

QCad

User Reference Manual

Author: Andrew Mustun
Copyright 2004 RibbonSoft, Inc.
All Rights Reserved.
Release: April 2004

Contents

1 License	7
Copyright	7
Trademarks	7
The GNU General Public License	7
2 Preface	8
Scope of This Manual	8
CAD - For the Uninitiated	8
Is QCad for me?	8
Who Is Behind QCad?	9
How can I get QCad?	9
QCad and Qt	9
3 Basic CAD Concepts	10
Entities	10
Attributes	10
Layers	10
Blocks	11
Drafting in CAD	11
Co-ordinate Systems	12
Origin	12
Cartesian Co-ordinates	12
Polar Co-ordinates	13
Object and Grid Snapping	14
Basic Editing	14
Entity Selection	14
Deletion	14
Modifications	14
Viewing in CAD	15
4 Starting a QCad Session	16
Windows Operating Systems	16
Linux / Unix Operating Systems	16
Mac OS X Operating System	16

5 QCad's User Interface	17
Main Application Window	17
Menu and CAD Toolbar	17
Layer- and Block List	18
Status Bar	18
Command Widget	18
Option Toolbar	18
Executing Commands	19
Modes	20
Command Mode	20
Normal Mode	20
6 Command Line	21
A Word About Keyboard Focus	21
7 File Handling	22
Create New Drawings	22
Opening Drawings	22
Saving Drawings	23
Exporting Drawings as Bitmaps	23
Closing Drawings	23
8 Printing	24
Print Preview	24
Print	25
Printing to Files	25
9 Basic Editing Commands	26
Undo	26
Redo	27
Copy	28
Cut	28
Paste	29
10 Viewing	30
Scrolling	30
Toggle Grid	31
Toggle Draft Mode	31
Redrawing	31
Zooming In / Out	32
Auto Zoom	33
Window Zoom	33
Panning	34
Previous View	34

11 Layers	35
Changing the Visibility of a Layer	36
Changing the Visibility of all Layers	36
Adding New Layers	36
Deleting Layers	37
Editing Layer Attributes	37
12 Blocks	38
Changing the Visibility of a Block	38
Changing the Visibility of all Blocks	39
Adding New Blocks	39
Removing Blocks	40
Renaming Blocks	40
Editing Blocks	40
Inserting Blocks	41
Creating Blocks from Existing Entities	42
13 Object Snap	43
Free Positioning	44
Snap to Grid	44
Snap to Endpoints	45
Snap to Points on Entities	46
Snap to Centers	47
Snap to Middle Points	48
Snap to Distance Points	49
Snap to Intersections	50
Snap to Intersections Manually	51
Snap Restrictions	52
Restrict Nothing	53
Restrict Orthogonally	53
Restrict Horizontally	53
Restrict Vertically	54
Relative Zero Point	55
Moving the Relative Zero Point	55
(Un-)locking the Relative Zero Point	55
14 Entity Drawing Commands	56
Points	56
Single Points	56
Lines	57
Sequence Of Lines	57
Line With Given Angle	58
Horizontal / Vertical Line	58
Rectangle	59
Parallel	59
Bisector	60

Tangent From Point to Arc / Circle	61
Tangent From Arc to Arc / Circle to Circle	61
Line Orthogonal to Another Entity	62
Line with Relative Angle to Another Entity	62
Freehand Line	63
Polygon with Center and Corner	64
Polygon with Two Corners	64
Arcs	65
Arc with Center, Radius and Angles	65
Arc with 3 Points	66
Concentric Arc	66
Circles	67
Circle with Center and Point	67
Circle with Center and Radius	67
Circle with 2 Points	68
Circle with 3 Points	68
Concentric Circle	69
Ellipses	70
Full Ellipse	70
Ellipse Arc	71
Texts	72
Advanced Text Commands	74
Dimensions	75
Aligned Dimension	76
Linear Dimensions	77
Horizontal / Vertical Dimensions	78
Radial Dimensions	78
Diametric Dimensions	79
Angular Dimensions	80
Leader	81
Hatches and Solid Fills	82
Images	84
 15 Entity Selection	 85
Deselecting All	85
Selecting All	85
Selecting Single Entities	86
Selecting Contours	86
Window Deselection	87
Window Selection	87
Inverting Selection	88
Deselect Intersected Entities	88
Select Intersected Entities	89
 16 Modification	 90
Move	91
Rotate	92
Scale	93
Mirror	94

Move and Rotate	95
Rotate and Counter-rotate (Rotate 2)	97
Trim / Extend	99
Trim Two Entities	100
Lengthen	101
Bevel	102
Round	103
Divide	104
Stretch	105
Properties	105
Attributes	106
Deleting Entities	107
Exploding Groups	108
17 Getting Measurements	109
Distance (Point, Point)	109
Distance (Entity, Point)	110
Angle	110
18 Application Preferences	111
Appearance	111
Language	111
Graphic View	111
Paths	112
19 Drawing Preferences	113
Paper	113
Units	113
Dimensions	114
20 Appendix	116
Hotkeys	116
Command Line	116
File Handling	116
Basic Editing	117
View	117
Layer Handling	117
Snapping	117
Construction	118
Dimensioning	118
Entity Selection	118
Modification	118
Fonts	119
Patterns	124
Mathematical Expressions	132
Supported Constants	132
Supported Functions	132

Drawing Units	135
Supported Length Units	135
Display Formats for Lengths	135
Supported Angle Units	136
Display Formats for Angles	136
Supported Paper Formats	137
Metric (ISO)	137
Other Formats	137
Migration from QCad 1	138
DXF Compatibility	138
Layer Handling	138
Bibliography	140
Index	141

License

Copyright

Copyright © 1999-2004 by RibbonSoft, Andrew Mustun.
Published 2004
Switzerland

Document ID: \$Date: 2004/04/12 12:26:08 \$
Release: April 2004

Trademarks

Intel is a registered trademark and 80286, 80386, 286, 386, 486, Pentium and Pentium Pro are trademarks of Intel Corp.
Linux is a registered trademark of Linus Torvalds.
TrueType is a registered trademark of Apple Computer, Inc.
Microsoft, and MS are registered trademarks and Windows, Windows 95, Windows 98 and Windows NT are trademarks of Microsoft Corp.
IBM is a registered trademark of International Business Machines Corp.
PostScript is a registered trademark of Adobe Systems. Inc.
Autodesk, DXF, AutoCAD are either registered trademarks or trademarks of Autodesk, Inc.
All other brand names, product names, or trademarks belong to their respective holders.

The GNU General Public License

The QCad Community edition for Linux, other Unix or X11 Systems and Mac OS X is released under the terms of the GNU General Public License (GPL), Version 2. QCad Profesional and all QCad versions for Windows are proprietary software. See <http://www.ribbonsoft.com> for details.

Preface

Scope of This Manual

This manual is designed to get you started using QCad and to help you using it more efficiently. Further it will give you an idea of the more advanced possibilities behind QCad. Chapter 3 will introduce you in the world of computer aided design (CAD). If you've already worked with a CAD program before, you might want to skip this chapter.

CAD - For the Uninitiated

If you take a look around you, most objects you will see once started off as a drawing in a CAD program. From your ball pen, your desk, furniture to your house and probably even your city - everything once had to be constructed either manually on paper or more likely with a CAD system.

CAD stands for Computer-Aided Design. It's essential to realize that CAD applications differ from painting or drawing programs. Drawings made in CAD applications represent real-world objects in their exact original measures. The most important factors are accuracy and to show as many details as necessary to produce the object. If you're used to drawing programs, you will most likely find it inconvenient that there's usually no or only limited drag-and-drop functionality in CAD systems. This is part of the idea, that everything is done accurately. The drawing doesn't only have to look accurate on the screen - it has to be accurate as a model. Moving an entity around means moving it from an exactly defined location to another exactly defined position. However, once you got used to this approach, you will never want to do technical drawings in an ordinary drawing program anymore.

Is QCad for me?

There are many different CAD systems out there and depending on what you intend to do, QCad might or might not fit your bill. If you are looking for a 3D modeling CAD application, you'll have to look somewhere else. QCad is a 2D CAD program. That means that everything is projected onto a plane. This doesn't mean that it's only suitable to represent 2D objects though. A couple of 2D drawings which represent an object from different perspectives (e.g. from the side, the top and from the front) provide usually enough information to fully define the object and all its dimensions.

Some of the main reasons why you might want to use QCad are its simplicity, the user friendly interface, its many features or simply the fact that it works on your favorite platform - be it Linux, an other Unix system, Windows or Mac OS X. Last but not least QCad is part of the open source movement. This means that if you're a programmer you can add your own functionality.

QCad has an estimated user base of over 100'000 people worldwide. Its users range from industrial companies in the fields of mechanical and electronic engineering to private users, teachers and students. There are many other capable CAD systems out there and most of them offer much more than QCad does. But most of them are bound to one single platform or are simply not affordable for many potential users. That's where QCad kicks in. QCad targets the hobbyists, occasional CAD users and people who are not CAD professionals but still need to draw plans once in a while. QCad is the CAD for the rest of us.

Who Is Behind QCad?

QCad is a product of RibbonSoft, a software engineering company based in Switzerland. However, many other people have contributed to QCad and it wouldn't be what it is today without the help of translators, developers and the feedback of QCad users. Please refer also to the company homepage of RibbonSoft for more information: www.ribbonsoft.com .

How can I get QCad?

You can get the latest version of QCad from www.qcad.org . On this web site you can also find more documentation and other resources for QCad.

QCad and Qt

Qt is the cross platform C++ application framework that QCad is based on. Much of the work involved in QCad is done by Trolltech (www.trolltech.com), the guys who develop Qt.

Basic CAD Concepts

This chapter is intended to introduce you into the basic concepts of a CAD system. If you're used to working with CAD system you might want to skip this chapter. Please note that the concepts mentioned in this chapter are all later described in detail.

Entities

Entities are graphical objects in a CAD system. Typical entities which are supported by most CAD systems are: points, lines and circular and elliptical arcs. More complex and CAD specific entities include polylines, texts, dimensioning, hatches and splines.

Attributes

Every entity has certain attributes such as its color, line type and line width.

Layers

A basic concept of computer aided drafting is the use of layers to organize a drawing. Every entity of a drawing is on exactly one layer and a layer can contain lots of entities. Typically entities with a common 'function' or common attributes are put on the same layer. E.g. you might want to put all axes in a drawing on a layer named 'axes' (see Figure 1). Layers can have attributes (color, line width, line style). Each entity can have its own attributes or have its attributes defined by the layer it is placed on. In the latter case you can change for example the color of all 'axes' entities by changing the color of the layer 'axes'.

In manual drafting, a similar approach was used. Different building systems, such as wiring and air conditioning were often drawn on separate transparent sheet of paper. These sheets were then overlaid on one another to produce final drawings.

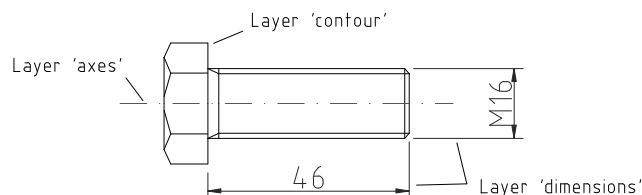


Figure 1: Example use of layers.

Blocks

A Block is a group of entities. Blocks can be inserted into the same graphic more than once with different Attributes and different locations, scaling factors and rotation angles (see Figure 2). Such an instance of a block is usually called an Insert. Inserts have attributes just like entities and layers. An Entity that is part of an Insert can have its own attributes or share the attributes of the Insert. Once created, Inserts are still linked to the Block they instantiate. The power of inserts is, that you can modify the Block once and all Inserts will be updated accordingly.

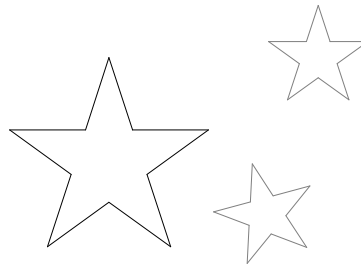


Figure 2: Three instances (Inserts) of a Block with different colors, angles and scalings.

Drafting in CAD

In many ways, CAD is similar to traditional drafting. When drawing a plan or a view of an object on a paper, you would use tools such as a ruler to draw lines. In CAD systems there is a variety of tools available to achieve the same goal. The big advantage of a CAD system is the fact that you can change every entity of your drawing easily after you've created it. This might be the most difficult thing to learn when moving from paper to CAD. When working with a CAD system you will very often create lines that won't be on the final printout or which don't have the correct length and need to be trimmed later (see Figure 3). A common mistake of CAD beginners is wanting to create the final drawing right away.

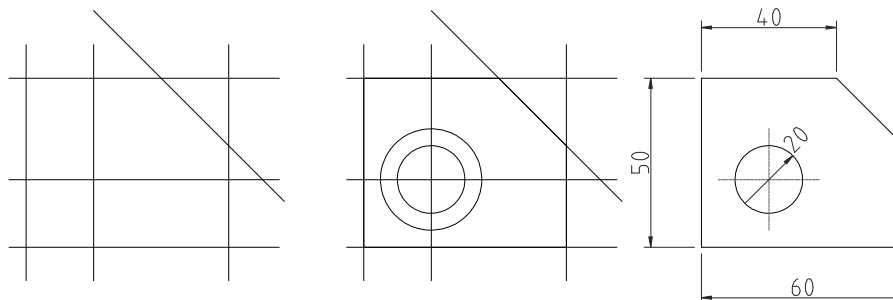


Figure 3: Three possible steps to quickly create a drawing using supportive constructions and the ability of a CAD system to change existing entities.

Co-ordinate Systems

A good understanding of how co-ordinates work is absolutely crucial if you are to make the best use of any CAD program. If you're not familiar with co-ordinates it's highly recommended that you take some time to familiarize yourself with the basic concepts.

Origin

The origin of the drawing is the point where the X and Y axis cross each other. It's the absolute zero of the drawing. In addition to the origin there's a relative zero point in QCad. This is a helpful spot that changes it's position depending on the current user interactions. The relative zero point can also be moved around by the user.

Cartesian Co-ordinates

(named after the French philosopher René Descartes 1596-1650)

The Cartesian co-ordinate system is the standard co-ordinate system where the position of a point can be described by its distance from two axes, X and Y. Cartesian co-ordinates in QCad are written in the format:

`x-ordinate,y-ordinate`

where x-ordinate can be a number such as 2.5 or a mathematical expression like $\cos(30)/2$ (see [Mathematical Expressions](#)).

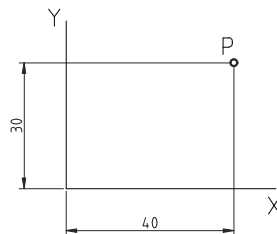


Figure 4: Absolute Cartesian Co-ordinates 40, 30 .

Cartesian co-ordinates can also originate in a position different from the origin. In that case we talk about 'relative co-ordinates'. In QCad, relative Cartesian co-ordinates are written in the format:

`@x-ordinate,y-ordinate`

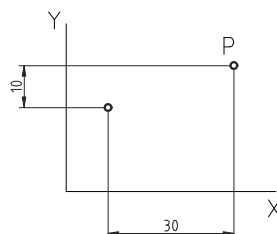


Figure 5: Relative Cartesian Co-ordinates @ 30, 10 .

Polar Co-ordinates

Polar co-ordinates use a distance and an angle to describe the position of a point.

In QCad, polar co-ordinates are noted in the format:

`distance<angle`

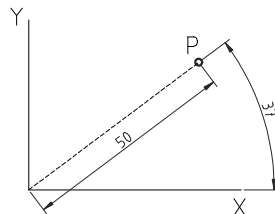


Figure 6: Absolute Polar Co-ordinates 50<37 .

Just like Cartesian co-ordinates, polar co-ordinates can also originate in a position different from the origin. In QCad, relative polar co-ordinates are written in the format:

`@distance<angle`

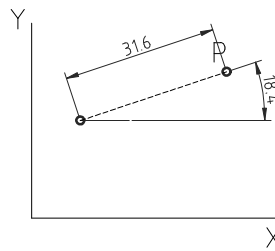


Figure 7: Relative Polar Co-ordinates @31.6<18.4 .

Object and Grid Snapping

Whenever you need to specify a co-ordinate in QCad, you can use the snap feature which allows you to precisely select grid points or significant points on existing objects: endpoints or midpoints of lines, etc.

The chapter "[Object Snap](#)" gives you an overview over all snap tools.

Basic Editing

Entities can be inserted, selected, and, once selected, can be deleted, transformed or duplicated. To insert an entity means to draw it by selecting the appropriate drawing tool, such as line, arc, etc., and by locating points that define the object to be drawn, such as the endpoints of a line.

The chapter "[Entity Drawing Commands](#)" lists all tools for adding new entities to a drawing.

Entity Selection

An entity must be selected before it can be deleted, duplicated, or transformed. Entity selection is one of the most basic of CAD operations. QCad offers a wide variety of selection tools to quickly select groups of entities, entities within a range, connected entities, etc.

The chapter "[Entity Selection](#)" introduces you into the selection tools of QCad.

Deletion

Deleting an entity means to remove it from the drawing.

Modifications

Basic modifications of a CAD systems include translation, rotation, reflection, and scaling. As well as these operations which don't alter the characteristic geometry of selected entities, there are some that do. You can break, trim, extend or stretch existing entities.

The chapter "[Modification](#)" lists and describes all tools for modifying and deleting entities.

Viewing in CAD

Unlike in manual drafting, there is no need in CAD to determine in advance the sheet size and drawing scale. There is **no** drawing scale: all sizes and distances are specified using their full-scale values. A 10 meter object is drawn as a 10 meter object. Only at the printing stage, the drawing scale needs to be determined based on sheet size and drawing size.

On the screen, the user can adjust the currently visible area of the drawing by zooming in to view more detail or zooming out to view a wider extent. Another important viewing operation in CAD is panning. To see another portion of the drawing without changing the display scale, a user pans to it by "moving" a rectangular display window until it's over the desired spot.

Chapter "Viewing" is all about zooming and viewing functions.

Starting a QCad Session

Windows Operating Systems

Start QCad by selecting it from the Start menu or by double-clicking on it in Explorer.

Linux / Unix Operating Systems

If you installed QCad in directory `/opt/qcad`, you can launch QCad from the console by typing:

```
cd /opt/qcad  
./qcad &
```

Mac OS X Operating System

Double-click on the QCad application icon in Finder.

QCAD's User Interface

When you launch QCad, it comes up with the main application window of the graphical user interface. This window hosts the drawing documents.

Main Application Window

QCAD comes with a graphical user interface (GUI) which offers most of the common features you might already know from other applications. In this chapter we take a closer look at the individual components of the QCad main application window (Figure 8).

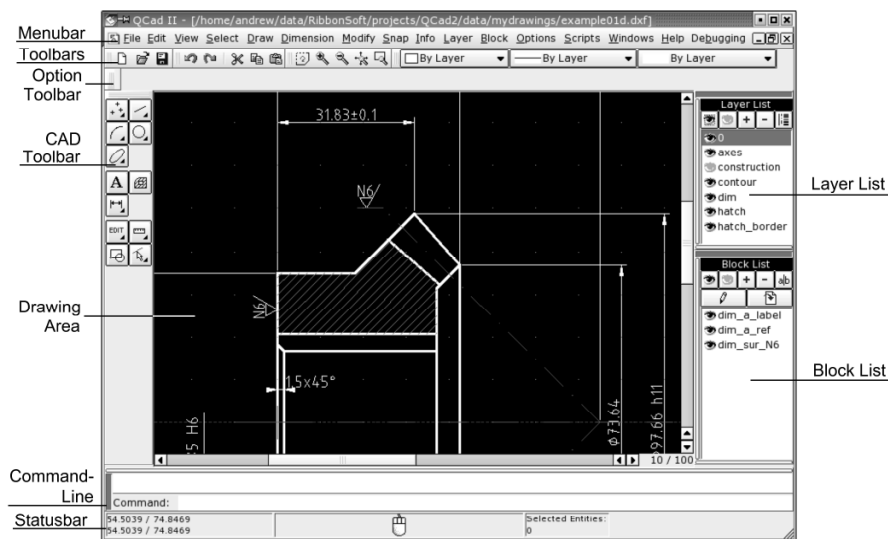


Figure 8: Main application window of QCad.

Menu and CAD Toolbar

Most functions of QCad can be accessed from its menu. For CAD related functions it might be more convenient to use the CAD Toolbar at the left. This Toolbar always shows the functions which currently make most sense to use. For example if you are about to set the startpoint of a line it shows the snapping functions which allow you to set the startpoint to an existing endpoint, a grid point or to use another positioning mode.

Layer- and Block List

To the right of the main window of QCad you can find the Layer- and the Block List. They show the layers and blocks of the currently active graphic document. You can switch the Layer- and Block Lists on and off using the menu `View - Views - Layer List` and `View - Views - Block List`.

Status Bar

The Status Bar at the bottom shows various information about the current state of QCad. At the left, the co-ordinate Widget displays the absolute and relative position of the mouse cursor as cartesian and polar co-ordinates. The Mouse Widget shows information about the current function of the left and right mouse buttons. The Selection Widget shows the number of entities that are currently selected. The Status Bar can be enabled / disabled using the menu `View - Views - Statusbar`.

Command Widget

Just above the status bar you can find the command line of QCad. It is used to type in commands and to notify you about warnings and errors. The command line can be enabled / disabled using the menu `View - Views - Command Line`.

Option Toolbar

The Option Toolbar is empty in Figure 8. Depending on the currently active tool, it shows the options that are available for the tool. If you prefer to change options and tool settings over the command line, you might want to hide this toolbar using the menu `View - Views - Option Toolbar`.

Executing Commands

Everything in QCad is dependent on a command being issued first. There are different ways to execute QCad commands:

- select a corresponding tool icon on one of the toolbars
- choose a menu item from one of the pull-down menus
- type the command at the command prompt

Please note that the menus and toolbars feature the most common commands.

Some actions require further input after launching. There are again three ways how those inputs can be requested by QCad:

- Dialog Windows. E.g. the text construction tool shows a dialog window where the user can choose the font and type in the string for the text entity.
- Option Toolbar. E.g. the dialog for drawing parallels requires the input of the distance from the original entity. It displays a text input box in the second toolbar at the top where the user can type the distance (see Figure 9).
- Command prompt. E.g. when drawing a sequence of lines you can close the sequence by typing 'close' at the command prompt.

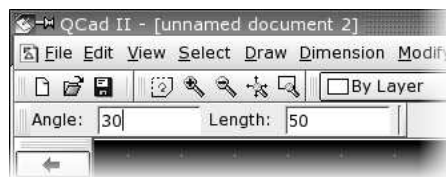


Figure 9: An optional toolbar may be shown for some tools.

Modes

QCAD operates in two modes: the **command mode** and the **normal mode**. In the command mode, it accepts commands over the command line. In normal mode all keyboard input goes to the application interface.

Command Mode

The command mode is indicated by a blue label at the left bottom corner of the command widget and a blinking cursor in the command line. In this mode, all keyboard input that is printable, goes into the command line. Use this mode to enter commands or co-ordinates. E.g. you can enter `line` to launch the line tool. If you press the Z key followed by the A key, the letters 'za' appear in the command line as opposed to triggering the auto zoom command.

Normal Mode

In this mode, QCAD behaves as any other standard application. All keyboard input is interpreted as hotkeys if specified or has no effect at all. If you press the `A` key in this mode, QCAD triggers the auto zoom command.

Command Line

QCAD can be controlled either using menus, the toolbar, hotkeys or the command line (Figure 10). The command line provides a very efficient way to work with a CAD system. Experienced users often key in commands with the left hand while operating the mouse with the right hand. However, if you're not (yet) used to it, it might only be in the way. In that case you can switch it off using the menu

View - Command Line

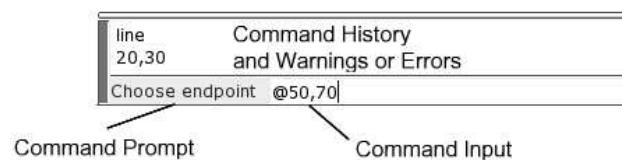


Figure 10: The Command Line of QCAD.

A Word About Keyboard Focus

Whenever the command line has the keyboard focus of the application, all keyboard input will be consumed by it. You can see that the command line has the keyboard focus the command prompt is blue and a text cursor (vertical line) blinks in the command input line.

Sometimes it's quicker to work with hotkeys than using the command line. In that case you will have to keep the focus away from the command line and give it to the main application window. The easiest way to achieve that is to press the Escape key. If there's text in the command line, you need to press Escape twice - the first will erase the text and the second will release the focus to the main application window.

Now you can for example press the Z-key followed by the A-key on your keyboard to trigger the auto zoom command.

To give the keyboard focus back to the command line, you can either click in it or press the Space bar.

File Handling

QCAD can load DXF files from older QCAD versions, AutoCAD® and other application that can produce DXF files. When saving files, QCAD follows the DXF 2000 standard [\[DXF\]](#).

Create New Drawings

Toolbar:



Menu:

File - New

Hotkey:

Ctrl-N

Create new drawings using this command. New drawings are completely empty (no entities, no layers, no blocks). Often, users create drawing templates with the layers and blocks they use most and sometimes already contain a drawing border. If you have such a template you might want to load that instead of creating a blank drawing from scratch.

Opening Drawings

Toolbar:



Menu:

File - Open

Hotkey:

Ctrl-O

Use this command to open DXF files. 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'.

Please note that QCad only loads those entities in the file which are also supported by the QCad engine. All other entities are ignored. If you import a DXF drawing created by another program, you should save that drawing under a new name before working with it. If you save such a drawing to the same file, non-supported entities will be lost.

Saving Drawings

Toolbar:



Menu:

File - Save

File - Save As

Hotkey:

Ctrl-S (save)

This command saves the current drawing to a file. Use the toolbar button or the menu `File - Save` to save the 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.

Exporting Drawings as Bitmaps

Menu:

File - Export..

This command exports the current drawing to a bitmap.

A dialog for selecting the output file is presented. After the file is selected, a second dialog asks the user for the bitmap size and the background color.

Closing Drawings

Menu:

File - Close

Hotkey:

Ctrl-W

This command closes the current drawing. You will be given the options to save the drawing before closing it, to save it to a new file or to discard all changes.

Printing

To print a drawing with QCad, you need to specify a scale factor for the printout. This can be 1:1 or any other desired scale. It's highly recommended to activate the print preview before actually printing a drawing. In the print preview you can adjust the scale factor and position the drawing on the paper.

Print Preview

Toolbar:



Menu:

File - Print Preview

Options Toolbar:



Opens a print preview window for the current drawing. The print preview shows how your drawing will look in the printout. Use the combo box in the options toolbar to adjust the scaling. The button in the middle toggles the black/white mode. When enabled all lines will appear black on the drawing rather than colored or gray. Use the button at the right to automatically center the drawing on the paper. You can also move the paper around by dragging it with the left mouse button.

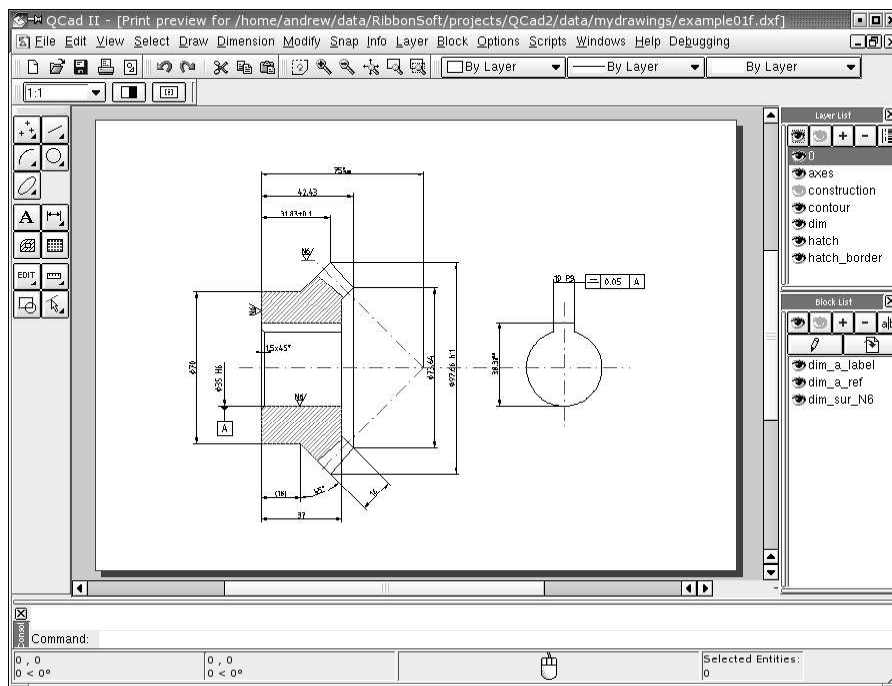


Figure 11: Print Preview.

Print

Toolbar:



Menu:

File - Print

Hotkey:

Ctrl-P

Use this command to print a drawing out of the print preview. A setup dialog is shown before anything is printed. Use this dialog to choose a printer. You can also still alter the paper format at this stage. This is not recommended however. The paper format can be changed in the drawing options before printing. If you change the paper format at this stage, the printout will not match the print preview.

Printing to Files

You can check "Print to file" in the printer setup dialog to create a file instead of actually printing on paper. Under Unix systems and Max OS X, this creates a Postscript file. Under Windows a raw binary print file is created (PRN). This file is printer dependent. To create PS files under Windows, please install a Postscript printer driver such as "APS-PS".

PS files can be easily converted to PDF files using a so-called distiller such as ps2pdf or Acrobat Distiller.

Basic Editing Commands

Undo

Toolbar:



Menu:

Edit - Undo

Hotkey:

oo, Ctrl-Z

Command:

undo, u

Description:

The Undo command takes back the last drawing or modification command that was executed. QCad can take back more than one command. E.g. if you've just created a line and a circle which you want to take back, launch the undo command twice. Undo cannot undo all actions. Some actions like file actions and layer actions can't be undone.

Redo

Toolbar:



Menu:

Edit - Redo

Hotkey:

uu, Ctrl-Shift-Z

Command:

redo

Description:

The redo command is the counterpart of the undo command. It makes changes undone before reappear.

Copy

Toolbar:



Menu:

Edit - Copy

Hotkey:

Ctrl-C

Command:

?

Description:

You might know the copy / paste mechanism from other applications. QCad offers a similar set of tools to copy entities from one drawing to another. Objects that are copied are stored on the QCad internal clipboard. This clipboard can only be accessed from QCad itself. You cannot paste graphical objects copied in other applications into QCad nor can you paste QCad entities into other applications.

Procedure:

1. After starting the copy tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to copy to the clipboard.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the reference point with the mouse or enter a co-ordinate in the command line. The reference point will be used to position the entities when pasting them into a drawing again.

Cut

Toolbar:



Menu:

Edit - Cut

Hotkey:

Ctrl-X

Command:

?

Description:

The cut command essentially works like the copy command described above. The only difference is that the selected entities will be removed from the current drawing after being copied to the clipboard.

Paste

Toolbar:**Menu:**

Edit - Paste

Hotkey:

Ctrl-V

Command:

?

Description:

The paste command inserts the entities previously copied to the clipboard. Paste is especially useful to transfer entities from one drawing to another.

All layers needed by the clipboard contents are pasted into the layerlist. Existing layers with the same name don't get overwritten. Sometimes this might lead to unexpected results. If an entity gets its attributes from the layer, it might be red in the original drawing and appear green in the drawing where you paste it. That's the case if a layer exists in both drawings with different attributes.

Inserts (references to blocks) that are on the clipboard will be pasted together with the blocks they refer to. Blocks in the current drawing don't get overwritten. This might lead to totally unexpected results in some cases. Make sure that different blocks never have the same name in drawings for which you use the copy / paste tools.

Viewing

The view menu and toolbar offer some tools to adjust the current viewport of the drawing. All these tools have absolutely no effect on the entities of the drawing. They only change the zoom factor and the visible area (viewport).

Viewing tools are probably the most often used functions in a CAD system. They can be easily accessed from the viewing toolbar (Figure 12). For the ones you use most, it's worth remembering and using the hotkey. To show the whole drawing, simply press `zA`. If the command line has the keyboard focus, you need to press the Escape key to release the focus first.



Figure 12: The viewing toolbar.

Scrolling

Use the two scrollbars at the right and the bottom of a drawing window to scroll around in a drawing. If you have a wheel mouse, you can also use the wheel to scroll up and down or left and right while holding down the shift key.

If your mouse has three mouse buttons, the middle mouse button can be used to change the current view by dragging the drawing around.

Toggle Grid

Toolbar:



Menu:

View - Grid

Command:

?

Toggles the grid visibility of the current drawing.

Toggle Draft Mode

Toolbar:



Menu:

View - Draft

Command:

?

Toggles the draft mode of the current drawing. In draft mode, all lines are shown with a width of 1 pixel. Hatches are not shown and large texts are only shown as rectangles. Use the draft mode if your drawing becomes very large and it takes long to redraw it.

Redrawing

Toolbar:



Menu:

View - Redraw

Hotkey:

zr, rg

Command:

regen

Redraws the current drawing. After moving or deleting a lot of entities the drawing can look incomplete or 'fuzzy'. With this function you can regenerate the graphic view.

Zooming In / Out

Toolbar:**Menu:**

View - Zoom in

View - Zoom out

Hotkey:

+ / -

Command:

?

This tool increases / decreases the current viewing factor by 1.5. The same effect can also be achieved by pressing the Control-Key on the keyboard and moving the mouse wheel (Figure 13).

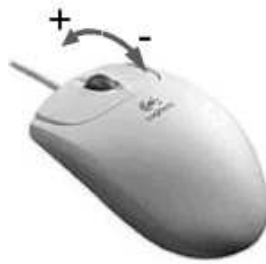


Figure 13: Use the Control-Key and the mouse wheel to zoom in / out quickly.

Auto Zoom

Toolbar:**Menu:**

View - Auto Zoom

Hotkey:

za

Command:

?

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

Window Zoom

Toolbar:**Menu:**

View - Window Zoom

Hotkey:

zw

Command:

?

This tool offers a quick way to view a certain area of the drawing.

Procedure:

1. Specify the first corner of the area you want to view. Click 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.

Panning

Toolbar:**Menu:**

View - Pan Zoom

Hotkey:

zp

Command:

?

Panning means moving around in the 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 doesn't have a middle mouse button you can use this tool instead and do the same with the left mouse button. Click the right mouse button when you're done.

Previous View

Toolbar:**Menu:**

View - Previous View

Hotkey:

zv

Command:

?

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

Layers

Layers are a useful concept for organizing drawings. It's worth learning how to work with layers - it will save you a lot of time especially when drafting more complex constructions. This chapter will introduce you into the concepts of layering and show you how QCad lets you manage the layers of your drawing.

Every drawing contains at least one layer: layer '0'. Additional layers can be created by the user at any time they are required. However, it's recommended to plan a drawing and to create all necessary layers with the correct attributes before starting to draft. The number of layers for a QCad drawing is not limited. Every entity of your drawing is placed on exactly one layer and a layer can contain an unlimited number of entities.

A Layer can be frozen to make all entities on it invisible. This way it is easily possible to temporarily hide all dimensions, texts or auxiliary constructions for a print out or to get a clearer overview of a complex drawing.

The current layer is the layer you are working on. It is highlighted in the layer list (Figure 14). Newly created entities are placed on the current layer.

Layers can be controlled from the layer menu or in the layer list which is usually shown at the right in the QCad application window (see Figure 14).



Figure 14: Layer list.

In Figure 14 the current layer is "construction". The eye icon at the left side of each layer name indicates whether the layer is currently visible or invisible (frozen). In this example, layer "auxiliary" is frozen, the others are visible.

Changing the Visibility of a Layer

The visibility of a layer can be changed by either double-clicking on it or by right-clicking on it and choosing "Toggle Visibility" from the context menu (Figure 15).



Figure 15: Context menu of the layer list.

Changing the Visibility of all Layers

Toolbar:



Menu:

Layer - Defreeze All
Layer - Freeze All

Hotkey:

th*, fr*

The two buttons shown above can be used to defreeze (thaw) or freeze all layers at once, which comes in handy when working with a drawing that has many layers. The same function is also available from the context menu.

Adding New Layers

Toolbar:



Menu:

Layer - Add Layer

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 (Figure 16).

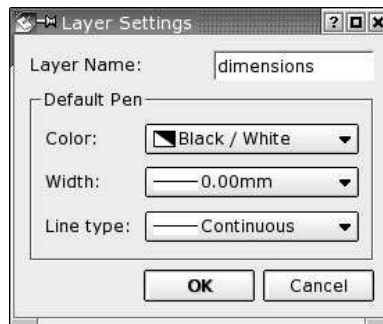


Figure 16: Dialog for creating new layers.

Deleting Layers

Toolbar:



Menu:

`Layer - Remove Layer`

You can delete the current layer with this command. Please note that all entities on that layer will be deleted as well. This command, unlike most other commands, can't be undone. However, if entities have been deleted with the layer, they will reappear on layer '0' if the action is undone. Layer '0' can never be deleted.

Editing Layer Attributes

Toolbar:



Menu:

`Layer - Edit Layer`

With this command you can trigger the same dialog that you used to create new layers (Figure 16). Use this to change the layer name or attributes. The name of the layer '0' cannot be changed.

Blocks

*Blocks are named groups of entities which can be inserted into a drawing multiple times with different attributes at different locations. Such an inserted instance of a block is called an **Insert** .*

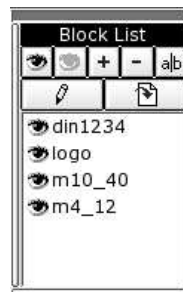


Figure 17: Block list.

Changing the Visibility of a Block

Double-click on a block in the block list to toggle its visibility. Alternatively, you can right-click on it and choose "Toggle Visibility" from the context menu (Figure 18).

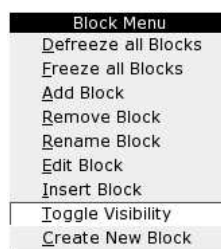


Figure 18: Context menu of the block list.

Changing the Visibility of all Blocks

Toolbar:



Menu:

Block - Defreeze All

Block - Freeze All

Description:

The two buttons shown above can be used to defreeze or freeze all blocks at once. The same function is also available from the context menu.

Adding New Blocks

Toolbar:



Menu:

Block - Add Block

Description:

This command creates a new empty block. The block dialog is shown where you can specify the name for the new block (Figure 19). If you want to create a block from existing entities, please use the "Create Block" command instead.



Figure 19: Dialog for creating new blocks.

Removing Blocks

Toolbar:



Menu:

Block - Remove Block

Description:

Removes the active block. Please note that this action cannot be undone. The block and all inserts of the block will be deleted.

Renaming Blocks

Toolbar:



Menu:

Block - Rename Block

Description:

Renames the active block. The same dialog is presented as for creating new blocks (Figure 19). Please note that this action cannot be automatically undone. The block and all inserts of the block will be renamed.

Editing Blocks

Toolbar:



Menu:

Block - Edit Block

Description:

This command opens the active block in a new document window where it can be modified just like any other drawing. To update the inserts that exist inside your drawing, simply activate the drawing window either by using the Windows menu or by clicking in it.

Inserting Blocks

Toolbar:



Menu:

Block - Insert Block

Command:

?

Options Toolbar:



Description:

Inserts the active block into the drawing.

Procedure:

1. Choose the block you want to insert from the list.
2. Click the insert button or choose 'Insert Block' from the menu.
3. Enter the rotation angle and scaling factor for the insert in the options toolbar.
4. To create an array of inserts, enter the number of columns and rows in the options toolbar and specify the column and row spacing.

Creating Blocks from Existing Entities

Toolbar:**Menu:**

Block - Create Block

Command:

?

Description:

Creates a block from existing entities.

Procedure:

1. Select the entities you want to use for the block.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the reference point of the block using the mouse or enter a co-ordinate in the command line.
4. Enter a name for the new block in the dialog that is shown (Figure 19) and click OK.

The block is now added to the block list and can be inserted into the drawing. The entities you selected in step 1 are removed and replaced by an insert of the block. If you don't want that you can use 'undo' to delete that insert. To restore the original entities, use 'undo' again. The undo / redo functions have no influence on the blocks themselves. Once a block is created it won't be removed again with undo.

Object Snap

This chapter lists all snap modes that can be activated to assist the user when setting points to exact co-ordinates.

For most constructions and modifications in QCad, it is necessary to specify the co-ordinates of reference points, startpoints, endpoints, centers or other identifying points. In order to make it possible to pick exact positions defined by previously drawn entities or by the grid, QCad offers you a set of object snap modes. The toolbar that features those modes, is automatically shown whenever it makes sense to use them (Figure 20).

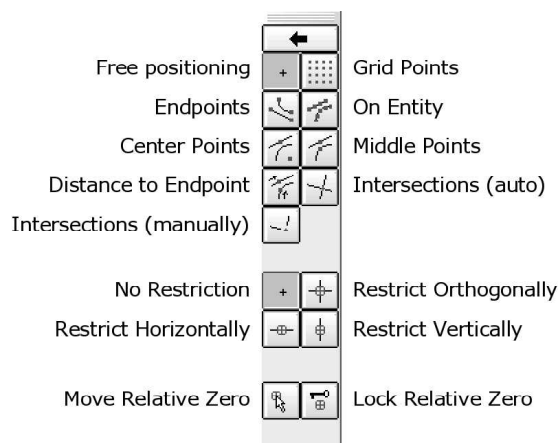


Figure 20: The snap toolbar.

Free Positioning

CAD Toolbar:



Menu:

Snap - Free

Hotkey:

os

Command:

?

Description:

Allows the user to set the co-ordinate freely using the mouse. Note, that this is almost never the recommended way of setting co-ordinates in a CAD system.

Snap to Grid

CAD Toolbar:



Menu:

Snap - Grid

Hotkey:

sg

Command:

?

Description:

Snaps to grid points.

Snap to Endpoints

CAD Toolbar:



Menu:

Snap - Endpoints

Command:

?

Hotkey:

se

Description:

Snap to endpoints of lines and arcs and to points (Figure 21).

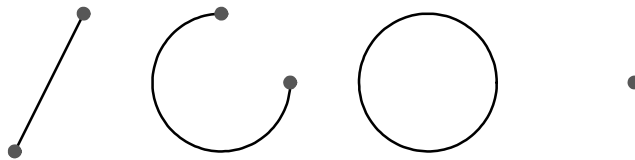


Figure 21: Endpoints found by the snap tool.

Snap to Points on Entities

CAD Toolbar:



Menu:

Snap - On Entity

Hotkey:

sn

Command:

?

Description:

Snap to the closest point that is on an entity (Figure 22).



Figure 22: 'On Entity' snaps to any point on the line of an entity.

Snap to Centers

CAD Toolbar:



Menu:

Snap - Centers

Hotkey:

so

Command:

?

Description:

Snap to centers of lines and arcs and to points (Figure 23).

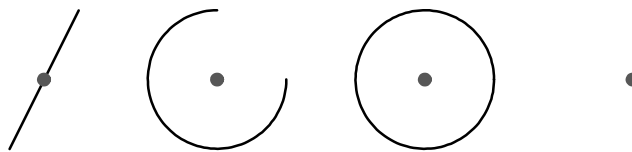


Figure 23: Centers found by the snap tool.

Snap to Middle Points

CAD Toolbar:



Menu:

Snap - Middle

Hotkey:

sm

Command:

?

Description:

Snaps to middle points of lines and arcs and to points (Figure 24). Note that the middle point of an arc is in the middle of the arc line while the center is at the center of the arc.



Figure 24: Middle points found by the snap tool.

Snap to Distance Points

CAD Toolbar:



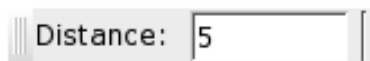
Menu:

Snap - Distance from Endpoint

Command:

?

Options Toolbar:



Description:

Snap to points with a given distance from endpoints (Figure 25). The distance can be adjusted in the options toolbar.

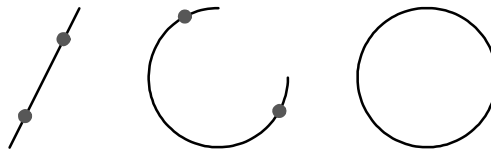


Figure 25: Example snap points for the snap tool 'Distance from Endpoint'.

Snap to Intersections

CAD Toolbar:



Menu:

Snap - Intersection

Hotkey:

si

Command:

?

Description:

Snap to intersections between entities (Figure 26).

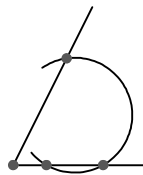


Figure 26: The Intersection Snap tool finds automatically all visible intersection points.

Snap to Intersections Manually

CAD Toolbar:



Menu:

Snap - Intersection Manually

Command:

?

Description:

Sometimes the intersection snap mode cannot be used to snap to an intersection point because the point is not on one or both of the entities. The second intersection tool lets you specify the two entities and then snaps the intersection point even if it's outside the entities (Figure 27).



Figure 27: The Manual Intersection Snap tool also lets you specify invisible intersection points.

Snap Restrictions

In addition to the snap functions described above, QCad can further restrict the position of the mouse cursor orthogonally to the position of the relative zero point. Figure 28 illustrates an example of how to use snap restrictions.

The 'Snap Point' is the point the user wants to specify. It's the point on the bottom line of the shape that has the same X-co-ordinate like the center of the full circle. The mouse cursor is located at the circle line. Because QCad is in the snap mode "Snap to Center Points", the center of the circle would normally be the snap point. A small rhombus highlights that point ("Center Point"). With the snap restriction "Restrict Horizontally" activated, QCad snaps to the point on the horizontal position of the relative zero point ("Snap Point").

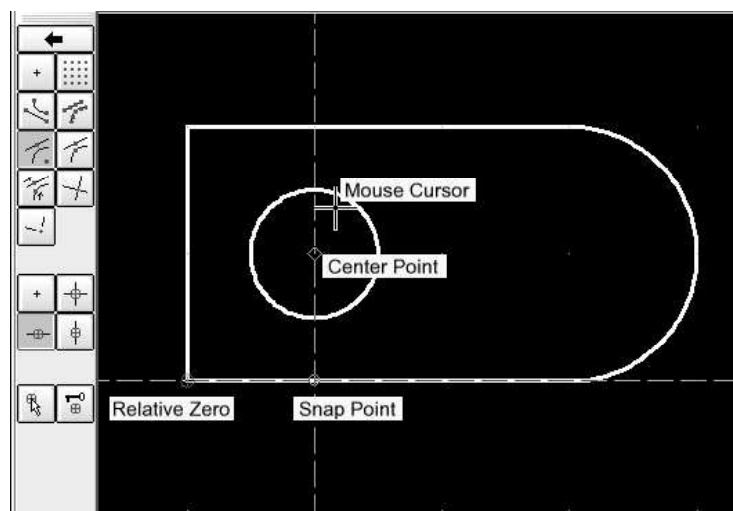


Figure 28: Snap restrictions further limit the snap position to imaginary lines orthogonally to the relative zero point.

Restrict Nothing

CAD Toolbar:



Menu:

Snap - Restrict Nothing

Command:

?

Description:

Disables all restrictions.

Restrict Orthogonally

CAD Toolbar:



Menu:

Snap - Restrict Orthogonally

Command:

?

Description:

Restricts orthogonally to relative zero point.

Restrict Horizontally

CAD Toolbar:



Menu:

Snap - Restrict Horizontally

Command:

?

Description:

Restricts horizontally to relative zero point.

Restrict Vertically

CAD Toolbar:



Menu:

Snap - Restrict Vertically

Command:

?

Description:

Restricts vertically to relative zero point.

Relative Zero Point

As mentioned previously, there is a relative Zero point in every QCad drawing. It appears as a small red circle with a hair cross. The relative zero point is the reference point for relative co-ordinates you enter in the command line and for snapping restrictions.

Normally, the relative zero point moves around automatically. For example if you're drawing a sequence of lines , the relative zero point always moves to the last point you've set. The next point of the sequence can then be defined in relative co-ordinates to the last point.

Sometimes this is not convenient and you would rather want to set the relative zero point to another position or lock its position so it stops moving around automatically. The snap toolbar offers two tools that do exactly this.

Moving the Relative Zero Point

CAD Toolbar:



Menu:

Snap - Set Relative Zero

Command:

?

Description:

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

(Un-)locking the Relative Zero Point

CAD Toolbar:



Menu:

Snap - (Un-)Lock Relative Zero

Command:

?

Description:

When enabled, this switch locks the position of the relative zero point. This means it doesn't move around automatically. You can still move it manually using the tool described above.

Entity Drawing Commands

This chapter describes the tools used in QCad for drawing graphic entities. Entities are the basic elements of a QCad drawing. Once drawn, entities can be manipulated and changed in various ways to suit the needs of the draftsman or designer. Subsequent chapters will explain these processes.

Points

Single Points

CAD Toolbar:



Menu:

Draw - Point - Points

Command:

point

Description:

This command is used to draw single points. Points are visually represented by a single 'dot'.

Procedure:

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

Lines

Click the line button in the CAD toolbar to show the line toolbar (Figure 29).

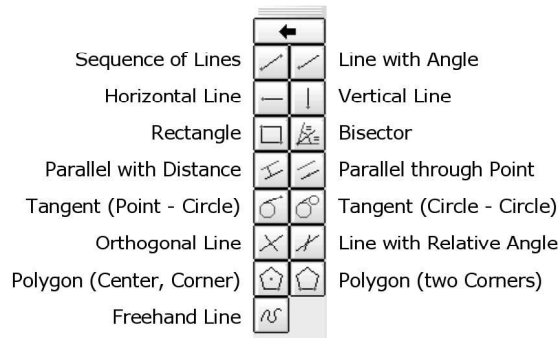


Figure 29: The line toolbar.

Sequence Of Lines

CAD Toolbar:



Menu:

Draw - Line - 2 Points

Command:

line

Options Toolbar:



Description:

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

Procedure:

1. Specify the startpoint of the first line segment. You can use the mouse or enter a co-ordinate in the console.
2. Specify the endpoint of the first line segment.
3. Specify the endpoints of additional line segments. Type **c** or **close** or click the 'Close' button in the Options Toolbar to close the sequence. If you need to undo a single line segment, you can do so by entering **u** or **undo** or clicking the 'Undo' button.

Line With Given Angle

CAD Toolbar:



Menu:

Draw - Line - Angle

Options Toolbar:

Angle: 30 Length: 50

Description:

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

1. Enter the angle and length of the line in the options toolbar. You can also use the command line: `angle` , `ang` or `a` to change the angle. `length` , `len` or `l` to change the length.
2. Place the line with the mouse or by entering a co-ordinate in the command line.

Horizontal / Vertical Line

CAD Toolbar:



Menu:

Draw - Line - Horizontal / Vertical

Options Toolbar:

Length: 50

Description:

Use this tool to create horizontal (vertical) lines. This tool essentially behaves like the one described above, except that you cannot input an angle.

Rectangle

CAD Toolbar:



Menu:

Draw - Line - Rectangle

Command:

rec, rectang, rectangle

Description:

Use this tool to create rectangular shapes more quickly than with the line tool. A rectangle consists of four separate lines.

Procedure:

1. Specify the first corner of the rectangle.
2. Drag the mouse to the second corner and click to specify the second corner of the rectangle. Alternatively you can enter the co-ordinate of the second corner in the command prompt. E.g. to create a rectangle with width 50 and height 25, enter @50,25 .

Parallel

CAD Toolbar:



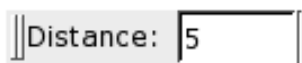
Menu:

Draw - Line - Parallel

Command:

?

Options Toolbar:



Description:

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

Procedure:

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

Bisector

CAD Toolbar:



Menu:

Draw - Line - Bisector

Command:

?

Options Toolbar:



Description:

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

Procedure:

1. In the options toolbar enter the length of the bisector(s), starting from the intersection point of the two lines. In the second text box, type the number of angle bisectors you want to create. The default is '1' but you can also create multiple bisectors as shown in Figure 30. To specify the length in the command line, enter `length`, `len` or `l`, followed by the length. To specify the number of bisectors in the command line, enter `number`, `num` or `n`, followed by the number.
2. Click the first line entity which defines the angle.
3. Click the second line entity so that the preview shows the angle bisector(s) you want to create.

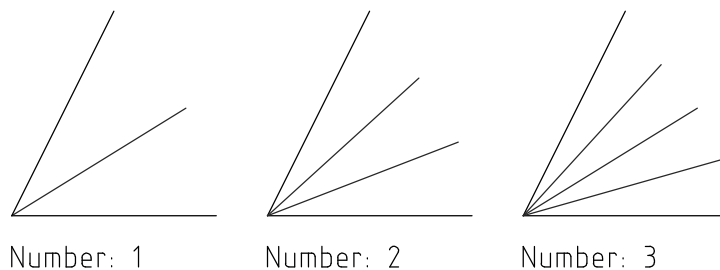


Figure 30: Construction of Bisectors.

Tangent From Point to Arc / Circle

CAD Toolbar:



Menu:

Draw - Line - Tangent (P,C)

Command:

?

Description:

Create tangents from a co-ordinate to an existing arc or circle entity with this tool.

Procedure:

1. Use the mouse to specify the location of the startpoint of the line or enter a co-ordinate in the command line.
2. Click the arc or circle 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 From Arc to Arc / Circle to Circle

CAD Toolbar:



Menu:

Draw - Line - Tangent (C,C)

Command:

?

Description:

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

Procedure:

1. Click the first arc or circle entity to which you want to create the tangent.
2. 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.

Line Orthogonal to Another Entity

CAD Toolbar:



Menu:

Draw - Line - Orthogonal

Command:

?

Description:

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

Procedure:

1. Click the existing entity.
2. Place the line with the mouse or enter a co-ordinate in the command line.

Line with Relative Angle to Another Entity

CAD Toolbar:



Menu:

Draw - Line - Relative Angle

Command:

?

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.

Procedure:

1. Enter the angle in the options toolbar.
2. Click the existing entity.
3. Place the line with the mouse or enter a co-ordinate in the command line.

Freehand Line

CAD Toolbar:



Menu:

Draw - Line - Freehand

Command:

?

Description:

With this tool you can draw freehand lines. Please use this tool sparingly since technical drawings usually require absolute accuracy. However, on some occasions this tool might come in handy (e.g. for broken edges like shown in Figure 31).

Procedure:

1. Place the mouse at the startpoint 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.

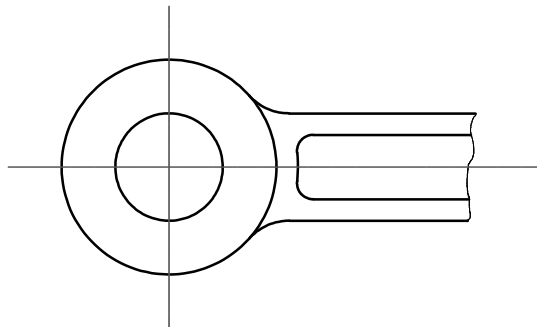


Figure 31: Example use for the freehand line tool.

Polygon with Center and Corner

CAD Toolbar:



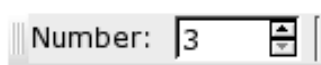
Menu:

Draw - Line - Polygon (Cen,Cor)

Command:

?

Options Toolbar:



Description:

Creates polygons with the center and one corner given.

Procedure:

1. Enter the degree of the polygon (number of edges / corners) in the options toolbar. Valid numbers range from 3 to 9999.
2. Set the center of the polygon using the mouse or enter a co-ordinate in the command line.
3. Set the first corner of the polygon.

Polygon with Two Corners

CAD Toolbar:



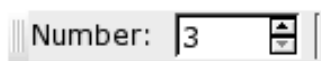
Menu:

Draw - Line - Polygon (Cor,Cor)

Command:

?

Options Toolbar:



Description:

Creates polygons with two corners given.

Procedure:

1. Enter the degree of the polygon (number of edges / corners) in the options toolbar. Valid numbers range from 3 to 9999.
2. Set the first corner of the polygon using the mouse or enter a co-ordinate in the command line.
3. Set the second corner of the polygon.

Arcs

Arc with Center, Radius and Angles

CAD Toolbar:



Menu:

Draw - Arc - Center, Point, Angles

Command:

?

Options Toolbar:



Description:

Draw arcs with center, radius and angles given with this tool.

Procedure:

1. Choose the direction of the arc in the options toolbar or using the command `reversed`, `rev` or `r`.
2. Set the center of the arc using the mouse or enter a co-ordinate in the command line.
3. Define the radius by clicking a point on the arc or by entering a co-ordinate into the command line. You may also enter the radius directly in the command line.
4. Set the start angle with the mouse or by entering a co-ordinate or the angle amount in the command line.
5. Set the end angle the same way as the start angle.

Arc with 3 Points

CAD Toolbar:



Menu:

Draw - Arc - 3 Points

Command:

?

Description:

If you know the startpoint, the endpoint and a point in the middle on the arc line, you can use this tool to create arcs.

Procedure:

1. Set the start point using the mouse or enter a co-ordinate 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 Arc

CAD Toolbar:



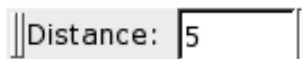
Menu:

Draw - Arc - Concentric

Command:

?

Options Toolbar:



Description:

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

Procedure:

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

Circles

Circle with Center and Point

CAD Toolbar:



Menu:

Draw - Circle - Center, Point

Command:

?

Hotkey:

ci

Description:

A circle with a given center and a point on the circle line can be created using this tool.

Procedure:

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

Circle with Center and Radius

CAD Toolbar:



Menu:

Draw - Circle - Center, Radius

Command:

?

Description:

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

Procedure:

1. Enter the radius in the option toolbar or use the command `radius`, `rad` or `r` to enter the radius on the command line.
2. Set the center of the circle using the mouse or enter a co-ordinate in the command line.

Circle with 2 Points

CAD Toolbar:



Menu:

Draw - Circle - 2 Points

Command:

?

Description:

Draws circles with two diametrically opposed points.

Procedure:

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

Circle with 3 Points

CAD Toolbar:



Menu:

Draw - Circle - 3 Points

Command:

?

Description:

If you know any three points on the circle line of a circle, you can use this tool to create circles.

Procedure:

1. Set the first point using the mouse or enter a co-ordinate in the command line.
2. Set the second point.
3. Set the third point.

Concentric Circle

CAD Toolbar:



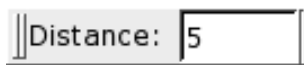
Menu:

Draw - Circle - Parallel

Command:

?

Options Toolbar:



Description:

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

Procedure:

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

Ellipses

Full Ellipse

CAD Toolbar:



Menu:

Draw - Ellipse - Ellipse

Command:

?

Description:

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

Procedure:

1. Set the center of the ellipse using the mouse or enter a co-ordinate 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 co-ordinate into the command line or enter an angle and major radius in the format @50 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

CAD Toolbar:



Menu:

Draw - Ellipse - Ellipse Arc

Command:

?

Description:

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

Procedure:

1. Set the center of the ellipse using the mouse or enter a co-ordinate 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 co-ordinate into the command line or enter an angle and major radius in the format @50 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 co-ordinate or the angle amount in the command line.
5. Set the end angle the same way as the start angle.

Texts

CAD Toolbar:



Menu:

Draw - Text

Command:

text

Description:

Use this tool to create single line and multi line texts.

Procedure:

1. Just after launching the tool, the text dialog is presented in which you can choose the font and enter the text you want to create (Figure 32).

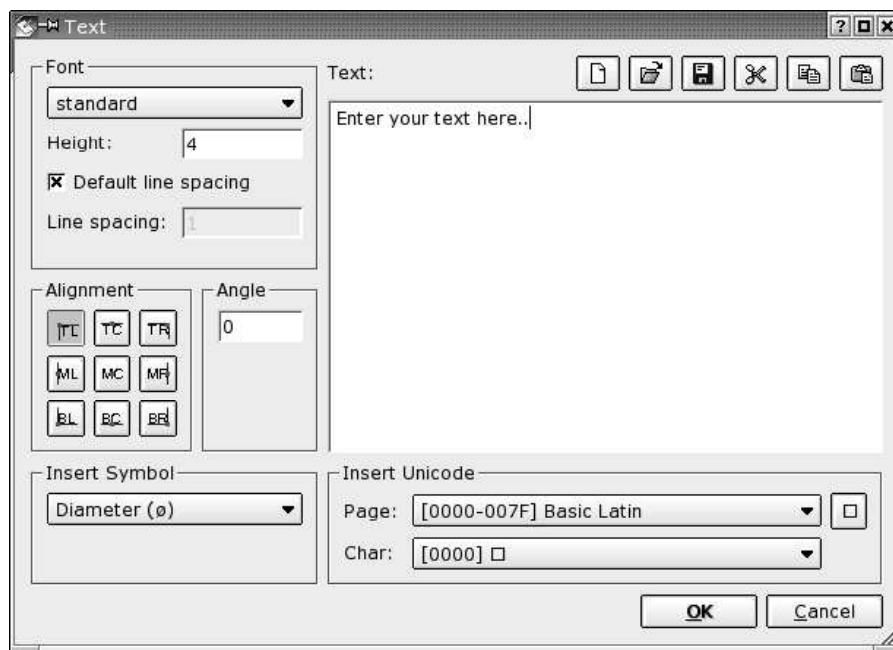


Figure 32: The text creation dialog.

1. Choose the font name and attributes in the upper left area of the dialog (Figure 33). For a complete list of all available fonts, please refer to the appendix. In the height field, enter the height of the text in the unit of your drawing. To choose normal line spacing, click the checkbox "Default line spacing". You can increase or decrease the line spacing by disabling the checkbox and entering a value in the "Line spacing" box. The value entered is a factor relative to the default (1.0). Valid values range from 0.25 to 4.00. Values outside this range are allowed but break compatibility to other CAD programs such as AutoCAD.

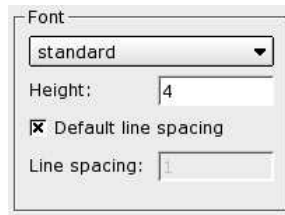


Figure 33: Font selection.

1. Choose the alignment of the text (Figure 34).

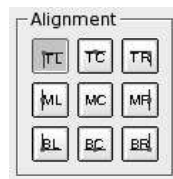


Figure 34: Text alignment.

1. Enter the angle for the text (Figure 35). Normal horizontal texts have an angle of zero degrees.

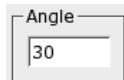


Figure 35: Text angle.

1. Enter the text you want to create in the large text box (Figure 36). You can also use the icons above the box to load texts, save texts you've already entered or to paste texts from the clipboard.
To enter special symbols that are not available on your keyboard, you can use the combobox at the bottom left of the dialog (Figure 37). Simply choose a symbol from the list and it will be placed at the current cursor position in the text box.
If the symbol list doesn't contain the symbol you need, please use the comboboxes at the bottom right to choose any character from the unicode char set (Figure 38). Note that the font you've chosen might not contain the characters you insert. In that case the characters will be ignored.
To insert a unicode character, select the unicode page in the first combobox and choose the character from the second combobox. Now click the button at the right to insert the character at the current cursor position.

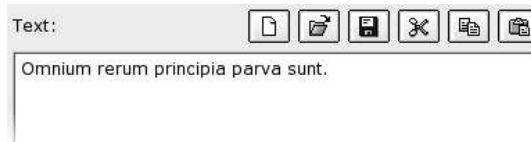


Figure 36: The text area where you can enter or load a text.

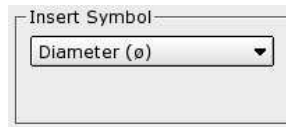


Figure 37: Insert special symbols using the Symbol combobox.



Figure 38: Insert any unicode characters with the unicode tools..

1. Click "OK" to exit the text dialog.
2. Use the mouse to specify the location of the text or enter a co-ordinate in the command line.
3. Often, users need to create a number of similar texts. For that reason the text tool doesn't terminate after creating the first text entity. You can change the text and the text angle in the options toolbar and create other texts with similar settings.
4. Right click or press Escape to stop the text tool.

Advanced Text Commands

Sometimes it's useful to put two texts on top of each other (e.g. for tolerances). There's a special escape sequence you can type anywhere in the text to achieve that. For example the sequence "10\S+0.05\ -0.05; Millimeter" would produce a tolerance symbol as shown in Figure 39.

10^{+0.05}_{-0.05}Millimeter

Figure 39: Text produced by the string "10\S+0.05\ -0.05; Millimeter".

Dimensions

This chapter is about constructing dimensions. Figure 40 shows the notation that is used when referring to dimension parts.

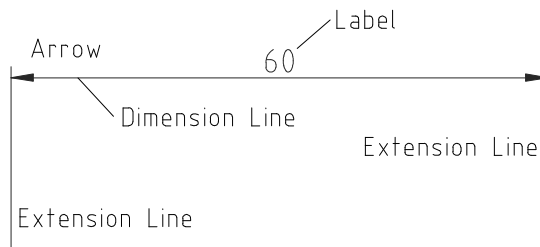
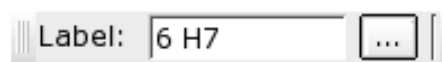


Figure 40: Notation for dimensions.

Options Toolbar:

The dimensioning tools may show individual option toolbars, but also share one common toolbar. It allows you to enter a text label for the dimension:



This text label can be any text. When left empty, the label is generated automatically and contains the measured distance of the dimension. To add additional signs or letters to a measured dimension, you can use the combination to insert the automatically generated label anywhere in the text you entered.

Examples:

Chosen label in the toolbar: Example for the generated label:

a	a
<>	20
<> H7	20 H7
∅<>	∅20

Aligned Dimension

CAD Toolbar:



Menu:

Dimension - Aligned

Hotkey:

da

Command:

?

Description:

Creates aligned dimensions. Aligned dimensions usually measure the length of an existing line (Figure 41). The dimension line is always parallel to the line between the two extension line endpoints.

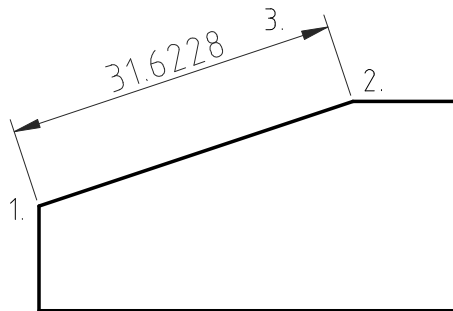


Figure 41: Sample for an aligned dimension.

Procedure:

1. Set the first extension line endpoint with the mouse or enter a co-ordinate in the command line.
2. Set the second extension line endpoint.
3. Set the position of the dimension line.

Linear Dimensions

CAD Toolbar:



Menu:

Dimension - Linear

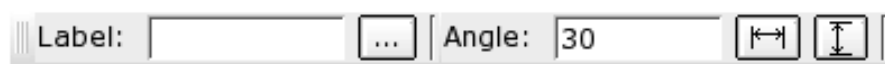
Hotkey:

dr

Command:

?

Options Toolbar:



Description:

Creates linear dimensions. Linear dimensions are usually used to measure vertical or horizontal distances but can also measure distances with any other angle. Figure 42 shows a horizontal version of a linear dimension.

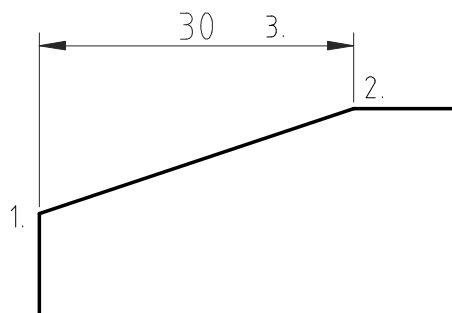


Figure 42: Example of a horizontal, linear dimension.

Procedure:

1. Enter the angle of the dimension in the options toolbar or click the vertical or horizontal button to set the angle to 0 or 90.
2. Set the first extension line endpoint with the mouse or enter a co-ordinate in the command line.
3. Set the second extension line endpoint.
4. Set the position of the dimension line.

Horizontal / Vertical Dimensions

CAD Toolbar:



Menu:

Dimension - Horizontal

Dimension - Vertical

Hotkey:

dh, dv

Command:

?

Description:

This tool behaves essentially like the one described above (Linear Dimension). The only difference is that you cannot enter an angle.

Radial Dimensions

CAD Toolbar:



Menu:

Dimension - Radial

Command:

?

Description:

Creates radial dimensions for circle or arc entities (Figure 43).

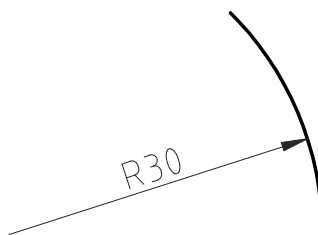


Figure 43: Example of a radial dimension.

Procedure:

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

Diametric Dimensions

CAD Toolbar:



Menu:

Dimension - Diametric

Command:

?

Description:

Creates diametric dimensions for circle or arc entities (Figure 44).

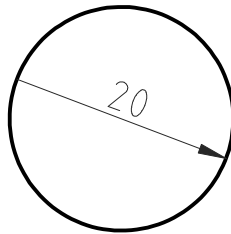


Figure 44: Example of a diametric dimension.

Procedure:

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

Angular Dimensions

CAD Toolbar:



Menu:

Dimension - Angular

Command:

?

Description:

Creates angular dimensions between two reference lines. (Figure 45).

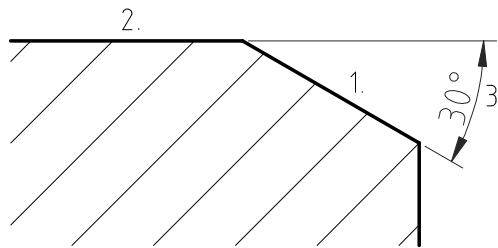


Figure 45: Example of an angular dimension.

Procedure:

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

Leader

CAD Toolbar:



Menu:

Dimension - Leader

Hotkey:

dl

Command:

?

Description:

Leaders are arrows that usually point from a text entity to an other entity (Figure 46). In the figure the text entity "N7" is connected to a surface with a leader.

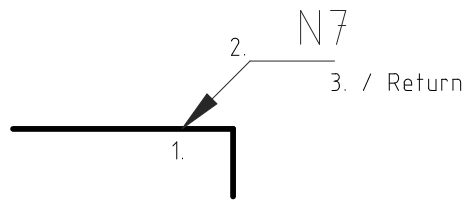


Figure 46: Example of a leader.

Procedure:

1. Set the location where the leader points to or enter a co-ordinate 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 press Return to finish.

Hatches and Solid Fills

CAD Toolbar:



Menu:

Draw - Hatch

Command:

?

Description:

Fills an area surrounded by existing entities with a hatch pattern or a solid color.

Procedure:

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 in the right sketch of Figure 47.

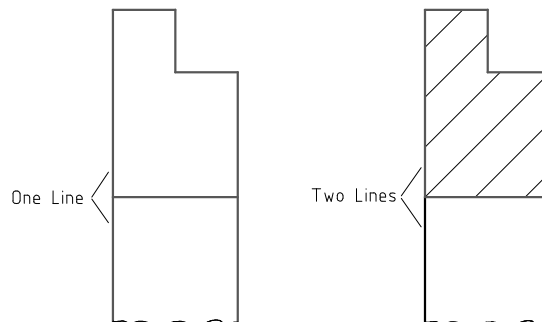


Figure 47: Preparation for the boundary entities.

1. Launch the hatching action.
2. Select the contour(s) you want to fill. Note that islands inside contours will get hatched if they are not selected (Figure 48)

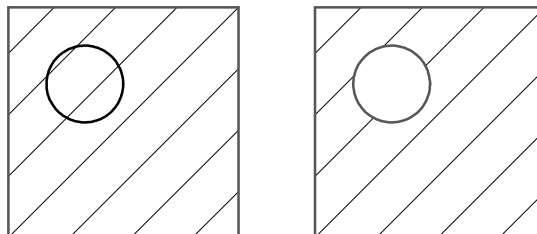


Figure 48: Islands which are selected won't get hatched through (right).

1. Click the right arrow button in the CAD toolbar to continue.
2. A dialog for the hatch options is displayed (Figure 49). Choose a hatch pattern, scale factor and a rotation angle of the pattern. If you want to fill the object with a solid color instead of a pattern, tick the checkbox "Solid Fill".
3. 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 pattern.

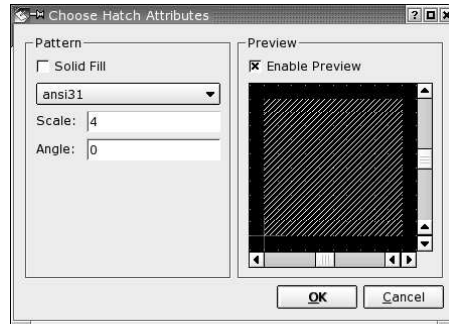


Figure 49: Hatch pattern selection.

Images

CAD Toolbar:



Menu:

Draw - Image

Command:

?

Options Toolbar:



Description:

Inserts a raster image (bitmap) into the drawing. Bitmaps always appear in the background of the rest of the drawing. The DXF file which is created when saving a drawing, contains just a reference to images. It's recommended to keep the image file and the DXF file in the same folder. Transparent background is supported for PNG images.

Procedure:

1. Choose the image file which you want to insert.
2. Type a scaling factor in the options toolbar. A factor of 1.0 means that one pixel of the image will be one unit large in the drawing.
3. Type a rotation angle in the options toolbar. Please note that rotated images can slow down the display process of QCad significantly. In any case it's more efficient to rotate the bitmap in a bitmap editing program before you insert it into QCad.
4. Specify the insertion point of the image. That's the left bottom corner of the image.

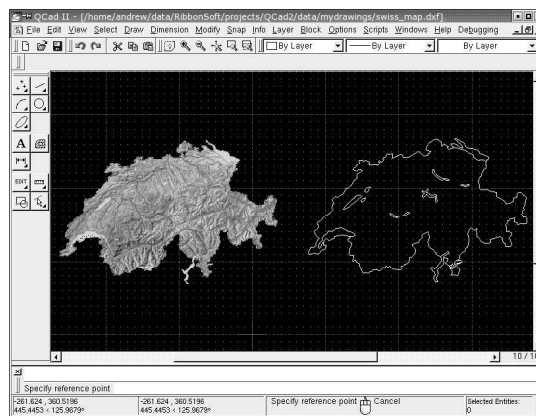


Figure 50: Images are often used in the background of a drawing to manually vectorize bitmaps.

Entity Selection

The selection tools documented in this chapter are used to select entities for later modification. For example if you want to move some entities, you need to select them before you can do so. The selection tells QCad which entities you intend to move.

Selected entities are highlighted in the graphic view with red color and a unique tight dotted line pattern.

Deselecting All

CAD Toolbar:



Menu:

Select - Deselect All

Command:

?

Description:

Deselects all entities on all currently visible layers. Use this tool to make sure that no entities are selected.

Selecting All

CAD Toolbar:



Menu:

Select - Select All

Command:

?

Description:

Selects all entities on all currently visible layers.

Selecting Single Entities

CAD Toolbar:



Menu:

Select - Select Entity

Command:

?

Description:

Selects single entities.

Procedure:

- Use the mouse to specify the entities to select. If you click an entity that is already selected it will get deselected.

Selecting Contours

CAD Toolbar:



Menu:

Select - (De-)Select Contour

Command:

?

Description:

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

Procedure:

- Use the mouse to pick one entity of the contour you want to select. The algorithm then searches from the endpoints of the selected entity in both directions for entities that are connected to this entity. All connected entities get selected if the entity you picked was unselected. Otherwise the whole contour will be deselected.

Window Deselection

CAD Toolbar:



Menu:

Select - Deselect Window

Command:

?

Description:

Deselects all entities that are entirely within a given window.

Procedure:

1. Click the first corner of the window using the left mouse button.
2. Drag the mouse to the other corner.
3. Click the second corner of the window.

Window Selection

CAD Toolbar:



Menu:

Select - Select Window

Command:

?

Description:

Selects all entities that are entirely within a given window.

Procedure:

Analog to deselecting entities in a window.

Inverting Selection

CAD Toolbar:



Menu:

Select - Invert Selection

Command:

?

Description:

Selects all unselected entities and deselects all selected entities.

Deselect Intersected Entities

CAD Toolbar:



Menu:

Select - Deselect Intersected

Command:

?

Description:

Deselects all entities that are intersected by a line.

Procedure:

1. Set the startpoint of the line using the left mouse button.
2. Drag the mouse to the endpoint of the line.
3. Set the endpoint of the line. All entities that are intersected by the line you've just drawn will be deselected.

Select Intersected Entities

CAD Toolbar:



Menu:

Select - Select Intersected

Command:

?

Description:

Selects all entities that are intersected by a line.

Procedure:

Analog to deselecting intersected entities.

Modification

Modification tools allow the user to modify existing entities. This chapter is a brief overview over the modification tools of QCad (Figure 51).

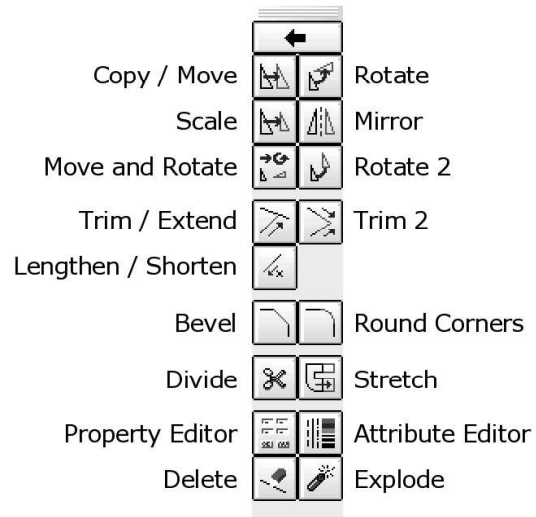


Figure 51: The modification toolbar.

Move

CAD Toolbar:



Menu:

Modify - Move

Command:

?

Description:

Moves or copies entities. This tool can create single or multiple copies of existing entities.

Procedure:

1. After starting the move tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to move or copy.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the reference point with the mouse or enter a co-ordinate in the command line.
4. Set the target point. To move the selected entities by a given amount, e.g. by 50 to the right, enter @50,0 in the command line.
5. The move dialog is displayed (Figure 52). To move the entities, choose "Delete Original", to copy them choose "Keep Original". Finally you can create a 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 entities in the end. The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".
6. Click OK to move / copy the entities. The selected entities will be deselected automatically.

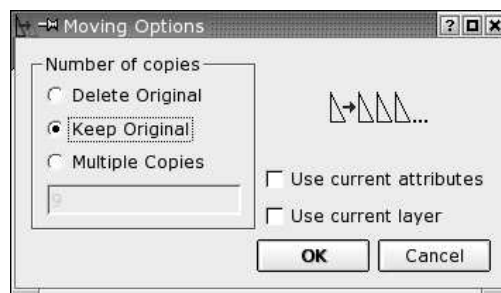


Figure 52: Move options dialog.

Rotate

CAD Toolbar:



Menu:

Modify - Rotate

Command:

?

Description:

Rotates entities by a given angle around a given center. This tool can create single or multiple copies of existing entities.

Procedure:

1. After starting the rotate tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to rotate.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the center of the rotation with the mouse or enter a co-ordinate in the command line.
4. The rotate dialog is displayed (Figure 53). To rotate the entities without keeping the original entities, choose "Delete Original", to copy them choose "Keep Original". Finally you can create a number of rotated 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 entities in the end. The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".
5. Click OK to rotate the entities. The selected entities will be deselected automatically.

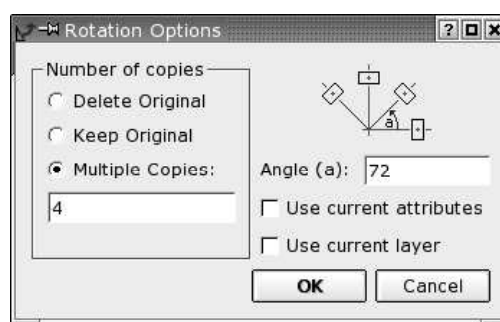


Figure 53: Rotate options dialog.

Scale

CAD Toolbar:



Menu:

Modify - Scale

Command:

?

Description:

Scales entities by a given factor from a given center. This tool can create single or multiple copies of existing entities.

Procedure:

1. After starting the scale tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to scale.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the center of the scaling with the mouse or enter a co-ordinate in the command line.
4. The scale dialog is displayed (Figure 54). To scale the entities without keeping the original entities, choose "Delete Original", to copy them choose "Keep Original". Finally you can create a number of scaled 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 entities in the end. The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".

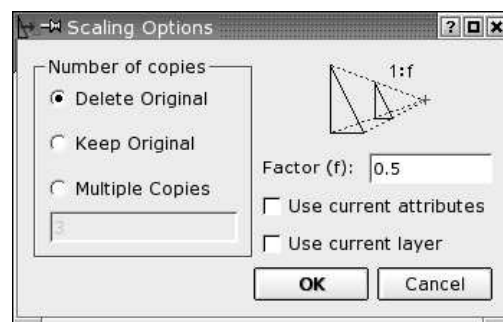


Figure 54: Scale options dialog.

1. Click OK to scale the entities. The selected entities will be deselected automatically.

Mirror

CAD Toolbar:



Menu:

Modify - Mirror

Command:

?

Description:

Mirrors entities at a given axis. This tool can create a mirrored copy of existing entities.

Procedure:

1. After starting the mirror tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to mirror.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the first endpoint of the mirror axes with the mouse or enter a co-ordinate in the command line.
4. Set the second endpoint of the mirror axes.
5. The mirror dialog is displayed (Figure 55). To mirror the entities without keeping the original entities, choose "Delete Original", to copy them choose "Keep Original". The copy will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".
6. Click OK to mirror the entities. The selected entities will be deselected automatically.

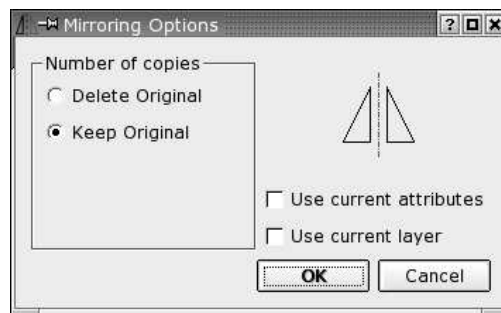


Figure 55: Mirror options dialog.

Move and Rotate

CAD Toolbar:



Menu:

Modify - Move and Rotate

Command:

?

Description:

Moves or copies and simultaneously rotates entities. This tool can create single or multiple copies of existing entities.

Procedure:

1. After starting the move and rotate tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to move or copy.
2. Click the right arrow button in the CAD toolbar to continue.
3. Enter the rotation angle in the options toolbar or use the command `angle`, `ang` or `a` in the command line.
4. Set the reference point with the mouse or enter a co-ordinate in the command line.
5. Set the target point. To move the selected entities by a given amount, e.g. by 50 to the right, enter `@50,0` in the command line. In Figure 56, 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 (Figure 59). To move the entities, choose "Delete Original", to copy them choose "Keep Original". Finally you can create a 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 entities in the end. The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".
7. Click OK to move and rotate the entities. The selected entities will be deselected automatically.

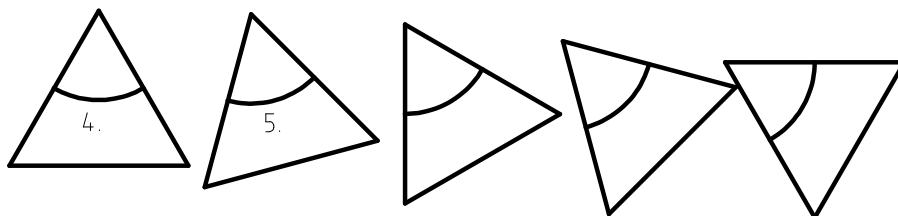


Figure 56: Move and Rotate reference point (4.) and target point (5.).

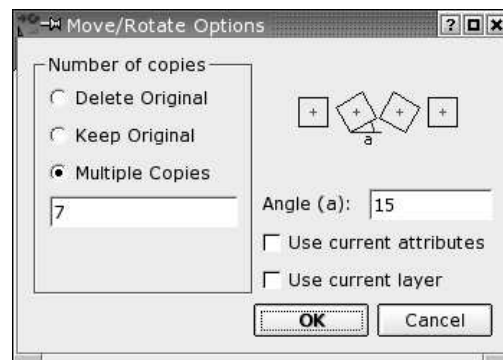


Figure 57: Move and Rotate options dialog.

Rotate and Counter-rotate (Rotate 2)

CAD Toolbar:



Menu:

Modify - Rotate 2

Command:

?

Description:

Rotates and counter-rotates entities. This tool can create single or multiple copies of existing entities. Use it if you want to rotate objects around a center without rotating the objects themselves.

Procedure:

1. After starting the rotate 2 tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to move or copy.
2. Click the right arrow button in the CAD toolbar to continue.
3. Set the center for the main rotation with the mouse or enter a co-ordinate in the command line.
4. Set the center of the rotation for the individual objects. The second center is rotated together with the entities around the first center (Figure 58).
5. The rotate 2 dialog is displayed (Figure fig:rotate2dialog01). To move the entities, choose "Delete Original", to copy them choose "Keep Original". Finally you can create a 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 entities in the end. The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can tick "Use current attributes" or "Use current layer".
6. Click OK to move and rotate the entities. The selected entities will be deselected automatically.

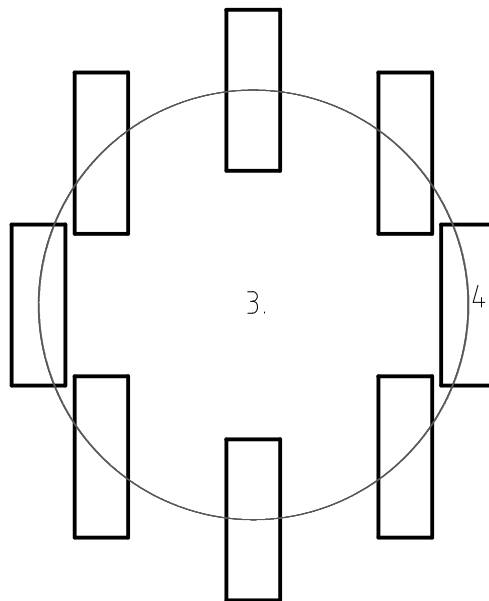


Figure 58: Rotate around two centers, an absolute center (3.) and a relative center (4.).

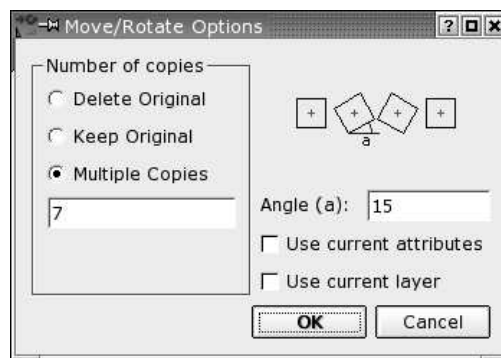


Figure 59: Move and Rotate options dialog.

Trim / Extend

CAD Toolbar:



Menu:

Modify - Trim

Command:

?

Description:

Trims or extends a line or arc to meet another entity (Figure 60).

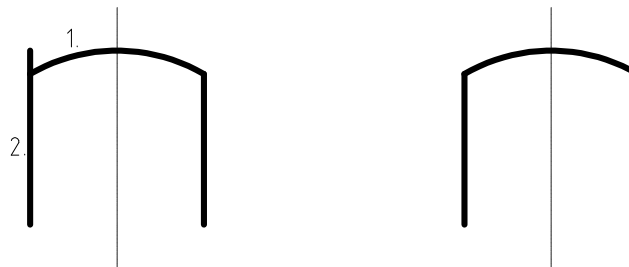


Figure 60: With the trim tool, the line at the left can be easily shortened to meet the arc.

Procedure:

1. Choose the limiting entity to which one or more other entities need to be trimmed.
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 Figure 60 you might want the top part of the line to stay and the lower part to disappear. In that case you would have to click the trim entity at the upper part. You always need to pick the part of the entity that you want to stay.
3. Right click twice or press Escape twice to stop the tool.

Trim Two Entities

CAD Toolbar:



Menu:

Modify - Trim Two

Command:

?

Description:

Trims or extends two line or arc entities at once to meet each other at their intersection point (Figure 61).

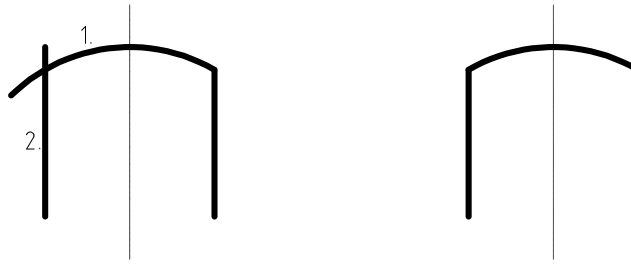


Figure 61: With the second trim tool, the line and the arc can both be trimmed to build an edge at once.

Procedure:

1. Choose the first entity to be trimmed to an edge.
2. Choose the second entity to be trimmed to an edge.
3. Right click twice or press Escape twice to stop the tool.

Lengthen

CAD Toolbar:



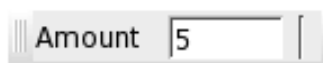
Menu:

Modify - Lengthen

Command:

?

Options Toolbar:



Description:

The Lengthen command can be used to either lengthen or shorten Lines or Arcs without the use of boundary edges.

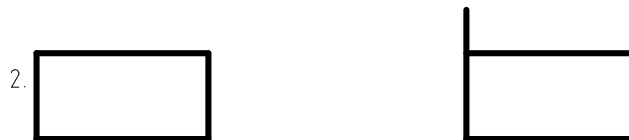


Figure 62: The Lengthen tool lets you trim an entity by a given amount.

Procedure:

1. Enter the distance you want to lengthen an entity by in the options toolbar. 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 parts of the drawing unit.
2. Choose the entity to be lengthened or shortened somewhere close to the end you want to modify.
3. Right click twice or press Escape twice to stop the tool.

Bevel

CAD Toolbar:



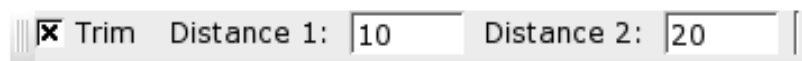
Menu:

Modify - Bevel

Command:

?

Options Toolbar:



Description:

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

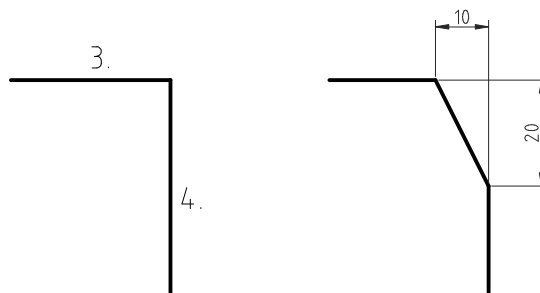


Figure 63: The bevel tool with trimming enabled.

Procedure:

1. Enter the geometry of the bevel in the options toolbar. 'Distance 1' is the distance the bevel line will have from the (imaginary) intersection of the two edges. 'Distance 2' is the same distance for the second edge.
2. Tick 'Trim' if you want to trim the entities automatically. If the option is disabled, the edges will stay untouched.
3. Choose the first edge entity (a line or arc).
4. Choose the second edge.

Round

CAD Toolbar:



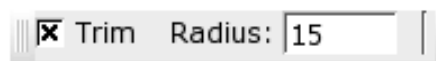
Menu:

Modify - Round

Command:

?

Options Toolbar:



Description:

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

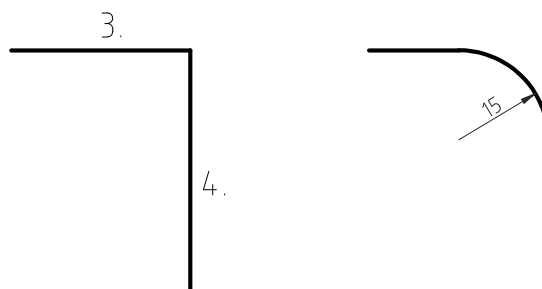


Figure 64: The round tool with trimming enabled.

Procedure:

1. Enter the radius of the rounding in the options toolbar.
2. Tick 'Trim' if you want to trim the entities automatically. If the option is disabled, the edges will stay untouched.
3. Choose the first edge entity (a line or arc).
4. Choose the second edge.

Divide

CAD Toolbar:



Menu:

Modify - Divide

Command:

?

Description:

Divides entities into two separate entities.

Procedure:

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

Stretch

CAD Toolbar:



Menu:

Modify - Stretch

Command:

?

Description:

Stretches contours.

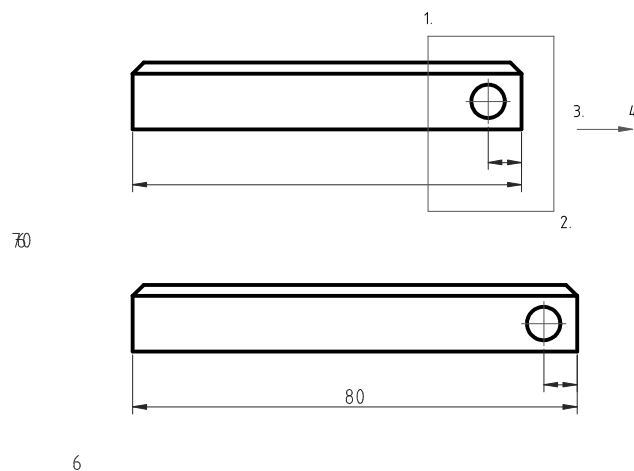


Figure 65: The stretch tool comes in handy when you need to lengthen or shorten complex constructions.

Procedure:

1. Set the first corner of the stretch area.
2. Set the second corner of the stretch area.
3. Set the reference point for the translation using the mouse or enter a co-ordinate in the command line.
4. Set the target point.

Properties

CAD Toolbar:



Menu:

Modify - Properties

Command:

?

Description:

Modify the attributes and geometry of entities in a dialog. Depending on the entity type, different dialogs are presented to modify important attributes and properties of the entity.

Procedure:

1. Click the entity to modify.

Attributes

CAD Toolbar:**Menu:**

Modify - Attributes

Command:

?

Description:

Modify the attributes of a selected group of entities.

Procedure:

1. After starting the tool, the CAD toolbar shows the selection tools. Use them to select the entities for which you want to change the attributes.
2. Click the right arrow button in the CAD toolbar to continue.
3. In the dialog shown, change only those attributes that you want to become the same for all selected entities. E.g. if you set the color to blue, all selected entities will become blue but their line width, layer and line style will stay the same.

Deleting Entities

CAD Toolbar:



Menu:

Modify - Delete

Command:

?

Description:

Deletes entities.

Procedure:

1. After starting the tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to remove from the drawing.
2. Click the right arrow button in the CAD toolbar to continue. The selected entities will be deleted.

Exploding Groups

CAD Toolbar:



Menu:

Modify - Explode

Command:

?

Description:

Converts blocks, polylines, texts, dimensions and hatches into basic atomic entities. Dimensions are split into lines, arrows and the text as still one entity. Texts are split into lines and arcs.

To split everything into atomic entities, use the command twice for all entities.

Procedure:

1. After starting the tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to explode.
2. Click the right arrow button in the CAD toolbar to continue.

Getting Measurements

Distance (Point, Point)

CAD Toolbar:



Menu:

Info - Distance from Point to Point

Command:

?

Description:

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

Procedure:

- Use the mouse to specify the location of the first point or enter a co-ordinate in the command line.
- Specify the second point.
- The measured distance is written to the command line.

Distance (Entity, Point)

CAD Toolbar:



Menu:

Info - Distance from Entity to Point

Command:

?

Description:

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

Procedure:

- Specify the entity.
- Use the mouse to specify the location of the point or enter a co-ordinate in the command line.
- The measured distance is written to the command line.

Angle

CAD Toolbar:



Menu:

Info - Angle between two lines

Command:

?

Description:

This tool measures the angle between two given lines.

Procedure:

- Specify the first line.
- Specify the second line.
- The measured angle (in degrees) is written to the command line.

Application Preferences

The general preferences dialog is used to change application options. There are options that apply to the whole Qcad application rather than just a single drawing.

Edit - General Preferences

Appearance

Language

The first language option changes the language of the Qcad GUI (menus, dialogs, ..). The second option changes the translation used for the commands that are entered in the command line. It is possible to e.g. use German menus and enter the English commands (Figure 66).

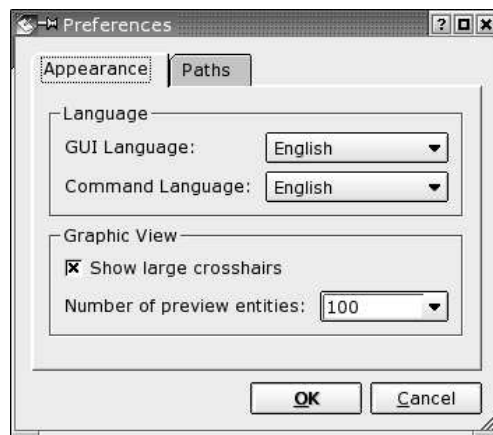


Figure 66: General Preferences for Qcad.

Graphic View

Tick "Show large crosshairs" to enable the crosshairs of the mouse cursor to extend to the drawing borders.

Sometimes it's useful to limit the number of preview entities. If you experience performance problems when moving or inserting entities in a drawing, please reduce the number of preview entities shown. You can set the value to 0 to disable the preview.

Paths

On the "Paths" tab, you can configure individual paths where QCad will search for translations files, hatch patterns, fonts, scripts, and part libraries (Figure 66b). You can enter multiple paths by separating them with a semicolon (;).

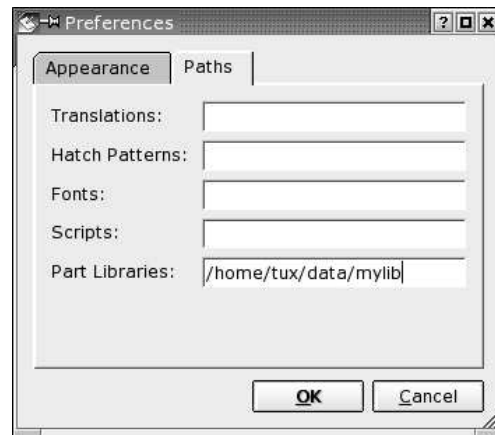


Figure 67: Configuration of individual paths.

Drawing Preferences

The drawing preferences dialog applies to the current drawing only. It's used to set the drawing units, paper size, dimensioning preferences, etc.

Edit - Drawing Preferences

Paper

The paper settings are only relevant when printing a drawing. The paper format and orientation that you choose in this dialog will be applied to the print preview (Figure 68).

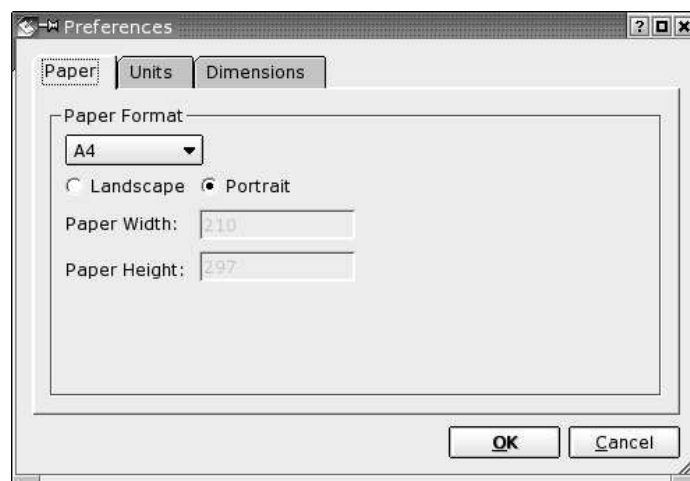


Figure 68: Paper settings for the current drawing.

Units

The chosen units and unit formats for lengths and angles are used for co-ordinates displayed in the statusbar, dimension labels and to scale inserted parts from the part library correctly. See [Appendix - Drawing Units](#) for a detailed overview of the available units and formats (Figure 69).

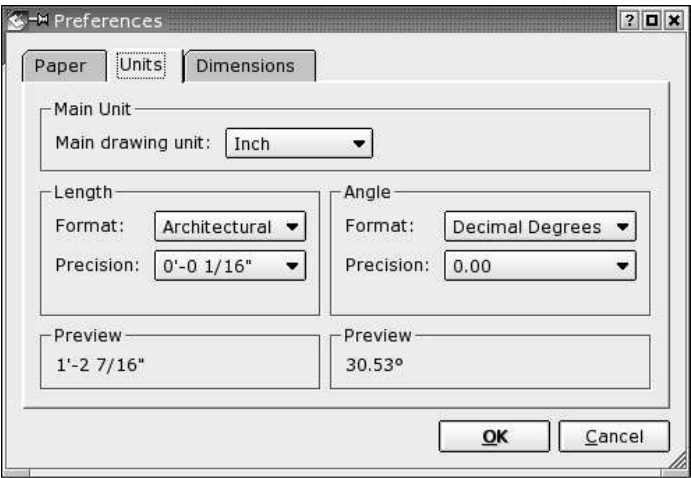


Figure 69: Unit and unit format.

Dimensions

On this tab you can change the measures for the dimension labels, the dimension arrow size and other meaasures used for dimensioning (Figure 70).

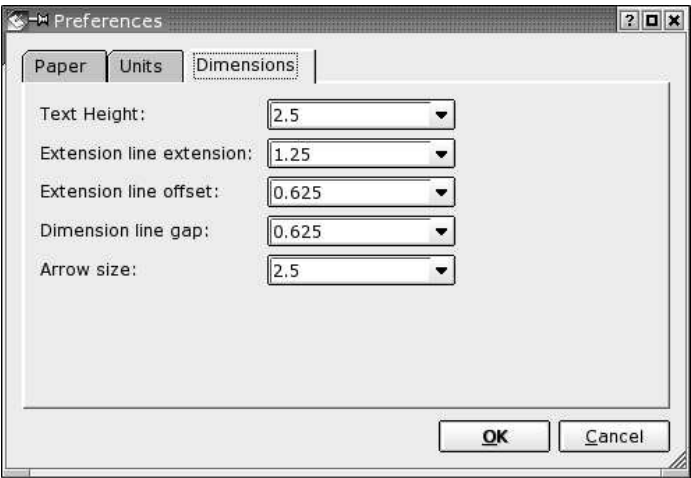


Figure 70: Dimension preferences.

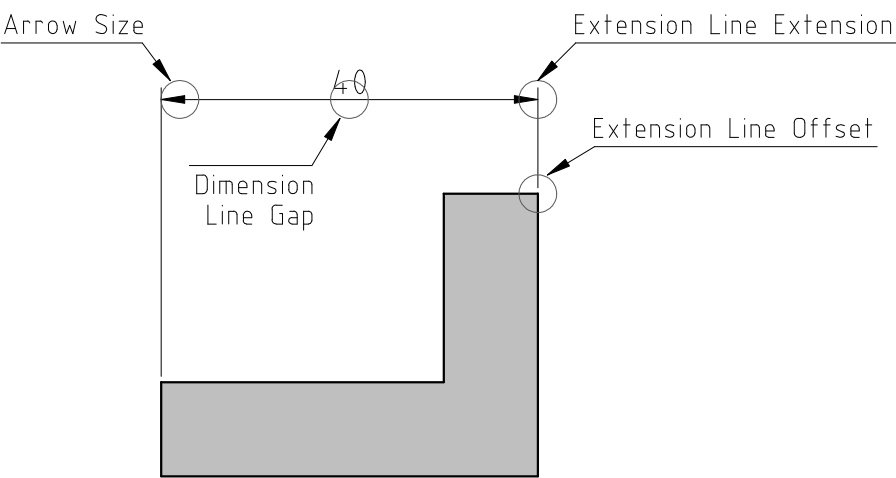


Figure 71: Dimension measures.

Appendix

Hotkeys

This is an overview of all hotkey combinations that can be used to trigger commands more efficiently. Two-letter combinations can only be entered if the command line has no keyboard focus, otherwise enter must be pressed after entering them. Two-letter combinations must be entered within a period of maximal 2 seconds. If you mistyped a two-letter combination, please press Escape or wait two seconds and start again.

Some commands have more than one key combination. That's mostly for compatibility with other programs.

In addition to the hotkey combinations listed below, the standard way of handling keys is used in dialogs. Under Windows and Unix operating systems, you can use Alt-Key combinations to press buttons (e.g. Alt-O for OK). If you're using Mac OS X this might be Option-O or Alt-O depending on your keyboard.

Command Line

Space	Activate command line
Escape	Deactivate command line or go one step back in the current action or show the main menu if no actions are active.
Arrow	Recall the last command inputs
Up / Down	
Tab	Show all currently available commands or complete the first letters of a command into the full command

File Handling

Ctrl-N	Create a new drawing
Ctrl-O	Open a drawing
Ctrl-S	Save current drawing
Ctrl-W	Close current drawing
Ctrl-P	Print current drawing
Ctrl-Q	Quit application

Basic Editing

oo, Ctrl-Z	Undo (oops)
uu, Ctrl-Shift-Z	Redo
Ctrl-X	Cut
Ctrl-C	Copy
Ctrl-P	Paste

View

rd	Redraw
zw	Zoom Window
za	Auto Zoom
zi, +	Zoom In
zo, -	Zoom Out
zp	Pan Zoom
zv	Back to previous view

Layer Handling

fr*	Freeze all layers
th*	Defreeze (thaw) all layers

Snapping

os	Free positioning
sg	Snap to grid
se	Snap to endpoints
sn	Snap to centers
sm	Snap to midpoints
si	Snap to intersections
np	Snap to nearest point on entity

Construction

po	Point
li	Line
re	Rectangle
rp	Regular Polygon
ci	Circle
c2	2 Point Circle
c3	3 Point Circle
ar	Arc
a3	3 Point Arc
ep	Ellipse
tx, mt	Text

Dimensioning

da	Aligned Dimension
dh	Horizontal Dimension
dv	Vertical Dimension
dr	Linear (rotated) dimension
ld	Leader

Entity Selection

tn, Ctrl-K	Select all
Ctrl-A	Select all

Modification

ch	Bevel (chamfer)
rm, xt	Trim, extend
mt	Multi line trim, extend
mv	Move
mi	Mirror
sz	Scale
ss	Stretch
er	Erase
xp, ex	Explode

Fonts

QCad comes with a variety of fonts. These fonts were especially designed for use within CAD applications.

A B C a b c

Figure 72: Courier

A B C a b c

Figure 73: Cursive

A B C a b c

Figure 74: Cyrilc2

A B C a b c

Figure 75: Cyrillic

A B C a b c

Figure 76: Cyrillic_ol

A B C a b c

Figure 77: Gothgnt



Figure 78: Gothgrt



Figure 79: Gothitt



Figure 80: Greek_ol

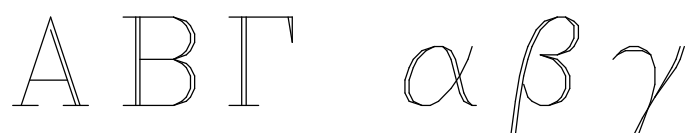


Figure 81: Greekc



Figure 82: Greekcs



Figure 83: Greekp

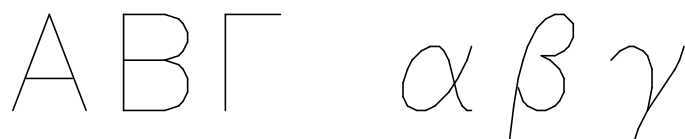


Figure 84: Greeks



Figure 85: Iso8859-11

A B C a b c

Figure 86: Italicc

A B C a b c

Figure 87: Italiccs

A B C a b c

Figure 88: Italict

A B C a b c

Figure 89: Kochigothic

A B C a b c

Figure 90: Kochimincho

A B C a b c

Figure 91: Normal

A B C a b c

Figure 92: Normallatin1

A B C a b c

Figure 93: Normallatin2

A B C a b c

Figure 94: Romanc

A B C a b c

Figure 95: Romancs

A B C a b c

Figure 96: Romand

A B C a b c

Figure 97: Romanp

A B C a b c

Figure 98: Romans

A B C a b c

Figure 99: Romans2

A B C a b c

Figure 100: Romant

A B C a b c

Figure 101: Scriptc

A B C a b c

Figure 102: Scripts

A B C a b c

Figure 103: Standard

A B X α β χ

Figure 104: Symbol



Figure 105: Symbol_astro



Figure 106: Symbol_misc1

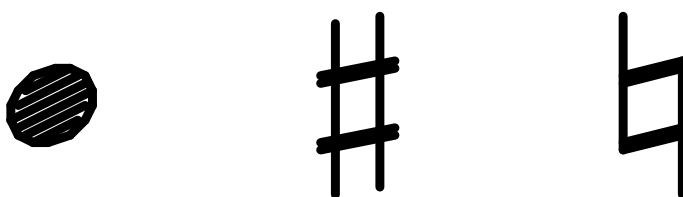


Figure 107: Symbol_misc2

A B C a b c л Д Й 兩 並 中 井

Figure 108: Unicode

Patterns

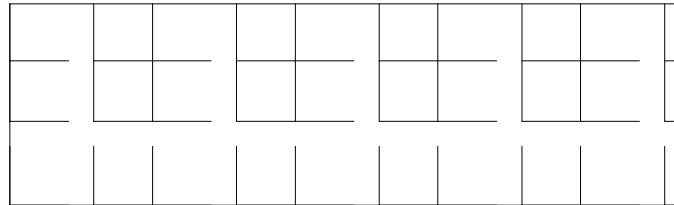


Figure 109: Angle

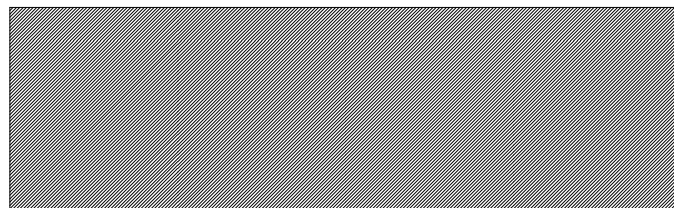


Figure 110: Ansi31

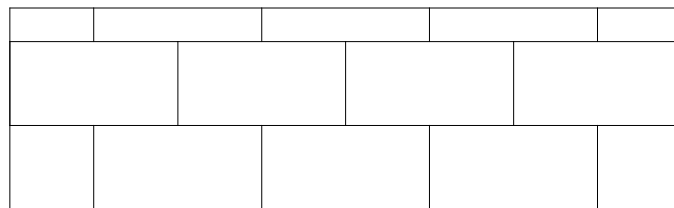


Figure 111: Ar-b816

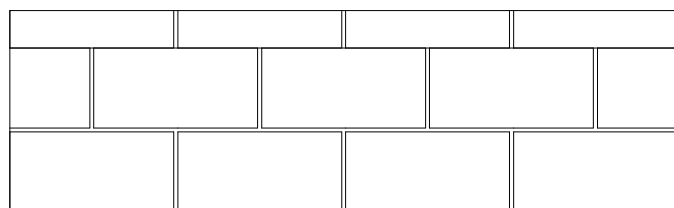


Figure 112: Ar-b816c

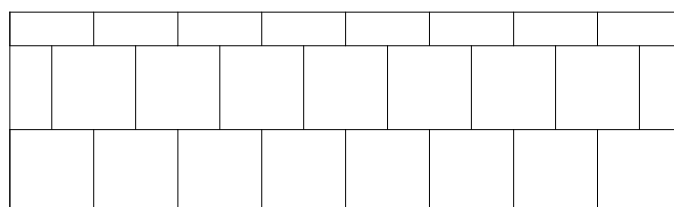


Figure 113: Ar-b88

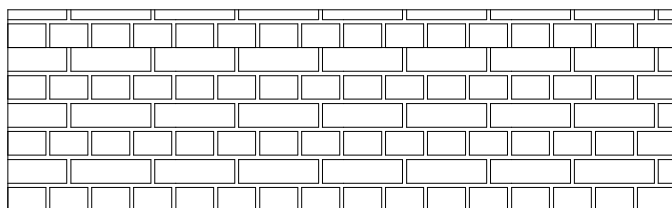


Figure 114: Ar-brelm

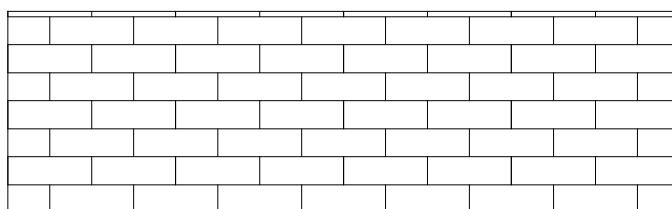


Figure 115: Ar-brstd

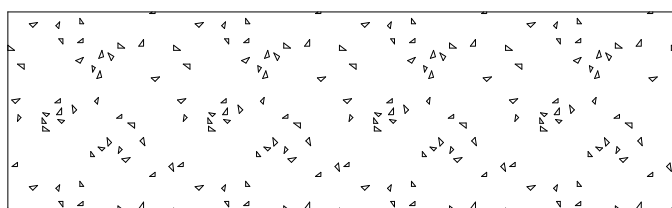


Figure 116: Ar-conc

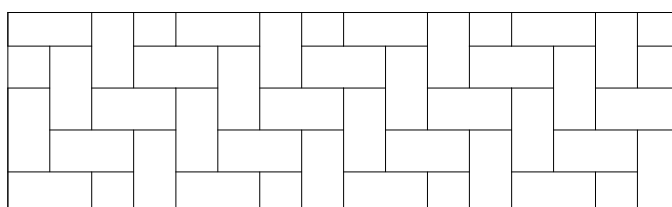


Figure 117: Ar-hbone

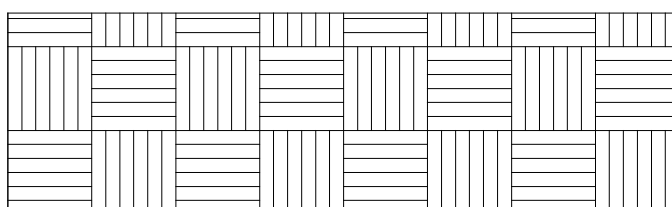


Figure 118: Ar-parq1

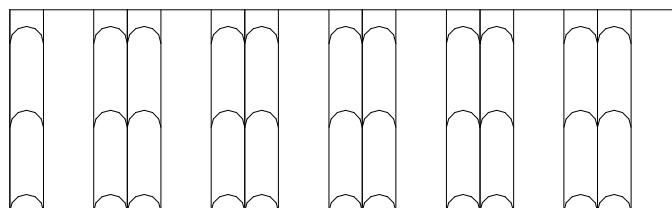


Figure 119: Ar-roof

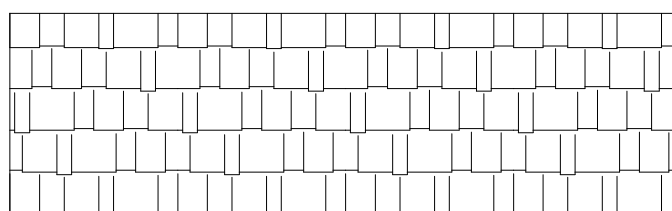


Figure 120: Ar-rshke

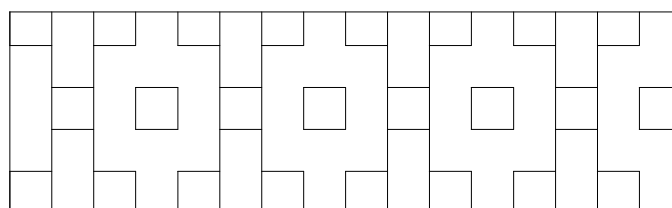


Figure 121: Box

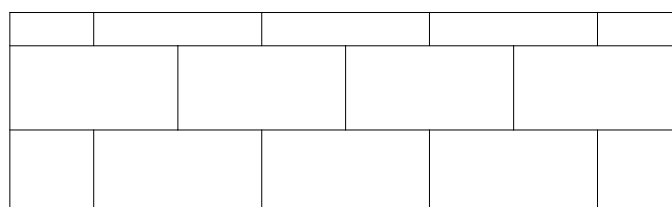


Figure 122: Brick

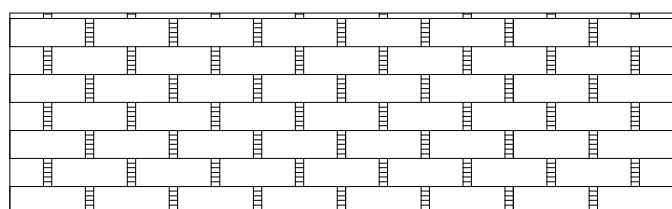


Figure 123: Brstone

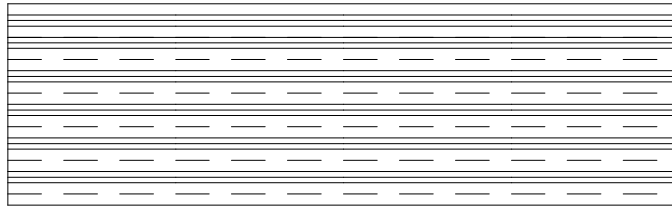


Figure 124: Clay

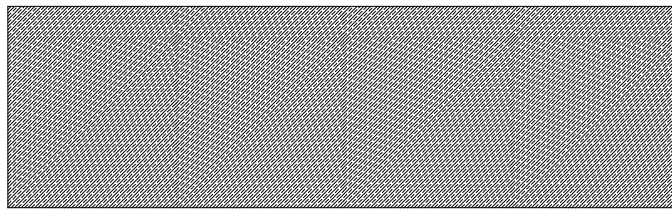


Figure 125: Concrete

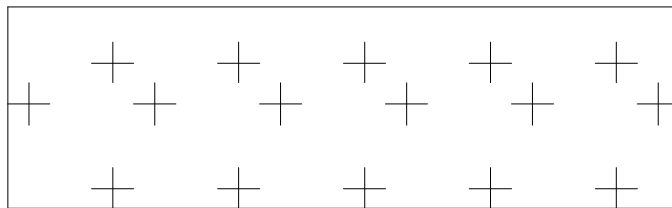


Figure 126: Cross

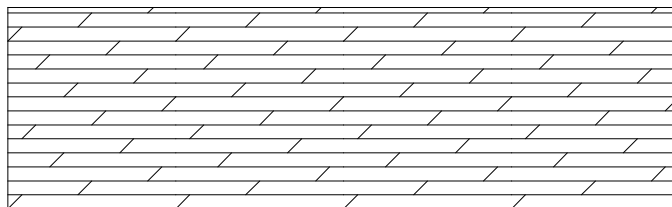


Figure 127: Dolmit

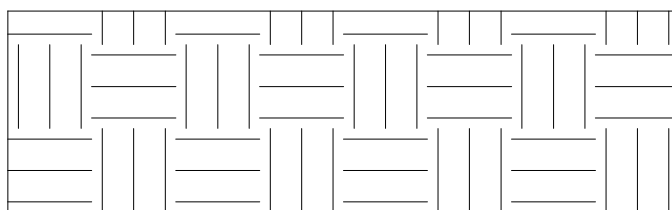


Figure 128: Earth

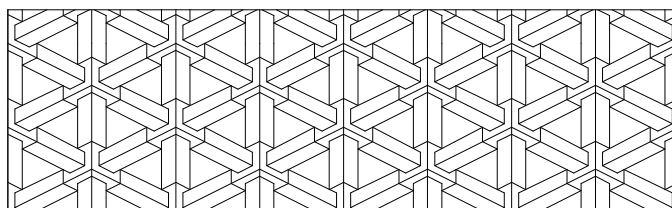


Figure 129: Escher

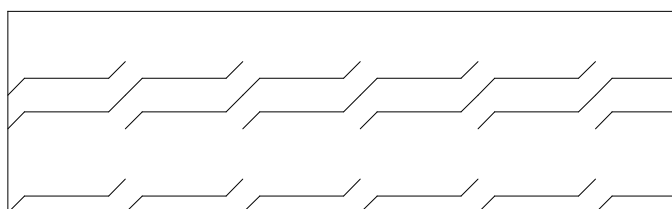


Figure 130: Flex

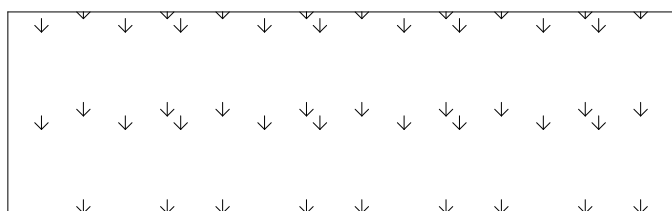


Figure 131: Grass

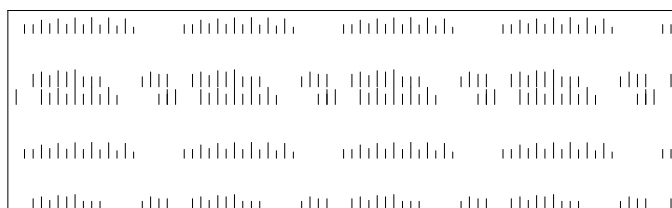


Figure 132: Grass_b

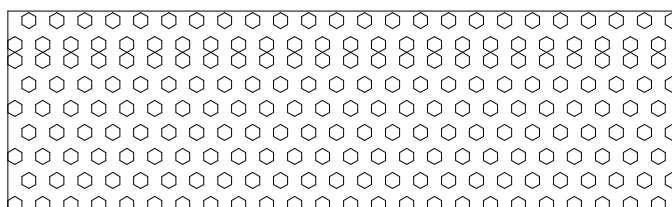


Figure 133: Hex

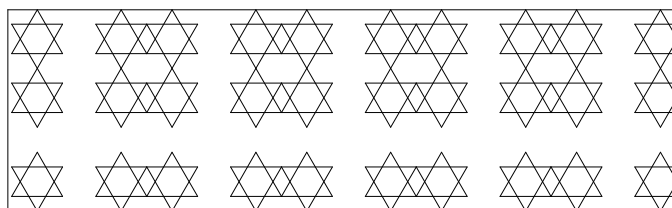


Figure 134: Hexagon_a

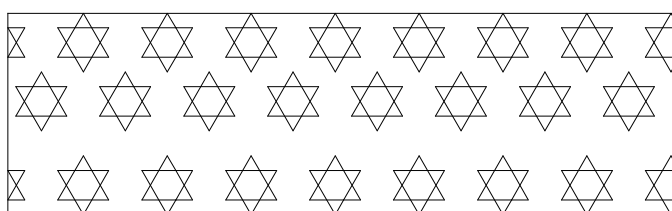


Figure 135: Hexagon_b

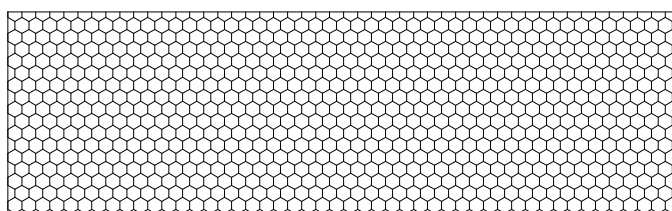


Figure 136: Honeycomb

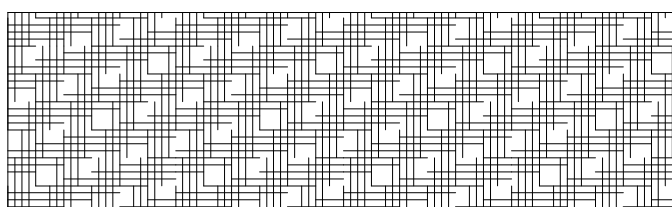


Figure 137: Hound

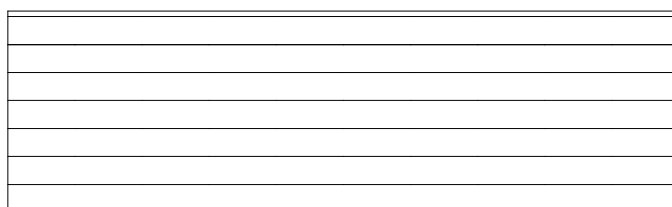


Figure 138: iso03w100

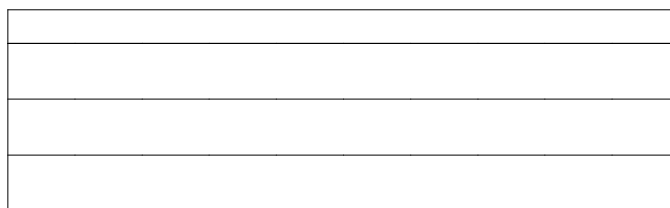


Figure 139: Iso03w100a

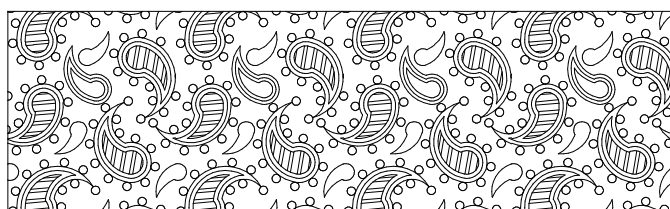


Figure 140: Paisley



Figure 141: Pantagon_a



Figure 142: Pantagon_b

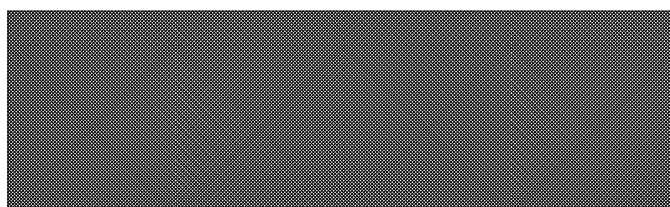


Figure 143: Plastic

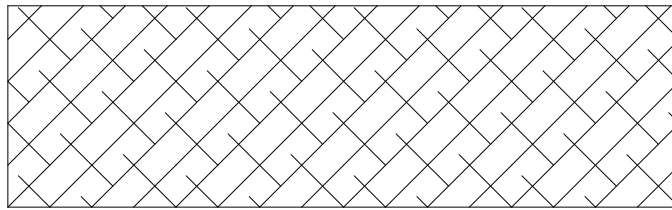


Figure 144: Sacncr

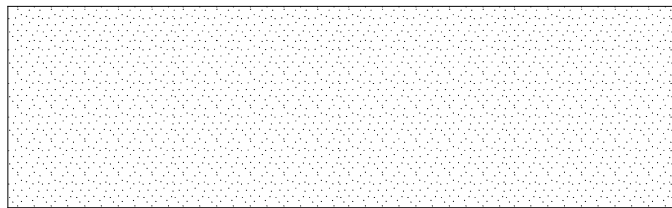


Figure 145: Sand

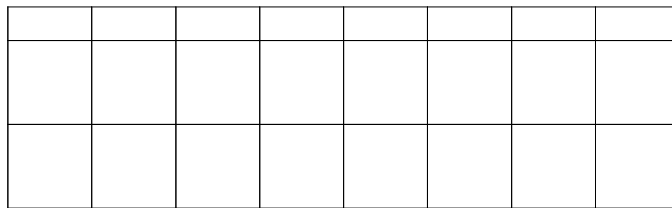


Figure 146: Square

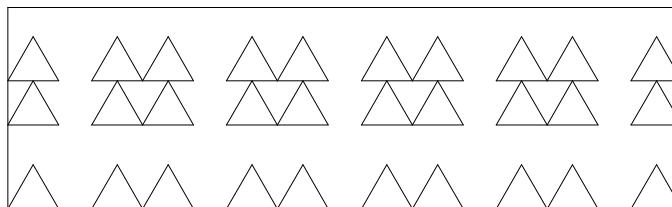


Figure 147: Triangle_a

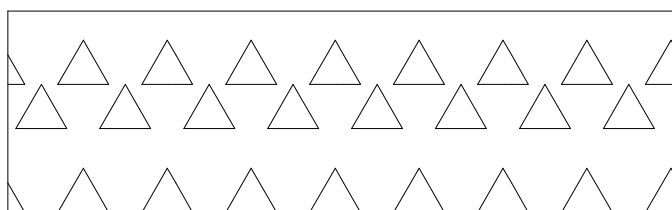


Figure 148: Triangle_b

Mathematical Expressions

QCAD comes with a built in mathematical expression parser called 'fparser'. The parser was written by [Juha Nieminen 'Warp'](#) of the [Tampere University of Technology](#) Finland. Whenever there are input fields which expect a real value from the user, you can type any expression that is recognized by the parser instead. This way you always have a powerful calculator at hand.

Supported Constants

<code>pi</code>	3.14159265
-----------------	------------

Supported Functions

The following table lists all functions supported by the expression parser. You can also find the same table in the [original documentation of 'fparser'](#).

<code>abs (A)</code>	Absolute value of A.
<code>acos (A)</code>	Arc-cosine of A. Returns the angle, measured in degrees, whose cosine is A.
<code>acosh (A)</code>	Same as <code>acos()</code> but for hyperbolic cosine.
<code>asin (A)</code>	Arc-sine of A. Returns the angle, measured in degrees, whose sine is A.
<code>asinh (A)</code>	Same as <code>asin()</code> but for hyperbolic sine.
<code>atan (A)</code>	Arc-tangent of (A). Returns the angle, measured in degrees, whose tangent is (A).
<code>atan2 (A, B)</code>	Arc-tangent of A/B. The two main differences to <code>atan()</code> is that it will return the right angle depending on the signs of A and B (<code>atan()</code> can only return values between $-\pi/2$ and $\pi/2$), and that the return value of $\pi/2$ and $-\pi/2$ are possible.
<code>atanh (A)</code>	Same as <code>atan()</code> but for hyperbolic tangent.
<code>ceil (A)</code>	Ceiling of A. Returns the smallest integer greater than A. Rounds up to the next higher integer.

<code>cos (A)</code>	Cosine of A. Returns the cosine of the angle A, where A is measured in degrees.
<code>cosh (A)</code>	Same as <code>cos()</code> but for hyperbolic cosine.
<code>cot (A)</code>	Cotangent of A (equivalent to $1/\tan(A)$).
<code>csc (A)</code>	Cosecant of A (equivalent to $1/\sin(A)$).
<code>eval (...)</code>	This a recursive call to the function to be evaluated. The number of parameters must be the same as the number of parameters taken by the function. Usually called inside <code>if()</code> to avoid infinite recursion.
<code>exp (A)</code>	Exponential of A. Returns the value of e raised to the power A where e is the base of the natural logarithm, i.e. the non-repeating value approximately equal to 2.71828182846.
<code>floor (A)</code>	Floor of A. Returns the largest integer less than A. Rounds down to the next lower integer.
<code>if (A, B, C)</code>	If <code>int(A)</code> differs from 0, the return value of this function is B, else C. Only the parameter which needs to be evaluated is evaluated, the other parameter is skipped; this makes it safe to use <code>eval()</code> in them.
<code>int (A)</code>	Rounds A to the closest integer. 0.5 is rounded to 1.
<code>log (A)</code>	Natural (base e) logarithm of A.
<code>log10 (A)</code>	Base 10 logarithm of A.
<code>max (A, B)</code>	If A > B, the result is A, else B.
<code>min (A, B)</code>	If A < B, the result is A, else B.
<code>sec (A)</code>	Secant of A (equivalent to $1/\cos(A)$).

<code>sin(A)</code>	Sine of A. Returns the sine of the angle A, where A is measured in degrees.
<code>sinh(A)</code>	Same as <code>sin()</code> but for hyperbolic sine.
<code>sqrt(A)</code>	Square root of A. Returns the value whose square is A.
<code>tan(A)</code>	Tangent of A. Returns the tangent of the angle A, where A is measured in degrees.
<code>tanh(A)</code>	Same as <code>tan()</code> but for hyperbolic tangent.

Drawing Units

QCAD supports various units for length and angles. Craftsmen from different professions are used to different ways of displaying numbers. QCAD supports the most commonly used formats for displaying lengths and angles and allows you to adjust the precision as required.

Please refer also to chapter [Drawing Options](#) to learn how to change the unit of a drawing.

Supported Length Units

Metric Units

Angstrom (10 E-10m),
Nanometers (10 E-09m),
Microns (10 E-06m),
Millimeters (0.001m),
Centimeters (0.01m),
Decimeters (0.1m),
Meters (1m),
Decameters (10m),
Hectometers (100m),
Kilometers (1000m),
Gigameters (10 E06m)

Imperial Units

Microinches (1/1 000 000 Inch),
Mil (1/1000 Inch),
Inch,
Feet (12 Inches),
Yards (3 Feet),
Mile (1 760 Yards)

Other Length Units

Astro (1.49600 * 10 E11 Meters),
Lightyear (9.4605 * 10 E15 Meters),
Parsec (3.0857 * 10 E16 Meters)

Display Formats for Lengths

Formats for All Length

Decimal (e.g. 3.5)
Scientific (e.g. 3.5E+01)
Fractional (e.g. 3 1/2)

Additional Formats for Lengths in Inches

Engineering (e.g. 2'-3.56")
Architectural (e.g. 2'-3 5/64")

Supported Angle Units

Degrees (full circle 360°)

Radians (full circle 2π)

Gradians (full circle 400g)

Display Formats for Angles

Decimal Degrees (e.g. 36.26°)

Degrees / Minutes / Seconds (e.g. 36° 15' 24")

Radians (e.g. 0.785r)

Gradians (e.g. 100g)

Supported Paper Formats

Paper formats are required to determine the scaling when printing a drawing. This chapter lists all supported formats and their size (width, height) in Millimeters.

Metric (ISO)

A0: 841.0 x 1189.0
A1: 594.0 x 841.0
A2: 420.0 x 594.0
A3: 297.0 x 420.0
A4: 210.0 x 297.0
A5: 148.0 x 210.0
A6: 105.0 x 148.0
A7: 74.0 x 105.0
A8: 52.0 x 74.0
A9: 37.0 x 52.0
A10: 26.0 x 37.0
B0: 1000.0 x 1414.0
B1: 707.0 x 1000.0
B2: 500.0 x 707.0
B3: 353.0 x 500.0
B4: 250.0 x 353.0
B5: 176.0 x 250.0
B6: 125.0 x 176.0
B7: 88.0 x 125.0
B8: 62.0 x 88.0
B9: 44.0 x 62.0
B10: 31.0 x 44.0

Other Formats

Letter: 215.9 x 279.4
Legal: 215.9 x 355.6
Executiv: 190.5 x 254.0
C5E: 163.0 x 229.0
Comm10: 105.0 x 241.0
DLE: 110.0 x 220.0
Folio: 210.0 x 330.0
Ledger: 432.0 x 279.0
Tabloid: 279.0 x 432.0

Migration from QCad 1

This chapter lists known problems when migrating from QCad 1 to QCad 2 and should be a help to those who convert drawings from QCad 1 to QCad 2.

DXF Compatibility

When opening DXF drawings made with QCad 1, you might experience problems with dimensions, hatches and texts. This is because the DXF format produced by QCad 1 is not 100% standard compliant. You can import DXF drawings from QCad 1 by changing the format in the File Open dialog to 'QCAD 1.x file (*.dxf *.DXF)' as shown in Figure 149.

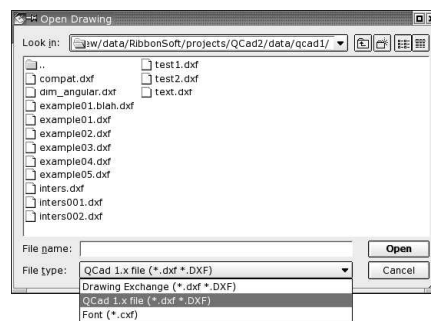


Figure 149: Opening QCad 1 drawings.

Rounded texts are no longer supported in QCad 2 for better compatibility with other programs. You can import rounded texts by splitting them up into single elements in QCad 1.

Layer Handling

Changing layer attributes in QCad 1 had no effect on already constructed entities. In QCad 2, changing layer attributes can also change the attributes of all entities that are on that layer and have attributes that are set to "ByLayer".

When migrating from QCad 1 to QCad 2, you might find that the attributes of the entities in old drawings are no longer appropriate. Usually it's recommendable to set all attributes to "ByLayer". To do that, choose **Attributes** from the **Modify** menu. Then select all entities and click the 'Continue' button. In the dialog, change all attributes to 'By Layer' as shown in Figure 150.

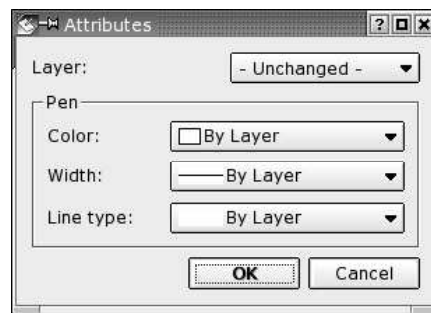


Figure 150: Changing all attributes to 'By Layer'.

Bibliography

[DXF] <http://www.autodesk.com/techpubs/autocad/acad2000/dxf> Autodesk DXF Reference

Index

- 2
- 2 Points Circle 68
- 2D 8
- 3
- 3 Points Arc 66
- 3 Points Circle 68
- 3D 8
- A**
- Aligned Dimension 76
- Angle Bisector 60
- Angle 13,58,65,110
- Angular Dimension 80
- Application Window 17
- Arc - 3 Points 66
- Arc - Center Point
- Angles 65
- Arc - Concentric 66
- Arc 65,66
- Area Patterning 82
- Area 82
- Arrow 81
- Attribute 10
- Attributes 106
- Auto zoom 33
- B**
- Bevel 102
- Bisector 60
- Bitmap 23
- Bitmaps 84
- Block List 18
- Block 11
- BMP 84
- C**
- CAD Toolbar 17
- CAD 8
- Cartesian
- Co-ordinates 12
- Center 65,67
- Centers 47
- Chamfer 102
- Circle - 2 Points 68
- Circle - 3 Points 68
- Circle - Center Point 67
- Circle - Center
- Radius 67
- Circle - Concentric 69
- Circle - Parallel 69
- Circle 67,68
- Close File 23
- Co-ordinate Systems 12
- Command Line -
- Hotkeys 116
- Command Line 21
- Command Widget 18
- Commands 21
- Computer-Aided
- Design 8
- Concentric Arc 66
- Concentric Circle 69
- Construction -
- Hotkeys 118
- Copy 28,91
- Create Drawing 22
- Crosshairs 111
- Cursor 111
- Cut 28,104
- D**
- Delete Entities 14
- Delete 107
- Deletion 14
- Deselect - Contour 86
- Deselect Intersected
- Entities 88
- Deselecting Area 87
- Deselecting All 85
- Deselecting Range 87
- Deselecting
- Rectangle 87
- Deselecting Window 87
- Diametric Dimension 79
- Dimension - Aligned 76
- Dimension - Angular 80
- Dimension -
- Diametric 79
- Dimension -
- Horizontal 78
- Dimension - Leader 81
- Dimension - Linear 77
- Dimension - Radial 78
- Dimension - Vertical 78
- Dimension 75
- Dimensioning -
- Hotkeys 118
- Display 15
- Distance from
- Endpoint 49
- Distance 109,110
- Divide 104
- Draft 31
- Drawing Area 17
- Drawing
- Preferences 113
- Drawing scale 15
- Drawing Units 135
- Duplicate 14
- DXF 22
- E**
- Edit - Copy 28
- Edit - Cut 28
- Edit - Hotkeys 117
- Edit - Paste 29

Edit - Redo 27	Info 109	Modify - Explode 108
Edit - Undo 26	Insert 11	Modify - Lengthen 101
Edit 90	Intersection 50,51	Modify - Mirror 94
Ellipse Arc 71	Inverting Selection 88	Modify - Move and Rotate 95
Ellipse 70		Modify - Move 91
Endpoints 45	J	Modify - Properties 105
Entity Properties 105	JPEG 23,84	Modify - Rotate 2 97
Entity Selection 14,85		Modify - Rotate 92
Entity Snap 43	L	Modify - Round 103
Entity 10	Language 111	Modify - Scale 93
Explode 108	Layer - Hotkeys 117	Modify - Stretch 105
Export 23	Layer List 18	Modify - Trim Two 100
Expressions 132	Layer 10	Modify - Trim 99
Extend 99,100	Leader 81	Move and Rotate 95
	Lengthen 101	Move Relative Zero 55
F	Line - Angle 58	Move 91
File - Close 23	Line - Bisector 60	
File - New 22	Line - Freehand 63	N
File - Open 22	Line - Horizontal 58	New Drawing 22
File - Print Preview 24	Line - Orthogonal 62	
File - Print 25	Line - Parallel 59	O
File - Save As 23	Line - Polygon 64	Object Snap 43
File - Save 23	Line - Rectangle 59	Object Snapping 14
File Handling -	Line - Relative Angle 62	On Entity 46
Hotkeys 116	Line - Sequence 57	Open Drawing 22
File 22,23	Line - Tangent 61	Option Toolbar 18
Fill 82	Line - Vertical 58	Options 111,113
Fillet 103	Line 57,58	Origin 12
Fonts 119	Linear Dimension 77	Orthogonal 62
Format 137	Lock Relative Zero 55	
Free Positioning 44		P
Freehand Line 63		Page Format 137
	M	Page 137
G	Mathematical	Panning 30,34
General	Expressions 132	Paper Format 137
Preferences 111	Measure Angle 110	Paper 137
GIF 23	Measure	Parallel Line 59
Graphic Object 10	Distance 109,110	Parallel 59,66
Grid Snapping 14	Measure 109	Paste 29
Grid 31,44	Menu 17	Paths 112
Group of entities 11	Middles 48	Pattern 82
GUI 17	Migration 138	Patterning 82
	Mirror 94	Patterns 124
H	Modification -	PDF 25
Hatch Patterns 124	Hotkeys 118	Platforms 9
Hatches 82	Modification 90	PNG 23,84
Horizontal Dimension 78	Modifications 14	Point 56
Horizontal 58	Modify - Attributes 106	Pointer 81
Hotkeys 116	Modify - Bevel 102	Polar Co-ordinates 13
	Modify - Delete 107	
I	Modify - Divide 104	
Images 84		

Polygon 64
 Postscript 24
 Preface 8
 Preferences 111,113
 Preview 24,111
 Previous 34
 Print Preview 24
 Print 24,25
 Printing 24
 Properties 105
 PS 24,25

Q
 QCad 1 138
 Qt 9

R
 Radial Dimension 78
 Radius 67
 Raster Graphic 23
 Raster Images 84
 Rectangle 59
 Redo 27
 Redraw 31
 Regen 31
 Relative Angle 62
 Relative zero 12
 Relative Zero 55
 Remove 107
 Restrict Horizontally 53
 Restrict Nothing 53
 Restrict Orthogonally 53
 Restrict Vertically 54
 Rotate and Counter-rotate 97
 Rotate 92
 Round 103

S
 Save File 23
 Scale 93
 Scissors 104
 Scrollbars 30
 Scrolling 30
 Select - Contour 86
 Select - Deselect All 85
 Select - Invert Selection 88
 Select - Select All 85
 Select - Select Entity 86

Select Entities 14
 Select Intersected Entities 89
 Select Objects 14
 Selecting Area 87
 Selecting All 85
 Selecting Contours 86
 Selecting Range 87
 Selecting Rectangle 87
 Selecting Single Entities 86
 Selecting Window 87
 Selection - Hotkeys 118
 Selection 85
 Sequence of Lines 57
 Set Relative Zero 55
 Sheet size 15
 Shorten 101
 Snap - Centers 47
 Snap - Distance from Endpoint 49
 Snap - Endpoints 45
 Snap - Free 44
 Snap - Grid 44
 Snap - Intersection Manually 51
 Snap - Intersection 50
 Snap - Lock Relative Zero 55
 Snap - Middle 48
 Snap - On Entity 46
 Snap - Relative Zero 55
 Snap - Restrict Horizontally 53
 Snap - Restrict Nothing 53
 Snap - Restrict Orthogonally 53
 Snap - Restrict Vertically 54
 Snapping - Hotkeys 117
 Snapping 14,43
 Solid Fill 82
 Status Bar 18
 Stretch 105

T
 Tagging 85
 Tangent 61
 Text 72

Three Points Arc 66
 Three Points Circle 68
 Toggle Draft Mode 31
 Toggle Grid 31
 Toolbar 17
 Transform 14
 Transformations 14
 Translate 91
 Trim 99,100
 Trolltech 9
 Two Points Circle 68

U
 Undo 26
 Units 135
 Unlock Relative Zero 55
 User Interface 17

V
 Vertical Dimension 78
 Vertical 58
 View - Command Line 21
 View - Draft 31
 View - Grid 31
 View - Hotkeys 117
 View - Pan Zoom 34
 View - Previous 34
 View - Views - Block List 18
 View - Views - Command Widget 18
 View - Views - Layer List 18
 View - Views - Option Toolbar 18
 View - Views - Statusbar 18
 View 31
 Viewing 15
 Visible area 15

W
 Window Deselection 87
 Window Selection 87
 Window zoom 33

X
 X-Ordinate 12
 XBM 23,84
 XPM 23,84

Y

Y-Ordinate 12

Z

Zero 12

Zoom auto 33

Zoom in 32

Zoom out 32

Zoom Pan 34

Zoom Previous 34

Zoom window 33

Zoom 32

Zooming 15