versa.

P2 and **P1** have to be numbers. Their quotient has to be greater than zero.

The parameter **P1** is optional. If it is omitted, PlanMaker sets it to 1 (one) automatically.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

DECIBEL(1000, 10) returns 20

DECIMAL (convert number from any base into decimal)

Syntax:

DECIMAL(Number, Base)

Description:

This converts a number from any base to decimal (base 10) – for example, a binary number (base 2) or a hexadecimal number (base 16).

Number is the number to be converted.

Base is the base of the number. Only integers between 2 and 36 are allowed.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

DECIMAL("00101010", 2) returns 42

DECIMAL("FF", 16) returns 255

See also:

BASE, BIN2DEC, HEX2DEC, OCT2DEC

DECIMALS (fractional part of a number)

Syntax:

DECIMALS(Number)

Description:

This returns the fractional part of the given number (i.e., the digits to the right of the decimal point).

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

DECIMALS(5.779) returns 0.779

Additional info:

To obtain the places before the decimal point, use the TRUNC function.

See also:

TRUNC

DEGREES (convert radians to degrees)

Syntax:

DEGREES(number)

Description:

This converts radians into degrees.

The following conversion formula is used:

DEGREES(x) = x * 180 / PI()

Example:

DEGREES(PI()) returns 180

See also:

RADIANS

DELTA (test for equality)

Syntax:

DELTA(Number1, Number2)

Description:

This tests if two values are equal. If they are, 1 is returned, otherwise 0 is returned.

Number1 and **Number2** must be numbers, otherwise a #VALUE! error value is returned.

However, text strings that are returned when using the TEXT function or some conversion functions, for example, are also permitted. The text strings may only contain numeric values.

Example:

DELTA(1, 2) returns 0

DELTA(2, 2) returns 1

DELTA(2, "2") returns 1

DELTA("Text", "Text") returns the #VALUE! error value

See also:

EXACT

DEVSQ (average square deviation)

Syntax:

DEVSQ(Number1 [, Number2, Number3 ...])

Description:

This returns the sum of squares of deviations of the specified numbers from their sample mean.

Empty cells, text strings and logical values are ignored.

Example:

DEVSQ(2, 2, 2, 3) returns 0.75

See also:

AVEDEV, STDEV.S/STDEV, SUM, VAR.S/VAR

DGET (database function)

Syntax:

DGET(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the exact value from a column of a database area that matches the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

Note:

If more than one value matches the specified criteria, the function returns a #NUM! error value.

DMAX (database function)

Syntax:

DMAX(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *largest* of those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

MAX, DMIN

DMIN (database function)

Syntax:

DMIN(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *smallest* of those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

MIN, DMAX

DOLLARDE (convert to dollar price decimal)

Syntax:

DOLLARDE(Value, Divisor)

Description:

This converts a dollar price expressed as a fraction into a decimal dollar price.

(In the USA, it is common practice for some types of financial transactions to represent monetary amounts as fractions – for example, 27 4/8 instead of 27.50.)

Value is the number to be converted.

Divisor is the denominator of the fraction. It must be greater than or equal to 1. **Divisor** should be an integer. Digits to the right of the decimal point are ignored.

Example:

DOLLARDE(27.4, 8) converts the fraction 27 4/8 into the decimal value 27.5.

See also:

DOLLARFR

DOLLARFR (convert to dollar price fraction)

Syntax:

DOLLARFR(Value, Divisor)

Description:

This converts a decimal dollar price into a dollar price expressed as a fraction.

(In the USA, it is common practice for some types of financial transactions to represent monetary amounts as fractions – for example, 27 4/8 instead of 27.50.)

Value is the number to be converted.

Divisor is the denominator of the fraction. It must be greater than or equal to 1. **Divisor** should be an integer. Digits to the right of the decimal point are ignored.

Example:

DOLLARFR(27.5, 8) returns 27.4 (since 27.5 equals the fraction 27 4/8)

See also:

DOLLARDE

DPRODUCT (database function)

Syntax:

DPRODUCT(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *product* of those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

PRODUCT, DAVERAGE, DSUM

DSTDEV (database function)

Syntax:

DSTDEV(DatabaseArea, Column, SearchCriteriaArea)

Description:

This estimates the *standard deviation* of a population based on a sample, using those values in a column of a database area that match the conditions specified in the search criteria area.

The function assumes that the determined values represent a sample from the population. On this basis, the standard deviation is estimated.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

STDEV.S/STDEV, DSTDEVP

DSTDEVP (database function)

Syntax:

DSTDEVP(DatabaseArea, Column, SearchCriteriaArea)

Description:

This calculates the *standard deviation of a population* based on the entire population using those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

STDEV.P/STDEVP, DSTDEV

DSUM (database function)

Syntax:

DSUM(DatabaseArea, Column, SearchCriteriaArea)

Description:

This returns the *sum* of those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

SUM, DAVERAGE

DVAR (database function)

Syntax:

DVAR(DatabaseArea, Column, SearchCriteriaArea)

Description:

This estimates the *variance* of a population based on a sample, using those values in a column of a database area that match the conditions specified in the search criteria area.

The function assumes that the determined values represent a sample from the population. On this basis, the variance is estimated.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

VAR.S/VAR, DVARP

DVARP (database function)

Syntax:

DVARP(DatabaseArea, Column, SearchCriteriaArea)

Description:

This calculates the *variance of a population* based on the entire population, using those values in a column of a database area that match the conditions specified in the search criteria area.

For more information on the use of database functions, see Working with database functions.

DatabaseArea is the cell range that contains the database. The first row of **DatabaseArea** must contain labels for the database columns (for example, Amount, Price, etc.).

Column is the desired column of the database area from which data is to be returned. You can specify **Column** by its relative column number or by its column label within double quotation marks (e.g., "Amount", "Price", etc.).

SearchCriteriaArea is a cell range containing one or more conditions that have to be fulfilled. The first row of SearchCriteriaArea has to contain column labels (e.g., Amount, Price, etc.).

See also:

VAR.P/VARP, DVAR

EDATE (date before/after n months)

Syntax:

EDATE(StartDate, m)

Description:

This returns the date that is **m** months before/after **StartDate**. This enables you to determine appointments that fall on the same day of another month.

m is the number of months.

If **m** is positive, the program calculates a date that is **m** months **after** the StartDate.

If **m** is negative, the program calculates a date that is **m** months **before** the StartDate.

m should be an integer. If this is not the case, PlanMaker automatically removes the decimal places.

Example:

EDATE("03/30/2018", 1) returns 04/30/2018

EDATE("03/31/2018", 1) *also* returns 04/30/2018 because April only has 30 days.

See also:

EOMONTH

EFFECT (effective interest rate)

Syntax:

EFFECT(NomRate, NPerYear)

Description:

This returns the effective annual interest rate, given the nominal annual interest rate and the number of compounding periods per year.

NomRate is the nominal annual interest rate.

NPerYear is the number of compounding periods per year (must be at least 1).

Note:

This function does not calculate the effective annual interest rate according to the regulations of some countries (for example, PAngV in Germany). It does not take into account any fees, commission, etc., but merely adds compound interest to the nominal interest rate over the year.

Example:

To calculate the effective annual interest rate of an investment with a nominal annual interest rate of 5%, compounded quarterly: How high is the annual percentage rate of charge?

EFFECT(5%, 4) returns approx. 5.09%

See also:

NOMINAL

EOMONTH (end of month in n months)

Syntax:

EOMONTH(StartDate, m)

Description:

This returns the last day of the month that is **m** months before/after **StartDate**.

The argument **StartDate** must be a valid date value, as the function will return the #VALUE! error value otherwise.

m is the number of months.

If **m** is zero, the last day of the same month will be returned.

If **m** is positive, the program calculates a date that is **m** months **after** the StartDate.

If **m** is negative, the program calculates a date that is **m** months **before** the StartDate.

m should be an integer. If this is not the case, PlanMaker automatically removes the decimal places.

Example:

EOMONTH("09/25/2018", 0) returns 09/30/2018

EOMONTH("09/25/2018", 1) returns 10/31/2018

EOMONTH("09/25/2018", -1) returns 08/31/2018

If cell A1 contains the date value 9/25/2018, then EOMONTH(A1, 0) returns 9/30/2018

See also:

EDATE

ERF (Gaussian error function)

Syntax:

ERF(LowerLimit [, UpperLimit])

Description:

This returns the Gaussian error function.

LowerLimit and **UpperLimit** (optional) determine the limits of the interval whose integral surface is to be calculated. If **UpperLimit** is omitted, the integral surface between zero and **LowerLimit** will be calculated.

See also:

ERFC

ERFC (complement to Gaussian error function)

Syntax:

ERFC(LowerLimit)

Description:

This returns the complement to the Gaussian error function.

For **LowerLimit**, the lower limit of the integral is to be specified. Only values ≥ 0 are allowed.

See also:

ERF

ERROR.TYPE (error type)

Syntax:

ERROR.TYPE(Cell)

Description:

This function can be used to query the type of error value in a cell.

As an argument, the function expects the address of the cell to be checked.

The function returns the following results:

Error value in cell	Result
No error value	#N/A
#NULL!	1
#DIV/0!	2
#VALUE!	3
#REF!	4
#NAME?	5
#NUM!	6
#N/A	7

For more information on error values, see Error values.

Example:

Cell A1 contains 1/0 and cell A2 contains 1/1.

Then the following applies:

ERROR.TYPE(A1) returns 2 (thus #DIV/0!)

ERROR.TYPE(A2) returns #N/A (thus no error)

The following formula returns the text "Error!" if any error occurs in cell A1, otherwise the value in A1 is returned.

IF(ISNA(ERROR.TYPE(A1)), A1, "Error!")

See also:

ISERR, ISERROR, ISNA, NA(), IFERROR, IFNA, TYPE, section Error values

EUROCONVERT (convert EU currencies)

Syntax:

EUROCONVERT(Number, From, To)

Description:

This function can convert the old currencies of all EU member states that have joined the monetary union, i.e., have introduced the single currency euro.

You can use this function to perform the following types of conversions:

a) Convert an amount from one of these currencies **into euros**, e.g., German Deutsche Mark (DM) into euros.

b) Convert an amount **from euros** into one of these currencies, e.g., euros into DM.

c) Convert an amount from one of these currencies **into another** by first converting it into euros and then into the other currency (a method called "triangulation"), e.g., Spanish pesetas -> euros -> DM. **Note:** This method can lead to rounding errors of up to 0.01.

Number is the amount to be converted.

From and **To** are codes representing the source and target currency (see table below). Use either PM codes or ISO codes.

Note: Microsoft Excel only supports ISO codes for this function. If you want your calculations to be Excel-compatible, ensure that you only use ISO codes (right column) for the currency codes.

The codes are not case-sensitive.

Country	PM code	ISO code
All euro countries (euro)	€	EUR
Belgium (franc)	BF	BEF
Germany (mark)	DM	DEM
Estonia (kroon)	kr	EEK
Finland (markka)	mk	FIM
France (franc)	F	FRF
Greece (drachma)	Dr	GRD
Ireland (pound)	IR£	IEP
Italy (lira)	L	ITL
Latvia (lats)	Ls	LVL
Lithuania (litas)	Lt	LTL
Luxembourg (franc)	LF	LUF
Malta (lira)	Lm	MTL
Netherlands (guilder)	fl	NLG
Austria (schilling)	öS	ATS
Portugal (escudo)	esc	PTE
Slovakia (koruna)	Sk	SKK

Slovenia (tolar)	SIT	SIT
Spain (peseta)	pts	ESP

Example:

EUROCONVERT(100, "DM", "€") converts 100 DM (German marks) into euros. The result is €51.13.

EUROCONVERT(100, "DEM", "EUR") does the same, but uses ISO codes. The result is of course identical: 51.13 (euros).

EUROCONVERT(100, "DEM", "ATS") returns 703.55 (Austrian schillings).

EVEN (round up to next even number)

Syntax:

EVEN(Number)

Description:

This rounds **Number** up to the nearest even integer.

Example:

EVEN(42) returns 42

EVEN(43) returns 44

EVEN(-43) returns -44

See also:

ODD, ROUNDUP, ROUND, MROUND, FIXED, INT, TRUNC, CEILING, FLOOR

EXACT (compare texts)

Syntax:

EXACT(Text1, Text2)

Description:

This returns TRUE if **Text1** and **Text2** are exactly the same.

This function is case-sensitive.

Example:

EXACT("aaa", "aaa") returns TRUE

EXACT("aaa", "Aaa") returns FALSE

EXACT(UPPER("aaa"), UPPER("Aaa")) returns TRUE

See also:

FIND, SEARCH, DELTA

EXP (power of e)

Syntax:

EXP(Number)

Description:

This returns *e* raised to the power of **Number**.

The constant *e* (2.71828...) is the base of the natural logarithm (LN).

Example:

EXP(5) returns 148.4132

EXP(LN(0.1234)) returns 0.1234

Additional info:

The inverse function is LN (natural logarithm).

See also:

LN, LOG, LOG10, POWER

EXPONDIST (exponential distribution)

Note: EXPONDIST is supplemented by the new identical function EXPON.DIST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

EXPONDIST(x, Lambda, Cumulative)

Description:

This returns the probabilities of an exponentially distributed random number.

An exponential distribution can be used to model time intervals like the time between phone calls or the time customers have to wait at a counter.

If, for example, it is known that a customer arrives at a bank counter every three minutes on average, you can use this function to calculate the probability that a customer will arrive after two minutes at the latest.

x is the value whose probability is to be calculated. **x** must be ≥ 0.

Lambda (λ) is the average number of successes per time interval. **Lambda** must be > 0.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

See also:

EXPON.DIST, POISSON.DIST/POISSON, WEIBULL.DIST/WEIBULL

EXPON.DIST (exponential distribution)

Syntax:

EXPON.DIST(x, Lambda, Cumulative)

Description:

This returns the probabilities of an exponentially distributed random number.

An exponential distribution can be used to model time intervals like the time between phone calls or the time customers have to wait at a counter.

If, for example, it is known that a customer arrives at a bank counter every three minutes on average, you can use this function to calculate the probability that a customer will arrive after two minutes at the latest.

x is the value whose probability is to be calculated. **x** must be ≥ 0.

Lambda (λ) is the average number of successes per time interval. **Lambda** must be > 0.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

EXPONDIST, POISSON.DIST/POISSON, WEIBULL.DIST/WEIBULL

FACT (factorial)

Syntax:

FACT(Number)

Description:

This returns the factorial of the specified **Number**.

The factorial of a number is the product of all positive integers in the range from 1 to the specified number.

The given number must be ≥ 0. Decimal places are automatically truncated.

Example:

FACT(9) equals 1*2*3*4*5*6*7*8*9, which is 362880.

See also:

FACTDOUBLE, MULTINOMIAL, POLYNOMIAL

FACTDOUBLE (double factorial)

Syntax:

FACTDOUBLE(Number)

Description:

This returns the double factorial of a **Number**.

The double factorial is calculated as follows:

If **Number** is even, its double factorial is 2*4*6*...**Number*.

If **Number** is odd, its double factorial is 1*3*5*...**Number*.

The given number must be ≥ 0. Decimal places are automatically truncated.

Example:

FACTDOUBLE(10) equals 2*4*6*8*10, which is 3840

FACTDOUBLE(11) equals 1*3*5*7*9*11, which is 10395

See also:

FACT

FALSE (logical value FALSE)

Syntax:

FALSE()

Description:

This returns the logical value FALSE.

See also:

TRUE

FDIST (F distribution)

Note: FDIST is supplemented by the new identical function F.DIST.RT, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

FDIST(X, DegreesFreedom1, DegreesFreedom2)

Description:

This returns the **right-tailed** F probability distribution function.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom1 is the numerator degrees of freedom.

DegreesFreedom2 is the denominator degrees of freedom.

Both degrees of freedom must be ≥ 1 and < 10^10. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

See also:

F.DIST.RT, F.INV.RT/FINV, F.TEST/FTEST

F.DIST.RT (F distribution)

Syntax:

F.DIST.RT(X, DegreesFreedom1, DegreesFreedom2)

Description:

This returns the **right-tailed** F probability distribution function.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom1 is the numerator degrees of freedom.

DegreesFreedom2 is the denominator degrees of freedom.

Both degrees of freedom must be ≥ 1 and < 10^10. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

FDIST, F.INV.RT/FINV, F.TEST/FTEST

FILENAME (file name of the document)

Syntax:

FILENAME([IncludePath])

Description:

This returns the file name of the current document.

IncludePath (optional) determines if the path (drive and folder) should be included:

FALSE or omitted: don't include path

TRUE: include path

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

If you save a document as `Balance.pmdx` in folder `C:\Documents`:

FILENAME() returns Balance.pmdx

FILENAME(TRUE) returns C:\Documents\Balance.pmdx

See also:

USERFIELD, SHEET, SHEETNAME, CELL

FIND (search for text in a text string)

Syntax:

FIND(SearchText, TextString [, StartPos])

Description:

This returns the position of **SearchText** within the **TextString**.

StartPos (optional) lets you specify the position in the text string, from which the search should start. If omitted, the search starts at the first character.

The FIND function is case-sensitive. To perform a non-case-sensitive search, use the SEARCH function instead.

Example:

FIND("a", "Banana") returns 2

FIND("A", "Banana") returns the #VALUE! error value, since the text string does not contain an uppercase "A".

FIND("a", "Banana", 3) returns 4

FIND("x", "Banana") returns the #VALUE! error value, since "x" does not occur in "Banana".

For example, if the word "banana" is in cell B5, you can enter the formula as follows:
FIND("a", B5) returns 2

See also:

SEARCH, REPLACE, SUBSTITUTE, EXACT, MID

FINV (percentiles of the F distribution)

Note: FINV is supplemented by the new identical function F.INV.RT, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

FINV(Probability, DegreesFreedom1, DegreesFreedom2)

Description:

This returns the inverse of the **right-tailed** F probability distribution function (i.e., percentiles of this distribution).

Probability is the given probability from the F distribution. The values must be within the range 0 to 1.

DegreesFreedom1 is the numerator degrees of freedom.

DegreesFreedom2 is the denominator degrees of freedom.

Both degrees of freedom must be ≥ 1 and < 10^10. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

F.INV.RT, F.TEST/FTEST, F.DIST.RT/FDIST

F.INV.RT (percentiles of the F distribution)

Syntax:

F.INV.RT(Probability, DegreesFreedom1, DegreesFreedom2)

Description:

This returns the inverse of the **right-tailed** F probability distribution function (i.e., percentiles of this distribution).

Probability is the given probability from the F distribution. The values must be within the range 0 to 1.

DegreesFreedom1 is the numerator degrees of freedom.

DegreesFreedom2 is the denominator degrees of freedom.

Both degrees of freedom must be ≥ 1 and < 10^10. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

FINV, F.TEST/FTEST, F.DIST.RT/FDIST

FISHER (Fisher transformation)

Syntax:

FISHER(Number)

Description:

This returns the Fisher transformation for the given number.

Number must be within the range from -1 to 1 (excluding -1 and 1).

Additional info:

The inverse function is FISHERINV

See also:

FISHERINV, CORREL, COVARIANCE.P/COVARIANCE.S/COVAR

FISHERINV (inverse of the Fisher transformation)

Syntax:

FISHERINV(Number)

Description:

This returns the inverse of the Fisher transformation for the given number.

Additional info:

This function is the inverse of the FISHER transformation.

See also:

FISHER, CORREL, COVARIANCE.P/COVARIANCE.S/COVAR

FIXED (format number as text with fixed decimals)

Syntax:

FIXED(Number [, n] [, NoThousandsSep])

Description:

This converts a number into a text string. The resulting text string contains the number rounded to **n** decimal places with thousands separators.

Number is the number to be converted.

n (optional) is the desired number of decimal places. If **n** is omitted, **Number** is rounded to 2 decimal places.

If **n** is a negative value, **Number** is rounded to the left of the decimal point. For example, if **n** is -2, **Number** is rounded to the nearest multiple of a hundred (see last example).

The optional logical value **NoThousandsSep** lets you specify if the resulting text string should contain thousands separators:

FALSE or omitted: Add thousands separators

TRUE: Do not add thousands separators

Note:

Note that the result of this function is a *text string*. To round a number without converting it to a text string, use one of the rounding functions (see below).

To only display a number as rounded without actually rounding it, you can also use the commands of the ribbon tab **Home** | group **Number**.

Example:

FIXED(1234.5678, 2) returns the text string 1,234.57

FIXED(1234.5678, 2, TRUE) returns the text string 1234.57 (without thousands separators)

FIXED(1234.5678, -2) returns the text string 1,200

See also:

ROUND, ROUNDDOWN, ROUNDUP, TRUNC, TEXT, INT

FLOOR (round down to a multiple of base)

Syntax:

FLOOR(Number, Base)

Description:

This rounds **Number** down (towards zero) to the nearest multiple of **Base**.

Positive numbers are rounded to the nearest multiple of **Base** that is smaller than **Number**.

Negative numbers are rounded to the nearest multiple of **Base** that is larger than **Number**.

Number and **Base** must have the same sign, otherwise the function returns a #NUM! error value.

Example:

FLOOR(42.5, 1) returns 42

FLOOR(-42.5, -1) returns -42

FLOOR(21, 5) returns 20

FLOOR(12.3456, 0.01) returns 12.34

See also:

CEILING, ROUNDDOWN, ROUNDUP, ROUND, MROUND, TRUNC

FORECAST (forecast using linear regression)

Syntax:

FORECAST(x, y_values, x_values)

Description:

This returns the y coordinate for the given **x** coordinate on a best-fit line based on the given values.

A *best-fit line* is the result of a *linear regression*, a statistical technique that adapts a line (called *"trendline"* or *"best-fit line"*) to a set of data points (for example, the results of a series of measurements).

The FORECAST function allows you to predict what value y (the dependent variable) will have approximately for a certain value x (the independent variable).

This function can be used to predict, for example, the resistance of a temperature-dependent resistor at a specific temperature after having measured the resistance at several other temperatures.

x is the value x for which a prediction is to be made.

For the arguments **y_values** and **x_values**, you usually specify a cell range.

y_values are the dependent variables (the resistance in the above example).

x_values are the independent variables (the temperature in the above example).

Note:

Note that this function first expects the **y_values** and then the **x_values** as second and third arguments – not the other way around.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

The resistance of a temperature-dependent resistor has been measured at several temperatures.

Cells A1:A4 contain the temperatures that were measured (the independent variables): 8, 20, 25, 28

Cells B1:B4 contain the resistances that were measured (the dependent variables): 261, 508, 608, 680

The resistance which will be delivered at 15°C can be estimated using the following formula:

FORECAST(15, B1:B4, A1:A4) returns 405.21805

At 15°, a resistance of 405.21805 (Ohm) would thus be expected.

Additional info:

INTERCEPT(y_values, x_values) equals FORECAST(0, y_values, x_values).

See also:

INTERCEPT, SLOPE, SKEW, STEYX, TREND

FORMULATEXT (display formula contained in cell)

Syntax:

FORMULATEXT(Cell)

Description:

This displays the formula that is in the specified cell.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

If cell A1 contains the formula =SUM(D2:D4), then FORMULATEXT(A1) will return the text "=SUM(D2:D4)".

See also:

ISFORMULA

FREQUENCY (frequency)

Syntax:

FREQUENCY(Values, Bins)

Description:

This returns a frequency distribution for the given set of values. You can use this function to determine how many values from a data set are within certain intervals.

The result of this function is a vertical array (see notes below).

Values is a cell range or an array containing the values to be evaluated. Blank cells and text strings are ignored.

Bins constitute a cell range or an array containing the intervals to be evaluated. If, for example, **Bins** is the array {10; 15; 42}, the function returns an array with the following values:

1. The number of values that are ≤ 10.

2. The number of values that are > 10 and ≤ 15.

3. The number of values that are > 15 and ≤ 42.

4. The number of values that are > 42.

Both **Values** and **Bins** have to be vertical arrays, as they must not have more than one column.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Please note that the array returned by this function contains *one element more* than the number of values in **Bins**.

See also:

COUNT

FTEST (F-test)

Note: FTEST is supplemented by the new identical function F.TEST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

FTEST(Array1, Array2)

Description:

This returns the test statistics of an F-test.

An F-test can be used to check the consistency of the variances of two samples.

Array1 and **Array2** are the two arrays (samples) to be evaluated.

They both have to consist of at least 2 values; otherwise the function returns a #DIV/0! error value (division by zero).

If the variance of **Array1** or **Array2** is 0, the function returns a #DIV/0! error value.

See also:

F.TEST, F.INV.RT/FINV, F.DIST.RT/FDIST

F.TEST (F-test)

Syntax:

F.TEST(Array1, Array2)

Description:

This returns the test statistics of an F-test.

An F-test can be used to check the consistency of the variances of two samples.

Array1 and **Array2** are the two arrays (samples) to be evaluated.

They both have to consist of at least 2 values; otherwise the function returns a #DIV/0! error value (division by zero).

If the variance of **Array1** or **Array2** is 0, the function returns a #DIV/0! error value.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

FTEST, F.INV.RT/FINV, F.DIST.RT/FDIST

FV (future value)

Syntax:

FV(Rate, NPer, Pmt [, PV] [, Type])

Description:

This returns the future value of an investment based on periodic constant payments and a fixed interest rate.

You can use this function to calculate the future value (terminal value) of a loan, for example.

The function expects the following arguments:

Rate is the interest rate (per payment period)

NPer is the total number of payment periods.

Pmt is the annuity (payment made for each period).

PV (optional) is the present value. If the argument **PV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

FV(6.5%/12, 10, -1000) returns 10,247.30

See also:

PV, PPMT, PMT, RATE, IPMT, FVSCHEDULE, NPER

FVSCHEDULE (future value)

Syntax:

FVSCHEDULE(PV, Rates)

Description:

This returns the future value of an initial principal after applying a series of *variable* compound interest rates.

This function is applicable to investments with just one payment (made at the beginning) and a fixed interest rate.

You can use this function to calculate the future value (terminal value) of a loan, for example.

The function expects the following arguments:

PV is the present value (initial amount)

Rates is a cell range or an array that contains the different interest rates per period (in their actual order, namely, rate for the first year, rate for the second year, etc.).

Example:

You have an investment of €10,000 that pays 5% interest in the first year, 5.2% in the second year, 5.5% in the third year and 5.9% in the fourth year. To calculate the future value of this investment, you enter the four interest rates (in the correct order) in cells A1 to A4, for example, and then apply the following formula:

FVSCHEDULE(10000, A1:A4) returns €12341.09

The different interest rates can also be specified directly using the following form:

FVSCHEDULE(10000, {5%, 5.2%, 5.5%, 5.9%})

See also:

FV

GAMMA (gamma function)

Syntax:

GAMMA(Number)

Description:

This calculates the gamma function for the given number.

Number must be > 0.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

GAMMALN, GAMMA.DIST/GAMMADIST, GAMMA.INV/GAMMAINV

GAMMADIST (gamma distribution)

Note: GAMMADIST is supplemented by the new identical function GAMMA.DIST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

GAMMADIST(X, Alpha, Beta, Cumulative)

Description:

This returns the probabilities of a gamma distributed random number.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

See also:

GAMMA.DIST, GAMMA.INV/GAMMAINV, GAMMA, GAMMALN

GAMMA.DIST (gamma distribution)

Syntax:

GAMMA.DIST(X, Alpha, Beta, Cumulative)

Description:

This returns the probabilities of a gamma distributed random number.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

The inverse function is GAMMA.INV

See also:

GAMMADIST, GAMMA.INV/GAMMAINV, GAMMA, GAMMALN

GAMMAINV (percentiles of the gamma distribution)

Note: GAMMAINV is supplemented by the new identical function GAMMA.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

GAMMAINV(Probability, Alpha, Beta)

Description:

This returns the percentiles of the gamma distribution.

Probability is the given probability from the gamma distribution. The values must be within the range 0 to 1.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

GAMMA.INV, GAMMA.DIST/GAMMADIST, GAMMA, GAMMALN

GAMMA.INV (percentiles of the gamma distribution)

Syntax:

GAMMA.INV(Probability, Alpha, Beta)

Description:

This returns the percentiles of the gamma distribution.

Probability is the given probability from the gamma distribution. The values must be within the range 0 to 1.

Alpha and **Beta** are shape parameters of the function. They both have to be greater than zero.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Additional info:

GAMMA.INV is the inverse function of GAMMA.DIST

See also:

GAMMAINV, GAMMA.DIST/GAMMADIST, GAMMA, GAMMALN

GAMMALN (logarithm of the gamma function)

Syntax:

GAMMALN(Number)

Description:

This returns the natural logarithm of the gamma function.

Number must be > 0.

See also:

GAMMA, GAMMA.DIST/GAMMADIST, GAMMA.INV/GAMMAINV

GAUSS (standard normal distribution)

Syntax:

GAUSS(x)

Description:

This returns the integral of the PHI function (density function of a standard normal distribution) minus 0.5.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

We thus recommend that you use the NORMSDIST function instead, since it is compatible with all versions of Excel:

GAUSS(x) equals NORMSDIST(x) - 0.5

See also:

PHI, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

GCD (greatest common divisor)

Syntax:

GCD(Number1 [, Number2, Number3 ...])

Description:

This returns the GCD (greatest common divisor) of the specified integers. The GCD is the largest integer that will divide all of the given integers without a remainder.

You can specify numbers or cell references as arguments.

All given values **Number1**, **Number2**, etc., have to be greater than zero. Furthermore, only integers should be used. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

GCD(25, 100, 250) returns 25

GCD(25, 105, 250) returns 5

If A1:A3 contains the values 4, 8 and 6, the following applies:

GCD(A1:A3) returns 2

See also:

LCM

GEOMEAN (geometric mean)

Syntax:

GEOMEAN(Number1 [, Number2, Number3...])

Description:

Returns the geometric mean of the specified numbers.

The geometric mean is the **n**-th root of the product of the given values (**n** is the total number of values).

All given values **Number1**, **Number2**, etc., have to be greater than zero. The values can be single values, cell ranges or an array.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

GEOMEAN(1, 2, 6) returns 2.28943

See also:

HARMEAN, TRIMMEAN, AVERAGE

GESTEP (number greater or equal to threshold value?)

Syntax:

GESTEP(Number, Step)

Description:

This returns 1 if **Number** ≥ **Step**, otherwise it returns 0.

Example:

GESTEP(41, 42) returns 0

GESTEP(42, 42) returns 1

GESTEP(42.1, 42) returns 1

See also:

DELTA

GETPIVOTDATA

Syntax:

GETPIVOTDATA(ValueField, PivotRange [, Field1, Label1, Field2, Label2,...])

Description:

This returns the values of an existing pivot table by retrieving certain fields from the pivot table structure.

Value field: This is the name of the field that you have placed in the *Values* area of the pivot table and whose data you want to retrieve. The argument must be specified within double quotation marks.

PivotRange: The referenced pivot table. Just address any cell of the pivot table target range to specify it. The argument is specified without quotation marks.

Field1, Label1, Field2, Label2,...: You can narrow the output of the values by retrieving a specific field in conjunction with an item from this field. You can add several combinations. The argument must be specified within double quotation marks.

ValueField and **PivotRange** are required as arguments, while **Fields/Labels** are optional.

Additional info:

For the **PivotRange**, you can also enter a *Named range*, but you first have to assign a name to the cell range of the pivot table report. To do so, select a cell in the pivot table report (you do not have to select the entire range) and follow the description in the Naming cell ranges section.

You cannot use the name automatically generated by PlanMaker (e.g., "Pivot table 1") as the *Named range*. Only names that have been created via the ribbon command **Formula |** group **Named areas | Edit names** can be actually assigned as names for the **PivotRange**.

Example:

Here is some sample data from the Pivot tables section. Assuming your pivot table report is in cell range F1:G12 and the "Amount" field is in the *Values* area, the "Employee" and "Order" fields are in the *Row labels* area:

	A	B	C	D	E	F	G	H
1	Date	Employee	Order	Amount		Row labels	Sum of Amount	
2	01/10/2018	Anna	Coffee	$35		Anna	123	
3	02/15/2018	Toni	Cream	$15		Coffee	75	
4	04/13/2018	Anna	Snacks	$20		Cream	12	
5	05/15/2018	Toni	Tea	$15		Snacks	36	
6	06/18/2018	Maria	Coffee	$30		Maria	30	
7	06/20/2018	Anna	Cream	$12		Coffee	30	
8	08/11/2018	Anna	Snacks	$16		Toni	47	
9	08/18/2018	Toni	Sweetener	$7		Cream	25	
10	10/28/2018	Anna	Coffee	$40		Sweetener	7	
11	11/15/2018	Toni	Cream	$10		Tea	15	
12						Grand total	200	

GETPIVOTDATA("AMOUNT", F1) returns 200, since the grand total for all values of *Amount* is displayed.

GETPIVOTDATA("AMOUNT", F1, "Employee", "Anna") returns 123, since the result of *Amount* is restricted to the *Employee* field containing the item *Anna*.

GETPIVOTDATA("AMOUNT", F1, "Employee", "Anna", "Order", "Coffee") returns 75, since the result of *Amount* is restricted to the field *Employee*/item *Anna* in combination with the field *Order*/item *Coffee*.

GROWTH (values of an exponential regression)

Syntax:

GROWTH(y_values [, x_values] [, New_x_values] [, Constant])

Description:

This returns the values of an exponential regression.

Exponential regression is a statistical technique that adapts an exponential curve to a set of data points (for example, the results of a series of measurements).

The GROWTH function returns an array with the y values of a best-fit exponential curve based on the given y values and x values.

To clarify, the function expects the following arguments:

y_values: Here, you specify the existing y coordinates, that is, the dependent values. **y_values** can be either a cell reference or an array. All values must be positive, otherwise a #NUM! error value is returned.

x_values: Here, you have the option of specifying the x coordinates, that is, the independent values. **x_values** also can be either a cell reference or an array. If **x_values** are omitted, the function automatically uses the values 1, 2, 3, ... (according to the number of existing **y_values**).

New_x_values (optional) are the x values for which you want to determine the corresponding y values on the curve. If omitted, the values in **x_values** will be used. If required, however, you can specify other x coordinates with **New_x_values**. **New_x_values** can also be either a cell reference or an array.

Constant (optional) is a logical value that lets you specify if the constant b should be set to 1 (one):

TRUE or omitted: b is automatically calculated from the available data.

FALSE: b is forced to equal 1 (one), the m values are adjusted accordingly.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

See also:

LINEST, LOGEST, TREND

HARMEAN (harmonic mean)

Syntax:

HARMEAN(Number1 [, Number2, Number3 ...])

Description:

This returns the harmonic mean of the given numbers.

This is defined as the reciprocal value of the arithmetic mean of the reciprocal values of the individual values.

All given values **Number1**, **Number2**, etc., have to be greater than zero. The values can be single values, cell ranges or an array.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

HARMEAN(1, 2, 6) returns 1.8

See also:

GEOMEAN, TRIMMEAN, AVERAGE

HEX2BIN (hexadecimal number to binary number)

Syntax:

HEX2BIN(Number [, Digits])

Description:

This converts the given hexadecimal number (base 16) to a binary number (base 2).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in hexadecimal notation):

80000000 ≤ *Number* ≤ 7FFFFFFF

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

HEX2BIN("2A") returns 101010

HEX2BIN("2A", 8) returns 00101010

HEX2BIN("2A", 2) returns the #NUM! error value because this number cannot be represented with fewer than 5 binary digits

HEX2BIN("FFFFFFD6") returns 11111111111111111111111111010110 (negative value)

See also:

BIN2HEX, HEX2DEC, HEX2OCT

HEX2DEC (hexadecimal number to decimal number)

Syntax:

HEX2DEC(Number)

Description:

This converts the given hexadecimal number (base 16) to a decimal number (base 10).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in hexadecimal notation):

80000000 ≤ *Number* ≤ 7FFFFFFF

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

Example:

HEX2DEC("2A") returns 42

HEX2DEC("FFFFFFD6") returns -42

See also:

DEC2HEX, HEX2BIN, HEX2OCT, DECIMAL

HEX2OCT (hexadecimal number to octal number)

Syntax:

HEX2OCT(Number [, Digits])

Description:

This converts the given hexadecimal number (base 16) to an octal number (base 8).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in hexadecimal notation):

80000000 ≤ *Number* ≤ 7FFFFFFF

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

HEX2OCT("2A") returns 52

HEX2OCT("2A", 4) returns 0052

HEX2OCT("2A", 1) returns the #NUM! error value because this number cannot be represented with fewer than 2 octal digits

HEX2OCT("FFFFFFD6") returns 37777777726 (negative value)

See also:

OCT2HEX, HEX2BIN, HEX2DEC

HLOOKUP (look up row-wise)

Syntax:

HLOOKUP(Crit, Range, n [, Sorted])

Description:

This function searches in the first row of **Range** for the first occurrence of **Crit**. If found, the function returns the content of the cell located in the same column and **n**-th row of **Range**.

Note: HLOOKUP can only search horizontally. To search vertically, use VLOOKUP instead.

Crit is the value for which the search is performed. The search is case-insensitive.

Range is the cell range or array to be evaluated. Its first row should contain the values to be scanned. These can be text strings, numbers or logical values.

n is the relative row number in **Range** from which a value will be returned if **Crit** is found.

If **n** is less than 1, a #VALUE! error value is returned. If **n** is greater than the total number of rows in **Range**, a #REF! error value is returned.

Sorted is a logical value that you use to specify whether the values in the first row of **Range** are sorted:

If **Sorted** is TRUE or omitted, values in the first column of **Range** have to be sorted in ascending order.

If **Sorted** is FALSE, the order of the values in the first column is irrelevant.

Important: If you do not explicitly set **Sorted** to FALSE, these values must be in sorted order!

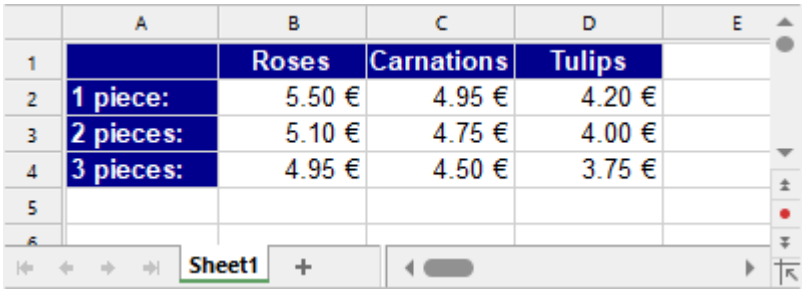
Note:

If **Sorted** is set to TRUE and VLOOKUP does not find a value that exactly matches **Crit**, the *next smaller* value is automatically taken.

If no smaller value exists because **Crit** is smaller than the smallest value available, the function returns the #N/A error value.

Example:

You sell flowers at different unit prices – depending on the amount of flowers the customer buys. To do so, you have created a table with a discount scale:



	A	B	C	D	E
1		Roses	Carnations	Tulips	
2	1 piece:	5.50 €	4.95 €	4.20 €	
3	2 pieces:	5.10 €	4.75 €	4.00 €	
4	3 pieces:	4.95 €	4.50 €	3.75 €	
5					

Sheet1

You can now use the HLOOKUP function to determine the price for a specific type of flower depending on the number of pieces.

Use the following arguments:

For **Crit**, enter the name of the flower type (i.e., "roses", "carnations" or "tulips").

For **Range**, you enter the cell range that contains the names of the flower types in the first row and the prices below – B1:D4 here.

For **n**, you enter the number of pieces *plus one*, as the first row (**n**=1) contains the names of the flowers rather than the prices.

Examples:

HLOOKUP("Roses", B1:D4, 2, FALSE) returns the price for one rose, i.e., €5.50.

HLOOKUP("Roses", B1:D4, 4, FALSE) returns the price for three roses, i.e., €4.95.

HLOOKUP("Carnations", B1:D4, 4, FALSE) returns the price for three carnations, i.e., €4,50.

HLOOKUP("Carnations", B1:D4, 5, FALSE) returns the #REF error value because **n** is greater than the row number of **Range**.

HLOOKUP("Petunias", B1:D4, 4, FALSE) returns the #N/A error value because "Petunias" does not appear in the first row of **Range**.

See also:

VLOOKUP, LOOKUP, INDEX, MATCH, TRUE

HOLIDAY (dates of movable Christian holidays)

Syntax:

HOLIDAY(Year, Type)

Description:

This returns the date of the movable Christian holidays in the specified year.

Year is the desired year (as an integer).

Type is an integer that determines the holiday for which the date should be returned:

0: Good Friday

1: Easter Sunday

2: Easter Monday

3: Ascension Day

4: Whit Sunday (Pentecost)

5: Whit Monday

6: (Feast of) Corpus Christi

7: Ash Wednesday

8: Repentance Day (Germany)

9: 1st Sunday of Advent

Example:

HOLIDAY(2014, 1) returns 04/20/2014 (Easter Sunday in 2014)

HOLIDAY(YEAR(TODAY()), 5) returns the date of Whit Monday in the current year

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

WORKDAY, NETWORKDAYS

HOUR (hour)

Syntax:

HOUR(Time)

Description:

This returns the hour of a time value.

Example:

HOUR("12:34:56 AM") returns 12

HOUR("09/25/2018 10:30:56") returns 10

If cell A1 contains a date with the time 10:30:56, then HOUR(A1) returns 10

See also:

MINUTE, SECOND, MILLISECONDS

HYPERLINK (hyperlink)

Syntax:

HYPERLINK(TargetAddress [, Text])

Description:

This displays a hyperlink (a link) to the specified **target address** in the cell. The displayed text is either the target address or the text specified in the optional parameter **Text**.

For example, you can use this function to insert a hyperlink to a web page in a cell. If the user clicks on it, their web browser starts automatically and presents this page.

Furthermore, you can insert links to another PlanMaker document, which is automatically opened when you click on this link. See examples below.

TargetAddress is the address to which the link should point. You can use a fixed text string, a calculation, or a reference to the cell that contains the target address. See examples below.

Text (optional) is the text to be displayed in the cell. If it is omitted, the **target address** will be displayed instead.

Example:

=HYPERLINK("http://www.softmaker.com")

The above hyperlink opens the web page www.softmaker.com in the web browser.

=HYPERLINK("http://www.softmaker.com", "Our Homepage")

Ditto, but instead of the target address, the text "Our Homepage" is displayed as the link text.

=HYPERLINK("http://www.softmaker.com#MyAnchor")

The above link opens the web page www.softmaker.com in the browser and jumps to the anchor (target) named "MyAnchor". Alternatively, you could also use the syntax =HYPERLINK("[http://www.softmaker.com] MyAnchor") in this case.

=HYPERLINK("C:\My Folder\Filename.pmdx")

The above link opens the file C:\My Folder\Filename.pmdx in PlanMaker.

Note: Not only can you open PlanMaker files with this function, but also any kind of document (provided that it is associated with an application that will open it).

=HYPERLINK("[C:\My Folder\Filename.pmdx]Sheet1!A5")

The above link opens the file C:\My Folder\Filename.pmdx in PlanMaker and jumps to the cell A5 in the worksheet "Sheet1". Note: The entire file path and name must be enclosed within square brackets. Directly after the file name, add the name of the worksheet, an exclamation mark and the address of the target cell.

=HYPERLINK("[C:\My Folder\Filename.pmdx]Sheet1!MyRange")

The above link opens the file C:\My Folder\Filename.pmdx in PlanMaker and jumps to the top left corner of the named range "MyRange" on the worksheet "Sheet1".

=HYPERLINK(E1, E2)

The above link opens the target address entered in cell E1. The content of cell E2 is displayed as a link text.

See also:

Links can also be created using the alternative method described in Using hyperlinks. However, the HYPERLINK function is more flexible.

HYPGEOMDIST (hypergeometric distribution)

Note: HYPGEOMDIST is supplemented by the new function HYPGEOM.DIST with additional functionality, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

HYPGEOMDIST(k, n, K, N)

Description:

This returns the probabilities of a hypergeometrically distributed random variable.

The hypergeometric distribution can be used to calculate the probability of, for example, obtaining **k** red balls when randomly drawing **n** balls (without replacement) from an urn containing **N** balls of which **K** are red.

k is the number of successes in the sample (success = drawing a red ball).

n is the size of the sample (the number of balls to be drawn)

K is the total number of successes in the population (the total number of *red* balls).

N is the size of the population (the total number of balls).

All of these values should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

An urn contains a total of 10 balls (N=10) of which 2 are red (K=2). To determine the probability of obtaining *exactly* 1 red ball (k=1) when drawing 5 balls (n=5), use the following calculation:

HYPGEOMDIST(1, 5, 2, 10) returns 0.55556

See also:

HYPGEOM.DIST, NEGBINOM.DIST/NEGBINOMDIST

HYPGEOM.DIST (hypergeometric distribution)

Syntax:

HYPGEOM.DIST(k, n, K, N, Cumulative)

Description:

This returns the (cumulative) probabilities of a hypergeometrically distributed random variable.

The hypergeometric distribution can be used to calculate the probability of, for example, obtaining **k** red balls when randomly drawing **n** balls (without replacement) from an urn containing **N** balls of which **K** are red.

k is the number of successes in the sample (success = drawing a red ball).

n is the size of the sample (the number of balls to be drawn)

K is the total number of successes in the population (the total number of *red* balls).

N is the size of the population (the total number of balls).

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative distribution function of the hypergeometric distribution is calculated. By entering the value FALSE, the probability density function of the hypergeometric distribution is calculated.

All of these values should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

An urn contains a total of 10 balls (N=10) of which 2 are red (K=2). To determine the probability of obtaining *exactly* or *at most* 1 red ball (k=1) when drawing 5 balls (n=5), use the following calculation:

HYPGEOM.DIST(1, 5, 2, 10, FALSE) returns 0.55556

HYPGEOM.DIST(1, 5, 2, 10, TRUE) returns 0.77778

The probability of obtaining *exactly* 1 red ball is 55.6% (density function). The probability of obtaining *at most* 1 red ball is 77.8% (cumulative distribution function).

Note:

The HYPGEOM.DIST function supplements the previous HYPGEOMDIST function with the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

HYPGEOMDIST, NEGBINOM.DIST/NEGBINOMDIST

IF (if-then-else condition)

Syntax:

IF(Condition, IfTrue [, IfFalse])

Description:

This function returns **IfTrue** if the specified **Condition** is TRUE, otherwise it returns **IfFalse** (if given).

Condition can be any kind of value or expression that returns TRUE or FALSE.

IfTrue and **IfFalse** can be any kind of value or expression (including formulas).

Example:

The formula IF(A1<5, "Order!", "Stock ok") returns the text string "Order!" if A1<5. If this is not the case (A1 is thus ≥ 5), it returns the text string "Stock ok".

See also:

CHOICE, IFERROR

IFEMPTY (return a value if cell is empty)

Syntax:

IFEMPTY(Cell, Value)

Description:

This normally returns the content of the **Cell**. If this cell is empty, however, the defined **Value** is returned instead.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

IFEMPTY(A1, "Cell is empty!") returns the content of cell A1 as long as it is not empty. For an empty cell, however, the entered text string "Cell is empty!" is returned instead.

See also:

ISBLANK, IFERROR, IFNA

IFERROR (return a value if error)

Syntax:

IFERROR(Value, ValueIfError)

Description:

This normally returns **Value**. If **Value** contains any type of error value, however, the defined **ValueIfError** is returned instead.

Compatibility notes:

This function is not supported by the **.xls** file format (used in Microsoft Excel 2003 and earlier). If you save a document in this format, all calculations using this function will be replaced with their current result as a *fixed value*.

If you want to use this function, you should not save your document in the "Microsoft Excel 97-2003 (.xls)" file format, but choose one of the following formats instead:

- "PlanMaker document (.pmdx or .pmd)"
- or: "Microsoft Excel 2007-2021 (.xlsx)"

Example:

IFERROR(A1, "Error!") returns the value in cell A1, as long as A1 doesn't contain an error value. If there is an error value in this cell, however, the text string "Error!" is returned instead.

See also:

IFNA, ISERR, ISERROR, ISNA, ERROR.TYPE, section Error values

IFNA (return defined value for #N/A)

Syntax:

IFNA(Cell, Value)

Description:

This normally returns the content of the **Cell**. If this cell is #N/A, however, the defined **Value** is returned instead.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

IFNA(A1, "not found!") returns the value in cell A1 as long as it doesn't contain the #N/A error value. If it does contain the #N/A error value, however, the text string "not found!" is returned instead.

See also:

IFERROR, ISERR, ISERROR, ISNA, ERROR.TYPE, section Error values

IFS (first condition met)

Syntax:

IFS(Condition1, Value1 [, Condition2, Value2 ...])

Description:

This function can test multiple conditions and returns the value of the first condition that is fulfilled. IFS allows shorter formulas than with a nested formula construction using the similar IF function.

Condition1 is the first tested condition and can be any expression.

Value1 is the result if Condition1 is fulfilled.

Condition2, Value2 (optional) is the second tested condition, etc.

As soon as the first fulfilled Condition is found, its corresponding Value is displayed as the result.

Note:

If none of the conditions are fulfilled, the #N/A error is returned.

Instead, you can use the condition TRUE at the end of the formula to specify a default value as the result, which is returned if none of the conditions are fulfilled (see example 2 below).

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

Example 1:

You have entered a sales volume in cell A1 and want to distribute a bonus according to the sales volume using the following distribution key:
Sales above 50000 = 2000 bonus; sales above 25000 = 1000 bonus; sales above 10000 = 500 bonus

The formula IFS(A1>50000, 2000, A1>25000, 1000, A1>10000, 500) then returns, for example:

A sales volume of 70000 in cell A1 results in 2000 as a bonus.

A sales volume of 30000 in cell A1 results in 1000 as a bonus.

A sales volume of 20000 in cell A1 results in 500 as a bonus.

If cell A1 contains a sales volume of 9000, the formula returns the result #N/A, since none of the conditions are fulfilled. Of course, you could also set a condition for this scenario. Alternatively, you can specify instead the default value which the formula generally returns if none of the conditions are fulfilled (see Example 2).

Example 2:

You have the same initial situation as in example 1. At the end of the formula, use the condition TRUE to specify a default value that appears if none of the conditions are fulfilled:

IFS(A1>50000, 2000, A1>25000, 1000, A1>10000, 500, TRUE, "check")

If, for example, cell A1 contains a sales volume of 9000 or -1000, the result is "check".

See also:

IF, SWITCH, VLOOKUP

IMABS (absolute value of a complex number)

Syntax:

IMABS(ComplexNumber)

Description:

This returns the absolute value (modulus) of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMABS("2+4i") returns 4.47214

See also:

COMPLEX, IMARGUMENT, IMEXP, IMCONJUGATE

IMAGINARY (imaginary coefficient of a complex number)

Syntax:

IMAGINARY(ComplexNumber)

Description:

This returns the imaginary coefficient of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMAGINARY("2+4i") returns 4

See also:

COMPLEX, IMREAL

IMARGUMENT (angle of a complex number)

Syntax:

IMARGUMENT(ComplexNumber)

Description:

This returns the angle (in radians) from the real axis to the representation of a complex number in polar coordinates.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMARGUMENT("2+4i") returns 1.10715

See also:

COMPLEX, IMABS, IMEXP, IMCONJUGATE

IMCONJUGATE (conjugate a complex number)

Syntax:

IMCONJUGATE(ComplexNumber)

Description:

This returns the complex conjugate of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMCONJUGATE("2+4i") returns 2-4i

See also:

COMPLEX, IMABS, IMARGUMENT, IMEXP

IMCOS (cosine of a complex number)

Syntax:

IMCOS(ComplexNumber)

Description:

This returns the cosine of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMCOS("2+4i") returns -11.3642347064011 - 24.8146514856342i

See also:

COMPLEX, IMSIN

IMDIV (division of complex numbers)

Syntax:

IMDIV(ComplexNumber1, ComplexNumber2)

Description:

This returns the quotient of two complex numbers, i.e., divides **ComplexNumber1** by **ComplexNumber2**.

The arguments must be complex numbers in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMDIV("4+12i", "2+4i") returns 2.8+0.4i

See also:

COMPLEX, IMPRODUCT, IMSUB, IMSUM

IMEXP (exponential of a complex number)

Syntax:

IMEXP(ComplexNumber)

Description:

This returns the exponential of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMEXP("2+4i") returns -4.82980938326939 - 5.59205609364098i

See also:

COMPLEX, IMABS, IMARGUMENT, IMCONJUGATE, IMLN

IMLN (natural logarithm of a complex number)

Syntax:

IMLN(ComplexNumber)

Description:

This returns the natural logarithm (base *e*) of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMLN("2+4i") returns 1.497866136777 + 1.107148717794i

See also:

COMPLEX, IMEXP, IMLOG10, IMLOG2

IMLOG10 (base-10 logarithm of a complex number)

Syntax:

IMLOG10(ComplexNumber)

Description:

This returns the base-10 logarithm of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMLOG10("2+4i") returns 0.650514997831991 + 0.480828578784234i

See also:

COMPLEX, IMLN, IMLOG2

IMLOG2 (base-2 logarithm of a complex number)

Syntax:

IMLOG2(ComplexNumber)

Description:

This returns the base-2 logarithm of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMLOG2("2+4i") returns 2.16096404744368 + 1.59727796468811i

See also:

COMPLEX, IMLN, IMLOG10

IMNEG (negative value of a complex number)

Syntax:

IMNEG(ComplexNumber)

Description:

This returns the negative of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMNEG("2+4i") returns -2-4i

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Tip: As an alternative to IMNEG(A1), you can also use the following formula:

COMPLEX(-IMREAL(A1), -IMAGINARY(A1))

See also:

COMPLEX

IMPOWER (power of a complex number)

Syntax:

IMPOWER(ComplexNumber, Exponent)

Description:

This returns a supplied complex number raised to a given power.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMPOWER("2+4i", 2) returns -12+16i

See also:

COMPLEX, IMSQRT

IMPRODUCT (product of complex numbers)

Syntax:

IMPRODUCT(ComplexNum1 [, ComplexNum2, ComplexNum3 ...])

Description:

Returns the product of the given complex numbers.

The arguments must be complex numbers in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMPRODUCT("2+4i", "1+2i") returns -6+8i

See also:

COMPLEX, IMDIV, IMSUB, IMSUM

IMREAL (real coefficient of a complex number)

Syntax:

IMREAL(ComplexNumber)

Description:

This returns the real coefficient of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMREAL("2+4i") returns 2

See also:

COMPLEX, IMAGINARY

IMSIN (sine of a complex number)

Syntax:

IMSIN(ComplexNumber)

Description:

This returns the sine of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMSIN("2+4i") returns 24.8313058489464 - 11.3566127112182i

See also:

COMPLEX, IMCOS

IMSQRT (square root of a complex number)

Syntax:

IMSQRT(ComplexNumber)

Description:

This returns the square root of a complex number.

ComplexNumber must be a complex number in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMSQRT("2+4i") returns 1.79890743994787 + 1.111785940502 84i

See also:

COMPLEX, IMPOWER

IMSUB (difference of complex numbers)

Syntax:

IMSUB(ComplexNumber1, ComplexNumber2)

Description:

This returns the difference of two complex numbers, i.e., subtracts **ComplexNumber2** from **ComplexNumber1**.

The arguments must be complex numbers in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMSUB("2+4i", "1+2i") returns 1+2i

See also:

COMPLEX, IMDIV, IMPRODUKT, IMSUM

IMSUM (sum of complex numbers)

Syntax:

IMSUM(ComplexNum1 [, ComplexNum2, ComplexNum3 ...])

Description:

This returns the sum of the given complex numbers.

The arguments must be complex numbers in x+yi or x+yj format. Tip: Complex numbers can be created with the COMPLEX function.

Example:

IMSUM("2+4i", "1+2i") returns 3+6i

See also:

COMPLEX, IMDIV, IMPRODUKT, IMSUB

INDEX (cell in a particular row/column of a range)

Syntax:

INDEX(Range, Row, Column [, PartRange])

Description:

This returns a reference to the cell at the intersection of the specified **Row** and **Column** of the **Range**.

The desired cell range is to be specified for **Range**.

You use **Row** and **Column** to determine the row and column of the **Range** in which the cell to be returned is located (starting from the top left corner of the **Range**).

If **Row** or **Column** refers to a cell outside the range, a #REF! error value is returned.

The optional argument **PartRange** is just a placeholder (for compatibility with Microsoft Excel). It can be specified, but PlanMaker will not evaluate it.

Compatibility notes:

This function is not fully compatible with the Excel function of the same name. The following restrictions apply:

The argument **Range** does not support multiple selections (nonadjacent ranges).

The argument **PartRange** is not evaluated.

Example:

INDEX(B2:D4, 2, 2) returns a reference to cell C3

See also:

MATCH, VLOOKUP, HLOOKUP, LOOKUP, AREAS, CHOOSE

INDIRECT (create reference from text)

Syntax:

INDIRECT(Text [, Mode])

Description:

This returns a cell reference from the specified text.

For example, the specified **Text** can contain a fixed cell address (e.g., "C1") or variable references that are combined from the contents of one or more cells.

The optional logical value **Mode** specifies the format of the given cell address:

TRUE or omitted: Normal cell address in *A1 notation* (columns are labeled alphabetically and rows numerically)

FALSE: Cell address in the *R1C1 notation* (columns and rows are labeled numerically; an alternative type of cell addressing supported only by Microsoft Excel)

Examples:

If cell C1 contains the text *Flower*
INDIRECT("C1") returns a reference to cell C1 and the result: *Flower*

If cell C1 contains the text *E5*
and cell E5 the text *Flower*
INDIRECT(C1) returns a reference to cell E5 and the result: *Flower*

If cell C1 contains the text *E*
and cell C2 the text *5*
and cell E5 the text *Flower*
INDIRECT(C1&C2) returns a reference to cell E5 and the result: *Flower*

If cell E5 contains the text *Flower*
INDIRECT("E"&5") returns a reference to cell E5 and the result: *Flower*

If cell C1 contains the text *5*
and E5 the text *Flower*
INDIRECT("E"&C1) returns a reference to cell E5 and the result: *Flower*

See also:

ADDRESS, OFFSET, COLUMN, ROW

INT (round down to nearest integer)

Syntax:

INT(Number)

Description:

This rounds **Number** down to the nearest integer.

Note:

When applying this function to *negative* numbers, note the difference between this function and other rounding functions such as ROUNDDOWN:

ROUNDDOWN(-1.234, 0) returns -1, because here *the amount* of the number is rounded down to the nearest integer.

INT(-1.234) results in -2, since here the negative number itself is rounded down to the nearest integer.

Example:

INT(5.9) returns 5

INT(-5.1) returns -6

See also:

TRUNC, ROUNDDOWN, ROUNDUP, ROUND, MROUND, FIXED, EVEN, ODD, CEILING, FLOOR

INTERCEPT (intercept point of a regression line)

Syntax:

INTERCEPT(y_values, x_values)

Description:

This returns the y coordinate of the point where a best-fit line based on the given values intersects the y axis.

A *best-fit line* is the result of a *linear regression*. This is a statistical technique that adapts a line (called *"trendline"* or *"best-fit line"*) to a set of data points (for example, the results of a series of measurements).

You can use the INTERCEPT function to predict what y (the dependent variable) will be approximately if x (the independent variable) is zero.

This function can be used to predict, for example, the resistance of a temperature-dependent resistor at 0° after having measured the resistance at several other temperatures.

For the arguments **y_values** and **x_values**, you usually specify a cell range.

y_values are the dependent variables (the resistance in the above example).

x_values are the independent variables (the temperature in the above example).

Note:

Note that this function expects the **y_values** first and then the **x_values** – not the other way around.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

The resistance of a temperature-dependent resistor has been measured at several temperatures.

Cells A1:A4 contain the temperatures that were measured (the independent variables): 8, 20, 25, 28

Cells B1:B4 contain the resistances that were measured (the dependent variables): 261, 508, 608, 680

The resistance which will be delivered at 0°C can be estimated using the following formula:

INTERCEPT(B1:B4, A1:A4) returns 93.69817

At 0°, a resistance of 93.69817 (Ohm) would thus be expected.

Additional info:

INTERCEPT(y_values, x_values) equals FORECAST(0, y_values, x_values).

See also:

FORECAST, SKEW, SLOPE, STEYX, PEARSON

INTERSECTION (intersection of two ranges)

Syntax:

INTERSECTION(Range1, Range2)

Description:

This returns a reference to the intersection of **Range1** and **Range2** (i.e. the cell range where **Range1** and **Range2** overlap).

Additional info:

Alternatively, you can use the *spacebar* operator to obtain the intersection of two ranges. Example:

SUM(INTERSECTION(A1:D4, C3:E6))

is identical to:

SUM(A1:D4 C3:E6).

Compatibility notes:

This function does not exist in Microsoft Excel. When you save a document in Excel format, all occurrences of this function will be automatically replaced by spacebar operators (see additional info above).

Example:

INTERSECTION(A1:D4, C3:E6) returns a reference to the range C3:D4

IPMT (interest payment)

Syntax:

IPMT(Rate, Per, NPer, PV [, FV] [, Type])

Description:

This returns the interest portion of the payment in the specified period, based on an investment with periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how large the interest portion of the annuity is in a particular period for a loan.

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

Per is the payment period to be considered.

NPer is the total number of payment periods.

PV is the present value (initial amount)

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

A loan of $100,000 is taken out under the following terms:

Annual interest rate: 10%

Payments: monthly

Duration: 6 years

What is the interest portion of the payment (annuity) in month 32?

IPMT(10%/12, 32, 6*12, 100000) returns -534

Note that all values have to use the same time unit, *months* in this case, since the payments are made monthly.

Thus, in the above formula, the annual interest rate was divided by 12 to get the *monthly* interest rate and the number of payment periods was multiplied by 12.

Additional info:

The PPMT function is the counterpart to IPMT: It returns the principle component of the payment in this period.

The PMT function, on the other hand, returns the total amount of the payment (annuity) per period, that is, the sum of the principal portion and the interest portion.

See also:

PV, ISPMT, PPMT, CUMPRINC, CUMIPMT, PMT, RATE, FV, NPER

IRR (internal rate of return)

Syntax:

IRR(Values [, Guess])

Description:

This returns an estimate of the internal rate of return for a series of cash flows. In contrast to the MIRR function, no finance or reinvestment rates are taken into account.

The calculation is based on the cash flows specified in the **Values** argument. Payments have to be entered as negative values, and income as positive values. The cash flows may vary in size, but they must occur at equal time intervals (e.g., monthly or annually).

The function expects the following arguments:

For **Values**, you specify a cell range or array in which the cash flows were entered. There must be at least one payment and one income.

Guess (optional) lets you specify an estimate for the result (see note below).

Note:

The result of this function is calculated using an iterative search technique. If the function does not find a result with an accuracy of 0.00001% or better after 20 iterations, it aborts and the #NUM! error value is returned. If this occurs, try specifying a different value for **Guess**.

See also:

MIRR, XIRR, RATE, NPV

ISBLANK (is empty?)

Syntax:

ISBLANK(Cell)

Description:

This returns TRUE if the given cell is empty.

Example:

If A1 contains any value and C7 is empty, the following applies:

ISBLANK(A1) returns FALSE

ISBLANK(C7) returns TRUE

See also:

ISNA, IFEMPTY, COUNT, COUNTA

ISERR (is an error value other than #N/A?)

Syntax:

ISERR(Cell)

Description:

Unlike the ISERROR function, **ISERR** only returns TRUE if the given cell returns an error value **other than #N/A**.

Example:

ISERR(A1) returns TRUE if A1 contains an error value other than #NA.

ISERR(A1) returns FALSE if A1 does not return an error or if the NA() function is entered there.

See also:

ISERROR, ISNA, NA(), ERRORTYPE, IFERROR, section Error values

ISERROR (is an error value?)

Syntax:

ISERROR(Cell)

Description:

This returns TRUE if the given cell contains any kind of error value.

Example:

ISERROR(A1) returns TRUE if A1 contains any kind of error value.

See also:

ISERR, ISNA, NA(), ERRORTYPE, ISREF, IFERROR, section Error values

ISEVEN (is an even number?)

Syntax:

ISEVEN(Number)

Description:

This returns TRUE if the specified **Number** is even. Otherwise, FALSE is returned.

Note:

Number should be an integer. If this is not the case, PlanMaker automatically removes the decimal places and checks if the remaining number is even.

Example:

ISEVEN(2) returns TRUE

ISEVEN(2.75) returns TRUE

ISEVEN(3) returns FALSE

ISEVEN(3.75) returns FALSE

See also:

ISODD

ISFORMULA (is a formula?)

Syntax:

ISFORMULA(Cell)

Description:

This returns TRUE if the given cell contains a formula.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

If A1 contains a formula and C7 contains a fixed number or plain text, the following applies:

ISFORMULA(A1) returns TRUE

ISFORMULA(C7) returns FALSE

See also:

ISREF, FORMULATEXT

ISLOGICAL (is a logical value?)

Syntax:

ISLOGICAL(Cell)

Description:

This returns TRUE if the given cell contains a logical value (i.e., TRUE or FALSE).

Example:

If A1 contains TRUE and C7 contains a text string, the following applies:

ISLOGICAL(A1) returns TRUE

ISLOGICAL(C7) returns FALSE

See also:

ISTEXT, ISNUMBER, ISREF, ISNONTEXT

ISNA (is not available?)

Syntax:

ISNA(Cell)

Description:

This returns TRUE if the given cell contains a #N/A error value.

Cells contain a #N/A error value if they contain the formula =NA() or a calculation resulting in a #N/A error.

For more information on the usage of #N/A error values, see the description of the NA (error value #N/A) function.

Example:

The function NA() was entered in cell A1 to indicate that a value that is absolutely necessary for another calculation is not yet available here.

Cell C7 performs a calculation with cell A1, for example, A1*2. Since cell A1 returns the error value #N/A, however, the result of this calculation is also the error value #N/A.

Thus, the following applies:

ISNA(A1) returns TRUE

ISNA(C7) also returns TRUE

See also:

NA(), ERROR.TYPE, ISERR, ISERROR, IFERROR, IFNA, section Error values

ISNONTEXT (is no text?)

Syntax:

ISNONTEXT(Cell)

Description:

This returns TRUE if the given cell does not contain text.

Example:

If A1 contains a fixed string or a calculation that returns a string, while C7 contains a number, the following applies:

ISNONTEXT(A1) returns FALSE

ISNONTEXT(C7) returns TRUE

Additional info:

ISTEXT is the counterpart of this function: It returns TRUE if the cell contains a text string.

See also:

ISTEXT, ISLOGICAL, ISNUMBER

ISNUMBER (is numeric?)

Syntax:

ISNUMBER(Cell)

Description:

This returns TRUE if the given cell contains a number.

Dates are also considered as numbers, whereas logical values (TRUE and FALSE) are not.

Example:

If the cells A1 to A4 contain the values:

"Text", 42, 09/25/2018 and TRUE

Then the following applies:

ISNUMBER(A1) returns FALSE

ISNUMBER(A2) returns TRUE

ISNUMBER(A3) returns TRUE

ISNUMBER(A4) returns FALSE

See also:

ISNUMBERP, ISLOGICAL, ISTEXT, ISNONTEXT, ISREF, COUNT

ISNUMBERP (PlanMaker 97 compatibility function)

Syntax:

ISNUMBERP(Cell)

Description:

This returns TRUE if the given cell contains a number or a logical value.

Dates are *also* considered as numbers.

Note:

This function was retained only for compatibility with PlanMaker 97. If possible, use the Excel-compatible function ISNUMBER (without "P") instead of this function.

The difference:

ISNUMBERP treats logical values as numbers, whereas ISNUMBER does not.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

If the cells A1 to A4 contain the values:

"Text", 42, 09/25/2018 and TRUE

Then the following applies:

ISNUMBERP(A1) returns FALSE

ISNUMBERP(A2) returns TRUE

ISNUMBERP(A3) returns TRUE

ISNUMBERP(A4) returns TRUE (!)

For comparison: ISNUMBER(A4) returns FALSE

See also:

ISNUMBER

ISODD (is an odd number?)

Syntax:

ISODD(Number)

Description:

This returns TRUE if the specified **Number** is odd. Otherwise, FALSE is returned.

Note:

Number should be an integer. If this is not the case, PlanMaker automatically removes the decimal places and checks if the remaining number is odd.

Example:

ISODD(1) returns TRUE

ISODD(1.75) returns TRUE

ISODD(2) returns FALSE

ISODD(2.75) returns FALSE

See also:

ISEVEN

ISOWEEK (ISO week number)

Syntax:

ISOWEEK(Date [, ReturnYear])

Description:

This returns the ISO week number of a date (i.e. the week number according to the ISO 8601 standard).

Unlike the WEEKNUM function, this function does not use the calculation method commonly used in the USA and many other parts of the world. On the contrary, it is based on the ISO standard 8601, which is binding in Germany, Austria and Switzerland, among other countries. It specifies the following:

1. Weeks always start on a Monday (not Sunday).

2. The first week of a year is defined as the week that includes January 4th.

Note:

This may lead to situations where December 29th, 30th and 31st are considered part of the first week of the next year – or where January 1st, 2nd and 3rd are considered part of the last week of the previous year.

For this reason, the ISOWEEK function has an additional parameter named **ReturnYear**, which has the following effect:

If set to 0 or omitted, the function returns the *week number*.

If set to 1, the function instead returns the *year* to which the date belongs (according to the ISO 8601 standard). See examples below.

Example:

ISOWEEK("01/01/2020") returns 1.

ISOWEEK("01/01/2017") returns 52 (since this day was a Sunday).

ISOWEEK("01/01/2017", 1) accordingly returns 2016 – not 2017.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Tip: As an alternative to this function, you can use the Excel-compatible ISOWEEKNUM function. It returns exactly the same results, but it does not have the optional parameter for displaying the year.

See also:

ISOWEEKNUM, WEEKNUM, YEAR, MONTH, DAY, WEEKDAY

ISOWEEKNUM (ISO week number)

Syntax:

ISOWEEKNUM(Date)

Description:

This returns the ISO week number of a date (i.e., the week number according to the ISO 8601 standard).

Unlike the WEEKNUM function, this function does not use the calculation method commonly used in the USA and many other parts of the world. On the contrary, it is based on the ISO standard 8601, which is binding in Germany, Austria and Switzerland, among other countries. It specifies the following:

1. Weeks always start on a Monday (not Sunday).

2. The first week of a year is defined as the week that includes January 4th.

Note:

This may lead to situations where December 29th, 30th and 31st are considered part of the first week of the next year – or where January 1st, 2nd and 3rd are considered part of the last week of the previous year. See examples below.

Tip: For this reason, PlanMaker provides an alternative (not Excel-compatible) function called ISOWEEK. This returns exactly the same results, but it can also return the year to which the date belongs (according to the ISO 8601 standard).

Example:

ISOWEEKNUM("01/01/2020") returns 1.

ISOWEEKNUM("01/01/2017") returns 52 (since this day was a Sunday).

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

ISOWEEK, WEEKNUM, YEAR, MONTH, DAY, WEEKDAY

ISPMT (interest payment)

Syntax:

ISPMT(Rate, Per, NPer, PV)

Description:

This returns the interest paid in the specified period for an investment with equal installments.

The function expects the following arguments:

Rate is the interest rate (per payment period)

Per is the payment period to be considered

NPer is the total number of payment periods

PV is the present value (initial amount)

See also:

PPMT, PMT, IPMT

ISREF (is a valid reference?)

Syntax:

ISREF(Value)

Description:

This returns TRUE if the specified value is a valid cell reference.

Example:

ISREF(A1) returns TRUE

ISREF($A$1) returns TRUE

ISREF(A1:C7) returns TRUE

ISREF("any text") returns FALSE

ISREF(FirstQuarter) returns FALSE until you assign the name "FirstQuarter" to a cell range – from then on, this calculation will also return TRUE.

See also:

ISFORMULA, ISERROR, ISLOGICAL, ISNUMBER, ISTEXT

ISTEXT (is text?)

Syntax:

ISTEXT(Cell)

Description:

This returns TRUE if the given cell contains text.

Example:

If A1 contains a fixed string or a calculation that returns a string, while C7 contains a number, the following applies:

ISTEXT(A1) returns TRUE

ISTEXT(C7) returns FALSE

Additional info:

ISNONTEXT is the counterpart of this function: It returns TRUE if there is *no* text string in the cell.

See also:

ISNONTEXT, ISLOGICAL, ISNUMBER, ISREF

KURT (kurtosis)

Syntax:

KURT(Number1, Number2, Number3, Number4 [, Number5 ...])

or

KURT(Range1 [, Range2, Range3 ...])

Description:

This returns the kurtosis of a probability distribution.

The kurtosis is a measure of the "peakedness" of a distribution compared to a normal distribution.

For a steeper curved ("narrow-peaked") distribution, the kurtosis is a positive value, while for a flatter curved ("broad-peaked") distribution, it is a negative value.

If the kurtosis approaches zero, the distribution has a good approximation to a normal distribution (see NORM.DIST/NORMDIST function).

Number1, **Number2**, **Number3**, etc., are the values to be evaluated. Empty cells, text and logical values are ignored.

At least four values have to be specified; otherwise, the function returns a #DIV/0! error value (division by zero).

Note:

This function does not accept value pairs (x value and y value) as arguments, but only the values of the distribution. If the same values appear multiple times, they must be repeated in the argument list accordingly (see example).

Example:

You measured the height of several test persons and obtained the following results: 1 x 1.60m, 2 x 1.65m, 4 x 1.70m, 2 x 1.75m and 1x1.80m.

To calculate the kurtosis of this distribution, the following formula can be used:

KURT(1.60, 1.65, 1.65, 1.70, 1.70, 1.70, 1.70, 1.75, 1.75, 1.80) returns 0.08036.

See also:

SKEW, NORM.DIST/NORMDIST

LARGE (k-th largest number)

Syntax:

LARGE(Range, k)

Description:

This returns the **k**-th largest value in a data set. If k=1, the largest value will be returned, if k=2, the second largest value will be returned, etc.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

k determines which number to return.

k must not be smaller than 1 or greater than the total number of values in **Range**. Otherwise, the function returns a #NUM! error value.

Example:

If A1:A5 contains the values 4, 2, 6, 5 and 9, the following applies:

LARGE(A1:A5, 1) returns 9

LARGE(A1:A5, 2) returns 6

LARGE(A1:A5, 3) returns 5

etc.

See also:

SMALL, MAX, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, RANK.EQ/RANK.AVG/RANK

LASTPRINTED (date last printed)

Syntax:

LASTPRINTED()

Description:

This returns the date on which the current document was last printed.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

LASTSAVED

LASTSAVED (date last saved)

Syntax:

LASTSAVED()

Description:

This returns the date on which the current document was last saved.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

LASTPRINTED

LCM (least common multiple)

Syntax:

LCM(Number1 [, Number2, Number3 ...])

Description:

This returns the LCM (least common multiple) of a set of numbers. The LCM is the smallest positive integer that is a multiple of the arguments.

You can specify numbers or cell references as arguments.

All given values **Number1**, **Number2**, etc., have to be ≥ 1. Furthermore, only integers should be used. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

LCM(5, 10, 15) returns 30

If A1:A3 contains the values 4, 8 and 6, the following applies:

LCM(A1:A3) returns 24

See also:

GCD

LEFT (left part of a text string)

Syntax:

LEFT(TextString [, n])

Description:

This returns the first **n** characters of a **TextString**.

If **n** is omitted, the function only returns the first character.

Example:

LEFT("peanut", 3) returns pea

LEFT("peanut") returns p

See also:

RIGHT, MID, REPLACE, LEN

LEN (length)

Syntax:

LEN(Value)

Description:

This returns the number of characters in a text string.

If the specified value is a number, a date or a logical value, it will be converted to a string automatically.

Note: For *dates*, the serial date number is converted to a string and not the formatted date itself (see last example).

Example:

LEN("Text") returns 4

LEN(42) returns 2

LEN(-42) returns 3

LEN(42.5) returns 4

LEN(42.50) also returns 4 (the superfluous zero is automatically removed!)

LEN(TRUE) returns 4

Please note when specifying dates: If cell A1 contains the date 09/25/2014 in any format, the following applies:

LEN(A1) returns 5

Reason: Date values are always stored as numbers in a spreadsheet – the date 09/25/2014, for example, as 41907. Thus, the result of the calculation is unexpected but completely correct – because:

LEN(41907) also returns 5

See also:

LEFT, RIGHT, MID

LINEST (statistics of a linear regression)

Syntax:

LINEST(y_values [, x_values] [, Constant] [, Stats])

Description:

This returns statistics of a linear regression.

Linear regression is a statistical technique that adapts a line (called *"trendline"* or *"best-fit line"*) to a set of data points (for example, the results of a series of measurements).

The equation for this trendline is as follows:

$y = m*x + b$

The LINEST function calculates a trendline from the given y values and x values, and then returns an array in the form {m,b}. The coefficient m is the slope of the trendline, and the constant b is the y-intercept point (the point where the trendline intersects the y axis).

If there are several ranges with x values, the equation is as follows:

$y = (m_1*x_1) + (m_2*x_2) + ... + (m_n*x_n) + b$

In this case, LINEST returns an array in the form $\{m_n , m_{n-1} , ... , m_2 , m_1 , b\}$.

If the optional argument **Stats** is used, several statistics of the regression (standard error, coefficient of determination, etc.) are added to the resulting array.

The LINEST function expects the following arguments:

y_values: Here, you specify the existing y coordinates, that is, the dependent values. **y_values** can be either a cell reference or an array.

x_values: Here, you have the option of specifying the x coordinates, that is, the independent values. **x_values** can also be either a cell reference or an array. If **x_values** are omitted, the function automatically uses the values 1, 2, 3, ... (according to the number of existing **y_values**).

Constant (optional) is a logical value that lets you specify if the constant b (the y-intercept point) should be set to zero:

TRUE or omitted: b is automatically calculated from the available data.

FALSE: b is set to zero. This forces the trendline to run through the zero point. The m coefficient(s) are adjusted accordingly.

Stats (optional) is a logical value that lets you specify if additional regression statistics should be returned:

FALSE or omitted: Do not return additional statistics.

TRUE: Return additional statistics. In this case, the function returns an array that not only includes m and b, but also the following values:

{m_n , m_{n-1} , ... , m_2 , m_1 , b ; sm_n , sm_{n-1} , ... , sm_2 , sm_1 , sb ; R^2 , sey ; F. df ; ssreg , ssresid}

A graphical representation of this array would look as follows:

m_n	m_{n-1}	...	m_2	m_1	b
sm_n	sm_{n-1}	...	sm_2	sm_1	sb
R^2	sey				
F	df				
ssreg	ssresid				

The above abbreviations represent the following values:

Abbreviation	Explanation
sm_1, sm_2, etc.	Standard error for the coefficients m_1, m_2, etc.
sb	Standard error for constant b
R^2	Square of the Pearson product moment correlation coefficient
sey	Standard error for y
F	F statistic
df	Degrees of freedom
ssreg	Regression sum of squares
ssresid	Residual sum of squares

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

LINEST({4; 5; 6}) returns the matrix {1.3}. This indicates that the coefficient m equals 1 and the constant b equals 3. (Since the x values were not specified, the LINEST function automatically uses the values 1, 2, 3, ...)

See also:

LOGEST, TREND, GROWTH

LN (natural logarithm)

Syntax:

LN(Number)

Description:

This returns the natural logarithm (base *e*) of **Number**.

Number must be > 0.

Example:

LN(0.7) returns -0.3567

LN(EXP(1.234)) returns 1.234

Additional info:

The inverse function is EXP

See also:

EXP, LOG, LOG10, POWER

LOG (logarithm)

Syntax:

LOG(Number [, Base])

Description:

This returns the logarithm of **Number** to the specified base.

Number must be > 0.

If **Base** is omitted, base 10 is used.

Example:

LOG(100) equals $\log_{10}$ (100) and returns 2

LOG(10 ^ 0.1234) returns 0.1234

LOG(8, 2) equals $\log_2$ (8) and returns 3

See also:

LOG10, LN, EXP, POWER

LOG10 (base-10 logarithm)

Syntax:

LOG10(Number)

Description:

This returns the base-10 logarithm of **Number**.

Number must be > 0.

Example:

LOG10(100) returns 2

LOG10(10 ^ 0.1234) returns 0.1234

See also:

LOG, LN, EXP

LOGEST (statistics of an exponential regression)

Syntax:

LOGEST(y_values [, x_values] [, Constant] [, Stats])

Description:

This returns statistics of an exponential regression.

Exponential regression is a statistical technique that adapts an exponential curve to a set of data points (for example, the results of a series of measurements).

The equation for this exponential curve is as follows:

y = b*m^x

If more than one range of x values is given, the equation is:

$y = b * (m_1\text{^}x_1) * (m_2\text{^}x_2) * ... * (m_n\text{^}x_n)$

The LOGEST function calculates this exponential line from the given y values and x values, and then returns an array in the form $\{m_n , m_{n-1} , ... , m_2 , m_1 , b\}$.

If the optional argument **Stats** is used, several statistics of the regression (standard error, coefficient of determination, etc.) are added to the resulting array.

The LOGEST function expects the following arguments:

y_values: Here, you specify the existing y coordinates, that is, the dependent values. **y_values** can be either a cell reference or an array.

x_values: Here, you have the option of specifying the x coordinates, that is, the independent values. **x_values** also can be either a cell reference or an array. If **x_values** are omitted, the function automatically uses the values 1, 2, 3, ... (according to the number of existing **y_values**).

Constant (optional) is a logical value that lets you specify if the constant b should be set to 1 (one):

TRUE or omitted: b is automatically calculated from the available data.

FALSE: b is forced to equal 1 (one). The m coefficients are adjusted accordingly.

Stats (optional) is a logical value that lets you specify if additional regression statistics should be returned:

FALSE or omitted: Do not return additional statistics.

TRUE: Return additional statistics. In this case, the function returns an array that not only includes m and b, but also the following values:

$\{m_n , m_{n-1} , ... , m_2 , m_1 , b ; sm_n , sm_{n-1} , ... , sm_2 , sm_1 , sb ; R^2 , sey ; F. df ; ssreg , ssresid\}$

A graphical representation of this array would look as follows:

m_n	m_{n-1}	...	m_2	m_1	b
sm_n	sm_{n-1}	...	sm_2	sm_1	sb
R^2	sey				
F	df				
ssreg	ssresid				

The above abbreviations represent the following values:

Abbreviation	Explanation
sm_1, sm_2, etc.	Standard error for the coefficients m_1, m_2, etc.
sb	Standard error for constant b
R^2	Square of the Pearson product moment correlation coefficient
sey	Standard error for y
F	F statistic
df	Degrees of freedom
ssreg	Regression sum of squares
ssresid	Residual sum of squares

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

See also:

LINEST, TREND, GROWTH

LOGINV (percentiles of the log-normal distribution)

Note: LOGINV is supplemented by the new identical function LOGNORM.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

LOGINV(Probability, Mean, StdDev)

Description:

This returns the inverse of the log-normal distribution function (i.e., percentiles of this distribution).

Probability is the given probability from the log-normal distribution. The values must be within the range 0 to 1.

Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x). This must be > 0.

See also:

LOGNORM.INV, LN, LOGNORM.DIST/LOGNORMDIST, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

LOGNORMDIST (log-normal distribution)

Note: LOGNORMDIST is supplemented by the new function LOGNORM.DIST with additional functionality, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

LOGNORMDIST(x, Mean, StdDev)

Description:

This returns the probabilities of a cumulative log-normally distributed random number.

You can use this function to view probability distributions in which the natural logarithm is normally distributed rather than the random variable itself.

x is the value to be evaluated. **x** must be > 0.

Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x). It must be > 0.

See also:

LOGNORM.DIST, LN, LOGNORM.INV/LOGINV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

LOGNORM.DIST (log-normal distribution)

Syntax:

LOGNORM.DIST(x, Mean, StdDev, Cumulative)

Description:

This returns the probabilities of a (cumulative) log-normally distributed random number.

You can use this function to view probability distributions in which the natural logarithm is normally distributed rather than the random variable itself.

x is the value to be evaluated. **x** must be > 0.

Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x). **StdDev** must be > 0.

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative normal distribution function is calculated. By entering the value FALSE, the normal probability density function is calculated.

Note:

The LOGNORM.DIST function supplements the previous LOGNORMDIST function with the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

The inverse function is LOGNORM.INV

See also:

LOGNORMDIST, LN, LOGNORM.INV/LOGINV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

LOGNORM.INV (percentiles of the log-normal distribution)

Syntax:

LOGNORM.INV(Probability, Mean, StdDev)

Description:

This returns the inverse of the log-normal distribution function (i.e., percentiles of this distribution).

Probability is the given probability from the log-normal distribution. The values must be within the range 0 to 1.

Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x). This must be > 0.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

LOGNORM.INV is the inverse function of LOGNORM.DIST

See also:

LOGINV, LN, LOGNORM.DIST/LOGNORMDIST, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

LOOKUP (search cell range)

Annotation:

The LOOKUP function searches a cell range for a search criterion (for example, a number or a text string) and then returns the contents of another cell with the corresponding position.

Important: The data to be searched must be available in sorted order. Tip: There are two alternatives to this function that can also search unsorted data and are more flexible in general: VLOOKUP and HLOOKUP

The LOOKUP function can be used in two different ways:

Usage with two one-dimensional ranges: Here, two vectors (i.e., one-dimensional ranges) are specified for the function: **Lookup Vector** and **ResultVector**. If the function finds the search criterion in the Lookup Vector, it returns the cell with the same position in the Result Vector as the result.

Usage with one two-dimensional range: Here, a two-dimensional range is specified for the function. If it finds the search criterion in the *first* row (or column) of this range, it returns the cell with the same position in the *last* row (or column) of this range as the result.

Usage with two one-dimensional ranges

Syntax:

LOOKUP(Crit, LookupVector, ResultVector)

Description:

This variant of LOOKUP scans **LookupVector** for the specified value in **Crit**. If found, the function returns the content of the cell located in the same row (or column) of the **ResultVector**.

Crit is the value for which the search is performed. The search is case-insensitive.

LookupVector is the cell range or array to be evaluated. This can contain text strings, numbers or logical values. It must be a vector, which means that **LookupVector** is a range of either only one row or one column.

Important: The data in **LookupVector** must be *sorted* in ascending order. Otherwise, this function may return incorrect results. If the values are not sorted, use VLOOKUP or HLOOKUP instead of LOOKUP.

ResultVector is the cell range or array from which the result of the function is to be returned. It also has to be a vector. In addition, its size and orientation (horizontal or vertical) must be identical to **LookupVector**.

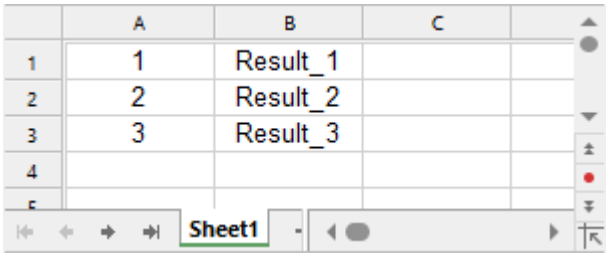
Note:

If LOOKUP does not find a value that exactly matches **Crit**, the *next smaller* value is automatically taken.

If no smaller value exists because **Crit** is smaller than the smallest value available, the function returns the #N/A error value.

Example:

In the following table ...



... LOOKUP returns the following results:

LOOKUP(1, A1:A3, B1:B3) returns Result_1

LOOKUP(2, A1:A3, B1:B3) returns Result_2

LOOKUP(3, A1:A3, B1:B3) returns Result_3

LOOKUP(2.5, A1:A3, B1:B3) returns Result_2 (because it is the next smallest value)

LOOKUP(10, A1:A3, B1:B3) returns Result_3 (because it is the next smallest value)

LOOKUP(0, A1:A3, B1:B3) returns #N/A

See also:

VLOOKUP, HLOOKUP, INDEX, MATCH

Usage with one two-dimensional range

Syntax:

LOOKUP(Crit, Range)

Description:

This variant of LOOKUP scans the *first* column (or row) of **Range** for the specified value in **Crit**. If found, the function returns the content of the cell located in the same row in the *last* column of **Range**.

Alternatively, this function can also search the range column by column. The search direction is determined by the dimensions of **Range**:

If **Range** contains more columns than rows, the function searches the first *row* and returns the value from the last *row*.

If **Range** contains more rows than columns or is square, the function searches the first *column* and returns the value from the last *column*.

Crit is the value for which the search is performed. The search is case-insensitive.

Range is the cell range or array on which the search is to be performed. In the first row (or column), it must contain the values for which the search is performed and, in the last row (or column), the results to be returned. These can be text strings, numbers or logical values. The range must be two-dimensional, that is, it must have at least two rows and two columns.

Important: The scanned data in the first row (or column) must be *sorted* in ascending order. Otherwise, this function may return incorrect results. If the values are not sorted, use VLOOKUP or HLOOKUP instead of LOOKUP.

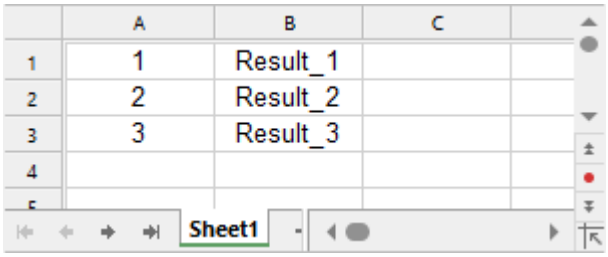
Note:

If LOOKUP does not find a value that exactly matches **Crit**, the *next smaller* value is automatically taken.

If no smaller value exists because **Crit** is smaller than the smallest value available, the function returns the #N/A error value.

Example:

In the following table ...



... LOOKUP returns the following results:

LOOKUP(1, A1:B3) returns Result_1

LOOKUP(2, A1:B3) returns Result_2

LOOKUP(3, A1:B3) returns Result_3

LOOKUP(2.5, A1:B3) returns Result_2 (because it is the next smallest value)

LOOKUP(10, A1:B3) returns Result_3 (because it is the next smallest value)

LOOKUP(0, A1:B3) returns #N/A

See also:

VLOOKUP, HLOOKUP, INDEX, MATCH

LOWER (convert text to lowercase)

Syntax:

LOWER(Text string)

Description:

This converts all characters in a text string to lowercase letters.

Example:

LOWER("PlanMaker") returns planmaker

See also:

UPPER, PROPER

MATCH (relative position in a range)

Syntax:

MATCH(Crit, Vector [, Mode])

Description:

This returns the relative position of **Crit** in **Vector**. If, for example, **Crit** is the fifth element of **Vector**, the function returns 5.

Crit is the value for which the search is performed. This can be any kind of value or a cell reference.

Vector is the cell range or array to be evaluated. This can contain either text strings, numbers or logical values. **Vector** must be a vector, which is an array or a cell range of either only *one* row or *one* column.

Mode specifies the type of search to be performed:

1 or omitted: Find the largest value that is equal to or *smaller* than **Crit**. ***Important:*** In this mode, the elements in **Vector** have to be sorted in *ascending* order. Otherwise, the function may return incorrect results.

0: Find the first value that is equal to **Crit**. In this mode, the elements in **Vector** do *not* have to be in sorted order.

-1: Find the smallest value that is equal to or *larger* than **Crit**. ***Important:*** In this mode, the elements in **Vector** have to be sorted in *descending* order. Otherwise, the function may return incorrect results.

Tip: If you use **Mode** 0 and you search for a text string, wildcard characters can be used in **Crit**: A question mark (?) stands for any single character, and an asterisk (*) stands for any sequence of characters.

Example:

MATCH("b", {"a";"b";"c";"d"}, 0) returns 2 because "b" is the second element of the given vector.

See also:

INDEX, LOOKUP, VLOOKUP, HLOOKUP

MAX (maximum)

Syntax:

MAX(Number1 [, Number2, Number3 ...])

Description:

This returns the largest value within the given numbers.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

MAX(5, 7, 4, 9) returns 9

If the cells A1:A3 contain the values -1, -2 and TRUE, the following applies:

MAX(A1:A3) returns -1 (the logical value TRUE is ignored)

MAX(A1:A3, TRUE) returns 1 (the entered value TRUE is evaluated as 1)

MAX(A1:A3, FALSE) returns 0 (the entered value FALSE is evaluated as 0)

See also:

MAXA, MIN, LARGE

MAXA (maximum)

Syntax:

MAXA(Number1 [, Number2, Number3 ...])

Description:

The **MAXA** function is almost identical to the MAX function (see note). Thus, it returns the largest value within the given numbers.

Number1, **Number2**, etc., are the values to be evaluated.

Note:

Unlike MAX, MAX**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Example:

MAXA(5, 7, 4, 9) returns 9

If the cells A1:A3 contain the values -1, -2 and TRUE, the following applies:

MAXA(A1:A3) returns 1 (since TRUE = 1)

MAX(A1:A3), on the other hand, results in -1 (because the logical value TRUE is ignored in the MAX function)

See also:

MAX

MAXIFS (conditional maximum)

Syntax:

MAXIFS(MaxRange, Range1, Criterion1 [, Range2, Criterion2 ...])

Description:

This function determines the largest value in a range that is evaluated according to one or more conditions.

MaxRange is the value range whose largest value is to be determined.

Range1 is the first cell range to be checked for the specified criterion. This range must have the same dimensions as **MaxRange**.

Criterion1 is the condition that the values in **Range1** have to fulfill to be included.

Range2, Criterion2, etc., (optional) are further cell ranges that can be checked for a condition.

Note:

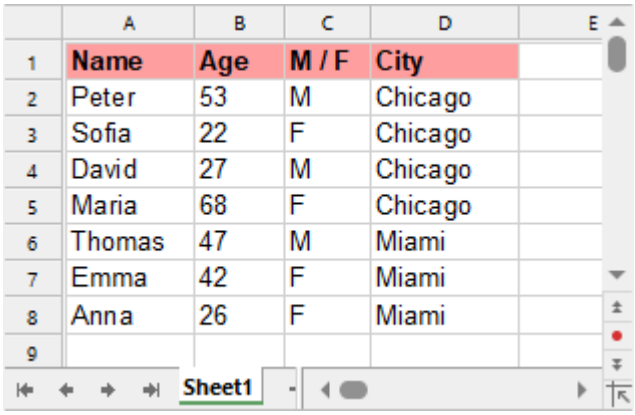
The search criterion can be a number, a text string or an expression, for example, "42", "screws", "03/12/2020", "TRUE", ">10" and should always be enclosed within double quotation marks (").

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

Example:

You have the following table with people of different ages:



	A	B	C	D	E
1	Name	Age	M / F	City	
2	Peter	53	M	Chicago	
3	Sofia	22	F	Chicago	
4	David	27	M	Chicago	
5	Maria	68	F	Chicago	
6	Thomas	47	M	Miami	
7	Emma	42	F	Miami	
8	Anna	26	F	Miami	
9					

Sheet1

MAXIFS(B2:B8, C2:C8, "M") returns 53 for the oldest man.

MAXIFS(B2:B8, C2:C8, "F", D2:D8, "Miami") returns 42 for the oldest woman from Miami.

MAXIFS(B2:B8, B2:B8, "<30") returns 27 for the oldest person under 30.

See also:

MINIFS, MAX, SUMIFS, AVERAGEIFS

MDETERM (matrix determinant)

Syntax:

MDETERM(Array)

Description:

This returns the determinant of a square matrix.

Array is a cell range or array containing the matrix to be evaluated. It must contain numeric values only and be square, i.e., have the same number of rows and columns.

Example:

MDETERM({9,8,8; 2,4,6; 1,2,1}) returns -40

See also:

Section Working with arrays, MINVERSE, MMULT, MSOLVE, TRANSPOSE

MEDIAN (Median)

Syntax:

MEDIAN(Number1 [, Number2, Number3 ...])

Description:

This returns the median of the given set of numbers.

The *median* is the number in the middle of a set of numbers. Half of the numbers are smaller than the median, while the other half are larger.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Note:

If the total number of values is *odd*, MEDIAN returns the number in the middle of the set of numbers.

If the total number of values is *even*, MEDIAN returns the average of the two numbers in the middle (see second example).

Example:

MEDIAN(1, 2, 3, 4, 5) returns 3

MEDIAN(1, 2, 3, 4, 5, 1000) returns 3.5

See also:

AVERAGE, MODE.SNGL/MODE, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE

MID (part of a text string)

Syntax:

MID(TextString, Start, Count)

Description:

This returns a part of a **TextString**.

The section starts at **Start** and is **Count** characters long.

Example:

MID("peanut", 2, 3) returns ean

MID("peanut", 2, 5) returns eanut

MID("peanut", 2, 100) also returns eanut

MID("peanut", 100, 100) returns an empty text string

See also:

LEFT, RIGHT, REPLACE, FIND, SEARCH, LEN

MILLISECONDS (milliseconds)

Syntax:

MILLISECONDS(Time)

Description:

This returns the milliseconds of a time value.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Example:

MILLISECONDS("12:34:56.555") returns 555

MILLISECONDS("09/25/2018 12:34:56.555") returns 555

If cell A1 contains a date with the time 12:34:56,555, then MILLISECONDS(A1) returns 555

See also:

HOUR, MINUTE, SECOND

MIN (minimum)

Syntax:

MIN(Number1 [, Number2, Number3 ...])

Description:

This returns the smallest value within the given numbers.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated.

If this function refers to empty cells, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Example:

MIN(5, 7, 4, 9) returns 4

If the cells A1:A3 contain the values 5, 2 and TRUE, the following applies:

MIN(A1:A3) returns 2 (the logical value TRUE is ignored)

MIN(A1:A3, TRUE) returns 1 (the entered value TRUE is evaluated as 1)

MIN(A1:A3, FALSE) returns 0 (the entered value FALSE is evaluated as 0)

See also:

MINA, MAX, SMALL

MINA (minimum)

Syntax:

MINA(Number1 [, Number2, Number3 ...])

Description:

The **MINA** function is almost identical to the MIN function (see note). It thus returns the smallest value within the given numbers.

Number1, **Number2**, etc., are the values to be evaluated.

Note:

Unlike MIN, MIN**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Example:

MINA(5, 7, 4, 9) returns 4

If the cells A1:A3 contain the values 5, 2 and TRUE, the following applies:

MINA(A1:A3) returns 1 (since TRUE = 1)

MIN(A1:A3), on the other hand, results in 2 (because the logical value TRUE is ignored in the MIN function)

See also:

MIN

MINIFS (conditional minimum)

Syntax:

MINIFS(MinRange, Range1, Criterion1 [, Range2, Criterion2 ...])

Description:

This function determines the smallest value in a range that is evaluated according to one or more conditions.

MinRange is the value range whose smallest value is to be determined.

Range1 is the first cell range to be checked for the specified criterion. This range must have the same dimensions as **MinRange**.

Criterion1 is the condition that the values in **Range1** have to fulfill to be included.

Range2, Criterion2, etc., (optional) are further cell ranges that can be checked for a condition.

Note:

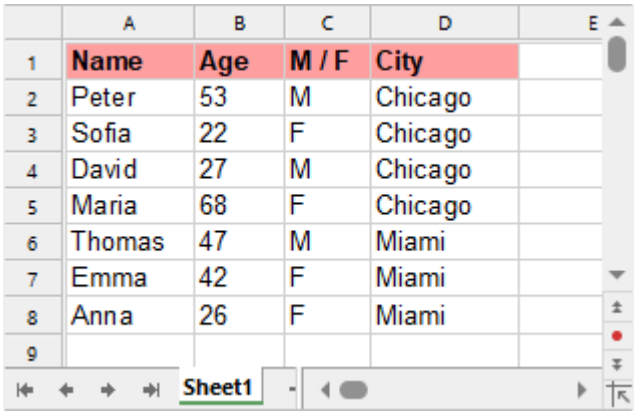
The search criterion can be a number, a text string or an expression, for example, "42", "screws", "03/12/2020", "TRUE", ">10" and should always be enclosed within double quotation marks (").

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

Example:

You have the following table with people of different ages:



	A	B	C	D
1	Name	Age	M / F	City
2	Peter	53	M	Chicago
3	Sofia	22	F	Chicago
4	David	27	M	Chicago
5	Maria	68	F	Chicago
6	Thomas	47	M	Miami
7	Emma	42	F	Miami
8	Anna	26	F	Miami
9				

Sheet1

MINIFS(B2:B8, C2:C8, "M") returns 27 for the youngest man.

MINIFS(B2:B8, C2:C8, "F", D2:D8, "Miami") returns 26 for the youngest woman from Miami.

MINIFS(B2:B8, B2:B8, ">30") returns 42 for the youngest person over 30.

See also:

MAXIFS, MIN, SUMIFS, AVERAGEIFS

MINUTE (minute)

Syntax:

MINUTE(Time)

Description:

This returns the minutes of a time value.

Example:

MINUTE("12:34:56 AM") returns 34

MINUTE("09/25/2018 10:30:56 AM") returns 30

If cell A1 contains a date with the time 10:30:56, then MINUTE(A1) returns 30

See also:

HOUR, SECOND, MILLISECONDS

MINVERSE (inverse matrix)

Syntax:

MINVERSE(Array)

Description:

This returns the inverse of a square matrix.

The inverse A^{-1} of matrix A is the matrix by which A must be multiplied to obtain the identity matrix 1.

Array is a cell range or array containing the matrix to be evaluated. It must contain numeric values only and be square, i.e., have the same number of rows and columns.

Additional info:

Matrices whose determinant is zero (singular matrices) cannot be inverted. In this case, the MINVERSE function returns a #NUM! error value.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Example:

MINVERSE({9,8,8;2,4,6;1,2,1}) returns:

{0.2,-0.2,-0.4;-0.1,-0.025,0.95;0,0.25,-0.5}

See also:

Section Working with arrays, MDETERM, MMULT, MSOLVE, TRANSPOSE

MIRR (modified internal rate of return)

Syntax:

MIRR(Values, FinanceRate, ReinvestRate)

Description:

This returns the modified internal rate of return for a series of cash flows. Unlike the IRR function, MIRR considers the finance rate (for investments) and the reinvest rate (for reinvestments).

The calculation is based on the cash flows specified in the **Values** argument. Payments have to be entered as negative values, and income as positive values. The cash flows may vary in size, but they must occur at equal time intervals (e.g., monthly or annually).

The function expects the following arguments:

For **Values**, you specify a cell range or array in which the cash flows were entered. There must be at least one payment and one income.

FinanceRate is the interest rate paid for investments.

ReinvestRate is the interest rate received for reinvestments.

See also:

IRR, NPV, XIRR, RATE

MMULT (product of matrices)

Syntax:

MMULT(Array1, Array2)

Description:

This returns the product of two matrices.

Array1 and **Array2** are cell ranges or arrays that contain the matrices to be evaluated. They must contain numeric values only.

Please note: The number of *columns* in **Array1** must equal the number of *rows* in **Array2**, as multiplication is not possible otherwise.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Example:

MMULT({1,2;3,4}, {5,6;7,8}) returns {19,22;43,50}

See also:

Section Working with arrays, MDETERM, MINVERSE, MSOLVE, TRANSPOSE

MOD (remainder of a division, Excel method)

Syntax:

MOD(x, y)

Description:

This returns the remainder of the division of **x** by **y** according to the "Excel method".

Important: This function returns incorrect results in some cases (see additional info below)!

Example:

MOD(5, 3) returns 2

MOD(5, 0) returns the #DIV/0! error value

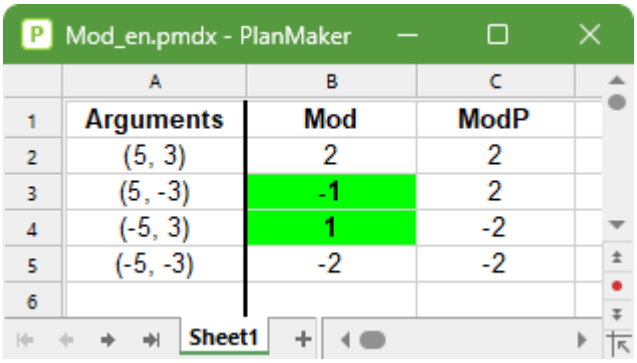
MOD(5, -3) returns -1 (which is incorrect – see additional info below!)

Additional info:

Microsoft Excel returns **incorrect results** when you use the MOD function with a negative numerator or denominator.

PlanMaker also returns **incorrect results** in this case (for compatibility reasons), but it offers an additional function named MODP that returns correct results.

The following table shows the differences between MOD and MODP (the results highlighted in green color are incorrect):



Compatibility notes:

Note: The MOD**P** function is not supported by Microsoft Excel. If you save a document in Excel format, all calculations that use this function will be replaced by their last result as a fixed value.

See also:

MODP, QUOTIENT

MODE (most frequently occurring value)

Note: MODE is supplemented by the new identical function MODE.SNGL, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

MODE(Number1 [, Number2, Number3 ...])

Description:

This returns the mode of a data set.

The mode is the value that occurs most frequently in a set of numbers.

Number1, **Number2**, etc., are the numbers to be evaluated.

If the given numbers contain more than one mode, their smallest mode will be returned.

If the given numbers do not contain any value occurring more than once, a #N/A error value will be returned.

Example:

MODE(1, 2, 4, 4, 4, 2, 1) returns 4

MODE(1, 2, 4, 4, 2, 1) returns 1 (if there are multiple modes, the smallest one is returned)

MODE(1, 4, 2) returns #N/A because no value occurs more than once.

See also:

MODE.SNGL, MEDIAN, NORM.DIST/NORMDIST

MODE.SNGL (most frequently occurring value)

Syntax:

MODE.SNGL(Number1 [, Number2, Number3 ...])

Description:

This returns the mode of a data set.

The mode is the value that occurs most frequently in a set of numbers.

Number1, **Number2**, etc., are the numbers to be evaluated.

If the given numbers contain more than one mode, their smallest mode will be returned.

If the given numbers do not contain any value occurring more than once, a #N/A error value will be returned.

Example:

MODE.SNGL(1, 2, 4, 4, 4, 2, 1) returns 4

MODE.SNGL(1, 2, 4, 4, 2, 1) returns 1 (if there are multiple modes, the smallest one is returned)

MODE.SNGL(1, 4, 2) returns #N/A because no value occurs more than once.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

MODE, MEDIAN, NORM.DIST/NORMDIST

MODP (remainder of a division, PlanMaker method)

Syntax:

MODP(x, y)

Description:

This returns the remainder of the division of **x** by **y** according to the "PlanMaker method" (see additional info below).

Example:

MODP(5, 3) returns 2

MODP(5, 0) returns the #DIV/0! error value

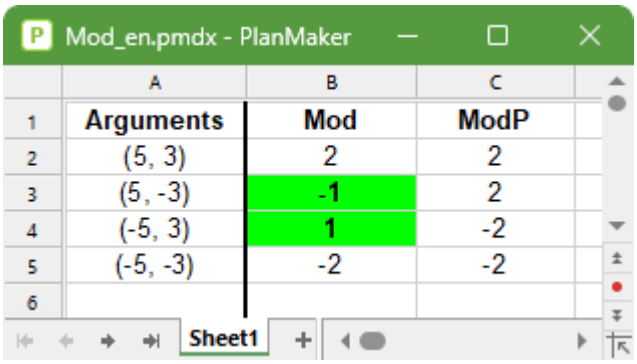
MODP(5, -3) returns 2 (see additional info below)

Additional info:

Microsoft Excel returns incorrect results when you use the MOD function with a negative numerator or denominator.

If you want to obtain correct results, you should use the MOD**P** function described here instead of MOD.

The following table shows the differences between MOD and MODP (the results highlighted in green color are incorrect):



Compatibility notes:

Note: The MOD**P** function is not supported by Microsoft Excel. If you save a document in Excel format, all calculations that use this function will be replaced by their last result as a fixed value.

See also:

MOD, QUOTIENT

MONTH (month of a date)

Syntax:

MONTH(Date)

Description:

This returns the month of a date.

Example:

MONTH("09/25/2018") returns 9

MONTH("August 17") returns 8

If cell A1 contains the date value 9/25/2018 then MONTH(A1) returns 9

See also:

ISOWEEKNUM, ISOWEEK, WEEKNUM, YEAR, DAY, WEEKDAY

MROUND (round to a multiple of base)

Syntax:

MROUND(Number, Base)

Description:

This rounds **Number** to the nearest multiple of **Base**.

Base must be > 0.

If the remainder of the division of **Number** by **Base** is smaller than half the value of **Base**, MROUND rounds down. If it is greater than or equal to half the value of **Base**, MROUND rounds up (see examples below).

Example:

MROUND(1234.5678, 1) returns 1235

MROUND(1234.5678, 0.01) returns 1234.57

MROUND(22, 5) returns 20

MROUND(23, 5) returns 25

MROUND(22.5, 5) returns 25

See also:

FLOOR, CEILING, ROUNDDOWN, ROUNDUP, ROUND, TRUNC, INT

MSOLVE (solution of matrix equation Ax=B)

Syntax:

MSOLVE(ArrayA, VectorB)

Description:

This returns the solution to a linear system of equations by solving the matrix equation Ax=B.

ArrayA is the cell range or array that contains matrix A. It must contain numeric values only and be square, i.e., have the same number of rows and columns.

The determinant of this matrix must not be zero, as the equation cannot be solved otherwise.

VectorB is the cell range or array that contains vector B. **VectorB** must be a vector, which is an array or cell range of just one column.

The result of this function is a solution vector.

Please note: The MSOLVE function only calculates correctly if the vector with the solutions is arranged as a *column*. To arrange the vector with the solutions in a *row*, you must also use the TRANSPOSE function with the following syntax:

TRANSPOSE(MSOLVE(ArrayA, VectorB))

Additional info:

The result of this function is calculated using singular value decomposition. When applied to larger matrices (more than 10-20 rows), significant rounding errors may occur.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Example:

To solve the following linear system of equations:

2x + 3y = 4
2x + 2y = 8

... use the following formula:

MSOLVE({2,3;2,2}, {4,8})

This results in the vector {8;-4}, which means that x=8 and y=-4.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

Section Working with arrays, MDETERM, MINVERSE, MMULT, TRANSPOSE

MULTINOMIAL (multinomial coefficient)

Syntax:

MULTINOMIAL(Number1 [, Number2, Number3 ...])

Description:

This returns the multinomial coefficient ("polynomial coefficient") of a set of numbers.

All given values **Number1**, **Number2**, etc., have to be ≥ 1.

Example:

MULTINOMIAL(1, 2, 3, 4, 5) returns 37837800

Note:

The MULTINOMIAL and POLYNOMIAL functions are completely identical.

See also:

POLYNOMIAL, FACT

N (convert value into number)

Syntax:

N(Value)

Description:

This converts a value into a number.

The specified value must be a number, a date value or a logical value, otherwise N returns zero.

Note:

This function was retained only for compatibility with older spreadsheet applications. It is **no** longer commonly used, as current spreadsheet applications (including PlanMaker) automatically convert values into numbers if necessary.

Example:

N("Text") returns 0

N("42") returns 0

N(42) returns 42

N(TRUE) returns 1

If cell A1 contains the date 09/25/2014, the following applies:

N(A1) returns 41907 (the serial number of that date)

See also:

T, VALUE, TEXT

NA (error value #N/A)

Syntax:

NA()

Description:

This returns the error value #N/A ("not available").

Entering =NA() in a cell indicates that the value in this cell is currently **n**ot **a**vailable. Use this function in a cell that is empty but which must be filled later in order to obtain correct results in calculations performed with this cell. As a result, all calculations that refer to this cell will also return a #N/A error value (instead of an incorrect result).

In this way, you can prevent calculations from being performed with empty cells, although these cells would have to be filled with values for a correct result to be returned.

Additional info:

Instead of entering =NA() in a cell, you can also type #N/A, which will lead to the same result.

Example:

You would like to calculate your annual sales using the sum of the four quarterly results. These are entered in cells A1 to A4; the annual sales could thus be calculated using the formula SUM(A1:A4).

There is only one problem: The result for the fourth quarter is not yet available, so cell A4 is empty. Thus, the annual sales may not yet be calculated.

However, PlanMaker persists in calculating the sum from A1 to A4 – even if one of the values is missing. The result: an apparently miserable annual result and irate shareholders.

It is much better practice to enter NA() in cell A4 rather than leave it empty. This marks the cell as "not available". This means that the SUM formula no longer has an incorrect numerical value as a result, but the error value #N/A. This indicates that one of the quarterly results is not yet available, i.e., the annual sales cannot yet be calculated. Result: *No* irate shareholders.

See also:

ISNA, ISERR, ISERROR, ERRORTYPE, section Error values

NEG (negative value)

Syntax:

NEG(Number)

Description:

This inverts the sign of a **Number**.

Example:

NEG(42) returns -42

NEG(-42) returns 42

NEG(0) returns 0

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Instead of using NEG, you can simply multiply the value by -1: NEG(*x*) equals (-1) * *x*.

See also:

ABS, SIGN

NEGBINOM.DIST (negative binomial distribution)

Syntax:

NEGBINOM.DIST(k, r, p, Cumulative)

Description:

This returns the negative binomial distribution (also known as Pascal distribution).

The negative binomial distribution can be used to calculate the probability of obtaining **k** failures before the **r**-th success in a random experiment with independent trials and only two possible outcomes (success or failure) with a fixed probability of **p** for each trial.

k is the number of failures. It must be ≥ 0.

r is the number of successes. It must be ≥ 0.

k and **r** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

p is the probability of success for each trial run. It must be within the range 0 to 1.

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative distribution function of the negative binomial distribution is calculated. By entering the value FALSE, the probability density function of the negative binomial distribution is calculated.

Example:

An urn contains 1 red ball and 5 black balls. What is the probability (with the balls being returned) of drawing *exactly* or *at most* 10 black balls (failures) before the 2nd red ball (success) is drawn?

NEGBINOM.DIST(10, 2, 1/6, FALSE) returns 0.04935

NEGBINOM.DIST(10, 2, 1/6, TRUE) returns 0.61867

The probability of having to draw *exactly* 10 black balls is 4.9% (probability density function). The probability of having to draw *at most* 10 black balls is 61.9% (cumulative distribution function).

Note:

The NEGBINOM.DIST function supplements the previous NEGBINOMDIST function with the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

NEGBINOMDIST, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST, HYPGEOM.DIST/HYPGEOMDIST

NEGBINOMDIST (negative binomial distribution)

Note: NEGBINOMDIST is supplemented by the new function NEGBINOM.DIST with additional functionality, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

NEGBINOMDIST(k, r, p)

Description:

This returns the negative binomial distribution (also known as Pascal distribution).

The negative binomial distribution can be used to calculate the probability of obtaining **k** failures before the **r**-th success in a random experiment with independent trials and only two possible outcomes (success or failure) with a fixed probability of **p** for each trial.

k is the number of failures. It must be ≥ 0.

r is the number of successes. It must be ≥ 0.

k and **r** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

p is the probability of success for each trial run. It must be within the range 0 to 1.

Example:

An urn contains 1 red ball and 5 black balls. What is the probability (with the balls being returned) of drawing *exactly* 10 black balls (failures) before the 2nd red ball (success) is drawn?

NEGBINOMDIST(10, 2, 1/6) returns 0.04935.

See also:

NEGBINOM.DIST, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST, HYPGEOM.DIST/HYPGEOMDIST

NETWORKDAYS (number of workdays)

Syntax:

NETWORKDAYS(StartDate, EndDate [, Holidays])

Description:

This returns the number of workdays between **StartDate** and **EndDate**.

This function counts workdays only, and Saturdays and Sundays are skipped. You also have the option of specifying a list of holidays to be skipped as well (see Holidays parameter).

StartDate and **EndDate** are the first and the last day of the time interval to be evaluated.

Holidays (optional) is a cell range or an array containing a list of dates to be skipped (e.g., holidays).

Example:

NETWORKDAYS("01/01/2010", "01/31/2010") returns 22.

January 2010 thus had 22 work days (not including public holidays).

If you want to include the holidays on 1/1/2010 and 1/6/2010 (in most European countries), you can enter these two dates in cells A1 and A2, for example, and then write them as follows:

NETWORKDAYS("01/01/2010, "01/31/2010", A1:A2)

The result is 20, as these holidays both fell on weekdays in 2010, so 2 days are to be deducted.

See also:

WORKDAY, DAYS/DAYSP, DAYS360, DATEDIF, HOLIDAY

NOMINAL (nominal interest rate)

Syntax:

NOMINAL(EffRate, NPerYear)

Description:

This returns the nominal annual interest rate based on the effective annual interest rate and the number of compounding periods per year.

EffRate is the effective annual interest rate.

NPerYear is the number of compounding periods per year (must be at least 1).

Example:

An investment with an effective annual interest rate of 5% is compounded quarterly. How high is the nominal annual interest rate?

NOMINAL(5%, 4) returns approx. 4.91%

See also:

EFFECT

NORMDIST (normal distribution)

Note: NORMDIST is supplemented by the new identical function NORM.DIST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

NORMDIST(x, Mean, StdDev, Cumulative)

Description:

This returns the normal distribution (also known as Gaussian distribution).

The normal distribution is one of the most important distributions in statistics, since many random variables are normally distributed or are at least approximately normally distributed.

x is the value to be evaluated.

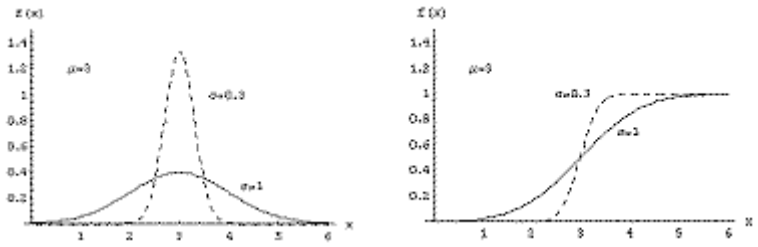
Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x).

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.



Left: density function ("bell curve"), right: distribution function

Additional info:

The reverse function to this function is NORMINV

See also:

NORM.DIST, NORM.INV/NORMINV, NORM.S.DIST/NORMSDIST, NORM.S.INV/NORMSINV

NORM.DIST (normal distribution)

Syntax:

NORM.DIST(x, Mean, StdDev, Cumulative)

Description:

This returns the normal distribution (also known as Gaussian distribution).

The normal distribution is one of the most important distributions in statistics, since many random variables are normally distributed or are at least approximately normally distributed.

x is the value to be evaluated.

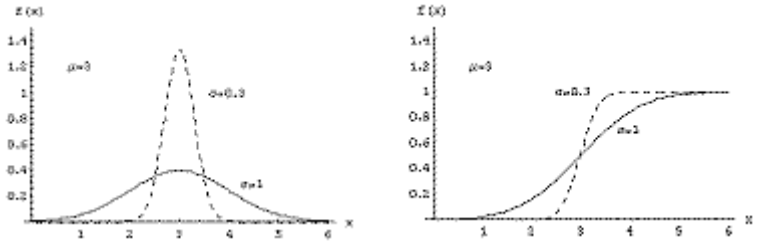
Mean is the mean of LN(x).

StdDev is the standard deviation of LN(x).

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.



Left: density function ("bell curve"), right: distribution function

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

The reverse function to this function is NORM.INV

See also:

NORMDIST, NORM.INV/NORMINV, NORM.S.DIST/NORMSDIST, NORM.S.INV/NORMSINV

NORMINV (percentiles of the normal distribution)

Note: NORMINV is supplemented by the new identical function NORM.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

NORMINV(Probability, Mean, StdDev)

Description:

This returns the inverse of the normal cumulative distribution function NORMDIST.

Probability is the given probability from the distribution. The values must be within the range 0 to 1.

Mean is the arithmetic mean of the distribution.

StdDev is the standard deviation of the distribution. This must be > 0.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

NORM.INV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST, NORM.S.INV/NORMSINV

NORM.INV (percentiles of the normal distribution)

Syntax:

NORM.INV(Probability, Mean, StdDev)

Description:

This returns the inverse of the normal cumulative distribution function NORM.DIST.

Probability is the given probability from the distribution. The values must be within the range 0 to 1.

Mean is the arithmetic mean of the distribution.

StdDev is the standard deviation of the distribution. This must be > 0.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

NORMINV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST, NORM.S.INV/NORMSINV

NORMSDIST (standard normal distribution)

Note: NORMSDIST is supplemented by the new function NORM.S.DIST with additional functionality, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

NORMSDIST(x)

Description:

This returns the standard normal cumulative distribution function.

The standard normal distribution is a normal distribution (see NORM.DIST/NORMDIST function) with a mean of zero and a standard deviation of 1.

x is the value whose probability is to be calculated.

Additional info:

The cumulative distribution function is returned.

The inverse function is NORMSINV

See also:

NORM.S.DIST, NORM.INV/NORMINV, NORM.S.INV/NORMSINV, STANDARDIZE, GAUSS, LOGNORM.INV/LOGINV, LOGNORM.DIST/LOGNORMDIST

NORM.S.DIST (standard normal distribution)

Syntax:

NORM.S.DIST(x, cumulative)

Description:

This returns the standard normal (cumulative) distribution function.

The standard normal distribution is a normal distribution (see NORM.DIST/NORMDIST function) with a mean of zero and a standard deviation of 1.

x is the value to be evaluated.

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative distribution function of the standard normal distribution is calculated. By entering the value FALSE, the probability density function of the standard normal distribution is calculated.

Note:

The NORM.S.DIST function supplements the previous NORMSDIST function with the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

Additional info:

The inverse function is NORM.S.INV

See also:

NORMSDIST, NORM.INV/NORMINV, NORM.S.INV/NORMSINV, STANDARDIZE, GAUSS, LOGNORM.INV/LOGINV, LOGNORM.DIST/LOGNORMDIST

NORMSINV (percentiles of the standard distribution)

Note: NORMSINV is supplemented by the new identical function NORM.S.INV, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

NORMSINV(Probability)

Description:

This returns the inverse of the standard normal cumulative distribution function NORMSDIST.

Probability is the given probability from the distribution. The values must be within the range 0 to 1.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

NORM.S.INV, NORM.INV/NORMINV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

NORM.S.INV (percentiles of the standard distribution)

Syntax:

NORM.S.INV(Probability)

Description:

This returns the inverse of the standard normal cumulative distribution function NORM.S.DIST.

Probability is the given probability from the distribution. The values must be within the range 0 to 1.

Note:

The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

NORMSINV, NORM.INV/NORMINV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

NOT (logical NOT function)

Syntax:

NOT(Value)

Description:

This performs a logical negation. TRUE becomes FALSE and FALSE becomes TRUE.

Value must be a logical value or a calculation that returns a logical value.

Example:

NOT(TRUE) returns FALSE

NOT(FALSE) returns TRUE

NOT(A1=0) returns TRUE if A1 contains a value other than zero.

See also:

OR, AND, XOR

NOW (current date and time)

Syntax:

NOW()

Description:

This returns the current date and time.

The displayed value is dynamic, so it is automatically updated when the document is opened.

Note:

The result is a number – like all dates in a spreadsheet. PlanMaker automatically formats it in time format so that it will be displayed as a time.

However, the result also contains the date. Thus, if you format the cell using the dialog box of the ribbon tab **Home** | group **Number** ◢, for example, in the number format MM/DD/YYYY hh:mm, the current date and time will be displayed.

See also:

TODAY, CREATEDATE

NPER (number of periods)

Syntax:

NPER(Rate, Pmt, PV [, FV] [, Type])

Description:

This returns the number of payment periods of an investment based on periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how many periodic payments (annuities) must be made for a loan.

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

Pmt is the annuity (payment made for each period).

PV is the present value (initial amount)

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

See also:

PV, PPMT, PMT, RATE, IPMT, FV

NPV (net present value)

Syntax:

NPV(Rate, Value1 [, Value2, Value3 ...])

Description:

This returns the net present value of an investment based on regular cash flows and a fixed interest rate.

The function expects the following arguments:

Rate is the interest rate (per payment period).

Value1, **Value2**, etc., are the cash flows. You can specify either single values or a cell range. ***Important:*** All cash flows have to occur at the *end* of each period.

The cash flows may vary in size and consist of incomes (positive values) and payments (negative values). However, they must occur at regular intervals (at the *end* of each period) and be entered in their actual order, namely, cash flow for the first period, cash flow for the second period, etc.

A zero has to be entered for periods where no cash flow occurred.

Differences to the PV function:

1. Cash flows of different amounts are possible for NPV, while they have to be constant amounts for PV.

2. For NPV, cash flows must occur at the end of each period; for PV, they can also be made at the start of the period.

Example:

NPV(6.5%, 4000, 5000) returns 8164.16

See also:

PV, XNPV, IRR

NUMBERVALUE (convert text into a number, locale)

Syntax:

NUMBERVALUE(Text [, DecimalSeparator] [, GroupSeparator])

Description:

This converts a text string into a number. The arguments **DecimalSeparator** and **GroupSeparator** allow you to format the text string in a locale-independent manner.

Text is the value that you want to convert into a number.

DecimalSeparator (optional) specifies which character is currently used to separate the decimal places in the text string.

GroupSeparator (optional) specifies which character is currently used in the text string to form groups of 1000s.

The result is returned as a default format: without a group separator and the zeros after the decimal separator are truncated.

Notes:

The following rules apply when using the NUMBERVALUE function:

- If the arguments **DecimalSeparator** and **GroupSeparator** are not specified, the text string is returned according to the current locale.
- If only the **DecimalSeparator** is specified, the **GroupSeparator** is returned according to the current locale.
- If several characters are specified in **DecimalSeparator** or **GroupSeparator**, only the first one is used.
- If both arguments **DecimalSeparator** and **GroupSeparator** are specified and they are the same, this results in the #VALUE! error value.
- An empty **Text** string returns 0 as the result.
- Space characters within the **Text** string are ignored.
- Multiple decimal separators in the **Text** string result in the #VALUE! error value.
- If the decimal separator appears before the group separator in the **Text** string, this results in the #VALUE! error value.
- If the **Text** string cannot be understood as a number, this returns the #VALUE! error value.
- The % sign at the end of the **Text** string is treated as a percentage (i.e., the number is divided by 100). Multiple % signs cause the number to be divided several times by 100.

Example:

NUMBERVALUE("42") returns 42

NUMBERVALUE("42.44") returns 42.44

NUMBERVALUE("42,44") returns 4244

NUMBERVALUE("42,44",",") returns 42.44

NUMBERVALUE("42,44",",",".") returns 42.44

NUMBERVALUE("42,000.44") returns 42000.44

NUMBERVALUE("42.000,44") returns #VALUE!

NUMBERVALUE("42.000,44",",",".") returns 42000.44

NUMBERVALUE("42,000.44",",",".") returns #VALUE!

NUMBERVALUE("42 0 0 0",",",".") returns 42000

NUMBERVALUE("42%") returns 0.42

NUMBERVALUE("42%%") returns 0.0042

Tip: Number value with unit of measure

If, for example, cell A1 contains the value "42,44 mm" and you want to obtain the value "42.44 mm" in cell B1 using the NUMBERVALUE function, this is not possible because the function expects a string that is recognizable as a number. In this case, proceed as follows:

1. Enter the value "42,44" (without mm) in cell A1, and use the formula NUMBERVALUE(A1,",",".") for cell B1 for converting according to the examples above.
2. Then format cell B1 with the custom number format @" mm" (For more information, see also Working with custom number formats).
3. You will get the value "42.44 mm" in cell B1 as a result.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

VALUE

OCT2BIN (octal number to binary number)

Syntax:

OCT2BIN(Number [, Digits])

Description:

This converts the given octal number (base 8) to a binary number (base 2).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in octal notation):

20000000000 ≤ *Number* ≤ 17777777777

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

OCT2BIN("52") returns 101010

OCT2BIN("52", 8) returns 00101010

OCT2BIN("52", 2) returns the #NUM! error value because this number cannot be represented with fewer than 6 binary digits

OCT2BIN("37777777726") returns 11111111111111111111111111010110 (negative value)

See also:

BIN2OCT, OCT2DEC, OCT2HEX

OCT2DEC (octal number to decimal number)

Syntax:

OCT2DEC(Number)

Description:

This converts the given octal number (base 8) to a decimal number (base 10).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in octal notation):

20000000000 ≤ *Number* ≤ 17777777777

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

Example:

OCT2DEC("52") returns 42

OCT2DEC("0052") also returns 42

OCT2DEC("37777777726") returns -42

See also:

DEC2OCT, OCT2BIN, OCT2HEX, DECIMAL

OCT2HEX (octal number to hexadecimal number)

Syntax:

OCT2HEX(Number [, Digits])

Description:

This converts the given octal number (base 8) to a hexadecimal number (base 16).

Number is the number to be converted.

Negative numbers must be represented in the 32-bit two's complement (see last example).

The permissible value range for **Number** is (in octal notation):

20000000000 ≤ *Number* ≤ 17777777777

This corresponds in decimal notation to:

-2,147,483,648 ≤ *Number* ≤ 2,147,483,647

The optional argument **Digits** lets you specify the number of places to be displayed. If omitted, PlanMaker automatically determines the required number of places.

Digits must be greater than zero. **Digits** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **Digits** are less than the minimum number of places required to display the number, the function returns a #NUM error value.

If **Number** is negative, the result is represented in 32-bit two's complement; the **Digits** argument is ignored in this case.

Example:

OCT2HEX("52") returns 2A

OCT2HEX("0052") also returns 2A

OCT2HEX("52", 1) returns the #NUM! error value because this number cannot be represented with fewer than 2 hexadecimal digits

OCT2HEX("37777777726") returns FFFFFFD6 (negative value)

See also:

HEX2OCT, OCT2BIN, OCT2DEC

ODD (round up to next odd number)

Syntax:

ODD(Number)

Description:

This rounds **Number** up (away from zero) to the nearest odd integer.

Example:

ODD(41) returns 41

ODD(42) returns 43

ODD(-42) returns -43

See also:

EVEN, ROUNDDOWN, ROUNDUP, ROUND, MROUND

OFFSET (shifted reference)

Syntax:

OFFSET(Reference, Rows, Columns [, Height] [, Width])

Description:

This returns a cell reference shifted by the specified number of rows and columns from the original cell reference. There is also the option of changing the size of the **Reference**.

Note: This does not physically move worksheet cells nor move the selection. The function only returns a shifted cell reference.

Reference is the original cell reference. You can specify either a single cell or a cell range (in the notation *Start cell:End cell*).

For **Rows**, specify the number of rows by which **Reference** is to be shifted. Positive values shift downwards, while negative values shift upwards.

For **Columns**, specify the number of columns by which **Reference** is to be shifted. Positive values shift to the right, while negative values shift to the left.

Height (optional) changes the height of **Reference** to the specified number of rows. **Height** must be a positive value.

Width (optional) changes the width of **Reference** to the specified number of columns. **Width** must be a positive value.

Example:

OFFSET(A1, 2, 0) returns a reference to A3

OFFSET(A1:C3, 1, 1) returns a reference to B2:D4

OFFSET(A1:C3, 1, 1, 5, 3) returns a reference to B2:D6

SUM(OFFSET(A1:C3, 1, 1, 5, 3)) returns the sum of the values in B2:D6

See also:

ADDRESS, INDIRECT, COLUMN, ROW

OR (logical OR function)

Syntax:

OR(Value1 [, Value2, Value3 ...])

Description:

This returns the logical value TRUE if at least one of the given arguments is TRUE, otherwise it returns FALSE.

You can use this function to verify that *at least one* of multiple conditions is fulfilled.

Example:

OR(TRUE, TRUE) returns TRUE

OR(TRUE, FALSE) returns TRUE

OR(FALSE, FALSE) returns FALSE

OR(A1=0, A1>10) returns TRUE if A1 is either zero or greater than ten.

See also:

XOR, AND, NOT

PEARSON (Pearson correlation coefficient)

Syntax:

PEARSON(Range1, Range2)

Description:

This returns the Pearson correlation coefficient *r* of two data sets.

This coefficient is a measure of the strength of the linear relationship between two data sets.

The result of the PEARSON function is within the range -1 to 1: 1 indicates a perfect linear relationship. -1 indicates a perfect inverse linear relationship (the values in **Range2** decrease as the values in **Range1** increase).

Range1 and **Range2** are cell references to the two data sets. The ranges should contain numbers. Empty cells, text and logical values are ignored.

Range1 and **Range2** must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 1, 2, 3 and B1:B3 contains 2, 3, 4, the following applies:

PEARSON(A1:A3, B1:B3) returns 1

If A1:A3 contains the values 1, 2, 3 and B1:B3 contains 4, 3, 2, the following applies:

PEARSON(A1:A3, B1:B3) returns -1

See also:

RSQ, SLOPE, INTERCEPT

PERCENTILE (percentiles of a data set)

Note: PERCENTILE is supplemented by the new identical function PERCENTILE.INC and the completely new function PERCENTILE.EXC, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

PERCENTILE(Range, k)

Description:

This returns the **k**-th percentile of a data set.

For example, if **k** is 0.5, the 0.5 percentile (i.e., the median) will be returned.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

k is the desired percentage. This must be within the range from 0 to 1 **(inclusive)** with 0=0% and 1=100%.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTILE(A1:A9, 0.75) returns 7

See also:

PERCENTILE.INC/PERCENTILE.EXC, QUARTILE.EXC/QUARTILE.INC/QUARTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE, RANK.EQ/RANK.AVG/RANK

PERCENTILE.EXC (percentiles of a data set)

Syntax:

PERCENTILE.EXC(Range, k)

Description:

This returns the **k**-th percentile of a data set.

For example, if **k** is 0.5, the 0.5 percentile (i.e., the median) will be returned.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

k is the desired percentage. This must be within the range from 0 to 1 **(exclusive)** with 0=0% and 1=100%.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTILE.EXC(A1:A9, 0.75) returns 7.5

Note:

The PERCENTILE.EXC and PERCENTILE.INC functions supplement the previous PERCENTILE function.

PERCENTILE.INC returns the same result as PERCENTILE (values 0 to 1 **inclusive**).

PERCENTILE.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

QUARTILE.EXC/QUARTILE.INC/QUARTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE, RANK.EQ/RANK.AVG/RANK

PERCENTILE.INC (percentiles of a data set)

Syntax:

PERCENTILE.INC(Range, k)

Description:

This returns the **k**-th percentile of a data set.

For example, if **k** is 0.5, the 0.5 percentile (i.e., the median) will be returned.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

k is the desired percentage. This must be within the range from 0 to 1 **(inclusive)** with 0=0% and 1=100%.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTILE.INC(A1:A9, 0.75) returns 7

Note:

The PERCENTILE.INC and PERCENTILE.EXC functions supplement the previous PERCENTILE function.

PERCENTILE.INC returns the same result as PERCENTILE (values 0 to 1 **inclusive**).

PERCENTILE.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

QUARTILE.EXC/QUARTILE.INC/QUARTILE,
PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE,
RANK.EQ/RANK.AVG/RANK

PERCENTRANK (percent rank in a data set)

Note: PERCENTRANK is supplemented by the new identical function PERCENTRANK.INC and the completely new function PERCENTRANK.EXC, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

PERCENTRANK(Range, Number [, Digits])

Description:

This returns the percentile rank (between 0 and 1 **inclusive**) of a value in a data set.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

Number is the value whose rank is to be determined. If **Number** is not included in the given values, interpolation will be used.

The optional argument **Digits** lets you specify to how many decimal places the result should be calculated. For this purpose, enter the desired number of decimal places. If omitted, 3 digits will be used.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTRANK(A1:A9, 5) returns 0.5 (50%) middle of the given values; median

PERCENTRANK(A1:A9, 1) returns 0 (0%) minimum value

PERCENTRANK(A1:A9, 9) returns 1 (100%) maximum value

See also:

PERCENTRANK.INC/PERCENTRANK.EXC, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, QUARTILE.EXC/QUARTILE.INC/QUARTILE, RANK.EQ/RANK.AVG/RANK

PERCENTRANK.EXC (percent rank in a data set)

Syntax:

PERCENTRANK.EXC(Range, Number [, Digits])

Description:

This returns the percentile rank (between 0 and 1 **exclusive**) of a value in a data set.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

Number is the value whose rank is to be determined. If **Number** is not included in the given values, interpolation will be used.

The optional argument **Digits** lets you specify to how many decimal places the result should be calculated. For this purpose, enter the desired number of decimal places. If omitted, 3 digits will be used.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTRANK.EXC(A1:A9, 5) returns 0.5	(50%) middle of the given values; median
PERCENTRANK.EXC(A1:A9, 1) returns 0.1	(10%) minimum value
PERCENTRANK.EXC(A1:A9, 9) returns 0.9	(90%) maximum value

Note:

The PERCENTRANK.EXC and PERCENTRANK.INC functions supplement the previous PERCENTRANK function.

PERCENTRANK.INC returns the same result as PERCENTRANK (values 0 to 1 **inclusive**).

PERCENTRANK.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, QUARTILE.EXC/QUARTILE.INC/QUARTILE, RANK.EQ/RANK.AVG/RANK

PERCENTRANK.INC (percent rank in a data set)

Syntax:

PERCENTRANK.INC(Range, Number [, Digits])

Description:

This returns the percentile rank (between 0 and 1 **inclusive**) of a value in a data set.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

Number is the value whose rank is to be determined. If **Number** is not included in the given values, interpolation will be used.

The optional argument **Digits** lets you specify to how many decimal places the result should be calculated. For this purpose, enter the desired number of decimal places. If omitted, 3 digits will be used.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

PERCENTRANK.INC(A1:A9, 5) returns 0.5	(50%) middle of the given values; median
PERCENTRANK.INC(A1:A9, 1) returns 0	(0%) minimum value
PERCENTRANK.INC(A1:A9, 9) returns 1	(100%) maximum value

Note:

The PERCENTRANK.INC and PERCENTRANK.EXC functions supplement the previous PERCENTRANK function.

PERCENTRANK.INC returns the same result as PERCENTRANK (values 0 to 1 **inclusive**).

PERCENTRANK.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, QUARTILE.EXC/QUARTILE.INC/QUARTILE, RANK.EQ/RANK.AVG/RANK

PERIOD (duration of fixed-interest investments)

Syntax:

PERIOD(Rate, PV, FV)

Description:

This returns the duration (i.e., the number of periods) required to obtain a specific future value with a fixed-interest investment.

This function is applicable to investments with just one payment (made at the beginning) and a fixed interest rate.

The function expects the following arguments:

Rate is the interest rate (per payment period)

PV is the present value (initial amount)

FV is the future value

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Tip: You can perform calculations with the PERIOD function using Excel-compatible functions as follows:

PERIOD(Rate, PV, FV) is equivalent to NPER(Rate, 0, PV, -FV)

Note the minus sign in front of FV!

Example:

You would like to obtain a payout of €5000 with a fixed-interest investment of €3000. The annual interest rate is 6%. How many periods (years) are required in this case?

PERIOD(6%, 3000, 5000) returns 8.76669 periods (i.e., years)

See also:

RRI

PERMUT (permutations)

Syntax:

PERMUT(n, k)

Description:

This returns the number of permutations of size **k** in a population of size **n**.

n is the total number of items. **n** must be > **k**.

k is the number of items in each combination. **k** must be > 0 and < **n**.

n and **k** should be integers. If this is not the case, PlanMaker automatically truncates the decimal places.

Additional info:

Permutations are similar to combinations (see COMBIN function), except that the *order* of the objects is important for permutations.

With PERMUT, for example, you can calculate how many possibilities there are for the first 3 places in a race with 10 participants. With COMBIN, on the other hand, you can determine how many possibilities there are when drawing 6 balls from 49.

The difference: While the order of the lottery numbers drawn can be changed at will (you could, for example, sort them in ascending order afterwards), the runners would naturally protest if you suddenly rearranged 1st-3rd places according to the first letters of their surnames.

The *order* plays a role in the permutations but not in the combinations.

Example:

Let's take a closer look at the example with the runners. You want to know how many permutations there are for the first 3 places in a race with 10 runners. The formula for this is as follows:

PERMUT(10, 3)

Result: There are 720 possible variations.

See also:

COMBIN

PHI (standard normal distribution)

Syntax:

PHI(x)

Description:

This returns the density function of the standard normal distribution.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

We thus recommend that you use the NORMDIST function instead, as it is compatible with all versions of Excel:

PHI(x) equals NORMDIST(x, 0, 1, FALSE)

See also:

NORM.DIST/NORMDIST, GAUSS

PI (constant value of pi)

Syntax:

PI()

Description:

This returns the mathematical constant π (pi), which expresses the ratio of a circle's circumference (2rπ) to its diameter (2r).

Example:

PI() returns 3.14159265...

PMT (payment)

Syntax:

PMT(Rate, NPer, PV [, FV] [, Type])

Description:

This returns the payment (principal plus interest) in the given period, applicable to loans with periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how large the (constant) payments are per period. These payments ("annuities") always consist of a principal component (see PPMT function) and an interest component (see IPMT function).

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

NPer is the total number of payment periods.

PV is the present value (initial amount)

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

PMT(6.5%, 10, 1000) returns -139.10

See also:

PV, ISPMT, PPMT, CUMPRINC, CUMIPMT, RATE, IPMT, FV, NPER

POISSON (Poisson distribution)

Note: POISSON is supplemented by the new identical function POISSON.DIST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

POISSON(x, Lambda, Cumulative)

Description:

This returns the probabilities of a Poisson distributed random variable.

The Poisson distribution describes the occurrence of independent, similar events within a given period (for example, the arrival of customers at a counter or the occurrence of incoming phone calls).

It is particularly suitable for probability distributions where there are a large number of results from a sample, and there is a low probability that the event in question will occur. Here, the Poisson distribution can simulate the binomial distribution (with **Lambda** = *n***p*). In contrast to the binomial distribution, however, Poisson requires only one parameter (apart from **x**): the expected value **Lambda** (see example).

x is the value whose probability is to be calculated. **x** must be ≥ 0. Furthermore, **x** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

Lambda (λ) is the expected value (mean value) for **x**. **Lambda** must be ≥ 0.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Example:

In a multi-story building, an average of 4 light bulbs become defective per week.

To determine the probability that just 3 light bulbs become defective in one week:

If you wanted to calculate this with the binomial distribution (function BINOM.DIST/BINOMDIST), you would have to write the following:

BINOMDIST(3, 100000, 4/100000, FALSE) returns 0.195368768

If you use the Poisson distribution, however, you can do so without the parameters **n** and **p** and simply write the following:

POISSON(3, 4, FALSE) returns 0,195366815

As expected, the two results are very similar (as described above).

Another example: What is the probability of a *maximum of* 3 light bulbs becoming defective?

Instead of the density function, you use the distribution function (specify the **Cumulative** argument with TRUE):

POISSON(3, 4, TRUE) returns 0.43347

See also:

POISSON.DIST, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST, EXPON.DIST/EXPONDIST

POISSON.DIST (Poisson distribution)

Syntax:

POISSON.DIST(x, Lambda, Cumulative)

Description:

This returns the probabilities of a Poisson distributed random variable.

The Poisson distribution describes the occurrence of independent, similar events within a given period (for example, the arrival of customers at a counter or the occurrence of incoming phone calls).

It is particularly suitable for probability distributions where there are a large number of results from a sample, and there is a low probability that the event in question will occur. Here, the Poisson distribution can simulate the binomial distribution (with **Lambda** = *n***p*). In contrast to the binomial distribution, however, Poisson requires only one parameter (apart from **x**): the expected value **Lambda** (see example).

x is the value whose probability is to be calculated. **x** must be ≥ 0. Furthermore, **x** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

Lambda (λ) is the expected value (mean value) for **x**. **Lambda** must be ≥ 0.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Example:

In a multi-story building, an average of 4 light bulbs become defective per week.

To determine the probability that just 3 light bulbs become defective in one week:

If you wanted to calculate this with the binomial distribution (function BINOM.DIST/BINOMDIST), you would have to write the following:

BINOM.DIST(3, 100000, 4/100000, FALSE) returns 0.195368768

If you use the Poisson distribution, however, you can do so without the parameters **n** and **p** and simply write the following:

POISSON.DIST(3, 4, FALSE) returns 0,195366815

As expected, the two results are very similar (as described above).

Another example: What is the probability of a *maximum of* 3 light bulbs becoming defective?

Instead of the density function, you use the distribution function (specify the **Cumulative** argument with TRUE):

POISSON.DIST(3, 4, TRUE) returns 0.43347

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

POISSON, BINOM.DIST.RANGE/BINOM.DIST/BINOMDIST, EXPON.DIST/EXPONDIST

POLYNOMIAL (polynomial coefficient)

Syntax:

POLYNOMIAL(Number1 [, Number2, Number3 ...])

Description:

This returns the polynomial coefficient ("multinomial coefficient") of a set of numbers.

All given values **Number1**, **Number2**, etc., have to be ≥ 1.

Example:

POLYNOMIAL(1, 2, 3, 4, 5) returns 37837800

Note:

The POLYNOMIAL and MULTINOMIAL functions are completely identical.

See also:

MULTINOMIAL, FACT

POWER (power)

Syntax:

POWER(x, y)

Description:

This returns x raised to the power of y.

x is the base.

y is the exponent.

Additional info:

The ^ operator does exactly the same:

POWER(x, y) is equivalent to x^y.

Example:

POWER(2, 4) is equivalent to 2^4, which is 16

See also:

LOG, LN, EXP, SERIESSUM, SQRT, ROOTN, section Operators in formulas

PPMT (payment on the principal)

Syntax:

PPMT(Rate, Per, NPer, PV [, FV] [, Type])

Description:

This returns the principal portion in the given period of an investment based on periodic constant payments and a fixed interest rate.

You can use this function, for example, to calculate how large the principal portion of the annuity is in a particular period for a loan.

"Period" means a payment period. For example, for a loan with monthly repayments, the third period is the third month.

The function expects the following arguments:

Rate is the interest rate (per payment period)

Per is the payment period to be considered.

NPer is the total number of payment periods.

PV is the present value (initial amount)

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

A loan of $100,000 is taken out under the following terms:

Annual interest rate: 10%

Payments: monthly

Duration: 6 years

What is the principal portion in the 32nd period (i.e., the 32nd month)?

PPMT(10%/12, 32, 6*12, 100000) returns -1.318.

Note that all values have to use the same time unit, *months* in this case, since the payments are made monthly.

Thus, in the above formula, the annual interest rate was divided by 12 to get the *monthly* interest rate and the number of payment periods was multiplied by 12.

Additional info:

The IPMT function is the counterpart to PPMT: It returns the interest component of the payment in this period.

The PMT function, on the other hand, returns the total amount of the payment (annuity) per period, that is, the sum of the principal portion and the interest portion.

See also:

PV, ISPMT, CUMPRINC, CUMIPMT, PMT, RATE, IPMT, FV, NPER

PROB (probability)

Syntax:

PROB(Values, Probabilities, k1 [, k2])

Description:

This returns the probability that a value is within the specified interval of values when the individual probabilities for each value are given.

In simple terms: This function sums up the respective **Probabilities** of the **Values** in this interval.

Values are the given values.

Probabilities are the probabilities associated with these values.

All **Probabilities** have to be within the range 0 (0%) to 1 (100%). Their sum has to be 1.

Values and **Probabilities** must have the same number of values, otherwise the function returns a #N/A error value.

k1 and **k2** determine the interval of values whose cumulative probabilities you want to calculate. If **k2** is omitted, only the probability of **k1** will be returned.

Example:

Cells A1:A3 contain the values 1, 2, 3. Cells B1:B3 contain the probabilities associated with these values: 17%, 33%, 50%.

To calculate the probability that a value is between 2 and 3:

PROB(A1:A3, B1:B3, 2, 3) returns 33% + 50% = 83%

PRODUCT (product)

Syntax:

PRODUCT(Number1 [, Number2, Number3 ...])

Description:

This returns the product of the specified numbers. **Number1**, **Number2**, etc., are the numbers to be multiplied.

Additional info:

When you use this function with cell references or arrays, text strings and logical values are ignored. However, the situation is different if you specify *fixed* values: Here, text strings containing numbers are counted. Furthermore, the logical value TRUE is treated as 1 and FALSE as 0. See examples below.

Example:

PRODUCT(2, 3, 4) equals 2*3*4, which is 24

When cells A1 to A3 are filled with the values 2, 3 and 4, the following applies:

PRODUCT(A1:A3) equals 2*3*4, which is 24

Note:

PRODUCT(2, "5", TRUE) equals 2*5*1, thus 10

However: If cells A1 to A3 are filled with 2, "5" and TRUE, the formula PRODUCT(A1:A3) returns a different result: It returns 2, as strings and logical values are only considered if they are specified as *fixed* values.

See also:

SUM

PROPER (convert text to upper/lower case)

Syntax:

PROPER(Text)

Description:

This converts all characters in a text string as follows: The first letter of each word is converted into an uppercase letter, all other letters are converted into lowercase letters.

Example:

PROPER("good morning") returns Good Morning

PROPER("Good Morning") returns Good Morning

PROPER("GoOd MoRnInG") returns Good Morning

See also:

UPPER, LOWER

PV (present value)

Syntax:

PV(Rate, NPer, Pmt [, FV] [, Type])

Description:

This returns the present value of an investment based on periodic constant payments and a fixed interest rate.

The function expects the following arguments:

Rate is the interest rate (per payment period)

NPer is the total number of payment periods.

Pmt is the annuity (payment made for each period).

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Example:

PV(6.5%/12, 12*10, 650) equals -57244.52

This formula calculates the present value of a loan with an interest rate of 6.5% (per year, thus the division by 12), a term of 10 years (=12*10 months) and an annuity of €650 (per month).

See also:

PPMT, NPV, PMT, RATE, IPMT, FV, NPER

QUARTILE (quartiles of a data set)

Note: QUARTILE is supplemented by the new identical function QUARTILE.INC and the completely new function QUARTILE.EXC, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

QUARTILE(Range, n)

Description:

This returns one of the quartiles of a data set, based on a percentile range from 0 to 1 **(inclusive)**.

Range is a range with numeric data – usually a reference to a cell range in which numbers to be evaluated are entered.

n determines which quartile is to be returned. The following values are permissible:

0: The minimum

1: The 1st quartile (25th percentile)

2: The 2nd quartile (50th percentile or median)

3: The 3rd quartile (75th percentile)

4: The maximum

If n is < 0 or > 4, the function will return a #NUM! error value.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

QUARTILE(A1:A9, 3) is equivalent to PERCENTILE(A1:A9, 0.75) and returns 7

See also:

QUARTILE.INC/QUARTILE.EXC, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE

QUARTILE.EXC (quartiles of a data set)

Syntax:

QUARTILE.EXC(Range, n)

Description:

This returns one of the quartiles of a data set, based on a percentile range from 0 to 1 **(exclusive)**.

Range is a range with numeric data – usually a reference to a cell range in which numbers to be evaluated are entered.

n determines which quartile is to be returned. The following values are permissible:

0: The minimum

1: The 1st quartile (25th percentile)

2: The 2nd quartile (50th percentile or median)

3: The 3rd quartile (75th percentile)

4: The maximum

If n is < 0 or > 4, the function will return a #NUM! error value.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

QUARTILE.EXC(A1:A9, 3) is equivalent to PERCENTILE.EXC(A1:A9, 0.75) and returns 7.5

Note:

The QUARTILE.EXC and QUARTILE.INC functions supplement the previous QUARTILE function.

QUARTILE.INC returns the same result as QUARTILE (values 0 to 1 **inclusive**).

QUARTILE.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE

QUARTILE.INC (quartiles of a data set)

Syntax:

QUARTILE.INC(Range, n)

Description:

This returns one of the quartiles of a data set, based on a percentile range from 0 to 1 **(inclusive)**.

Range is a range with numeric data – usually a reference to a cell range in which numbers to be evaluated are entered.

n determines which quartile is to be returned. The following values are permissible:

0: The minimum

1: The 1st quartile (25th percentile)

2: The 2nd quartile (50th percentile or median)

3: The 3rd quartile (75th percentile)

4: The maximum

If n is < 0 or > 4, the function will return a #NUM! error value.

Example:

If the cells A1:A9 contain the values 1, 2, 3, 4, 5, 6, 7, 8, 9, the following applies:

QUARTILE.INC(A1:A9, 3) is equivalent to PERCENTILE.INC(A1:A9, 0.75) and returns 7

Note:

The QUARTILE.INC and QUARTILE.EXC functions supplement the previous QUARTILE function.

QUARTILE.INC returns the same result as QUARTILE (values 0 to 1 **inclusive**).

QUARTILE.EXC (values 0 to 1 **exclusive**) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, MEDIAN, MODE.SNGL/MODE

QUOTIENT (quotient of a division)

Syntax:

QUOTIENT(x, y)

Description:

This returns the integer part of the division of **x** by **y**.

Example:

QUOTIENT(6, 4) returns 1

See also:

MOD, MODP

RADIANS (convert degrees to radians)

Syntax:

RADIANS(Number)

Description:

This converts degrees to radians.

The following conversion formula is used:

RADIANS(x) = x * PI() / 180

Example:

RADIANS(180) returns 3.14159... (the value of the number Pi)

See also:

DEGREES

RAND (random value)

Syntax:

RAND()

Description:

This returns a random number ≥ 0 and < 1.

A new random number will be generated each time the document is recalculated.

Tip: By choosing the ribbon command **Formula** | group **Update** | **Update data** | **Update calculations**, you can manually recalculate a document.

Example:

INT() returns a random number.

The following formula returns a random integer between 1 and 6, thus simulating the roll of a dice.

INT(RAND()*6+1)

Tip: However, this is easier with RANDBETWEEN(16, 6).

See also:

RANDBETWEEN

RANDBETWEEN (random value)

Syntax:

RANDBETWEEN(LowerLimit, UpperLimit)

Description:

This returns a random integer x in the range **LowerLimit** $\leq x \leq$ **UpperLimit**.

A new random number will be generated each time the document is recalculated.

Tip: By choosing the ribbon command **Formula** | group **Update** | **Update data** | **Update calculations**, you can manually recalculate a document.

Example:

RANDBETWEEN(1, 6) returns a random integer between 1 and 6, thus simulating the roll of a dice.

See also:

RAND

RANK (rank in a data set)

Note: RANK is supplemented by the new identical function RANK.EQ and the completely new function RANK.AVG, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

RANK(Number, Range [, Ascending])

Description:

This returns the rank of a number in a data set. The largest number's rank is 1, the 2nd largest number's rank is 2, etc. If there are duplicate values in the list, they are given the **higher rank**.

Number is the number whose rank is to be determined. If **Number** does not occur in **Range**, the function returns a #N/A error value.

Range is a range with numeric data – usually a reference to a cell range in which the given numbers are entered.

Ascending (optional) is a logical value that lets you specify whether to use ascending or descending order:

FALSE or omitted: Use *descending* order (the *largest* number's rank is 1).

TRUE: Use *ascending* order (the *smallest* number's rank is 1).

Example:

If the cells A1:A6 contain the values 3, 2, 7, 5, 9, 7, the following applies:

RANK(9, A1:A6) returns 1

RANK(9, A1:A6, TRUE) returns 6

RANK(5, A1:A6) returns 4

RANK(7, A1:A6) returns 2

See also:

RANK.EQ/RANK.AVG, SMALL, LARGE, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK

RANK.AVG (rank in a data set)

Syntax:

RANK.AVG(Number, Range [, Ascending])

Description:

This returns the rank of a number in a data set. The largest number's rank is 1, the 2nd largest number's rank is 2, etc. If there are duplicate values in the list, they are given the **average rank**.

Number is the number whose rank is to be determined. If **Number** does not occur in **Range**, the function returns a #N/A error value.

Range is a range with numeric data – usually a reference to a cell range in which the given numbers are entered.

Ascending (optional) is a logical value that lets you specify whether to use ascending or descending order:

FALSE or omitted: Use *descending* order (the *largest* number's rank is 1).

TRUE: Use *ascending* order (the *smallest* number's rank is 1).

Example:

If the cells A1:A6 contain the values 3, 2, 7, 5, 9, 7, the following applies:

RANK.AVG(9, A1:A6) returns 1

RANK.AVG(9, A1:A6, TRUE) returns 6

RANK.AVG(5, A1:A6) returns 4

RANK.AVG(7, A1:A6) returns 2.5

Note:

The RANK.AVG and RANK.EQ functions supplement the previous RANK function.

RANK.EQ returns the same result as RANK (**higher rank** for duplicate values in the list).

The RANK.AVG (**average rank** for duplicate values in the list) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

RANK/RANK.EQ, SMALL, LARGE, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK

RANK.EQ (rank in a data set)

Syntax:

RANK.EQ(Number, Range [, Ascending])

Description:

This returns the rank of a number in a data set. The largest number's rank is 1, the 2nd largest number's rank is 2, etc. If there are duplicate values in the list, they are given the **higher rank**.

Number is the number whose rank is to be determined. If **Number** does not occur in **Range**, the function returns a #N/A error value.

Range is a range with numeric data – usually a reference to a cell range in which the given numbers are entered.

Ascending (optional) is a logical value that lets you specify whether to use ascending or descending order:

FALSE or omitted: Use *descending* order (the *largest* number's rank is 1).

TRUE: Use *ascending* order (the *smallest* number's rank is 1).

Example:

If the cells A1:A6 contain the values 3, 2, 7, 5, 9, 7, the following applies:

RANK.EQ(9, A1:A6) returns 1

RANK.EQ(9, A1:A6, TRUE) returns 6

RANK.EQ(5, A1:A6) returns 4

RANK.EQ(7, A1:A6) returns 2

Note:

The RANK.EQ and RANK.AVG functions supplement the previous RANK function.

RANK.EQ returns the same result as RANK (**higher rank** for duplicate values in the list).

The RANK.AVG (**average rank** for duplicate values in the list) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

RANK/RANK.AVG, SMALL, LARGE, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK

RATE (interest rate per period)

Syntax:

RATE(NPer, Pmt, PV [, FV] [, Type] [, Guess])

Description:

This returns an estimate for the interest portion of the payment per period, based on an investment with periodic constant payments and a fixed interest rate.

The function expects the following arguments:

NPer is the total number of payment periods.

Pmt is the annuity (payment made for each period).

PV is the present value (initial amount)

FV (optional) is the future value (the total value after the last payment). If the argument **FV** is omitted, it will be set to zero.

Type (optional) is the timing of the payments:

0 or omitted: Payment at the end of each period.

1: Payment at the beginning of each period.

Guess (optional) lets you specify an estimate of how high the interest rate is likely to be (see note below).

Note:

The result of this function is calculated using an iterative search technique. If the function does not find a result with an accuracy of 0.00001% or better after 20 iterations, it aborts and the #NUM! error value is returned. If this occurs, try specifying a different value for **Guess**.

Example:

RATE(12*10, -650, 60000) returns 0.455087%

This formula calculates the interest rate for a loan of €60,000 with a term of 10 years (12*10 months) and a payment of €650.00 (per month) using an iterative search technique.

The result is the interest rate *per month*. To obtain the annual interest rate, it must thus be multiplied by 12.

See also:

PV, PPMT, PMT, XIRR, IPMT, RRI, FV, NPER, IRR, MIRR

REPLACE (replace text in a text string)

Syntax:

REPLACE(Text, Start, Count, NewText)

Description:

This replaces a specific part of a text string with other text.

Text is the original text string.

Start is the position of the first character to be replaced.

Count is the number of characters to be replaced.

NewText is the replacement text.

Example:

REPLACE("aaaaaa", 3, 2, "bb") returns aabbaa

REPLACE("aaaaaa", 3, 4, "bb") returns aabb

REPLACE("cccccc", 3, 2, "dddddddd") returns ccddcc

REPLACE("cccccc", 3, 4, "dddddddd") returns ccdddd

REPLACE("cccccc", 3, 6, "dddddddd") returns ccdddddd

See also:

FIND, SEARCH, MID, SUBSTITUTE, LEFT, RIGHT, REPT

REPT (repeat text string)

Syntax:

REPT(Text, n)

Description:

This repeats the specified text string **n** times.

You use the argument **n** to specify the number of repetitions. **n** should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

Example:

REPT("Text", 3) returns TextTextText

See also:

REPLACE, SUBSTITUTE

RIGHT (right part of a text string)

Syntax:

RIGHT(TextString [, n])

Description:

This returns the last **n** characters of a **TextString**.

If **n** is omitted, the function only returns the last character.

Example:

RIGHT("peanut", 3) returns nut

RIGHT("peanut") returns t

See also:

LEFT, MID, REPLACE, LEN

ROMAN (Roman numeral)

Syntax:

ROMAN(Number [, Format])

Description:

This converts an Arabic numeral into a Roman numeral (as a text string).

Number must be ≥ 0 and < 4000.

Number should be an integer. If this is not the case, PlanMaker automatically removes all decimal places and converts the remaining number into a Roman numeral.

If **Number** equals zero, an empty text string will be returned.

If the optional argument **Format** is used, PlanMaker tries to create a shorter form of the Roman numeral. **Format** can be any integer from 0 (do not shorten) to 4 (shorten as much as possible).

Example:

ROMAN(1999) returns MCMXCIX

ROMAN(1999, 0) also returns MCMXCIX

ROMAN(1999, 1) returns MLMIL

ROMAN(1999, 2) returns MXMIX

ROMAN(1999, 3) returns MVMIV

ROMAN(1999, 4) returns MIM

ROOTN (n-th root)

Syntax:

ROOTN(Number [, n])

Description:

This returns the **n**-th root of a number. If **n** is omitted, the function returns the square root (n=2).

n should be an integer. If this is not the case, PlanMaker automatically truncates the decimal places.

If **n** is even, **Number** must be ≥ 0.

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Tip: Alternatively, you can calculate the n-th root of a number using the following Excel-compatible function:

ROOTN(Number, n) equals POWER(Number, 1/n)

Example:

ROOTN(25) returns 5

ROOTN(-25) returns the #NUM! error value

ROOTN(125, 3) returns 5

ROOTN(-125, 3) returns -5

See also:

SQRT, POWER

ROUND (round to n digits)

Syntax:

ROUND(Number, n)

Description:

This rounds **Number** to **n** decimal places.

n is the desired number of decimal places.

If **n** is omitted, **Number** is rounded to the nearest integer.

If **n** is a negative value, **Number** is rounded to the left of the decimal point. For example, if **n** is -2, **Number** is rounded to the nearest multiple of a hundred (see last example).

Example:

ROUND(1.234, 2) returns 1.23

ROUND(1.235, 2) returns 1.24

ROUND(444.222, 0) returns 444

ROUND(444.222, -2) returns 400

See also:

ROUNDDOWN, ROUNDUP, MROUND, TRUNC, FIXED, INT, EVEN, ODD, CEILING, FLOOR

ROUNDDOWN (round down to n digits)

Syntax:

ROUNDDOWN(Number, n)

Description:

This rounds a number down (towards zero) to **n** decimal places.

n is the desired number of decimal places.

If **n** is omitted, **Number** is rounded down to the nearest integer.

If **n** is a negative value, **Number** is rounded down to the left of the decimal point. For example, if **n** is -2, **Number** is rounded to the nearest multiple of a hundred (see second last example).

Note:

Negative numbers are rounded up rather than rounded down (see last example). This is not mathematically correct, but it is implemented exactly as in Microsoft Excel for compatibility reasons.

Example:

ROUNDDOWN(1.234, 2) returns 1.23

ROUNDDOWN(1.235, 2) returns 1.23

ROUNDDOWN(888.999, 0) returns 888

ROUNDDOWN(888.999, -2) returns 800

ROUNDDOWN(-7.8, 0) returns -7

See also:

ROUNDUP, ROUND, MROUND, TRUNC, FIXED, INT, EVEN, ODD, CEILING, FLOOR

ROUNDUP (round up to n digits)

Syntax:

ROUNDUP(Number, n)

Description:

This rounds a **Number** up to **n** decimal places.

n is the desired number of decimal places.

If **n** is omitted, **Number** is rounded up to the nearest integer.

If **n** is a negative value, **Number** is rounded up to the left of the decimal point. For example, if **n** is -2, **Number** is rounded up to the nearest multiple of a hundred (see second last example).

Note:

Negative numbers are rounded down rather than rounded up (see last example). This is not mathematically correct, but it is implemented exactly as in Microsoft Excel for compatibility reasons.

Example:

ROUNDUP(1.234, 2) returns 1.24

ROUNDUP(1.235, 2) returns 1.24

ROUNDUP(444.222, 0) returns 445

ROUNDUP(444.222, -2) returns 500

ROUNDUP(-7.3, 0) returns -8

See also:

ROUNDDOWN, ROUND, MROUND, TRUNC, FIXED, INT, EVEN, ODD, CEILING, FLOOR

ROW (row number of cells)

Syntax:

ROW([Reference])

Description:

This returns the row number(s) of a cell or cell range:

If **Reference** is a single cell, the row number of this cell is returned.

If **Reference** is a range of cells, their row numbers are returned as a vertical array. Note: In this case, the formula has to be entered as an *array formula* (see Working with arrays).

If **Reference** is omitted, the row number of the cell that contains this function is returned.

Example:

ROW(D2) returns 2

ROW(D2:F4) returns {2;3;4}

ROW() returns 5 when you use this function in cell A5

See also:

COLUMN, ROWS, CELL, ADDRESS, INDIRECT, OFFSET

ROWS (number of rows in a range)

Syntax:

ROWS(Range)

Description:

This returns the number of rows in the specified cell range.

Example:

ROWS(A1:D5) returns 5

See also:

ROW, COLUMNS, AREAS

RRI (interest for fixed-interest investments)

Syntax:

RRI(Per, PV, FV)

Description:

This returns the interest rate required to obtain a specific future value after a specific period.

This function is applicable to investments with just one payment (made at the beginning) and a fixed interest rate.

The function expects the following arguments:

Per is the number of periods. If, for example, the annual interest rate is considered, a period is one year.

PV is the present value (initial amount)

FV is the future value

Example:

You would like to obtain $5000 with a fixed-interest investment of $3000 after 10 years. How high does the annual interest rate have to be?

RRI(10, 3000, 5000) returns 0.05241 (i.e., about 5.24%).

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

In older versions of PlanMaker, this function was called PCF.

See also:

PERIOD, RATE

RSQ (square of Pearson)

Syntax:

RSQ(Range1, Range2)

Description:

This returns the determination coefficient of two data sets.

This coefficient is a measure of the strength of the linear relationship between data sets. The closer it is to 1, the greater the correlation.

Range1 and **Range2** are cell references to the two data sets. They must contain numbers. Empty cells, text and logical values are ignored.

The data sets must have the same number of values, otherwise the function returns a #N/A error value.

Example:

If A1:A3 contains the values 2, 5, 3 and B1:B3 contains 2, 7, 4, the following applies:

RSQ(A1:A3, B1:B3) returns 0.99436

Additional info:

RSQ is the square of PEARSON (Pearson's correlation coefficient).

See also:

PEARSON

SEARCH (search for text in a text string)

Syntax:

SEARCH(SearchText, TextString [, StartPos])

Description:

This returns the position of **SearchText** within the **TextString**.

StartPos (optional) lets you specify the position in the text string, from which the search should start. If omitted, the search starts at the first character.

The SEARCH function is not case-sensitive, i.e., it does not distinguish between uppercase and lowercase letters. To perform a case-sensitive search, use the FIND function instead.

Example:

SEARCH("a", "Banana") returns 2

SEARCH("A", "Banana") returns 2

SEARCH("a", "Banana", 3) returns 4

SEARCH("x", "Banana") returns the #VALUE! error value, since "x" does not occur in "Banana".

For example, if the word "banana" is in cell B5, you can enter the formula as follows:
SEARCH("a", B5) returns 2

See also:

FIND, REPLACE, SUBSTITUTE, EXACT, MID

SECOND (second)

Syntax:

SECOND(Time)

Description:

This returns the seconds of a time value.

Example:

SECOND("12:34:56 AM") returns 56

SECOND("09/25/2018 10:30:36") returns 36

If cell A1 contains a date with the time 10:30:36, then SECOND(A1) returns 36

See also:

HOUR, MINUTE, MILLISECONDS

SERIESSUM (sum of a power series)

Syntax:

SERIESSUM(x, n, m, Coefficients)

Description:

This returns a power series based on the following formula:

SERIESSUM(x,n,m,a) =

a1*x^n + a2*x^(n+m) + a3*x^(n+2m) + a4*x^(n+3m) + ...

x is the base.

n is the exponent for the first element in the series.

m is the increment for **n**.

Coefficients are the coefficients by which the individual elements of the series are multiplied. **Coefficients** can be either a cell reference or a single-column array.

The number of coefficients also determines the number of elements within the power series. If, for example, 10 coefficients are specified, the series will consist of 10 elements.

Example:

SERIESSUM(2, 2, 1, {1; 2; 3; 4; 5}) returns 516

See also:

POWER

SHEET (index of a worksheet)

Syntax:

SHEET([SheetName])

Description:

This returns the index of a worksheet in the current document. The first worksheet in a document returns 1, the second sheet returns 2, etc.

If the optional argument **SheetName** is given, the index of the worksheet with the specified name is returned. Otherwise, the index of the current worksheet is returned.

Note:

The SHEET function replaces the (almost) identical SHEETNUMBER function

Background info: The SHEETNUMBER function was previously available in PlanMaker but not in Excel. In Excel 2013, Microsoft introduced the same function – but with a different name: SHEET.

There is only one small difference: When the specified parameter is a cell reference, SHEET will return the index of the worksheet that contains this reference. SHEETNUMBER, on the other hand, will read the content of the referenced cell and return the index of the worksheet with this name.

Example:

For the third worksheet in a document, SHEET() returns 3.

SHEET("Expenses") returns the index of the worksheet named "Expenses".

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

SHEETNUMBER, SHEETNAME, SHEETS, FILENAME

SHEETNAME (name of a worksheet)

Syntax:

SHEETNAME([n])

Description:

This returns the name of a worksheet.

n is the number of the worksheet whose name you want to determine. The first worksheet in the workbook is number 1, the second is number 2, etc. If **n** is omitted, the current worksheet is used.

Example:

A document contains four worksheets: "Spring", "Summer", "Autumn" and "Winter" (in that order). The cursor is currently located in a cell of the third worksheet ("Autumn"). The following applies:

SHEETNAME() returns Autumn

SHEETNAME(1) returns Spring

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

SHEET, SHEETS, FILENAME

SHEETNUMBER (compatibility function)

Syntax:

SHEETNUMBER([SheetName])

Description:

Compatibility function for older PlanMaker documents.

This function has been replaced by the (almost) identical SHEET function

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Tip: Instead of this function, use the Excel-2013-compatible function SHEET in future.

See also:

SHEET, SHEETS, SHEETNAME

SHEETS (number of worksheets)

Syntax:

SHEETS([Reference])

Description:

This returns the number of referenced worksheets (both visible and hidden sheets).

If the optional argument **Reference** is not specified, the function returns the number of worksheets in the current workbook.

Example:

A document contains 3 worksheets.

SHEET() returns 3.

SHEET(A1) returns 1 because of the reference to a cell in the current worksheet.

SHEET(Table1:Table2!A1) returns 2 because of the reference to cells in worksheet1 and worksheet2.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

See also:

SHEET, SHEETNAME

SIGN (sign of a number)

Syntax:

SIGN(Number)

Description:

This function returns the sign of a **Number**. The following values may be returned:

1 if Number is positive

-1 if Number is negative

0 if Number is zero

Example:

SIGN(-5) returns -1

See also:

ABS, NEG

SIN (sine)

Syntax:

SIN(Number)

Description:

This returns the sine of an angle.

The angle must be specified in radians.

Tip:

You can use the RADIANS function to convert an angle given in degrees into radians.

Example:

SIN(PI()/2) returns 1

SIN(RADIANS(90)) returns 1

See also:

ASIN, COS, TAN

SINH (hyperbolic sine)

Syntax:

SINH(Number)

Description:

This returns the hyperbolic sine of **Number**.

Example:

SINH(0.45) returns 0.4653

See also:

ASINH, COSH, TANH

SKEW (skewness of a distribution)

Syntax:

SKEW(Number1, Number2, Number3 [, Number4 ...])

or

SKEW(Range1 [, Range2, Range3 ...])

Description:

This returns the skewness of the density function of a probability distribution.

The skewness is a measure of the asymmetry of a distribution.

If there are more values to the right of the arithmetic mean of the distribution, you have a "right-skewed" distribution. SKEW returns a positive value here.

The opposite case is a "left-skewed" distribution. Here, SKEW returns a negative value.

As the SKEW approaches zero, the distribution becomes approximately symmetric.

Number1, **Number2**, **Number3**, etc., are the values to be evaluated. Empty cells, text and logical values are ignored.

At least three values have to be specified, otherwise the function returns a #DIV/0! error value.

Note:

This function does not accept value pairs (x value and y value) as arguments, but *only* the values of the distribution. If the same values appear multiple times, they must be repeated in the argument list accordingly (see example).

Example:

You measured the height of several test persons and obtained the following results: 1 x 1.60m, 2 x 1.65m, 4 x 1.70m, 2 x 1.75m and 1x1.80m.

To calculate the skewness of this distribution, use the following formula:

SKEW(1.60, 1.65, 1.65, 1.70, 1.70, 1.70, 1.70, 1.75, 1.75, 1.80) returns 4.66562E-15.

See also:

INTERCEPT, FORECAST, KURT, NORM.DIST/NORMDIST

SLN (straight-line depreciation)

Syntax:

SLN(Cost, Salvage, Life)

Description:

This returns the depreciation per period of an asset, using the straight-line depreciation method.

Cost = Initial cost of asset

Salvage = Salvage value (value at the end of the depreciation)

Life = Life of asset (in number of periods)

Example:

Initial values: Initial cost = 6000, Salvage value = 3000, Life of asset = 5 years

SLN(6000, 3000, 5) returns 600 (per year)

See also:

SYD, DDB, DB

SLOPE (slope of a linear trend)

Syntax:

SLOPE(y_values, x_values)

Description:

This returns the slope of a best-fit line for the given values. The slope of a line is the value by which the y value grows when the x value is increased by 1.

A *best-fit line* is the result of a *linear regression*, a statistical technique that adapts a line (called *"trendline"* or *"best-fit line"*) to a set of data points (for example, the results of a series of measurements).

For the arguments **y_values** and **x_values**, you usually specify a cell range.

Y_values are the dependent variables.

X_values are the independent variables.

Note:

Note that this function expects the **y_values** first and then the **x_values** – not the other way around.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

The resistance of a temperature-dependent resistor has been measured at several temperatures.

Cells A1:A4 contain the temperatures that were measured (the independent variables): 8, 20, 25, 28

Cells B1:B4 contain the resistances that were measured (the dependent variables): 261, 508, 608, 680

The following calculation returns the slope of a best-fit line based on these values:

SLOPE(B1:B4, A1:A4) returns 20.76799

See also:

INTERCEPT, FORECAST, PEARSON, STEYX

SMALL (k-th smallest number)

Syntax:

SMALL(Range, k)

Description:

This returns the **k**-th smallest value in a data set. If k=1, the smallest value will be returned, if k=2, the second smallest value will be returned, etc.

Range is a range with numeric data – usually a reference to a cell range in which numbers are entered.

k determines which number to return.

k must not be smaller than 1 or greater than the total number of values in **Range**. Otherwise, the function returns a #NUM! error value.

Example:

If A1:A5 contains the values 4, 2, 6, 5 and 9, the following applies:

SMALL(A1:A5, 1) returns 2

SMALL(A1:A5, 2) returns 4

SMALL(A1:A5, 3) returns 5

etc.

See also:

LARGE, MIN, PERCENTILE.EXC/PERCENTILE.INC/PERCENTILE, PERCENTRANK.EXC/PERCENTRANK.INC/PERCENTRANK, RANK.EQ/RANK.AVG/RANK

SORTM (sort)

Syntax:

SORTM(Range, Key [, Mode] [, Columnwise] [, ColumnNumber])

Description:

This function sorts a cell range or an array according to one or more criteria and returns an array with the sorted values.

Range is the cell range or array containing the data to be sorted. You can specify a one-dimensional or two-dimensional cell range or an array for this purpose.

Key determines by which rows the cell range or array should be sorted. To do so, specify a two-row cell range or a two-row array with the following contents:

The first row must contain the numbers of the rows by which you want to sort. If, for example, you enter {1,3} for the sort key, the cell range will be sorted by the first and by the third row of **Range**.

The second row (optional) can contain logical values that specify the sort order:

FALSE or omitted: Ascending order

TRUE: Descending order

If, for example, you use {1,3; true,false} as the sort key, the cell range will be sorted by the first row in *descending* order and by the third row in *ascending* order.

Mode (optional) is a logical value that lets you specify if PlanMaker should distinguish between uppercase and lowercase letters:

FALSE or omitted: Ignore case of letters

TRUE: Distinguish between uppercase and lowercase letters, i.e., sort terms where the first letter is lowercase before terms that begin with the same uppercase letter.

Columnwise (optional) is a logical value that lets you specify if the data should be sorted row-wise or column-wise:

FALSE or omitted: Sort row-wise

TRUE: Sort column-wise

If you use the optional argument **ColumnNumber**, the function only returns a specific column (or row – if sorted column-wise) instead of the entire data:

If omitted, all columns are returned.

If set to n, only the n-th column is returned.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Example:

SORTM({4;2;3}, {1}) returns {2;3;4}

SORTM({4;2;3}, {1;TRUE}) returns {4;3;2}

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

SORTV, section Sorting cells

SORTV (sort)

Syntax:

SORTV(Range [, Descending] [, Mode] [, Data])

Description:

This returns a vector that contains the values of the given one-dimensional cell range in sorted order.

Unlike the SORTM function, SORTV is only applicable to *one*-dimensional cell ranges (vectors). These are areas that comprise either only one row or only one column.

Range is the one-dimensional cell range or array that contains the data to be sorted.

Descending (optional) lets you specify the sort order:

FALSE or omitted: Ascending order

TRUE: Descending order

Mode (optional) is a logical value that lets you specify if PlanMaker should distinguish between uppercase and lowercase letters:

FALSE or omitted: Ignore case of letters

TRUE: Distinguish between uppercase and lowercase letters, i.e., sort terms where the first letter is lowercase before terms that begin with the same uppercase letter.

If you use the optional argument **Data**, the function returns the data from **Data** as the result rather than the data from **Range**. **Data** must be of exactly the same size as **Range**.

This is useful if, for example, you create a worksheet that contains a column with the names of athletes and a column with their scores next to it. If you enter the column with their scores as the **Range** here, but the column with their names as the **Data**, you will obtain a list of names sorted by scores.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Example:

SORTV({4;2;3}) returns {2;3;4}

SORTV({4;2;3}, TRUE) returns {4;3;2}

SORTV({4,2,3}) returns {2,3,4}

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

SORTM, section Sorting cells

SQRT (square root)

Syntax:

SQRT(Number)

Description:

This returns the square root of a **Number**.

The given number must be ≥ 0.

Example:

SQRT(25) returns 5

SQRT(-25) returns the #NUM! error value

See also:

ROOTN, SQRTPI, POWER

SQRTPI (square root of x*Pi)

Syntax:

SQRTPI(Number)

Description:

This returns the square root of (**Number** * pi).

The given number must be ≥ 0.

Example:

SQRTPI(42) equals SQRT(42*PI()), which is 11.48681381

See also:

PI, SQRT, ROOTN

STANDARDIZE (standardized value)

Syntax:

STANDARDIZE(x, Mean, StdDev)

Description:

This returns the standardized value of **x** based on a distribution defined by its mean and standard deviation.

x is the value to be standardized.

Mean is the arithmetic mean of the distribution.

StdDev is the standard deviation of the distribution.

Example:

STANDARDIZE(5, 4, 1.25) returns 0.8

See also:

AVERAGE, STDEV.S/STDEV, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

STDEV (standard deviation of a sample)

Note: STDEV is supplemented by the new identical function STDEV.S, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

STDEV(Number1, Number2 [, Number3 ...])

Description:

This estimates the standard deviation of a population based on a sample.

The standard deviation indicates the average amount by which the specified values deviate from the mean.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The standard deviation is calculated using the "unbiased" or "n-1" method.

Example:

STDEV(2, 2, 3, 4, 1) returns 1.14018

See also:

STDEV.S, STDEVA, STDEV.P/STDEVP, VAR.P/VARP, VAR.S/VAR, AVEDEV, STANDARDIZE, DEVSQ

STDEVA (standard deviation of a sample)

Syntax:

STDEVA(Number1, Number2 [, Number3 ...])

Description:

The **STDEVA** function is almost identical to the STDEV function (see note). It thus provides an estimate of the standard deviation of the population based on a sample.

The standard deviation indicates the average amount by which the specified values deviate from the mean.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

Note:

Unlike the STDEV function, STDEV**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Additional info:

The standard deviation is calculated using the "unbiased" or "n-1" method.

Example:

STDEVA(2, 2, 3, 4, 1) returns 1.14018

See also:

STDEV.S/STDEV, STDEVPA

STDEVP (standard deviation of entire population)

Note: STDEVP is supplemented by the new identical function STDEV.P, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

STDEVP(Number1, Number2 [, Number3 ...])

Description:

This returns the standard deviation of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the standard deviation of a sample, use the STDEV.S/STDEV function instead. This returns an estimate of the standard deviation.

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The standard deviation is calculated using the "biased" or "n" method.

Example:

STDEVP(2, 2, 3, 4, 1) returns 1.0198

See also:

STDEV.P, STDEVPA, STDEV.S/STDEV, VAR.P/VARP, VAR.S/VAR

STDEV.P (standard deviation of entire population)

Syntax:

STDEV.P(Number1, Number2 [, Number3 ...])

Description:

This returns the standard deviation of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the standard deviation of a sample, use the STDEV.S/STDEV function instead. This returns an estimate of the standard deviation.

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The standard deviation is calculated using the "biased" or "n" method.

Example:

STDEV.P(2, 2, 3, 4, 1) returns 1.0198

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

STDEVP, STDEVPA, STDEV.S/STDEV, VAR.P/VARP, VAR.S/VAR

STDEVPA (standard deviation of entire population)

Syntax:

STDEVPA(Number1, Number2 [, Number3 ...])

Description:

The **STDEVPA** function is almost identical to the STDEVP function (see note). It returns the standard deviation of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the standard deviation of a sample, use the STDEVA function instead. This returns an estimate of the standard deviation.

Note:

Unlike the STDEVP function, STDEVP**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Additional info:

The standard deviation is calculated using the "biased" or "n" method.

Example:

STDEVPA(2, 2, 3, 4, 1) returns 1.0198

See also:

STDEV.P/STDEVP

STDEV.S (standard deviation of a sample)

Syntax:

STDEV.S(Number1, Number2 [, Number3 ...])

Description:

This provides an estimate of the standard deviation of a population based on a sample.

The standard deviation indicates the average amount by which the specified values deviate from the mean.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The standard deviation is calculated using the "unbiased" or "n-1" method.

Example:

STDEV.S(2, 2, 3, 4, 1) returns 1.14018

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

STDEV, STDEV.P/STDEVP, VAR.P/VARP, VAR.S/VAR

STEYX (standard error of a linear regression)

Syntax:

STEYX(y_values, x_values)

Description:

This returns the standard error of a linear regression.

The standard error is a measure of the reliability of a linear regression: the greater the standard error, the more the estimated values deviate from the actual values.

For the arguments **y_values** and **x_values**, you usually specify a cell range.

Y_values are the dependent variables.

X_values are the independent variables.

Note:

Note that this function expects the **y_values** first and then the **x_values** – not the other way around.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

The resistance of a temperature-dependent resistor has been measured at several temperatures.

Cells A1:A4 contain the temperatures that were measured (the independent variables): 8, 20, 25, 28

Cells B1:B4 contain the resistances that were measured (the dependent variables): 261, 508, 608, 680

The following calculation returns the standard error predicted by linear regression based on these values:

STEYX(B1:B4, A1:A4) returns 4.97351

See also:

INTERCEPT, SLOPE, FORECAST

SUBSTITUTE (replace text in a text string)

Syntax:

SUBSTITUTE(Text, OldText, NewText [, n])

Description:

This replaces the **OldText** in a text string with the **NewText**.

Text is the original text string.

OldText is the text to be replaced. Note: This function is case-sensitive, thus it distinguishes between uppercase and lowercase letters.

NewText is the replacement text.

The optional argument **n** specifies which occurrence of OldText should be replaced. If **n** is omitted, all occurrences will be replaced.

Example:

SUBSTITUTE("aabbaabb", "aa", "xx") returns xxbbxxbb

SUBSTITUTE("aabbaabb", "aa", "xx", 2) returns aabbxxbb

SUBSTITUTE("aabbaabb", "AA", "xx") returns aabbaabb

See also:

FIND, SEARCH, REPLACE, REPT

SUBTOTAL (calculations without hidden cells)

Syntax:

SUBTOTAL(FunctionID, Range1 [, Range2, Range3 ...])

Description:

This applies one of the arithmetic functions listed below to one or more cell ranges, ignoring all cells that are currently *hidden*.

In detail: If you use the commands of the ribbon tab **Data |** group **Filter** to apply a filter to a cell range and then perform arithmetic calculations on this range, cells that have been filtered out will still be included. If you use the SUBTOTAL function, however, filtered cells will be ignored.

Range1, **Range2**, etc., are the cell ranges you want to evaluate.

FunctionID is a number between 1 and 11 that specifies which arithmetic function to apply:

Value	Function	Explanation
1:	AVERAGE	(arithmetic mean)
2:	COUNT	(number of cells filled with numbers)
3:	COUNTA	(number of cells filled)
4:	MAX	(maximum)
5:	MIN	(minimum)
6:	PRODUCT	(product)
7:	STDEV	(standard deviation of a sample)
8:	STDEVP	(standard deviation of entire population)
9:	SUM	(sum)
10:	VAR	(variance of a sample)
11:	VARP	(variance of entire population)

If, for example, you set **FunctionID** to 9, the SUBTOTAL function returns the sum of the specified cell range(s) – but ignores all cells that have been hidden via the commands of the **Sort** button on the ribbon tab **Data |** group **Filter**.

Ignoring manually hidden cells also:

There's a second variant of this function. In this case, all rows are ignored that have been hidden *manually* (using, e.g., the ribbon command **Home |** group **Cells | Visibility | Hide rows** or the ribbon command **Data |** group **Outliner | Group**).

To use this variant, simply set the **FunctionID** to a value between 101 and 111 instead of a value between 1 and 11, as shown in the following table:

Value	Function	Explanation
101:	AVERAGE	(arithmetic mean)
102:	COUNT	(number of cells filled with numbers)
103:	COUNTA	(number of cells filled)
104:	MAX	(maximum)
105:	MIN	(minimum)
106:	PRODUCT	(product)
107:	STDEV	(standard deviation of a sample)
108:	STDEVP	(standard deviation of entire population)
109:	SUM	(sum)
110:	VAR	(variance of a sample)
111:	VARP	(variance of entire population)

If, for example, you set **FunctionID** to 109, SUBTOTAL returns the sum of the specified cell range(s) – but ignores all cells that have been hidden via the commands of the ribbon tab **Data** | group **Filter**.

In addition, all rows will be ignored that were hidden manually via the ribbon command **Home** | group **Cells** | **Visibility** | **Hide rows** or via the ribbon command **Data** | group **Outliner** | **Group**.

Please note: Only hidden *rows* will be ignored, while hidden *columns* will still be included in the calculations.

Example:

SUBTOTAL(9, A1:A10) returns the sum of the range A1:A10, but ignores all cells filtered out by the commands of the ribbon tab **Data** | group **Filter**.

See also:

For more information on filters, see Filtering cells by their content.

SUM (Sum)

Syntax:

SUM(Number1 [, Number2, Number3 ...])

Description:

This returns the sum of the given numbers.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be summed.

If you use this function with *cell references* (for example A1:A3), only numbers are considered, while text strings and logical values occurring in the referenced cells are ignored.

However, the situation is different if you type *fixed values* directly into the formula: Here, numbers entered as a text string (for example, "42") are included. Furthermore, the logical value TRUE is treated as 1 and FALSE as 0. See examples below.

Example:

SUM(1, 2, 3, 4) equals 1+2+3+4, which is 10

If the cells A1:A4 contain the values 1, 2, 3 and 4, the following applies:

SUM(A1:A4) equals 1+2+3+4, which is 10

Please note:

If the cells A1:A3 contain 1, "2" and TRUE:

SUM(A1:A3) returns 1, since the text string and the logical value are ignored.

They are counted only if they were typed as *fixed* values directly into the formula:

SUM(1, "2", TRUE) returns 1+2+1, which is 4

Additional info:

There are some special terms for the SUM formula, where PlanMaker automatically completes the formula with additional functions. The following examples illustrate how it works:

If you enter SUM(A5:A), all values in column A are added, except A1:A4.

If you enter SUM(E2:2), all values in row 2 are added, except A2:D2.

If you enter SUM(A:A10), all values of A1:A10 are added.

If you enter SUM(2:E2), all values of A2:E2 are added.

Tip:

You can also use the **AutoSum** icon on the ribbon tab **Home** | group **Contents** to quickly calculate the sum of a specific cell range:

Select an empty cell just below a group of cells containing numbers and then click on the AutoSum icon. The Sum function for the cell range above is now automatically inserted into the selected cell. You just have to press the Enter key↵ to display the result.

Alternative method: Select a range of cells with numbers and then click on the AutoSum icon, which will insert the sum of this cell range just below these cells.

You can also apply the AutoSum icon for several other arithmetic functions. To do so, click on the arrow to the right of the icon. PlanMaker will open a dropdown list containing the available functions – for example, Count, Product, Average, etc. Select the desired function from the list to apply it to a cell range using the methods just described.

Note: The last selected function from the list will be applied again when you click directly on the AutoSum icon.

See also:

PRODUCT, SUMPRODUCT, SUMIF, SUMIFS, SUMXMY2, SUMX2MY2, SUMX2PY2

SUMIF (sum if condition is true)

Syntax:

SUMIF(Range, Criterion [, SumRange])

Description:

This returns the sum of those values in a cell range that fulfill the specified criterion.

Range is the cell range to be evaluated.

Criterion is the condition that the values in **Range** have to fulfill to be included:

Use numbers or text (like "42" or "screws") to obtain the sum of all cells that contain the relevant value.

Use conditions (like ">10" or "<=5") to sum all cells that match the specified condition.

Note: Criteria always have to be enclosed within double quotation marks (").

SumRange (optional) lets you specify the cell range that contains the values to be summed. If **SumRange** is omitted, the values in **Range** will be summed.

Example:

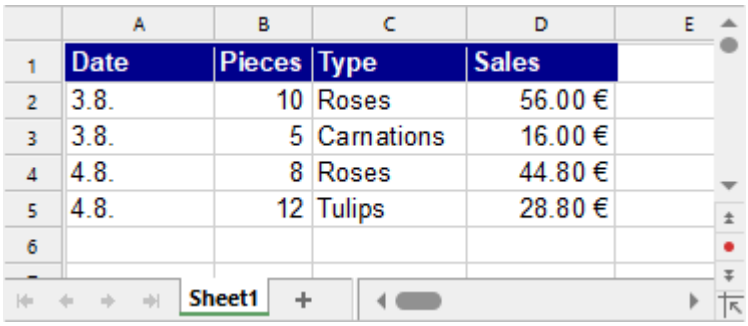
The cells A1:A5 contain the values 1, 2, 3, 2, 1.

Then the following applies:

SUMIF(A1:A5, "2") returns 2+2, which is 4

SUMIF(A1:A5, ">=2") returns 2+3+2, which is 7

You enter your daily income in a table as follows:



	A	B	C	D	E
1	Date	Pieces	Type	Sales	
2	3.8.	10	Roses	56.00 €	
3	3.8.	5	Carnations	16.00 €	
4	4.8.	8	Roses	44.80 €	
5	4.8.	12	Tulips	28.80 €	
6					

Sheet1

How much is the sum of the sales if you only consider sales larger than €40?

SUMIF(D2:D5, ">40") results in €100.80

How many flowers of the type "roses" were sold?

SUMIF(C2:C5, "Roses", B2:B5) returns 18

In the above formula, the optional argument **SumRange** was used: A search was performed on the **Range** C2:C5 (that is, the type), but the numbers to be added were taken from the **SumRange** B2:B5 (that is, the number of pieces).

See also:

AVERAGEIF, AVERAGEIFS, SUM, SUMIFS, COUNTIF, COUNTIFS

SUMIFS (sum if conditions are true)

Syntax:

SUMIFS(SumRange, Range1, Criterion1 [, Range2, Criterion2 ...])

Description:

This returns the sum of those values in a cell range that fulfill *all* of the specified criteria.

This function is similar to the SUMIF function, but it allows you to specify multiple search ranges (including one criterion for each range).

Note: The order of the parameters is different here: In SUMIF, **SumRange** is the last parameter – while it is the first parameter in SUMIFS.

SumRange is the cell range that contains the values to be summed.

Range1 is the first cell range to be checked for the specified criterion. This range must have the same dimensions as **SumRange**.

Criterion1 is the condition that the values in **Range1** have to fulfill to be included:

Use numbers or text (like "42" or "screws") to obtain the sum of all cells that contain this value.

Use conditions (like ">10" or "<=5") to sum all cells that match the specified condition.

Note: Criteria always have to be enclosed within double quotation marks (").

Unlike the SUMIF function, SUMIFS allows you to specify multiple search ranges (including one criterion for each range). If you do so, only occurrences where *all* criteria are fulfilled are included in the calculation of the sum.

Compatibility notes:

This function is not supported by the **.xls** file format (used in Microsoft Excel 2003 and earlier). If you save a document in this format, all calculations using this function will be replaced with their current result as a *fixed value*.

If you want to use this function, you should not save your document in the "Microsoft Excel 97-2003 (.xls)" file format, but choose one of the following formats instead:

- "PlanMaker document (.pmdx or .pmd)"
- or: "Microsoft Excel 2007-2021 (.xlsx)"

Example:

SUMIFS(A1:A10, C1:C10, ">1", C1:C10, "<2")

This formula returns the sum of only those cells in the cell range A1:A10 where the corresponding cell in the range C1:C10 is greater than 1 and less than 2.

See also:

AVERAGEIF, AVERAGEIFS, SUM, SUMIF, ZÄHLENWENN, COUNTIFS

SUMPRODUCT (sum of products)

Syntax:

SUMPRODUCT(Range1 [, Range2, Range3 ...])

Description:

This multiplies the corresponding elements of two or more arrays and returns the sum of these products.

Range1, **Range2**, etc., are cell ranges or arrays that contain the values to be evaluated. As a rule, you must specify the cell ranges which contain the values here.

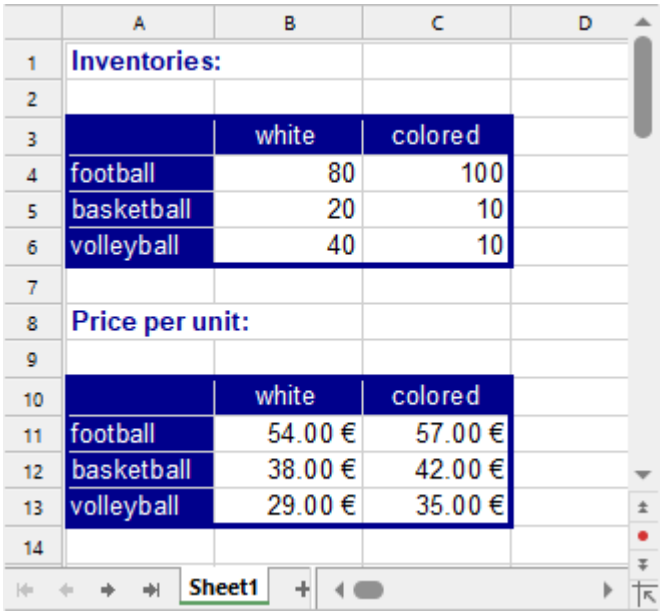
All ranges must have the same number of rows and columns, otherwise the function returns a #N/A error value.

Empty cells and cells with non-numeric content (strings, logical values, etc.) are interpreted as zero.

Example:

You have created a document in which you enter your inventory.

The document contains two areas of equal size. In the first area, the quantities of the articles are entered. In the second area, the corresponding unit prices are entered.



	A	B	C	D
1	Inventories:			
2				
3		white	colored	
4	football	80	100	
5	basketball	20	10	
6	volleyball	40	10	
7				
8	Price per unit:			
9				
10		white	colored	
11	football	54.00 €	57.00 €	
12	basketball	38.00 €	42.00 €	
13	volleyball	29.00 €	35.00 €	
14				

Sheet1

In order to calculate the total value of all articles, the quantities of each item must be multiplied by their respective unit price. Finally, the sum of these products must be calculated.

This entire calculation can be combined with the SUMPRODUCT function:

SUMPRODUCT(B4:C6, B11:C13) returns 12710.

Thus, there are currently articles with a total value of €12710 in your inventory.

See also:

SUM, SUMSQ, SUMXMY2, SUMX2MY2, SUMX2PY2

SUMSQ (sum of squares)

Syntax:

SUMSQ(Number1 [, Number2, Number3 ...])

Description:

This returns the sum of the squares of the given numbers.

Example:

SUMSQ(1, 2, 3) equals 1^2 + 2^2 + 3^2, which is 14.

See also:

SUM, SUMPRODUCT, SUMXMY2, SUMX2MY2, SUMX2PY2

SUMX2MY2 (sum of x^2 - y^2)

Syntax:

SUMX2MY2(Range1, Range2)

Description:

This returns the sum of the difference of squares of corresponding values in two arrays.

The equation for this function is:

SUMX2MY2 = ∑(x^2 - y^2)

Range1 and **Range2** are the arrays whose SumX2MY2 is to be calculated. As a rule, you must specify the cell ranges which contain the values here.

Both ranges must have the same number of rows and columns, otherwise the function returns a #N/A error value.

Pairs of values where at least one value is an empty cell or a non-numeric value (text string or logical value) are ignored.

Example:

Cells A1 to A3 contain the following values: 9, 4, 7

Cells B1 to B3 contain the following values: 1, 0, 5

If you calculate SUMX2MY2(A1:A3, B1:B3) here, this is equivalent to the following calculation:

(9^2) - (1^2) + (4^2) - (0^2) + (7^2) - (5^2)

The result is 120.

See also:

SUM, SUMPRODUCT, SUMX2PY2, SUMXMY2, SUMSQ

SUMX2PY2 (sum of x^2 + y^2)

Syntax:

SUMX2PY2(Range1, Range2)

Description:

This returns the sum of the sum of squares of corresponding values in two arrays.

The equation for this function is:

SUMX2PY2 = ∑(x^2 + y^2)

Range1 and **Range2** are the arrays whose sumX2PY2 is to be calculated. As a rule, you must specify the cell ranges which contain the values here.

Both ranges must have the same number of rows and columns, otherwise the function returns a #N/A error value.

Pairs of values where at least one value is an empty cell or a non-numeric value (text string or logical value) are ignored.

Example:

Cells A1 to A3 contain the following values: 9, 4, 7

Cells B1 to B3 contain the following values: 1, 0, 5

If you calculate SUMX2PY2(A1:A3, B1:B3) here, this is equivalent to the following calculation:

(9^2) + (1^2) + (4^2) + (0^2) + (7^2) + (5^2)

The result is 172.

See also:

SUM, SUMPRODUCT, SUMX2MY2, SUMXMY2, SUMSQ

SUMXMY2 (sum of (x - y)^2)

Syntax:

SUMXMY2(Range1, Range2)

Description:

This returns the sum of squares of differences of corresponding values in two arrays.

The equation for this function is:

SUMXMY2 = ∑(x - y)^2

Range1 and **Range2** are the arrays whose SumXMY2 is to be calculated. As a rule, you must specify the cell ranges which contain the values here.

Both ranges must have the same number of rows and columns, otherwise the function returns a #N/A error value.

Pairs of values where at least one value is an empty cell or a non-numeric value (text string or logical value) are ignored.

Example:

Cells A1 to A3 contain the following values: 9, 4, 7

Cells B1 to B3 contain the following values: 1, 0, 5

If you calculate SUMXMY2(A1:A3, B1:B3) here, this is equivalent to the following calculation:

(9-1)^2 + (4-0)^2 + (7-5)^2

The result is 84.

See also:

SUM, SUMPRODUCT, SUMX2MY2, SUMX2PY2, SUMSQ

SWITCH (categorize initial value)

Syntax:

SWITCH(Calculation, Result1, Value1 [, Result2, Value2 ...] [, DefaultValue])

Description:

This function checks an initial value and compares it to a series of defined values for the first match.

Calculation is the initial value and can be any expression.

Result1 is the first element to be compared with the initial value for **Calculation**.

Value1 is to be returned if **Result1** matches the initial value for **Calculation**.

Result2; Value2 (optional) is the second compared element, etc.

Once the first match is found, its assigned value is returned.

Note:

If no match is found, the #N/A error is returned.

However, you can set a **DefaultValue** instead at the end of the formula, which will be returned if no match is found (see example 2 below).

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

Example 1:

If a specific number is found in cell A1, PlanMaker should output a name defined for this purpose. The following key applies: 1 stands for Lisa, 2 stands for Peter, 3 stands for Anna.

The formula SWITCH(A1, 1, "Lisa", 2, "Peter", 3, "Anna"), for example, then returns the following:

If cell A1 contains the number 1 = Lisa

If cell A1 contains the number 2 = Peter

If cell A1 contains the number 3 = Anna

If cell A1 contains anything other than 1, 2, or 3, the formula returns the result #N/A because no match was found. Alternatively, you can determine which default value the formula generally returns if no match is found (see example 2).

Example 2:

You have the same initial situation as in example 1. At the end of the formula, set a default value that will be returned if no match is found:

SWITCH(A1, 1, "Lisa", 2, "Peter", 3, "Anna", "no match")

If, for example, cell A1 contains 4, the result "no match" is returned.

See also:

IF, IFS, VLOOKUP, CHOOSE

SYD (sum-of-years' digits depreciation)

Syntax:

SYD(Cost, Salvage, Life, Per)

Description:

This returns the depreciation of an asset for the specified period, using the sum-of-years' digits method.

The function expects the following arguments:

Cost = Initial cost of asset

Salvage = Salvage value (value at the end of the depreciation)

Life = Life of asset (in number of periods)

Per = Period to evaluate (in the same time unit as **Life**)

Note:

This function is only partially applicable to the tax law of some countries (for example, Germany)!

Example:

In the case of a machine purchased for €200,000 that is used for 10 years and then has a salvage value of €10,000, the following annual depreciation amount must be applied for the first year:

SYD(200000, 10000, 10, 1) returns €34,545.45 (depreciation in year 1)

See also:

DBB, DB, SLN

T (convert value into text string)

Syntax:

T(Value)

Description:

This converts a value into a text string.

The specified value must be a text string, otherwise the function returns an empty text string.

Note:

This function was retained only for compatibility with older spreadsheet applications. It is **no** longer commonly used, as current spreadsheet applications (including PlanMaker) automatically convert values into text strings if necessary.

Example:

T("Text") returns Text

T("42") returns 42

T(42) returns an empty text string

T(TRUE) returns an empty text string

See also:

N, VALUE, TEXT

TAN (tangent)

Syntax:

TAN(Number)

Description:

This returns the tangent of an angle.

The angle must be specified in radians.

Tip:

You can use the RADIANS function to convert an angle given in degrees into radians.

Example:

TAN(PI()/4) returns 1

TAN(RADIANS(45)) returns 1

See also:

ATAN, ATAN2, COT, SIN, COS

TANH (hyperbolic tangent)

Syntax:

TANH(Number)

Description:

This returns the hyperbolic tangent of a **Number**.

Example:

TANH(0.45) returns 0.4219

See also:

ATANH, COSH, SINH

TDIST (t-distribution)

Note: TDIST is supplemented by the new functions T.DIST.2T, T.DIST.RT and T.DIST with additional functionalities, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

TDIST(X, DegreesFreedom, Tails)

Description:

This returns the probability of the Student's t-distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Tails is the number of distribution tails:

1: One-tailed distribution

2: Two-tailed distribution

See also:

T.DIST.2T/T.DIST.RT/T.DIST, T.INV.2T/T.INV/TINV, T.TEST/TTEST

T.DIST (t-distribution)

Syntax:

T.DIST(X, DegreesFreedom, Cumulative)

Description:

This returns the probability of the Student's t-distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Cumulative determines the type of function as a switch: If you enter the value TRUE, the cumulative distribution function of the left-tailed Student's t-distribution is calculated. By entering the value FALSE, the probability density function of the Student's t-distribution is calculated.

Note:

The T.DIST, T.DIST.2T and T.DIST.RT functions supplement the previous TDIST function, whereby T.DIST has the additional argument **Cumulative**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

T.DIST.2T/T.DIST.RT/TDIST, T.INV/T.INV.2T/TINV, T.TEST/TTEST

T.DIST.2T (t-distribution two-tailed)

Syntax:

T.DIST.2T(X, DegreesFreedom)

Description:

This returns the probability of the **two-tailed** Student's t-distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The T.DIST.2T function, T.DIST.RT and T.DIST functions supplement the previous TDIST function, whereby T.DIST.2T returns the same result as TDIST with the argument **Tails=2**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

T.DIST.RT/T.DIST/TDIST, T.INV.2T/T.INV/TINV, T.TEST/TTEST

T.DIST.RT (t-distribution right-tailed)

Syntax:

T.DIST.RT(X, DegreesFreedom)

Description:

This returns the probability of the **right-tailed** Student's t-distribution.

X is the value whose probability is to be calculated. **X** must be ≥ 0.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Note:

The T.DIST.RT, T.DIST.2T and T.DIST functions supplement the previous TDIST function, whereby T.DIST.RT returns the same result as TDIST with the argument **Tails=1**.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

T.DIST.2T/T.DIST/TDIST, T.INV.2T/T.INV/TINV, T.TEST/TTEST

TEXT (convert number into text string)

Syntax:

TEXT(Number, Format)

Description:

This converts a number into a text string in the specified number format.

Use the argument **Format** to specify the desired format. Enter a corresponding text string here. This must be structured like a text string that you create when creating custom number formats in the dialog box of the ribbon tab **Home** | group **Number** ◢ (see Structure of a custom number format).

Additional info:

To change the format of numbers without having to convert them into a text string, choose the commands of the ribbon tab **Home** | group **Number** and select the desired number format.

Example:

TEXT("9/25/2018", "MM/DD/YY") returns 09/25/18 (as a text string)

TEXT(2.715, "0.00 ""USD""") returns 2.72 USD (as a text string)

See also:

VALUE, T, N, FIXED

TEXTJOIN (concatenate strings with separators)

Syntax:

TEXTJOIN(Separator, IgnoreEmptyCells, String1 [, String2 ...])

Description:

This combines two or more text strings into one text string. Unlike the CONCAT function, you can use TEXTJOIN to specify separators between text strings. You can also specify whether empty cells should be ignored.

Separator is the desired separator between text strings. If omitted, no separator is used.

IgnoreEmptyCells determines whether empty cells should be ignored. TRUE or omitted ignores empty cells, while FALSE also concatenates empty cells.

String1, String2 etc., are the cells with text strings to be connected.

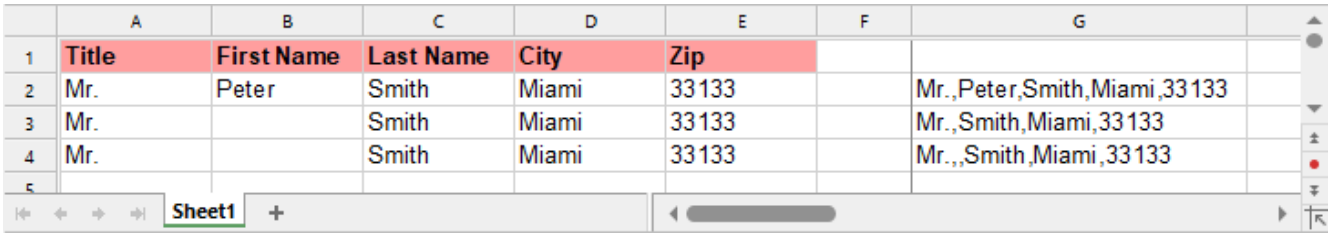
Example:

In the figure below, column G shows the result for the following formulas:

In row2, the formula is TEXTJOIN(",", TRUE, A2:E2)

In row3 (with an empty cell B3), the formula is TEXTJOIN(",", TRUE, A3:E3)

In row4 (with an empty cell B4), the formula is TEXTJOIN(",", FALSE, A4:E4)



	A	B	C	D	E	F	G
1	Title	First Name	Last Name	City	Zip		
2	Mr.	Peter	Smith	Miami	33133		Mr.,Peter,Smith,Miami,33133
3	Mr.		Smith	Miami	33133		Mr.,Smith,Miami,33133
4	Mr.		Smith	Miami	33133		Mr.,,Smith,Miami,33133
5							

Sheet1

Compatibility notes:

Microsoft Excel supports this function only in version 2019 or later. The function is unknown in older versions.

See also:

CONCAT, CONCATENATE

TIME (create time)

Syntax:

TIME(Hour, Minute, Second)

Description:

This returns a serial date value based on the specified **Hour**, **Minute** and **Second**.

Example:

TIME(0, 20, 0) returns 0.01389, the serial date value for the corresponding time.

However, PlanMaker automatically formats this date value in the number format "Date/Time", thus the cell displays: 00:20:00.

See also:

TIMEVALUE, DATE, DATEVALUE

TIMEDIFF (time difference)

Syntax:

TIMEDIFF(StartTime, EndTime)

Description:

This returns the time elapsed between two time values.

A time must be specified for **StartTime** and **EndTime**.

You can also specify a date with a time (see last example), but in this case both **StartTime** and **EndTime** have to include a date, otherwise a #VALUE! error value is returned.

Note:

Compared to simply subtracting time values, this function has the following advantage when **EndTime** is smaller than **StartTime**:

An employee starts his shift, for example, at 10:00 pm and leaves at 6:00 am. If calculating his hours of work by subtracting 10:00 pm from 6:00 am, the result would be -16 hours. If the TIMEDIFF function is used, the correct result (8 hours) is returned.

Example:

TIMEDIFF("8:00 AM", "4:00 PM") returns 8:00

TIMEDIFF("10:00 PM", "6:00 AM") returns 8:00

TIMEDIFF("10/01/2018 10:00 PM", "10/02/2018 6:00 AM") returns 8:00

Compatibility notes:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

See also:

TIME, TIMEVALUE, DAYS/DAYSP, DAYS360

TIMEVALUE (convert text into time)

Syntax:

TIMEVALUE(Text)

Description:

This converts a text string representing a date into a serial time value.

Note:

This function was retained only for compatibility with older spreadsheet applications. It is **no** longer commonly used, as current spreadsheet applications (including PlanMaker) automatically convert times specified as text into date/time values if necessary.

Example:

TIMEVALUE("10:30:00 am") returns 0.4375.

If you now format this using the options of the ribbon tab **Home** | group **Number** in one of the date/time formats (for example, h:mm), the cell will display the time behind this value: 10:30.

See also:

DATEVALUE, DATE, TIME

TINV (percentiles of the t-distribution)

Note: TINV is supplemented by the new identical function T.INV.2T and the completely new function T.INV, which are available in newer versions of Microsoft Excel (2010 or later).

Syntax:

TINV(Probability, DegreesFreedom)

Description:

This returns the inverse of the **two-tailed** Student's t-distribution function (i.e., percentiles of this distribution).

Probability is the two-tailed probability to be evaluated. The values must be within the range 0 to 1.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Please note: The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

See also:

T.INV.2T/T.INV, T.TEST/TTEST, T.DIST.2T/T.DIST.RT/T.DIST/TDIST

T.INV (percentiles of the t-distribution left-tailed)

Syntax:

T.INV(Probability, DegreesFreedom)

Description:

This returns the inverse of the **left-tailed** Student's t-distribution function (i.e., percentiles of this distribution).

Probability is the two-tailed probability to be evaluated. The values must be within the range 0 to 1.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Please note: The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Note:

The T.INV and T.INV.2T functions supplement the previous TINV function.

T.INV.2T returns the same result as TINV (**two-tailed** percentiles).

T.INV (**left-tailed** percentiles) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

TINV/T.INV.2T, T.TEST/TTEST, T.DIST.2T/T.DIST.RT/T.DIST/TDIST

T.INV.2T (percentiles of the t-distribution two-tailed)

Syntax:

T.INV.2T(Probability, DegreesFreedom)

Description:

This returns the inverse of the **two-tailed** Student's t-distribution function (i.e., percentiles of this distribution).

Probability is the two-tailed probability to be evaluated. The values must be within the range 0 to 1.

DegreesFreedom is the number of degrees of freedom. This value must be ≥ 1. Integers are expected; if this is not the case, PlanMaker automatically truncates the decimal places.

Please note: The result of this function is calculated using an iteration method. If, after 100 iterations, PlanMaker does not find a result that deviates less than ±3x10^-7 from **Probability**, it displays the error value #N/A and cancels the approximation.

Note:

The T.INV.2T and T.INV functions supplement the previous TINV function.

T.INV.2T returns the same result as TINV (**two-tailed** percentiles).

T.INV (**left-tailed** percentiles) is completely new.

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

TINV/T.INV, T.TEST/TTEST, T.DIST.2T/T.DIST.RT/T.DIST/TDIST

TODAY (current date)

Syntax:

TODAY()

Description:

This returns the current system date (without time).

The displayed value is dynamic, so it is automatically updated when the document is opened.

Note:

The result is a number – like all dates in a spreadsheet. PlanMaker automatically formats it in the date format MM.DD.YYYY so that it will be displayed as a date. You can use the options from the ribbon tab **Home** | group **Number** to adjust the displayed date format to your needs.

Example:

TODAY() results in 07/31/2018

See also:

NOW, CREATEDATE

TRANSPOSE (transpose matrix)

Syntax:

TRANSPOSE(Array)

Description:

This returns the transpose of a matrix, i.e., swaps its columns with its rows. When you transpose, for example, a 2x4 matrix, the result is a 4x2 matrix.

Array is a cell range or array containing the matrix to be evaluated.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Please note that the range to be selected before entering the formula must have the correct *dimensions*. If the matrix consists of **x** rows and **y** columns, the resulting range must have **y** rows and **x** columns.

Example:

TRANSPOSE({1,2,3;4,5,6}) returns {1,4;2,5;3,6}

See also:

Section Working with arrays, MDETERM, MINVERSE, MMULT, MSOLVE

TREND (values of a linear regression)

Syntax:

TREND(y_values [, x_values] [, New_x_values] [, Constant])

Description:

This returns the values of a linear regression.

Linear regression is a statistical technique that adapts a line (called *"trendline"* or *"best-fit line"*) to a set of data points (for example, the results of a series of measurements).

The TREND function returns an array with the y values of a trendline calculated from the given y values and x values.

To clarify, the function expects the following arguments:

y_values: Here, you specify the existing y coordinates, that is, the dependent values. **y_values** can be either a cell reference or an array.

x_values: Here, you have the option of specifying the x coordinates, that is, the independent values. **x_values** can also be either a cell reference or an array. If **x_values** are omitted, the function automatically uses the values 1, 2, 3, ... (according to the number of existing **y_values**).

New_x_values (optional) are the x values for which you want to determine the corresponding y values on the curve. If omitted, the values in **x_values** will be used. If required, however, you can specify other x coordinates with **New_x_values**. **New_x_values** can also be either a cell reference or an array.

Constant (optional) is a logical value that lets you specify if the constant b (the y-intercept point) should be set to zero:

TRUE or omitted: b is automatically calculated from the available data.

FALSE: b is set to zero. This forces the trendline to run through the zero point. The m coefficient(s) are adjusted accordingly.

Note:

Formulas using this function have to be entered as an *array formula*. For more information, see Working with arrays.

Additional info:

The linear regression is performed with this function using the least squares method.

Example:

The resistance of a temperature-dependent resistor has been measured at several temperatures.

Cells A1:A4 contain the temperatures that were measured (the independent variables): 8, 20, 25, 28

Cells B1:B4 contain the resistances that were measured (the dependent variables): 261, 508, 608, 680

The following calculation returns an array with the y coordinates of a best-fit line calculated from these values:

TREND(B1:B4, A1:A4)

See also:

LINEST, LOGEST, GROWTH, FORECAST

TRIM (remove spaces from text)

Syntax:

TRIM(Text)

Description:

This removes all spaces at the beginning and end of the specified text string.

Example:

TRIM(" Text ") returns text

See also:

CLEAN

TRIMMEAN (mean without marginal values)

Syntax:

TRIMMEAN(Area, Percentage)

Description:

This returns the mean of the values in the middle of a data set (by excluding the specified percentage of extreme values).

"Extreme values" are the smallest and largest values of a data set. These are often "outliers" (unusually low or high values, for example, due to a measurement error).

Area is the cell range or array to evaluate.

Percentage is the percentage of values to be excluded.

If, for example, 100 values are given and **Percentage** is set to 10%, 10 values will be excluded: the 5 lowest and the 5 highest.

Note: For symmetrical purposes, the number of excluded values will automatically be rounded down to a multiple of 2. If, for example, 30 values are given and **Percentage** is set to 10%, only 2 values will be excluded instead of 3.

Example:

Cells A1:A10 contain the following values: 43, 45, 42, **0**, 44, 45, 42, **1234**, 40, 41

As you can see, two values are exceptionally low/high (e.g., due to a measurement error): 0 and 1234. If you used the AVERAGE function, you would thus obtain an unusable average:

AVERAGE(A1:A10) returns 159.2

If you use TRIMMEAN instead and, for example, exclude 20% of the top and bottom values – in this case, these are conveniently exactly the two outliers among the 10 values – a useful average value is returned:

TRIMMEAN(A1:A10, 20%) returns 42.75

See also:

AVERAGE, GEOMEAN, HARMEAN

TRUE (logical value TRUE)

Syntax:

TRUE()

Description:

This returns the logical value TRUE.

See also:

FALSE

TRUNC (truncate a number)

Syntax:

TRUNC(Number [, n])

Description:

This returns the given number truncated to **n** digits after the decimal point.

n is the desired number of decimal places.

If **n** is omitted, **Number** is truncated to an integer.

If **n** is a negative value, **Number** is truncated to the left of the decimal point (see last example).

Example:

TRUNC(5.779) returns 5

TRUNC(5.779, 2) returns 5.77

TRUNC(1234, -2) returns 1200

Additional info:

To obtain the decimal places, use the DECIMALS function.

See also:

ROUNDDOWN, ROUNDUP, DECIMALS, ROUND

TTEST (t-test)

Note: TTEST is supplemented by the new identical function T.TEST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

TTEST(Array1, Array2, Tails, Type)

Description:

This returns the result of a Student's t-test.

A t-test can be used to check the consistency of the means of two samples.

Array1 and **Array2** are the two arrays (samples) to be evaluated.

Tails is the number of distribution tails:

1: One-tailed distribution

2: Two-tailed distribution

Type is the type of t-test:

1: dependent samples ("paired")

2: independent samples, equal variances ("homoscedastic")

3: independent samples, different variances ("heteroscedastic")

See also:

T.TEST, T.INV.2T/T.INV/TINV, T.DIST.2T/T.DIST.RT/T.DIST/TDIST

T.TEST (t-test)

Syntax:

T.TEST(Array1, Array2, Tails, Type)

Description:

This returns the result of a Student's t-test.

A t-test can be used to check the consistency of the means of two samples.

Array1 and **Array2** are the two arrays (samples) to be evaluated.

Tails is the number of distribution tails:

1: One-tailed distribution

2: Two-tailed distribution

Type is the type of t-test:

1: dependent samples ("paired")

2: independent samples, equal variances ("homoscedastic")

3: independent samples, different variances ("heteroscedastic")

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

TTEST, T.INV.2T/T.INV/TINV, T.DIST.2T/T.DIST.RT/T.DIST/TDIST

TYPE (type of a value)

Syntax:

TYPE(Value)

Description:

This returns the type of the specified **Value** in the form of a number.

You can see which number stands for which type in the following table:

"Value" is a ...	Result
Number or empty cell	1
Text string	2
Boolean	4
Error value	16
Array	64

Example:

TYPE(42) returns 1

If A1 contains the text string "Text", the following applies:

TYPE(A1) returns 2

See also:

ERRORTYPE

UPPER (convert text to uppercase)

Syntax:

UPPER(Text)

Description:

This converts all letters in a text string into uppercase letters.

Example:

UPPER("PlanMaker") returns PLANMAKER

See also:

PROPER, LOWER

USERFIELD (output user info)

Syntax:

USERFIELD(FieldName)

Description:

This returns the content of one of the fields with your user information (name, address, etc.) entered in PlanMaker's settings dialog.

Note: You can edit your user information by choosing the command **File | Options** , switching to the **General** tab and clicking on the **User info** button.

FieldName is the name of the field to be returned (see table below).

It is not case-sensitive. Note, however, that **FieldName** must always be enclosed within double quotation marks (") – as in USERFIELD("User.City").?

FieldName can be one of the following text strings:

User info field	Field name
Title	User.Title
Name	User.Name
Initials	User.Initials
Company	User.Company
Department	User.Department
Address 1	User.Address 1
Address 2	User.Address 2
State, ZIP	User.ZIP
City	User.City
Country	User.Country
Phone 1	User.Phone 1
Phone 2	User.Phone 2
Phone 3	User.Phone 3

Fax	User.Fax
E-mail 1	User.E-mail 1
E-mail 2	User.E-mail 2
E-mail 3	User.E-mail 3
Web site	User.Website

Compatibility note 1:

This function does not exist in Microsoft Excel. If you save a document in Excel format, all calculations using this function will be replaced with their current result as a *fixed* value.

Compatibility note 2:

In PlanMaker 2016 and older, there were two sets of user data: "User (home)" and "User (business)". These have been reduced to one set since PlanMaker 2018. When you open an old document that uses this function, its parameters are automatically converted accordingly.

Example:

USERFIELD("User.Website") returns www.softmaker.com

VALUE (convert text into a number)

Syntax:

VALUE(Text)

Description:

This converts a text string into a number.

Text is the text string to be converted. It must contain the text representation of a number, otherwise a #VALUE! error value is returned.

Note:

This function was retained only for compatibility with older spreadsheet applications. It is **no** longer commonly used, as current spreadsheet applications (including PlanMaker) automatically convert text strings into numbers if necessary.

Example:

VALUE("42") returns 42

VALUE("42 USD") returns 42, since "USD" is a valid currency unit.

VALUE("42 screws") returns a #VALUE! error value.

VALUE("TRUE") returns 1

VALUE("09/25/2014") returns 41907, the serial date value for the date 09/25/2014.

See also:

NUMBERVALUE, TEXT, T, N

VAR (variance of a sample)

Note: VAR is supplemented by the new identical function VAR.S, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

VAR(Number1, Number2 [, Number3 ...])

Description:

This provides an estimate of the variance of a population based on a sample.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The variance is calculated using the "unbiased" or "n-1" method.

Example:

VAR(2, 2, 3, 4, 1) returns 1.3

See also:

VAR.S, VARA, VARPA, VAR.P/VARP, STDEV.S/STDEV, STDEV.P/STDEVP

VARA (variance of a sample)

Syntax:

VARA(Number1, Number2 [, Number3 ...])

Description:

The **VARA** function is almost identical to the VAR function (see note). It thus provides an estimate of the variance of a population based on a sample.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

Note:

Unlike the VAR function, VAR**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Additional info:

The variance is calculated using the "unbiased" or "n-1" method.

Example:

VARA(2, 2, 3, 4, 1) returns 1.3

See also:

VAR.S/VAR, VARPA

VARP (variance of entire population)

Note: VARP is supplemented by the new identical function VAR.P, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

VARP(Number1, Number2 [, Number3 ...])

Description:

This returns the variance of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the variance of a sample, use the VAR.S/VAR function instead. This returns an estimate of the variance.

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The variance is calculated using the "biased" or "n" method.

Example:

VARP(2, 2, 3, 4, 1) returns 1.04

See also:

VAR.P, VAR.S/VAR, STDEV.S/STDEV, STDEV.P/STDEVP

VAR.P (variance of entire population)

Syntax:

VAR.P(Number1, Number2 [, Number3 ...])

Description:

This returns the variance of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the variance of a sample, use the VAR.S/VAR function instead. This returns an estimate of the variance.

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The variance is calculated using the "biased" or "n" method.

Example:

VAR.P(2, 2, 3, 4, 1) returns 1.04

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

VARP, VAR.S/VAR, STDEV.S/STDEV, STDEV.P/STDEVP

VARPA (variance of entire population)

Syntax:

VARPA(Number1, Number2 [, Number3 ...])

Description:

The **VARPA** function is almost identical to the VARP function (see note). It returns the variance of an entire population.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated. Please note that *all* values of the entire population have to be specified. To calculate the variance of a sample, use the VARA or VAR.S/VAR function instead. This returns an estimate of the variance.

Note:

Unlike the VARP function, VARP**A** also evaluates logical values and text: FALSE is evaluated as 0, while TRUE is evaluated as 1. Text is evaluated as 0.

Empty cells, however, are also ignored here.

Example:

VARPA(2, 2, 3, 4, 1) returns 1.04

See also:

VAR.P/VARP, VAR.S/VAR, VARA

VAR.S (variance of a sample)

Syntax:

VAR.S(Number1, Number2 [, Number3 ...])

Description:

This provides an estimate of the variance of a population based on a sample.

Number1, **Number2**, etc., are single values, cell ranges or arrays that contain the values to be evaluated (sample).

If this function refers to empty cells or cells with error values, they are ignored. Likewise, cells that contain text strings or logical values, but these can be entered directly into the list of arguments and thus taken into account.

Additional info:

The variance is calculated using the "unbiased" or "n-1" method.

Example:

VAR.S(2, 2, 3, 4, 1) returns 1.3

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

VAR, VARA, VAR.P/VARP, VARPA, STDEV.S/STDEV, STDEV.P/STDEVP

VLOOKUP (look up column-wise)

Syntax:

VLOOKUP(Crit, Range, n [, Sorted])

Description:

This function searches in the first column of **Range** for the first occurrence of **Crit**. If found, the function returns the content of the cell located in the same row and **n**-th column of **Range**.

Note: VLOOKUP can only search vertically. To search horizontally, use HLOOKUP instead.

Crit is the value for which the search is performed (case-insensitive).

Range is the cell range or array to be evaluated. Its first column should contain the values to be scanned. These can be text strings, numbers or logical values.

n is the relative column number in **Range** from which a value will be returned if **Crit** is found.

If **n** is less than 1, a #VALUE! error value is returned. If **n** is greater than the total number of columns in **Range**, a #REF! error value is returned.

Sorted is a logical value that you use to specify whether the values in the first column of **Range** are sorted:

If **Sorted** is TRUE or omitted, values in the first column of **Range** have to be sorted in ascending order.

If **Sorted** is FALSE, the order of the values in the first column is irrelevant.

Important: If you do not explicitly set **Sorted** to FALSE, these values must be in sorted order!

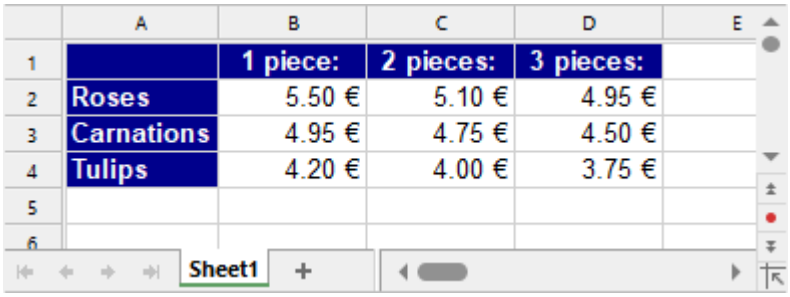
Note:

If **Sorted** is set to TRUE and VLOOKUP does not find a value that exactly matches **Crit**, the *next smaller* value is automatically taken.

If no smaller value exists because **Crit** is smaller than the smallest value available, the function returns the #N/A error value.

Example:

You sell flowers at different unit prices – depending on the amount of flowers the customer buys. To do so, you have created a table with a discount scale:



You can now use the VLOOKUP function to determine the price for a specific type of flower depending on the number of pieces.

Use the following arguments:

For **Crit**, you enter the name of the flower type (i.e., "roses", "carnations" or "tulips").

For **Range**, you enter the cell range that contains the names of the flower types in the first column and the prices to the right of them – A2:D4 here.

For **n**, you enter the number of pieces *plus one*, as the first column (**n**=1) contains the names of the flowers rather than the prices.

Examples:

VLOOKUP("Roses", A2:D4, 2, FALSE) returns the unit price when purchasing one rose, i.e., €5.50.

VLOOKUP("Roses", A2:D4, 3, FALSE) returns the unit price when purchasing two roses, i.e., €5.10.

VLOOKUP("Roses", A2:D4, 4, FALSE) returns the unit price when purchasing three roses, i.e., €4.95.

VLOOKUP("Carnations", A2:D4, 4, FALSE) returns the unit price when purchasing three carnations, i.e., €4.50.

VLOOKUP("Carnations", A2:D4, 5, FALSE) returns the #REF error value because **n** is greater than the column number of **Range**.

VLOOKUP("Petunias", A2:D4, 4, FALSE) returns the #N/A error value because "Petunias" does not appear in the first column of **Range**.

See also:

HLOOKUP, LOOKUP, INDEX, MATCH, SWITCH, CHOOSE, IFS

WEEKDAY (weekday)

Syntax:

WEEKDAY(Date [, Mode])

Description:

This returns the day of the week of a date as a number (from 1 to 7 or from 0 to 6).

Mode (optional) lets you change the type of values to be returned:

1 or omitted: Numbers from 1 (Sunday) to 7 (Saturday) are returned.

2: Numbers from 1 (Monday) to 7 (Sunday) are returned.

3: Numbers from 0 (Monday) to 6 (Sunday) are returned.

Example:

WEEKDAY("09/25/1966") returns 1 (i.e., Sunday)

WEEKDAY("09/25/1966", 2) returns 7 (i.e., Sunday)

See also:

ISOWEEKNUM, ISOWEEK, WEEKNUM, YEAR, MONTH, DAY

WEEKNUM (week number)

Syntax:

WEEKNUM(Date [, WeekStartsOn])

Description:

This returns the week number of a **Date**.

For this purpose, the calculation method commonly used in the USA and many other parts of the world is applied: The first calendar week of the year is always the week that contains January 1st.

According to this method, the first calendar week cannot start in the previous year and does not include a full 7 days in most cases.

Note:

In Germany (and some other countries like Switzerland and Austria), a different method of calculation is normally used; here, it is based on the ISO standard. See ISOWEEKNUM function (or see below **WeekStartsOn** with value variant "21").

Important:

Also note that by default, this function considers *Sunday* as the start of the week – not Monday.

However, you can change this with the optional argument **WeekStartsOn**. **WeekStartsOn** lets you specify on which day a week starts:

1 or omitted: Week starts on a Sunday.

2: Week starts on a Monday.

In addition, the following variants are also defined:

11: Week starts on a Monday.

12: Week starts on a Tuesday.

13: Week starts on a Wednesday.

14: Week starts on a Thursday.

15: Week starts on a Friday.

16: Week starts on a Saturday.

17: Week starts on a Sunday.

21: Week starts on a Monday. Here, the first calendar week of the year is the week that contains January 4th, in accordance with the ISO standard.

Compatibility notes:

Microsoft Excel supports the values 11-17 and 21 for the argument **WeekStartsOn** only in version 2010 or later. In older versions, these values are not known.

Example:

On Sunday, January 5, 2020:

WEEKNUM("01/05/2020", 1) returns 2

WEEKNUM("01/05/2020", 2) returns 1

See also:

ISOWEEKNUM, ISOWEEK, YEAR, MONTH, DAY, WEEKDAY

WEIBULL (Weibull distribution)

Note: WEIBULL is supplemented by the new identical function WEIBULL.DIST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

WEIBULL(x, Alpha, Beta, Cumulative)

Description:

This returns the probabilities of a Weibull distributed random variable.

The Weibull distribution is an extension of the exponential distribution (see function EXPON.DIST/EXPONDIST) to events that are not purely random and do not exactly match the Poisson distribution (see function POISSON.DIST/POISSON).

It is used especially for reliability analysis.

x is the value whose probability is to be calculated. **x** must be ≥ 0.

Alpha is a parameter of the distribution. **Alpha** determines the shape of the distribution curve. If Alpha = 1, the Weibull distribution equals an exponential distribution with Lambda = 1/**Beta**.

Beta is another parameter of the distribution. **Beta** determines the scaling of the distribution curve, i.e., it stretches or compresses the function graph.

Both parameters have to be greater than zero.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Example:

WEIBULL(42, 2, 100, TRUE) returns 0.16172

WEIBULL(42, 2, 100, FALSE) returns 0.00704

See also:

WEIBULL.DIST, EXPON.DIST/EXPONDIST, POISSON.DIST/POISSON

WEIBULL.DIST (Weibull distribution)

Syntax:

WEIBULL.DIST(x, Alpha, Beta, Cumulative)

Description:

This returns the probabilities of a Weibull distributed random variable.

The Weibull distribution is an extension of the exponential distribution (see function EXPON.DIST/EXPONDIST) to events that are not purely random and do not exactly match the Poisson distribution (see function POISSON.DIST/POISSON).

It is used especially for reliability analysis.

x is the value whose probability is to be calculated. **x** must be ≥ 0.

Alpha is a parameter of the distribution. **Alpha** determines the shape of the distribution curve. If Alpha = 1, the Weibull distribution equals an exponential distribution with Lambda = 1/**Beta**.

Beta is another parameter of the distribution. **Beta** determines the scaling of the distribution curve, i.e., it stretches or compresses the function graph.

Both parameters have to be greater than zero.

You use the logical value **Cumulative** to specify which type of function will be returned:

FALSE: The probability density function is returned.

TRUE: The cumulative distribution function is returned.

Example:

WEIBULL.DIST(42, 2, 100, TRUE) returns 0.16172

WEIBULL.DIST(42, 2, 100, FALSE) returns 0.00704

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

WEIBULL, EXPON.DIST/EXPONDIST, POISSON.DIST/POISSON

WORKDAY (date after x workdays)

Syntax:

WORKDAY(StartDate, Days [, Holidays])

Description:

This determines the date that is *x* workdays before or after the **StartDate**.

This function counts workdays only, thus skipping Saturdays and Sundays. You also have the option of specifying a list of holidays to be skipped as well (see **Holidays** parameter).

StartDate is the start date.

Days is the number of workdays. A positive value will return a date in the future, while a negative value will return a date in the past.

Holidays (optional) is a cell range or an array containing a list of dates to be skipped (for example, holidays). (This is usually done using a cell range or an array containing the corresponding date values.)

Example:

WORKDAY("10/1/2016", 9) returns 10/13/2016.

The date that comes 9 work days after 10/1/2016 is thus 10/13/2016.

However, 10/3/2016, for example, was a holiday in Germany. If you wanted to take this into account, you would have to write:

WORKDAY("10/1/2016", 9, "10/3/2016") – which results in 10/14 instead of 10/13, since the holiday on 10/3 fell on a weekday and thus is skipped in the count.

See also:

HOLIDAY, NETWORKDAYS

XIRR (internal rate of return)

Syntax:

XIRR(Values, Dates [, Guess])

Description:

This returns an estimate of the internal rate of return of an investment with *irregular* cash flows.

The calculation is based on the cash flows specified in the **Values** argument. Payments have to be entered as negative values, and income as positive values. Payments may be made at any time. The date of each cash flow has to be specified using the **Dates** argument.

The function expects the following arguments:

For **Values**, you specify a cell range or array in which the cash flows were entered. There must be at least one payment and one income.

Dates is a cell range or array containing the dates for each cash flow specified in **Values**.

Guess (optional) lets you specify an estimate for the result (see note below).

Note:

The result of this function is calculated using an iterative search technique. If the function does not find a result with an accuracy of 0.00001% or better after 20 iterations, it aborts and the #NUM! error value is returned. If this occurs, try specifying a different value for **Guess**.

See also:

IRR, NPV, MIRR, XNPV, RATE

XNPV (net present value)

Syntax:

XNPV(Rate, Values, Dates)

Description:

This returns the net present value of an investment with *irregular* cash flows.

The calculation is based on the cash flows specified in the **Values** argument. Payments have to be entered as negative values, and income as positive values. Payments may be made at any time. The date of each cash flow has to be specified using the **Dates** argument.

The function expects the following arguments:

Rate is the discount rate to apply to the cash flows.

For **Values**, you specify a cell range or array in which the cash flows were entered. There must be at least one payment and one income.

Dates is a cell range or array containing the dates for each cash flow specified in **Values**.

See also:

NPV, XIRR

XOR (logical XOR function)

Syntax:

XOR(Value1 [, Value2, Value3 ...])

Description:

XOR ("exclusive or") returns the logical value TRUE if an *odd* number of the arguments are TRUE, otherwise it returns FALSE.

This function is often used with only two arguments. Here, the following applies: If *either* one *or* the other argument is TRUE, it returns TRUE. On the other hand, if *both* are TRUE or *both* are FALSE, it returns FALSE.

Compatibility notes:

Microsoft Excel supports this function only in version 2013 or later. The function is unknown in older versions.

Example:

XOR(TRUE, TRUE) returns FALSE

XOR(TRUE, FALSE) returns TRUE

XOR(FALSE, TRUE) returns TRUE

XOR(FALSE, FALSE) returns FALSE

See also:

OR, AND, NOT

YEAR (year of a date)

Syntax:

YEAR(Date)

Description:

This returns the year of a date.

Example:

YEAR("09/25/2018") returns 2018

If cell A1 contains the date value 9/25/2018: YEAR(A1) returns 2018

See also:

ISOWEEKNUM, ISOWEEK, WEEKNUM, MONTH, DAY, DAYSPERYEAR, WEEKDAY

ZTEST (z-test)

Note: ZTEST is supplemented by the new identical function Z.TEST, which is available in newer versions of Microsoft Excel (2010 or later).

Syntax:

ZTEST(Area, x [, Sigma])

Description:

This returns the one-tailed probability of a z-test.

You can use this function to check the probability that a sample comes from a certain population.

Area is a cell reference to the area against which **x** is to be tested. There should be at least 30 values.

x is the value to be tested.

Sigma (optional) is the standard deviation σ of the population (if known). If omitted, PlanMaker automatically uses the standard deviation of the given sample (i.e., the values in **Area**).

See also:

Z.TEST, STANDARDIZE, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

Z.TEST (z-test)

Syntax:

Z.TEST(Area, x [, Sigma])

Description:

This returns the one-tailed probability of a z-test.

You can use this function to check the probability that a sample comes from a certain population.

Area is a cell reference to the area against which **x** is to be tested. There should be at least 30 values.

x is the value to be tested.

Sigma (optional) is the standard deviation σ of the population (if known). If omitted, PlanMaker automatically uses the standard deviation of the given sample (i.e., the values in **Area**).

Compatibility notes:

Microsoft Excel supports this function only in version 2010 or later. The function is unknown in older versions.

See also:

ZTEST, STANDARDIZE, NORM.DIST/NORMDIST, NORM.S.DIST/NORMSDIST

Appendix

The appendix of the manual contains the following information:

- **Ribbon commands and corresponding menu commands**

  In this section, you will find a table of all commands in the ribbon interface and the respective corresponding menu command in the classic menu interface.

- **Shortcut keys**

  This section contains tables with the most important shortcut keys that you can use in the program.

- **Command-line parameters**

  Here, you will find an overview of parameters that can be used to directly trigger a specified program option when you start PlanMaker.

Ribbon commands and corresponding menu commands

In this section, you will find a table of all commands in the ribbon interface and the respective corresponding menu command in the classic menu interface.

Tip 1: You can switch the user interface between *Ribbon* and *Classic menus* at any time. To do so, choose the command **File | Options** in the ribbon (or choose **Tools > Options** in the classic menu interface). In the dialog box, switch to the **Appearance** tab and click on the **User interface** button. The program will display a dialog box in which you can choose the desired user interface.

Tip 2: Use the "hamburger menu" (the icon ☰ on the left in the Quick access toolbar) if you still want to access the classic menu commands from the ribbon interface.

The subsequent table contains the following columns:

- **Left column:** The left column lists all **ribbon commands** in the program, listed according to the order of ribbon tabs and in the following format: **Ribbon tab | Command group | Command**
- **Right column:** The right column lists all corresponding **classic menu commands** in the program in the following format: **Menu > Command**

  Example: The ribbon command **File | Document | Save** can be found under **File > Save** in the classic menu interface.

  Another entry >> is also added for some commands if the command can only be found in the submenu of an icon or in a dialog box.

Thus, here is the aforementioned table:

	Ribbon	Menu
	File | File | New	File > New
	File | File | Open	File > Open
	File | File | Close	File > Close
	File | Document | Save	File > Save
	File | Document | Save as	File > Save as
	File | Document | Save all	File > Save all
	File | Document | PDF export	File > Export as PDF
	File | Document | Send	File > Send
	File | Print | Page setup	File > Page setup
	File | Print | Define print range	File > Print range > Define print range
	File | Print | Define print range >> Remove print range	File > Print range > Remove print range
	File | Print | Print preview	File > Print preview
	File | Print | Print	File > Print
	File | File management | Properties	File > Properties
	File | File management | Versions	File > File versions
	File | File management | File manager	File > File manager
	File | File management | Recalculate >> Recalculate workbook	Tools > Recalculate
	File | File management | Recalculate >> Recalculate sheet	*Shift+F9*
	File | File Management | Compress all pictures	Object > Compress all pictures
	File | Scripts | Edit script	Tools > Edit script
	File | Scripts | Run script	Tools > Run script

	Ribbon	Menu
	File | Settings | Options	Tools > Options
	File | Settings | Customize >> Customize ribbon	Tools > Customize
	File | File | Exit (if no document is open)	File > Exit
	in the Quick access toolbar	Edit > Undo
	in the Quick access toolbar	Edit > Redo
	in the Quick access toolbar	Edit > Repeat
	in the Quick access toolbar	View > Touch mode
	in the Quick access toolbar	View > Dark document mode
	Home | Edit | Paste	Edit > Paste
	Home | Edit | Paste >> Paste special	Edit > Paste special
	Home | Edit | Cut	Edit > Cut
	Home | Edit | Copy	Edit > Copy
	Home | Edit | Format painter	Format > Transfer formatting
	Home | Character | Bold	*Toolbar: Formatting*
	Home | Character | Italic	*Toolbar: Formatting*
	Home | Character | Underline	*Toolbar: Formatting*
	Home | Character | Font color	*Toolbar: Formatting*
	Home | Character | Increase font size	
	Home | Character | Decrease font size	
	Home | Alignment | Align top	
	Home | Alignment | Cell orientation	
	Home | Alignment | Line wrap	*Toolbar: Formatting*

	Ribbon	Menu
	Home | Alignment | Default alignment	*Toolbar: Formatting*
	Home | Alignment | Merge cells	*Toolbar: Formatting*
	Home | Number | Percent format	
	Home | Number | Currency format	
	Home | Number | Increase decimal places	*Toolbar: Formatting*
	Home | Number | Decrease decimal places	*Toolbar: Formatting*
	Home | Format | Shading	Format > Shading
	Home | Format | Borders	Format > Borders
	Home | Format | Conditional formatting	*Toolbar: Formatting*
	Home | Format | Conditional formatting >> New rule	Format > Conditional formatting > New rule
	Home | Format | Conditional formatting >> Manage rules	Format > Conditional formatting > Manage rules
	Home | Format | Conditional formatting >> Delete rules in selected cells	Format > Conditional formatting > Delete rules in selected cells
	Home | Format | Cell styles	
	Home | Format | Cell styles >> Manage cell styles	Format > Cell style
	Home | Cells | Insert	Worksheet > Insert cells
	Home | Cells | Insert >> Insert copied cells	Worksheet > Insert copied cells
	Home | Cells | Insert >> Insert worksheet	Worksheet > Worksheet > Insert
	Home | Cells | Delete	Worksheet > Delete cells
	Home | Cells | Delete >> Delete worksheet	Worksheet > Worksheet > Delete
	Home | Cells | Visibility	
	Home | Cells | Visibility >> Hide rows	Worksheet > Row > Hide
	Home | Cells | Visibility >> Show rows	Worksheet > Row > Show

	Ribbon	Menu
	Home | Cells | Visibility >> Hide columns	Worksheet > Column > Hide
	Home | Cells | Visibility >> Show columns	Worksheet > Column > Show
	Home | Cells | Visibility >> Hide sheet	Worksheet > Worksheet > Hide
	Home | Cells | Visibility >> Show sheet	Worksheet > Worksheet > Show
	Home | Contents | Sort and filter	*Toolbar: Standard*
	Home | Contents | Sort and filter >> Sort	Worksheet > Sort
	Home | Contents | Sort and filter >> Sort ascending	*Toolbar: Standard*
	Home | Contents | Sort and filter >> Sort descending	*Toolbar: Standard*
	Home | Contents | Sort and filter >> AutoFilter	Worksheet > Filter > AutoFilter
	Home | Contents | Sort and filter >> Show all	Worksheet > Filter > Show all
	Home | Contents | Sort and filter >> Reapply filter	Worksheet > Filter > Reapply filter
	Home | Contents | Sort and filter >> Special filter	Worksheet > Filter > Special filter
	Home | Contents | Autosum	*Toolbar: Standard*
	Home | Contents | Fill	Edit > Fill
	Home | Contents | Fill >> Down	Edit > Fill > Down
	Home | Contents | Fill >> Right	Edit > Fill > Right
	Home | Contents | Fill >> Up	Edit > Fill > Up
	Home | Contents | Fill >> Left	Edit > Fill > Left
	Home | Contents | Fill >> Series	Edit > Fill > Series
	Home | Contents | Delete	Edit > Delete special
	Home | Contents | Delete >> All	Edit > Delete special > All
	Home | Contents | Delete >> Formatting	Edit > Delete special > Formatting

	Ribbon	Menu
	Home | Contents | Delete >> Contents	Edit > Delete special > Contents
	Home | Contents | Delete >> Comments	Edit > Delete special > Comments
	Home | Contents | Delete >> Conditional formatting	Edit > Delete special > Conditional formatting
	Home | Contents | Delete >> Input validation	Edit > Delete special > Input validation
	Home | Contents | Delete >> Reset character formatting	Format > Standard
	Home | Contents | Delete >> Remove link	Format > Remove link
	Home | Search | Search	Edit > Search
	Home | Search | Replace	Edit > Replace
	Home | Search | Search again	Edit > Search again
	Home | Search | Go to	Edit > Go to
	Home | Selection | Select all	Edit > Select all
	Insert | Tables | Table	Worksheet > New table
	Insert | Tables | Pivot table	Worksheet > New pivot table
	Insert | Tables | Sheet >> Insert	Worksheet > Worksheet > Insert
	Insert | Tables | Sheet >> Copy	Worksheet > Worksheet > Copy
	Insert | Tables | Sheet >> Move	Worksheet > Worksheet > Move
	Insert | Tables | Sheet >> Delete	Worksheet > Worksheet > Delete
	Insert | Tables | Sheet >> Rename	Worksheet > Worksheet > Rename
	Insert | Tables | Sheet >> Show	Worksheet > Worksheet > Show
	Insert | Tables | Sheet >> Hide	Worksheet > Worksheet > Hide
	Insert | Tables | Sheet >> Properties	Worksheet > Worksheet > Properties
	Insert | Objects | Picture frame	Object > New picture frame

	Ribbon	Menu
	Insert | Objects | Picture frame >> From scanner	File > Acquire
	Insert | Objects | Picture frame >> Select source	File > Select source
	Insert | Objects | Text frame	Object > New text frame
	Insert | Objects | Chart frame	Object > New chart frame
	Insert | Objects | AutoShape	Object > New drawing > AutoShape
	Insert | Objects | Lines	Object > New drawing > Line
	Insert | Objects | TextArt object	Object > New drawing > TextArt
	Insert | Objects | QR code/Barcode frame	Object > New QR code/Barcode frame
	Insert | Objects | Equation Editor object	Object > New OLE object frame >> SoftMaker Equation Editor
	Insert | Objects | Form object >> Checkbox	Object > New form object > Checkbox
	Insert | Objects | Form object >> Radio button	Object > New form object > Radio button
	Insert | Objects | Form object >> Dropdown	Object > New form object > Dropdown
	Insert | Objects | Form object >> Listbox	Object > New form object > Listbox
	Insert | Objects | Form object >> Pushbutton	Object > New form object > Pushbutton
	Insert | Objects | Form object >> Spinner	Object > New form object > Spinner
	Insert | Objects | Form object >> Scrollbar	Object > New form object > Scrollbar
	Insert | Objects | Form object >> Label	Object > New form object > Label
	Insert | Objects | Form object >> Groupbox	Object > New form object > Groupbox
	Insert | Objects | OLE object frame	Object > New OLE object frame
	Insert | Objects | OLE object frame >> TextMaker object	Object > New OLE object frame >> TextMaker Document
	Insert | Objects | OLE object frame >> Presentations object	Object > New OLE object frame >> Presentations Slide

Ribbon	Menu
Insert | Links | Hyperlink	Format > Link
Insert | Comments | Comment	Insert > Comment
Insert | Text | Header / footer	Insert > Header and footer
Insert | Text | SmartText	Insert > SmartText
Insert | Text | Symbol	Insert > Symbol
Insert | Text | Character	
Layout | Page setup | Page margins	File > Page setup
Layout | Page setup | Page orientation	File > Page setup
Layout | Page setup | Page size	File > Page setup
Layout | Page setup | Page break	Insert > Page break
Layout | Page Setup | Header / footer	Insert > Header and footer
Layout | Cells | Cell size: Row height	Worksheet > Row > Height
Layout | Cells | Cell size: Column width	Worksheet > Column > Width
Layout | Cells | Optimum height	Worksheet > Row > Optimum height
Layout | Cells | Optimum width	Worksheet > Column > Optimum width
Layout | Position | Bring to front	Object > Order > Bring to front
Layout | Position | Bring to front >> Bring forward one level	Object > Order > Bring forward one level
Layout | Position | Send to back	Object > Order > Send to back
Layout | Position | Send to back >> Send backward one level	Object > Order > Send backward one level
Layout | Position | Rotate object	Object > Rotate or flip
Layout | Position | Align objects	Object > Align or distribute
Layout | Objects | Group	Object > Group

	Ribbon	Menu
	Layout | Objects | Duplicate	Edit > Duplicate
	Formula | Function | Function	Insert > Function
	Formula | Function | Autosum	*Toolbar: Standard*
	Formula | Function | Recently used	Insert > Function >> Category
	Formula | Function | Financial	Insert > Function >> Category
	Formula | Function | Logic	Insert > Function >> Category
	Formula | Function | Strings	Insert > Function >> Category
	Formula | Function | Date and time	Insert > Function >> Category
	Formula | Function | Statistics	Insert > Function >> Category
	Formula | Function | Mathematics	Insert > Function >> Category
	Formula | Function | More functions	Insert > Function >> Category
	Formula | Named areas | Edit names	Worksheet > Names > Edit
	Formula | Named areas | Create names	Worksheet > Names > Import
	Formula | Named areas | Apply names	Worksheet > Names > Apply
	Formula | Named areas | Export names	Worksheet > Names > Insert list
	Formula | Traces | Trace to error	Tools > Formula auditing > Trace to error
	Formula | Traces | Trace predecessors	Tools > Formula auditing > Trace predecessors
	Formula | Traces | Trace predecessors >> Select predecessors	Tools > Formula auditing > Select predecessors
	Formula | Traces | Trace successors	Tools > Formula auditing > Trace successors
	Formula | Traces | Trace successors >> Select successors	Tools > Formula auditing > Select successors
	Formula | Traces | Remove all traces	Tools > Formula auditing > Remove all traces

	Ribbon	Menu
	Formula | Traces | Remove all traces >> Remove traces to predecessors	Tools > Formula auditing > Remove traces to predecessors
	Formula | Traces | Remove all traces >> Remove traces to successors	Tools > Formula auditing > Remove traces to successors
	Formula | Traces | Previous error	Tools > Formula auditing > Go to previous error
	Formula | Traces | Next error	Tools > Formula auditing > Go to next error
	Formula | Watch | Show watch list	View > Watch window
	Formula | Update | Update data >> Update charts	Tools > Update charts
	Formula | Update | Update data >> Update external references	Tools > Update external references
	Formula | Update | Update data >> Update pivot tables	Tools > Refresh all pivot tables
	Formula | Update | Recalculate >> Recalculate workbook	Tools > Recalculate
	Formula | Update | Recalculate >> Recalculate sheet	*Shift+F9*
	Data | Filter | Sort	Worksheet > Sort
	Data | Filter | Ascending	*Toolbar: Standard*
	Data | Filter | Descending	*Toolbar: Standard*
	Data | Filter | AutoFilter	Worksheet > Filter > AutoFilter
	Data | Filter | Special filter	Worksheet > Filter > Special filter
	Data | Filter | Show all	Worksheet > Filter > Show all
	Data | Filter | Reapply AutoFilter	Worksheet > Filter > Reapply filter
	Data | Edit | External references	Worksheet > External references
	Data | Edit | Transpose	Worksheet > Transpose
	Data | Edit | Text to columns	Worksheet > Text to columns
	Data | Edit | Remove empty rows	Worksheet > Remove > Empty rows

	Ribbon	Menu
	Data | Edit | Remove empty rows >> Remove duplicate rows	Worksheet > Remove > Duplicate rows
	Data | Analyze | Pivot table	Worksheet > New pivot table
	Data | Analyze | Data consolidation	Worksheet > Data consolidation
	Data | Analyze | Scenario manager	Tools > Scenarios
	Data | Analyze | Goal seek	Tools > Goal seek
	Data | Outliner | Group	Worksheet > Outliner > Group
	Data | Outliner | Ungroup	Worksheet > Outliner > Ungroup
	Data | Outliner | Remove all groups	Worksheet > Outliner > Clear outline
	Data | Outliner | Show details	Worksheet > Outliner > Show details
	Data | Outliner | Hide details	Worksheet > Outliner > Hide details
	Data | Input validation | Input validation	Format > Input validation
	Data | Input validation | Previous invalid cell	Tools > Formula auditing > Go to previous invalid cell
	Data | Input validation | Next invalid cell	Tools > Formula auditing > Go to next invalid cell
	Review | Spelling | Spell check	Tools > Check spelling
	Review | Spelling | Spell check >> Edit user dictionaries	Tools > Edit user dictionaries
	Review | Spelling | Research	*Toolbar: Standard*
	Review | Spelling | Auto correction	Tools > Auto correction
	Review | Comments | Add	Tools > Comments > Insert comment
	Review | Comments | Edit	Tools > Comments > Edit comment
	Review | Comments | Delete	Next > Comments > Delete comments
	Review | Comments | Previous comment	Tools > Comments > Previous comment
	Review | Comments | Next comment	Tools > Comments > Next comment

	Ribbon	Menu
	Review | Comments | Comment visibility >> Show all comments	Tools > Comments > Show all comments
	Review | Comments | Comment visibility >> Show this comment	Tools > Comments > Show this comment
	Review | Comments | Comment visibility >> Restore comment positions	Tools > Comments > Restore comment positions
	Review | Protection | Cell protection	Format > Cell >> Protection tab: Protect cell
	Review | Protection | Hide formula	Format > Cell >> Protection tab: Hide formula
	Review | Protection | Sheet protection	Tools > Sheet protection
	Review | Protection | Workbook protection	Tools > Workbook protection
	Review | Speech | Speak	Tools > Text to speech
	Review | Input validation | Input validation	Format > Input validation
	Review | Input validation | Mark invalid data	Tools > Formula auditing > Mark invalid data
	Review | Input validation | Remove marks	Tools > Formula auditing > Remove invalid data marks
	Review | Input validation | Previous invalid cell	Tools > Formula auditing > Go to previous invalid cell
	Review | Input validation | Next invalid cell	Tools > Formula auditing > Go to next invalid cell
	View | Mode | Object mode	View > Object mode
	View | View | Row and column headers	View > Row & column headers
	View | View | Show formulas	View > Show formulas
	View | View | Syntax highlighting	View > Syntax highlighting
	View | View | Show zero values	Worksheet > Worksheet > Properties >> Show zero values
	View | View | Show sheet grid	Worksheet > Worksheet > Properties >> Gridlines
	View | View | Show page breaks	Worksheet > Worksheet > Properties >> Page breaks

	Ribbon	Menu
	View | View | Page break preview	View > Page break preview
	View | Zoom | Set zoom	View > Zoom level
	View | Zoom | Actual size	View > Actual size
	View | Zoom | Fit to selection	View > Zoom level >> Fit to selection
	View | Windows | Freeze cells	View > Freeze
	View | Windows | Pivot table sidebar	View > Pivot table sidebar
	View | Windows | Pivot table sidebar >> Show at left	View > Pivot table sidebar > Show at left
	View | Windows | Pivot table sidebar >> Show at right	View > Pivot table sidebar > Show at right
	View | Windows | Full screen	View > Full screen
	View | Windows | Windows	Window >
	View | Windows | Windows >> View side by side	Window > View side by side
	View | Windows | Windows >> Close all	Window > Close all
	Table | Table | Range	Worksheet > Table > Table range
	Table | Table | Properties	Worksheet > Table > Table settings
	Table | Table | Summarize as pivot	Worksheet > Table > Create pivot table
	Table | Table | Convert to range	Worksheet > Table > Convert to range
	Table | Table | Delete table	Worksheet > Table > Delete table
	Table | Selection | Select rows	Worksheet > Table > Select > Select rows
	Table | Selection | Select columns	Worksheet > Table > Select > Select columns
	Table | Selection | Select table	Worksheet > Table > Select > Select table
	Table | Edit | Insert above	Worksheet > Table > Insert > Insert rows above
	Table | Edit | Insert below	Worksheet > Table > Insert > Insert rows below

Ribbon	Menu
Table | Edit | Delete rows	Worksheet > Table > Delete > Delete rows
Table | Edit | Insert left	Worksheet > Table > Insert > Insert columns at left
Table | Edit | Insert right	Worksheet > Table > Insert > Insert columns at right
Table | Edit | Delete columns	Worksheet > Table > Delete > Delete columns
Table | Style | Table styles	*Toolbar: Table*
Table | Style | Header row	Worksheet > Table > Header row
Table | Style | Total row	Worksheet > Table > Total row
Table | Style | First column	Worksheet > Table > Table settings
Table | Style | Last column	Worksheet > Table > Table settings
Table | Style | Banded rows	Worksheet > Table > Table settings
Table | Style | Banded columns	Worksheet > Table > Table settings
Object | Insert | New object	*Toolbar: Standard*
Object | Format | AutoShape templates	*Toolbar: AutoShape*
Object | Format | AutoShape templates >> More	Object > Properties
Object | Format | AutoShape effects	*Toolbar of the respective object*
Object | Format | Fill color	*Toolbar of the respective object*
Object | Format | Line color	*Toolbar of the respective object*
Object | Format | Line style	*Toolbar of the respective object*
Object | Format | Line thickness	*Toolbar of the respective object*
Object | Format | Line endings	*Toolbar: AutoShape*
Object | Edit | Change AutoShape	*Toolbar: AutoShape*
Object | Edit | Edit points	*Toolbar: AutoShape*

Ribbon	Menu
Object | Edit | Combine objects	*Via the context menu*
Object | Inner text | Text rotation	*Toolbar: AutoShape*
Object | Inner text | Vertical alignment	*Toolbar: AutoShape*
Object | Inner text | Change text area	*Toolbar: AutoShape*
Object | Inner text | Add/edit data link	Object > Link to a cell
OLE object | OLE | OLE object	Edit > OLE object
OLE object | OLE | Edit link	Edit > Link
OLE object | Size | Cropping	*Toolbar: Picture*
OLE object | Adjust | Recolor picture	*Toolbar: Picture*
Picture | Adjust | Compress pictures	Object > Compress selected pictures
Picture | Adjust | Change source	Object > Properties >> Picture tab: File...
Picture | Adjust | Reset	*Toolbar: Picture*
Picture | Format | Photo frame	*Toolbar: Picture*
Chart | Type | Chart type	*Toolbar: Chart*
Chart | Type | Chart subtype	*Toolbar: Chart*
Chart | Data | Series in columns	Object > Chart > Data in columns
Chart | Data | Series in rows	Object > Chart > Data in rows
Chart | Design | Chart colors	*Toolbar: Chart*
Chart | Chart elements | Add chart element	*Toolbar: Chart*
Chart | Chart elements | Select chart element	*Toolbar: Chart*
Chart | Chart elements | Edit properties	Object > Chart > xy element: Properties
Chart | Chart elements | Fill color	

Ribbon	Menu
Chart | Chart elements | Line color	
Chart | Chart elements | Line style	
Chart | Chart elements | Line thickness	
Chart | Position | Chart location	Object > Chart > Change chart location
Chart | Export | Save chart as a picture	Object > Chart > Save as picture
not available	Edit > Delete
Home | Number >> group arrow	Format > Cell
Home | Character >> group arrow	Format > Character
Home | Alignment >> group arrow	Format > Paragraph
Via the context menu	Format > Character style
not available	Format > AutoFormat
Via the context menu of the worksheet register	Worksheet > Worksheet > Show > Show all
Data | Outliner >> group arrow	Worksheet > Outliner > Automatically show outline pane
Data | Outliner >> group arrow	Worksheet > Outliner > Options
Via the context menu	Worksheet > Table > Select > Select column data
not available	Tools > Edit lists

Shortcut keys

The following pages contain tables with Planmaker's most important shortcut keys:

- Shortcut keys in Windows and Linux versions
- Shortcut keys in Mac version

Tip: If you choose the ribbon command **File | Customize** , you can customize the default shortcut keys and create new shortcut keys. For more information, see Customizing shortcut keys.

Shortcut keys in Windows and Linux versions

In the **Windows** and the **Linux** version, the following shortcut keys are available:

Shortcut keys for commands

Command	Shortcut key
Quick access toolbar: Undo last command	**Ctrl+Z**
Quick access toolbar: Redo	**Ctrl+Y**
Quick access toolbar: Repeat last command	**Ctrl+Shift+Y**
**File	New**	**Ctrl+N**
**File	Open**	**Ctrl+O**
**File	Close**	**Ctrl+W** or **Ctrl+F4**
**File	Save**	**Ctrl+S**
**File	Print**	**Ctrl+P**
**File	File manager**	**F12**
**File	Exit**	**Ctrl+Q** or **Alt+F4**
**Home	Paste**	**Ctrl+V**
**Home	Paste Special**	**Ctrl+Alt+V**
**Home	Cut**	**Ctrl+X**

Command	Shortcut key
Home | **Copy**	**Ctrl+C**
Home | **Format painter** (copying step)	**Ctrl+Shift+C**
Home | **Format painter** (pasting step)	**Ctrl+Shift+V**
Home | group **Character:**	
Open the "Character" dialog box	**Ctrl+2**
Select font from the list	**Ctrl+D**
Bold on/off	**Ctrl+B**
Italics on/off	**Ctrl+I**
Underline on/off	**Ctrl+U**
Superscript	**Ctrl+Shift+Plus key** (numeric keypad)
Subscript	**Ctrl+Shift+Minus key** (numeric keypad)
Remove superscript or subscript	**Ctrl+Shift+Multiply key** (numeric keypad)
Change uppercase/lowercase of selected text (from "test" to "Test" to "TEST")	**Shift+F3**
Home | group **Alignment:**	
Open the "Cell" dialog box	**Ctrl+1**
Alignment: Left	**Ctrl+L**
Alignment: Centered	**Ctrl+E**
Alignment: Right	**Ctrl+R**
Alignment: Justified	**Ctrl+J**
Line break for current cell	**Ctrl+Enter**
Home | group **Number:**	
Open the "Cell" dialog box	**Ctrl+1**
Number format: Number	**Ctrl+Shift+1**
Number format: Scientific	**Ctrl+Shift+2**
Number format: Currency	**Ctrl+Shift+4**
Number format: Percentage	**Ctrl+Shift+5**

Command	Shortcut key
Home| group **Cells:**	
Insert cells	**Ctrl+Plus**
Delete cells	**Ctrl+Minus**
Hide current row	**Ctrl+9**
Show current row	**Ctrl+Shift+9**
Hide current column	**Ctrl+8**
Show current column	**Ctrl+Shift+8**
Home | Search	**Ctrl+F**
Home | Replace	**Ctrl+H**
Home | Search again (or **Replace again**)	**F3**
Home | Go to	**F5 or Ctrl+G**
Go to the next error	**Ctrl+F3**
Home| Select all	**Ctrl+A**
Insert | Table	**Ctrl+T**
Insert | Hyperlink	**Ctrl+K**
Insert | Comment	**Shift+F2**
Insert | Character | Em dash (—)	**Ctrl+Alt+Minus key** (numeric keypad)
Insert | Character | Non-breaking space	**Ctrl+Shift+Space**
Formula | Insert function	**F7**
Formula | Update data:	
Updating charts	**F8**
Update calculations	**F9**
Recalculate the current worksheet only	**Shift+F9**
View | Full screen	**F6**

Other useful shortcut keys

Command	Shortcut key
Display help content	**F1**
Edit the content of the current cell	**F2**
To the next open document	**Ctrl+F6**
To the previous open document	**Ctrl+Shift+F6**
Show/hide ribbon	**Ctrl+F1**
Move to the next ribbon tab	**Ctrl+F12**
Move to the previous ribbon tab	**Ctrl+Shift+F12**
Move to the next worksheet (for open dialog boxes: move to the next tab)	**Ctrl+Tab**
Move to the previous worksheet (for open dialog boxes: move to the previous tab)	**Ctrl+Shift+Tab**
Select the next object (in object mode)	**Tab**
Select the previous object (in object mode)	**Shift+Tab**
Move to the next cell	**←/→/↑/↓**
Move to the next / previous cell horizontally	**Tab / Shift+Tab**
Move to the next / previous cell vertically	**Enter / Shift+Enter**
Move to the next *filled* cell	**Ctrl+←/→/↑/↓**
Move to the first cell in the current column	**Ctrl+PageUp↑**
Move to the last cell in the current column	**Ctrl+PageDown↓**
Move to the first cell in the current row	**Home**
Move to the last *filled* cell in the current row	**End**
Move to the first cell in the worksheet (cell A1)	**Ctrl+Home**
Move to the last *filled* cell in the worksheet	**Ctrl+End**
Move to the next circular reference	**Shift+F5**
Go to the next error	**Ctrl+F3**

Command	Shortcut key
Copy cell above	**Ctrl+,** (comma)
Copy value from cell above	**Ctrl+Shift+,** (comma)
Enter an array formula	**Ctrl+Shift+**↵
Select current array formula	**Ctrl+7**
Select current conditional formatting	**Ctrl+6**
Open a dropdown list with the contents of the cells above or below a cell	**Alt+Shift+**↓
Switch between AUTO and TEXT input mode (see Status bar)	**Ctrl+Shift+F4**
Insert current date	**Ctrl+.** (period)
Insert current time	**Ctrl+Shift+.** (period)
Convert a hexadecimal character code into Unicode characters*	**Ctrl+Alt+Shift+X**

* If, for example, you type 20AC and then press this shortcut key, a euro sign € will appear (since this character has the code 20AC in the Unicode character set).

Shortcut keys in Mac version

Note: For most shortcut keys with the **Cmd** key, you can also use the **Ctrl** key instead of the **Cmd** key. For the shortcut key **Cmd+S**, for example, you can also press **Ctrl+S**.

In the **Mac** version, the following shortcut keys are available:

Shortcut keys for commands

Command	Shortcut key
Quick access toolbar: Undo last command	**Cmd+Z**
Quick access toolbar: Redo	**Cmd+Y**
Quick access toolbar: Repeat last command	**Cmd+Shift+Y**
File | New	**Cmd+N**
File | Open	**Cmd+O**

Command	Shortcut key
File | Close	**Cmd+W** or **Cmd+F4**
File | Save	**Cmd+S**
File | Print	**Cmd+P**
File | File manager	**F12**
File | Exit	**Cmd+Q** or **Alt+F4**
Home | Paste	**Cmd+V**
Home | Paste Special	**Cmd+Alt+V**
Home | Cut	**Cmd+X**
Home | Copy	**Cmd+C**
Home | Format painter (copying step)	**Cmd+Shift+C**
Home | Format painter (pasting step)	**Cmd+Shift+V**
Home | group **Character:**	
Open the "Character" dialog box	**Cmd+2**
Select font from the list	**Cmd+D**
Bold on/off	**Cmd+B**
Italics on/off	**Cmd+I**
Underline on/off	**Cmd+U**
Superscript	**Cmd+Shift+Plus key** (numeric keypad)
Subscript	**Cmd+Shift+Minus key** (numeric keypad)
Remove superscript or subscript	**Cmd+Shift+Multiply key** (numeric keypad)
Change uppercase/lowercase of selected text (from "test" to "Test" to "TEST")	**Shift+F3**
Home | group **Alignment:**	
Open the "Cell" dialog box	**Cmd+1**
Alignment: Left	**Cmd+L**
Alignment: Centered	**Cmd+E**
Alignment: Right	**Cmd+R**

Command	Shortcut key
Alignment: Justified	**Cmd+J**
Line break for current cell	**Cmd+Enter**
Home | group **Number:**	
Open the "Cell" dialog box	**Cmd+1**
Number format: Number	**Cmd+Shift+1**
Number format: Scientific	**Cmd+Shift+2**
Number format: Currency	**Cmd+Shift+4**
Number format: Percentage	**Cmd+Shift+5**
Home| group **Cells:**	
Insert cells	**Cmd+Plus**
Delete cells	**Cmd+Minus**
Hide current row	**Cmd+9**
Show current row	**Cmd+Shift+9**
Hide current column	**Cmd+8**
Show current column	**Cmd+Shift+8**
Home | Search	**Cmd+F**
Home | Replace	**Cmd+H**
Home | Search again (or **Replace again**)	**F3**
Home | Go to	**Cmd+G** or **F5**
Go to the next error	**Cmd+F3**
Home| Select all	**Ctrl+A**
Insert | Table	**Cmd+T**
Insert | Hyperlink	**Cmd+K**
Insert | Comment	**Shift+F2**
Insert | Character | Em dash (—)	**Cmd+Alt+Minus key** (numeric keypad)
Insert | Character | Non-breaking space	**Cmd+Shift+Space**
Formula | Insert function	**F7**

Command	Shortcut key
Formula | Update data:	
Updating charts	**F8**
Update calculations	**F9**
Recalculate the current worksheet only	**Shift+F9**
View | Full screen	**F6**

Other useful shortcut keys

Command	Shortcut key
Display help content	**F1**
Edit the content of the current cell	**F2**
To the next open document	**Cmd+F6**
To the previous open document	**Cmd+Shift+F6**
Expand or collapse the ribbon	**Cmd+F1**
Move to the next ribbon tab	**Ctrl+F12**
Move to the previous ribbon tab	**Ctrl+Shift+F12**
Move to the next worksheet (for open dialog boxes: move to the next tab)	**Ctrl+Tab**
Move to the previous worksheet (for open dialog boxes: move to the previous tab)	**Ctrl+Shift+Tab**
Select the next object (in object mode)	**Tab**
Select the previous object (in object mode)	**Shift+Tab**
Move to the next cell	**←/→/↑/↓**
Move to the next / previous cell horizontally	**Tab / Shift+Tab**
Move to the next / previous cell vertically	**Enter / Shift+Enter**
Move to the next *filled* cell	**Cmd+←/→/↑/↓**
Move to the first cell in the current column	**Cmd+PageUp↑**
Move to the last cell in the current column	**Cmd+PageDown↓**

Command	Shortcut key
Move to the first cell in the current row	**Home**
Move to the last *filled* cell in the current row	**End**
Move to the first cell in the worksheet (cell A1)	**Cmd+Home**
Move to the last *filled* cell in the worksheet	**Cmd+End**
Move to the next circular reference	**Shift+F5**
Go to the next error	**Cmd+F3**
Copy cell above	**Ctrl+,** (comma)
Copy value from cell above	**Ctrl+Shift+,** (comma)
Enter an array formula	**Cmd+Shift+**↵
Select current array formula	**Cmd+7**
Select current conditional formatting	**Cmd+6**
Open a dropdown list with the contents of the cells above or below a cell	**Alt+Shift+**↓
Switch between AUTO and TEXT input mode (see Status bar)	**Cmd+Shift+F4**
Insert current date	**Ctrl+.** (period)
Insert current time	**Ctrl+Shift+.** (period)
Convert a hexadecimal character code into Unicode characters*	**Cmd+Alt+Shift+X**

* If, for example, you type 20AC and then press this shortcut key, a euro sign € will appear (since this character has the code 20AC in the Unicode character set).

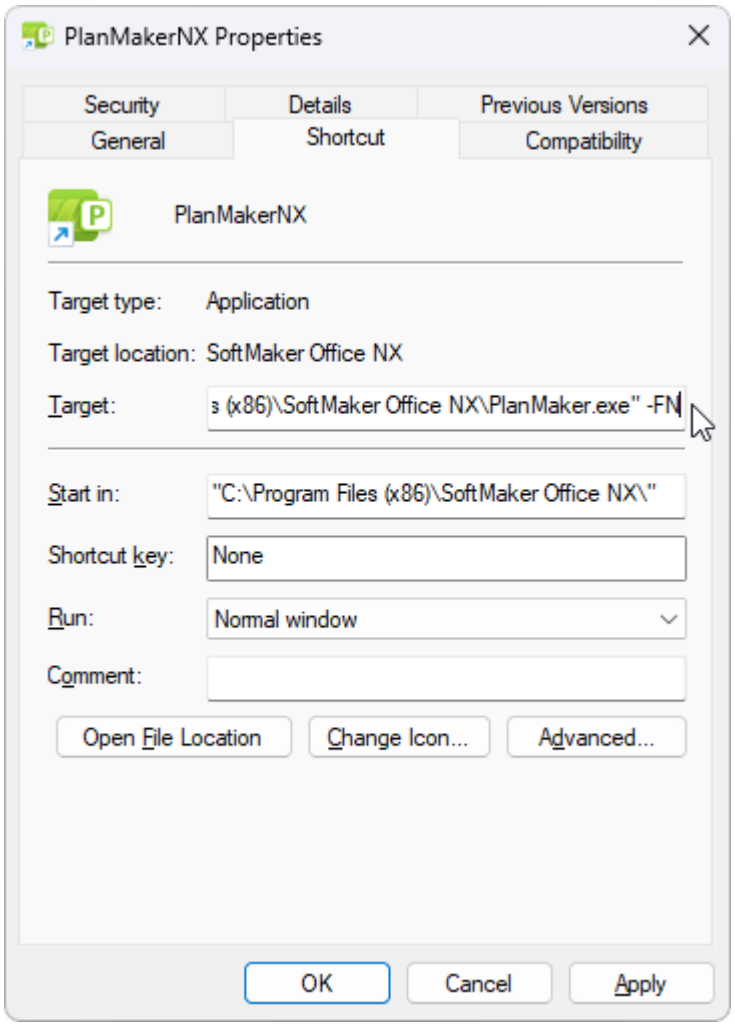
Command-line parameters

Note: The *command-line parameters* described here only work on Windows.

Command-line parameters allow you to specify that PlanMaker is started immediately with a specific program option rather than with the usual default behavior. Example: You would like PlanMaker to switch to the dialog box for selecting a document template directly upon startup.

To insert a command-line parameter, proceed as follows:

1. Create a new shortcut to PlanMaker on the desktop. Ideally, give the link a distinctive name in order to distinguish it more clearly.
2. Choose the properties of the shortcut: Right-click on the link for the context menu and select the entry **Properties**.
3. The following dialog box with the **Shortcut** tab will then appear:



4. In the input field **Target** after the file path `"...\PlanMaker.exe"`, enter the desired parameter from the table below. In the figure, for example, this is the parameter **-FN**.

   Important: There must be a **space** in front of the parameter and the parameter starts with a hyphen.

5. Confirm with **OK**.

When you open PlanMaker via this newly created link (double-click the link), the program will start directly with the behavior of the parameter that was used.

Command-line parameters

Parameter	Description
-N	PlanMaker starts without opening a new default document.
-FO	PlanMaker starts with an open dialog box for selecting a file.
-FN	PlanMaker starts with an open dialog box for selecting a document template.
-P"Path\FileName"	PlanMaker starts and prints the specified file directly on the *default printer*.
-Q"PrinterName","Path\FileName" Note: *No* space is inserted within the parameter.	PlanMaker starts and prints the specified file directly on the *specified printer*.

#

#DIV/0! 520
#N/A 520, 613, 672, 675, 718
#NAME? 520
#NULL! 520
#NUM! 520
#REF! 520
#VALUE! 520

$

$ sign 515

%

% sign (operator) 512

3

3D effect (for objects) 288
3D view (for charts) 362

A

ABS (absolute value) 526
Absolute cell references 515
Absolute value (ABS) 526
Accounting (number format) 192
ACOS (arccosine) 526
ACOSH (inverse hyperbolic cosine) 527
ACOT (arccotangent) 528
ACOTH (inverse hyperbolic cotangent) 528
Actual size (view) 483
Add or remove spaces when pasting 453
Add spacing around punctuation marks (French) 386
Add/edit text
  for AutoShapes 315
  for text frames 315
Addition (operator) 512
ADDRESS (cell address as text) 529
After input, move selection... 456
Align (object)
  align and distribute objects 274
  snap to cells 451
  to other objects with guides 451
Alignment
  of cell contents 205
  of text in text frames 292, 294
All caps 212
Analyze sheets 107
AND (logical AND function) 530
Android/iOS
  context menu 36
  input mode 34
  select 69
  share a document 424
  unlock commands 34
  unlock locked commands 22
  user interface 34
Annuity
  see PMT 747
ANSI code of a character (CODE) 560
Anti-aliasing 458
Arabic scripts 442, 453
Arccosine (ACOS) 526
Arccotangent (ACOT) 528
Arcsine (ASIN) 531
Arctangent
  ATAN 532
  ATAN2 533
Area chart 326
AREAS (number of areas) 531
Array functions
  MDETERM (determinant) 702
  MINVERSE (inverse) 708
  MMULT (product) 709
  MSOLVE (solution of matrix equation) 715
  TRANSPOSE (transpose matrix) 816
  working with arrays 522
Asian scripts 453
ASIN (arcsine) 531
ASINH (inverse hyperbolic sine) 532
Associate file types 465
ATAN (arctangent) 532
ATAN2 (arctangent) 533
ATANH (inverse hyperbolic tangent) 534
Auto correction 386, 453
Auto input mode 39
Autocomplete cells 63, 456
AutoFill 81
AutoFilter 99
AutoFit cell sizes 188
AutoFormat 234
Automatic percent input 456
Automatically recalc (worksheets) 114, 477
Auto-Recover 463, 464
AutoShape
  add/edit text 315

AutoShape
AutoShape effects 316
AutoShape templates 316
change on ribbon tab Object 316
change on the ribbon tab Object 276
change text area 281
change via the dialog box 289, 318
drawing 313
AutoSum 795
AutoText 392
AVEDEV (average deviation) 534
Average
arithmetic mean (AVERAGE) 535
arithmetic mean (AVERAGEA) 535
if condition is true (AVERAGEIF) 536
if conditions are true (AVERAGEIFS) 537
AVERAGE (arithmetic mean) 535
Average deviation (AVEDEV) 534
Average square deviation (DEVSQ) 603
AVERAGEA (arithmetic mean) 535
AVERAGEIF (average if condition is true) 536
AVERAGEIFS (average if conditions are true) 537
Axes (in charts) 332, 351, 352, 354, 356
Axis title (in charts) 332, 355

B

B (compatibility function) 538
Background color
of cells 203
of text 214
of the program 458
Backup copies 405, 464
Backup folder 405, 464
BAK files 405
Banded rows and columns 137
Bar chart 326
Barcode, generate 321
BASE (decimal number to another base) 539
Base-10 logarithm (LOG10) 690
Based on (for cell styles) 227
Based on (for character styles) 224
BasicMaker 445
Beep on errors (for spell check) 462
Beep on errors (for warning messages) 458
BESSELI (modified Bessel function In(x)) 540
BESSELJ (Bessel function Jn(x)) 541
BESSELK (modified Bessel function Kn(x)) 541
BESSELY (Bessel function Yn(x)) 542
Beta distribution
BETA.DIST 543
BETA.INV 544
BETADIST 542
BETAINV 544
BETA.DIST (beta distribution) 543
BETA.INV (percentiles of the beta distribution) 544
BETADIST (beta distribution) 542
BETAINV (percentiles of the beta distribution) 544
BIN2DEC (binary to decimal) 545
BIN2HEX (binary to hexadecimal) 546
BIN2OCT (binary to octal) 547
Binary to decimal (BIN2DEC) 545
Binary to hexadecimal (BIN2HEX) 546
Binary to octal (BIN2OCT) 547
Binomial distribution
BINOM.DIST 549
BINOM.DIST.RANGE 550
BINOM.INV 551
BINOMDIST 548
CRITBINOM 582
NEGBINOM.DIST 719
NEGBINOMDIST 720
Bold (font) 212
Boolean (number format) 192
Borders
of cells 201
of objects 286
Box and whisker plot 326
Boxplot chart 326
Bring forward one level (object) 273
Bring to front (object) 273
Bubble chart 326

C

Calculation in status bar 456
Calculations
iterative 477
update 114, 477
Capitalize first letter of sentences 386
Category axis (in charts) 332, 351
CEILING (round up to a multiple of base) 552
Cell
alignment 205
borders 201
change size 187
delete 71
do not display (protection) 426
do not print (protection) 426
enter data 63
fill automatically 81, 506
insert 73

Cell
merge 205
number format 190, 192, 195
protect 209, 426
select 67
shading 203
CELL (information about a cell) 553
Cell address as text (ADDRESS) 529
Cell contents
copy 75
cut 75
delete 75
paste 75
Cell formatting, transfer 248
Cell frame 44, 451
Cell marker 451
Cell protection 426
Cell references
absolute/relative 515
paste from clipboard 77
show cell contents in objects 281
to dynamic tables 141
Cell references, external 516
Enter 517
manage 518
update 518
Cell styles 226, 227
apply 226
based on 227
create 227
default style "Normal" 229
delete 229
edit 229
rename 229
shortcut keys 227
Centered 205
Change shape 276
Change text area
of AutoShapes 281
of text frames 281
CHAR (character from ANSI code) 554
Character format 209
all caps 212
background color 214
bold 212
character pitch 216
font 211
font color 214
font size 211
italic 212
kerning 216
letter spacing 216
small caps 212
strikethrough 212
superscripts and subscripts 215
text styles 212
underline 212
Character formatting 209
apply to entire words 453
change 209
reset 217, 223
Character pitch 216
Character styles 221
apply 222
based on 224
create 221
default style "Normal" 224
delete 222
edit 222
rename 222
shortcut keys 221, 222
Chart (dialog box) 359
Chart (ribbon tab) 324
Chart area 342
Chart title 333, 356, 361
Charts 323
add/delete data series 360
change arrangement of data series 331, 360
change chart location 363
change chart type 326
change properties 359
data in columns 331
data in rows 331
edit 324
insert 324
save as picture 364
select chart elements 340
update 363, 477
Check for updates 465
Checkboxes (form object) 368
CHIDIST (chi-square distribution) 555
CHIINV (percentiles of the chi-square distribution) 555
CHISQ.DIST.RT (chi-square distribution) 556
CHISQ.INV.RT (percentiles of the chi-square distribution) 557
CHISQ.TEST (chi-square test for independence) 557
CHITEST (chi-square test for independence) 558
CHOICE (x>0, x=0, x<0?) 558
CHOOSE (choose value from list) 559
Circles, draw 313
Circular reference 39, 477
Classic menus with toolbars 22, 458
list of all ribbon commands and menu commands 841
user interface 33

CLEAN (remove non-printable characters) 560
Close (File) 449
Close document 449
CODE (ANSI code of a character) 560
Color
  of cells 203
  of objects 284
  of text 214
Color gradient (for objects) 284
Column
  change width 187, 188
  delete 71
  insert 73
  optimum width 188
  select 67
  show/hide 189
COLUMN (function: column number of cells) 561
Column chart 326
Column header
  change column width 188
  print 252
  show/hide 481
  use for selection 67
COLUMNS (function: number of columns in a range) 561
COMBIN (combinations) 562
Combine (object shapes) 277
Combine shapes 277
Command-line parameters 866
Commands, repeat 66
Comments 116
  always show 116, 475
  delete 116
  edit 116
  insert 116
  show in PDF export 420
  show in the printout 252
Complex numbers
  absolute value (IMABS) 655
  angle (IMARGUMENT) 656
  conjugate (IMCONJUGATE) 657
  cosine (IMCOS) 657
  create complex number (COMPLEX) 563
  difference (IMSUB) 664
  division (IMDIV) 658
  exponential (IMEXP) 658
  imaginary coefficient (IMAGINARY) 655
  logarithm, base 10 (IMLOG10) 659
  logarithm, base 2 (IMLOG2) 660
  negative value (IMNEG) 660
  power (IMPOWER) 661
  product (IMPRODUCT) 661
  real coefficient (IMREAL) 662
  sine (IMSIN) 663
  square root (IMSQRT) 663
  sum (IMSUM) 664
Complex scripts 453
Compress pictures 299
CONCAT (concatenate strings from list) 563
CONCATENATE (concatenate text strings) 564
Concatenation (operator) 512
Concatenation of text strings (functions)
  CONCAT 563
  CONCATENATE 564
  TEXTJOIN 810
Condition
  CHOICE 558
  IF 651
  IFS 654
Conditional formatting 235
  delete rules in selected cells 242
  manage rules 240
  new rule 235
  types of conditions 237
Cone chart 326
CONFIDENCE (confidence interval) 565
Confidence interval
  CONFIDENCE 565
  CONFIDENCE.NORM 566
CONFIDENCE.NORM (confidence interval) 566
Connectors, draw 313
Consolidate data 124
Context menu 36
CONVERT (unit conversion) 567
Convert EU currencies (EUROCONVERT) 614
Copy
  cell contents 75
  formula 75
  insert copied cells 74
  object 75
  text 75
Corners (in charts) 345
Correct first TWo UPpercase LEtters 386
CORREL (correlation coefficient) 571
Correlation coefficient
  CORREL 571
  PEARSON 738
COS (cosine) 571
COSH (hyperbolic cosine) 572
Cosine (COS) 571
COT (cotangent) 572
Cotangent (COT) 572
COTH (hyperbolic cotangent) 573
Count

Count
COUNT 574
COUNTA 574
COUNTBLANK 575
COUNTIF 575
COUNTIFS 576
COUNT (number of cells filled with numbers) 574
COUNTA (number of cells filled) 574
COUNTBLANK (number of empty cells) 575
COUNTIF (count if condition is true) 575
COUNTIFS (count if conditions are true) 576
COUNTP (PlanMaker 97 compatibility function) 578
COVAR (population covariance) 579
Covariance
COVAR (population) 579
COVARIANCE.P (population) 579
COVARIANCE.S (sample) 580
COVARIANCE.P (population covariance) 579
COVARIANCE.S (sample covariance) 580
Create complex number (COMPLEX) 563
Create new document 447
CREATEDATE (date the document was created) 581
Creation date 256
CRITBINOM (binomial distribution) 582
Cropping (of pictures) 300, 302
CSV file format 433, 435
CUMIPMT (cumulative interest) 583
CUMPRINC (cumulative principal) 584
CURRENCY (format number as currency) 585
Currency (number format) 192
Current date (dynamic)
with time: NOW function 729
without time: TODAY function 816
Current date (static)
insert date via shortcut key 63
insert time via shortcut key 63
Curves, draw 313
Custom number formats 192, 195, 196
Customize
create user-defined icons 503
customize shortcut keys 503
customize toolbars 498
ribbon 488
Cut
cell contents 75
formula 75
object 75
text 75
Cylinder chart 326

D

Dark mode
for the user interface 458
for worksheets 485
Data consolidation 124, 125, 128, 130
Data labels (in charts) 334, 345
Data points (in charts) 345
Data series (in charts) 360
add 360
delete 360
edit 345
Data series, fill automatically 81, 506
Data source area, change
for charts 360
for pivot tables 178
Data validation 113, 243
Database functions 524
DAVERAGE 588
DCOUNT 594
DCOUNTA 595
DGET 603
DMAX 604
DMIN 604
DPRODUCT 606
DSTDEV 607
DSTDEVP 608
DSUM 608
DVAR 609
DVARP 610
Date
create a date value (DATE function) 586
current (static), insert via shortcut key 63
enter in cell 63
updated, with time: NOW function 729
updated, without time: TODAY function 816
DATE (create a date value) 586
Date before/after n months (EDATE) 610
Date difference (DATEDIF) 586
Date filter (AutoFilter) 99
Date last changed 256
Date the document was created (CREATEDATE) 581
Date/time (number format) 192
DATEDIF (date difference) 586
DATEVALUE (convert text into date value) 588
DAVERAGE (database function) 588
DAY (day of a date) 589
DAYS (days between two dates) 589
DAYS360 (days between two dates) 590
DAYSP (compatibility function) 591
DAYSPERMONTH (days per month) 592

DAYSPERYEAR (days per year) 593
DB (fixed-declining balance depreciation) 594
dBASE file format 433
DCOUNT (database function) 594
DCOUNTA (database function) 595
DDB (double-declining balance depreciation) 596
DEC2BIN (decimal to binary) 596
DEC2HEX (decimal to hexadecimal) 597
DEC2OCT (decimal to octal) 598
DECIBEL (decibel value of two quantities) 599
DECIMAL (convert number from any base into decimal) 600
Decimal number to another base (BASE) 539
Decimal places, increase/decrease 190
Decimal places, increase/decrease 190
Decimal point after input 475
Decimal separator, change 456
Decimal to binary (DEC2BIN) 596
Decimal to hexadecimal (DEZ2HEX) 597
Decimal to octal (DEC2OCT) 598
DECIMALS (fractional part of a number) 601
Default cell style "Normal" 229
Default character style "Normal" 224
Default currency 475
Default file format when saving 463
Default language for spell check 379, 462
Default number format 192
Default program to open 465
Default template (document template)
- "Normal" 46, 233
- change 46
- switch 233

Default unit of measure 458
Defined names 89, 92
- apply names to existing formulas 93
- assign names 90
- create a list of all named ranges 92

DEGREES (convert radians to degrees) 601
Degrees to radians (RADIANS) 759
Delete 44
- cell content 70
- cells 70, 71
- columns/rows 71
- duplicate rows 72
- empty rows 72
- object 75
- text 75

Delete special (cell content) 70
DELTA (test for equality) 602
Depreciation
- double-declining balance (DDB) 596
- fixed-declining balance (DB) 594
- straight-line (SLN) 781
- sum-of-years' digits (SYD) 804

Design Science 311
Desktop mode (Android/iOS) 458
Determinant of a matrix (MDETERM) 702
DEVSQ (average square deviation) 603
DGET (database function) 603
Dialog language 458
Dictionaries for spell check
- edit user dictionaries 382
- export/import user dictionaries 470
- install 508

Display formula contained in cell (FORMULATEXT) 629
Distance to edge 250
Distribute (object) 274
Division (operator) 512
DMAX (database function) 604
DMIN (database function) 604
Document management 398
Document properties 473
- Calculate tab 477
- Colors tab 473
- Fonts tab 480
- Options tab 475
- Protection tab 480
- Statistics tab 475
- Summary tab 400, 473

Document protection 430
Document statistics 475
Document summary
- enter 400
- prompt for summary information when saving 463

Document tabs 38, 446
Document templates
- apply 232
- change template folder 231, 463
- create 231
- default template "Normal" 233
- edit 232

Document, print 50, 416
Document, save 50
Documents, view side by side 448
Dollar (number format) 190
DOLLARDE (convert to dollar price decimal) 605
DOLLARFR (convert to dollar price fraction) 606
Doughnut chart 326
DPRODUCT (database function) 606
Drag and drop 75

Drawing object (dialog box) 318
Drawing object (ribbon tab) 316
Drawings 313
  change properties 316, 318
  combine 277
  edit retroactively 279
  insert 313
  rotate 272
Dropdown lists (form object) 371
DSTDEV (database function) 607
DSTDEVP (database function) 608
DSUM (database function) 608
Duplicate (objects) 274
Duplicates
  remove duplicate rows 72
  show duplicate values 237
DVAR (database function) 609
DVARP (database function) 610
Dynamic tables 132

E

EDATE (date before/after n months) 610
Edit chart elements 340
  axis title 355
  chart area 342
  chart title 356, 361
  corners 345
  data labels 345
  data series 345
  error bars 345
  floor 344
  gridlines 357
  legend 357
  lines 345
  plot area 343
  trendline 349
  up/down bars 345
  walls 344
  X axis 351
  Y axis 352
  Z axis 354
Edit color palette 473
Edit lists (for filling) 81, 506
Edit mode and object mode 270
Edit points (of objects) 279
Edit script 445
Edit shapes retroactively 279
Edit toolbar 37
EFFECT (effective interest rate) 611
Effective interest rate (EFFECT) 611
Effects (for objects) 288
Ellipses, draw 313
Ellipsis ("...") 184
E-mail a document 424
Empty rows, delete 72
En dashes, different widths 184
Enable OpenGL engine (Linux only) 465
Enable OpenGL for AutoShapes 465
Encryption 430
Enter data in cells 63
EOMONTH (end of month in n months) 612
Equality (DELTA) 602
Equation Editor 311
ERF (Gaussian error function) 612
ERFC (complement to Gaussian error function) 613
Error bar (in charts) 335, 345
Error function
  ERF 612
  ERFC (complement) 613
Error type (ERROR.TYPE) 613
Error values in calculations 112, 520, 672
ERROR.TYPE (error type) 613
Euro (number format) 190
EUROCONVERT (convert EU currencies) 614
EVEN (round up to next even number) 616
EXACT (compare texts) 616
Excel file format 368, 433, 438, 440
Exit (file) 51
EXP (power of e) 617
Expand the input field 37
EXPON.DIST (exponential distribution) 618
EXPONDIST (exponential distribution) 618
Exponential distribution
  EXPON.DIST 618
  EXPONDIST 618
Exponential regression
  statistics (LOGEST) 691
  values (GROWTH) 640
Exponentiation (operator) 512
Export (other file format) 433
Export/Import of the settings 470
Extended support
  for Arabic text 442, 453
  for Asian fonts 453
External cell references 516
  enter 517
  manage 518
  update 518

F

F distribution
F.DIST.RT 621
F.INV.RT 624
FDIST 621
FINV 623
F.DIST.RT (F distribution) 621
F.INV.RT (percentiles of the F distribution) 624
F.TEST (F-test) 631
FACT (factorial) 619
FACTDOUBLE (double factorial) 620
FALSE (logical value FALSE) 620
FDIST (F distribution) 621
Fields (in headers/footers) 256
File
export as PDF 420
send by e-mail 424
File formats 433
File manager 401
File path, show in title bar 458
File properties 473
Calculate tab 477
Colors tab 473
Fonts tab 480
Options tab 475
Protection tab 430, 480
Statistics tab 475
Summary tab 400, 473
File type 433
File versions 405, 464
File: Close 449
File: Exit 51
File: New 46, 232
File: Open 48, 433
File: Page setup 250, 252, 256
File: Print 50, 416
File: Print preview 415
File: Properties 473
File: Save 50
File: Save all 50
File: Save as 50, 433
FILENAME (file name of the document) 622
Fill
of objects 284
Fill cells (automatically) 81, 506
Filling
of cells 203
Filter 99
AutoFilter 99
special filter 104
SUBTOTAL (function) 794
FIND (search for text in a text string) 623
FINV (percentiles of the F distribution) 623
First column (highlight) 137
First column, freeze 182
FISHER (Fisher transformation) 625
FISHERINV (inverse of the Fisher transformation) 625
FIXED (format number as text with fixed decimals) 626
Flip (objects) 272
Floor (in charts) 344
FLOOR (round down to a multiple of base) 627
Folder for file versions 405, 464
Font 211
bold 212
italic 212
underline 212
Font color 214
Font lists, hide fonts 468
Font size 211
Footers 256
FORECAST (forecast using linear regression) 628
Form object (ribbon tab) 378
Form objects 366
edit 367
evaluate 367
Form objects, insert 366
checkboxes 368
dropdown lists 371
groupboxes 377
labels 377
listboxes 372
pushbuttons 374
radio buttons 370
scrollbars 375
spinners 374
Format painter 248
Format URLs as hyperlinks 386
Formatting toolbar 33
Forms 365
Formula auditing 110, 111, 112, 113
Formulas and functions from A to Z 511
FORMULATEXT (display formula contained in cell) 629
Fractile (see PERCENTILE) 739, 740, 741
Fraction (number format) 192
Freehand, draw 313
Freeze cells 182
Freeze rows/columns 182

FREQUENCY (frequency) 629
F-test
F.TEST 631
FTEST 630
Full screen mode (view) 484
Functions 511
functions from A to Z 526
insert 513
Future value
FV 631
FVSCHEDULE 632
FV (future value) 631
FVSCHEDULE (future value) 632

G

GAMMA (gamma function) 633
Gamma distribution
GAMMA.DIST 634
GAMMA.INV 636
GAMMADIST 634
GAMMAINV 635
Gamma function
GAMMA 633
GAMMALN 636
GAMMA.DIST (gamma distribution) 634
GAMMA.INV (percentiles of the gamma distribution) 636
GAMMADIST (gamma distribution) 634
GAMMAINV (percentiles of the gamma distribution) 635
GAMMALN (logarithm of the gamma function) 636
GAUSS (standard normal distribution) 637
Gaussian distribution (NORM.DIST) 723
Gaussian distribution (NORMDIST) 722
Gaussian error function
ERF 612
ERFC complement 613
GCD (greatest common divisor) 637
GEOMEAN (geometric mean) 638
Geometric mean (GEOMEAN) 638
GESTEP (number greater or equal to threshold value?) 639
GETPIVOTDATA 639
Glow effect (for objects) 288
Go to 266
Goal seek 118
Greatest common divisor (GCD) 637
Gridlines between worksheet cells
paint behind cells 451
print 252
show 481
Gridlines in charts 335, 357
Group
cells (outliner) 409
in pivot tables 174
objects 275
show/hide cells 410
Groupboxes (form object) 377
GROWTH (values of an exponential regression) 640
Guidelines for text frames, show 475
Guides for objects 451

H

Hamburger menu 29, 31
HARMEAN (harmonic mean) 641
Harmonic mean (HARMEAN) 641
Header row in tables 137
Headers and footers 256
Height of a row, change 187, 188
Help content, display (F1 key) 857, 861
HEX2BIN (hexadecimal to binary) 642
HEX2DEZ (hexadecimal to decimal) 643
HEX2OCT (hexadecimal to octal) 644
Hexadecimal to binary (HEX2BIN) 642
Hexadecimal to decimal (HEX2DEZ) 643
Hexadecimal to octal (HEX2OCT) 644
Hide
cell contents (protection) 426
formulas (protection) 426
objects 276, 289
rows/columns 189
Hide cells
manually 189
using an outliner 408
Hide formula (protection) 426
High-low chart 326
HLOOKUP (look up row-wise) 645
HOLIDAY (dates of movable Christian holidays) 646
Horizontal alignment
of cell contents 205
of text in AutoShapes 219
of text in text frames 219
HOUR (hour) 647
HTML format 440
Hunspell dictionaries 462, 508
Hyperbolic cosine (COSH) 572
Hyperbolic cotangent (COTH) 573
Hyperbolic sine (SINH) 779
Hyperbolic tangent (TANH) 807

Hypergeometric distribution
HYPGEOM.DIST 650
HYPGEOMDIST 649
HYPERLINK (function: hyperlink) 648
Hyperlinks (links) 386, 412
applied to cells 412
for objects 289
for OLE objects 307
format URLs as hyperlinks 386
HYPERLINK function 648
HYPGEOM.DIST (hypergeometric distribution) 650
HYPGEOMDIST (hypergeometric distribution) 649
Hyphen
non-breaking hyphen 184
soft hyphen 184
Hyphenation 384
in cells 385
in text frames 384
set language 379

I

IF (if-then-else condition) 651
IFEMPTY (return a defined value if cell is empty) 652
IFERROR (return a value if error) 652
IFNA (return defined value for #N/A) 653
IFS (first condition met) 654
Ignore words that start with a number 462
IMABS (absolute value of a complex number) 655
Images
copy to document's folder 297
insert 297
save within document 297
IMAGINARY (imaginary coefficient of a complex number) 655
IMARGUMENT (angle of a complex number) 656
IMCONJUGATE (conjugate a complex number) 657
IMCOS (cosine of a complex number) 657
IMDIV (division of complex numbers) 658
IMEXP (exponential of a complex number) 658
IMLN (natural logarithm of a complex number) 659
IMLOG10 (base-10 logarithm of a complex number) 659
IMLOG2 (base-2 logarithm of a complex number) 660
IMNEG (negative value of a complex number) 660
Import (other file format) 433
Import/Export of the settings 470
IMPOWER (power of a complex number) 661
IMPRODUCT (product of complex numbers) 661
IMREAL (real coefficient of a complex number) 662
IMSIN (sine of a complex number) 663
IMSQRT (square root of a complex number) 663
IMSUB (difference of complex numbers) 664
IMSUM (sum of complex numbers) 664
In-cell editing 456
Indents (for text frames) 218
INDEX (cell in a particular row/column) 665
INDIRECT (create reference from text) 666
Information about a cell (CELL) 553
Inner margins
of cells 205
of text frames 292, 294
of text in AutoShapes 319
Input field, expand 37
Input method (Android) 34
Input validation 113, 243, 244, 247
Insert
cells 73
characters 184
charts 324
columns 73
comments 116
copied cells 74
drawings 313
Equation Editor objects 311
form objects 366
functions 513
headers and footers 256
objects 269
OLE objects 305
page break 260
pictures 297
rows 73
SmartText entries 392
special characters 184
text frames 292
Insert characters 184
Insert frames and drawings immediately 453
Insert mode / overwrite mode 39
disable 453
Insert special characters 184
Installation 26
Android 28
iOS 28
Linux 28
macOS 27
Windows 26
INT (round down to nearest integer) 667
INTERCEPT (intercept point of a regression line) 667
Interest payment
CUMIPMT (cumulative) 583
IPMT 669
ISPMT 680

Interest rate
PCF 772
per period (RATE) 764
RRI 772
Internal rate of return
IRR 670
MIRR 709
XIRR 837
Intersection
INTERSECTION (function: intersection of two ranges) 668
operator 512
Inverse hyperbolic cosine (ACOSH) 527
Inverse hyperbolic cotangent (ACOTH) 528
Inverse hyperbolic sine (ASINH) 532
Inverse hyperbolic tangent (ATANH) 534
Inverse matrix (MINVERSE) 708
iOS/Android
context menu 36
input mode 34
select 69
share a document 424
unlock commands 34
unlock locked commands 22
user interface 34
IPMT (interest payment) 669
IRR (internal rate of return) 670
ISBLANK (is empty?) 671
ISERR (is an error value other than #N/A?) 520, 672
ISERROR (is an error value?) 520, 672
ISEVEN (is an even number?) 673
ISFORMULA (is a formula?) 673
ISLOGICAL (is a logical value?) 674
ISNA (is not available?) 675
ISNONTEXT (is no text?) 675
ISNUMBER (is numeric?) 676
ISNUMBERP (PlanMaker 97 compatibility function) 677
ISODD (is an odd number?) 678
ISOWEEK (ISO week number) 678
ISOWEEKNUM (ISO week number) 679
ISPMT (interest payment) 680
ISREF (is a valid reference?) 681
ISTEXT (is text?) 681
Italic (font) 212
Iterations for calculations 477

J

Jump target (hyperlink) 412
Justification (in cells) 205
Justification (in text frames) 292, 294

K

Keep aspect ratio 283
Keep scaling 283
Kerning 216
KURT (kurtosis) 682
Kurtosis (KURT) 682

L

Labels (form object) 377
Landscape format 250
Language
for spell check and hyphenation 379
for user interface 458
LARGE (k-th largest number) 683
Last column (highlight) 137
LASTPRINTED (date last printed) 684
LASTSAVED (date last saved) 684
LCM (least common multiple) 685
Least common multiple (LCM) 685
LEFT (function: left part of a text string) 685
Left align 205
Left arrow key never exits cell editing 456
Legend (in charts) 336, 357
LEN (length) 686
Letter spacing 216
Line break 205
Line chart 326
Line spacing (for text frames) 218
Linear regression
FORECAST 628
INTERCEPT 667
show trend in charts 349
statistics (LINEST) 687
values (TREND) 817
Lines
as borders of objects 286
drawing 313
Lines (in charts) 337, 345
LINEST (statistics of a linear regression) 687
Link to cell content, show in object 281
Links (hyperlinks)
applied to cells 412
for objects 289
format URLs as hyperlinks 386
HYPERLINK function 648
Links (OLE objects)

Links (OLE objects)
edit 307
update 307
Listboxes (form object) 372
Live preview 458
LN (natural logarithm) 689
Locking objects 289
LOG (logarithm) 689
LOG10 (base-10 logarithm) 690
Logarithm
any base (LOG) 689
base 10 (LOG10) 690
natural (LN) 689
LOGEST (statistics of an exponential regression) 691
LOGINV (percentiles of the log-normal distribution) 692
LOGNORM.DIST (log-normal distribution) 694
LOGNORM.INV (percentiles of the log-normal distribution) 695
Log-normal distribution
LOGINV 692
LOGNORM.DIST 694
LOGNORM.INV 695
LOGNORMDIST 693
LOGNORMDIST (log-normal distribution) 693
LOOKUP (search cell range) 695
LOWER (convert text to lowercase) 698
Lowercase
LOWER 698
PROPER 755

M

Macros 368, 440
Mail the document 424
Manage (button in the settings) 469
Manual (PlanMaker) 24
Margins
inner margins of AutoShapes 319
inner margins of cells 205
inner margins of text frames 292, 294
page margins 250
Mark invalid data 247
MATCH (relative position in a range) 699
MathType 311
Matrix equation (MSOLVE) 715
MAX (maximum) 699
MAXA (maximum) 700
MAXIFS (conditional maximum) 701
Maximum number of undo steps 453
MDETERM (matrix determinant) 702
Mean
geometric (GEOMEAN) 638
harmonic (HARMEAN) 641
without marginal values (TRIMMEAN) 819
MEDIAN 703
Menu bar 33
Menus with toolbars or ribbon? 29, 458
Merge cells 205
MID (part of a text string) 703
Middle mouse button 453
MILLISECONDS (milliseconds) 704
MIN (minimum) 705
MINA (minimum) 705
MINIFS (conditional minimum) 706
MINUTE (minute) 707
MINVERSE (inverse matrix) 708
MIRR (modified internal rate of return) 709
Mirror (objects) 272
MMULT (product of matrices) 709
MOD (remainder of a division, Excel method) 710
MODE (most frequently occurring value) 711
MODE.SNGL (most frequently occurring value) 712
Modified internal rate of return (MIRR) 709
MODP (remainder of a division, PlanMaker method) 713
Modulo
MOD (Excel method) 710
MODP (PlanMaker method) 713
MONTH (month of a date) 714
Move by ... decimals 475
MROUND (round to a multiple of base) 714
MSOLVE (solution of matrix equation Ax=B) 715
Multinomial coefficient
MULTINOMIAL 716
POLYNOMIAL 751
Multiplication (operator) 512

N

N (convert value into number) 717
NA (error value #N/A) 718
Name of a worksheet (SHEETNAME) 776
Names (for cell ranges) 89, 92
apply to existing formulas 93
assign 90
create from selection 91
edit 90
insert list 92
Names (of objects) 274
Names (of tables) 137, 141, 274

NEG (negative value) 719
Negative binomial distribution
NEGBINOM.DIST 719
NEGBINOMDIST 720
Negative value (NEG) 719
Net present value
NPV 731
XNPV 837
NETWORKDAYS (number of workdays) 721
Neumann function (BESSELY) 542
New (File) 46, 231, 232
New pivot table, create 145
New program window 447
New table (create tables in worksheets) 132, 133, 134
NOMINAL (nominal interest rate) 722
Non-breaking hyphen 184
Non-breaking space 184
Non-printable characters, remove (CLEAN) 560
NORM.DIST (normal distribution) 723
NORM.INV (percentiles of the normal distribution) 725
NORM.S.DIST (standard normal distribution) 727
NORM.S.INV (percentiles of the standard distribution) 728
Normal
cell style 229
character style 224
Normal distribution
NORM.DIST 723
NORM.INV 725
NORMDIST 722
NORMINV 724
Normal distribution, logarithmic
LOGINV 692
LOGNORM.DIST 694
LOGNORM.INV 695
LOGNORMDIST 693
Normal.pmvx 46, 233
NORMDIST (normal distribution) 722
NORMINV (percentiles of the normal distribution) 724
NORMSDIST (standard normal distribution) 726
NORMSINV (percentiles of the standard distribution) 727
NOT (logical NOT function) 729
NOW (current date and time) 729
NPER (number of periods) 730
NPV (net present value) 731
Number (number format) 192
Number filter (AutoFilter) 99
Number format 190, 192, 195, 196
Number of periods (NPER) 730
Number of worksheets (SHEETS) 777
Number separators 456
Number series, fill automatically 81, 506
Numbers, enter 63
NUMBERVALUE (convert text into a number) 732

O

Object
copy 75
cut 75
delete 75
insert 269
paste 75
select 269
Object (dialog box) 283
Object mode and edit mode 270
Object properties
change default settings 290
for charts 359
for checkboxes 369
for drawings 316, 318
for dropdown lists 371
for groupboxes 377
for labels 377
for listboxes 373
for OLE objects 308, 310
for pictures 302
for pushbuttons 374
for radio buttons 370
for scrollbars 376
for spinners 375
for text frames 292, 294
general properties 283
Objects 268
align and distribute 274
change order 273
change position 271, 283
change properties 283
combine 277
distribute 274
duplicate 274
edit retroactively 279
group 275
guides for alignment 451
hide 276
resize 271, 283
rotate 283
rotate or flip 272
show hidden objects 276, 475
show linked cell contents 281
show text frame guidelines 475

Objects 268
ungroup 275
OCT2BIN (octal to binary) 734
OCT2DEC (octal to decimal) 735
OCT2HEX (octal to hexadecimal) 735
Octal to binary (OCT2BIN) 734
Octal to decimal (OCT2DEC) 735
Octal to hexadecimal (OCT2HEX) 735
ODD (round up to next odd number) 736
OFFSET (shifted reference) 737
OLE object (dialog box) 310
OLE object (ribbon tab) 308
OLE objects 305
change properties 308, 310
edit 307
edit links 307
Equation Editor 311
insert 305
update links 307
warning when loading 453
Open (File) 48, 232, 433
Operators in formulas 512
Optimum height 188
Optimum width 188
OR (logical OR function) 738
Order (of objects) 273
Ordinal numbers, add superscript (auto correction) 386
Orientation
of printed pages 250
Other file formats 433
Outline pane 408, 410
Outlines 408
automatically show outline pane 410
change settings 410
group 409
options 410
protect 410
remove all groups 409
show/hide grouped cells 410
ungroup 409
Overlapping objects 294, 319
Overwrite mode / insert mode 39
disable 453

P

Page break preview 260
Page breaks
adjust manually 260
show/hide 481
Page setup 250
copy to other worksheets 87
headers and footers 256
number of pages 256
page format 250
page margins 250
page number 252, 256
page orientation 250
page size 250
print options 252
Paper size 250
Paper trays 250
Paragraph format (for text frames) 217
indents 218
line spacing 218
spacing above/below 220
text alignment 219
Pascal distribution
NEGBINOM.DIST 719
NEGBINOMDIST 720
Paste copied content 75
Paste reference 77, 517
Paste special 77
Payment (PMT) 747
PCF (interest rate) 772
PDF export 420
PDF/A, create 420
Peakedness of a distribution (KURT) 682
PEARSON (Pearson correlation coefficient) 738
Percent sign (operator) 512
Percentage (number format) 190, 192
PERCENTILE (percentiles of a data set) 739
PERCENTILE.EXC (percentiles of a data set) 740
PERCENTILE.INC (percentiles of a data set) 741
PERCENTRANK (percent rank in a data set) 742
PERCENTRANK.EXC (percent rank in a data set) 742
PERCENTRANK.INC (percent rank in a data set) 743
PERIOD (duration of fixed-interest investments) 744
PERMUT 745
PHI (standard normal distribution) 746
Photo frames 276, 300
PI (constant value of pi) 747
Picture (dialog box) 302
Picture (ribbon tab) 300
Pictures 297
change properties 302
change source 300
compress 299
copy to document's folder 297
export 302
from scanner 298
insert 297

Pictures 297
insert from the gallery/camera (Android/iOS) 298
recolor 300, 302
rotate or flip 272
save within document 297
select source 298
Pie chart 326
Pinned files 48
Pinned templates 46
Pivot tables 143
change the source data area 178
column labels 149
copy 179
create from existing data 146
create from external data 148
create from tables in worksheets 145
custom name for fields 155, 156, 160, 165
custom name for pivot table 168
defer layout update 171
delete 179
field list: areas section 156
field list: configure the pivot table 159
field list: exercises 150
field list: fields section 155
field list: further options 158
field list: initial screen 149
field settings 160
fields: move and remove 156
filter 172
filter labels/filter values 172
form (field settings) 160
GETPIVOTDATA 639
group 174
layout (pivot table settings) 168
modify pivot table areas 178
move 179
number format 165
outline form 160
pivot table settings 168
pivot table sidebar 149
refresh when opening the file 168, 178
report filter 149, 172
row labels 149
save source data with file 168
sort A-Z (fields in the field list) 155
sort A-Z or Z-A (fields in the pivot report) 172
sort by fill lists (fields in the pivot report) 168, 172
styles for pivot tables 168
subtotals 160
tabular form 160
update 178
value settings 165
values area 149
Plain text file format 433, 435
PlanMaker
basics 42
customize 450
set as default program 465
settings 451
user interface 29
PlanMaker basics 42
PlanMaker file format 433
PlanMaker manual 24
PlanMaker Tour 52
Plot area (in charts) 343
PMBAK files 405
PMD file format 433
PMDX file format 433
PMT (payment) 747
Poisson distribution
POISSON 748
POISSON.DIST 749
Polynomial coefficient
MULTINOMIAL 716
POLYNOMIAL 751
Portrait format 250
Position (of objects) 271, 283
Position of a value in a range (MATCH) 699
Power (POWER function) 751
PPMT (payment on the principal) 752
Precision as displayed 477
Present value (PV) 755
Preview (of a document) 48
Primary axes (in charts) 361
Principal
CUMPRINC (cumulative) 584
PPMT 752
Print a document 50, 416
Print date 256
Print order 252
Print preview 415
Print range
define print range 252
remove print range 252
Print time 256
Printable (object property) 289
PROB (probability) 753
PRODUCT (product) 754
Product of matrices (MMULT) 709
Prompt for summary information when saving 463
PROPER (convert text to upper/lower case) 755
Properties (document) 473
Calculate tab 477
Colors tab 473
Fonts tab 480

Properties (document) 473
Options tab 475
Protection tab 430, 480
Statistics tab 475
Summary tab 400, 473
Properties (object)
for charts 359
for checkboxes 369
for drawings 316, 318
for dropdown lists 371
for groupboxes 377
for labels 377
for listboxes 373
for OLE objects 308, 310
for pictures 302
for pushbuttons 374
for radio buttons 370
for scrollbars 376
for spinners 375
for text frames 292, 294
Properties (objects in general)
change default settings 290
general properties 283
Properties (worksheet) 481
Protect 426
cell protection 426
document protection 430
outline 410
sheet protection 426
workbook protection 429
Protection indicator 475
Pushbuttons (form object) 374
PV (present value) 755
Pyramid chart 326

Q

QR code, generate 321
QUARTILE (quartiles of a data set) 756
QUARTILE.EXC (quartiles of a data set) 757
QUARTILE.INC (quartiles of a data set) 758
Quick access toolbar 31
change position 458, 492
customize icons 492
Quick paths 398
Quotes, smart 386
QUOTIENT (quotient of a division) 759

R

Radar chart 326, 362
RADIANS (convert degrees to radians) 759
Radians to degrees (DEGREES) 601
Radio buttons (form object) 370
RAND (random value) 760
RANDBETWEEN (random value) 761
Rank
PERCENTRANK 742
PERCENTRANK.EXC 742
PERCENTRANK.INC 743
RANK 761
RANK.AVG 762
RANK.EQ 763
RANK (rank in a data set) 761
RANK.AVG (rank in a data set) 762
RANK.EQ (rank in a data set) 763
RATE (interest rate per period) 764
Rate of return
IRR 670
MIRR 709
XIRR 837
Read aloud 396
Recalc only before ... 114, 477
Recalculate
all worksheets 114, 477
current worksheet 114
Rectangles, draw 313
Redo 66
Redo (changes) 66
References, external 516
Reflection effect (for objects) 288
Refresh data for pivot table 178
Regression, exponential
statistics (LOGEST) 691
values (GROWTH) 640
Regression, linear
FORECAST 628
INTERCEPT 667
show trend in charts 349
statistics (LINEST) 687
values (TREND) 817
Relative cell references 515
Remove duplicate rows 72
Remove empty rows 72
Remove spaces (TRIM) 818
Repeat commands 66
Repeat search 264
Repeated columns 252
Repeated rows 252
REPLACE (replace text in a text string) 765
Replace (ribbon command Home | Search) 263, 264
Replace again (ribbon command Home | Search) 264

Replace text in a text string
REPLACE 765
SUBSTITUTE 793
REPT (repeat text string n times) 766
Research via the Internet 395
Reset
character formatting 217, 223
PDF export options 420
picture settings 300
ribbon 488
SoftMaker Office settings 470
toolbars 497
Revert to previous version (file) 405
Ribbon 22, 31
create user-defined icons 503
customize 486
customize groups 488
customize icons 488
list of all ribbon commands and menu commands 841
manage 488
reset 488
show/hide 487
user interface 31, 458, 486
Ribbon or menus with toolbars? 29, 458
RIGHT (right part of a text string) 766
Right align 205
ROMAN (Roman numeral) 767
Root
n-th root (ROOTN) 768
square root (SQRT) 786
square root of complex number (IMSQRT) 663
square root of x*Pi (SQRTPI) 786
ROOTN (n-th root) 768
Rotate 205
axis labels 351, 352, 354, 356
cell contents 205
drawings 283
objects 272, 283
pictures 283
text in AutoShapes 319
text in text frames 292, 294
Rotate or flip (objects) 272
Round
CEILING 552
EVEN 616
FIXED 626
FLOOR 627
INT 667
MROUND 714
ODD 736
ROUND 768
ROUNDDOWN 769
ROUNDUP 770
TRUNC 820
ROUND (round to n digits) 768
ROUNDDOWN (round down to n digits) 769
Rounding
display numbers rounded (per number format) 190
final result 477
intermediate results 477
precision as displayed 477
ROUNDUP (round up to n digits) 770
Row
change height 187, 188
delete 71, 72
insert 73
optimum height 188
select 67
show/hide 189
Row & column headers 481
ROW (function: row number of cells) 771
Row header
change row height 188
print 252
show/hide 481
use for selection 67
ROWS (function: number of rows) 771
RRI (interest rate) 772
RSQ (square of Pearson) 773

S

Save (File) 50
Save all (File) 50
Save as (File) 50, 433
Scale document 458
Scaling (of printed pages) 252
Scan pictures 298
Scenarios 120
Scientific (number format) 192
Scribble mode on iOS 34
Scripts 368, 440, 445
Scrollbars (form object) 375
Scrollbars, show/hide 475
Search (file manager) 402
Search (function)
FIND (search for text in a text string) 623, 773
HLOOKUP 645
LOOKUP 695
SEARCH (search for text in a text string) 773
VLOOKUP 830
Search (ribbon command Home | Search) 262, 264
cell contents 262
search again 264

Search files (file manager) 402
SECOND (second) 774
Secondary axes (in charts) 360, 361
Select
  always select whole word 453
  cells 67
  in the Android/iOS version 69
  objects 269
Select all 67, 270
Select chart elements 340
Select whole words when selecting 453
Send a document by e-mail 424
Send backward one level (object) 273
Send to back (object) 273
Series axis (in charts) 332, 354, 356
SERIESSUM (sum of a power series) 775
Set as default
  document template 46
  object properties 290
Settings (PlanMaker) 450, 451
  Appearance tab 458
  Backup tab 464
  Edit tab 456
  Files tab 463
  Fonts tab 468
  General tab 453
  Language tab 462
  System tab 465
  View tab 451
Settings (SoftMaker Office)
  export/import 470
  reset 470
Shading (of cells) 203
Shadow (for objects) 287
Share a document (via Android/iOS and via Windows 10 and higher) 424
Sheet
  change properties 87
  copy 87
  delete 87
  hide sheet 87
  insert 86
  move 87
  protect 426
  rename 87
  show sheet 87
  visibility 87
SHEET (index of a worksheet) 775
Sheet direction 443, 481
Sheet grid (gridlines), show 481
Sheet protection 426
  disable 428
  enable 427
Sheet register 39, 86
  change color 481
  display 475
  font size 451
SHEETNAME (name of a worksheet) 776
SHEETNUMBER (compatibility function) 777
SHEETS (number of worksheets) 777
Shortcut keys
  [export/import 470
  customize 503
  for cell styles 227
  for character styles 221, 222
  for special characters 184
  predefined shortcuts: Mac 857
  predefined shortcuts: Windows/Linux 857
  view list of all shortcut keys 503
Show cells 189
Show duplicate values 237
Show fonts in font list 458
Show formulas (ribbon tab View) 107
Show full path in title bar 458
Show hidden files and folders 465
Show hidden objects 276, 475
Show indicators (for comments) 475
Show tooltips
  for commands 458
  for formulas 456
Show warning when loading OLE objects 453
Show watch list 109
Show/hide chart elements 331
  axes (X, Y, Z) 332
  axis title 332
  chart title 333, 361
  data labels 334
  error bars 335
  gridlines 335
  legend 336
  lines 337
  trendline 338
  up/down bars 339
Show/hide keyboard automatically 465
Sidebar for the pivot table 149
Sign (operator) 512
SIGN (sign of a number) 778
SIN (sine) 779
Sine (SIN) 779
SINH (hyperbolic sine) 779
Size
  of cells 187
  of objects 271, 283
SKEW (skewness of a distribution) 780

SLN (straight-line depreciation) 781
SLOPE (slope of a linear trend) 781
SMALL (k-th smallest number) 782
Small caps 212
Smart quotes 386
SmartText 392
  export/import 470
SmartText entries
  create 392
  edit 394
  expand 386
  insert 393
  replace automatically 393
Smooth edges of pictures 465
Smooth edges of screen fonts 458
Snapshots (backup copies) 405, 464
Soft edges effect (for objects) 288
Soft hyphen 184
SoftMaker Basic 445
SoftMaker Equation Editor 311
SoftMaker support 23
Sort and filter 95, 99
SORTM (sort function) 783
SORTV (sort function) 784
Source data for charts 360
Source data for pivot tables 178
Source for picture files 298
Spaces, different widths 184
Speak 396
Special filter 104
Spell check 380
  as you type 382, 462
  edit user dictionaries 382
  install additional dictionaries 508
  manual 380
  set language 379
Spinners (form object) 374
SQLite file format 433
SQRT (square root) 786
SQRTPI (square root of x*Pi) 786
Square of Pearson (RSQ) 773
Standard deviation
  entire population (STDEV.P) 790
  entire population (STDEVP) 789
  entire population (STDEVPA) 790
  sample (STDEV) 787
  sample (STDEV.S) 791
  sample (STDEVA) 788
Standard error of a linear regression (STEYX) 792
Standard normal distribution
  GAUSS 637
  NORM.S.DIST 727
  NORM.S.INV 728
  NORMSDIST 726
  NORMSINV 727
  PHI 746
Standard toolbar 33
STANDARDIZE (standardized value) 787
Start script 445
Statistics (of document) 475
Statistics of a regression
  LINEST (linear) 687
  LOGEST (exponential) 691
Status bar 39
  show/hide 458
STDEV (standard deviation of a sample) 787
STDEV.P (standard deviation of entire population) 790
STDEV.S (standard deviation of a sample) 791
STDEVA (standard deviation of a sample) 788
STDEVP (standard deviation of entire population) 789
STDEVPA (standard deviation of entire population) 790
STEYX (standard error of a linear regression) 792
Stock chart 326
Strikethrough 212
Student's t-distribution
  T.DIST 808
  T.DIST.2T (two-tailed) 808
  T.DIST.RT (right-tailed) 809
  T.INV (left-tailed) 814
  T.INV.2T (two-tailed) 815
  TDIST 807
  TINV 813
Student's t-test (T.TEST) 822
Student's t-test (TTEST) 821
Styles
  cell styles 226
  character styles 221
  table styles 137
Subscripts 215
SUBSTITUTE (replace text in a text string) 793
SUBTOTAL (calculations without hidden cells) 794
Subtraction (operator) 512
SUM (Sum) 795
SUMIF (sum if condition is true) 797
SUMIFS (sum if conditions are true) 798
SUMPRODUCT (sum of products) 799
SUMSQ (sum of squares) 800
SUMX2MY2 (sum of x^2 - y^2) 801
SUMX2PY2 (sum of x^2 + y^2) 802

SUMXMY2 (sum of (x - y)^2) 802
Superscript of ordinal numbers (auto correction) 386
Superscripts 215
Support 23
Surface chart 326
SWITCH (categorize initial value) 803
SYD (sum-of-years' digits depreciation) 804
Syntax highlighting 108, 475
  when opening Excel files 438
System requirements 24

T

T (convert value into text string) 805
T.DIST (t-distribution) 808
T.DIST.2T (t-distribution two-tailed) 808
T.DIST.RT (t-distribution right-tailed) 809
T.INV (percentiles of the t-distribution left-tailed) 814
T.INV.2T (percentiles of the t-distribution two-tailed) 815
T.TEST (t-test) 822
Tab width (in text frames) 475
Table (ribbon tab) 134
Tables in worksheets 132
  banded rows and columns 137
  cell references 141
  convert to range 135
  create 133
  delete rows/columns 135
  delete table 135
  expand automatically 135, 456
  first column/last column, highlight 137
  header row 137
  insert rows/columns 135
  name of table 141
  names and identifiers 141
  select rows/columns 135
  select table 135
  summarize as pivot table 134
  table name 137
  table properties 137
  table range 135
  table styles 137
  total row 137, 139
Tablet mode (Android/iOS) 458
Tabs for documents 38, 446
Tabs, close 449
TAN (tangent) 806
Tangent (TAN) 806
TANH (hyperbolic tangent) 807
TDIST (t-distribution) 807
t-distribution
  T.DIST 808
  T.DIST.2T (two-tailed) 808
  T.DIST.RT (right-tailed) 809
  T.INV (left-tailed) 814
  T.INV.2T (two-tailed) 815
  TDIST 807
  TINV 813
Technical support 23
Template folder for document templates 463
Templates
  document templates 231, 433
Text
  copy 75
  cut 75
  delete 75
  paste 75
TEXT (convert number into text string) 810
Text (number format) 192
Text (Object properties tab) 320
Text alignment horizontal (for objects) 219
Text alignment vertical (for objects) 281
Text direction 442, 443
Text filter (AutoFilter) 99
Text frames 292
  add/edit text 315
  change properties 292, 294
  change text area 281
  hyphenation 384
  insert 292
  paragraph format 217
Text input mode 39
Text rotation for text frames 281
Text strings
  compare (EXACT) 616
  concatenate (CONCAT) 563
  concatenate (CONCATENATE) 564
  concatenate (TEXTJOIN) 810
  convert into a number (NUMBERVALUE) 732
  convert into a number (VALUE) 825
  cut part (LEFT) 685
  cut part (MID) 703
  cut part (RIGHT) 766
  repeat n times (REPT) 766
Text styles 212
Text to columns, split 94
Text, enter in cell 63
TextArt objects
  change properties 320
  drawing 313
  transformation 320
TEXTJOIN (concatenate strings with separators) 810

TextMaker file format 433
Thousands separator
  change 456
  show 190
Three-dimensional calculations 57, 89
Threshold value (GESTEP) 639
Time
  current (static), insert via shortcut key 63
  enter in cell 63
  updated, with date: NOW function 729
TIME (create time) 811
TIMEDIFF (time difference) 812
TIMEVALUE (convert text into time) 813
TINV (percentiles of the t-distribution) 813
Title (of charts) 361
Title bar 30
TODAY (current date) 816
Toolbars for Android/iOS
  customize 34
  user interface 34, 458
Toolbars for classic menus
  create user-defined icons 503
  customize 494, 498
  delete 497
  locked 496
  manage 497
  position 496
  rename 497
  reset 497
  show/hide 494
  user interface 33, 458, 494
  visible in mode 494
Top row, freeze 182
Total row in tables 137, 139
Touch mode 458
Trace to error 110
Transformation (of TextArt objects) 320
Transparency (of pictures) 302
Transpose
  matrix (MTRANS) 816
  swap columns and rows 77, 94
TRANSPOSE (transpose matrix) 816
TREND (values of a linear regression) 817
Trendline (in charts) 338, 349
TRIM (remove spaces from text) 818
TRIMMEAN (mean without marginal values) 819
TRUE (logical value TRUE) 820
TRUNC (truncate a number) 820
TTEST (t-test) 821
TYPE (type of a value) 822

U

Underline (font) 212
Undo (changes) 66
Ungroup
  cells (outliner) 409
  objects 275
Unit conversion (CONVERT) 567
Unit of measure 458
Unpinned files 48
Unpinned templates 46
Up/down bars (in charts) 339, 345
Update
  charts 363, 477
  external references 518
  pivot tables 178
  worksheet (calculations) 114, 477
Updates 465
UPPER (convert text to uppercase) 823
Uppercase
  character format option 212
  PROPER (function) 755
  UPPER (function) 823
Use left Alt for shortcut keys 453
Use SHM extensions 465
Use system file dialogs 458
Use XIM 465
User dictionaries
  edit 382
  export/import 470
User info 453, 824
User info, output (USERFIELD) 824
User interface (ribbon or menus?) 29, 458
User interface on Android/iOS 34, 458
User interface size 458
USERFIELD (output user info) 824

V

VALUE (convert text into a number) 825
Value axis (in charts) 332, 352
VAR (variance of a sample) 826
VAR.P (variance of entire population) 828
VAR.S (variance of a sample) 830
VARA (variance of a sample) 827
Variance
  of a sample (VAR) 826
  of a sample (VAR.S) 830
  of a sample (VARA) 827
  of entire population (VAR.P) 828

Variance
of entire population (VARP) 827
of entire population (VARPA) 829
VARP (variance of entire population) 827
VARPA (variance of entire population) 829
VBA scripts 368, 440, 445
Version management 405, 464
Vertical alignment
of cell contents 205
of text in AutoShapes 281, 319
of text in text frames 281, 292, 294
Vertical text 205
View
edit mode 270
full screen 484
object mode 270
touch mode 458
windows 448
zoom level 483
View side by side (windows) 448
Visibility
objects 289
rows/columns 189
worksheet 87
Visual Basic 445
VLOOKUP (look up column-wise) 830

W

Walls (in charts) 344
Warning if a formula contains errors 456
Watch window 109
Weber function (BESSELY) 542
Week number
ISOWEEK 678
ISOWEEKNUM 679
WEEKNUM 833
WEEKDAY (weekday) 832
WEEKNUM (week number) 833
WEIBULL (Weibull distribution) 834
WEIBULL.DIST (Weibull distribution) 835
Whole word, select 453
Width of a column, change 187, 188
Window
close all 449
view side by side 448
Workbook 85
Workbook properties 473
Workbook protection 429
disable 430
enable 429
WORKDAY (date after x workdays) 836
Workdays
count (NETWORKDAYS) 721
date after x workdays (WORKDAY) 836
Worksheet 85, 481
alternating shade 481
change properties 87, 481
change sheet direction 481
copy 87
delete 87
hide sheet 87
insert 86
move 86, 87
protect 426
rename 86, 87
select multiple 86
show sheet 87
visibility 87
Worksheet register 39, 86
change color 481
display 475
font size 451
Worksheets
edit 61
format 186
protect 426, 430
select 67
update calculations 114, 477
Workspace color 458
Wrap text 205

X

X axis (in charts) 332, 351
XIRR (internal rate of return) 837
XLS file format 433
XLSX file format 433
XNPV (net present value) 837
XOR (logical XOR function) 838
XY scatter chart 326

Y

Y axis (in charts) 332, 352
YEAR (year of a date) 839

Z

Z axis (in charts) 332, 354, 356
Zero values, show/hide
for number formats of cells 190
worksheet properties 481
Zoom level (view) 483

z-test
Z.TEST 840
ZTEST 839