- Convert Drawing Unit (CU)
- Escape
- Reset / Idle (QQ)
- Drawing Preferences
- Application Preferences

- View
  - Draft Mode (DF)
  - Screen-based Linetypes (NL)
  - Anti-aliasing (NT)
  - Grid (GR)
  - Zoom
    - Zoom In
    - Zoom Out
    - Auto Zoom (ZA)
    - Zoom to Selection (ZS)
    - Previous View (ZV)
    - Window Zoom (ZW)
    - Pan Zoom (ZP)
  - Overlays
    - Direction (VD)
    - Order (VO)
    - Startpoint (VS)
  - Stored Views
    - Add View (VA)
    - Remove View (VR)
    - Edit View (VE)
    - Pick View (VP)
  - Isometric Grid Off (JO)
  - Top Projection (JT)
  - Right Projection (JR)
  - Left Projection (JL)
  - Layer List (GY)
  - Block List (GB)
  - View List (GV)
  - Property Editor (GP)
  - Selection Filter (GF)
  - Library Browser (GL)
  - Command Line (GM)
  - Clipboard Display (GC)
  - Status Bar (GS)
  - Focus on Command Line
  - Focus on Options Toolbar

        - Focus on Tool Matrix
        - Display Distance/Angle
    - Select
        - Deselect All (TN)
        - Select All (TA)
        - Select View (TV)
        - Invert Selection (TI)
        - (De-)Select Rectangular Area (TR)
        - (De-)Select Polygon (TP)
        - (De-)Select Contour (TC)
        - (De-)Select Intersected Entities (TX)
        - (De-)Select Layer (TL)
    - Draw
        - Point
            - Single Point (PO)
            - N Points on Line (PN)
            - MxN Points (PM)
        - Line
            - Line from 2 Points (LI)
            - Line from Angle (LA)
            - Horizontal Line (LH)
            - Vertical Line (LV)
            - Angle Bisector (LB)
            - Parallel (with Distance) (LP)
            - Parallel (through Point) (LG)
            - Tangent (Point, Circle) (LT1)
            - Tangent (Two Circles) (LT2)
            - Orthogonal / Tangent (LN)
            - Relative Angle (LR)
            - Orthogonal (LO)
            - Freehand Line (LF)
        - Arc
            - Center, Point, Angles (AR)
            - 2 Points and Radius (AD)
            - 2 Points and Angle (A2)
            - 2 Points and Length (AL)
            - 2 Points and Height (AH)
            - 3 Points (A3)
            - Concentric (with Distance) (AC)
            - Concentric (through Point) (AG)
            - Tangentially Connected (AN)
            - Tangent, Point, Radius (AT)

- Circle
  - Center, Point (CI)
  - Center, Radius (CR)
  - Center, Diameter (CA)
  - 2 Points and Radius (CD)
  - 2 Points (C2)
  - 3 Points (C3)
  - Concentric (with Distance) (CC)
  - Concentric (through Point) (CG)
  - Tangent and 2 Points (CT1)
  - Tangent, Point, Radius (CTP)
  - 2 Tangents and Point (CT2)
  - 2 Tangents and Radius (CTR)
  - 3 Tangents (CT3)
- Ellipse
  - Ellipse (Center, Point, Ratio) (EP)
  - Ellipse Arc (EA)
  - Ellipse with Radii (EI)
  - Ellipse with Diameters (ED)
  - Parallel Curve (with Distance) (EC)
  - Parallel Curve (through Point) (EG)
  - Inscribed in Quadrilateral (EQ)
- Spline
  - Spline (Control Points) (SP)
  - Spline (Fit Points) (SL)
  - Insert Fit Point (NI)
  - Append Fit Point(s) (NA)
  - Remove Fit Point (NR)
  - Simplify (NP)
- Polyline
  - Draw Polyline (PL)
  - Polyline from Segments (OG)
  - Polyline from Selection (OC)
  - Partial Polyline Along Selection (OL)
  - Insert Node (OA)
  - Append Node(s) (OP)
  - Delete Node(s) (OD)
  - Remove Nodes between Two Nodes (OB)
  - Trim Segments (OT)
  - Relocate Start Point (OR)
  - Change Segment Type (OX)
  - Arc to Line Segments

        - Offset (OQ)
        - Morph (OM)
        - Simplify (OS)
        - Normalize Polylines (ON)
        - Logically close Polylines (OE)
        - Logically open Polylines (OJ)
    - Shape
        - Rectangle (RE)
        - Rectangle with Size (RS)
        - Rectangle (3 Points) (R3)
        - Polygon (Center, Corner) (PG1)
        - Polygon (2 Points of Side) (PG2)
        - Polygon (Center, Side) (PG3)
        - Polygon (Side, Side) (PG4)
        - Star (PT)
    - Hatch
        - Hatch from Selection (HA)
        - Hatch from Segments (HS)
    - Viewport
        - Add Viewport (VT)
    - Ring (RI)
    - Text (TE)
    - Insert Bitmap (IM)
    - Trace Bitmap
- Dimension
    - Aligned (DA)
    - Rotated (Linear) (DL)
    - Horizontal (DH)
    - Vertical (DV)
    - Baseline (DB)
    - Continue (DC)
    - Ordinate (X/Y) (DO)
    - Leader (DE)
    - Datum (DT)
    - Tolerance (TO)
    - Radial (DR)
    - Diametric (DD)
    - Angular (DN)
    - Arc Length (DG)
    - Reset Label Position (DS)
    - Remove Style Overrides (DX)
- Modify

- Move / Copy (MV)
- Rotate (RO)
- Scale (SZ)
- Mirror (MI)
- Flip Horizontal (FH)
- Flip Vertical (FV)
- Move and Rotate (MR)
- Rotate Two (R2)
- Align Reference Points (AE)
- Align (MA)
- Offset (with Distance) (OF)
- Offset (through Point) (OH)
- Trim (RM)
- Trim Both (TM)
- Lengthen / Shorten (LE)
- Stretch (SS)
- Clip to Rectangle (CLR)
- Chamfer / Bevel (CH)
- Round (RN)
- Divide (DI)
- Split Entities (MS)
- Break out Segment (D2)
- Break out Manual (B2)
- Auto Trim (AX)
- Break out Gap (D3)
- Reverse (RV)
- Edit Text (MT)
- Edit Tolerance (ML)
- Edit Hatch (MH)
- Explode (XP)
- Draw Order
  - Bring to Front (MF)
  - Send to Back (MB)
  - Reorder (MO)
  - Order Connected Entities (MC)
- Detection
  - Detect Duplicates (MD)
  - Detect Zero-Length Entities (MZ)
- Projection
  - Orthographic Projection (Cylindrical) (PR)
  - Isometric Projection (PJ)
  - Isometric Projection (Cylindrical) (PC)

        - Matrix Transformation (MX)
    - Property Painter (PI)
- Snap
    - Free (SF)
    - Grid (SG)
    - End (SE)
    - On Entity (ST)
    - Perpendicular (SU)
    - Tangential (SB)
    - Center (SC)
    - Reference (SR)
    - Middle (SM)
    - Middle Manual (SN)
    - Distance (SD)
    - Distance Manual (SH)
    - Intersection (SI)
    - Intersection Manual (SY)
    - Auto (SA)
    - Coordinate (SX)
    - Polar Coordinate (SO)
    - X/Y from Points (.X)
    - Y/X from Points (.Y)
    - Center of Selection (SK)
    - Restrictions Off (EN)
    - Restrict Orthogonally (EO)
    - Restrict Horizontally (EH)
    - Restrict Vertically (EV)
    - Restrict Angle or Length (EL)
    - Set Relative Zero (RZ)
    - Lock Relative Zero (RL)
    - Lock Snap (SQ)
- Info
    - Position (IO)
    - Relative Position (IV)
    - Polar Position (IL)
    - Relative Polar Position (IT)
    - Distance Point to Point (IP)
    - Distance Entity to Point (IE)
    - Distance Entity to Entity (IN)
    - Angle (IA)
    - Total Length of Selection (IS)
    - Polygonal Area (IR)

        - Arc/Circle/Ellipse Area (IC)
        - Polyline Area (II)
    - Layer
        - Toggle Visibility (YV)
        - Toggle Frozen Status (YF)
        - Toggle Lock Status (YL)
        - Show Only Active (YO)
        - Show All Layers (YS)
        - Hide All Layers (YH)
        - Thaw All Layers (YW)
        - Freeze All Layers (YZ)
        - Lock All Layers (YK)
        - Unlock All Layers (YN)
        - Add Layer (YA)
        - Add Sublayer (YU)
        - Create Layer from Selection (YT)
        - Move Selection to Current Layer (YJ)
        - Duplicate Layer (YY)
        - Delete Layer(s) (YR)
        - Purge Unused Layers (YP)
        - Edit Layer (YE)
        - Select Layer Entities (YC)
        - Deselect Layer Entities (YD)
        - Layer States
            - Add... (YM)
    - Block
        - Explode (XP)
        - Toggle Visibility (BV)
        - Show All Blocks (BS)
        - Hide All Blocks (BH)
        - Add Empty Block (BA)
        - Create Block from Selection (BC)
        - Add Layout Block (BL)
        - Remove Block (BR)
        - Purge Unused Blocks (BP)
        - Rename Block (BN)
        - Edit Block (BE)
        - Edit Block from Reference (BD)
        - Edit Block In-place (BB)
        - Duplicate Block (BY)
        - Return to Main Drawing (EM)
        - Select Block References (BK)

        - Deselect Block References (BX)
        - Insert Block (BI)
        - Create Library Item (BT)
        - Attributes
            - Define Block Attribute (UA)
            - Synchronize Attributes (UY)
    - Window
        - Close All
        - Next
        - Previous
        - Full Screen
    - CAM
        - CAM Configuration... (KK)
        - CAM Export (KX)
        - CAM Legacy Export
        - G-Code Import...
        - Gerber Import...
        - Display Toolpaths (KV)
        - Add Tool (KA)
        - Edit Tool (KE)
        - Remove Tool (KR)
        - Export Tools (KO)
        - Import Tools (KI)
        - Add Profile Toolpath (KP)
        - Add Drill Toolpath (KL)
        - Edit Toolpath (KD)
        - Remove Toolpath (KM)
        - Move Toolpath Up
        - Move Toolpath Down
        - Regenerate Toolpath
        - Regenerate Toolpath from Tool
        - Regenerate All Toolpaths
        - Update Toolpath from Selection
        - Add Tab (KT)
        - Show CAM Tool List (GO)
        - Show CAM Toolpath List (GA)
        - Show 3D Simulation View (G3)
        - Show Simulation Panel (GI)
        - Nesting
        - Set Nesting Quantity

File Tools

Toolbar / Icon:

Commands: `filemenu`

Description

The file menu contains all tools to open, save, import, export and print files.

New

Toolbar / Icon:


Menu: `File > New`
Shortcuts: `Ctrl+N (Mac: ⌘N) | Ctrl+T (Mac: ⌘T)`
Commands: `new`

Description

Create new, empty drawings using this command. New drawings are completely empty (no entities, only one default layer, no blocks). As a frequent user of QCAD, you might instead want to create drawing templates with the layers and blocks you use the most and probably a drawing border. Once you have such a template you can load that instead of starting a new drawing from scratch.
Please refer also to the command New from Template to learn how to use drawing templates.

New from Template...

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `File > New from Template...`

Shortcut: `Ctrl+Shift+N (Mac: ⇧⌘N)`

Commands: `newfromtemplate`

Description

This tool opens a drawing template as a starting point for a new drawing. Any regular drawing can be used as drawing template if it is stored in the library/templates directory or another template directory configured in the application preferences dialog.

Open

Toolbar / Icon:

Menu: `File > Open`

Shortcut: `Ctrl+O (Mac: ⌘O)`

Commands: `open`

Description

Use this command to open a drawing file. The file is loaded and shown in a new window within the QCAD application window. You can switch between different loaded drawings using the menu "Windows" or the tabs at the top of the drawing area.

QCAD Professional

PDF Files

If you open a PDF file, QCAD show a dialog. In this dialog, you can specify the page to import, if you want to import images and if clipping should be applied. Imported images are

stored in a new folder called "pdf_images" located where the PDF file is located. Without clipping, all vector data is imported from the PDF file. This might include data outside the paper.

Editing CXF Fonts

QCAD comes with a collection of line fonts in its own CXF format. Line fonts are sometimes used in CAD instead of TrueType fonts. Line fonts are also often used for engraving or other further processing.
QCAD can load and edit fonts in CXF format. To load an existing font, simply use the menu File > Open and choose the format filter "CXF Font Files (*.cxf)". Then choose the font file you want to open. The fonts that come with QCAD are stored in directory "fonts" of your QCAD installation.
The font glyphs (representation of the various letters and symbols) are loaded into blocks. Make sure the block list is visible (View > Block List), so you can navigate through the font glyphs.
Each block has a name in the format "[Hexadecimal Unicode] [Symbol]", for example "0x0041 A" for a capital "A" with Unicode 65 (41 in hexadecimal).
By convention, a capital "A" should be constructed with a height of 9 drawing units. When the font is used for a text, it is scaled by QCAD according to the text height chosen.
Glyphs should have their reference point 0,0 at the lower left corner of the base line. In the CXF format, glyphs can contain lines, arcs and polylines. Splines may be used, but are converted into polylines during export.

Import

Toolbar / Icon:

Menu: `File > Import`
Shortcut: `Ctrl+Shift+I (Mac: ⇧⌘I)`
Commands: `import`

Description

The import command inserts a drawing from a file on disk into the current drawing.
All layers that are used by the imported drawing are added to the layer list of the current drawing. Existing layers with the same name can be overwritten if desired ("overwrite layers"

option in the options tool bar).
Block references of the imported drawing will be inserted together with the block definitions they refer to. Blocks in the current drawing can be overwritten if desired ("overwrite blocks" option in the options tool bar).
The options tool bar also offers some tools to scale, rotate or flip the imported drawing while positioning it.

Usage

Set the target point for the imported drawing with the mouse or enter a coordinate in the command line. The target point corresponds to the absolute zero point of the imported drawing.

Revert

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `File > Revert`
Shortcut: `Ctrl+R (Mac: ⌘R)`
Commands: `revert`

Description

Reloads the current drawing from disk. All unsaved changes are lost. Note that this operation cannot be undone. This function can for example be used to refresh the view if QCAD is used as file viewer for files which are modified by another application or process.

Show in Explorer / Finder / File Manager

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `File > Show in Explorer / Finder / File Manager`

Shortcut: `F, I`

Commands: `showfile`

Description

This tool can be used to open the folder that contains the current drawing in a file manager window. This can be Explorer under Windows, Finder under macOS or a file manager such as Nautilus under Linux. Explorer and Finder will also highlight the drawing file in the folder.

Save

Toolbar / Icon:

Menu: `File > Save`

Shortcut: `Ctrl+S (Mac: ⌘S)`

Commands: `save`

Description

This command saves the current drawing to the same file it was loaded from. If you want to save a newly created drawing or save the current drawing to a new file, use the menu File - Save As instead. You will then be asked for a file name before the drawing is saved.

Save As...

Toolbar / Icon:

Menu: `File > Save As...`
Shortcut: `Ctrl+Shift+S (Mac: ⇧⌘S)`
Commands: `saveas`

Description

Saves the current drawing as a new file. The dialog that is shown also allows you to choose the format and format version that you want to use to save the file.

Close

Toolbar / Icon:

Menu: `File > Close`
Shortcut: `Ctrl+W (Mac: ⌘W)`
Commands: `close`

Description

This command closes the current drawing. If the current drawing contains unsaved changes, you will be given the option to save the drawing before closing it or to discard those changes.

Bitmap Export...

Toolbar / Icon:
 
Menu: `File > Bitmap Export...`
Shortcut: `X, B`
Commands: `bitmapexport`

Description

This tool exports the current drawing as a bitmap file.

Usage

A dialog for selecting the output file is presented. After the file name has been specified, a second dialog asks you for the bitmap size and the preferred background color.
Note that exporting to bitmaps with a very large size can take a long time, depending on your hardware. The maximum size for bitmaps is width x height ≤ 2.147.483.647.

Quick SVG Export

Toolbar / Icon:
 
Add-on: **QCAD Professional**
Menu: `File > Quick SVG Export`
Shortcut: `X, S`
Commands: `svgexport`

Description

Exports the current drawing to an SVG file. While the SVG format is not ideal for storing CAD data, it is very popular for storing various types of vector graphics mainly for presentation and sometimes for further processing.
This tool uses the parameters that can be configured in the application preferences under Load/Save - Quick SVG Export.

Advanced SVG Export...

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `File > Advanced SVG Export...`

Commands: `svgexportas`

Description

Exports the current drawing in the SVG format. An advanced export dialog is shown that can be used to configure various options for the export.
Most notably, the option "Preserve Geometry" can be used to export the drawing for further processing. The visual appearance of the output might not exactly match that of the CAD drawing in this case, but as much of the geometry as possible is carried across to the SVG file.

SVG Import...

Toolbar / Icon:

Menu: `File > SVG Import...`

Commands: `svgimport`

Description

This tool imports an SVG file into the current drawing. Note that only a very limited set of SVG tags and arguments is supported. The idea is to import as much of the raw geometry in the SVG file as possible.

PDF Export

Toolbar / Icon:

Menu: `File > PDF Export`
Shortcut: `X, D`
Commands: `pdf`

Description

This tool exports the current drawing to a PDF file. Exporting to PDF is very similar to printing a drawing. You will most likely have to adjust your drawing scale and drawing position as well as page settings before exporting a drawing to PDF.
For this reason, QCAD automatically switches to the print preview when this tool is used. If the print preview does not match your expectations, cancel the export, adjust the page settings and start the PDF export again.

Print Preview

Toolbar / Icon:

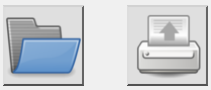
Menu: `File > Print Preview`
Shortcut: `Ctrl+Shift+P (Mac: ⇧⌘P)`
Commands: `printpreview`

Description

The print preview shows how your drawing will look like in the printout. Use the combo box in the options tool bar to adjust the scale factor of your drawing. This option does not actually scale or otherwise modify your drawing entities, it only sets up the scale factor that is applied for fitting your drawing on paper.
The buttons right of the scale combo box can be used to automatically fit your drawing on the chosen paper or to automatically add pages, so your entire drawing can be printed on multiple

pages.
After choosing the "Move Paper Position" tool, you can move the paper around freely by dragging it with the left mouse button. Click the right mouse button or hit the escape key to terminate the tool.
The options tool bar offers some other tools to quickly change some common drawing preferences that are also accessible in the drawing preferences dialog.

Print

Toolbar / Icon:


Menu: `File > Print`
Shortcut: `Ctrl+P (Mac: ⌘P)`
Commands: `print`

Description

Use this command to print a drawing. Make sure to always switch to the print preview before printing to check if the drawing scale and position is set up as desired.
A setup dialog is shown before anything is printed. Use this dialog to choose the printer and adjust printer specific settings.

Print Current View

Toolbar / Icon:


Menu: `File > Print Current View`
Shortcut: `Ctrl+Alt+P (Mac: ⌘Alt+P)`
Commands: `printview`

Description

Use this command to print the drawing portion that is currently shown in the graphics view.

Edit Tools

Toolbar / Icon:

Commands: `editmenu`

Description

The edit menu contains tools for basic editing such as copy / paste and deleting.

Application Preferences

Toolbar / Icon:


Menu: `Edit > Application Preferences`
Shortcut: `Ctrl+, (Mac: ⌘,)`
Commands: `preferences`

Description

In the application preferences dialog, you can adjust the look and behavior of QCAD and its tools and graphics views.
QCAD also stores some of these settings inside the drawings. Those settings can be changed in the application preferences as default settings for new drawings. In other words, they don't affect existing drawings but only drawings that are created after the preferences have been changed.

Undo [-]

Toolbar / Icon:

Menu: `Edit > Undo [-]`
Shortcuts: `Ctrl+Z (Mac: ⌘Z) | O, O`
Commands: `undo | oops`

Description

This tool takes back the last change of the drawing. QCAD can take back more than one command. For example if you have just created a line and a circle and you want to undo both, use the undo command twice.

Redo [-]

Toolbar / Icon:

Menu: `Edit > Redo [-]`
Shortcuts: `Ctrl+Shift+Z (Mac: ⇧⌘Z) | U, U`
Commands: `redo | uu`

Description

Redo reverts the effects of the previous Undo Action.

Delete

Toolbar / Icon:

Menu: `Edit > Delete`
Shortcuts: `Del | Backspace | E, R`
Commands: `delete | er`

Description

This tool deletes the entities that are currently selected.

Cut

Toolbar / Icon:

Menu: `Edit > Cut`
Shortcut: `Ctrl+X (Mac: ⌘X)`
Commands: `cut`

Description

The Cut tool works like the Copy tool with the only difference that the selected entities are removed from the current drawing after being copied to the clipboard.

Cut with Reference

Toolbar / Icon:


Menu: `Edit > Cut with Reference`
Shortcuts: `Ctrl+Shift+X (Mac: ⇧⌘X) | R, T`
Commands: `cutwithreference | rt`

Description

The Cut with Reference tool works like the Copy with Reference tool with the only difference that the selected entities are removed from the current drawing after being copied to the clipboard.

Copy

Toolbar / Icon:


Menu: `Edit > Copy`
Shortcuts: `Ctrl+C (Mac: ⌘C) | C, P`
Commands: `copy | cp`

Description

QCAD offers a set of tools to copy entities from one drawing to another, similar like other applications.

Usage

1. Use the selection tools to prepare a selection of entities you want to copy to the clipboard.
2. Choose the copy tool.

The standard copy tool of QCAD acts like the copy tool of other applications. It does not allow you to choose a reference point for your selection but instead computes the reference point automatically at the center of the selection. This is not always desirable as you most likely

want to place the pasted entities at a precisely defined position. For this, QCAD offers and alternative, more CAD specific copy tool: Copy with Reference.

The selection is now on the QCAD internal clipboard and can be pasted into the same or a different drawing that is opened with the same running instance of QCAD. Note that if you quit the QCAD application, the contents of the clipboard is lost. The QCAD clipboard can only be accessed by QCAD itself. You cannot exchange drawing entities with other applications through copy and paste. Please use the export and import functionality of QCAD for this.

Copy with Reference

Toolbar / Icon:


Menu: `Edit > Copy with Reference`
Shortcuts: `Ctrl+Shift+C (Mac: ⇧⌘C) | R, C`
Commands: `copywithreference | rc`

Description

Copies the current selection to the QCAD clipboard. This tool lets you specify a reference point that is used to position the selection when pasting it into a drawing.

Usage

1. Use the selection tools to prepare a selection of entities you want to copy to the clipboard.
2. Start the Copy with Reference tool.
3. Specify the reference point that you want to use when pasting the selection.

Paste

Toolbar / Icon:
 

Menu: `Edit > Paste`
Shortcuts: `Ctrl+V (Mac: ⌘V) | P, S`
Commands: `paste | ps`

Description

The paste command inserts the entities that were previously copied to the clipboard. Paste is especially useful to transfer entities from one drawing to another.
All layers that are used by the clipboard contents are pasted into the layer list of the current drawing. Existing layers with the same name can be overwritten if desired ("overwrite layers" option in the options tool bar).
Block references that are on the clipboard will be pasted together with the block definitions they refer to. Blocks in the current drawing can be overwritten if desired ("overwrite blocks" option in the options tool bar).
The options tool bar also offers some tools to scale, rotate or flip the clipboard contents while pasting.

Usage

Set the target point for the pasted entities with the mouse or enter a coordinate in the command line. The target point corresponds to the reference point that was chosen when copying or cutting the entities if Copy / Cut with Reference was used. If Cut or Copy without reference point was used, the center of the selection is used as reference point.

Paste along Entity

Toolbar / Icon:


Add-on: QCAD Professional
Menu: `Edit > Paste along Entity`
Shortcuts: `Ctrl+Shift+V (Mac: ⇧⌘V) | P, E`
Commands: `pastealongentity`

Description

This tool pastes the clipboard contents along an existing line, arc, polyline or spline entity. The clipboard is pasted multiple times in such a way that the reference point of the clipboard contents is placed on the entity.

The clipboard can be inserted multiple times with a given distance between items or a given number of times, equally spread along the entity.

The "Align" option can be used to rotate the clipboard to align its angle with the angle along the entity.

The "Include end points" option causes the first and last items to be placed on the start and end point of the entity.

In addition, the clipboard can be scaled or rotated while pasting.

Duplicate

Toolbar / Icon:


Menu: `Edit > Duplicate`
Shortcut: `D, P`
Commands: `duplicate | dp`

Description

This tool duplicates the selected entities and selects the created copy.

Usage

1. Use the selection tools to prepare a selection of entities you want to duplicate.
2. Choose the duplication tool.

Quick Modify Tools

Toolbar / Icon:

Shortcut: `W, Q`
Commands: `quickmodifymenu`

Move Left

Toolbar / Icon:

Add-on: **QCAD Professional**
Menu: `Edit > Quick Modify > Move Left`
Shortcut: ←

Description

This function can be used to quickly move the current selection by a previously configured distance.
The distance can be configured under Edit > Application Preferences > Edit > Move.

Move Right

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Edit > Quick Modify > Move Right`

Shortcut: →

Description

This function can be used to quickly move the current selection by a previously configured distance.
The distance can be configured under Edit > Application Preferences > Edit > Move.

Move Up

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Edit > Quick Modify > Move Up`

Shortcut: ↑

Description

This function can be used to quickly move the current selection by a previously configured distance.
The distance can be configured under Edit > Application Preferences > Edit > Move.

Move Down

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Edit > Quick Modify > Move Down`
Shortcut: `↓`

Description

This function can be used to quickly move the current selection by a previously configured distance.
The distance can be configured under Edit > Application Preferences > Edit > Move.

Rotate Counter-Clockwise

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Edit > Quick Modify > Rotate Counter-Clockwise`
Shortcut: `F5`
Commands: `rotateccw`

Description

This function can be used to quickly rotate the current selection by a previously configured angle or to rotate entities on the fly while pasting or inserting a block or library item.
The angle can be configured under Edit > Application Preferences > Edit > Rotate.

Rotate Clockwise

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Edit > Quick Modify > Rotate Clockwise`
Shortcut: `F6`
Commands: `rotatecw`

Description

This function can be used to quickly rotate the current selection by a previously configured angle or to rotate entities on the fly while pasting or inserting a block or library item.
The angle can be configured under Edit > Application Preferences > Edit > Rotate.

Find/Replace

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Edit > Find/Replace`
Shortcuts: `Ctrl+F (Mac: ⌘F) | R, P`

Description

This tool replaces all occurrences of a text string in text entities with another text string. If one or more entities are selected, the search is limited to only those entities. If nothing is selected, all text entities are searched. Entities on locked or invisible layers are not searched.

Substitute Fonts

Toolbar / Icon:
 

Add-on: QCAD Professional
Menu: `Edit > Substitute Fonts`
Shortcut: `F, F`

Description

With this tool, you can easily replace all uses of a particular font with another one. This is particularly useful if an imported drawing uses fonts which are not available on your system or which do not support all glyphs that are required.

Scale Text Heights

Toolbar / Icon:
 

Add-on: QCAD Professional
Menu: `Edit > Scale Text Heights`
Shortcut: `M, G`
Commands: `scaletextheight | mg`

Description

This tool scales the text height of all selected texts, without changing the position of the texts.

Convert Drawing Unit

Toolbar / Icon:


Menu: `Edit > Convert Drawing Unit`
Shortcut: `C, U`
Commands: `convertunit | cu`

Escape

Toolbar / Icon:

Menu: `Edit > Escape`
Shortcut: `Esc`
Commands: `escape`

Description

The escape key can be used to terminate the current tool and return to the idle state of QCAD. If the current tool requires multiple steps, escape returns to the previous step of the tool. To terminate the tool, escape might have to be pressed multiple times in this case.

Reset / Idle

Toolbar / Icon:

Menu: `Edit > Reset / Idle`
Shortcut: `Q, Q`
Commands: `reset | qq`

Description

This tool terminates all active tools and returns QCAD to its idle state in which you can use the mouse cursor to select entities, drag and drop entities, etc.

Drawing Preferences

Toolbar / Icon:

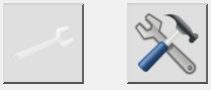
Menu: `Edit > Drawing Preferences`
Shortcut: `Ctrl+I (Mac: ⌘I)`
Commands: `drawingpreferences`

Description

The drawing preferences dialog allows you to change preferences that affect various aspects of the current drawing. These preferences are stored in your drawing.

Application Preferences

Toolbar / Icon:



Menu: `Edit > Application Preferences`
Shortcut: `Ctrl+, (Mac: ⌘,)`
Commands: `preferences`

Description

In the application preferences dialog, you can adjust the look and behavior of QCAD and its tools and graphics views.
QCAD also stores some of these settings inside the drawings. Those settings can be changed in the application preferences as default settings for new drawings. In other words,

they don't affect existing drawings but only drawings that are created after the preferences have been changed.

View Tools

Toolbar / Icon:

Shortcut: `W, V`
Commands: `viewmenu`

Description

The view menu contains all tools for zooming, panning and adjusting the display.

Draft Mode

Toolbar / Icon:


Menu: `View > Draft Mode`
Shortcut: `D, F`
Commands: `draftmode | df`

Description

Toggles the draft mode of the current drawing. In draft mode, all lines are shown with a width of 1 pixel. Large texts are simplified. Use the draft mode if your drawing becomes very large and it takes long to redraw it.

Screen-based Linetypes

Toolbar / Icon:
 

Menu: `View > Screen-based Linetypes`
Shortcut: `N, L`
Commands: `screenlinetype | nl`

Description

Toggles the screen-based linetype mode of the current drawing.
If this mode is activated, linetypes are optimized for a computer screen. All lines are displayed with a screen based line width (lines don't get wider when zooming in) and all patterns are shown as fixed, pixel based patterns on screen (dashes don't get longer when zooming in).
If this mode is off, line widths and linetypes are shown in drawing units (default).

Anti-aliasing

Toolbar / Icon:
 

Menu: `View > Anti-aliasing`
Shortcut: `N, T`
Commands: `antialiasing | nt`

Description

Toggles anti-aliasing for the current drawing. With anti-aliasing, skewed lines, arcs and texts are displayed smoother.

Grid

Toolbar / Icon:

Menu: `View > Grid`

Shortcut: `G, R`

Description

Toggles the grid visibility of the current drawing.

Zoom Tools

Toolbar / Icon:

Shortcut: `W, Z`

Commands: `zoommenu`

Zoom In

Toolbar / Icon:

Menu: `View > Zoom > Zoom In`

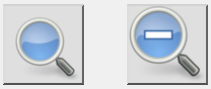
Shortcuts: `+ | = | Ctrl++ (Mac: ⌘+)`

Commands: `zoomin`

Description

This tool increases the current viewing factor. The same effect can also be achieved by turning the mouse wheel away from you.

Zoom Out

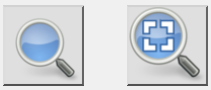
Toolbar / Icon:


Menu: `View > Zoom > Zoom Out`
Shortcuts: `-` | `Ctrl+-` (Mac: ⌘-)
Commands: `zoomout`

Description

This tool decreases the current viewing factor. The same effect can also be achieved by turning the mouse wheel toward you.

Auto Zoom

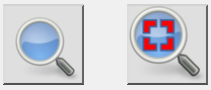
Toolbar / Icon:


Menu: `View > Zoom > Auto Zoom`
Shortcut: `Z, A`
Commands: `zoomauto` | `za`

Description

Scales the drawing view so that all entities that are on visible layers fit on the screen.

Zoom to Selection

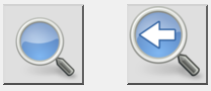
Toolbar / Icon:


Menu: `View > Zoom > Zoom to Selection`
Shortcut: `Z, S`
Commands: `zoomselection | zs`

Description

This tool zooms to the current selection, ensuring that all selected entities are visible.

Previous View

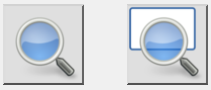
Toolbar / Icon:


Menu: `View > Zoom > Previous View`
Shortcut: `Z, V`
Commands: `zoomprevious | zv`

Description

Shows the last used view. Use this to switch back to the previous view for example after performing an auto zoom or after zooming in with the window zoom.

Window Zoom

Toolbar / Icon:


Menu: `View > Zoom > Window Zoom`
Shortcut: `Z, W`
Commands: `zoomwindow | zw`

Description

This tool offers a quick way to zoom in on a certain area of the drawing.

Usage

1. Specify the first corner of the area you want to view. Press the left mouse button at that corner and keep it down.
2. Drag the mouse to the second corner of the area.
3. Let go of the mouse button at the second corner.

Note: Alternatively you can specify the first and the second corner each with a single mouse click instead of dragging the mouse around with the mouse button pressed.

Pan Zoom

Toolbar / Icon:


Menu: `View > Zoom > Pan Zoom`
Shortcut: `Z, P`
Commands: `zoompan | zp`

Description

Panning means moving (scrolling) around in a drawing. The quickest way to do so is using the middle mouse button and move the view similar like you would move a paper around: press the middle mouse button and hold it while moving the drawing around. If your mouse does not have a middle mouse button you can press the Control key (Mac OS X: Command

key) and use the left mouse button instead.
Alternatively, you can activate this tool and then move the view with the left mouse button without pressing any keys.
Click the right mouse button when you are done.

Overlay Tools

Toolbar / Icon:


Commands: `overlaymenu`

Direction

Toolbar / Icon:


Add-on: QCAD Professional
Menu: `View > Overlays > Direction`
Shortcut: `V, D`
Commands: `overlaydirection | vd`

Description

This tool shows a direction overlay over the current drawing. An arrow is shown over each entity with a direction. Entities with a direction are: lines, arcs, ellipse arcs, splines and polylines.

Order

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `View > Overlays > Order`

Shortcut: `V, O`

Commands: `overlayorder | vo`

Description

This tool shows an order overlay over the current drawing. An order number is shown over each entity.

Startpoint

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `View > Overlays > Startpoint`

Shortcut: `V, S`

Commands: `overlaystartpoint | vs`

Description

This tool shows a startpoint overlay over the current drawing. An arrow is shown over each startpoint of an entity with a start / end point. Entities with a startpoint are: lines, arcs, ellipse arcs, splines and polylines.

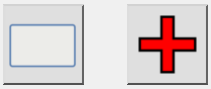
Stored Views Tools

Toolbar / Icon:

Commands: `storedviewsmenu`

Add View

Toolbar / Icon:



Add-on: QCAD Professional

Menu: `View > Stored Views > Add View`

Shortcut: `V, A`

Commands: `viewadd`

Description

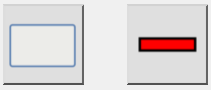
Saves the current view (zoom and offset) as an entry in the view list. Saved views can quickly be recovered by clicking the view in the view list instead of zooming and scrolling.

Usage

1. Zoom in on a part of your drawing you frequently want to view.
2. Launch this tool.
3. Enter a name for the view and click OK.

Remove View

Toolbar / Icon:



Add-on: QCAD Professional

Menu: `View > Stored Views > Remove View`

Shortcut: `V, R`

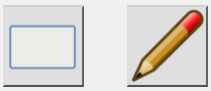
Commands: `viewremove`

Description

Removes the active view from the view list.

Edit View

Toolbar / Icon:



Add-on: QCAD Professional

Menu: `View > Stored Views > Edit View`

Shortcut: `V, E`

Commands: `viewedit`

Description

This tool edits the name of a previously saved view.

Usage

1. Activate that view you want to edit in the view list.
2. Launch this tool.
3. Edit the name of the view and click "OK".

Pick View

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `View > Stored Views > Pick View`

Shortcut: `V, P`

Commands: `viewpick`

Description

This tool updates an existing saved view to match the current view.

Usage

1. Activate the view you want to update in the view list.
2. Zoom and pan to display the new view you want to use.
3. Launch this tool to update the active view in the view list.

Isometric Grid Off

Toolbar / Icon:

Menu: `View > Isometric Grid Off`

Shortcut: `J, O`

Commands: `isometricgridoff | jo`

Top Projection

Toolbar / Icon:

Menu: `View > Top Projection`
Shortcut: `J, T`
Commands: `isometricgridtop | jt`

Right Projection

Toolbar / Icon:

Menu: `View > Right Projection`
Shortcut: `J, R`
Commands: `isometricgridright | jr`

Left Projection

Toolbar / Icon:

Menu: `View > Left Projection`
Shortcut: `J, L`
Commands: `isometricgridleft | jl`

Layer List

Toolbar / Icon:

Menu: `View > Layer List`
Shortcut: `G, Y`
Commands: `gy`

Description

This tool shows / hides the layer list widget.

QCAD Professional

By default, the layer list shows columns for the layer visibility (on/off) and the layer lock (locked/unlocked). More columns for other layer states (frozen, snappable, plottable) can be shown under Edit > Application Preferences > Layer List.

Block List

Toolbar / Icon:

Menu: `View > Block List`
Shortcut: `G, B`
Commands: `gb`

View List

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `View > View List`

Shortcut: `G, V`

Commands: `gv`

Property Editor

Toolbar / Icon:

Menu: `View > Property Editor`

Shortcut: `G, P`

Commands: `gp`

Selection Filter

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `View > Selection Filter`

Shortcut: `G, F`

Commands: `gf`

Description

The selection filter widget can be used to select entities based on their type or their properties. With the selection filter, you can for example easily select all circles with a radius between 3 and 4 or all text entities with a height of 5 units.
You can also remove entities from a selection or select only those entities that match the filter and are already selected (intersection).

Usage

1. Display the selection filter widget using View > Selection Filter.
2. Choose the entity type to filter by or select "Any" to match any entity type.
3. Choose the property name of the property to filter by.
4. Choose the comparison operator. Depending on the property type chosen, you can use logical comparisons, mathematical comparisons (=, ≠, >, ≥, < ≤) or string comparisons (is, is not, contains, contains not, starts with, ends with, regular expression match).
5. Choose the value to compare with.
6. Click one of the buttons at the bottom to apply the filter.
   The first button at the left selects all entities that match the filter. The existing selection is ignored.
   The second button adds the entities that match to the selection.
   The third button deselects the matching entities.
   The last button selects only those entities that match the filter and are already selected (boolean intersection).

Examples

- **To select all texts with text height 5**
  Entity type: Text
  Property: Text Height
  Comparison: = (equal to)
  Value: 5
- **To select all texts with text height larger than 5**
  Entity type: Text
  Property: Text Height
  Comparison: > (greater than)
  Value: 5
- **To select all circles with a radius between 5 and 6 (inclusive)**
  Entity type: Circle
  Property: Radius
  Comparison: ≥ (greater than or equal to)
  Value: 5
  Click first button at the left
  Entity type: Circle

Property: Radius
Comparison: ≤ (less than or equal to)
Value: 6
Click last button at the right

Special properties

The property color matches entities by their color value. This can be "By Layer" in which case the displayed color is irrelevant. To match by the displayed color (regardless if the value is "By Layer" or fixed), choose the property "Displayed Color" instead.

The boundary and size properties can be used to select entities based on their location or size.

Library Browser

Toolbar / Icon:


Add-on: QCAD Professional
Menu: `View > Library Browser`
Shortcut: `G, L`
Commands: `gl`

Description

The part libraries of QCAD are collections of symbols and other drawing files that can be used in your drawings. To insert an item from a part library, you need to show the library browser first using this tool.
Parts can be inserted into your drawing using drag and drop. Once the mouse cursor is inside the drawing area, you can use the regular snap tools to position the inserted part and the options tool bar to scale, flip or rotate the part.

Command Line

Toolbar / Icon:

Menu: `View > Command Line`
Shortcut: `G, M`
Commands: `gm`

Description

The command line of QCAD allows you to start commands, enter coordinates or enter values such as distances or radii.

Entering Coordinates

Absolute coordinates are entered in the format "x,y":

40,5

Relative coordinates are entered in the format "@x,y":

@10,6

Absolute polar coordinates are entered in the format "distance<angle":

10<30

Relative polar coordinates are entered in the format "@distance<angle":

@10<45

When entering coordinates or values, mathematical expressions may be used to calculate a coordinate based on known values. For example the coordinate 10,5 may also be entered as:

5+5,30/6

Calculator

The command line can also be used as a calculator. To do this, enter a mathematical expression proceeded by an equal sign:

=3+4
7

Variables may be used to store values:

=a=5+6
11
=a/2
5.5

Mathematical Expressions

Mathematical constants available are:

PI, LN2, LN10, LOG2E, LOG10E, SQRT1_2, SQRT2

Mathematical functions available are:

abs, ceil, floor, exp, log, max, min, pow, sqrt, random, round, rad2deg, deg2rad, sin, cos, tan, asin, acos, atan, atan2, log10, log1p, log2, sign, cosh, sinh, tanh, acosh, asinh, atanh, expm1, hypot, cbrt, trunc

Most of these constants and functions are standard ECMAScript (JavaScript) functions and documented online. In standard ECMAScript, these functions are part of the Math class so the function *abs* would have to written as *Math.abs*. In the QCAD command line, you can leave away the *Math.* part for convenience. Trigonometric functions (sin, cos, tan, asin, acos, atan, atan2) accept or return angles in degrees. If you prefer the radian versions of these functions, use the original *Math.* functions instead.

The functions *rad2deg* and *deg2rad* can be used to convert angles between radian and degrees.

Clipboard Display

Toolbar / Icon:

Add-on: **QCAD Professional**
Menu: `View > Clipboard Display`
Shortcut: `G, C`
Commands: `gc`

Status Bar

Menu: `View > Status Bar`
Shortcut: `G, S`
Commands: `gs`

Focus on Command Line

Toolbar / Icon:


Menu: `View > Focus on Command Line`
Shortcuts: `Space | Ctrl+M (Mac: ⌘M)`

Description

This command activates the command line for input, for example to enter a coordinate when specifying a point. This is usually done by pressing the Space key.

Focus on Options Toolbar

Toolbar / Icon:


Menu: `View > Focus on Options Toolbar`
Shortcut: `Meta+Space`

Focus on Tool Matrix

Toolbar / Icon:

Menu: `View > Focus on Tool Matrix`
Shortcut: `Meta+Shift+Space (Mac: Meta+⇧Space)`

Description

This command activates the filter (search) field of the tool matrix for input.

Display Distance/Angle

Toolbar / Icon:

Menu: `View > Display Distance/Angle`
Shortcut: `F8`

Selection Tools

Toolbar / Icon:

Shortcut: `W, S`
Commands: `selectionmenu`

Description

Cross Selection

Some selection tools can be used in cross selection mode. In this mode, not only entities that are completely inside a given area are selected but also entities that are only partly inside the area. This selection is also known as "cross selection".

Selection Mode

Some selection tools allow you to choose a selection mode in the options toolbar. The selection modes that are available are:

- Replace selection:
  
  Replaces the current selection with the new selection made by the active selection tool (default).
- Add to selection:
  
  Adds the selection to the current selection.
- Remove from selection:
  
  Removes (subtracts) the selection from the current selection.
- Intersect:
  
  Only entities that were already selected and that match the criteria of the selection tool are selected.

Deselect All

Toolbar / Icon:


Menu: `Select > Deselect All`
Shortcuts: `T, N | Ctrl+K (Mac: ⌘K) | Ctrl+D (Mac: ⌘D) | Ctrl+Shift+A (Mac: ⇧⌘A)`
Commands: `deselectall | tn`

Description

Deselects all entities. Use this tool to make sure that no entities are selected. Alternatively, you can also click into an empty area of your drawing.

Select All

Toolbar / Icon:

Menu: `Select > Select All`
Shortcuts: `T, A | Ctrl+A (Mac: ⌘A)`
Commands: `selectall | ta`

Description

Selects all entities on all visible, unlocked layers.

Select View

Toolbar / Icon:

Add-on: **QCAD Professional**
Menu: `Select > Select View`
Shortcut: `T, V`
Commands: `selectview | tv`

Description

Selects all entities that are completely inside the currently visible area.

Invert Selection

Toolbar / Icon:

Menu: `Select > Invert Selection`
Shortcut: `T, I`
Commands: `selectinvert | invertselection | ti`

Description

Selects all entities that are not currently selected and deselects all selected entities.

(De-)Select Rectangular Area

Toolbar / Icon:

Menu: `Select > (De-)Select Rectangular Area`
Shortcuts: `T, R | T, W`
Commands: `selectrectangle | selectwindow | tr | tw`

Description

Selects all entities that are within a given rectangular area.

Usage

1. Choose the selection mode in the options tool bar.
2. Tick the check box for cross selection to also select entities that intersect the given rectangle.
3. Use the mouse to draw the selection rectangle. All matching entities are selected or deselected depending on the selection mode.

(De-)Select Polygon

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Select > (De-)Select Polygon`

Shortcut: `T, P`

Commands: `selectpolygon | tp`

Description

Selects or deselects all entities that are within a given polygon.

Usage

1. Choose the selection mode in the options tool bar.
2. Tick the check box for "cross selection" to also select entities that intersect the given polygon.
3. Use the mouse to draw the selection polygon. All matching entities are selected or deselected depending on the selection mode.

(De-)Select Contour

Toolbar / Icon:

Menu: `Select > (De-)Select Contour`

Shortcut: `T, C`

Commands: `selectcontour | tc`

Description

Selects or deselects entities that are connected to each other and form a contour (closed or open).

Usage

1. Choose the selection mode in the options tool bar.
2. Use the mouse to pick one entity of the contour you want to select. The tool then searches from the endpoints of the selected entity in both directions for entities that are directly or indirectly connected to this entity. All connected entities are selected or deselected depending on the chosen selection mode.

Instead of using this tool, you can simply double-click an entity to select it and all connected entities.

(De-)Select Intersected Entities

Toolbar / Icon:


Menu: `Select > (De-)Select Intersected Entities`
Shortcut: `T, X`
Commands: `selectintersected | tx | ux`

Description

Selects or deselects all entities that are intersected by a line.

Usage

1. Choose the selection mode in the options tool bar.
2. Click the start point of the intersection line.
3. Click the end point of the intersection line.
4. All intersected entities are selected or deselected depending on the selection mode.

(De-)Select Layer

Toolbar / Icon:

Menu: `Select > (De-)Select Layer`
Shortcut: `T, L`
Commands: `selectlayerbyentity | tl`

Description

Selects or deselects all entities on the same layer as a chosen entity.

Usage

1. Choose the selection mode in the options tool bar.
2. Use the mouse to pick one entity on the layer you want to (de-)select. All entities that are on the same layer as the entity you have picked are selected or deselected depending on the selection mode.

Drawing Tools

Point Tools

Toolbar / Icon:

Shortcut: `W, P`
Commands: `pointmenu`

Single Point

Toolbar / Icon:
 
Menu: `Draw > Point > Single Point`
Shortcut: `P, O`
Commands: `point | po`

Description

This command is used to draw single points. Points are visually represented by a small cross.

Usage

- Use the mouse to specify the location of the point or enter a coordinate in the command line.

N Points on Line

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Point > N Points on Line`
Shortcut: `P, N`
Commands: `npoints | pn`

Description

With this tool you can draw a given number of points that are evenly distributed on a line.

Usage

1. Enter the number of points you want to create.

2. Specify the start and end point of the line with the mouse or by entering the coordinates in the command line.

MxN Points

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Point > MxN Points`
Shortcut: `P, M`
Commands: `mnpoints | pm`

Description

With this tool you can draw a grid of N times M points. The points are evenly distributed in an irregular quadrilateral.

Usage

1. Enter the number of points along the first side of the quadrilateral as "M" in the options tool bar.
2. Enter the number of points along the second side of the quadrilateral as "N" in the options tool bar.
3. Specify the four corner points of the quadrilateral with the mouse or by entering their coordinates in the command line.

Line Tools

Toolbar / Icon:

Shortcut: `W, L`
Commands: `linemenu`

Description

Line Types

Most line tools allow you to choose the type of line to create in the options toolbar. The line types available are:

- Auto:

  Automatically create a line of the same type as a chosen other line. This applies for the parallel line tool.
- Line Segment:

  Creates line segments from a start point to an end point.
- Infinite Line:

  Creates infinite lines that go through two given points. These are sometimes called X-lines or construction lines.
- Rays:

  Creates rays from a given start point, through another point with an infinite length.

Line from 2 Points

Toolbar / Icon:

Menu: `Draw > Line > Line from 2 Points`
Shortcut: `L, I`
Commands: `line | ln | li | l`

Description

This tool lets you draw a sequence of one or more straight lines.

Usage

1. Choose the desired line type in the options toolbar.
2. Specify the start point of the first line segment. You can use the mouse or enter a coordinate in the console.
3. Specify the endpoint of the first line segment.
4. Specify the endpoints of additional line segments. Click the "Close" button in the options tool bar to close the sequence:

   If you need to undo a single line segment, you can do so by clicking the "Undo" button:

   
5. You can set the length or angle of the line to fixed values using the appropriate inputs in the options toolbar.

Line from Angle

Toolbar / Icon:

Menu: `Draw > Line > Line from Angle`
Shortcut: `L, A`
Commands: `lineangle | la`

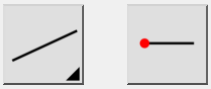
Description

Use this tool to create lines with a given angle. The line is usually trimmed to the desired length after creating it.

Usage

1. Choose the desired line type in the options toolbar.
2. Enter the angle and length of the line in the options tool bar.
3. In the options tool bar, choose the reference point on the line which you want to use for positioning the line. 'Start' means that the line will have its start point at the point where you position it.
4. Place the line with the mouse or by entering a coordinate in the command line.

Horizontal Line

Toolbar / Icon:


Menu: `Draw > Line > Horizontal Line`
Shortcut: `L, H`
Commands: `linehorizontal | lh`

Description

Use this tool to create horizontal lines. This tool is used like the tool for lines at any given angle, except that the angle is fixed to be horizontal.

Vertical Line

Toolbar / Icon:

Menu: `Draw > Line > Vertical Line`
Shortcut: `L, V`
Commands: `linevertical | lv`

Description

Use this tool to create vertical lines. This tool is used like the tool for lines at a given angle, except that the angle is fixed to be vertical.

Angle Bisector

Toolbar / Icon:

Menu: `Draw > Line > Angle Bisector`
Shortcut: `L, B`
Commands: `linebisector | bisector | lb`

Description

Use this tool for creating angle bisectors between two line entities.

Usage

1. Choose the desired line type in the options toolbar.
2. In the options tool bar enter the length of the bisector(s), starting from the intersection point of the two lines. In the second text box, choose the number of angle bisectors you want to create. The default is '1' but you can also create multiple bisectors as shown

here:

3. Click the first line entity which limits the angle.
4. Click the second line entity with the mouse cursor on that side of the line on which you want to create the angle bisector(s). The preview shows the angle bisector(s) before they are created.

Parallel (with Distance)

Toolbar / Icon:

Menu: `Draw > Line > Parallel (with Distance)`
Shortcuts: `L, P | P, A`
Commands: `lineparallel | lineoffset | parallel | par | lp | pa`

Description

With this tool you can create parallels to existing lines (or concentric arcs and circles).

Usage

1. Choose the desired line type in the options toolbar.
2. Enter the distance of the concentric or parallel entity from the original entity in the options tool bar shown at the top.
3. Enter the number of parallel or concentric entities to create in the options tool bar.
4. Click the base entity. The parallel or concentric entities are created on that side on which the mouse cursor is located while selecting the entity.

Parallel (through Point)

Toolbar / Icon:

Menu: `Draw > Line > Parallel (through Point)`
Shortcut: `L, G`
Commands: `lineparallelthrough | lineoffsetthrough | parallelthrough | lg`

Description

With this tool you can create parallels to existing lines or concentric arcs and circles. The parallel or concentric arc or circle goes through a given point.

Usage

1. Choose the desired line type in the options toolbar.
2. Enter the number of entities to create in the options tool bar.
3. Click the base entity.
4. Click the position through which the first parallel or concentric arc or circle runs.

Tangent (Point, Circle)

Toolbar / Icon:

Menu: `Draw > Line > Tangent (Point, Circle)`
Shortcut: `L, T, 1`
Commands: `linetangent | tangent | lt1`

Description

Create tangents from a coordinate to an existing arc, circle or ellipse entity with this tool.

Usage

1. Choose the desired line type in the options toolbar.

2. Use the mouse to specify the location of the start point of the line or enter a coordinate in the command line.
3. Click the entity to which you want to create the tangent. Usually, two tangents are possible. When moving the mouse around, you can see a preview of the tangent that will be created.

Tangent (Two Circles)

Toolbar / Icon:

Menu: `Draw > Line > Tangent (Two Circles)`
Shortcut: `L, T, 2`
Commands: `linetangent2 | tangent2 | lt2`

Description

Create tangents from one existing arc or circle entity to another one with this tool.

Usage

1. Choose the desired line type in the options toolbar.
2. Click the first arc or circle entity to which you want to create the tangent.
3. Click the second arc or circle entity to which you want to create the tangent. Usually, four tangents are possible. When moving the mouse around, you can see a preview of the tangent that will be created.

Orthogonal / Tangent

Toolbar / Icon:
 

Menu: `Draw > Line > Orthogonal / Tangent`
Shortcut: `L, N`
Commands: `lineorthogonaltangent | orthotangent`

Description

Use this tool to create a line that is orthogonal to another line and tangent to an existing arc, circle or ellipse entity.

Usage

1. Choose the desired line type in the options toolbar.
2. Click the line entity to which the new line should be orthogonal.
3. Click the arc, circle or ellipse entity to which the line should be a tangent. There are two possible solutions. When moving the mouse around, you can see a preview of the tangent that will be created.

Relative Angle

Toolbar / Icon:
 

Menu: `Draw > Line > Relative Angle`
Shortcut: `L, R`
Commands: `linerelativeangle | lr`

Description

Create lines with a relative angle to existing entities with this tool. The existing entity can be a line or an arc / circle. Lines with a relative angle of 0 degrees to an arc are tangents or parallels to tangents. Lines with a relative angle of 90 degrees to an arc or line are orthogonal lines.

Usage

1. Choose the desired line type in the options toolbar.
2. Enter the relative angle in the options tool bar.
3. Click the existing base entity.
4. Place the line with the mouse or enter a coordinate in the command line.

Orthogonal

Toolbar / Icon:


Menu: `Draw > Line > Orthogonal`
Shortcut: `L, O`
Commands: `lineorthogonal | lo`

Description

This tool lets you create lines orthogonal to an existing base entity. The base entity can be a line, arc or circle.

Usage

1. Choose the desired line type in the options toolbar.
2. Click the existing base entity.
3. Place the line with the mouse or enter a coordinate in the command line.

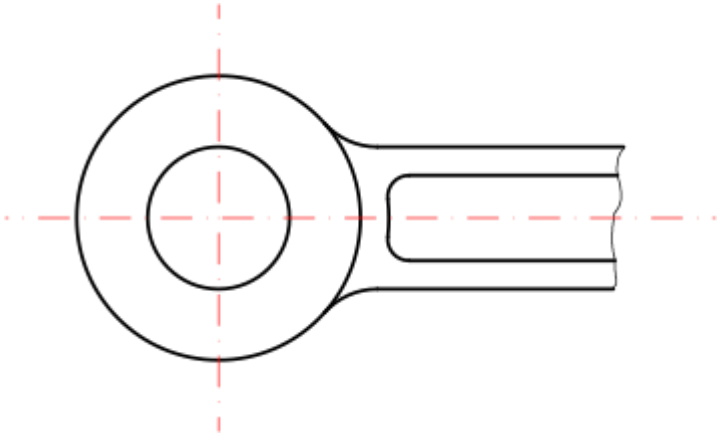
Freehand Line

Toolbar / Icon:

Menu: `Draw > Line > Freehand Line`
Shortcut: `L, F`
Commands: `linefree | freehand | lf`

Description

With this tool you can draw freehand lines. This tool is usually used only sparingly since technical drawings require absolute accuracy. However, on some occasions this tool might come in handy, for example for broken edges like shown here:



Usage

1. Place the mouse at the start point of the freehand line, press the left mouse button and keep it down.
2. Now move the mouse around to draw the line. At the endpoint of the freehand line, let go of the mouse button.

Arc Tools

Toolbar / Icon:

Shortcut: `W, A`
Commands: `arcmenu`

Center, Point, Angles

Toolbar / Icon:


Menu: `Draw > Arc > Center, Point, Angles`
Shortcut: `A, R`
Commands: `arcc | ar`

Description

Draws an arc from its center point, a point on the arc line, start angle and end angle.

Usage

1. Choose the direction of the arc in the options toolbar.
2. Set the center of the arc using the mouse or enter a coordinate in the command line.
3. Define the radius by clicking a point on the arc or by entering a coordinate into the command line. You may also enter the radius in the command line.
4. Set the start angle with the mouse or by entering a coordinate or the angle in the command line.
5. Set the end angle the same way as the start angle.

2 Points and Radius

Toolbar / Icon:
 

Menu: `Draw > Arc > 2 Points and Radius`
Shortcut: `A, D`
Commands: `arcradius | ad`

Description

Draws an arc using the start point, end point and the radius.

Usage

1. Type the arc radius into the options tool bar and choose the arc direction (clockwise or counter-clockwise) and the solution (larger arc or smaller arc).
2. Specify the start point of the arc.
3. Specify the end point of the arc. If the end point is too far away from the start point for a solution, the closest solution is drawn (a half circle with the given radius and direction).

2 Points and Angle

Toolbar / Icon:
 

Menu: `Draw > Arc > 2 Points and Angle`
Shortcut: `A, 2`
Commands: `arc2 | a2`

Description

Draws an arc using the start point, end point and the included angle.

Usage

1. Type the included angle into the options tool bar and choose the arc direction (clockwise or counter-clockwise).

2. Specify the start point of the arc.
3. Specify the end point of the arc.

2 Points and Length

Toolbar / Icon:
 

Menu: `Draw > Arc > 2 Points and Length`
Shortcut: `A, L`
Commands: `arclength | al`

Description

Draws an arc using the start point, end point and the arc length.

Usage

1. Type the arc length into the options tool bar and choose the arc direction (clockwise or counter-clockwise).
2. Specify the start point of the arc.
3. Specify the end point of the arc. If the end point is too far away from the start point for a solution, nothing is drawn.

2 Points and Height

Toolbar / Icon:
 

Menu: `Draw > Arc > 2 Points and Height`
Shortcut: `A, H`
Commands: `archeight | ah`

Description

Draws an arc using the start point, end point and the arc height.

Usage

1. Type the arc height into the options tool bar and choose the arc direction (clockwise or counter-clockwise).
2. Specify the start point of the arc.
3. Specify the end point of the arc.

3 Points

Toolbar / Icon:
 

Menu: `Draw > Arc > 3 Points`
Shortcut: `A, 3`
Commands: `arc3 | a3`

Description

If you know the start point, the end point and a point in between on the arc line, you can use this tool to draw an arc.

Usage

1. Set the start point using the mouse or enter a coordinate in the command line.
2. Set the second point to a known point on the arc line.
3. Set the endpoint of the arc.

Concentric (with Distance)

Toolbar / Icon:
 

Menu: `Draw > Arc > Concentric (with Distance)`
Shortcut: `A, C`
Commands: `arcconcentric | ac`

Description

With this tool you can create one or multiple concentric arcs with a given distance to an existing arc.

Usage

1. Enter the distance of the concentric arc from the original base arc in the options tool bar.
2. Enter the number of concentric arcs to create in the options tool bar.
3. Click the base arc. The concentric arc(s) are created on that side on which the mouse cursor is located while clicking the base arc.

Concentric (through Point)

Toolbar / Icon:
 

Menu: `Draw > Arc > Concentric (through Point)`
Shortcut: `A, G`
Commands: `arcconcentricthrough | ag`

Description

With this tool you can create concentric arcs that go through a specified point.

Usage

1. Enter the number of concentric arcs to create in the options tool bar.
2. Click the base arc.

3. Click the location through which the first concentric arc should go.

Tangentially Connected

Toolbar / Icon:

Menu: `Draw > Arc > Tangentially Connected`
Shortcut: `A, N`
Commands: `arctangential | an`

Description

Draws an arc that connects tangentially to an existing arc or line.

Usage

1. Enter the radius of the arc in the options tool bar.
2. Specify the end point of the arc.

Tangent, Point, Radius

Toolbar / Icon:

Menu: `Draw > Arc > Tangent, Point, Radius`
Shortcut: `A, T`
Commands: `arctangentpointradius | at`

Description

Draws an arc with a given radius, tangential to an entity and through a point.

Usage

1. Enter the radius in the options tool bar.
2. Specify the tangential entity.
3. Specify the point on the arc.
4. Choose the desired solution by clicking close to it with the left mouse button.

Circle Tools

Toolbar / Icon:


Shortcut: `W, C`
Commands: `circlemenu`

Center, Point

Toolbar / Icon:


Menu: `Draw > Circle > Center, Point`
Shortcut: `C, I`
Commands: `circle | ci`

Description

Draws a circle with a given center and a point on the circle line.

Usage

1. Set the center of the circle using the mouse or enter a coordinate in the command line.
2. Define the radius by clicking a point on the circle line or by entering a coordinate into the command line. You may also enter the radius in the command line.

Center, Radius

Toolbar / Icon:


Menu: `Draw > Circle > Center, Radius`
Shortcut: `C, R`
Commands: `circlecr | cr`

Description

This tool lets you create circles with a given center and radius.

Usage

1. Enter the radius in the option tool bar.
2. Set the center of the circle using the mouse or enter a coordinate in the command line.

Center, Diameter

Toolbar / Icon:


Menu: `Draw > Circle > Center, Diameter`
Shortcut: `C, A`
Commands: `circlediameter | ca`

Description

This tool lets you create circles with a given center and diameter.

Usage

1. Enter the diameter in the option tool bar.
2. Set the center of the circle using the mouse or enter a coordinate in the command line.

2 Points and Radius

Toolbar / Icon:
 

Menu: `Draw > Circle > 2 Points and Radius`
Shortcut: `C, D`
Commands: `circleradius | cd`

Description

Draws a circle from two points on the circle line and a radius.

Usage

1. Type the radius into the options tool bar.
2. Specify the first point on the circle line.
3. Specify the second point on the circle line.

2 Points

Toolbar / Icon:
 

Menu: `Draw > Circle > 2 Points`
Shortcut: `C, 2`
Commands: `circle2p | c2`

Description

Draws a circle with two diametrically opposed points.

Usage

1. Set the first point using the mouse or enter a coordinate in the command line.
2. Set the second point.

3 Points

Toolbar / Icon:
 

Menu: `Draw > Circle > 3 Points`
Shortcut: `C, 3`
Commands: `Circle3P | c3`

Description

Draws a circle from three known points on the circle line.

Usage

1. Set the first point on the circle line using the mouse or enter a coordinate in the command line.
2. Set the second point on the circle line.
3. Set the third point.

Concentric (with Distance)

Toolbar / Icon:
 

Menu: `Draw > Circle > Concentric (with Distance)`
Shortcut: `C, C`
Commands: `circleconcentric | cc`

Description

With this tool you can create one or multiple concentric circles with a given distance to an existing circle.

Usage

1. Enter the distance of the concentric circle from the original base circle in the options toolbar.
2. Enter the number of concentric circles to create in the options toolbar.
3. Click the base circle. The concentric circle(s) are created on that side on which the mouse cursor is located while clicking the base circle.

Concentric (through Point)

Toolbar / Icon:


Menu: `Draw > Circle > Concentric (through Point)`
Shortcut: `C, G`
Commands: `circleconcentricthrough | cg`

Description

With this tool you can create concentric circles that go through a specified point.

Usage

1. Enter the number of concentric circles to create in the options toolbar.
2. Click the base circle.
3. Click the location through which the first concentric circle should go.

Tangent and 2 Points

Toolbar / Icon:


Menu: `Draw > Circle > Tangent and 2 Points`
Shortcut: `C, T, 1`
Commands: `circletangent2p | ct1`

Description

Draws a circle that is tangential to an entity and goes through two points.

Usage

1. Specify the tangential entity.
2. Specify the first point on the circle line.
3. Specify the second point on the circle line.

Tangent, Point, Radius

Toolbar / Icon:


Menu: `Draw > Circle > Tangent, Point, Radius`
Shortcut: `C, T, P`
Commands: `circletangentpointradius | ctp`

Description

Draws an arc with a given radius, tangential to an entity and through a point.

Usage

1. Enter the radius in the options tool bar.
2. Specify the tangential entity.
3. Specify the point on the arc.
4. Choose the desired solution by clicking close to it with the left mouse button.

2 Tangents and Point

Toolbar / Icon:

Menu: `Draw > Circle > 2 Tangents and Point`
Shortcut: `C, T, 2`
Commands: `circletangent2 | ct2`

Description

Draws a circle that is tangential to two entities and goes through a point.

Usage

1. Specify the first tangential entity.
2. Specify the second tangential entity.
3. Specify the point on the circle line.

2 Tangents and Radius

Toolbar / Icon:

Menu: `Draw > Circle > 2 Tangents and Radius`
Shortcut: `C, T, R`
Commands: `circletangent2radius | ctr`

Description

Draws a circle with a given radius that is tangential to two entities.

Usage

1. Type the radius into the options tool bar.
2. Specify the first tangential entity.
3. Specify the second tangential entity.

3 Tangents

Toolbar / Icon:
 

Menu: `Draw > Circle > 3 Tangents`
Shortcut: `C, T, 3`
Commands: `circletangent3 | ct3`

Description

Draws a circle that is tangential to three entities.

Usage

1. Specify the first tangential entity.
2. Specify the second tangential entity.
3. Specify the third tangential entity.

Ellipse Tools

Toolbar / Icon:


Shortcut: `W, E`
Commands: `ellipsemenu`

Ellipse (Center, Point, Ratio)

Toolbar / Icon:
 

Menu: `Draw > Ellipse > Ellipse (Center, Point, Ratio)`
Shortcut: `E, P`
Commands: `ellipse | ep`

Description

Draws ellipses with a given center, major axis and minor axis.

Usage

1. Set the center of the ellipse using the mouse or enter a coordinate in the command line.
2. Define the major axis by clicking the endpoint of the axis, which is a point on the ellipse. You can also enter a coordinate into the command line or enter an angle and major radius in the format @50<30 where 50 is the major radius and 30 is the ellipse angle.
3. Define the endpoint of the minor axis which is also a point on the ellipse.

Ellipse Arc

Toolbar / Icon:
 

Menu: `Draw > Ellipse > Ellipse Arc`
Shortcut: `E, A`
Commands: `ellipsearc | ea`

Description

Draws ellipse arcs with a given center, major and minor axis and start and end angles.

Usage

1. Set the center of the ellipse using the mouse or enter a coordinate in the command line.

2. Define the major axis by clicking the endpoint of the axis, which is a point on the ellipse. You can also enter a coordinate into the command line or enter an angle and major radius in the format @50<30 where 50 is the major radius and 30 is the ellipse angle.
3. Define the endpoint of the minor axis which is also a point on the ellipse or enter the length of the minor axis.
4. Set the start angle with the mouse or by entering a coordinate or the angle in the command line.
5. Set the end angle the same way as the start angle.

Ellipse with Radii

Toolbar / Icon:


Menu: `Draw > Ellipse > Ellipse with Radii`
Shortcut: `E, I`
Commands: `ellipseradii | ei`

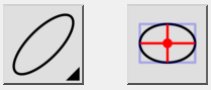
Description

Draws ellipses with given major and minor radius.

Usage

1. Enter the radius in X (major radius), the radius in Y (minor radius) and the ellipse angle in the options toolbar.
2. Set the center of the ellipse using the mouse or enter a coordinate in the command line.

Ellipse with Diameters

Toolbar / Icon:


Menu: `Draw > Ellipse > Ellipse with Diameters`
Shortcut: `E, D`
Commands: `ellipsediameters | ed`

Description

Draws ellipses with given major and minor diameter (width / height).

Usage

1. Enter the diameter in X (major diameter), the diameter in Y (minor diameter) and the ellipse angle in the options toolbar.
2. Set the center of the ellipse using the mouse or enter a coordinate in the command line.

Parallel Curve (with Distance)

Toolbar / Icon:


Menu: `Draw > Ellipse > Parallel Curve (with Distance)`
Shortcut: `E, C`
Commands: `ellipseoffset | ec`

Description

With this tool you can create one or multiple parallel curves with a given distance to an existing ellipse.

Usage

1. Enter the distance of the parallel curves from the original base ellipse in the options toolbar.
2. Enter the number of parallel curves to create in the options toolbar.

3. Click the base ellipse. The concentric curve(s) are created on that side on which the mouse cursor is located while clicking the base ellipse.

Parallel Curve (through Point)

Toolbar / Icon:


Menu: `Draw > Ellipse > Parallel Curve (through Point)`
Shortcut: `E, G`
Commands: `ellipseoffsetthrough | eg`

Description

With this tool you can create curves that are parallel to an ellipse and go through a specified point.

Usage

1. Enter the number of parallel curves to create in the options toolbar.
2. Click the base ellipse.
3. Click the location through which the first parallel curve should go.

Inscribed in Quadrilateral

Toolbar / Icon:


Add-on: QCAD Professional
Menu: `Draw > Ellipse > Inscribed in Quadrilateral`
Shortcut: `E, Q`
Commands: `ellipseinscribedquad | eq`

Description

Draws an ellipse that is inscribed in a quadrilateral.

Usage

1. Specify the first line of the quadrilateral.
2. Specify the second line of the quadrilateral.
3. Specify the third line of the quadrilateral.
4. Specify the fourth line of the quadrilateral.

Spline Tools

Toolbar / Icon:


Shortcut: `W, N`
Commands: `splinemenu`

Spline (Control Points)

Toolbar / Icon:

Menu: `Draw > Spline > Spline (Control Points)`
Shortcut: `S, P`
Commands: `spline | sp`

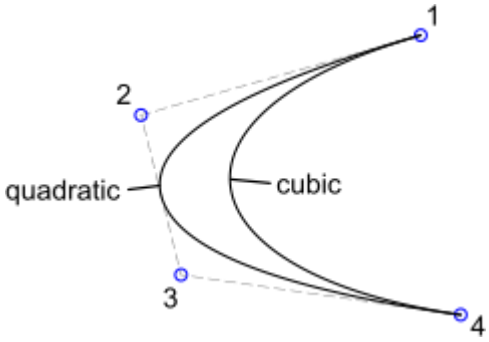
Description

Draws spline curves from control points. Non-uniform rational B-Splines (NURBS) with homogeneous weighting factors are the only splines that are supported.

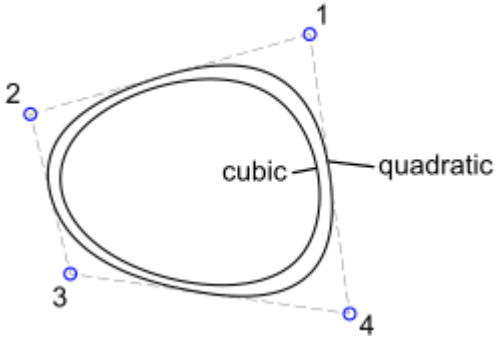
Usage

1. Choose the degree of the spline in the options tool bar. Supported degrees are 2 (quadratic b-spline) and 3 (cubic b-spline). The higher the degree, the 'smoother' the curve becomes.
   Note that a quadratic b-spline requires by definition at least 3 control points while a cubic b-spline has at least 4 control points.
2. Check the 'Closed' check box in the options tool bar if you want to create a closed spline. Closed splines are continuous closed loops.
3. Specify the control points. After defining the third (for quadratic b-splines) or fourth (for cubic b-splines) control point, a preview will be shown. You can remove the last control point again by clicking the 'Undo' button in the options tool bar.
4. Hit escape or click the right mouse button after setting the last point to confirm the spline shape.
5. You can now create another spline or right-click again to terminate the tool.

Example for two open splines (quadratic and cubic):



Example for two closed, periodic splines (quadratic and cubic):



Spline (Fit Points)

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Spline > Spline (Fit Points)`
Shortcut: `S, L`
Commands: `splinefit | sl`

Description

Draws cubic spline curves from fit points (points on the spline curve).

Usage

1. Check the "Closed" check box in the options tool bar if you want to create a closed spline. Closed splines are continuous closed loops.
2. Specify the fit points. You can remove the last fit point by clicking the "Undo" button in the options tool bar.
3. Hit escape or click the right mouse button after setting the last point to confirm the spline shape.
4. You can not create another spline or right-click again to terminate the tool.

Insert Fit Point

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Spline > Insert Fit Point`
Shortcut: `N, I`
Commands: `splineinsertfit | ni`

Description

With this tool you can insert fit points into existing splines.

Usage

- Click the position on the spline at which a new fit point should be inserted. The fit point is inserted at the point on the spline which is closest to the point that is clicked. If the spline is selected, the new fit points are visualized.

Append Fit Point(s)

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Spline > Append Fit Point(s)`
Shortcut: `N, A`
Commands: `splineappend | na`

Description

With this tool you can append fit points to existing splines.

Usage

- Click a position near the start of a spline to append fit points at the beginning. Click a position near the end of a spline to append fit points at the end.
- Click the positions of the new fit points.

Remove Fit Point

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Spline > Remove Fit Point`
Shortcut: `N, R`
Commands: `splineremovefit | nr`

Description

With this tool you can remove fit points from splines.

Usage

- Click the fit point of a spline to remove it.

Simplify

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Spline > Simplify`
Shortcut: `N, P`
Commands: `splinesimplify | np`

Polyline Tools

Toolbar / Icon:

Shortcut: `W, O`
Commands: `polylinemenu`

Draw Polyline

Toolbar / Icon:


Menu: `Draw > Polyline > Draw Polyline`
Shortcut: `P, L`
Commands: `polyline | pl`

Description

This tool lets you draw polylines. A polyline is a continuous line formed from one or more connected line or arc segments.
Please note that it is almost always more convenient to use the regular line and arc tools to draw a new polyline. The loosely connected line and arc segments can then be converted into a polyline entity using the "Create Polyline from Segments" tool.

Usage

1. Specify the start point of the first polyline segment. You can use the mouse or enter a coordinate in the console.
2. Specify the endpoint of the first polyline segment.
3. Specify the endpoints of additional polyline segments.

If the polyline contains arc segments, you can tick the "Arc" check box and enter the arc radius in the options tool bar when setting the endpoint of an arc segment. The arc will connect smoothly (tangential) to the last segment.

Polyline from Segments

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Polyline from Segments`
Shortcut: `O, G`
Commands: `polylinefromsegments | og`

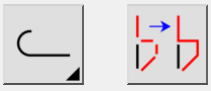
Description

This tool lets you create polylines from existing segments (lines and arcs that are connected). Connected splines are automatically converted to polylines with arc segments.

Usage

Click one of the segments. A polyline is created from the selected entity and all line or arc entities that are connected to its end- or start point.
The polyline is created on the same layer with the same attributes as the entity that is clicked. If that layer is locked, the polyline is created on the current layer instead.

Polyline from Selection

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Polyline from Selection`
Shortcut: `O, C`
Commands: `polylinefromselection | oc`

Description

This tool lets you create polylines from selected segments (connected lines, arcs and polylines).

Usage

1. Select the entities you want to join into polylines.
2. Launch this tool.
3. All connected entities are joined into a polyline. If not all selected entities are connected, multiple polylines are created.
4. The number of polylines created is printed in the command line history.

Partial Polyline Along Selection

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Draw > Polyline > Partial Polyline Along Selection`

Shortcut: `O, L`

Commands: `polylinealongselection | ol`

Description

This tool lets you create polylines along a partial length along a selected path (connected lines, arcs and polylines).

Usage

1. Select the entities along which you want to create a polyline.
2. Launch this tool.
3. Click the start point of the polyline. To start point can be at any position along the selected entities.
4. Click the end point of the polyline.
5. The polyline is created between the chosen points along the selected entities. Disconnected entities which are not part of the new polyline are deselected.

Insert Node

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Insert Node`
Shortcuts: `O, A | O, I`
Commands: `polylineinsert | polylineadd | oa | oi`

Description

With this tool you can add nodes to existing polylines. Adding a new node to a polyline means to split up one segment into two segments. The two new segments are both straight, even if one of original segments was an arc segment.

Usage

1. Specify the segment of the polyline in which you want to add the new node.
2. Specify the position of the new node with your mouse or by entering a coordinate.

Append Node(s)

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Append Node(s)`
Shortcut: `O, P`
Commands: `polylineappend | op`

Description

With this tool you can append nodes to the beginning or end of existing polylines. Appending a node to a polyline means to extend it by one segment at the beginning or end.

Usage

1. Click the polyline somewhere near the end to which you want to append a node.
2. Specify the position of the new node(s) with your mouse or by entering a coordinate.

Delete Node(s)

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Delete Node(s)`
Shortcut: `O, D`
Commands: `polylinedel | od`

Description

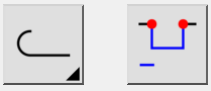
Use this tool to remove nodes from existing polylines. This way two segments of a polyline can be joined into one.

Usage

1. Specify the polyline from which you want to delete nodes.
2. Specify the position of the node that you want to remove.

Remove Nodes between Two Nodes

Toolbar / Icon:



Add-on: QCAD Professional
Menu: `Draw > Polyline > Remove Nodes between Two Nodes`
Shortcut: `O, B`
Commands: `polylinedelbetween | ob`

Description

Use this function to delete all nodes between two nodes of an existing polyline.

Usage

1. Specify the polyline from which you want to delete segments.
2. Specify the position of the first limiting node. That is the node from where you want to start deleting segments. The node itself will not be deleted. In most cases it is appropriate to use the endpoint snap function to specify the node.
3. Specify the position of the node where you want to stop deleting. All segments that are between the first node and the second node are removed from the polyline. After the segments are deleted, the two limiting nodes are connected by a straight line segment so the polyline is still one continuous sequence of line segments.

Trim Segments

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Draw > Polyline > Trim Segments`

Shortcut: `O, T`

Commands: `polylinetrim | ot`

Description

Use this function to trim two polyline segments. This means to extend or shorten the two segments, in a way that they meet at one common point.

Usage

1. Specify the first polyline segment to trim by clicking it with the mouse.
2. Specify the second segment to be trimmed from the same polyline. All segments that are between the first segment and the second segment are removed to keep the polyline a continuous line string.

Relocate Start Point

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Relocate Start Point`
Shortcut: `O, R`
Commands: `polylinestart | or`

Description

Use this tool to choose a new start point for a closed polyline.

Usage

1. Click the closed polyline of which the start point should be relocated.
2. Click the new position of the start point anywhere along the polyline.

Change Segment Type

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Change Segment Type`
Shortcut: `O, X`
Commands: `polylinechange | ox`

Description

With this tool you can change the type of a polyline segment from an arc to a line and vice versa.

Usage

- Click an existing polyline segment. An arc segment is immediately converted into a line segment. For a line segment, the arc can be defined by clicking a point on the arc.

Arc to Line Segments

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Polyline > Arc to Line Segments`
Commands: `polylinearctoline`

Description

Use this tool to convert all arc segments in polylines into straight line segments.

Usage

1. Select the polyline(s) from which you want to convert all segments into line segments.
2. Start this tool.

Offset

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Draw > Polyline > Offset`
Shortcut: `O, Q`
Commands: `polylineequidistant | polylineoffset | oq`

Description

This tool lets you create equidistant polylines to existing polylines.

Usage

1. Enter the distance of the equidistant and the number of equidistant polylines you want to create at once in the options tool bar.
2. Click the original polyline. The equidistant will be created on the side on which you click next to the polyline.

Morph

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Draw > Polyline > Morph`

Shortcut: `O, M`

Commands: `polylinemorph | om`

Description

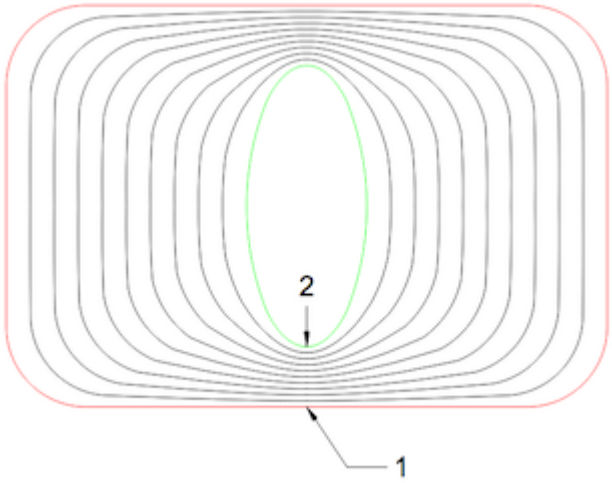
With this tool, you can morph between two polylines. A number of new intermediate polylines are created. Each intermediate polyline is a mix of the two chosen polylines. With each step of the morphing operation, the position and shape of the first polyline is transformed to be more like the second polyline.

Usage

1. Prepare the two polylines.
   Usually the two polylines should have the same direction or orientation. For closed polylines, you would typically want the start point to be aligned, so that no rotation occurs while morphing. More complex polylines can be split at sharp corners to maintain the position of those corners throughout the morphing process.
2. Launch this tool.
3. Enter the number of desired steps in the options toolbar.
4. Choose the desired easing function in the options toolbar. The normal easing function is "linear" which interpolates linearly between the two polylines. Easing functions can be used to create various uneven types of interpolations.
   For example to start morphing at the first polyline, first advancing slowly toward the second polyline and then accelerate toward the second polyline in a quadratic function, choose "in quadratic".
5. Click the first polyline. You can also select a line, an arc, a circle, an ellipse or a spline.

6. Click the second polyline.
7. The entered number of intermediate polylines is created.

Example:
In this example, the rounded rectangle is morphed to the ellipse in nine steps. Both shapes are first converted into polylines (the ellipse is exploded into a polyline with arc segments). The startpoint of the rounded rectangle is set to be at the bottom center (1). The start point of the ellipse is set at its bottom center (2). Both polylines are made counter-clockwise.



Simplify

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Simplify`
Shortcut: `O, S`
Commands: `polylinesimplify | os`

Description

Simplifies all selected polylines by joining consecutive segments that are in line.

Usage

1. Select the polyline(s) you want to simplify.
2. Start this tool.

3. Enter the desired tolerance in the Options toolbar. The tolerance is the maximum distance between the original polyline and the simplified polyline.

Normalize Polylines

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Normalize Polylines`
Shortcut: `O, N`
Commands: `polylinenormalize | on`

Description

Use this tool to automatically remove duplicate nodes from existing polylines.

Usage

1. Select the polyline(s) from which you want to delete duplicate nodes.
2. Start this tool.

Logically close Polylines

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Logically close Polylines`
Shortcut: `O, E`
Commands: `polylinetoclosed | oe`

Description

Polylines can be geometrically closed or logically closed. A geometrically closed polyline is essentially an open polyline for which the last node just happens to be at the same position as the first node. In a logically closed polyline with the same shape, the last node is omitted and the information that the polyline is closed is stored in the polyline instead. Logically closed polylines require one node less than geometrically closed ones. Logically closed polylines are often better suitable for further processing (CAM) than geometrically closed ones.

This tool converts all geometrically closed polylines into logically closed polylines.

Logically open Polylines

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Polyline > Logically open Polylines`
Shortcut: `O, J`
Commands: `polylinetoopen | oj`

Description

This tool converts all logically closed polylines into logically open (but geometrically closed) polylines.

See also Draw > Polyline > Logically close Polylines

Shape Tools

Toolbar / Icon:

Shortcut: `W, H`
Commands: `shapemenu`

Rectangle

Toolbar / Icon:
 

Menu: `Draw > Shape > Rectangle`
Shortcut: `R, E`
Commands: `rectangle | linerectangle | rect | re`

Description

Use this tool to create rectangular shapes from two diagonally opposite corners.

Usage

1. Specify the first corner of the rectangle.
2. Move the mouse to the second corner and click to specify the second corner of the rectangle. Alternatively you can enter the coordinate of the second corner in the command prompt. E.g. to create a rectangle with width 50 and height 25, enter the relative coordinate of the second corner as follows:
   @50,25

Rectangle with Size

Toolbar / Icon:
 

Menu: `Draw > Shape > Rectangle with Size`
Shortcut: `R, S`
Commands: `rectanglesize | linerectanglesize | rs`

Description

Use this tool to create rectangular shapes with a given size.

Usage

1. Enter the size of the rectangle in the options tool bar.
2. Choose the reference point you want to use to position the rectangle.
3. Specify the position of the rectangle with the mouse or by entering a coordinate in the command line.

Rectangle (3 Points)

Toolbar / Icon:
 

Add-on: QCAD Professional

Menu: `Draw > Shape > Rectangle (3 Points)`

Shortcut: `R, 3`

Commands: `rectangle3p | linerectangle3p | rect3p | r3`

Description

Use this tool to create rectangular shapes from two corners of the first edge and a point on the opposite side.

Usage

1. Specify the first corner of the rectangle.
2. Move the mouse to the second corner and click to specify the second corner of the first edge of the rectangle. This defines the length and rotation angle of the rectangle.
3. Click a third point that is on the opposite edge. This defines the width of the rectangle.

Polygon (Center, Corner)

Toolbar / Icon:

Menu: `Draw > Shape > Polygon (Center, Corner)`
Shortcuts: `P, G, 1 | H, C`
Commands: `linepolygon | polygon | pg1`

Description

Creates polygons with the center and one corner given.

Usage

1. Enter the degree of the polygon (number of edges / corners) in the options tool bar. Valid numbers range from 3 to 99.
2. Set the center of the polygon using the mouse or enter a coordinate in the command line.
3. Set the first corner of the polygon.

Polygon (2 Points of Side)

Toolbar / Icon:

Menu: `Draw > Shape > Polygon (2 Points of Side)`
Shortcuts: `P, G, 2 | H, P`
Commands: `linepolygon2 | polygon2 | pg2`

Description

Creates polygons with two corners given.

Usage

1. Enter the degree of the polygon (number of edges / corners) in the options tool bar. Valid numbers range from 3 to 99.

2. Set the first corner of the polygon using the mouse or enter a coordinate in the command line.
3. Set the second corner of the polygon.

Polygon (Center, Side)

Toolbar / Icon:


Menu: `Draw > Shape > Polygon (Center, Side)`
Shortcut: `P, G, 3`
Commands: `polygoncs | pg3`

Description

Creates polygons with the center and middle point of one side.

Usage

1. Enter the degree of the polygon (number of edges / corners) in the options tool bar. Valid numbers range from 3 to 99.
2. Set the center of the polygon using the mouse or enter a coordinate in the command line.
3. Set the middle point of one side of the polygon.

Polygon (Side, Side)

Toolbar / Icon:


Menu: `Draw > Shape > Polygon (Side, Side)`
Shortcut: `P, G, 4`
Commands: `polygonss | pg4`

Description

Creates polygons from two opposite corners or sides.

Usage

1. Enter the degree of the polygon (number of edges / corners) in the options tool bar. Valid numbers range from 3 to 99.
2. Choose the option Corner to Corner to define the polygon with two opposite corners.
3. Set the first corner or middle of a side of the polygon using the mouse or enter a coordinate in the command line.
4. Set the second, opposite corner or middle of a side of the polygon.

Star

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Shape > Star`
Shortcut: `P, T`
Commands: `star | pt`

Description

Creates stars from a given outer and inner corner.

Usage

1. Enter the number of outermost corners of the star shape in the options tool bar. Valid numbers range from 3 to 99.
2. Tick the option "symmetrical" to create a regular, symmetrical star. Without this option, the inner corner can be placed freely.
3. Set the first outer corner of the star using the mouse or enter a coordinate in the command line.
4. Set the inner corner of the star.

Hatch Tools

Toolbar / Icon:

Shortcut: `W, X`
Commands: `hatchmenu`

Hatch from Selection

Toolbar / Icon:

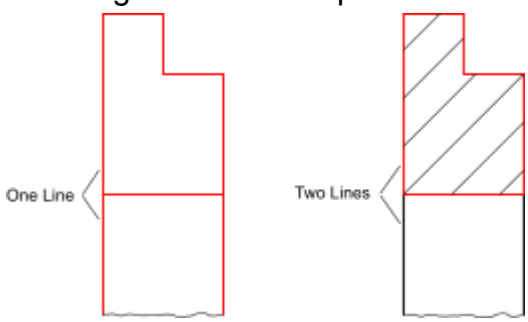
Menu: `Draw > Hatch > Hatch from Selection`
Shortcut: `H, A`
Commands: `hatch | ha`

Description

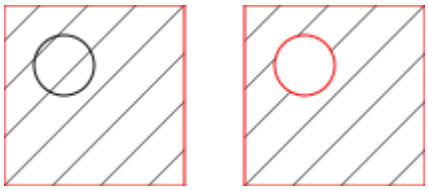
This tool fills an area surrounded by existing entities with a hatch pattern or a solid color.

Usage

1. Prepare the entities that surround the hatching area so they form a closed contour. The contour must be closed in a way that one entity is connected to the next one as shown at the right in this example:



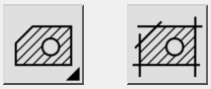
2. Select the contour(s) you want to fill. Note that islands inside contours will get hatched if they are not selected:



3. Launch the hatch tool.
4. A dialog for the hatch options is displayed. Choose a hatch pattern, scale factor and a rotation angle for the hatch pattern. If you want to fill the object with a solid color instead of a pattern, check the check box "Solid Fill".
5. Click 'OK' to proceed with the hatching. Depending on the complexity of the contour and the scale factor of the chosen pattern, it might take a while to create the hatch.

Hatch from Segments

Toolbar / Icon:



Add-on: QCAD Professional

Menu: `Draw > Hatch > Hatch from Segments`

Shortcut: `H, S`

Commands: `hatchsegments | hs`

Description

This tool fills an area with a hatch pattern or a solid color. The area is defined by clicking existing entities or untrimmed entity segments which are part of the hatch boundary.

Usage

1. Make sure that the area to be filled is enclosed on all sides with entities (e.g. lines, arcs, splines, etc.). The entities do not have to be trimmed to corners for this tool. You can use partial entities or segments of entities as part of the boundary.
2. Start this tool.
3. Click OK.
4. A dialog for the hatch options is displayed. Choose a hatch pattern, scale factor and a rotation angle for the hatch pattern. If you want to fill the object with a solid color instead of a pattern, check the check box "Solid Fill".
5. Click all entity segments that are part of the hatch or fill boundary. The selected segments are highlighted. You can also remove a segment from the boundary by

clicking it again. The boundary segments do not have to be clicked in any particular order and you can also define islands inside an area.

Viewport Tools

Toolbar / Icon:


Shortcut: `W, T`
Commands: `viewportmenu`

Add Viewport

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Viewport > Add Viewport`
Shortcut: `V, T`
Commands: `viewport | vt`

Description

Creates a new viewport in a layout block. A viewport is a rectangular area that displays a part of the drawing at a given scale and angle. Viewports can only exist in layout blocks, special blocks dedicated to printing the drawing or parts of it, often along with a border and header.

Usage

1. After starting this tool, the model space (main drawing) is shown to define the rectangular area of the drawing the viewport should display.
2. Adjust the rotation angle of the viewport in the options toolbar to create a viewport that shows a part of the drawing rotated by a given angle. Note that rotated viewports are

not supported in other CAD systems.
3. Click the first corner of the viewport area.
4. Click the second corner of the viewport area.
5. QCAD now switches automatically to a layout block where the viewport can be created. If there are multiple layout blocks and this tool was not started from a layout block, a dialog is shown to let you choose in which layout block you want to place the viewport.
6. Adjust the viewport scale in the options toolbar if necessary. If the drawing is larger or much smaller than the available paper size, you will likely want to scale it up or down to fit on the paper.
7. Click the center position of the viewport.

The viewport is now added to the layout. To change properties such as scale or rotation angle of the viewport, you can click it to select it and adjust properties using the property editor.

The rectangular border of the viewport is shown if the layer the viewport is on is shown. If the layer the viewport is on is switched off, the border is hidden but the contents of the viewport is still shown. Viewports are typically placed on a dedicated layer (e.g. "viewports") which can be switched off for printing.

Ring

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Ring`
Shortcut: `R, I`
Commands: `ring | donut | ri`

Description

This tool lets you create rings (donuts) with a given center and an inner and outer diameter.

Usage

1. Enter the inner and outer diameters in the option tool bar.
2. Set the center of the ring using the mouse or enter a coordinate in the command line.

Text

Toolbar / Icon:


Menu: `Draw > Text`
Shortcut: `T, E`
Commands: `text | te`

Description

Use this tool to create single line or multi line text entities.

Usage

1. After launching the tool, the text dialog is presented in which you can choose the font and enter the text you want to create.
2. Choose the font name, text height and initial text attributes in the upper left area of the dialog.
   CAD fonts that are made up of lines and arcs are listed at the top of the font list, TrueType fonts are listed below.
   In the "Height" field, enter the height of the text in the unit of your drawing. To choose normal line spacing, enter a line spacing factor or "1". You can increase or decrease the line spacing by entering an alternative value. The value entered is a factor relative to the default (1.0).
3. Enter the rotation angle for the text. Normal horizontal texts have an angle of 0 degrees and texts that are readable from the right 90 degrees.
4. Choose the alignment of the text.
5. Check the "Simple Text" check box to create a single line text without any advanced formatting.
6. Enter the text you want to create in the large text box.
   You can use the various tools above the box to further format the text that you have entered or to paste texts from the clipboard. Use the button at the right to quickly insert symbols or to choose a Unicode character ("Others...").
   Note that the font you have chosen might not contain the characters you insert. In that case the characters will not be displayed in the drawing.
7. Click "OK" to exit the text dialog.
8. Use the mouse to specify the location of the text or enter a coordinate in the command line.

9. Often, users need to create a number of similar texts. For that reason the text tool does not terminate after creating the first text entity. You can change the text and the text angle in the options tool bar and create other texts with similar settings.
10. Right click or press Escape to stop the text tool.

Insert Bitmap

Toolbar / Icon:

Menu: `Draw > Insert Bitmap`
Shortcut: `I, M`
Commands: `insertbitmap | bitmap | im`

Description

Inserts a raster image (bitmap) into the drawing.
Note that the drawing file which is created when saving the drawing, contains only references to inserted images. It is recommended to keep the image file and the drawing file in the same folder, so QCAD can find the image again when the drawing file is loaded later.
Transparent backgrounds are supported for PNG images.
Note that large bitmaps can cause the drawing display to become very slow.
Usually you will want images to be in the background of other entities. Please refer to the 'Modify' - 'Send to back' tool to learn how to do this.

Usage

1. Choose the image file which you want to insert.
2. Enter the desired height or width for the image in drawing units in the options tool bar.
3. Type a rotation angle in the options tool bar.
4. Specify the insertion point of the image. That is the left bottom corner of the image.

Trace Bitmap

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Draw > Trace Bitmap`
Commands: `tracebitmap | trace`

Description

This tool traces an inserted image with polylines. This can be used as a means to vectorize in image, for example to quickly trace the outlines of an inserted image from a silhouette.

Usage

1. Insert the image you want to trace into your drawing if you have not done so already.
2. Select the image and start this tool.
3. Choose the desired options in the dialog:
4. **Monochrome:** Enabling this option converts the image to monochrome on the fly before tracing. This can yield better results for silhouette type images that are not completely monochrome already. You can also use this option if you are only interested in the outline of an image with a transparent or white background.
5. **Alpha Threshold:** The maximum alpha (transparency) value of a pixel to be considered white when converting to monochrome. Valid values range from 0 (everything not completely transparent is black) to 255 (everything is white).
6. **Brightness Threshold:** The maximum brightness of a pixel to be considered black. Valid values range from 0 (everything is black) to 255 (everything is white).
7. **Number of colors:** Images are reduced in color before tracing. This number indicates the number of colors to be used for the palette. Lower numbers yield less details or less noise and higher numbers more details or noise.
8. You can click the Update button anytime to update a small preview of the image tracing.
9. Click OK to trace the image. The polylines are added on the current layer.

Dimension Tools

Toolbar / Icon:


Shortcut: `W, D`
Commands: `dimensionmenu`

Description

Dimensions are used to add measurements to a drawing.
Dimensions typically consist of a text label and some lines and arrows to indicate the precise location of the measurement. The text label may be accompanied by tolerance indications or symbols. The options tool bar that is shown while a dimension is being created supports you to define the dimension label contents and its tolerances.

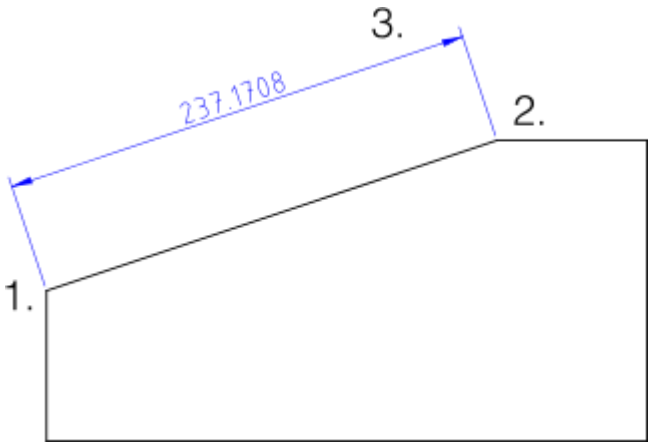
Aligned

Toolbar / Icon:


Menu: `Dimension > Aligned`
Shortcut: `D, A`
Commands: `dimaligned | da`

Description

Creates aligned dimensions. Aligned dimensions usually measure the length of an existing line. The dimension line is always parallel to the line between the two chosen points 1. and 2.:



Usage

1. Set the first extension line end point with the mouse or enter a coordinate in the command line.
2. Set the second extension line end point.
3. Set the position of the dimension line.

Rotated (Linear)

Toolbar / Icon:


Menu: `Dimension > Rotated (Linear)`
Shortcut: `D, L`
Commands: `dimlinear | dimrotated | dl`

Description

Creates rotated (linear) dimensions. Linear dimensions are usually used to measure vertical or horizontal distances but can also measure distances with any other angle.

Usage

1. Enter the angle of the dimension in the options toolbar.
2. Set the first extension line endpoint with the mouse or enter a coordinate in the command line.
3. Set the second extension line endpoint.
4. Set the position of the dimension line.

Horizontal

Toolbar / Icon:
 

Menu: `Dimension > Horizontal`
Shortcut: `D, H`
Commands: `dimhor | dimhorizontal | dh`

Description

This tool is provided for convenience and behaves essentially like the tool for rotated dimensions. The only difference is that the angle is fixed to 0 degrees (horizontal).

Vertical

Toolbar / Icon:
 

Menu: `Dimension > Vertical`
Shortcut: `D, V`
Commands: `dimver | dimvertical | dv`

Description

This tool is provided for convenience and behaves essentially like the tool for rotated dimensions. The only difference is that the angle is fixed to 90 degrees (vertical).

Baseline

Toolbar / Icon:

Add-on: QCAD Professional

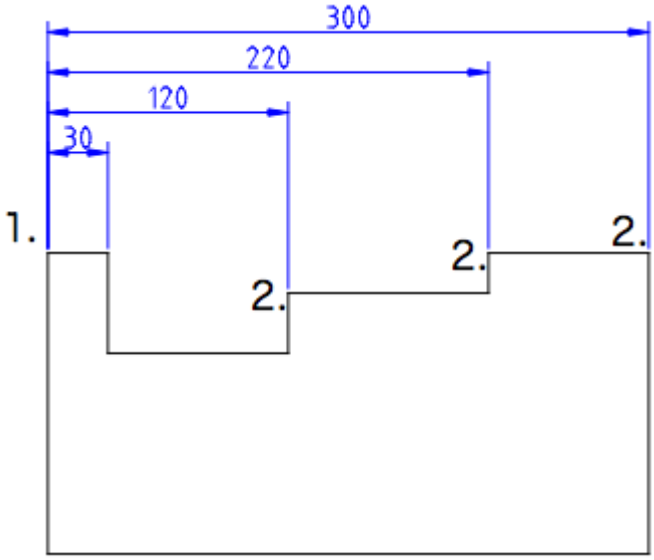
Menu: `Dimension > Baseline`

Shortcut: `D, B`

Commands: `dimbase | baselinedimension | db`

Description

Creates a rotated (linear), angular, or ordinate dimension, starting at the baseline of an existing dimension.



Usage

1. Click the base line of an existing linear, angular or ordinate dimension.
2. Set the second extension line end point with the mouse or enter a coordinate in the command line. The dimension lines are automatically stacked with a fixed distance between them from each other. You can adjust this distance (the dimension line increment) under *Edit* > *Drawing Preferences* > *Dimension* > *Dimension Settings* > *Dimension line increment*. You can also override this setting for individual dimensions in the options toolbar by unchecking *Default spacing* and entering a value for *Spacing*.

Continue

Toolbar / Icon:

Add-on: QCAD Professional

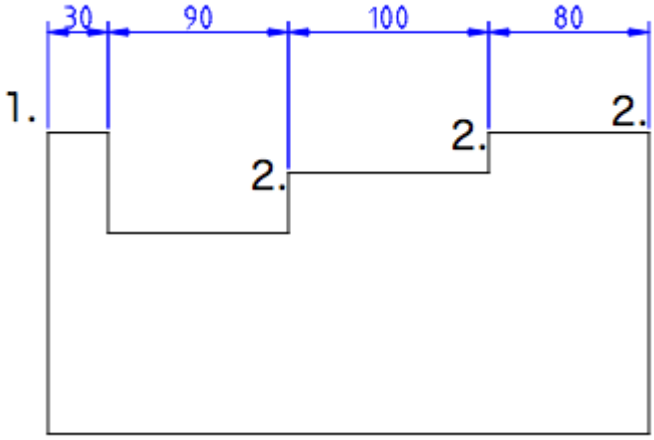
Menu: `Dimension > Continue`

Shortcut: `D, C`

Commands: `dimcontinue | continuedimension | dc`

Description

Creates a dimension that starts from an extension line of a selected linear, angular or ordinate dimension.



Usage

1. Click the base line of an existing linear, angular or ordinate dimension.
2. Set the second extension line end point with the mouse or enter a coordinate in the command line.
3. Continue to click more extension line end points if desired to create a chain of connected dimensions.

Ordinate (X/Y)

Toolbar / Icon:

Menu: `Dimension > Ordinate (X/Y)`
Shortcut: `D, O`
Commands: `dimordinate | do`

Description

This tool draws an ordinate dimension. Ordinate dimensions indicate the X-ordinate or Y-ordinate of a feature in the drawing, relative to an origin point.

Usage

1. Specify the location of the feature with the mouse or by entering a coordinate.
2. Specify the location of the text label and end point of the leader. If the leader is closer to being vertical, the X datum is indicated, if it is closer to being horizontal, the Y datum is indicated.
3. The origin of the ordinate dimension is at the absolute zero point. To move the origin by mouse, select the ordinate dimension entity and move the origin with the mouse (drag and drop). To specify a coordinate for the origin, use the property editor.

QCAD Professional:

- Click the mouse arrow symbol in the options tool bar to set the origin for new ordinate dimensions:

  You can then set the new origin location by clicking in the drawing or entering a coordinate in the command line. All new ordinate dimensions created after this measure the distance from that origin.

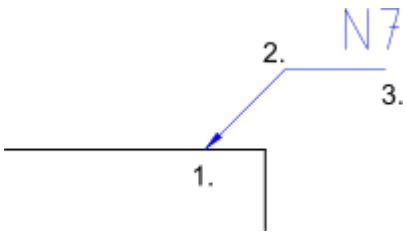
Leader

Toolbar / Icon:

Menu: `Dimension > Leader`
Shortcuts: `D, E | L, D`
Commands: `leader | dimlea | de | ld`

Description

Leaders are arrows that usually point from a text entity to another entity as shown below. In the example, the text entity "N7" is describing a surface property by pointing to it with a leader.



Usage

1. Set the location where the leader points to or enter a coordinate in the command line.
2. Set the location of the next edge of the leader line.
3. Set the other edges of the leader line and click the right mouse button or press the Escape key to finish.

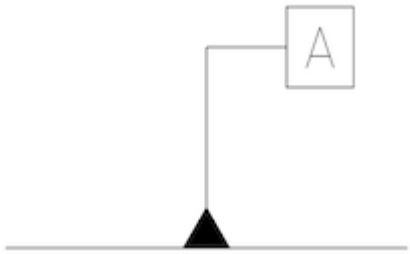
Datum

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Dimension > Datum`
Shortcut: `D, T`
Commands: `datum | dt`

Description

Creates a special type of leader typically used to attach a datum identifier symbol to an edge or surface in a mechanical drawing.



Usage

1. Click the start point of the datum leader. That's the point that indicates which feature the datum refers to.
2. Set the location of the next edge of the leader line.
3. Set the other edges of the leader line and click the right mouse button or press the Escape key to finish.

Tolerance

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Dimension > Tolerance`
Shortcut: `T, O`
Commands: `tolerance | to`

Description

Creates a feature control frame typically used for geometric tolerances or datum identifier symbols.

Usage

1. After starting this tool, the Geometric Tolerance dialog is shown.

2. Choose the desired tolerance symbol and fill in the tolerance values and datum reference letters as desired. To create a datum identifier symbol (for example [A]), only fill in the field "Datum Identifier".
3. Click OK.
4. Click the location for the feature control frame.
5. Click the right mouse button or press the Escape key to terminate the tool.

Radial

Toolbar / Icon:


Menu: `Dimension > Radial`
Shortcut: `D, R`
Commands: `dimrad | dimradial | dr`

Description

Creates radial dimensions for circle or arc entities.

Usage

1. Choose a circle or arc entity.
2. Set the position of the radial dimension line using the mouse or by entering a coordinate or an angle in the command line.

Diametric

Toolbar / Icon:

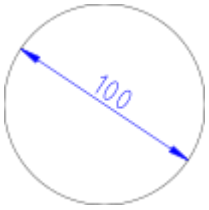
Menu: `Dimension > Diametric`

Shortcut: `D, D`

Commands: `dimdia | dimdiametric | dd`

Description

Creates diametric dimensions for circle or arc entities.



Usage

1. Choose a circle or arc entity.
2. Set the position of the diametric dimension line using the mouse or by entering a coordinate or an angle in the command line.

Angular

Toolbar / Icon:

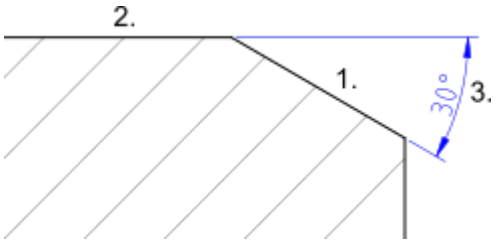
Menu: `Dimension > Angular`

Shortcut: `D, N`

Commands: `dimangular | dn`

Description

Creates angular dimensions between two reference lines as shown here:

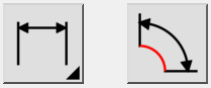


Usage

1. Choose the first line entity.
2. Choose the second line entity.
3. Set the position of the angular dimension arc using the mouse or by entering a coordinate in the command line.

Arc Length

Toolbar / Icon:



Add-on: QCAD Professional

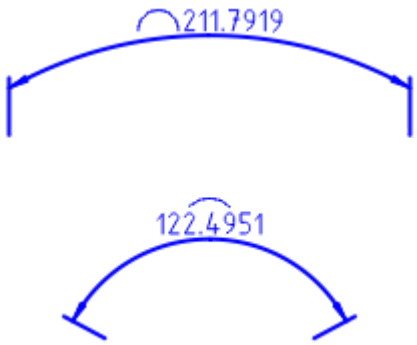
Menu: `Dimension > Arc Length`

Shortcut: `D, G`

Commands: `dimarclength | dg`

Description

Creates arc length dimensions as shown here:



Usage

1. Choose the arc entity.

2. Set the position of the dimension arc using the mouse or by entering a coordinate in the command line.

The position of the arc symbol can be adjusted in the property editor (property "Arc Symbol Type").

Reset Label Position

Toolbar / Icon:

Menu: `Dimension > Reset Label Position`
Shortcut: `D, S`
Commands: `dimregen | ds`

Description

This tool resets the label position of all selected dimension entities and places their label back to their automatically computed position.

Remove Style Overrides

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Dimension > Remove Style Overrides`
Shortcut: `D, X`
Commands: `dimremovestyle | dx`

Description

Removes all dimension style overrides from selected entities. These are dimension style properties that differ from the drawing default.

Modification Tools

Toolbar / Icon:

Shortcut: `W, M`
Commands: `modifymenu`

Move / Copy

Toolbar / Icon:


Menu: `Modify > Move / Copy`
Shortcut: `M, V`
Commands: `move | mv`

Description

Moves or copies entities.

Usage

1. Select the entities you want to move or copy.
2. Launch this tool.
3. Set the reference point with the mouse or enter a coordinate in the command line.
4. Set the target point. To move the selected entities by a given amount, enter a relative coordinate. For example to move it by 50 drawing units to the right, enter @50,0 in the command line.
5. The move dialog is displayed.
   To move the entities, choose "Delete Original", to copy them choose "Keep Original".
   You can also create any given number of copies at once by choosing "Multiple Copies" and entering the number of copies in the text line below. Note that "9" will create 9

copies and keep the original - so there will be 10 instances of the selected entities in the end.
The new entities are placed on the same layer as the originals and have the same attributes. To use the current layer and current attributes instead, tick "Use current layer and attributes".

6. Click "OK" to move or copy the entities.

Rotate

Toolbar / Icon:
 

Menu: `Modify > Rotate`
Shortcut: `R, O`
Commands: `rotate | ro`

Description

Rotates entities by a given angle around a given center.

Usage

1. Select the entities you want to rotate.
2. Launch this tool.
3. Set the center of the rotation with the mouse or enter a coordinate in the command line.
4. The rotate dialog is displayed where you can enter the number of copies and the rotation angle.
   If you want to specify the rotation angle with the mouse cursor, check the mouse cursor button that is displayed beside the angle input:
   
   To delete the original entities, check "Delete Original", to copy them choose "Keep Original". You can also create any given number of copies by choosing "Multiple Copies".
   The new entities are placed on the same layer as the originals and have the same attributes. To use the current layer and current attributes instead, tick "Use current layer and attributes".
5. Click "OK".

6. If you have previously chosen to specify the rotation angle by mouse, you now have to first specify a reference point for the rotation and then a target point. The rotation angle is the angle enclosed by the reference point, center of rotation and target point.

Scale

Toolbar / Icon:
 

Menu: `Modify > Scale`
Shortcut: `S, Z`
Commands: `scale | sz`

Description

Scales entities by a given factor towards a given center.

Usage

1. Select the entities you want to scale.
2. Launch this tool.
3. Set the center of the scaling with the mouse or enter a coordinate in the command line.
4. The scale dialog is displayed where you can enter the scale factor.
   If you want to scale with two different factors in X and Y direction, uncheck the button for proportional scaling:
   
   You can then enter two different scaling factors for X and Y.
   If you want to scale the selection by mouse, check the mouse cursor button:
   
5. Click "OK".
6. If you have previously chosen to scale the selection by mouse, you now have to specify a reference point and a target point for the scaling operation.

Mirror

Toolbar / Icon:
 

Menu: `Modify > Mirror`
Shortcut: `M, I`
Commands: `mirror | mi`

Description

Mirrors entities along a given axis.

Usage

1. Select the entities you want to mirror.
2. Launch this tool.
3. Specify a first point on the mirror axis with the mouse or enter a coordinate in the command line.
4. Specify a second point on the mirror axis.
5. The mirror dialog is displayed.
   To mirror the entities without keeping the original entities, choose "Delete Original", to copy them choose "Keep Original".
   The new entities are placed on the same layer as the originals and have the same attributes. To use the current layer and current attributes instead, tick "Use current layer and attributes".
6. Click "OK" to mirror the entities.

Flip Horizontal

Toolbar / Icon:
 

Menu: `Modify > Flip Horizontal`
Shortcut: `F, H`
Commands: `fliphorizontally | fh`

Description

This function flips (mirrors) the current selection horizontally.

Flip Vertical

Toolbar / Icon:
 

Menu: `Modify > Flip Vertical`
Shortcut: `F, V`
Commands: `flipvertically | fv`

Description

This function flips (mirrors) the current selection vertically.

Move and Rotate

Toolbar / Icon:
 

Menu: `Modify > Move and Rotate`
Shortcut: `M, R`
Commands: `moverotate | mr`

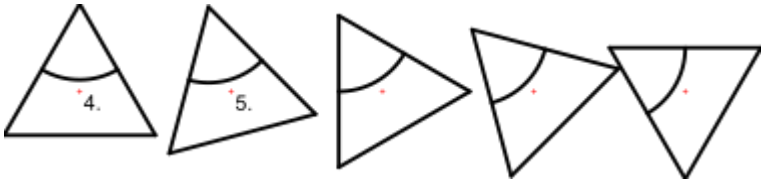
Description

Moves or copies and simultaneously rotates entities.

Usage

1. Select the entities you want to move or copy.
2. Launch this tool.

3. Enter the rotation angle in the options toolbar
4. Set the reference point with the mouse or enter a coordinate in the command line.
5. Set the target point.
   In the figure below, the two reference points are labeled. The rotation angle in the example is 15 degrees and the number of copies four. This results in a total rotation angle of 60 degrees.
6. The move and rotate dialog is displayed.
   To move the entities, choose "Delete Original", to copy them choose "Keep Original".
   You can also create any given number of copies at once by choosing "Multiple Copies" and entering the number of copies in the text line below.
   The new entities are placed on the same layer as the originals and have the same attributes. To use the current layer and current attributes instead, tick "Use current layer and attributes".
7. Click "OK" to move and rotate the entities.



Rotate Two

Toolbar / Icon:


Menu: `Modify > Rotate Two`
Shortcut: `R, 2`
Commands: `rotate2 | r2`

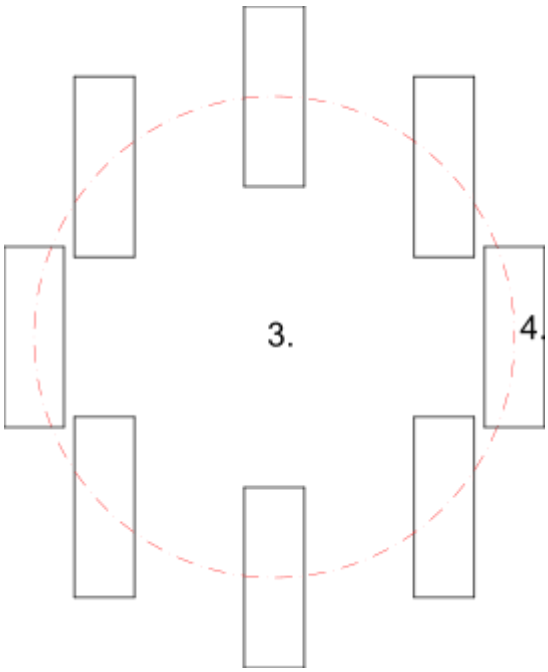
Description

Rotates and counter-rotates entities. This tool can be useful to rotate entities around a center while maintaining the original orientation of the entities themselves.

Usage

1. Select the entities you want to rotate and counter-rotate.
2. Launch this tool.

3. Set the center for the main rotation with the mouse or enter a coordinate in the command line.
4. Set the center of the rotation for the individual objects. This second center of rotation is rotated together with the entities around the first center.
5. The rotate two dialog is displayed where you can enter the rotation angle and counter-rotation angle.
   To delete the original entities, check "Delete Original", to copy them choose "Keep Original". You can also create any given number of copies by choosing "Multiple Copies".
   The new entities are placed on the same layer as the originals and have the same attributes. To use the current layer and current attributes instead, tick "Use current layer and attributes".
6. Click "OK" to rotate the entities.



Align Reference Points

Toolbar / Icon:


Add-on: QCAD Professional

Menu: `Modify > Align Reference Points`

Shortcut: `A, E`

Commands: `align | ae`

Description

Aligns the selected entities with other, existing entities.

Usage

1. Select the entities you want to align.
2. Launch this tool.
3. Check "Scale" to also scale the selection to fit between the two target points.
4. Check "Copy" to create an aligned copy of the selected entities.
5. Set the first reference point with the mouse or enter a coordinate in the command line. This point will be aligned with the first target point.
6. Set the first target point.
7. Set the second reference point to be aligned with the second target point.
8. Set the second target point.
9. The selection is moved, rotated and optionally scaled in such a way that the two chosen reference points align with the two chosen target points.

Align

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Modify > Align`
Shortcut: `M, A`
Commands: `alignhv`

Description

With this tool, the current selection can be aligned relative to an entity in the drawing, relative to the drawing bounding box or relative to the biggest or smallest entity in the selection.

Usage

1. Select the entities you want to align.
2. Start the tool.
3. In the options tool bar, choose to what you want to align the selection (a picked entity, the whole drawing or the biggest or smallest entity in the selection).

4. Choose how you want to align the selection (left, center, right, top, middle, bottom).
5. Tick the check box "Treat selection as group" to treat the geometry of the selected entities as one.
6. If you have chosen to align to a picked entity, click the entity now, otherwise, click the tick button in the options tool bar:


Offset (with Distance)

Toolbar / Icon:

Menu: `Modify > Offset (with Distance)`
Shortcut: `O, F`
Commands: `offset | of`

Description

With this tool you can create parallels to existing lines (or concentric arcs and circles).

Usage

1. Enter the distance of the concentric or parallel entity from the original entity in the options tool bar shown at the top.
2. Enter the number of parallel or concentric entities to create in the options tool bar.
3. Click the base entity. The parallel or concentric entities are created on that side on which the mouse cursor is located while selecting the entity.

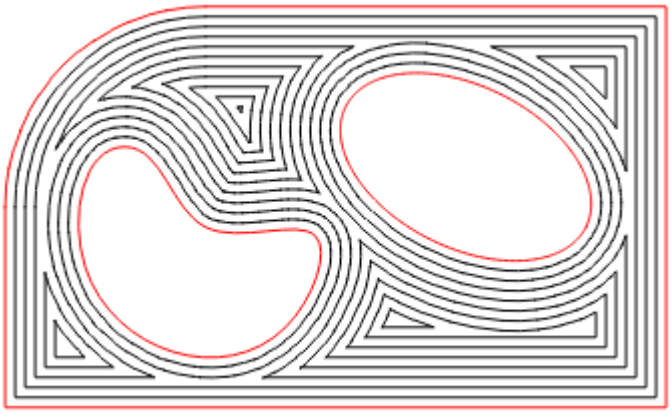
QCAD Professional:

Additional options for the handling of polylines are shown in the options tool bar:

Mode

-  Polyline Mode: In this mode, the offset tool offsets entire polylines and other contours at once.

If multiple polylines or contours are selected, they are all offset together as shown here:



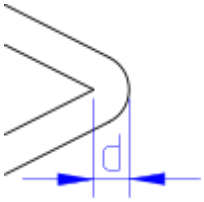
-  Segment Mode: In this mode, individual segments of polylines can be offset.

Join Types

- Round:

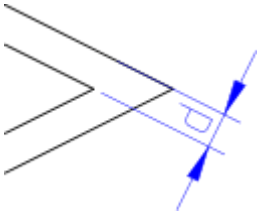
  Edges are rounded with a radius that equals the offset distance.

- Miter:

  Edges are trimmed.

Offset (through Point)

Toolbar / Icon:

Menu: `Modify > Offset (through Point)`
Shortcut: `O, H`
Commands: `offsetthrough | oh`

Description

With this tool you can create parallels to existing lines (or concentric arcs and circles).

Usage

1. Enter the number of entities to create in the options tool bar.
2. Click the base entity.
3. Click the position through which the first parallel or concentric arc or circle runs.

QCAD Professional:

The same additional options for the handling of polylines apply as for the Offset (with Distance) tool.

Trim

Toolbar / Icon:

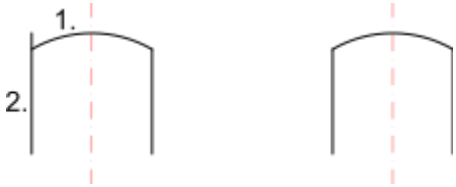
Menu: `Modify > Trim`
Shortcuts: `R, M | X, T`
Commands: `trim | extend | rm | xt`

Description

Trims or extends a line, arc or ellipse to another entity.

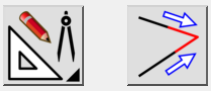
Usage

1. Choose the limiting entity to which you want to trim or extend one or more other entities.
2. Choose the entities that you want to trim to the limiting entity. Often there are two possibilities for the entity to be trimmed. In the example below, you might want the top part of the line to remain in the drawing and the lower part to be trimmed away. In that case you would have to click the trim entity at the upper part. Always click that part of the entity which you want to retain.



3. Right click twice or press Escape twice to terminate the tool.

Trim Both

Toolbar / Icon:


Menu: `Modify > Trim Both`
Shortcut: `T, M`
Commands: `trim2 | extend2 | tm`

Description

Trims or extends two lines, arcs or ellipses to their common intersection point.

Usage

1. Choose the first entity to be trimmed.
2. Choose the second entity to be trimmed.

3. Right click twice or press Escape twice to terminate the tool.

Lengthen / Shorten

Toolbar / Icon:

Menu: `Modify > Lengthen / Shorten`
Shortcut: `L, E`
Commands: `lengthen | shorten | trimamount | le`

Description

This tool can be used to either lengthen or shorten lines or arcs by a given amount.

Usage

1. Enter the distance you want to lengthen an entity by in the options tool bar. A positive value lengthens the entity, while a negative value shortens it. E.g. a value of '5' will lengthen the chosen entity by 5 units.
2. Choose the entity to be lengthened or shortened somewhere close to the end you want to modify.
3. Right click or press the Escape key to terminate the tool.

Stretch

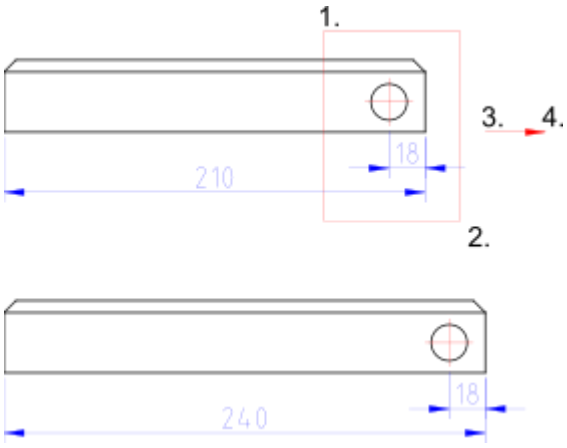
Toolbar / Icon:

Menu: `Modify > Stretch`
Shortcut: `S, S`
Commands: `stretch | ss`

Description

This tool stretches contours and dimension entities. This can also be described as moving all endpoints within a given rectangular or polygonal area.
If there is a selection of entities, this tool only affects the selected entities. Otherwise, this tool operates on all entities in the given area.



Usage

1. Choose in the options toolbar if you want to stretch a rectangular area (rectangle symbol) or a polygonal area (polygon symbol).
2. Set the first corner of the stretch area.
3. Set the second corner of the stretch area.
4. For polygonal areas, set the other corners of the stretch area and right-click the last corner when finished.
5. Set the reference point for moving using the mouse or enter a coordinate in the command line.
6. Set the target point.

QCAD Professional:

- In the options toolbar you can choose to stretch a rectangular or a polygonal area.
- For polygonal areas, all corner points of the polygon must be clicked. The last corner point is clicked with the right mouse button.

Clip to Rectangle

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Modify > Clip to Rectangle`
Shortcut: `C, L, R`
Commands: `cliprectangle | cliprect | clr`

Description

Clips the current selection to the boundaries of a rectangular area.

Usage

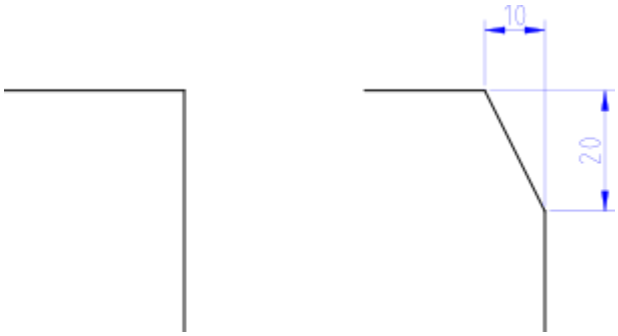
1. Select the entities you want to clip.
2. Specify the first corner of the rectangular area.
3. Move the mouse to the second corner and click to specify the second corner of the rectangular area.

Chamfer / Bevel

Toolbar / Icon:
 
Menu: `Modify > Chamfer / Bevel`
Shortcut: `C, H`
Commands: `bevel | chamfer | ch`

Description

Bevels a corner shaped by two entities. Optionally, the edge entities of the corner can be trimmed automatically to fit the new shape.



Usage

1. Enter the geometry of the bevel in the options tool bar. "Distance 1" is the distance the bevel line will have from the (imaginary) intersection of the two edges (10 in the example shown). "Distance 2" is the same distance for the second edge (20 in the example shown).
2. Check "Trim" if you want to trim the entities automatically. If the option is disabled, the two entities will remain untouched.
3. Choose the first edge entity (a line or an arc).
4. Choose the second entity.

Round

Toolbar / Icon:


Menu: `Modify > Round`
Shortcut: `R, N`
Commands: `round | rn`

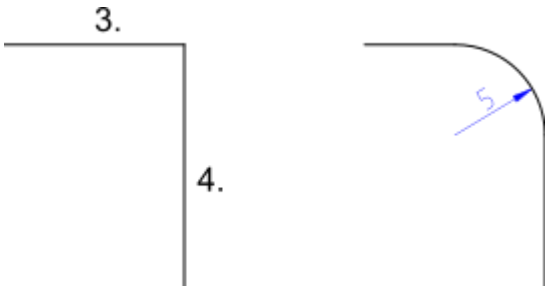
Description

Rounds the corner of a contour. Optionally, the edge entities of the corner can be trimmed automatically to fit the new shape.

Usage

1. Enter the radius of the rounding in the options tool bar (e.g. 5).
2. Check "Trim" if you want to trim the entities automatically. If the option is disabled, the two corner entities will remain untouched.
3. Choose the first edge entity (a line or arc).

4. Choose the second entity.



Divide

Toolbar / Icon:
 

Menu: `Modify > Divide`
Shortcut: `D, I`
Commands: `divide | di`

Description

Divides entities into two separate entities.

Usage

1. Choose the entity you want to divide.
2. Set the dividing point using the mouse. The dividing point is usually the intersection point with another entity. Choose the intersection snap mode to automatically snap to intersections.

Split Entities

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Modify > Split Entities`
Shortcut: `M, S`
Commands: `split | ms`

Description

This tool splits the selected lines, arcs and circles up into N equally long segments.

Usage

1. Select the entities you want to split up.
2. Launch this tool.
3. Use the options tool bar to specify into how many segments you want to split up the entities. If any circles are selected, you may also enter an angle offset for splitting up these circles.
4. Click the button with the tick to split up the selected entities:


Break out Segment

Toolbar / Icon:
 
Menu: `Modify > Break out Segment`
Shortcut: `D, 2`
Commands: `break | breakout | d2`

Description

Divides entities by cutting out a segment that is limited by intersecting entities.

Usage

Choose the entity you want to divide by clicking the segment you want to cut out. The segment is removed automatically if the check box "Remove Segment" is checked. If not, the entity is only cut at the intersections without removing the segment.

QCAD Professional:

- In the options toolbar you can activate the line mode to break out all segments intersected by a line.
- In this mode, you can click a start and an end point to break out all segments that are intersected by the line between the two points.

Break out Manual

Toolbar / Icon:
 

Menu: `Modify > Break out Manual`
Shortcut: `B, 2`
Commands: `brk | b2`

Description

Divides objects by cutting out a segment in between two user defined points.

Usage

1. Select the entity you want to break up.
2. Click the first point at which the entity is to be divided.
3. If the point does not lie on the entity, the separating point is assumed to be the closest point on the entity.
4. Click the second point at which the entity is to be divided.
5. The segment between the two points is removed if the "Remove Segment" option in the options toolbar is active. Otherwise, the entity is only separated at the given points.

Auto Trim

Toolbar / Icon:
 

Menu: `Modify > Auto Trim`
Shortcut: `A, X`
Commands: `autotrim | ax`

Description

Trims or extends an entity on both sides to the next limiting entities.

Usage

- Choose the entity you want to trim or extend by clicking the segment you want to keep. The entity is automatically trimmed or extended, so that the two end points align with the next two entities.

Break out Gap

Toolbar / Icon:
 

Menu: `Modify > Break out Gap`
Shortcut: `D, 3`
Commands: `breakoutgap | d3`

Description

Breaks out a gap from a line, arc or circle.

Usage

- Pick the entity from which you wish to break out a gap.
- Click the center of the gap on the entity.

Reverse

Toolbar / Icon:
 

Menu: `Modify > Reverse`
Shortcut: `R, V`
Commands: `reverse | rv`

Description

This tool reverses the direction of all selected line, arc and ellipse arc entities. This is mostly useful for drawings that are being prepared for further processing (e.g. CAM).

Edit Text

Toolbar / Icon:
 

Menu: `Modify > Edit Text`
Shortcut: `M, T`
Commands: `edittext | modifytext | mt`

Description

This tool can be used to edit existing text entities.

Usage

1. Click the text entity you want to edit.
2. The text dialog is shown where you can adjust the text and its attributes.

Note that instead of using this tool, you can also double-click the text entity that you want to edit.

Edit Tolerance

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Modify > Edit Tolerance`

Shortcut: `M, L`

Commands: `edittolerance | modifytolerance | ml`

Description

This tool can be used to edit existing text entities.

Usage

1. Click the text entity you want to edit.
2. The text dialog is shown where you can adjust the text and its attributes.

Note that instead of using this tool, you can also double-click the text entity that you want to edit.

Edit Hatch

Toolbar / Icon:

Menu: `Modify > Edit Hatch`

Shortcut: `M, H`

Commands: `edithatch | modifyhatch | mh`

Description

This tool can be used to edit existing hatch entities.

Usage

1. Click the hatch entity you want to edit.
2. The hatch dialog is shown where you can adjust the hatch and its attributes.

Note that instead of using this tool, you can also double-click the hatch entity that you want to edit.

Explode

Toolbar / Icon:
 

Menu: `Block > Explode`
Shortcut: `X, P`
Commands: `explode | xp`

Description

Converts block references, polylines, splines, ellipses, texts, dimensions and hatches into more basic entities. Dimensions are split up into lines, triangles and text entities. Text entities are split into polylines (or lines, arcs and splines). Splines are converted into polylines with tangentially connected arc segments. Ellipses are converted into polylines with arc segments. Polylines are split up into loose segments. Hatches are split up into lines. Solid fills are converted to the outline of the solid fill.

Usage

1. Select the entities you want to explode.
2. Launch this tool to explode the selected entities.
3. Repeat step 2 to further break up the selected entities if desired. If an entity cannot be further broken up, it remains unchanged.

Draw Order

Toolbar / Icon:


Bring to Front

Toolbar / Icon:


Menu: `Modify > Draw Order > Bring to Front`
Shortcut: `M, F`
Commands: `tofront | mf`

Description

Changes the order in which entities are displayed. Use this tool to bring entities into the foreground if they are hidden behind images, solid fills or other entities.

Usage

1. Select the entities to bring into the foreground of the drawing.
2. Launch this tool.

Send to Back

Toolbar / Icon:


Menu: `Modify > Draw Order > Send to Back`
Shortcut: `M, B`
Commands: `toback | mb`

Description

Changes the order in which entities are displayed. Use this tool to send entities that are hiding other entities into the background.

Usage

1. Select the entities to move into the background of the drawing.
2. Launch this tool.

Reorder

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Modify > Draw Order > Reorder`
Shortcut: `M, O`
Commands: `reorder | mo`

Description

With this tool you can reorder entities manually. The draw order of entities is usually not important unless an entity is hiding another one. However, for certain types of further processing (such as CAM), the order of entities can be crucial.

Usage

1. To reorder all entities in your drawing, make sure that nothing is selected. Otherwise, select the entities you wish to reorder. Note that if you reorder only selected entities, the order of all other entities remains unchanged. Other entities might be drawn before, after or in between selected entities.
2. Click the entity that should be drawn or processed first and subsequently click all entities in the desired order. When the last entity is clicked, the tool automatically terminates. The numerical order of the entities is shown as an overlay while this tool is running.

Order Connected Entities

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Modify > Draw Order > Order Connected Entities`
Shortcut: `M, C`
Commands: `orderconnected | mc`

Detection Tools

Toolbar / Icon:

Detect Duplicates

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Modify > Detection > Detect Duplicates`
Shortcut: `M, D`
Commands: `duplicates`

Description

This tool can be used to find, select and optionally delete identical entities in the current drawing.

Usage

1. After launching the tool, the options tool bar displays controls to enter tolerance values for the comparison of entities. Adjust these if appropriate.
2. Tick the option "Ignore Layer" if you want two identical entities on different layers to be considered equal.
3. Click the refresh button to run the search again if you have made any adjustments to the tolerances and the "Auto" check box is not checked:

   If the "Auto" check box is checked, the search for duplicates is automatically repeated if you change any of the options.
4. Click the delete button if you wish to delete the selected duplicates:

   
5. Click the close button to terminate this tool:

Detect Zero-Length Entities

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Modify > Detection > Detect Zero-Length Entities`

Shortcut: `M, Z`

Commands: `zerolength | zero`

Description

This tool can be used to find, select and optionally delete very small entities (length almost 0) in the current drawing.

Usage

1. After launching the tool, the options tool bar displays controls to enter tolerance values that are used to detect zero length entities. Adjust these if appropriate.

2. Click the refresh button to run the search again if you have made any adjustments to the tolerances and the "Auto" check box is not checked:
   
   If the "Auto" check box is checked, the search for duplicates is automatically repeated if you change any of the options.
3. Click the delete button if you wish to delete the selected zero length entities:
   
4. Click the close button to terminate this tool:
   

Projection Tools

Toolbar / Icon:


Shortcut: `W, J`
Commands: `projectmenu`

Orthographic Projection (Cylindrical)

Toolbar / Icon:

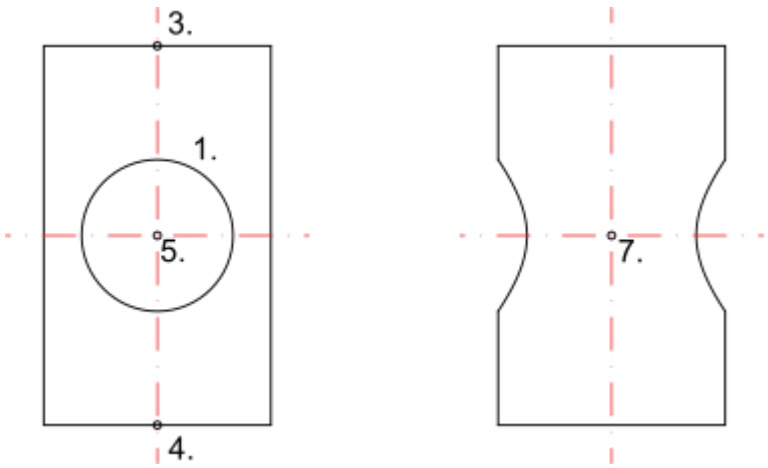

Add-on: QCAD Professional
Menu: `Modify > Projection > Orthographic Projection (Cylindrical)`
Shortcut: `P, R`
Commands: `orthocylindrical | orthocyl | pr`

Description

Creates an orthographic projection of the selected entities onto a cylindric surface. This is typically used to create the edge of a drilling in a side view of a cylinder as shown in the figure below on the right.

Usage

1. Select the entities you want to project. In the example below, that is the circle at the left.
2. Start this tool.
3. Click the start point of the unprojected cylinder axis. In the example below that is at the top center of the cylinder at the left.
4. Click the end point of the cylinder axis.
5. Click the reference point. The reference point in the selection will be used to place the projected entities into the side view.
6. Adjust the parameters in the options tool bar as desired.
   The cylinder diameter is the diameter of the cylindric shape we are projecting onto.
   The cylinder rotation is the rotation of the projected entities around the cylinder axis. Typically 90 or -90 degrees.
   Since the resulting shapes are likely neither lines or arcs, projected entities are split up into polylines with small segments. You can adjust the segment length used in the options tool bar.
   The clipping option can be enabled to automatically hide the part of the projected shape that would lay behind the X/Y plane.
7. Click the target point to position the projection.



Isometric Projection

Toolbar / Icon:

Menu: `Modify > Projection > Isometric Projection`
Shortcut: `P, J`
Commands: `isometric | pj`

Description

This tool creates isometric projections (and other types of projections) of the current selection in the drawing.

Usage

1. Select the entities you want to project.
2. Launch this tool.
3. Choose a projection type and a viewing direction for the projection in the options tool bar.
4. Set the reference point for the projection. This is the point you will use to position the projection in the next step.
5. Move the mouse cursor to the position where you want to create the projection and click to set the position.

Isometric Projection (Cylindrical)

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Modify > Projection > Isometric Projection (Cylindrical)`
Shortcut: `P, C`
Commands: `isometriccylindrical | isocyl | pc`

Description

This tool creates an isometric projection of the current selection onto a cylindric surface.

Usage

1. Select the entities you want to project.
2. Launch this tool.
3. Choose a viewing direction for the projection in the options tool bar.
4. Enter the diameter of the cylinder in the options tool bar.
5. Enter the rotation of the projected entities around the cylinder axis in degrees in the options tool bar.
6. Set the start point of the unprojected cylinder axis.
7. Set the end point of the cylinder axis.
8. Set the reference point for the projection. This is the point you will use to position the projection in the next step.
9. Move the mouse cursor to the position where you want to create the projection and click to set the position. Before setting the definitive position, you might still want to adjust the cylinder diameter, rotation angle and viewing direction in the options tool bar, until the preview matches your expectations.

Matrix Transformation

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Modify > Projection > Matrix Transformation`

Shortcut: `M, X`

Commands: `matrixtransform | mx`

Description

This tool transforms the selection using an arbitrary user defined transformation matrix.

Usage

1. Select the entities you want to transform.
2. Launch this tool.
3. Enter the transformation matrix in the options tool bar.

4. For convenience, you may additionally enter a factor that is applied to each element of the matrix.
5. Set the reference point for the transformation. This is the point you will use to position the transformed entities in the next step.
6. Move the mouse cursor to the position where you want to create the transformation and click to set the position. Before setting the definitive position, you might still want to adjust the transformation matrix in the options tool bar, until the preview matches your expectations.

Property Painter

Toolbar / Icon:
 

Add-on: **QCAD Professional**
Menu: `Modify > Property Painter`
Shortcut: `P, I`
Commands: `propertypainter | pi`

Description

This tool applies properties of a chosen entity to the current selection. Properties that can be copied for all entities are:

- Layer
- Color
- Lineweight
- Linetype
- Custom properties

If the source and destination entities are both text entities, the following text specific properties can be copied:

- Text font
- Text height
- Bold property
- Italic property

For hatch entities, the pattern can be copied:

- Solid property
- Pattern
- Scale
- Angle

Usage

1. Select the target entities to which you want to apply the properties.
2. Start this tool.
3. Choose in the options toolbar which properties you want to copy.
4. Click the entity from which the properties are copied.

Snap Tools

Toolbar / Icon:


Commands: `snapmenu`

Free

Toolbar / Icon:


Menu: `Snap > Free`
Shortcut: `S, F`
Commands: `snapfree | sf`

Description

Allows free positioning using the mouse. Note that this is almost never the recommended way of setting coordinates in a CAD system except for drawing freehand lines.

Grid

Toolbar / Icon:

Menu: `Snap > Grid`

Shortcut: `S, G`

Commands: `snapgrid | sg`

Description

Snaps to grid points.

End

Toolbar / Icon:

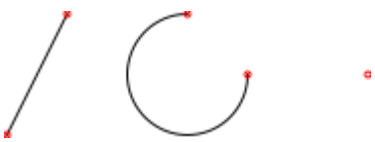
Menu: `Snap > End`

Shortcut: `S, E`

Commands: `snapend | se`

Description

Snaps to end points of lines, arcs polyline segments, splines, ellipse arcs and to points.



On Entity

Toolbar / Icon:

Menu: `Snap > On Entity`
Shortcut: `S, T`
Commands: `snaponentity | st`

Description

Snaps to the closest point on an entity.

Perpendicular

Toolbar / Icon:

Menu: `Snap > Perpendicular`
Shortcut: `S, U`
Commands: `snapperpendicular | su`

Description

Snaps to the perpendicular point on a line, arc, circle or ellipse. This only applies when drawing lines.

Tangential

Toolbar / Icon:

Menu: `Snap > Tangential`
Shortcut: `S, B`
Commands: `snaptangential | sb`

Description

Snaps to the point of tangency on an arc, circle or ellipse. This only applies when drawing lines. The snap point is the tangent point of an imaginary line from the relative zero point, tangential to the clicked arc or circle.

Center

Toolbar / Icon:

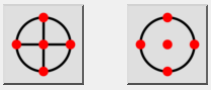
Menu: `Snap > Center`
Shortcut: `S, C`
Commands: `snapcenter | sc`

Description

Snaps to the center of arcs, circles and ellipses as well as to the middle of lines and to points.

Reference

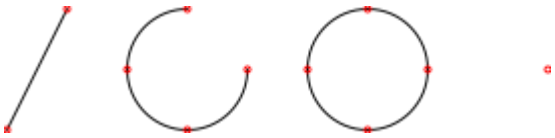
Toolbar / Icon:



Menu: `Snap > Reference`
Shortcut: `S, R`
Commands: `snapreference | sr`

Description

Snaps to reference points. Reference points are the blue points that are shown when an entity is selected. This is particularly useful to snap to reference points of circles and arcs, for example when drawing center lines.



Middle

Toolbar / Icon:

Menu: `Snap > Middle`
Shortcut: `S, M`
Commands: `snapmiddle | sm`

Description

Snaps to middle points of lines and arcs and to point entities. Note that the middle of an arc is at the middle of the arc line on the arc, not at its center.

Middle Manual

Toolbar / Icon:

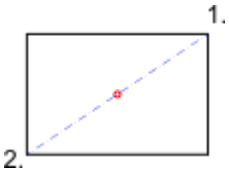
Menu: `Snap > Middle Manual`
Shortcut: `S, N`
Commands: `snapmiddlemanual | sn`

Description

This tool allows you to snap to a point that is in the middle between two points. This is most commonly used to snap to the center of a rectangle or polygon by choosing two diagonally opposite corners.

Usage

1. After activating this snap tool, specify the first of the two points that define the middle point. For example one corner of a rectangle.
2. Click the second of the two points, for example the diagonally opposite corner of the rectangle.



Distance

Toolbar / Icon:

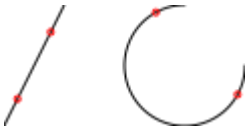
Menu: `Snap > Distance`
Shortcut: `S, D`
Commands: `snapdistance | sd`

Description

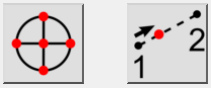
Snaps to points with a given distance to the end point of lines or arcs.

Usage

1. Enter the distance into the options toolbar.
2. Move the cursor to the entity along which the distance should be measured.
3. When the desired position / preview is shown, click the left mouse button to set the coordinate.



Distance Manual

Toolbar / Icon:


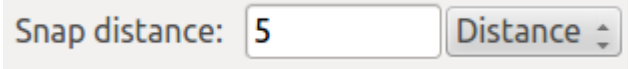
Menu: `Snap > Distance Manual`
Shortcut: `S, H`
Commands: `snapdistancemanual | sh`

Description

This tool allows you to snap to a point that lays on the imaginary line between two points at a given distance from the first point.
This snap tool does not restrict the distance, percentage or fraction entered. Entering a distance greater than the distance between the two points, a percentage greater than 100 or a fraction greater than 1.0 will allow you to snap to a point beyond the second point.
Negative values can be entered to snap to points beyond the first point.
Fractions may be entered as decimal numbers (0.5, 0.7) or as fractions (1/7, 3/11).

Usage

1. After activating this snap tool, choose "Distance", "Percentage", or "Fraction" and enter the desired value in the options tool bar.



2. Specify the first point. That is the point to measure the distance from.
3. Click the second point.

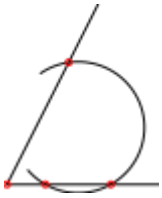
Intersection

Toolbar / Icon:

Menu: `Snap > Intersection`
Shortcut: `S, I`
Commands: `snapintersection | si`

Description

Snaps to intersection points between entities.



Intersection Manual

Toolbar / Icon:

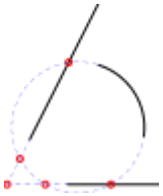
Menu: `Snap > Intersection Manual`
Shortcut: `S, Y`
Commands: `snapintersectionmanual | sy`

Description

Sometimes the intersection snap mode cannot be used to snap to an intersection point because the intersection point lays outside one or both of the entities. This snap tool lets you explicitly specify two intersecting entities and snap to their intersection point.

Usage

1. After activating this snap tool, click the first of the two intersecting entities.
2. Click the second of the two intersecting entities. If two intersection points are possible, make sure to click the second entity somewhere closer to the intersection point you want to snap to.



Auto

Toolbar / Icon:

Menu: `Snap > Auto`

Shortcut: `S, A`

Commands: `snapauto | sa`

Description

The auto snap tool snaps to the closest intersection point, end point, middle point, perpendicular point, reference point, grid point or point on an entity in that order of priority. The types of reference points the auto snap tool snaps to can be configured under *Edit > Application Preferences > Snap > Auto snap*.

Coordinate

Toolbar / Icon:


Menu: `Snap > Coordinate`
Shortcut: `S, X`
Commands: `snapcoordinate | sx`

Description

Defines a point by entering an absolute or relative Cartesian coordinate.

Usage

1. Start this tool when you have to specify a point and want to do so by entering a pair of coordinates (x/y).
2. Enter the coordinates in the options tool bar and choose if they are absolute coordinates or a relative ones (relative to the relative zero point).
3. Click the OK button or press Enter to confirm the input and set the coordinates:

Polar Coordinate

Toolbar / Icon:


Menu: `Snap > Polar Coordinate`
Shortcut: `S, O`
Commands: `snappolar | so`

Description

Defines a point by entering an absolute or relative polar coordinate.

Usage

1. Start this tool when you have to specify a point and want to do so by entering a polar coordinate.
2. Enter the polar coordinate (radius and angle) in the options tool bar and choose if it is an absolute coordinate or a relative one (relative to the relative zero point).
3. Click the OK button or press Enter to confirm the input and set the coordinate:


X/Y from Points

Toolbar / Icon:

Menu: `Snap > X/Y from Points`
Shortcut: `., X`
Commands: `.x`

Description

This snap tool can be used to extract one coordinate (e.g. X) from the location of one existing entity and the other coordinate (e.g. Y) from the location of another object.
This can for example be used to locate the center of a rectangle.
This concept is also referred to as 'combining coordinates', 'point filtering' or 'coordinate filtering'.

Usage

1. When specifying a position, start this tool.
2. Click a position in the drawing to lock the X coordinate at that position.
3. Click a position in the drawing to lock the Y coordinate at that position.

Y/X from Points

Toolbar / Icon:

Menu: `Snap > Y/X from Points`
Shortcut: `., Y`
Commands: `.y`

Description

This tool works the same way as the tool X/Y from Points, but with opposite order of coordinate input.

Center of Selection

Toolbar / Icon:

Menu: `Snap > Center of Selection`
Shortcut: `S, K`
Commands: `snapselectioncenter | sk`

Description

Snaps to the center of the current selection. This is the point at the center between the leftmost and the rightmost edge in X direction and the lowermost and the uppermost edge in Y direction of the selection.

Restrictions Off

Toolbar / Icon:


Menu: `Snap > Restrictions Off`
Shortcut: `E, N`
Commands: `restrictoff | en`

Description

Switches off all snap restrictions.

Restrict Orthogonally

Toolbar / Icon:

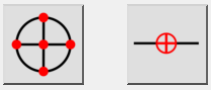

Menu: `Snap > Restrict Orthogonally`
Shortcut: `E, O`
Commands: `restrictorthogonal | eo`

Description

Restricts the cursor vertically or horizontally to the same X or Y position as the relative zero point.

Restrict Horizontally

Toolbar / Icon:



Menu: `Snap > Restrict Horizontally`
Shortcut: `E, H`
Commands: `restrictHorizontal | eh`

Description

Restricts the cursor horizontally to the same Y position as the relative zero point.

Restrict Vertically

Toolbar / Icon:

Menu: `Snap > Restrict Vertically`
Shortcut: `E, V`
Commands: `restrictVertical | ev`

Description

Restricts the cursor vertically to the same X position as the relative zero point.

Restrict Angle or Length

Toolbar / Icon:

Menu: `Snap > Restrict Angle or Length`
Shortcut: `E, L`
Commands: `restrictanglelength | el`

Description

Restricts the cursor movement by angle and distance in relation to the relative zero point.

Set Relative Zero

Toolbar / Icon:

Menu: `Snap > Set Relative Zero`
Shortcut: `R, Z`
Commands: `setrelativezero | rz`

Description

Lets you set a new location for the relative zero point.

Usage

- Click the new location of the relative zero point.

Lock Relative Zero

Toolbar / Icon:

Menu: `Snap > Lock Relative Zero`
Shortcut: `R, L`
Commands: `lockrelativezero | rl`

Description

When enabled, this switch locks the position of the relative zero point. This means it does not move around automatically but you can still move it manually using the tool to set the position of the relative zero point.

Lock Snap

Toolbar / Icon:

Add-on: **QCAD Professional**
Menu: `Snap > Lock Snap`
Shortcut: `S, Q`
Commands: `locksnap | sq`

Description

When enabled, this switch keeps the current snap tool active even when a new tool is started.

Information Tools

Toolbar / Icon:

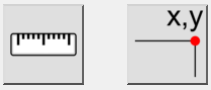
Shortcut: `W, I`
Commands: `infomenu | informationmenu | measuringmenu`

Description

The tools in the Info menu can be used to measure distances, areas and query other information from the drawing.
Some of the tools offer an option to add the measured result to the drawing as text if desired.

Position

Toolbar / Icon:



Menu: `Info > Position`
Shortcut: `I, O`
Commands: `infopos | io`

Description

This tool outputs the absolute, Cartesian coordinates of chosen points in the drawing.

Relative Position

Toolbar / Icon:


Menu: `Info > Relative Position`
Shortcut: `I, V`
Commands: `infoposrel | iv`

Description

This tool outputs the relative, Cartesian coordinates of chosen points in the drawing.

Polar Position

Toolbar / Icon:


Menu: `Info > Polar Position`
Shortcut: `I, L`
Commands: `infopospol | il`

Description

This tool outputs the absolute, polar coordinates of chosen points in the drawing.

Relative Polar Position

Toolbar / Icon:

Menu: `Info > Relative Polar Position`
Shortcut: `I, T`
Commands: `infopospolrel | it`

Description

This tool outputs the relative, polar coordinates of chosen points in the drawing.

Distance Point to Point

Toolbar / Icon:

Menu: `Info > Distance Point to Point`
Shortcut: `I, P`
Commands: `infodist | ip`

Description

This tool measures the exact distance between two points given by the user.

Usage

1. Use the mouse to specify the location of the first point or enter a coordinate in the command line.
2. Specify the second point.
3. The measured distance is printed on screen and in the command line.

Distance Entity to Point

Toolbar / Icon:

Menu: `Info > Distance Entity to Point`
Shortcut: `I, E`
Commands: `infodistep | ie`

Description

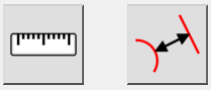
This tool measures the exact distance between an entity and a point given by the user.

Usage

1. Specify the entity.
2. Use the mouse to specify the location of the point or enter a coordinate in the command line.
3. The measured distance is printed on screen and in the command line.

Distance Entity to Entity

Toolbar / Icon:



Menu: `Info > Distance Entity to Entity`
Shortcut: `I, N`
Commands: `infodistee | in`

Description

This tool measures the exact distance between an entity and a point given by the user.

Usage

1. Specify the entity.
2. Use the mouse to specify the location of the point or enter a coordinate in the command line.

3. The measured distance is printed on screen and in the command line.

Angle

Toolbar / Icon:

Menu: `Info > Angle`
Shortcut: `I, A`
Commands: `infoangle | ia`

Description

This tool measures the angle between two given lines.

Usage

1. Specify the first line.
2. Specify the second line.
3. The measured angle (in degrees) is printed on screen and in the command line.

Total Length of Selection

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Info > Total Length of Selection`
Shortcut: `I, S`
Commands: `infosum | is`

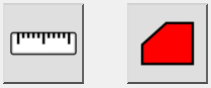
Description

This tool calculates the total length of a selection of entities.

Usage

1. Select the entities you want to measure the total length from.
2. Start the tool.
3. The total length of the selected entities is printed in the command line.

Polygonal Area

Toolbar / Icon:


Menu: `Info > Polygonal Area`
Shortcut: `I, R`
Commands: `infoarea | ir`

Description

This tool measures the area covered by a polygon and its circumference.

Usage

1. Specify the first point of the area.
2. Specify the other points that define the area and right click when you are finished.
3. The measured area is printed on screen and in the command line. The circumference is also printed in the command line.

Arc/Circle/Ellipse Area

Toolbar / Icon:


Menu: `Info > Arc/Circle/Ellipse Area`
Shortcut: `I, C`
Commands: `acearea | ic`

Description

Author: Robert S.

This command calculates the area of arcs, circles, ellipses and elliptical arcs, and optionally adds the information to the current layer of a drawing. If you select any other type of entity a warning is shown in the command line.

For arcs, there are two options for calculating areas, namely Segment or Sector. The area of a segment (or slice) is the area bound by the arc and two lines drawn from the arc's startpoint and endpoint to the arc's centre. The area of a sector is the area bound by the arc and a chord drawn from the arc's startpoint to the endpoint.

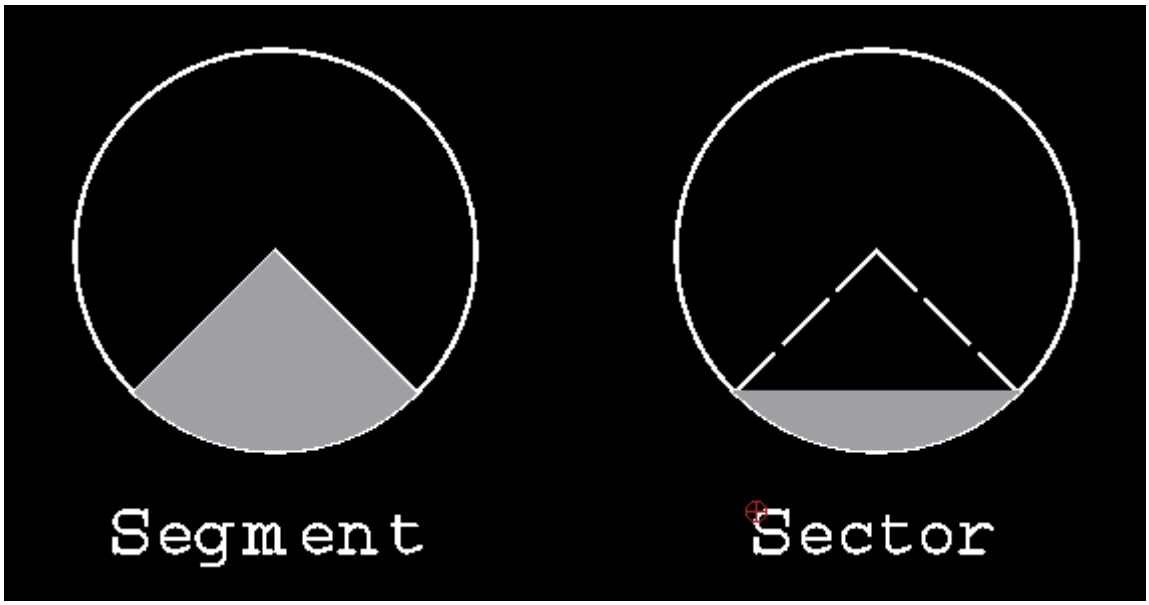


Figure 1: a segment (slice) and sector.

Options

This command has the standard "*Add measurement to drawing*" and "*Text Height*" options.

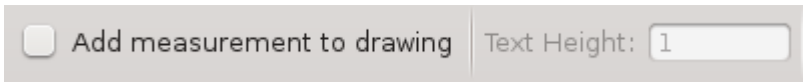


Figure 2: Standard Info options

Whether this is off or not, when you click on an arc, circle, ellipse or elliptical arc, the information is displayed in the command line widget.

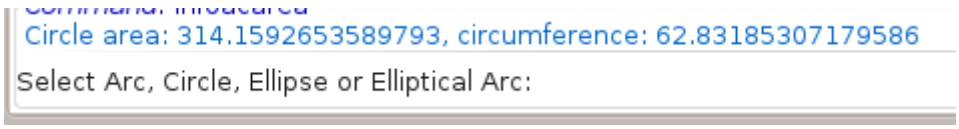


Figure 3: Info displayed in command line

(Note: Every time you start an info command the "*Add measurement to drawing*" checkbox is always unchecked. You must remember to switch it on if you want the info added to the

drawing).

It also adds the following options.

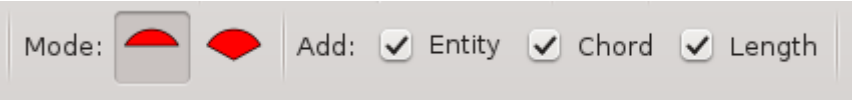


Figure 4: Added options

Mode:

The mode only applies to arcs.

Sector: This mode returns the area of the sector. This is useful when used in conjunction with the "*Polygonal Area*" command.

 *Segment:* This mode returns the area of the segment.

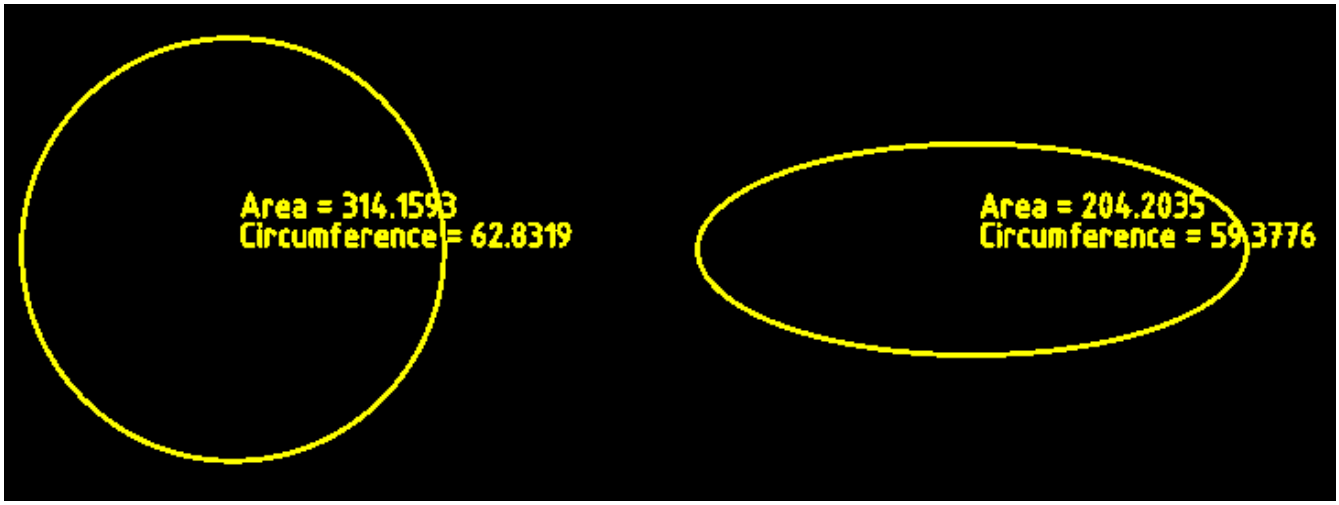
Add:

Entity: This selects whether you want to add a copy of the measured entity to the current layer of the drawing.

Chord: This selects if you want the chord (sector), or radius lines (segment), added to the drawing.

Length: By default the area is always printed on the drawing. This option selects whether you want extra information added to the drawing.

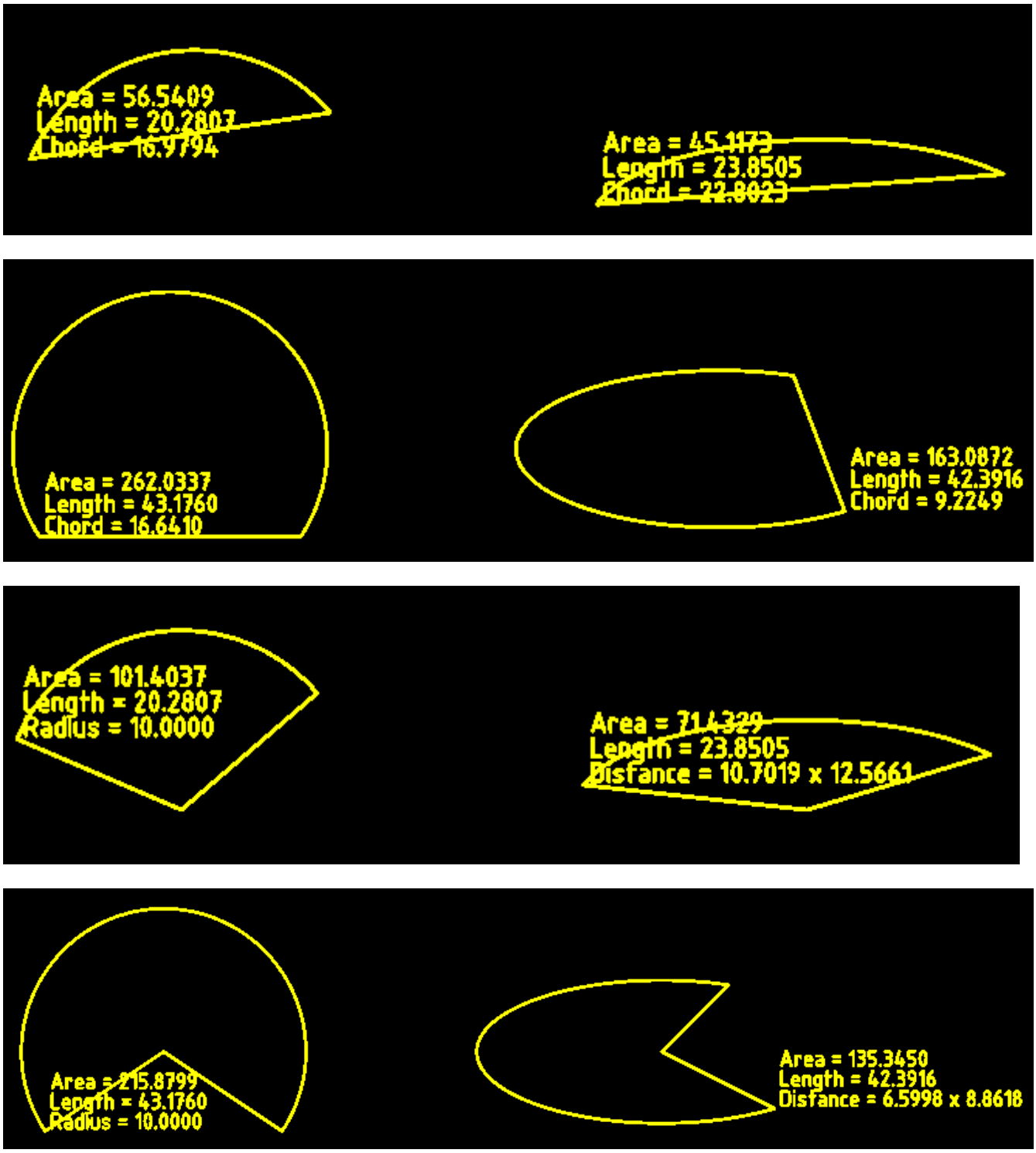
For circles and full ellipses the extra information is the circumference. This uses the centre point as the lower left corner for the added text.



For circular arcs the extra information is the length of the arc and the length of the chord (in sector mode), or the radius (in segment mode).

For ellipses the extra information is the length of the arc and the length of the chord (in sector mode), or the distance from the centre point to the startpoint of the arc, followed by the distance from the centre point to the endpoint of the arc (in segment mode).

The lower left corner for the added text is the endpoint of the arc.



Polyline Area

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Info > Polyline Area`
Shortcut: `I, I`
Commands: `infopolylinearea | ii`

Description

This tool measures the area of a polyline.

Usage

1. Specify the polyline entity.
2. The measured area is printed on screen and in the command line.

Layer Tools

Toolbar / Icon:

Shortcut: `W, Y`
Commands: `layermenu`

Description

The tools in the Layer menu can be used to add, modify and delete layers. Some of the tools are also available as tool buttons at the top of the layer list.

Toggle Visibility

Toolbar / Icon:

Menu: `Layer > Toggle Visibility`
Shortcut: `Y, V`
Commands: `layertogglevisibility | yv`

Description

This tool toggles the visibility of the current layer. You may also toggle layers by clicking on the eye icon beside the layer name in the layer list. A black eye icon means that the layer is visible, a gray icon that the layer is hidden.

Toggle Frozen Status

Toolbar / Icon:


Add-on: **QCAD Professional**
Menu: `Layer > Toggle Frozen Status`
Shortcut: `Y, F`
Commands: `layertogglefrozen | yf`

Description

This tool toggles the frozen status of the current layer. You may also toggle the frozen status of layers by clicking on the ice or sun icon beside the layer name in the layer list.

Note that block references on frozen layers are not shown while block references on layers that are off are still shown (provided that they contain any entities on layers which are on and not frozen).

Toggle Lock Status

Toolbar / Icon:


Menu: `Layer > Toggle Lock Status`
Shortcut: `Y, L`
Commands: `layertogglelock | yl`

Description

This tool toggles the locked status of the current layer. You may also toggle the locked status of layers by clicking on the lock icon beside the layer name in the layer list.

Entities on locked layers cannot be selected, deleted or modified.

Show Only Active

Toolbar / Icon:


Menu: `Layer > Show Only Active`
Shortcut: `Y, O`
Commands: `layershowactive | yo`

Description

This tool hides all layers except for the selected (active) layer in the layer list.

Show All Layers

Toolbar / Icon:
 

Menu: `Layer > Show All Layers`
Shortcut: `Y, S`
Commands: `layershowall | ys`

Description

This tool can be used to conveniently show all layers at once. This can be useful while working with a drawing with many layers.

Hide All Layers

Toolbar / Icon:
 

Menu: `Layer > Hide All Layers`
Shortcut: `Y, H`
Commands: `layerhideall | yh`

Description

This tool can be used to conveniently hide all layers at once. This can be useful while working with a drawing with many layers.

Thaw All Layers

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Layer > Thaw All Layers`

Shortcut: `Y, W`

Commands: `layerthawall | yw`

Description

This tool can be used to conveniently thaw all layers at once. This can be useful while working with a drawing with many layers.

Freeze All Layers

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Layer > Freeze All Layers`

Shortcut: `Y, Z`

Commands: `layerfreezeall | yz`

Description

This tool can be used to conveniently hide (freeze) all layers at once. This can be useful while working with a drawing with many layers.

Lock All Layers

Toolbar / Icon:

Menu: `Layer > Lock All Layers`
Shortcut: `Y, K`
Commands: `layerlockall | yk`

Description

This tool can be used to conveniently lock all layers at once.

Unlock All Layers

Toolbar / Icon:

Menu: `Layer > Unlock All Layers`
Shortcut: `Y, N`
Commands: `layerunlockall | yn`

Description

This tool can be used to conveniently unlock all layers at once.

Add Layer

Toolbar / Icon:

Menu: `Layer > Add Layer`
Shortcut: `Y, A`
Commands: `layeradd | addlayer | ya`

Description

This command creates a new layer. The layer dialog is shown where you can specify the name for the new layer and adjust its attributes.

QCAD Professional

In addition, you can choose if objects on this layer can be snapped to and if this layer should be printed.
A layer can be made a sub-layer of a parent layer. The layer list will then show a tree structure of hierarchical layers.

Add Sublayer

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `Layer > Add Sublayer`
Shortcut: `Y, U`
Commands: `layeraddsub | yu`

Create Layer from Selection

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Layer > Create Layer from Selection`

Shortcut: `Y, T`

Commands: `layerfromselection | yt`

Move Selection to Current Layer

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Layer > Move Selection to Current Layer`

Shortcut: `Y, J`

Commands: `layertocurrent | yj`

Duplicate Layer

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Layer > Duplicate Layer`

Shortcut: `Y, Y`

Commands: `layercopy | layerduplicate | yy`

Description

This tool creates a copy of the layer currently selected in the layer list. A dialog is shown that allows you to enter a name for the new layer.

Delete Layer(s)

Toolbar / Icon:

Menu: `Layer > Delete Layer(s)`
Shortcut: `Y, R`
Commands: `layerdelete | yr`

Description

You can delete the currently selected layer from the layer list with this command. Please note that all entities on that layer will be deleted as well.
Layer "0" can never be deleted.

Purge Unused Layers

Toolbar / Icon:

Add-on: **QCAD Professional**
Menu: `Layer > Purge Unused Layers`
Shortcut: `Y, P`
Commands: `purgelayers`

Edit Layer

Toolbar / Icon:


Add-on: QCAD/CAM
Menu: `Layer > Edit Layer`
Shortcut: `Y, E`
Commands: `layeredit | ye`

Select Layer Entities

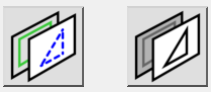
Toolbar / Icon:


Menu: `Layer > Select Layer Entities`
Shortcut: `Y, C`
Commands: `layerselect | selectlayer | yc`

Description

This tool adds all entities on the current layer to the selection. Only entities on the current block can be selected, entities on other blocks are not selected.

Deselect Layer Entities

Toolbar / Icon:


Menu: `Layer > Deselect Layer Entities`
Shortcut: `Y, D`
Commands: `layerdeselect | deselectlayer | y- | yd`

Description

This tool deselects all entities on the current layer.

Layer States

Toolbar / Icon:

Add...

Add-on: QCAD Professional
Menu: `Layer > Layer States > Add...`
Shortcut: `Y, M`

Block Tools

Toolbar / Icon:

Shortcut: `W, B`
Commands: `blockmenu`

Explode

Toolbar / Icon:
 

Menu: `Block > Explode`
Shortcut: `X, P`
Commands: `explode | xp`

Description

Converts block references, polylines, splines, ellipses, texts, dimensions and hatches into more basic entities. Dimensions are split up into lines, triangles and text entities. Text entities are split into polylines (or lines, arcs and splines). Splines are converted into polylines with tangentially connected arc segments. Ellipses are converted into polylines with arc segments. Polylines are split up into loose segments. Hatches are split up into lines. Solid fills are converted to the outline of the solid fill.

Usage

1. Select the entities you want to explode.
2. Launch this tool to explode the selected entities.
3. Repeat step 2 to further break up the selected entities if desired. If an entity cannot be further broken up, it remains unchanged.

Toggle Visibility

Toolbar / Icon:
 

Menu: `Block > Toggle Visibility`
Shortcut: `B, V`
Commands: `blocktogglevisibility | bv`

Description

Click the eye icon in the block list to toggle the visibility of a block. Block references of invisible blocks are not displayed.

Show All Blocks

Toolbar / Icon:

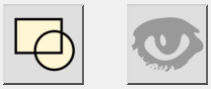
Menu: `Block > Show All Blocks`
Shortcut: `B, S`
Commands: `blockshowAll | bs`

Description

This tool makes sure that all block references in the drawing are shown.

Hide All Blocks

Toolbar / Icon:



Menu: `Block > Hide All Blocks`
Shortcut: `B, H`
Commands: `blockhideAll`

Description

This tool hides all block references in the drawing.

Add Empty Block

Toolbar / Icon:

Menu: `Block > Add Empty Block`
Shortcut: `B, A`
Commands: `blockadd | ba`

Description

This command creates a new empty block. The block dialog is shown where you can specify a name for the new block. If you want to create a block from existing entities, please use the Create Block command instead.

Create Block from Selection

Toolbar / Icon:

Menu: `Block > Create Block from Selection`
Shortcut: `B, C`
Commands: `blockcreate | bc`

Description

Creates a new block from existing entities.

Usage

1. Select the entities you want to use for the new block.
2. Specify the reference point of the block.
3. Enter a name for the new block in the dialog that is shown and click OK.

The block is now added to the block list and instances of it can be inserted into the drawing. The entities you have selected in the first step are removed and replaced by a block reference of the newly created block.

Add Layout Block

Toolbar / Icon:
 
Add-on: QCAD Professional
Menu: `Block > Add Layout Block`
Shortcut: `B, L`
Commands: `layoutadd | bl`

Description

This tool creates a new layout block. Layout blocks are typically used to arrange different parts of a drawing on a sheet of paper for printing. Layout blocks typically contain a drawing border, header and one or multiple viewports.

Usage

- Layout blocks have a name (shown in the block list) and a tab order (order in the block list). Both can be entered when creating the layout block.

Remove Block

Toolbar / Icon:

Menu: `Block > Remove Block`
Shortcut: `B, R`
Commands: `blockremove | br`

Description

Removes the active block. The block, all entities on it and all block references of the block will be deleted.

QCAD Professional

Multiple blocks can be selected to delete all selected blocks at once.

Purge Unused Blocks

Toolbar / Icon:
 

Add-on: QCAD Professional
Menu: `Block > Purge Unused Blocks`
Shortcut: `B, P`
Commands: `purgeblocks`

Rename Block

Toolbar / Icon:
 

Menu: `Block > Rename Block`
Shortcut: `B, N`
Commands: `blockrename | bn`

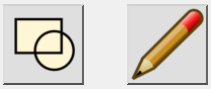
Description

Renames the active block. The same dialog is presented as for creating new blocks. Block references are automatically updated to reference the renamed block.

QCAD Professional

If the currently selected block is a layout block, the layout can be renamed with this tool. In this case, the dialog also shows the order number of the layout. Layouts in the block list are ordered by this number.

Edit Block

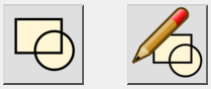
Toolbar / Icon:


Menu: `Block > Edit Block`
Shortcut: `B, E`
Commands: `blockedit | be`

Description

This tool opens the active block for editing. It can be modified just like the main drawing. To return to the main drawing (called "*Model_Space"), activate block "*Model_Space" and click the same button again to edit that block or start the tool "Edit Main Drawing" in menu "Block".

Edit Block from Reference

Toolbar / Icon:


Menu: `Block > Edit Block from Reference`
Shortcut: `B, D`
Commands: `blockeditfromreference | bd`

Description

With this tool, a block can be edited by choosing an existing block reference entity. This is useful and usually faster if you don't know the name of the block you want to edit.

Usage

- Click any block reference entity that references the block you want to edit.

The block is opened for editing. To return to the main drawing (called "*Model_Space"), edit block "*Model_Space" or start the tool "Edit Main Drawing" in menu "Block".

Edit Block In-place

Toolbar / Icon:


Add-on: QCAD Professional
Menu: `Block > Edit Block In-place`
Shortcut: `B, B`
Commands: `blockeditinplace | bb`

Description

With this tool a block can be edited at the location where it was inserted into the drawing. The block can be inserted uniformly scaled or rotated.

This may appear as if the contents of a block reference was being edited. This is not the case. It's important to keep in mind that it is the block definition that is being edited. Any changes made to the block will affect not only the block reference that is being edited but also all other block references that are using that same block.

Usage

1. Click the block reference you want to edit in-place.
2. QCAD switches into block editing mode with the block entities as working set. The working set is the set of entities that can currently be modified. Various tool buttons are shown in the options toolbar. These tools are only shown in block editing mode when no other tool is active (neutral mode of QCAD). In this mode, the block entities (working set) are shown in their regular colors while the rest of the drawing is faded. This indicates that the faded entities can still be used as references for example for trimming or to create offsets but they cannot be modified as they are not in the current working set.
3. You can now change the block as desired using the normal CAD tools to draw, modify or delete entities.
4. When you are done, click "Save and Close" to exit the block editor and return to the main drawing.

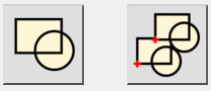
5. Other tools in block editing mode are:
   Save and Update to save the block and update all block references accordingly. This can be used to update the drawing and preview what effect the current modifications have. The block editing mode is not closed.
   Auto Update to automatically update all other block references whenever the contents of the block is edited.
   Add to Working Set to add selected entities from outside the current working set to the working set and add them to the block.
   Remove from Working Set to remove the selected entities from the working set and from the block.
   Cancel to cancel all changes since the last save and return to the main drawing.

Preferences

Edit > Application Preferences > Graphics View > Behavior

- *Edit block reference in-place:* Check this preference to open the in-place block editor when double-clicking a block. To exit the in-place block editor, you can double-click into an empty space of your drawing.

Duplicate Block

Toolbar / Icon:

Add-on: QCAD Professional
Menu: `Block > Duplicate Block`
Shortcut: `B, Y`
Commands: `blockcopy | blockduplicate | by`

Description

This tool creates a copy of the block currently selected in the block list. A dialog is shown that allows you to enter a name for the new block.

Return to Main Drawing

Toolbar / Icon:

Menu: `Block > Return to Main Drawing`
Shortcut: `E, M`
Commands: `editmain | em`

Description

Choose this tool when you are editing a block and want to return to the main drawing (block "*Model_Space").

Select Block References

Toolbar / Icon:

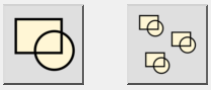
Menu: `Block > Select Block References`
Shortcut: `B, K`
Commands: `blockselect | selectblock | bk`

Description

This tool selects all block references of the block that is currently selected in the block list. Only block references which are part of the block that is currently being edited are selected.

Deselect Block References

Toolbar / Icon:



Menu: `Block > Deselect Block References`
Shortcut: `B, X`
Commands: `blockdeselect | deselectblock | bx`

Description

This tool deselects all block references of the block that is currently selected in the block list.

Insert Block

Toolbar / Icon:

Menu: `Block > Insert Block`
Shortcut: `B, I`
Commands: `blockinsert | minsert | insert | bi`

Description

This tool inserts the active block into the drawing. One or multiple block references are created in the drawing to represent the active block.

Usage

1. Choose the block you want to insert from the block list.
2. Click the insert button or choose 'Insert Block' from the menu.
3. Enter the rotation angle and scale factor for the block reference in the options tool bar.
4. Specify the position of the inserted block by clicking a coordinate or entering a coordinate in the console.

Create Library Item

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Block > Create Library Item`

Shortcut: `B, T`

Commands: `librarycreate | bt`

Description

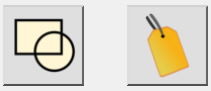
Creates a new part library item from a selection in the drawing.

Usage

1. Select the entities you want to use for the new library item.
2. Specify the reference point of the item.
3. Enter a name for the new item and choose a library and location where to store the item. Click OK.

Attribute Tools

Toolbar / Icon:



Shortcut: `W, U`

Commands: `attributemenu`

Define Block Attribute

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Block > Attributes > Define Block Attribute`

Shortcut: `U, A`

Commands: `attributedefinition | attdef | ua`

Description

Use this tool to add an attribute definition to a block definition.

Usage

1. The attribute definition dialog is shown.
2. Choose the font name, text height and text properties as described in the documentation of the text tool.
3. Enter a tag for the attribute definition.
   This identifier is used to link attributes to their attribute definition. You can also think of the tag as a variable name.
4. Enter a prompt for the attribute definition. This text is shown whenever the user is requested to enter a value for this attribute.
5. Enter a default value for attributes that are created from this attribute definition. This can be a commonly used value for the attribute or a placeholder such as "?" or "#".
6. Click "OK" to exit the dialog.
7. Use the mouse to specify the location of the attribute definition or enter a coordinate in the command line.
8. Right click or press Escape to stop the tool.

Synchronize Attributes

Toolbar / Icon:
 

Add-on: QCAD Professional
Menu: `Block > Attributes > Synchronize Attributes`
Shortcut: `U, Y`
Commands: `attributesynchronization | attsync | uy`

Description

This tool synchronizes attributes in block references with the attribute definitions in the appropriate block definition.
Attributes for which no definition is available are deleted.
For new attribute definitions, a new attribute is created with the attribute definition default value as value.
The value of existing attributes is not changed. The text style, position, alignment and angle are reset to those of the attribute definition.
If nothing is selected, this tool synchronizes all attributes of all block references.
If there is a selection, only those block references that are selected are synchronized with their block definitions.

Window Tools

Close All

Toolbar / Icon:

Menu: `Window > Close All`

Shortcut: `Ctrl+Shift+W (Mac: ⇧⌘W)`

Description

This tool closes all open drawings.

Next

Toolbar / Icon:

Menu: `Window > Next`

Shortcuts: `Ctrl+Right (Mac: ⌘Right) | Ctrl+PgUp (Mac: ⌘PgUp)`

Description

This tool activates the next open drawing window / tab.

Previous

Toolbar / Icon:

Menu: `Window > Previous`

Shortcuts: `Ctrl+Left (Mac: ⌘Left) | Ctrl+PgDown (Mac: ⌘PgDown)`

Description

This tool activates the previous open drawing window / tab.

Full Screen

Toolbar / Icon:

Add-on: QCAD Professional

Menu: `Window > Full Screen`

Shortcut: `Ctrl+Shift+F (Mac: ⇧⌘F)`

Commands: `fullscreen`

Description

Switches the application window to full screen mode or back to normal mode. The application menu can be accessed by clicking at the very top of the screen.

CAM Tools

Toolbar / Icon:

Commands: `cammenu`

CAM Configuration...

Toolbar / Icon:


Add-on: QCAD/CAM
Menu: `CAM > CAM Configuration...`
Shortcut: `K, K`
Commands: `camconfig | kk`

Description

Use this function to display a dialog with various preferences for the current configuration. These preferences apply to all tools and all toolpaths.

CAM Export

Toolbar / Icon:


Add-on: QCAD/CAM
Menu: `CAM > CAM Export`
Shortcut: `K, X`
Commands: `camexport | kx`

Description

This tool exports all available toolpaths in the current drawing to a CAM output file using the currently selected configuration (combo box at the top left).
If the drawing has been previously exported using the same configuration, the export starts immediately. Otherwise, a file dialog is shown to choose a file name and location.

CAM Legacy Export

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > CAM Legacy Export`

Commands: `camexportv1`

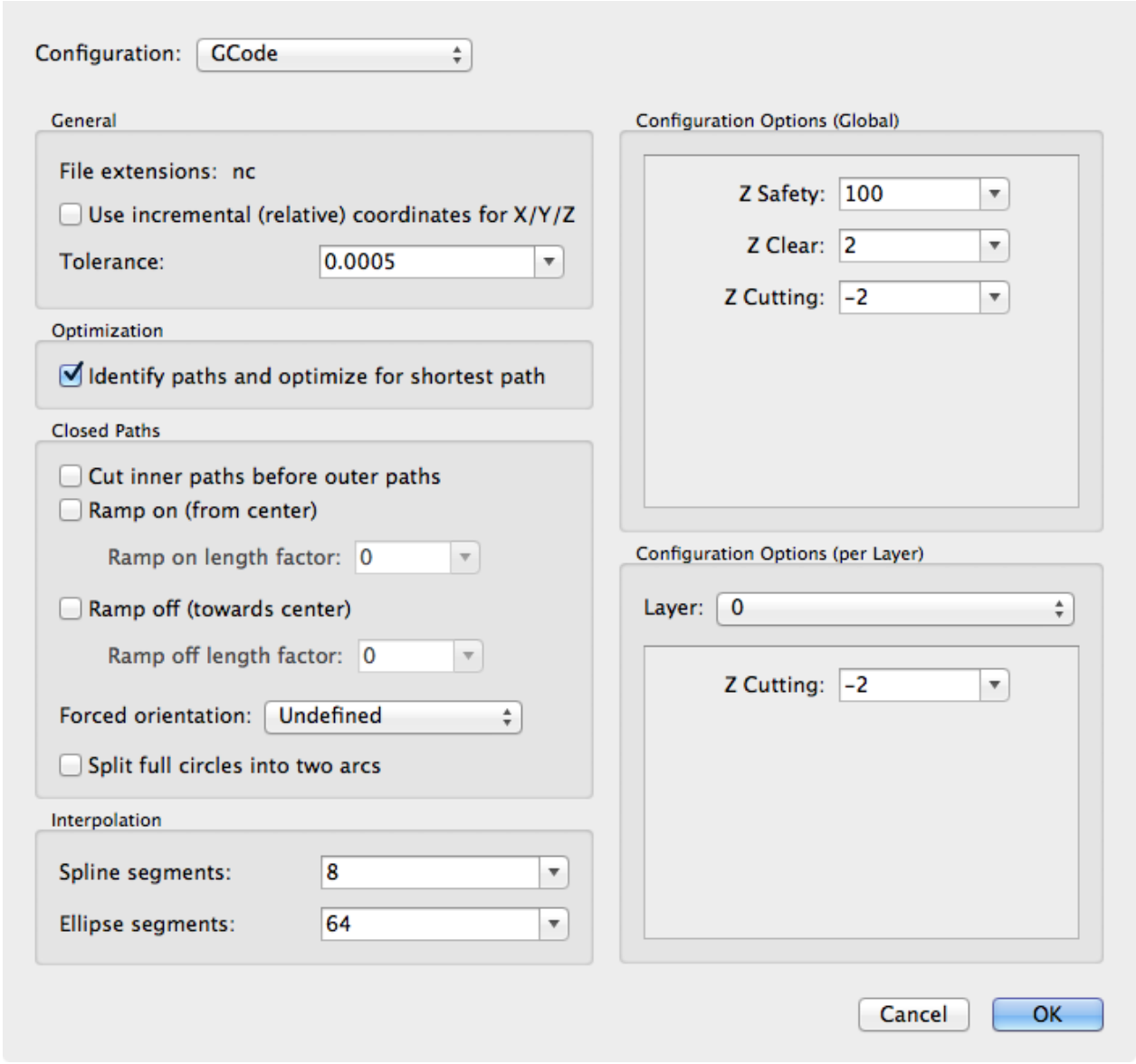
Description

QCAD/CAM can export CAD drawings to various formats used in CAM (computer-aided manufacturing). These formats can be dialects of G-Code or other, completely different formats. Each supported CAM format is defined in a configuration file which is an ECMAScript (JavaScript) file that implements the format details. All available configurations are stored inside the directory scripts/Cam/CamConfigurations. New format implementations can be added to the same location.
More information about how to create new configurations can be found on our web site at:
http://www.qcad.org/qcad-cam-configuration

Usage

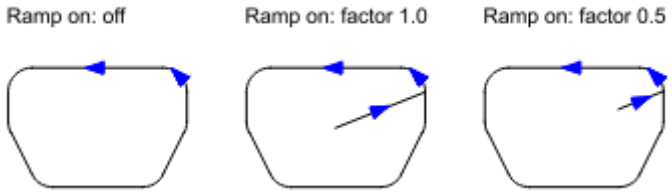
1. Load the drawing (or Gerber file) you want to export to a CAM format.
2. Start the tool by clicking its tool button or choosing its menu (see above).

3. The CAM configuration dialog is shown:



4. Choose the file format you want to use. The name of the file format directly corresponds to the name of the configuration file stored on disk. In the dialog shown above, the configuration *GCode* is chosen which corresponds to the file *scripts/Cam/CamConfigurations/GCode.js*. The various CAM export parameters displayed in the CAM configuration dialog are automatically initialized to the default parameters used by the chosen configuration.
5. In the *General* section at the top of the dialog, you can choose if you want to use relative or absolute coordinates and the tolerance used to identify connected paths. The tolerance is the maximum gap between two end points to be treated as connected.
6. In the *Closed Path* section, you can activate some options that only apply to closed paths. A closed path or contour consists of a number of entities which are connected to each other. The last entity connects to the first entity of the path. Full circles are also treated as closed paths.
    - The *Ramp on* option adds an additional linear move before the first entity of a closed path from the direction of the path center. The figure below illustrates the cutting path and order without a ramp on, with a ramp on with factor 1.0 (from

center) and with a ramp on with factor 0.5.



- The same option is also available for *Ramp off,*, an additional linear move at the end of the closed path, towards the center of the path.
- With option *Forced orientation*, it is possible to force all contours to be cut in the same orientation (clockwise or counterclockwise). This is usually required if the chosen CAM configuration uses the same tool radius compensation mechanism (e.g. G40, G41, G42) for all closed contours.
- Option *Split full circles in two arcs* cuts all full circles as two halves. This is required for some controllers which cannot handle circles with an angle of over 180 degrees.

7. Click *OK* to start the conversion. Depending on the size and complexity of the drawing, the conversion might take several minutes to complete. You can follow its progress in the command line history output.
8. QCAD/CAM opens a new drawing with the conversion result upon complete conversion. The drawing contains two layers, one for regular linear and circular moves (shown in black or white) and one for rapid moves (shown in blue). You can also use the CAM simulation tool panel to simulate the output.

Note: the unit used for coordinates in the CAM output is defined by the CAM configuration that is used. The coordinates in the drawing that is used as source are automatically converted to the unit defined by the CAM configuration.

G-Code Import...

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > G-Code Import...`
Commands: `camimport`

Description

This tool imports the geometry from an existing G-code file. The expected syntax of the G-code can be configured in the application preferences.

Gerber Import...

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > Gerber Import...`

Commands: `gbximport`

Description

The QCAD/CAM module can import Gerber files of format RS-274X. Only paths and aperture flashes are imported.

Usage

1. Start the tool by choosing the menu or tool button shown above.
2. Select the Gerber file you want to import. Gerber files usually have the file ending .gbx.
3. The Gerber file is loaded and shown as a new drawing.

Note: The unit of the Gerber file is not converted during the import. If the Gerber file is stored in Inches, the file is imported into a new drawing in Inches. If the Gerber file uses Millimeters, a drawing in Millimeters is created.

Display Toolpaths

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Display Toolpaths`
Shortcut: `K, V`
Commands: `togglecam | kv`

Description

This tool toggles the display of all CAM toolpaths. If toolpaths are on, all toolpaths are shown in the drawing area. This can be distracting when changing the geometry of your drawing.

Add Tool

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Add Tool`
Shortcut: `K, A`
Commands: `camaddtool | addtool | ka`

Description

This function adds a tool for CAM processing to the current drawing. The new tool can then be used to create CAM toolpaths. The main property of a tool is its diameter. The diameter is used to compute the desired toolpath offset (cutter radius compensation).

Usage

1. Start this tool.
2. A dialog for entering the tool parameters is displayed.
    - The tool name is usually a number that is used in the G-code output for tool changes (e.g. "T1").

- The tool diameter is required for the calculation of the tool radius compensation and for the simulation.

Edit Tool

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Edit Tool`
Shortcut: `K, E`
Commands: `camedittool | edittool | ke`

Description

This function can be used to edit an existing tool. The same dialog with tool parameters is shown as when a new tool is added.

Remove Tool

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Remove Tool`
Shortcut: `K, R`
Commands: `camdeletetool | tooldelete | kr`

Description

This function removes the selected tool from the tool list.

Export Tools

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Export Tools`
Shortcut: `K, O`
Commands: `camexporttools | exporttools | ko`

Description

This function exports all tools of the selected configuration of the current drawing into a separate file. The tools can then for example be imported into another, existing drawing.

Import Tools

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Import Tools`
Shortcut: `K, I`
Commands: `camimporttools | importtools | ki`

Description

This function imports tools from an external file. This file may have been created previously via CAM < Export Tools or can be any other drawing with tools.

Add Profile Toolpath

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Add Profile Toolpath`
Shortcut: `K, P`
Commands: `camaddprofiletoolpath | addprofiletoolpath | profiletoolpathadd | kp`

Description

This tool creates a new profile tool path, based on the current selection. A profile tool path is tool path that follows a profile with the tool center either on the geometry or using a tool radius correction to the left or right side of the geometry.

Usage

1. Select the profile(s) you want to use as a basis for the profile toolpath. You can select one or multiple open or closed profiles. A profile typically consists of a polyline or a selection of connected lines, arcs, ellipse arcs and splines. If you need full control of the start point and direction of a profile toolpath, make sure that all your profiles consist of polylines with the desired start point and direction.
2. Start this tool.
3. The profile toolpath dialog is shown with the various parameters that define a toolpath.
    - Name: Name of the toolpath, unique per configuration.
    - Tool: The tool number of the tool used to process the toolpath. You can also add a tool by clicking on the plus symbol.
    - Feed Rate: The feed rate to use for this particular tool path. This defaults to the feed rate of the chosen tool but can be adjusted for each toolpath individually if desired.
    - Plunge Rate: The feed rate used for downward moves. This defaults to the plunge rate of the chosen tool but can be adjusted for each toolpath individually if desired.

    Cutting Depths:
    - Safe Z (a): The Z level used to approach the toolpath at rapid move (clearance, approaching level).
    - Start Depth (b): The Z level at which the material begins (typically 0).
    - Cut Depth (c): The final cutting depth, measured from the start depth. Note that cut depth is a distance (positive), not a Z level.

- Passes: The number of passes to use to reach the final cutting depth or 1 to cut to the final depth in a single pass.

Side / Direction:

- Side: The side for the tool radius compensation or *On* to cut with the tool center on the profile.
- Direction: Cutting direction climbing (left of profile) or conventional (right of profile).

Leads:

- Lead In Type: Shape of lead in to use. Leads are an additional moves inserted before / after the profile to approach or leave the profile. If tool radius compensation is used, leads must be used, so the tool radius compensation can be switched on / off.
- Lead In Type: Shape of lead out to use.
- Lead In Length / Radius: The size of the lead in shape to use.
- Lead Out Length / Radius: The size of the lead out shape to use.
- Overcut: Overlapping element used at the end of a profile to cut slightly further than required. This can lead to a smoother end result.

4. Click OK to create the profile toolpath. QCAD/CAM generates the toolpath and displays the path of the tool center in the graphics view as a dashed line.

Add Drill Toolpath

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Add Drill Toolpath`
Shortcut: `K, L`
Commands: `camadddrilltoolpath | adddrilltoolpath | drilltoolpathadd | kl`

Description

This tool creates a new drill tool path, based on the current selection. A drill tool path is a tool path for drilled holes. The center of the hole can be defined by a point entity or the center of a circle entity.

Usage

1. Select the points or circles you want to use as a basis for the drill toolpath. You can select one or multiple points or circles.
2. Start this tool.
3. The drill toolpath dialog is shown with the various parameters that define a drill toolpath.
    - Name: Name of the toolpath, unique per configuration.
    - Tool: The tool number of the tool used to process the toolpath. You can also add a tool by clicking on the plus symbol.
    - Plunge Rate: The feed rate used for downward moves. This defaults to the plunge rate of the chosen tool but can be adjusted for each toolpath individually if desired. For upward moves, the feed rate of the tool is used.

    Cutting Depths:
    - Safe Z (a): The Z level used to approach the toolpath at rapid move (clearance, approaching level).
    - Start Depth (b): The Z level at which the material begins (typically 0).
    - Cut Depth (c): The final cutting depth, measured from the start depth. Note that cut depth is a distance (positive), not a Z level.
    - Passes: The number of passes to use to reach the final cutting depth or 1 to cut to the final depth in a single pass.
4. Click OK to create the drill toolpath. QCAD/CAM generates the toolpath and displays the path of the tool center in the graphics view as a dashed line.

Edit Toolpath

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Edit Toolpath`
Shortcut: `K, D`
Commands: `camedittoolpath | edittoolpath | toolpathedit | kd`

Description

Use this function to edit the parameters of existing toolpaths.

Remove Toolpath

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Remove Toolpath`
Shortcut: `K, M`
Commands: `camremoveprofiletoolpath | removeprofiletoolpath | toolpathremove | km`

Description

This tool removes the currently selected toolpath from the toolpath list.

Move Toolpath Up

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Move Toolpath Up`
Commands: `toolpathmoveup`

Description

This tool moves a toolpath further up in the order of processing. Toolpaths are exported in the exact order as shown in the toolpath list.

Move Toolpath Down

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > Move Toolpath Down`

Commands: `toolpathmovedown`

Description

This tool moves a toolpath further down in the order of processing. Toolpaths are exported in the exact order as shown in the toolpath list.

Regenerate Toolpath

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > Regenerate Toolpath`

Commands: `toolpathregen`

Description

This tool regenerates the toolpath that is currently selected in the toolpath list based on its current parameters. Use this after changing the parameters or geometry of a toolpath.

Regenerate Toolpath from Tool

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > Regenerate Toolpath from Tool`

Commands: `toolpathregentool`

Description

Regenerate All Toolpaths

Toolbar / Icon:

Add-on: QCAD/CAM

Menu: `CAM > Regenerate All Toolpaths`

Commands: `toolpathregenall`

Description

This tool regenerates all toolpaths based on their current parameters. Use this after changing configuration global parameters or after updating the geometry of your drawing.

Update Toolpath from Selection

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Update Toolpath from Selection`
Commands: `toolpathupdatefromselection`

Description

This tool updates the currently selected toolpath to use the selected geometry as a new basis. Use this after deleting (part of) the geometry of a toolpath to regenerate the toolpath based on a different or updated geometry.

Add Tab

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Add Tab`
Shortcut: `K, T`
Commands: `camaddtab | kt`

Description

This function adds a tab to the current drawing. Tabs are small connections between a cutout shape and its surrounding material. Tabs can be used to hold a piece in place while cutting it out of a sheet material. This prevents the piece from moving while the tool is still cutting. Tabs are typically very small, so the piece can be easily broken out of the sheet material once the toolpath is complete. To securely hold a piece in place, three or more tabs are typically used.

Usage

1. Start this tool.

2. Enter the desired length and thickness of the tab.
3. Use the mouse to position the tab in your drawing. Tabs are usually positioned on the original contour (not on the offset contour). However, they also affect the toolpath if a radius compensation is used.

Show CAM Tool List

Toolbar / Icon:


Add-on: QCAD/CAM
Menu: `CAM > Show CAM Tool List`
Shortcut: `G, O`
Commands: `camtoollist | go`

Description

The tool list shows all tool that have been created for the current drawing and chosen configuration. You can create as many tools as required, for each configuration. When changing the configuration at the top left, the tool list is automatically updated to only show the tools available for that configuration.

Show CAM Toolpath List

Toolbar / Icon:


Add-on: QCAD/CAM
Menu: `CAM > Show CAM Toolpath List`
Shortcut: `G, A`
Commands: `camtoolpathlist | ga`

Description

The toolpath list shows all toolpath that have been created for the current drawing and chosen configuration. Toolpaths are shown in the same order as they will be processed in.

Show 3D Simulation View

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Show 3D Simulation View`
Shortcut: `G, 3`
Commands: `simulation3d | g3`

Description

The 3D view can be used to observe Z movements during CAM simulation.

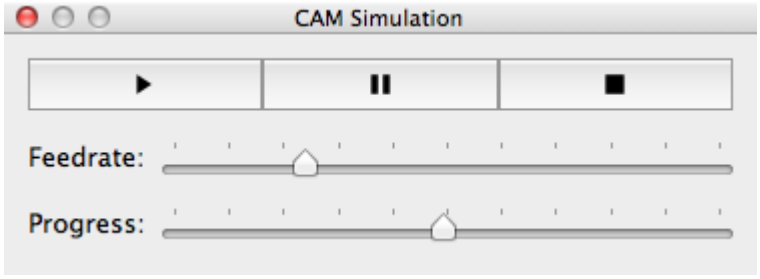
Show Simulation Panel

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Show Simulation Panel`
Shortcut: `G, I`
Commands: `camsimulation | gi`

Description

With the CAM Simulation widget, the order in which entities are processed can be visualized in slow motion.



The play button starts the simulation. The current position of the tool is shown as yellow circle in the drawing area.
The pause button can be used to pause the simulation. Hit the play button to continue the simulation again.
The stop button stops the simulation.
The feedrate slider can be used to adjust the speed of the simulation while the simulation is in progress.
With the progress slider, the progress of the simulation can be adjusted directly to the desired position and continued from there.

Nesting

Toolbar / Icon:

Add-on: QCAD/CAM
Menu: `CAM > Nesting`
Commands: `nesting`

Description

This tool automatically arranges parts on one or multiple chosen sheets optimizing the space used on the sheet.

Usage

1. Optional: Define the nesting quantity of the parts to nest. To do this, select all entities of a part, choose *CAM > Set Nesting Quantity* and enter the desired quantity.
2. Select the parts to nest. Parts can consist of lines, arcs, circles, ellipses, splines or polylines. Parts may contain islands. Each part is expected to be surrounded by a closed path of connected entities.
3. Start this tool.

4. Adjust the nesting parameters in the options toolbar at the top if desired:
    - Rotations: The number of rotations to try when inserting parts into the sheets. 4 means that each part can be rotated 0 degrees, 90 degrees, 180 degrees or 270 degrees.
    - Sheet Margin: Minimum distance between sheets and parts
    - Margin: Minimum distance between parts
    - Tolerance: Tolerance when converting arcs to polylines. A larger tolerance will not lead to overlapping parts but might lead to larger margins between parts.
    - Alignment: Parts are arranged with the chosen alignment in mind. An alignment of "Left - Bottom" means that parts are shuffled to the left (primary alignment) and towards the bottom (secondary alignment).
    - Strategy: Choose the desired nesting strategy. The "Hull" strategy tries to minimize the total surface area occupied by the parts.
5. Click the sheet to be used to arrange the parts in. This can be any entity that is part of a closed shape. Multiple sheets may be chosen. Only one entity of each sheet has to be clicked. Multiple sheets can be chosen. If a sheet lies within another sheet, it is considered to be an island. Islands are not used to place parts.
6. Right-click when all sheets have been chosen.
7. QCAD/CAM starts the nesting process and lists possible solutions as they are found.
8. As soon as the first solutions are found and listed, you can click a solution in the list to preview it. Solutions are listed in order of quality (from best to worst).
9. The nesting process can be stopped when a good solution is identified using the "Stop" button.
10. QCAD/CAM now shows all solutions again and lets you choose the preferred solution.
11. Click "Apply Solution" to apply the chosen solution.

Preferences

Edit > Application Preferences > CAM > Nesting

- *Number of threads:* Number of CPU threads to use for nesting. Default is "Auto" which means the number of threads depends on the number of CPU cores available on the system.
- *Maximum number of iterations:* Number of iterations for which threads are created to compute new, better solutions. This can be set to a lower number to cause QCAD/CAM to stop even when the user never clicks the "Stop" button.
- *Weighting of primary alignment:* The factor by how much the primary alignment is weighted more than the secondary alignment. Set to 1 to weigh both alignments equally. Set to a higher number to squeeze in one direction only.
- *Place small parts into holes of other parts:* If activated, allows placing of smaller parts inside holes of larger parts.
- Align part angles to axis: Automatically rotate parts to be aligned to X / Y axis.

- *Disable AppNap if QCAD runs in background mode:* Keep QCAD/CAM threads active at normal priority when QCAD/CAM runs in the background (i.e. QCAD/CAM is not the top window). This preference is only available on macOS.

Set Nesting Quantity

Toolbar / Icon:

Add-on: **QCAD/CAM**
Menu: `CAM > Set Nesting Quantity`
Commands: `nestingquantity`

Description

This tool allows you to assign a quantity to a part to be nested. The part is then inserted several times into the sheet during nesting.