QCad User Reference Manual

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1. Abstract

This user reference manual for QCad 2.1 is intended to give the reader a brief overview over all functions and features of QCad. It is not a beginners guide to CAD in general or to QCad. The instructions explain each tool but not how to create a complete drawing from scratch.

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The source code of the QCad Community edition for Linux, other Unix or X11 Systems and Mac OS X
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3. Preface

3.1. CAD - For the Uninitiated

If you take a look around you, most man-made objects you see once started off as a drawing in a CAD program. Your ball pen, your desk, your furniture, 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 is essential to realize that CAD applications differ from simple painting or drawing programs. Drawings made in CAD applications represent real-world objects in their exact original measures. The most important aspects are accuracy and to provide a level of details that defines the object well enough that it can be manufactured based on the drawing. If you are used to simple drawing programs, you will most likely find it inconvenient that many steps are required to perform an operation such as moving part of the drawing. This is part of the idea, that everything is done accurately. The drawing does not only have to look accurate on the screen - it has to be accurate as a model of the real-world object. Moving an entity around means moving it from an exactly defined location to another exactly defined position.

Once you are used to this approach, you will most likle understand why this approach is superior to using other drawing applications for many tasks.

3.2. How can I get QCad?

You can get the latest version of QCad from RibbonSoft.com. On this web site you can also find more documentation, mailing lists and many other resources for QCad.

3.3. 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 needs. If you are looking for a 3D modeling CAD application, you will definitely have to look somewhere else. QCad is a 2D CAD program. That means that everything you draw is projected onto a plane. However, this does not imply that QCad is only suitable to represent 2D objects. A couple of 2D drawings which show 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 prefer QCad over other CAD applications are its simplicity, the user friendly interface, its features or 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 its source code is available free of charge and if you are a programmer you can even 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 many of them offer more features than QCad. But most of them are bound to one single platform, hard to learn or are simply not affordable for many potential users. 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 has been designed to be the CAD for the rest of us.

3.4. QCad and Qt

Qt is the cross platform C++ application framework that QCad is based on. Much of the portability and user friendlyness of QCad would not have been possible without the work of Trolltech (trolltech.com), the company who develops Qt.

3.5. Scope of This Manual

This manual is designed to give an overview over the features of QCad. Its main objective is to be complete and cover all features and commands that QCad offers.

3.6. Who Is Behind QCad?

QCad is a product of RibbonSoft GmbH, a software engineering company based in Switzerland. Many people from the open source community have also contributed to QCad and it would not be what it is today without the help of many voluntary translators, developers and the feedback of QCad users. Please refer also to the company homepage of RibbonSoft for more information: RibbonSoft.com.

4. Basic CAD Concepts

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

4.1. 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, dimensions, hatches and splines.

4.2. Attributes

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

4.3. Layers

A basic concept of computer aided drafting is the use of layers to organize and structure a drawing. Every entity of a drawing is on exactly one layer and each layer can contain any number of entities. Typically, all entities with a common 'function' or with common attributes are constructed on the same layer. Every layer has attributes (color, line width, line style) and the entities on a layer usually inherit their attributes from the layer they are placed on.

Figure 1 shows an example drawing that uses layers. All dimensions of the drawing are placed on a layer named 'dimensions'. The color of all 'dimension' entities isdefined by the layer and can be easily changed by changing the color of the layer 'dimensions'. The drawing can also be shown without any dimensions by hiding layer 'dimensions'.

Historically, in manual drafting, a similar approach has been 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 the final drawing.
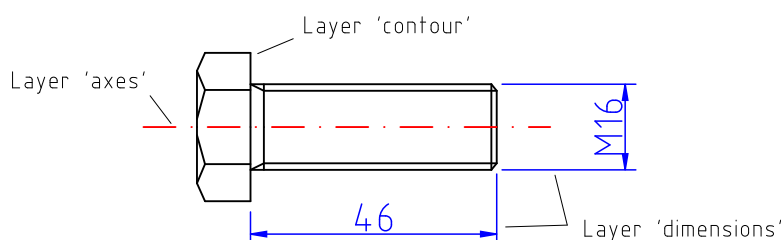
Figure 1: Example drawing with layers.

4.4. Blocks

A block is a named group of entities. Blocks can be inserted into the same drawing more than once at different locations, with different attributes, scaling factors and rotation angles (see Figure 2). Such instances of a block are usually called 'inserts'. Inserts have attributes just like other entities. An entity that is part of an insert can have its own attributes, inherit its attributes from the layer it is placed on, or inherit them from the insert it is part of. Once created, inserts are still dependent on the block they instantiate. The power of inserts is, that you can modify a block and all inserts that were previously made from that block will be updated to reflect the changes.
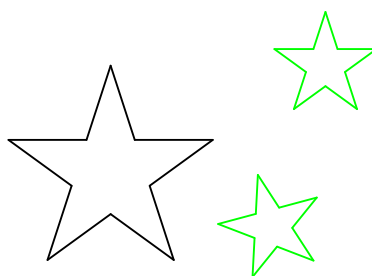
Figure 2: Three instances of a block (inserts) with different attributes, angles and scaling factors.

4.5. 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. CAD systems offer many tools 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 have created it. This might be one of the more difficult things to learn when moving from paper to CAD. When working with a CAD system you will very often create lines that will not 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. Never hesitate to create an auxiliary construction if it helps you to define or verify a part of your drawing.
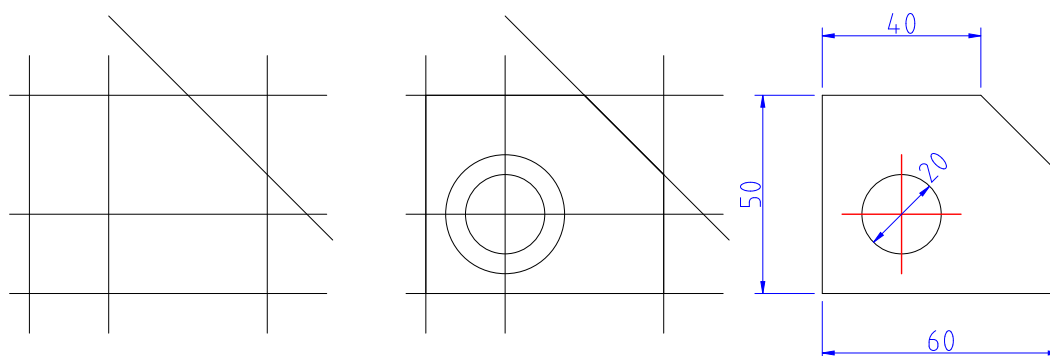
Figure 3: Three possible steps to quickly create a drawing using supportive constructions and the ability of a CAD system to change existing entities.

4.6. Coordinate Systems

A good understanding of how the most common coordinate systems work is absolutely essential if you are to make the best use of any CAD program. If you are not familiar with coordinates it is highly recommended that you take some time to familiarize yourself with this fundamental concept.

Origin

The origin (or origin point) of the drawing is the point where the X and Y axis cross each other. It is also the absolute zero point of the drawing.

Relative Zero Point

In addition to the absolute zero point, there is also a relative zero point in QCad. The relative zero point can be moved to any location by the user as a temporary reference for a local construction.

Cartesian Coordinates

The Cartesian coordinate system is the standard coordinate system that is usually used if no other system is specified. In the Cartesian coordinate system, the position of a point can be described by its distance from two axes, X and Y. Cartesian coordinates are usually written in the format:

`x-ordinate,y-ordinate`

For example the coordinate 3.5,7 is located 3.5 units to the right and 7 unit to the top of the origin.
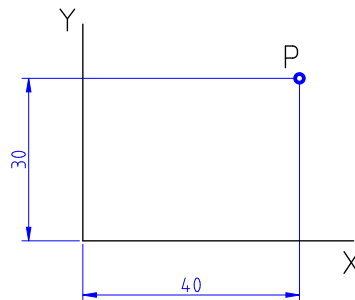
Figure 4: Absolute Cartesian coordinate 40,30.

Cartesian coordinates can also originate in a position different from the origin. In that case we talk about 'relative coordinates'. There is no standard notation for relative Cartesian coordinates, but in QCad they are written in the format:

`@x-ordinate,y-ordinate`
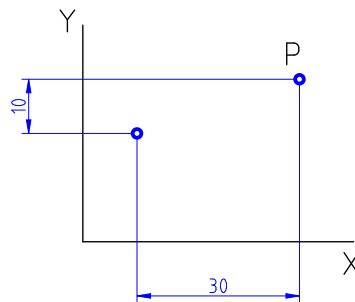
Figure 5: Relative Cartesian coordinate @30,10.

Polar Coordinates

Polar coordinates use a distance and an angle to describe the position of a point. The angle 0 is always pointing to the right on your screen (east). In QCad, polar coordinates are noted in the format:

`distance<angle`
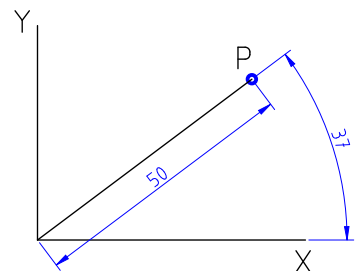
Figure 6: Absolute Polar coordinate 50<37.

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

```
@distance<angle
```
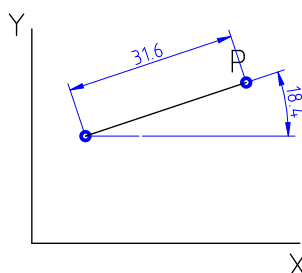

Figure 7: Relative Polar coordinate @31.6<18.4

4.7. Object and Grid Snapping

Whenever you need to specify a coordinate in QCad, you can use the snap functions. They allow you to precisely select grid points or significant points on existing objects such as endpoints, midpoints, centers or intersections between entities.

4.8. Basic Editing

Entity Construction

Entities can be added to a drawing with various drawing tools or by duplicating existing entities. To draw an entity means to define all points and parameters that define the entity, such as the endpoints of a line.

Entity Selection

Before an entity can be deleted, duplicated, or transformed, it must be selected. Entity selection is one of the most basic of CAD operations. However, selecting the right entities for an operation is not always trivial. QCad offers a wide variety of selection tools to quickly select groups of entities, entities within a range, connected entities, etc.

Deletion

Deleting an entity means to remove it from the drawing. In QCad, all features that have to do with deleting entities are grouped with the modification features.

Modifications

Existing entities can be modified in many ways. Basic modifications include translation, rotation, reflection, and scaling. These operations do not alter the characteristic geometry of the affected entities. Other modifications change an entity more fundamentally (for example trimming, extending or stretching)

4.9. 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, a drawing scale needs to be specified to fit the drawing on a paper. The drawing model itself is not affected by this and always remains in the original 1:1 scale.

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 is over the desired spot.

5. Starting a QCad Session

5.1. Linux / Unix Operating Systems

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

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

5.2. Mac OS X Operating System

Double-click on the QCad application icon in Finder.

5.3. Windows Operating Systems

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

6. The QCad User Interface

When you launch QCad, it comes up with the main application window of the graphical user interface. This chapter gives an overview over the different sections of the main application window.

6.1. Main Application Window

Some of the components of the main application window are also commonly available in other applications (e.g. the menubar or the toolbars). Other components are specific to CAD or even to QCad only. Figure 8 shows the names by which this manual refers to the individual components of the QCad user interface.
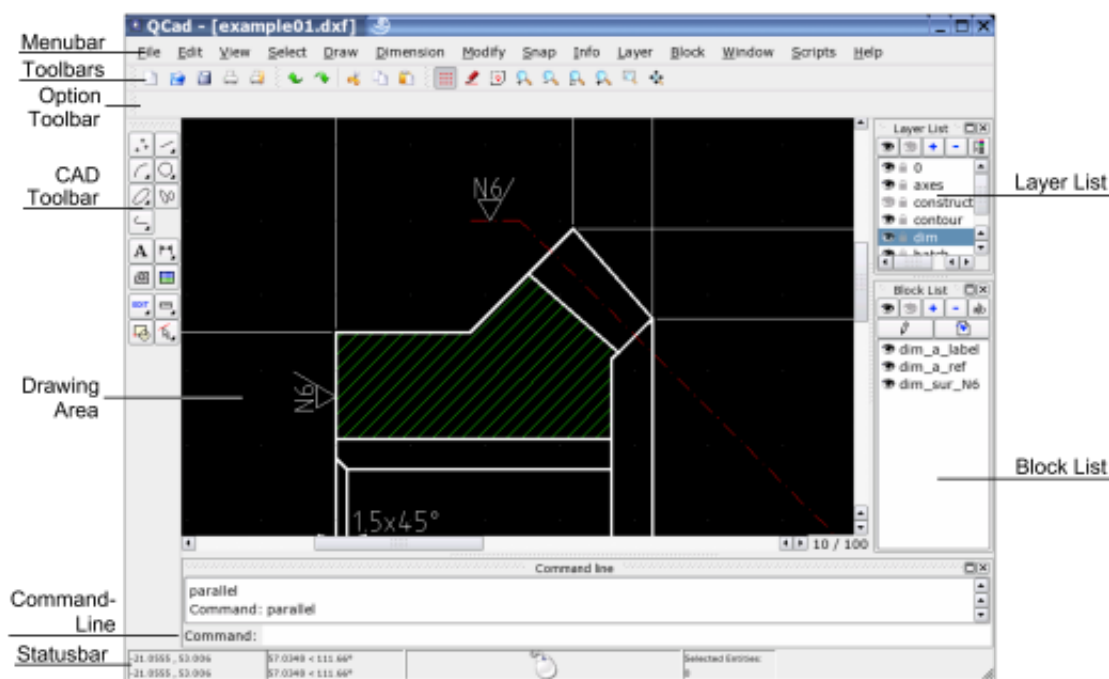
Figure 8: Main application window of QCad.

6.2. Menu and CAD Toolbar

Most functions of QCad can be accessed from its menu at the top. 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.

6.3. Layer List and Block List

To the right of the main window of QCad you can find the layer list and the block list. They list the layers and blocks of the currently active drawing and offer some basic tools that are layer or block specific.
You can switch the lists on and off using the menu View - Views - Layer List and View - Views - Block List.

6.4. Status Bar

The status bar at the bottom of the application window shows various information about the current status of QCad. At the left, the coordinate widget displays the absolute and relative position of the mouse cursor as cartesian and polar coordinates. The mouse widget in the middle shows information about the current function of the left and right mouse buttons. The selection widget at the right shows the number of entities that are currently selected.
The status bar can be enabled / disabled using the menu View - Views - Statusbar.

6.5. Command Widget

Just above the status bar you can find the command line of QCad (Professional version only). 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.

6.6. 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.

6.7. Executing Commands

Every action in QCad depends on a command being issued first. There are three different ways to execute QCad commands:

select a corresponding tool icon from one of the toolbars choose a menu item from one of the pull-down menus enter a command in the command prompt

Most actions require further user inputs after being launched. There are three ways how those inputs can be requested by QCad:

Dialogs. For example the text construction tool shows a dialog to choose a font and type in the string for the text entity. Option toolbar. For example the tool for drawing parallels requires the input of the distance of the parallel from the original entity. It displays a text input box in the tool options toolbar at the top where the user can type the distance (see Figure 9). Command line. For example when drawing a sequence of lines you can close the sequence by typing 'close' into the command line.
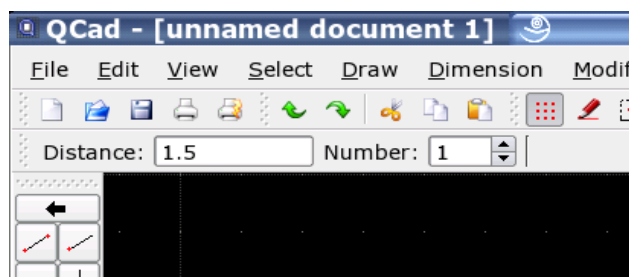
Figure 9: An optional toolbar may be shown for some tools.

6.8. 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 coordinates. 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. If in normal more, you can enter the command mode by pressing the space key or by clicking into the command line input field.

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 Z key, followed by the A key in this mode, QCad triggers the auto zoom command. To switch from command mode to normal mode, press the Escape key. If there is some text in the command line input field, you need to press Escape twice.

7. Command Line

7.1. Using the Command Line

QCad can be controlled either using menus, the toolbars, hotkeys or the command line. 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 are not (yet) used to using a command line, you might want to switch it off using the menu View - Command Line.
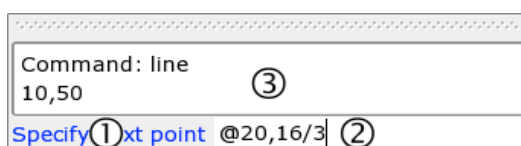
Figure 10: The Command Line of QCad.

Figure 10 shows the command line of QCad. It consists of the command line prompt (1), the command line input field (2) and the output area (3) that shows the command line history as well as potential errors, warnings, ...

7.2. 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 if the command prompt is blue and a text cursor (vertical line) blinks in the command line input field.

Sometimes it is quicker to work with hotkeys than using the command line. In this case you will have to keep the focus away from the command line and give it back to the main application window. The easiest way to achieve that is to press the Escape key. If there is 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 into it with the mouse or press the Space bar.

8. File Handling

QCad can load DXF files from older QCad versions, AutoCAD and other applications that can produce DXF files. When saving files, QCad follows the DXF 2000 standard.

8.1. Creating New Drawings

Toolbar:

Menu:

File - New

Hotkey:

Ctrl-N

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

8.2. Opening Drawings

Toolbar:

Menu:

File - Open

Hotkey:

Ctrl-O

Use this command to open a DXF file. The file is loaded and shown in a new window within the QCad application window. You can switch between different loaded drawings using the menu 'Windows'.

Please note that only those entities from the file are loaded that are supported by QCad. 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, all not supported entities will be lost.

8.3. 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. You will then be asked for a file name before the drawing is saved.

8.4. 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 you for the bitmap size and the preferred background color.

8.5. Closing Drawings

Menu:

`File - Close`

Hotkey:

`Ctrl-W`

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

9. 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. Before printing you should always switch to the print preview mode. In the print preview, you can adjust the scale factor and position the drawing on the paper.

9.1. Print Preview

Opens a print preview window for the current drawing.

Toolbar:

Menu:

`File - Print Preview`

Options Toolbar:

The print preview shows how your drawing will look in the printout. Use the combo box in the options toolbar to adjust the scale factor. The first button at the right of the combo box toggles the black/white mode. When enabled all lines will appear black on the drawing rather than colored or gray. Use the button in the middle to automatically center the drawing on the paper or the button at the right to automatically fit the drawing on paper.

You can also move the paper around by dragging it with the left mouse button.
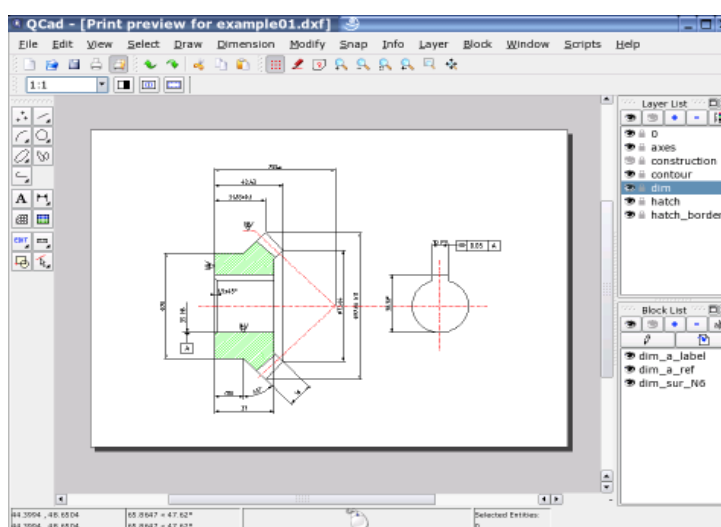
Figure 11: Print Preview.

9.2. Printing a Drawing

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. Do not change the paper format at this stage. 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 this creates a Postscript (PS) file that can also be converted to PDF. Under Windows a raw binary print file is created (PRN). The format of this file depends on the print driver. To create PS files under Windows, please install a Postscript printer driver such as "APS-PS" which comes with Windows.
To create high quality PDF files, you can use the menu 'File' - 'PDF Export' (QCad 2.1.0.1 and later).

10. Basic Editing Commands

This chapter lists the basic edit commands of QCad. These commands are not CAD specific and often also available in other applications. For CAD specific editing commands, please refer to chapter "Modification".

10.1. 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. For example if you have just created a line and a circle which you want to take back, launch the undo command twice.
Undo cannot undo all actions. Some operations on files, layers and blocks cannot be undone.

10.2. Redo

Toolbar:

Menu:

Edit - Redo

Hotkey:

uu, Ctrl-Shift-Z

Command:

redo, r

Description:

The redo command is the counterpart of the undo command. It restores changes that were previously undone.

10.3. Copy

Toolbar:

Menu:

Edit - Copy

Hotkey:

Ctrl-C

Command:

copy, cp

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 coordinate in the command line. The reference point will be used to position the entities when pasting them into a drawing again.

10.4. Cut

Toolbar:

Menu:

Edit - Cut

Hotkey:

Ctrl-X

Command:

cut, ct

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.

10.5. Paste

Toolbar:

Menu:

Edit - Paste

Hotkey:

Ctrl-V

Command:

paste, ps

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 is 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.

Procedure:

1.   Activate the paste function.
2.   Set the target point of for the pasted entities with the mouse or enter a coordinate in the command line. The target point corresponds to the reference point that was chosen when copying or cutting the entities.

11. Viewing

The view menu and toolbar offer some tools to adjust the current viewport of the drawing. All these tools have no effect on the geometry of 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 is worth remembering and using the hotkeys. For example 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.

11.1. Scrolling

Use the two scrollbars at the right and the bottom of the 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 (hold down the Ctrl key) or left and right (hold 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 (panning).

11.2. Toggle Grid

Toolbar:

Menu:
```
View - Grid
```
Description:

Toggles the grid visibility of the current drawing.

11.3. Toggle Draft Mode

Toolbar:

Menu:
```
View - Draft
```
Description:

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 bounding rectangles. Use the draft mode if your drawing becomes very large and it takes long to redraw it.

11.4. Redrawing

Toolbar:

Menu:
```
View - Redraw
```

Hotkey:

`zr, rg`

Command:

`regen`

Description:

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.

11.5. Zooming In / Out

Toolbar:
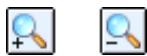
Menu:

`View – Zoom in`
`View – Zoom out`

Hotkey:

`+ / –`
`zi / zo`

Description:

This tool increases / decreases the current viewing factor by 1.5. The same effect can also be achieved by turning the mouse wheel (Figure 13).

Figure 13: Use the mouse wheel to zoom in / out quickly.

11.6. Auto Zoom

Toolbar:

Menu:

`View – Auto Zoom`

Hotkey:

`za`

Command:

zoomauto

Description:

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

11.7. Previous View

Toolbar:

Menu:

View - Previous View

Hotkey:

zv

Command:

zoomprevious

Description:

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.

11.8. Window Zoom

Toolbar:

Menu:

View - Window Zoom

Hotkey:

zw

Command:

zoomwindow

Description:

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.

11.9. Panning

Toolbar:

Menu:

View - Pan Zoom

Hotkey:

zp

Command:

zoompan

Description:

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 does not have a middle mouse button you can activate this tool instead and then do the same with the left mouse button. Click the right mouse button when you are done.

12. Layers

Layers are a useful concept for organizing drawings. It is crucial to learn 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 layers and show you how QCad lets you manage the layers of your drawing.

Every drawing contains at least one layer: layer '0'. This is the default layer that can never be removed. Additional layers can be created at any time they are required. Usually you will plan a drawing and create all necessary layers with the correct attributes before starting to draft. The number of layers for a QCad drawing is not limited but usually you will not need more than 10 layers. Note that every entity of your drawing is placed on exactly one layer. 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 currently working on. It is highlighted in the layer list (see Figure 14). Newly created entities are always 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, all other layers are visible. You can quickly show / hide layers by clicking on the eye icon.

The other icon beside the layer name is a symbol for a lock. In Figure 14, layer "hatches" is locked. Entities on locked layers cannot be selected and therefore cannot be modified or deleted. To quickly lock / unlock a layer, click on its lock icon.

12.1. Changing the Visibility of a Layer

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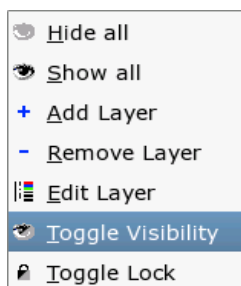
Figure 15: Context menu of the layer list.

12.2. Changing the Visibility of all Layers

Toolbar (Layer List):

Menu:

Layer - Show all Layer - Hide all

Hotkey:

th*, fr*

The two buttons shown above can be used to show (defreeze) or hide (freeze) all layers at once. This can be useful when working with a drawing that has many layers. The same function is also available from the context menu.

12.3. Adding New Layers

Toolbar (Layer List):

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).
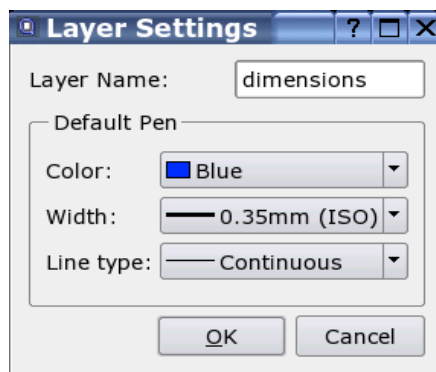
Figure 16: Dialog for creating new layers.

12.4. Deleting Layers

Toolbar (Layer List):

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, cannot 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.

12.5. Editing Layer Attributes

Toolbar (Layer List):

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.

13. 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.

13.1. 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).
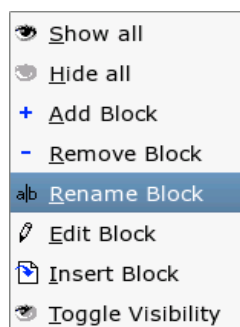
Figure 18: Context menu of the block list.

13.2. Changing the Visibility of all Blocks

Toolbar (Block List):

Menu:

```
Block - Show All Block - Hide All
```

Description:

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

13.3. Adding New Blocks

Toolbar (Block List):

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.

13.4. Removing Blocks

Toolbar (Block List):

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.

13.5. Renaming Blocks

Toolbar (Block List):

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.

13.6. Editing Blocks

Toolbar (Block List):

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.

13.7. Inserting Blocks

Toolbar (Block List):

Menu:

```
Block - Insert Block
```

Command:

```
?
```

Options Toolbar:
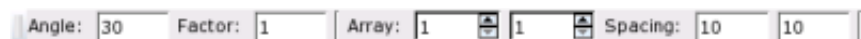
Description:

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.

13.8. Creating Blocks from Existing Entities

Toolbar:

Menu:

```
Block - Create Block
```

Description:

Creates a new 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 coordinate 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 instances of it can be inserted into the drawing. The entities you have selected in step 1 are removed and replaced by an instance of the block. If you don't

want that you can use the 'undo' function to delete that insert. To restore the original entities, use 'undo' again. Note that the undo / redo functions have no influence on the blocks themselves. Once a block is created it will not be removed again when using undo.

14. Object Snap

This chapter lists all snap modes that can be activated to assist you to position the mouse cursor to an exact location.

For most constructions and modifications in QCad, it is necessary to specify the coordinates 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 a set of object snap modes. The toolbar that features those modes is shown in Figure 20. It is automatically shown whenever object snap modes can be used .
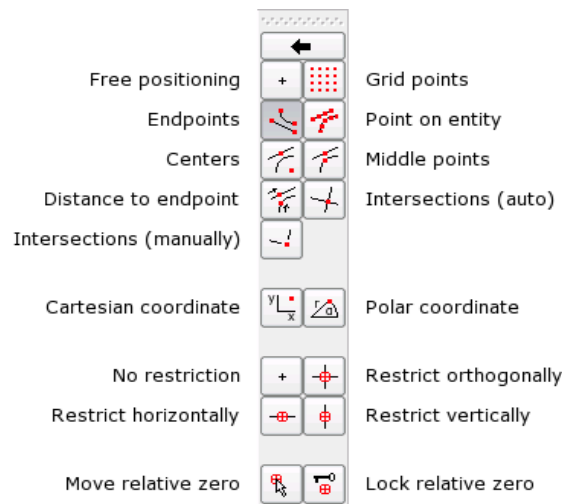
Figure 20: The object snap toolbar.

14.1. Free Positioning

CAD Toolbar:

Menu:

Snap - Free

Hotkey:

os

Command:

free, fre

Description:

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

14.2. Snap to Grid

CAD Toolbar:

Menu:

Snap - Grid

Hotkey:

sg

Command:

grid, gri

Description:

Snaps to grid points.

14.3. Snap to Endpoints

CAD Toolbar:

Menu:

Snap - Endpoints

Hotkey:

se

Command:

endpoint, end

Description:

Snaps to endpoints of lines and arcs and to points (Figure 21).
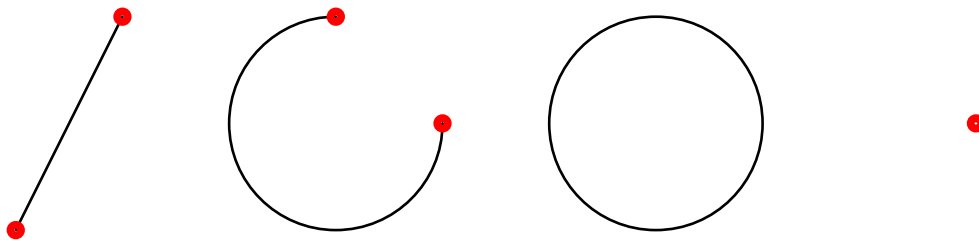
Figure 21: Endpoints found by the snap tool.

14.4. Snap to Points on Entities

CAD Toolbar:

Menu:

Snap - On Entity

Hotkey:

sn

Command:

near, nea

Description:

Snaps to the closest point that is on an entity (Figure 22).
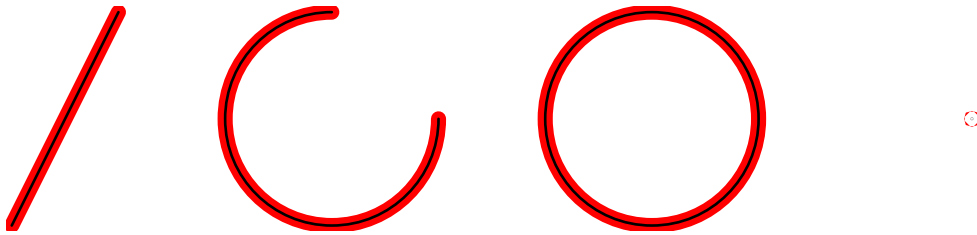
Figure 22:'On Entity' snaps to any point on the line of an entity.

14.5. Snap to Centers

CAD Toolbar:

Menu:

Snap - Centers

Hotkey:

so

Command:

center, cen

Description:

Snaps to centers of lines and arcs and to points (Figure 23).
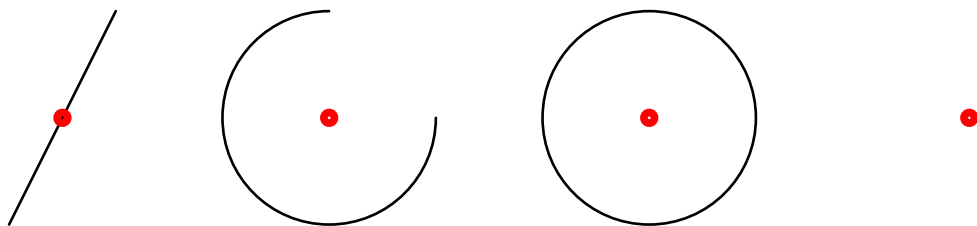
Figure 23: Centers found by the snap tool.

14.6. Snap to Middle Points

CAD Toolbar:

Menu:

Snap - Middle

Hotkey:

sm

Command:

middle, mid

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.
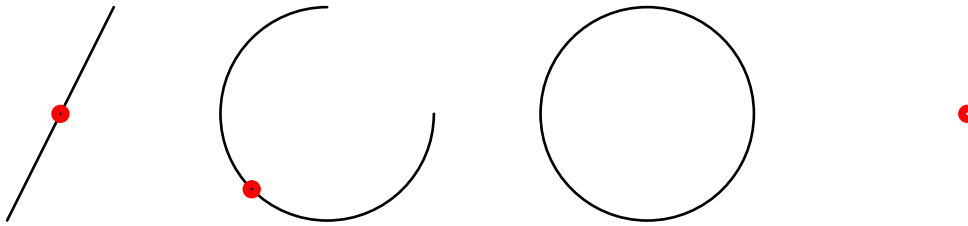
Figure 24: Middle points found by the snap tool.

14.7. Snap to Distance Points

CAD Toolbar:

Menu:

Snap - Distance from Endpoint

Command:

distance, dist

Options Toolbar:
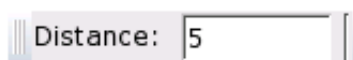
Description:

Snaps to points with a given distance from endpoints (Figure 25). The distance can be adjusted in the options toolbar.
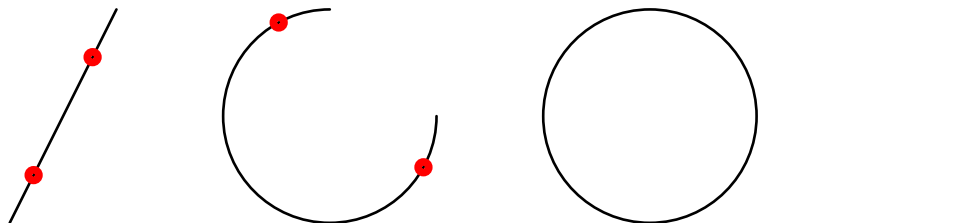
Figure 25: Example snap points for the snap tool 'Distance from Endpoint'.

14.8. Snap to Intersections

CAD Toolbar:

Menu:

Snap - Intersection

Hotkey:

si

Command:

intersection, int

Description:

Snaps to intersections between entities (Figure 26).
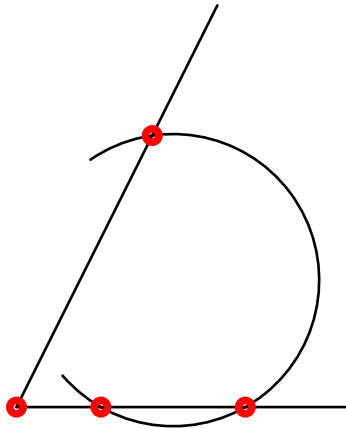
Figure 26: The Intersection Snap tool finds automatically all visible intersection points.

14.9. Snap to Intersections Manually

CAD Toolbar:

Menu:

`Snap - Intersection Manually`

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 is outside the entities (Figure 27).
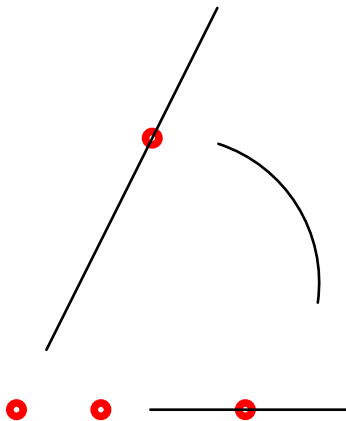
Figure 27: The Manual Intersection Snap tool also lets you specify invisible intersection points.

14.10. 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 is the point on the bottom line of the shape that has the same X-coordinate 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").
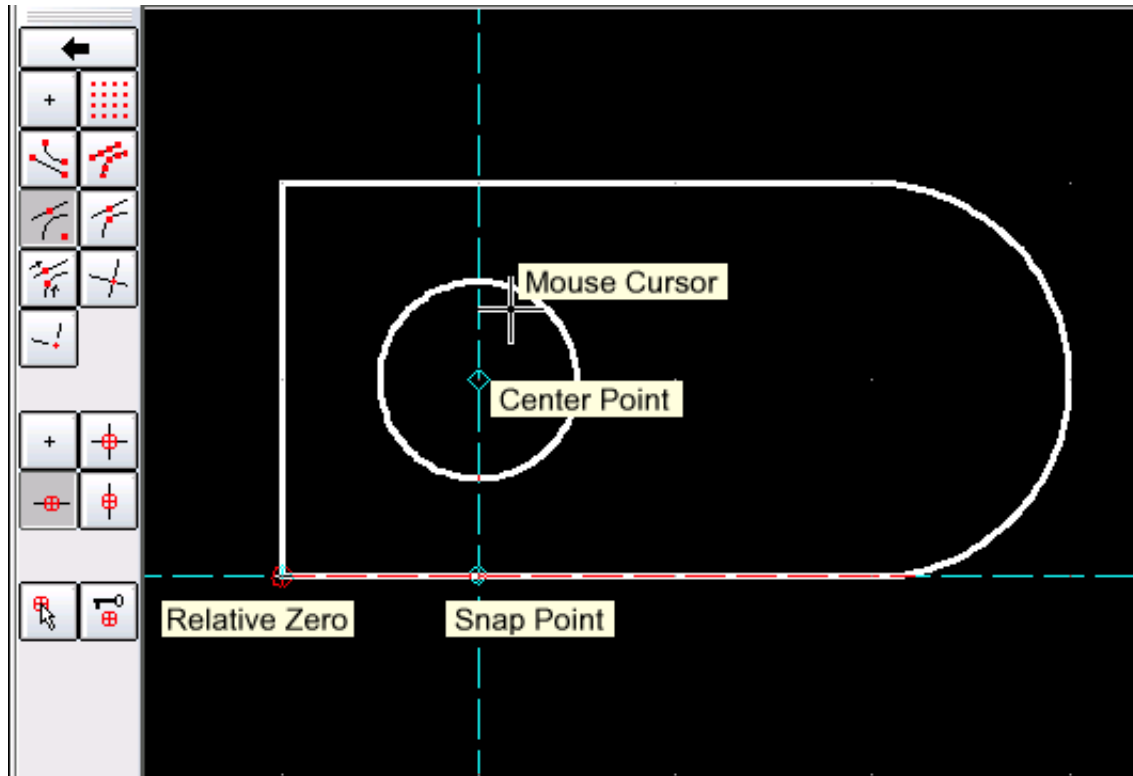
Figure 28: Snap restrictions further limit the snap position
to imaginary lines orthogonally to the relative zero point.

Restrict Nothing

CAD Toolbar:

Menu:

`Snap - Restrict Nothing`

Description:

Disables all restrictions.

Restrict Orthogonally

CAD Toolbar:

Menu:

`Snap - Restrict Orthogonally`

Description:

Restricts orthogonally to relative zero point.

Restrict Horizontally

CAD Toolbar:

Menu:

```
Snap - Restrict Horizontally
```

Description:

Restricts horizontally to relative zero point.

Restrict Vertically

CAD Toolbar:

Menu:

```
Snap - Restrict Vertically
```

Description:

Restricts vertically to relative zero point.

14.11. 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 coordinates you enter in the command line and for snapping restrictions.
Normally, the relative zero point moves around automatically. For example if you are drawing a sequence of lines, the relative zero point always moves to the last point you have just set. The next point of the sequence can then be defined in coordinates relative to that 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 to do this.

Moving the Relative Zero Point

CAD Toolbar:

Menu:

```
Snap - Set Relative Zero
```

Command:

```
relativezero, relz
```

Description:

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

(Un-)locking the Relative Zero Point

CAD Toolbar:

Menu:

```
Snap - (Un-)Lock Relative Zero
```

Command:

```
lockrelativezero, lrel
```

Description:

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

15. Drawing Commands

The following chapters describe the tools used in QCad for drawing graphical 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 processes to modify or copy existing entities.

16. Points

16.1. 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 coordinate in the command line.

17. Lines

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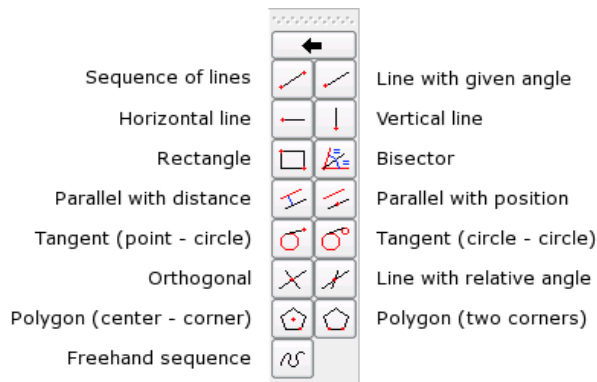
Figure 29: The line toolbar.

17.1. Sequences Of Lines

CAD Toolbar:

Menu:

Draw – Line – 2 Points

Command:

line, ln, l

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 coordinate 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.

17.2. Lines With Given Angle

CAD Toolbar:

Menu:

Draw – Line – Angle

Command:

lineangle, la

Options Toolbar:

Angle: 15   Length: 50   Snap Point: Start ▼

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. Choose the snap point on the line which you want to use for positioning the line. Start means that the line will start at the point where you position it.
3. Place the line with the mouse or by entering a coordinate in the command line.

17.3. Horizontal / Vertical Lines

CAD Toolbar:

Menu:

Draw - Line - Horizontal / Vertical

Command:

linehor, lhlinever, lv

Options Toolbar:

Length: 50   Snap Point: Start ▼

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.

17.4. Rectangles

CAD Toolbar:

Menu:

Draw - Line - Rectangle

Command:

rectangle, rectang, rec

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 coordinate of the second corner in the command prompt. E.g. to
      create a rectangle with width 50 and height 25, enter @50,25.

17.5. Bisectors

CAD Toolbar:

Menu:

`Draw - Line - Bisector`

Command:

`bisector`

Options Toolbar:

Length: 50   Number: 2

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.
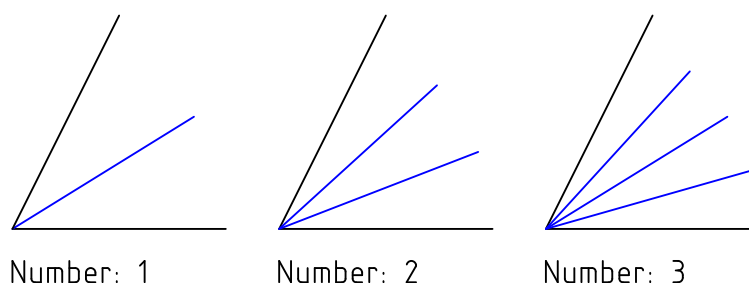

Number: 1   Number: 2   Number: 3
Figure 30: Construction of Bisectors.

17.6. Parallels

CAD Toolbar:

Menu:

`Draw - Line - Parallel`

Command:

`lineparallel, parallel, par, lineoffset, offset, o`

Options Toolbar:

Distance: 1.5   Number: 1

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.

17.7. Parallels Through Point

CAD Toolbar:

Menu:

```
Draw - Line - Parallel
```

Options Toolbar:
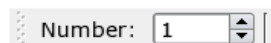
Number: 1

Description:

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

Procedure:

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

17.8. Tangents From Point to Arc / Circle

CAD Toolbar:

Menu:

```
Draw - Line - Tangent (P,C)
```

Command:

```
linetangent, tangent, lt
```

Description:

Create tangents from a coordinate 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 coordinate 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.

17.9. Tangents from Arc to Arc / Circle to Circle

CAD Toolbar:

Menu:

Draw - Line - Tangent (C,C)

Command:

linetangent2, tangent2, lt2

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.

17.10. Lines Orthogonal to Other Entities

CAD Toolbar:

Menu:

Draw - Line - Orthogonal

Command:

lineorthogonal, lo

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 coordinate in the command line.

17.11. Lines with Relative Angle to Other Entities

CAD Toolbar:

Menu:

Draw - Line - Relative Angle

Command:

linerelangle, lre

Description:

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

Procedure:

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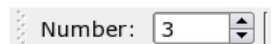3.   Place the line with the mouse or enter a coordinate in the command line.

17.12. Polygons with Center and Corner

CAD Toolbar:

Menu:

Draw - Line - Polygon (Cen,Cor)

Command:

linepolygon, polygon

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 coordinate in the command line.
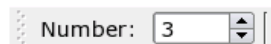3.   Set the first corner of the polygon.

17.13. Polygons with Two Corners

CAD Toolbar:

Menu:

Draw - Line - Polygon (Cor,Cor)

Command:

linepolygon2, polygon2

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 coordinate in the command line.
3. Set the second corner of the polygon.

17.14. Freehand Lines

CAD Toolbar:

Menu:

`Draw - Line - Freehand`

Command:

`linefree, freehand`

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.
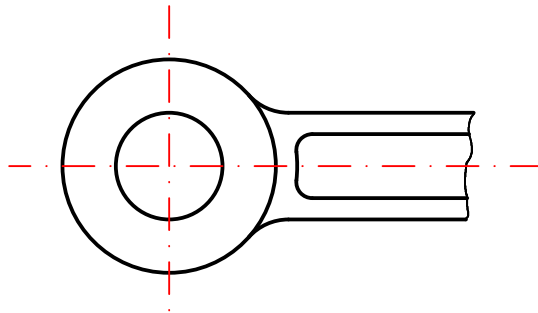

Figure 31: Example use for the freehand line tool.

18. Arcs

18.1. Arc with Center, Radius and Angles

CAD Toolbar:

Menu:

Draw - Arc - Center, Point, Angles

Hotkey:

ar

Command:

arcc

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 coordinate in the command line.
3.   Define the radius by clicking a point on the arc or by entering a coordinate into the command line. You may also enter the radius directly in the command line.
4.   Set the start angle with the mouse or by entering a coordinate or the angle amount in the command line.
5.   Set the end angle the same way as the start angle.

18.2. Arc with 3 Points

CAD Toolbar:

Menu:

Draw - Arc - 3 Points

Hotkey:

a3

Command:

arc

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 coordinate in the command line.
2.   Set the second point to a known point on the arc line.
3.   Set the endpoint of the arc.

18.3. Concentric Arc

CAD Toolbar:

Menu:

```
Draw – Arc – Concentric
```

Hotkey:

```
of
```

Command:

```
offset, of
```

Options Toolbar:

Description:

With this tool you can create concentric arcs.

Procedure:

1.   Enter the distance of the concentric arc from the original base arc in the options toolbar shown at
      the top or in the command line.
2.   Enter the number of concentric arcs to create in the options toolbar.
3.   Click the base arc. The concentric arc(s) are created on that side on which the mouse cursor is
      located while selecting the base arc.

19. Circles

19.1. Circle with Center and Point

CAD Toolbar:

Menu:

Draw - Circle - Center, Point

Hotkey:

ci

Command:

circle

Description:

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

Procedure:

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

19.2. Circle with Center and Radius

CAD Toolbar:

Menu:

Draw - Circle - Center, Radius

Command:

circlecr

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 coordinate in the command line.

19.3. Circle with 2 Points

CAD Toolbar:

Menu:

Draw - Circle - 2 Points

Command:

circle2p

Description:

Draws circles with two diametrically opposed points.

Procedure:

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

19.4. Circle with 3 Points

CAD Toolbar:

Menu:

Draw - Circle - 3 Points

Command:

circle3p

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 coordinate in the command line.
2.   Set the second point.
3.   Set the third point.

19.5. Concentric Circle

CAD Toolbar:

Menu:

Draw - Circle - Concentric

Command:

offset, of

Options Toolbar:

Distance: 1.5   Number: 1

Description:

With this tool you can create concentric circles.

Procedure:

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

20. Ellipses

20.1. Full Ellipses

CAD Toolbar:

Menu:

Draw – Ellipse – Ellipse

Command:

ellipse

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

20.2. Ellipse Arcs

CAD Toolbar:

Menu:

Draw – Ellipse – Ellipse Arc

Command:

ellipsearc

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

21. Polylines

Click the polyline button in the CAD toolbar to show the polyline toolbar.
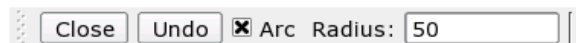
21.1. Creating a Polyline

CAD Toolbar:

Menu:

```
Draw - Polyline - Create
```

Command:

```
polyline
```

Options Toolbar:

Description:

This tool lets you draw polylines. A polyline is a continuous line formed from one or more connected line or arc segments.

Procedure:

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

Note: If your polyline contains arc segments, you can tick the 'Arc' checkbox and enter the arc radius in the options toolbar when setting the endpoint of an arc segment. The arc will connect smoothly (tangential) to the last segment.

21.2. Adding Nodes to Polylines

CAD Toolbar:

Menu:

```
Draw - Polyline - Add node
```

Description:

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

Procedure:

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

21.3. Appending Nodes to Polylines

CAD Toolbar:

Menu:

Draw - Polyline - Append node

Description:

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

Procedure:

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

For appending nodes, the same options are available as when creating new polylines.

21.4. Deleting Nodes from Polylines

CAD Toolbar:

Menu:

Draw - Polyline - Delete Polyline node

Description:

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

Procedure:

1.   Specify the polyline from which you want to delete nodes.
2.   Specify the position of the node that you want to remove. In most cases it is appropriate to use the endpoint snap function to specify the node. The tool automatically activates that snap function by default.

21.5. Deleting Segments Between Two Nodes

CAD Toolbar:

Menu:

Draw - Polyline - Delete between two nodes

Description:

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

Procedure:

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

21.6. Trimming Polyline Segments

CAD Toolbar:

Menu:

```
Draw - Polyline - Trim segments
```

Description:

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

Procedure:

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

21.7. Creating Equidistant Polylines

CAD Toolbar:

Menu:

```
Draw - Polyline - Equidistant
```

Options Toolbar:

Distance: 12   Number: 1

Description:

This tool lets you create equidistant polylines to existing polylines.

Procedure:

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

21.8. Creating a Polyline from Existing Segments

CAD Toolbar:

Menu:

Draw – Polyline – Create from Segments

Description:

This tool lets you create polylines from existing segments (lines and arcs that are connected).

Procedure:

1.   Click one of the segments. A polyline is created from the selected entity and all entities that are
     connected to its end- or startpoint.

22. Splines

CAD Toolbar:

Menu:

`Draw - Spline`

Command:

`spline`

Options Toolbar:
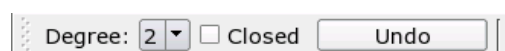
Description:

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

Procedure:

1. Choose the degree of the spline in the options toolbar. Supported degrees are 2 (quadratic b-spline) and 3 (cubic b-spline). The higher the degree, the 'smoother' the curve becomes. Note, that a quadratic b-spline requires by definition at least 3 control points while a cubic b-spline needs at least 4 control points.
2. Check the 'Closed' checkbox in the options toolbar if you want to create a closed spline. Closed splines are continuous closed loops.
3. Specify the control points. After defining the third (for quadratic b-splines) or fourth (for cubic b-splines) control point, a preview will be shown. You can remove the last control point again if it was accidently put in the wrong place by clicking the 'Undo' button in the options toolbar.
4. Click the right mouse button to finish the spline tool.
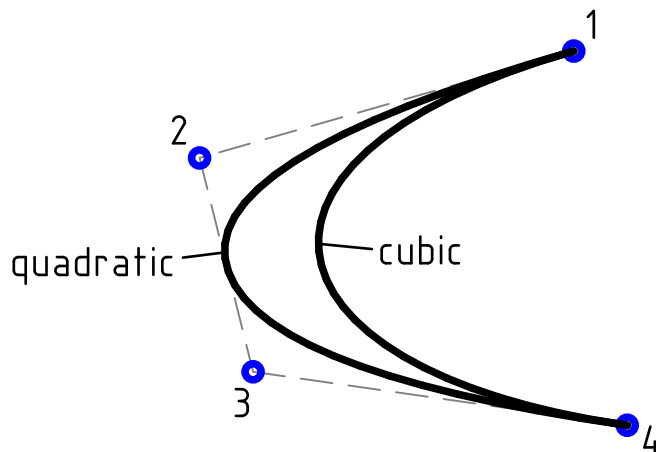
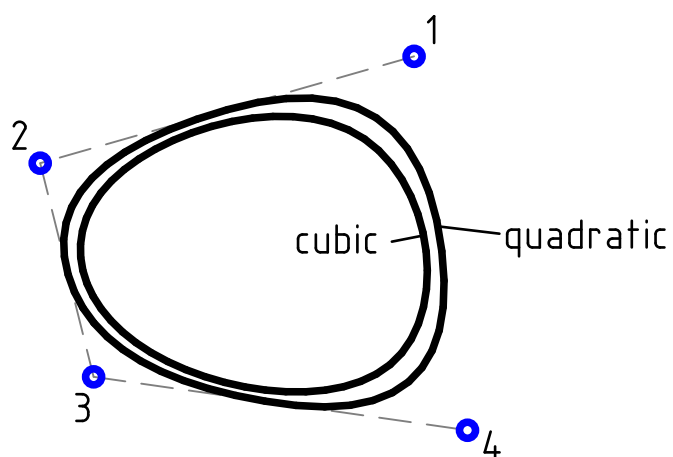**Figure 32:** Open splines with four control points.

Figure 33: Closed splines with the same control points.

23. Texts

CAD Toolbar:

A

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 34).
2.   Choose the font name and attributes in the upper left area of the dialog (Figure 34/1). 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.
3.   Choose the alignment of the text (Figure 34/2).
4.   Enter the angle for the text (Figure 34/3). Normal horizontal texts have an angle of zero degrees.
5.   Enter the text you want to create in the large text box (Figure 34/4). You can also use the icons above the box to load texts from files, save texts that you have 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 34/5). 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 does not contain the symbol you need, please use the comboboxes at the bottom right to choose any character from the unicode char set (Figure 34/6). Note that the font you have chosen might not contain the characters you insert. In that case the characters will not be displayed in the drawing.
      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.
6.   Click "OK" to exit the text dialog.
7.   Use the mouse to specify the location of the text or enter a coordinate in the command line.
8.   Often, users need to create a number of similar texts. For that reason the text tool does not terminate after creating the first text entity. You can change the text and the text angle in the options toolbar and create other texts with similar settings.
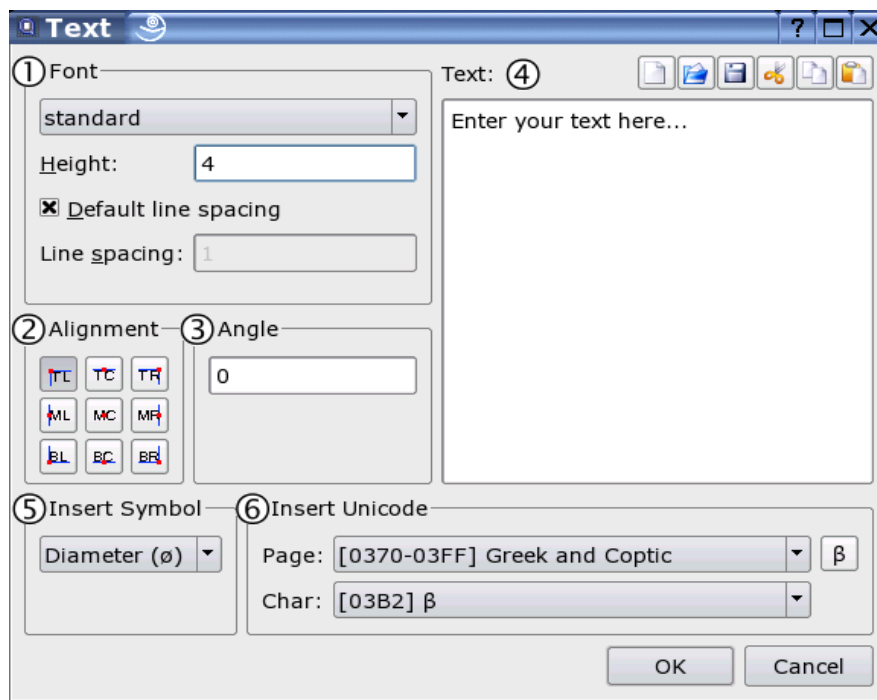9.   Right click or press Escape to stop the text tool.

Figure 34: The text creation dialog.

23.1. Advanced Text Commands

Sometimes it can be useful to embed two small text lines on top of each other in a text (e.g. for tolerances). There is a sepcial 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 35.

Figure 35: Text produced by the string "10\S+0.05\-0.05; Millimeter".

24. Dimensions

This chapter is about constructing dimensions. Figure 36 shows the notation that is used when referring to dimension parts.
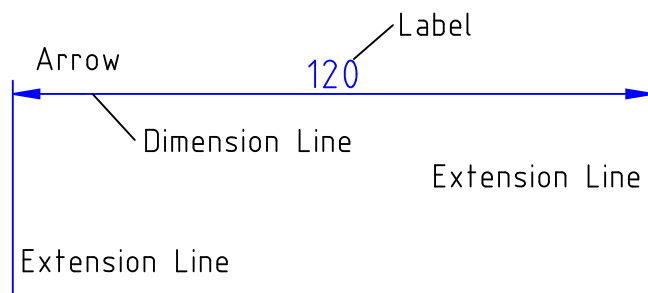
Figure 36: Notation for dimensions.

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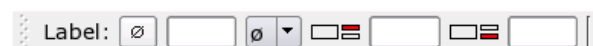
This text label can be any text. When left empty, the label is generated automatically and contains the measured distance (or angle) of the dimension. To add additional signs or letters to a measured dimension, you can use the combination <> to insert the automatically generated label anywhere in the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

Examples:

Chosen label in the toolbar:	Example for the generated label:
a	a
<>	20
<> H7	20 H7
#<>	#20

The toggle button with the diameter sign can be used to show a diameter sign before the dimension label. Use the edit field right after that button to enter any text to be displayed as the label. The combo box contains some often used symbols. Select a symbol from the list to insert it into the edit field for the label. The two edit fields at the right are usually used to indicate tolerances. The first for the upper tolerance of a value (e.g. +0.1) and the second one for the lower tolerance (e.g. -0.05). See Figures 37 and 38 for an example.

Figure 37: These example options generate a dimension label as shown in Figure 38.

Figure 38: Example dimension label with advanced options.

24.1. Aligned Dimensions

CAD Toolbar:

Menu:

Dimension - Aligned

Hotkey:

da

Command:

dimaligned

Description:

Creates aligned dimensions. Aligned dimensions usually measure the length of an existing line (Figure 39). The dimension line is always parallel to the line between the two extension line endpoints.
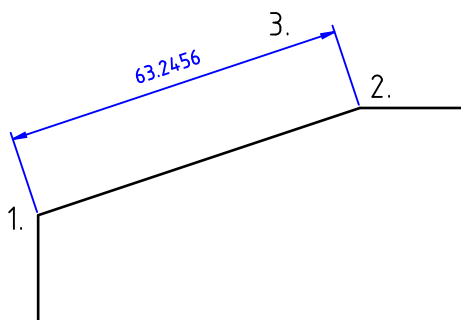

Figure 39: Example for an aligned dimension.

Procedure:
1. Set the first extention line endpoint with the mouse or enter a coordinate in the command line.
2. Set the second extension line endpoint.
3. Set the position of the dimension line.

24.2. Linear Dimensions

CAD Toolbar:

Menu:

Dimension - Linear

Hotkey:

dr

Command:

dimlin

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 40 shows a horizontal version of a linear dimension.
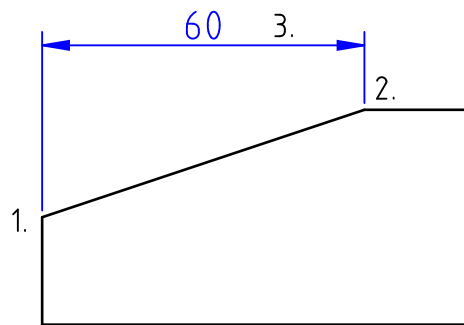
Figure 40: Example of a horizontal, linear dimension.

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 degrees.
2. Set the first extension line endpoint with the mouse or enter a coordinate in the command line.
3. Set the second extension line endpoint.
4. Set the position of the dimension line.

24.3. Horizontal / Vertical Dimensions

CAD Toolbar:

Menu:

```
Dimension – Horizontal
Dimension – Vertical
```

Hotkey:

```
dh, dv
```

Command:

```
dimhor
dimver
```

Description:

These tools are provided for convenience and behave essentially like the tool described above (Linear Dimension). The only difference is that you don't have to enter an angle.

24.4. Radial Dimensions

CAD Toolbar:

Menu:

```
Dimension – Radial
```

Command:

dimrad

Description:

Creates radial dimensions for circle or arc entities (Figure 41).
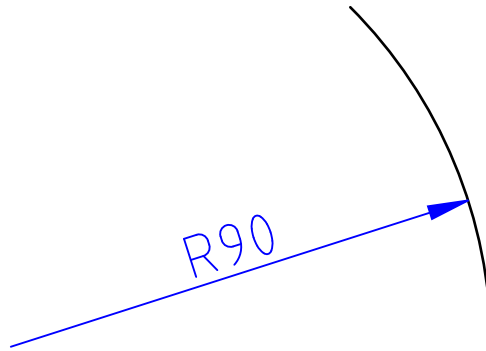
Figure 41: Example of a radial dimension.

Procedure:

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

24.5. Diametric Dimensions

CAD Toolbar:

Menu:

Dimension - Diametric

Command:

dimdia

Description:

Creates diametric dimensions for circle or arc entities as shown in Figure 42.
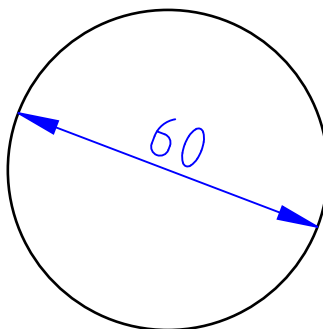
Figure 42: Example of a diametric dimension.

Procedure:

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

24.6. Angular Dimensions

CAD Toolbar:

Menu:

Dimension - Angular

Command:

dimang

Description:

Creates angular dimensions between two reference lines as shown in Figure 43.
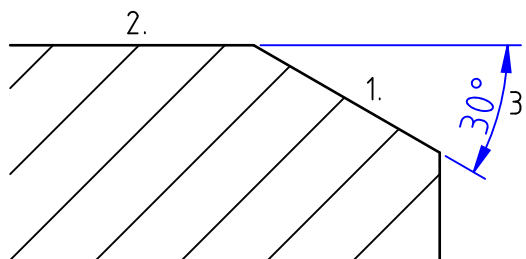
Figure 43: Example of an angular dimension.

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 coordinate in the command line.

24.7. Leaders

CAD Toolbar:

Menu:

Dimension - Leader

Hotkey:

dl

Command:

dimlea

Description:

Leaders are arrows that usually point from a text entity to an other entity as shown in Figure 44. In the example, the text entity "N7" is describing a surface property by pointing to it with a leader.
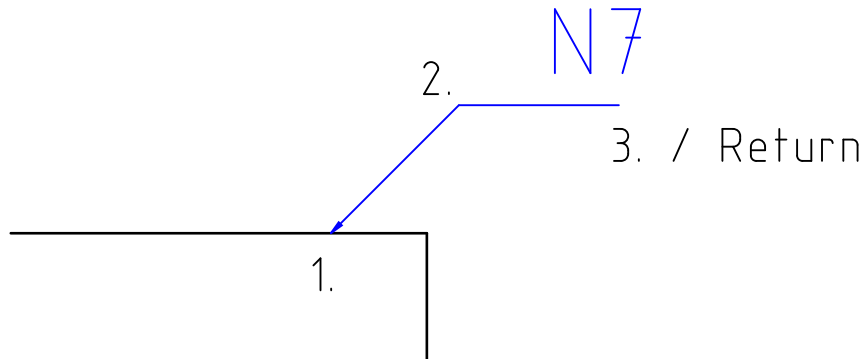
Figure 44: Example of a leader.

Procedure:

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

25. Hatches And Solid Fills

CAD Toolbar:

Menu:

Draw - Hatch

Command:

hatch

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 45.
2. Launch the hatching action.
3. Select the contour(s) you want to fill. Note that islands inside contours will get hatched if they are not selected (Figure 46)
4. Click the right arrow button in the CAD toolbar to continue.
5. A dialog for the hatch options is displayed (Figure 47). Choose a hatch pattern, scale factor and a rotation angle for the hatch pattern. If you want to fill the object with a solid color instead of a pattern, check the checkbox "Solid Fill".
6. Click 'OK' to proceed with the hatching. Depending on the complexity of the contour and the scale factor of the chosen pattern, it might take a while to create the hatch.
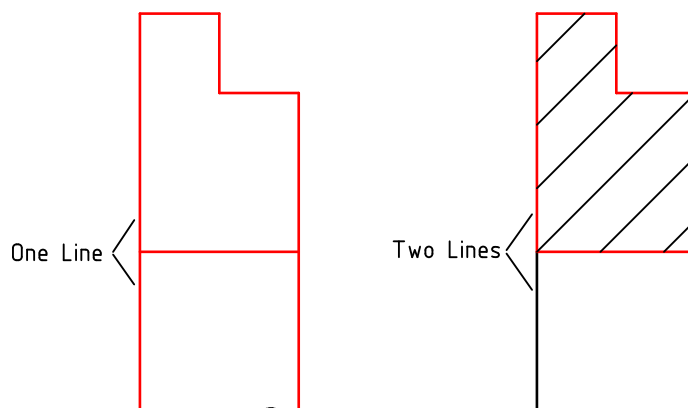
Figure 45: Preparation for the boundary entities.
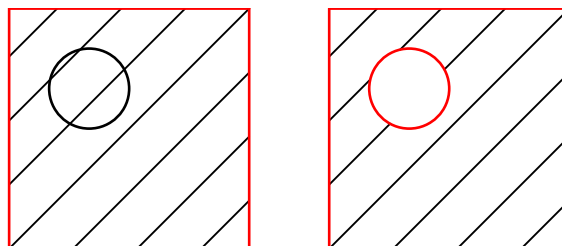
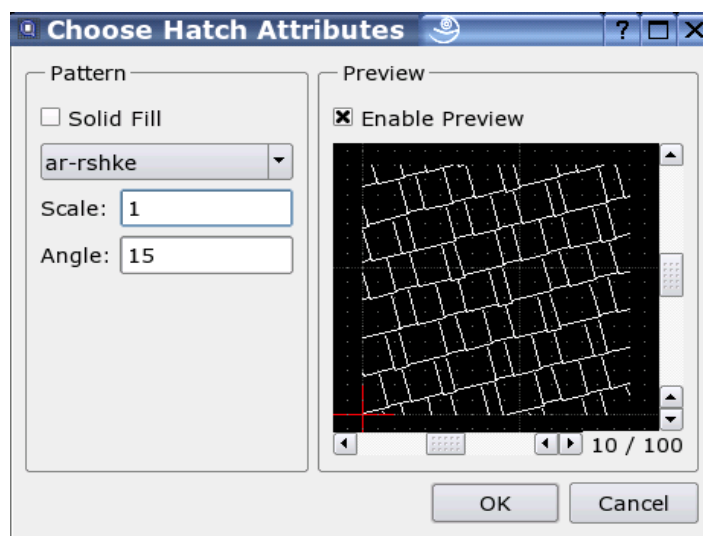**Figure 46:** Islands which are selected will not get hatched through (right).

Figure 47: Hatch pattern selection.

26. Images

CAD Toolbar:

Menu:

Draw - Image

Command:

image

Options Toolbar:

Angle: 30   Factor: 0.5

Description:

Inserts a raster image (bitmap) into the drawing.

Note that the DXF file which is created when saving a drawing, contains only a reference to images. It is recommended to keep the image file and the DXF file in the same folder, so QCad can find the image again when the DXF file is loaded later.

Transparent backgrounds are supported for PNG images.

Note that large bitmaps can cause the drawing display to become very slow. Rotating large bitmaps with QCad is generally not advisable. You might want to use a bitmap graphic application to rotate the image before you insert it into a QCad drawing.

Usually you will want images to be in the background of other entities. Please refer to the 'Modify' - 'Send to back' function to learn how to do that.

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 is more efficient to rotate the bitmap in a bitmap editing program before you insert it into QCad.
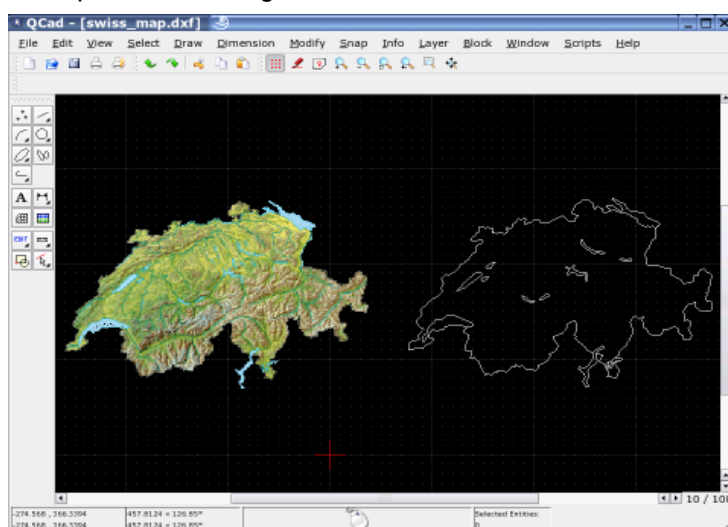4. Specify the insertion point of the image. That is the left bottom corner of the image.

Figure 48: Images are often used in the background of a drawing to manually vectorize bitmaps.

27. 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.

27.1. Deselecting All

CAD Toolbar:

Menu:
Select - Deselect All

Hotkey:
Ctrl-K

Command:
deselectall

Description:

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

27.2. Selecting All

CAD Toolbar:

Menu:
Select - Select All

Hotkey:
Ctrl-A

Command:
selectall

Description:

Selects all entities on all currently visible layers.

27.3. Selecting Single Entities

CAD Toolbar:

Menu:
Select - Select Entity

Command:

selectsingle

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.

27.4. Selecting Contours

CAD Toolbar:

Menu:

Select - (De-)Select Contour

Command:

selectcontour

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.

27.5. Window Deselection

CAD Toolbar:

Menu:

Select - Deselect Window

Command:

deselectwindow

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.

27.6. Window Selection

CAD Toolbar:

Menu:

Select - Select Window

Command:

selectwindow

Description:

Selects all entities that are entirely within a given window.

Procedure:

Analog to deselecting entities in a window.

27.7. Inverting Selection

CAD Toolbar:

Menu:

Select - Invert Selection

Command:

selectinvert

Description:

Selects all unselected entities and deselects all selected entities.

27.8. Deselect Intersected Entities

CAD Toolbar:

Menu:

Select - Deselect Intersected

Command:

deselectintersected

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.

27.9. Select Intersected Entities

CAD Toolbar:

Menu:

```
Select - Select Intersected
```

Command:

```
selectintersected
```

Description:

Selects all entities that are intersected by a line.

Procedure:

Analog to deselecting intersected entities.

27.10. Selecting Layers

CAD Toolbar:

Menu:

```
Select - (De-)Select Layer
```

Command:

```
selectlayer
```

Description:

Selects or deselects all entities on a layer.

Procedure:

Use the mouse to pick one entity of the layer you want to (de-)select. All entities that are on the same layer as the entity you have picked get selected if the entity you picked was unselected. Otherwise the whole layer will be deselected.

28. Modification

Modification tools allow the user to modify existing entities. This chapter is a brief overview over the modification tools of QCad (Figure 49).
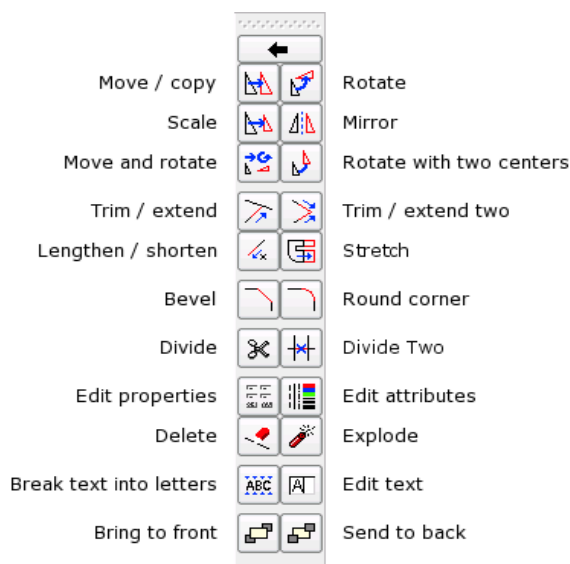
Figure 49: The modification toolbar.

28.1. Move / Copy

CAD Toolbar:

Menu:

`Modify – Move / Copy`

Hotkey:

`mv`

Command:

`move, copy`

Description:

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

Procedure:
1.   After starting the 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 coordinate in the command line.
4.   Set the target point. To move the selected entities by a given amount, enter a relative coordinate. E.g. to move it 50 units to the right, enter @50,0 in the command line.
5.   The move dialog is displayed (Figure 50). To move the entities, choose "Delete Original", to copy them choose "Keep Original". You can also 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 selected 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 check "Use current attributes" or "Use current layer".
6.   Click OK to move / copy the entities. The selected entities will be deselected automatically.
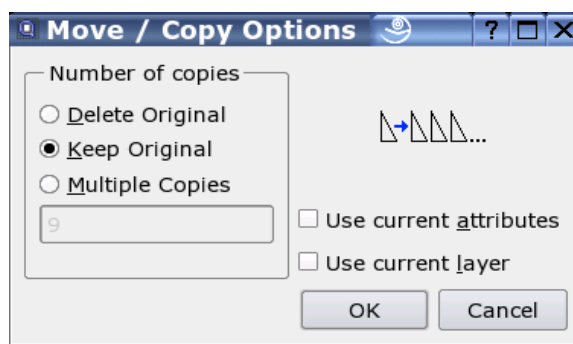
Figure 50: Move options dialog.

28.2. Rotate

CAD Toolbar:

Menu:

Modify - Rotate

Hotkey:

ro

Command:

rotate

Description:

Rotates entities by a given angle around a given center. This tool can create single or multiple rotated 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 coordinate in the command line.
4.   The rotate dialog is displayed (Figure 51). 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 check "Use current attributes" or "Use current layer".
5.   Click OK to rotate the entities. The selected entities will be deselected automatically.
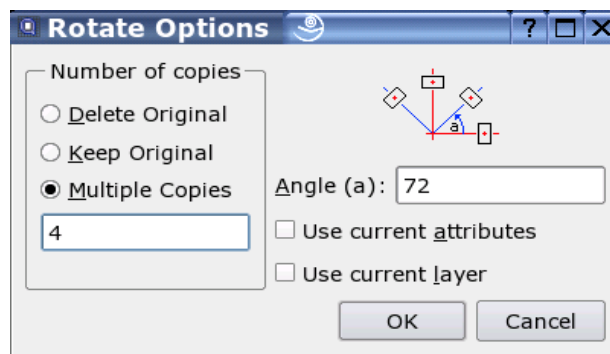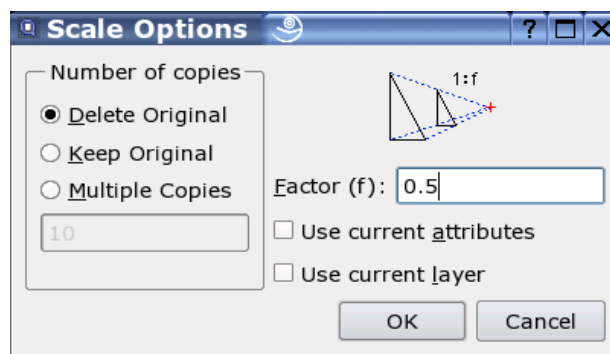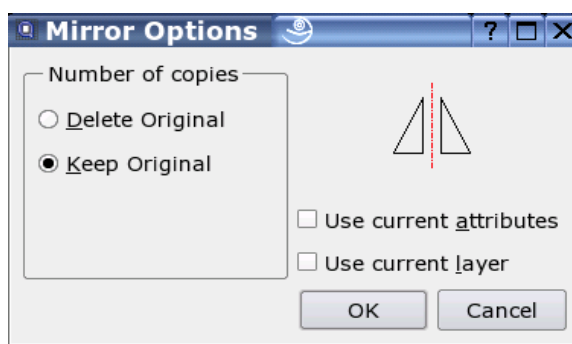

Figure 51: Rotate options dialog.

28.3. Scale

CAD Toolbar:

Menu:

Modify - Scale

Scale:

sz

Command:

scale

Description:

Scales entities by a given factor towards 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 coordinate in the command line.
4. The scale dialog is displayed (Figure 52). 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 check "Use current attributes" or "Use current layer".
5. Click OK to scale the entities. The selected entities will be deselected automatically.

Figure 52: Scale options dialog.

28.4. Mirror

CAD Toolbar:

Menu:

Modify - Mirror

Hotkey:

mi

Command:

`mirror`

Description:

Mirrors entities along 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 axis with the mouse or enter a coordinate in the command line.
4.   Set the second endpoint of the mirror axis.
5.   The mirror dialog is displayed (Figure 53). 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 check "Use current attributes" or "Use current layer".
6.   Click OK to mirror the entities. The selected entities will be deselected automatically.

Figure 53: Mirror options dialog.

28.5. Move and Rotate

CAD Toolbar:

Menu:

`Modify - Move and Rotate`

Command:

`moverotate`

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 coordinate 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 54, 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 55). 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 check "Use current attributes" or "Use current layer".

7.   Click OK to move and rotate the entities. The selected entities will be deselected automatically.
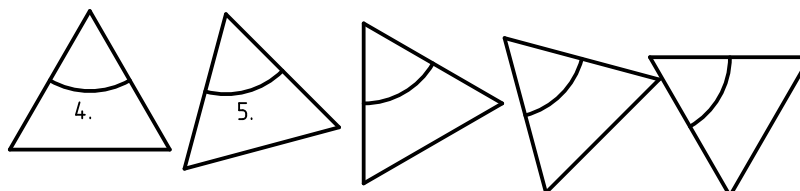

Figure 54: Move and Rotate reference point (4.) and target point (5.).
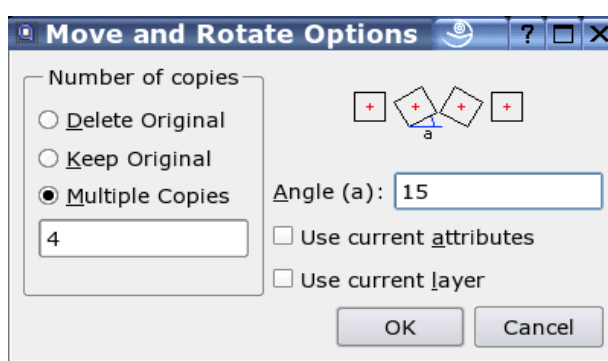
Figure 55: Move and Rotate options dialog.

28.6. Rotate and Counter-rotate (Rotate Two)

CAD Toolbar:

Menu:

Modify - Rotate Two

Command:

rotate2

Description:

Rotates and counter-rotates entities. This tool can create single or multiple copies of existing entities. This tool can be useful 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 coordinate 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 56).
5.   The rotate 2 dialog is displayed (Figure 57). 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 '7' will create 7 copies and keep the original - so there will be 8 instances of the entities in the end

(8 instances evenly distributed over the full circle in the example). The copies will have the same attributes and be on the same layer as the original entities. To change that default behavior, you can check "Use current attributes" or "Use current layer".

6.   Click OK to move and rotate the entities. The selected entities will be deselected automatically.
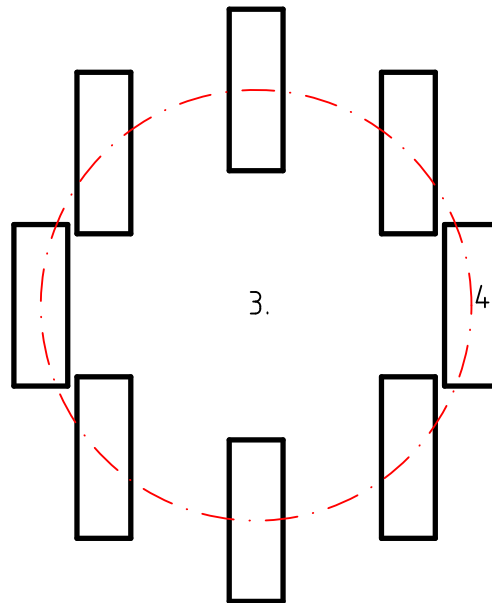
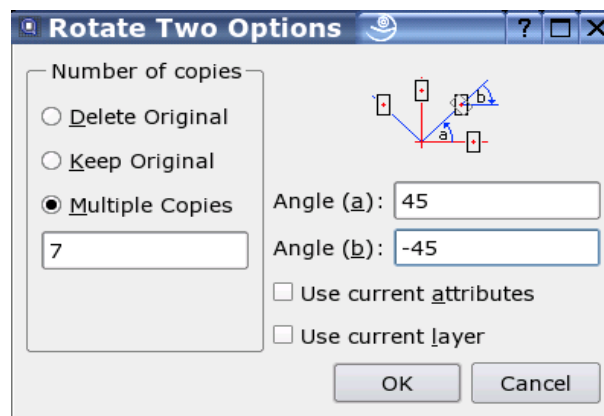**Figure 56:** Rotate around two centers, an absolute center (3.) and a relative center (4.).

Figure 57: Rotate two options dialog.

28.7. Trim / Extend

CAD Toolbar:

Menu:

Modify - Trim

Hotkey:

xt

Command:

trim, extend

Description:

Trims or extends a line or arc to meet another entity (Figure 58).
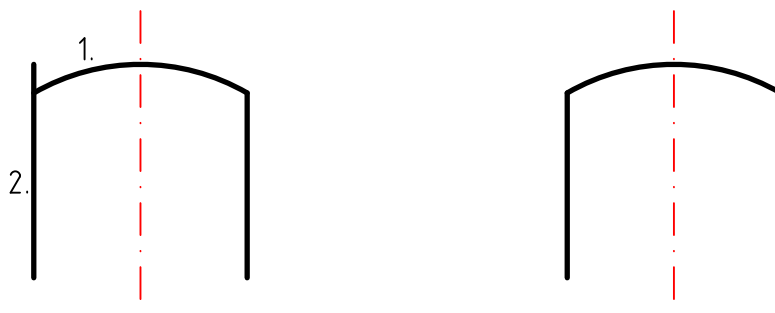
Figure 58: With the trim tool, the line at the left can be easily shortened to meet the arc.

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 58 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.

28.8. Trim Two Entities

CAD Toolbar:

Menu:

Modify - Trim Two

Hotkey:

tm

Command:

trim2

Description:

Trims or extends two line or arc entities at once to meet each other at their intersection point (Figure 59).
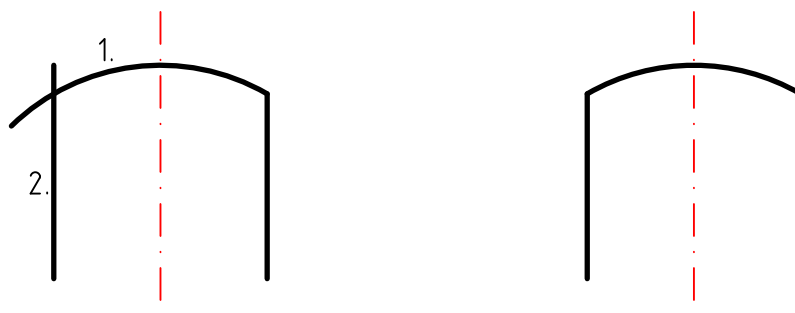
Figure 59: With the second trim tool, the line and the arc can both be trimmed to build an edge in one step.

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.

28.9. Lengthen

CAD Toolbar:

Menu:

Modify - Lengthen

Command:

lengthen, trimamount

Options Toolbar:
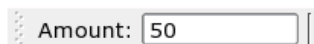
Amount: 50

Description:

The lengthen command can be used to either lengthen or shorten lines or arcs by a given amount without the use of limiting entities.

Figure 60: 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 units.
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.

28.10. Stretch

CAD Toolbar:

Menu:

Modify - Stretch

Hotkey:

ss

Command:

stretch

Description:

Stretches contours. This can also be decribed as moving all endpoints within a given window.
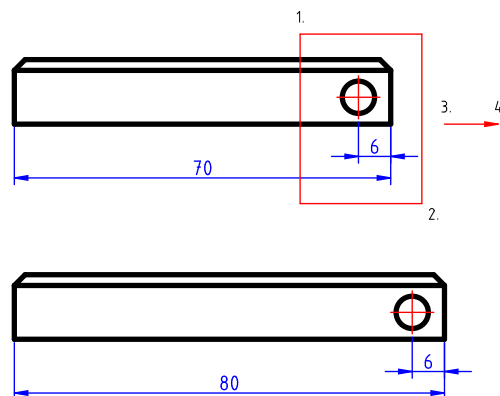
Figure 61: The stretch tool lets you lengthen or shorten complex constructions with little effort.

Procedure:

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

28.11. Bevel

CAD Toolbar:

Menu:

Modify - Bevel

Hotkey:

ch

Command:

bevel, chamfer

Options Toolbar:

Description:

Bevels a corner shaped by two entities. Optionally, the edge entities of the corner can be trimmed automatically to fit the new shape.
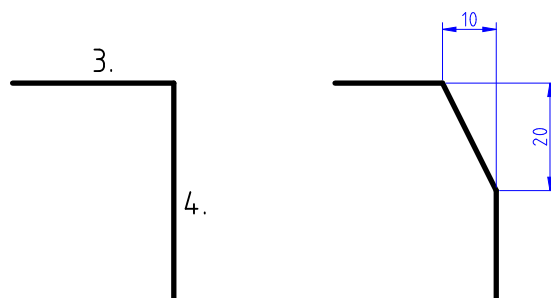
Figure 62: The bevel tool with trimming enabled.

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 (10 in the example shown). 'Distance 2' is the same distance for the second edge (20 in the example shown).
2.   Check 'Trim' if you want to trim the entities automatically. If the option is disabled, the two entities will remain untouched.
3.   Choose the first edge entity (a line or an arc).
4.   Choose the second entity.

28.12. Round

CAD Toolbar:

Menu:

Modify - Round

Hotkey:

rn

Command:

round

Options Toolbar:
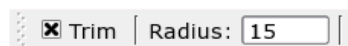
Description:

Rounds the corner of a contour. Optionally, the edge entities of the corner can be trimmed automatically to fit the new shape.
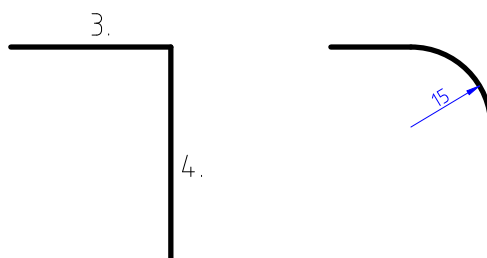
Figure 63: The round tool with trimming enabled.

Procedure:
1.   Enter the radius of the rounding in the options toolbar.

2.   Check 'Trim' if you want to trim the entities automatically. If the option is disabled, the two corner entities will remain untouched.
3.   Choose the first edge entity (a line or arc).
4.   Choose the second entity.

28.13. Divide

CAD Toolbar:

Menu:

Modify - Divide

Hotkey:

di

Command:

divide

Description:

Divides entities into two separate entities.

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

28.14. Divide Two

CAD Toolbar:

Menu:

Modify - Divide 2

Hotkey:

d2

Command:

divide2

Description:

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

Procedure:
1.   Choose the entity you want to divide by clicking the segment you want to cut out. The segment is removed.

28.15. Properties

CAD Toolbar:

Menu:

Modify - Properties

Hotkey:

pr

Command:

properties

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 the geometry of the entity.

Procedure:

1.   Click the entity to modify.
2.   Change the attributes or geometry values
3.   Click OK

28.16. Attributes

CAD Toolbar:

Menu:

Modify - Attributes

Hotkey:

at

Command:

attributes

Description:

Modify the attributes of a selection 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 remain untouched.

28.17. Deleting Entities

CAD Toolbar:

Menu:

Modify - Delete

Hotkey:

er

Command:

```
delete, erase
```

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.

28.18. Exploding Groups

CAD Toolbar:

Menu:

```
Modify - Explode
```

Hotkey:

```
xp, ex
```

Command:

```
explode
```

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.

28.19. Explode Text into Letters

CAD Toolbar:

Menu:

```
Modify - Explode Text into Letters
```

Command:

```
explodetext
```

Description:

Converts text entities into several smaller text entities, each containing one letter of the original text.

Procedure:

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

28.20. Edit Text

CAD Toolbar:

Menu:

```
Modify - Modify Text
```

Hotkey:

```
pr
```

Command:

```
properties
```

Description:

This function is equivalent to 'Modify, Properties' for text entities.

28.21. Bring to Front / Send to Back

CAD Toolbar:

Menu:

```
Modify - Bring to Front
Modify - Send to Back
```

Command:

```
tofront
toback
```

Description:

Changes the order in which entities are displayed. Use these functions to bring entities into the foreground if they are hidden behind images, solid fills or other entities or to send entities that are hiding other entities into the background.

Procedure:

1. After starting the tool, the CAD toolbar shows the selection tools. Use them to select the entities you want to bring into the foreground / send into the background.
2. Click the right arrow button in the CAD toolbar to continue.

29. Getting Measurements

29.1. Distance (Point, Point)

CAD Toolbar:

Menu:

Info - Distance Point to Point

Command:

infodist, distance

Description:

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

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

29.2. Distance (Entity, Point)

CAD Toolbar:

Menu:

Info - Distance Entity to Point

Command:

infodist2, distance2

Description:

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

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

29.3. Angle

CAD Toolbar:

Menu:

Info - Angle between two lines

Command:

```
infoangle, angle
```

Description:

This tool measures the angle between two given lines.

Procedure:

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

29.4. Total Length of Selected Entities

CAD Toolbar:

Menu:

```
Info - Total length of selected entities
```

Command:

```
infosum
```

Description:

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

Procedure:

1.   After starting the tool, the CAD toolbar shows the selection tools. Use them to select the entities you want calculate the total length of.
2.   Click the right arrow button in the CAD toolbar to continue.
3.   The total length of the selected entities is printed in the command line.

29.5. Area

CAD Toolbar:

Menu:

```
Info - Polygonal Area
```

Command:

```
infoarea, area
```

Description:

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

Procedure:

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

30. Part Library

The part library of QCad is a collection of symbols and other drawing files that can be reused in your drawings. To insert an item from the part library, you need to show the part library browser first using the menu:

`View - Toolbars - Library Browser`

After that, the part library browser window is displayed as shown in Figure 64.
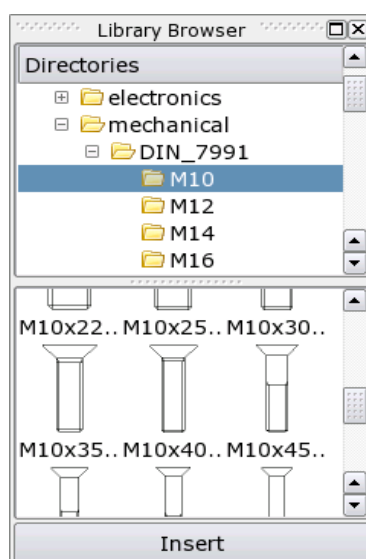
Figure 64: The Part Library Browser.

30.1. Inserting Parts

Options Toolbar:

Procedure:
1. The upper part of the browser shows a tree structure of all available part libraries. To show the contents of a folder, click on the name of the folder. Double-click on a folder to show its sub-folders.
2. Choose the item you want to insert from the lower part of the browser.
3. Click the 'Insert' button.
4. Before you place the item in your drawing, specify a scale factor and rotation angle for it in the options toolbar.
5. Now, place the item in your drawing with the mouse or by typing a coordinate in the command line.

Note that inserting a part from the library browser creates a block in your drawing for that part. If you need more than one instance of a part, please insert instances of the block from the block list of the drawing. Inserting the same part or symbol from the part library browser more than once will create multiple blocks in your drawing - something that should be avoided.

30.2. Extending The Part Library

You can easily extend the part library of QCad by saving any drawing into the directory tree in the 'library' folder of your QCad installation. Part library drawings are stored in the same format as any

other drawing (DXF). Part library drawings should not contain blocks. When a part is inserted into a QCad drawing, a block is automatically created for the whole part.

31. Application Preferences

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

```
Edit - Application Preferences
```

31.1. Appearance

Figure 65 shows the application preferences dialog.
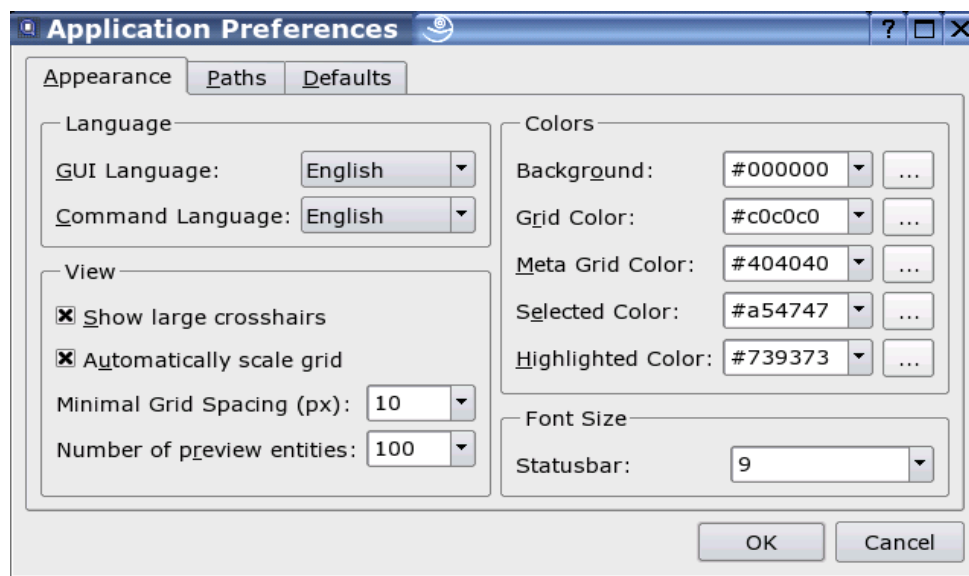
Figure 65: Application preferences for QCad.

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.

View

Check "Show large crosshairs" to enable the crosshairs of the mouse cursor to extend to the drawing borders.
Check "Automatically scale grid" to change the grid spacing automatically to best fit the current drawing view. You might want to switch this off if you prefer a fixed grid spacing.
The grid is switched off or increases its spacing when the distance between two grid points in pixels falls below "Minimal Grid Spacing" in pixels.
If you experience performance problems for example 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.

Colors

In the color area of the 'Appearance' tab, you can adjust the colors used for elements of the graphic view. Use the button at the right side of the combo box to show a color selection dialog.

31.2. 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 66). You can enter multiple paths by separating them with a semicolon (;).
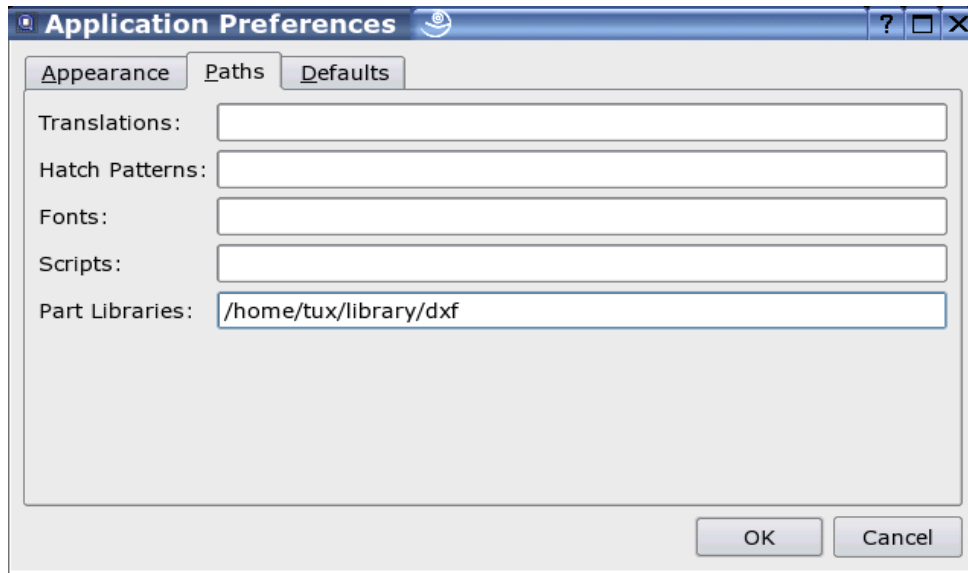
Figure 66: Configuration of individual paths.

32. Drawing Preferences

The drawing preferences dialog applies to the current drawing only. It is used to set the drawing units, paper size, dimensioning preferences and other drawing specific settings.

```
Edit - Current Drawing Preferences
```

32.1. 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 67).
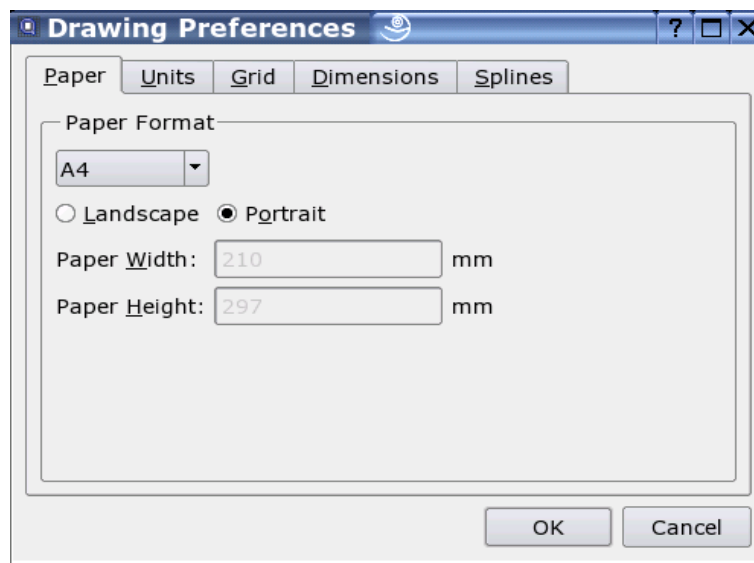
Figure 67: Paper settings for the current drawing.

32.2. Units

The chosen units and unit formats for lengths and angles are used for displaying coordinates in the statusbar as well as for formatting 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 68).
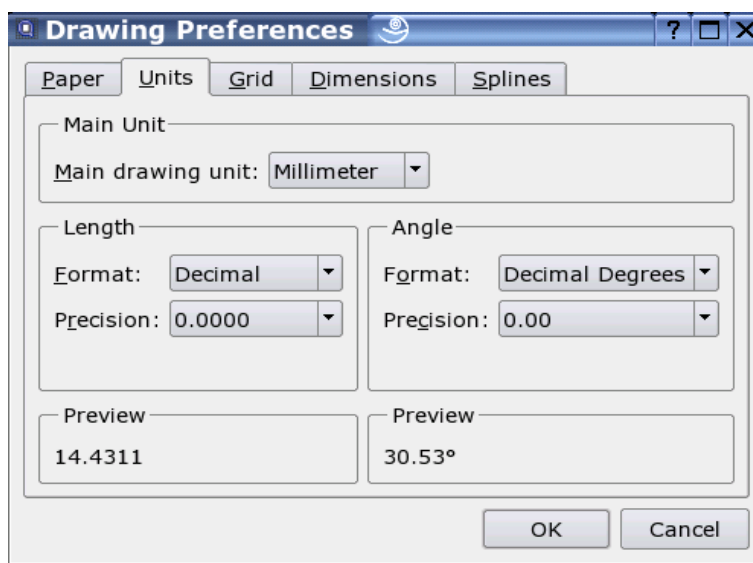
Figure 68: Unit and unit format.

32.3. Grid

In this section of the drawing preferences dialog you can adjust the grid spacing and visibility (Figure 69). The default grid spacing is 'auto' which means that the grid scales automatically to a reasonable spacing for the current scale factor of the drawing. You can also set the grid spacing to any preferred fixed values.
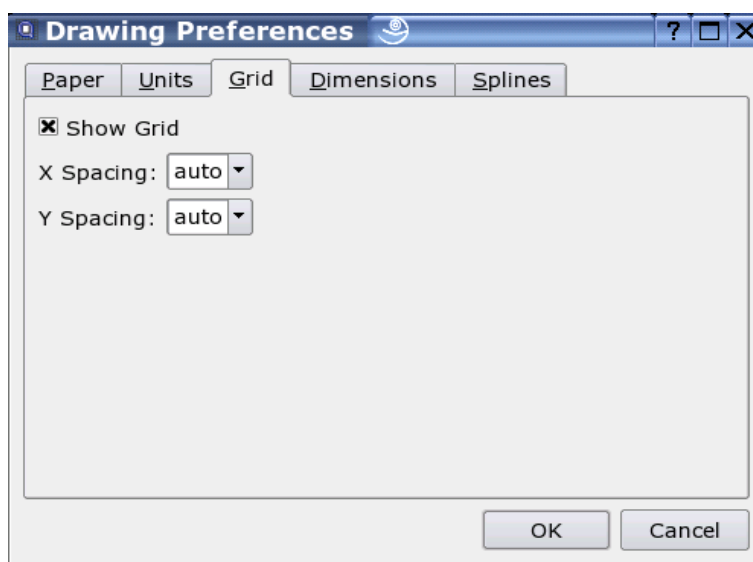
Figure 69: Grid settings.

32.4. Dimensions

On this tab you can change the measures for the dimension labels, the dimension arrow size and other measures used for dimensioning (Figure 70).
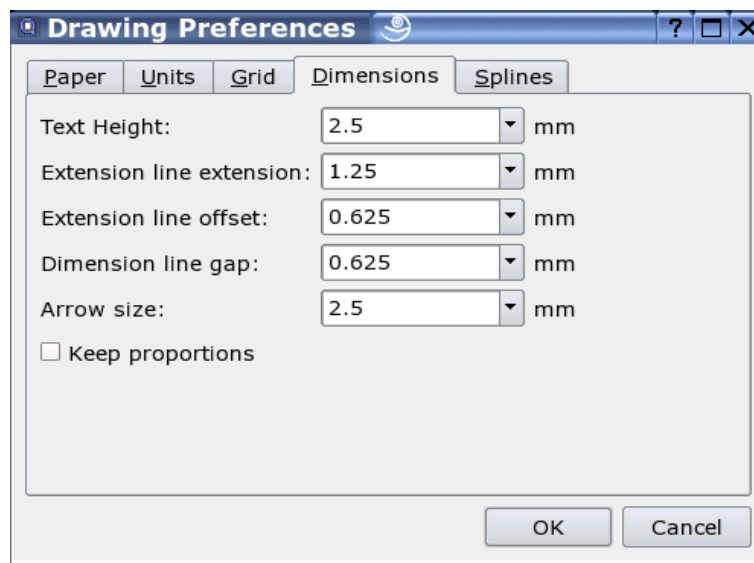
Figure 70: Dimension preferences.
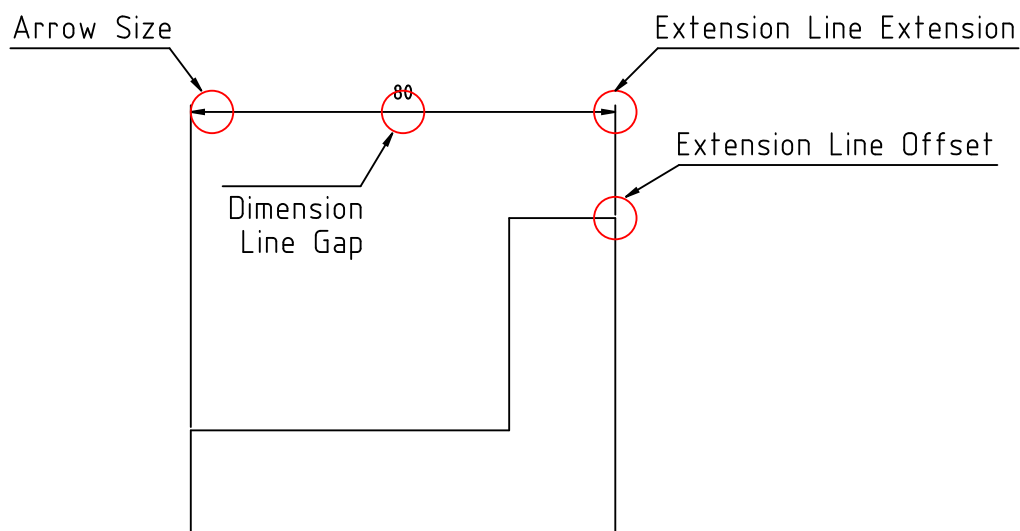
Figure 71: Dimension measures.

32.5. Splines

QCad interpolates spline curves with small line segments. In this tab you can adjust the accuracy of this interpolation. A lower value means that the spline quality is poor but the display performance is better. A higher value means that the splines are shown in a higher quality which can reduce the display performance (Figure 72).
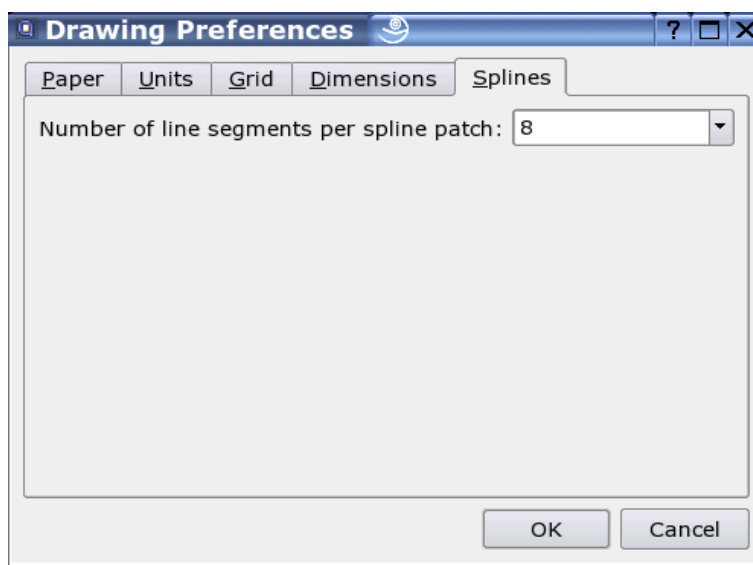
Figure 72: Spline settings.

33. Appendix

The following chapters contain some lists and tables with fonts, hatch patterns, supported mathematical expressions, supported units and paper formats.

34. Hotkeys

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

Some commands have more than one key combination. This is mostly for better 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 are using Mac OS X this might be Option-O or Alt-O depending on your keyboard.

Under Mac OS X, use the Option key where the Ctrl key is indicated.

34.1. Command Line

Space / Option-M	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 Up / Down	Recall the last command inputs
Tab	Show all currently available commands or complete the first letters of a command into the full command

34.2. 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

34.3. Basic Editing

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

34.4. View

rd	Redraw
zw	Zoom Window

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

34.5. Layer Handling

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

34.6. Snapping

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

34.7. Construction

po	Point
li	Line
la	Line with given angle
pa, of	Parallel / Offset
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

34.8. Dimensioning

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

34.9. Entity Selection

tn, Ctrl-K	Deselect all

Ctrl-A   Select all

34.10. Modification

ch   Bevel (chamfer)
rn   Round
di   Divide
rm, xt   Trim, extend
tm   Multi line trim, extend
mv   Move
mi   Mirror
ro   Rotate
sz   Scale
ss   Stretch
pr   Entity properties
at   Entity attributes
er   Erase (Delete)
xp, ex   Explode

35. Fonts

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

Figure 73: Courier

Figure 74: Cursive

Figure 75: Cyrillic II

Figure 76: Gothgbt

Figure 77: Gothgrt

Figure 78: Gothitt

Figure 79: Greek_ol

Figure 80: Greekc

Figure 81: Greekcs

Figure 82: Greekp

Figure 83: Greeks

Figure 84: Iso8859-11

A B C   a b c
Figure 85: Italicc

A B C   a b c
Figure 86: Italiccs

A B C   a b c
Figure 87: Italict

ABC abc
Figure 88: Kochigothic

ABC abc
Figure 89: Kochimincho

ABC abc
Figure 90: Normal

ABC abc
Figure 91: Normallatin1

ABC abc
Figure 92: Normallatin2

A B C   a b c
Figure 93: Romanc

A B C   a b c
Figure 94: Romancs

ABC abc
Figure 95: Romand

A B C   a b c
Figure 96: Romanp

ABC abc
Figure 97: Romans

ABC abc
Figure 98: Romans2

A B C   a b c

Figure 99: Romant

A B C   a b c

Figure 100: Scriptc

A B C   a b c

Figure 101: Scripts

ABC abc

Figure 102: Standard

ABX αβχ

Figure 103: Symbol

Ω ℳ ↗

Figure 104: Symbol_astro

| \ ⟍   ○ □ △

Figure 105: Symbol_misc1

● ♯ ♮ ♭ ▬ - ♦ ♪ 𝄞 𝄢 𝄡

Figure 106: Symbol_misc2

ABC abc 123
ΛДЙ 両並中丰

Figure 107: Unicode

36. Patterns
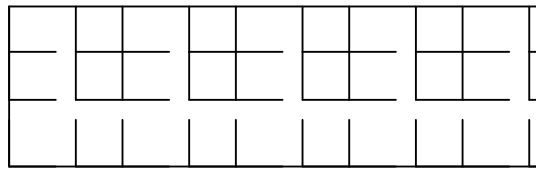
Figure 108: Angle
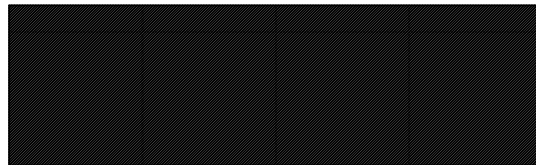
Figure 109: Ansi31
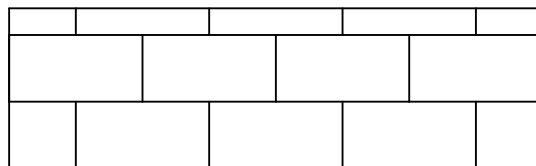
Figure 110: Ar-b816
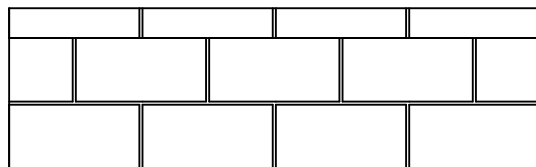
Figure 111: Ar-b816c
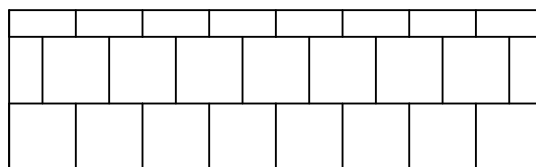
Figure 112: Ar-b88
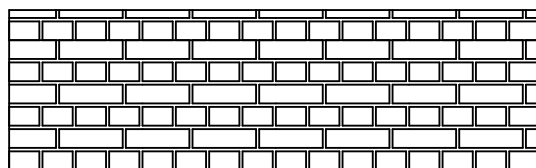
Figure 113: Ar-brelm
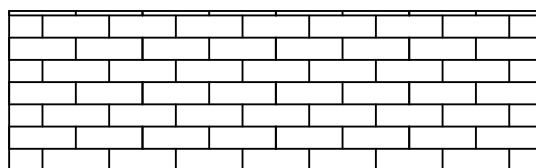
Figure 114: Ar-brstd

Figure 115: Ar-conc
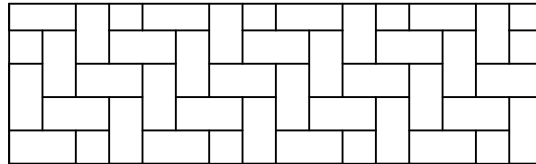
Figure 116: Ar-hbone
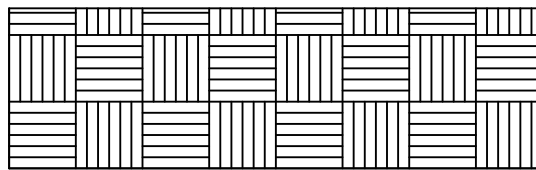
Figure 117: Ar-parq1
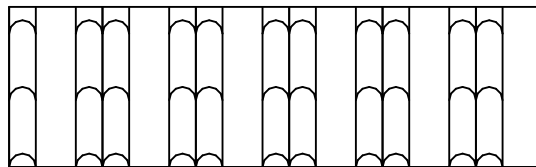
Figure 118: Ar-roof
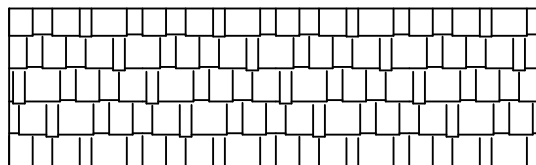
Figure 119: Ar-rshke
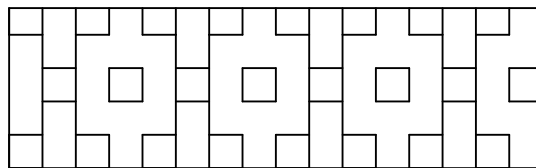
Figure 120: Box
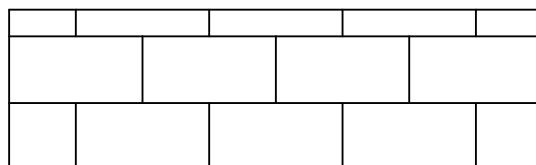
Figure 121: Brick
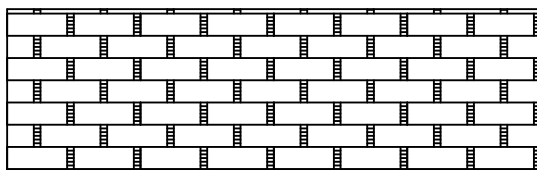
Figure 122: Brstone
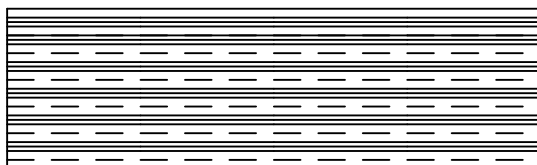
Figure 123: Clay
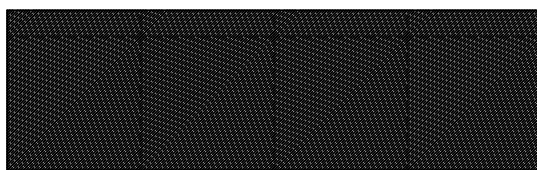
Figure 124: Concrete
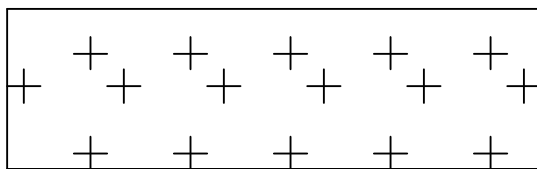
Figure 125: Cross
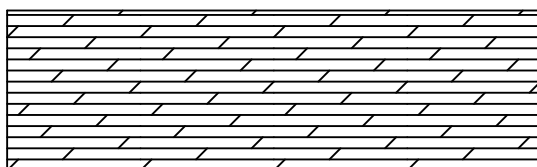
Figure 126: Dolmit
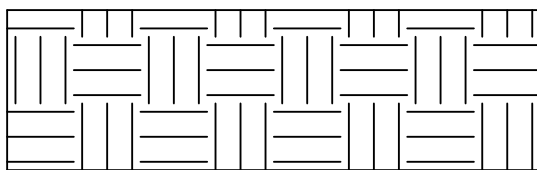
Figure 127: Earth
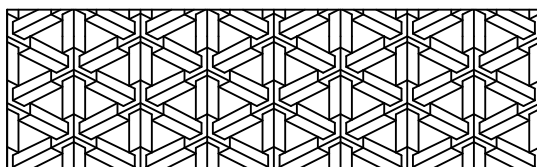
Figure 128: Escher
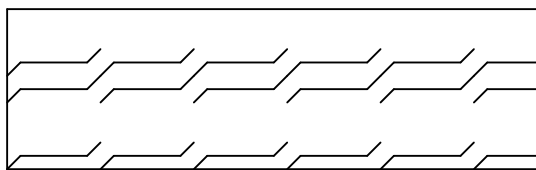
Figure 129: Flex
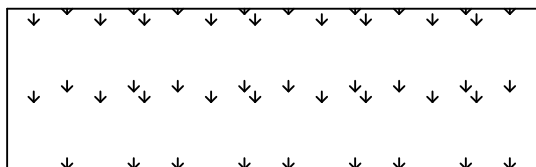
Figure 130: Grass
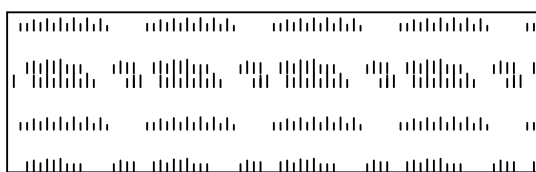
Figure 131: Grass_b
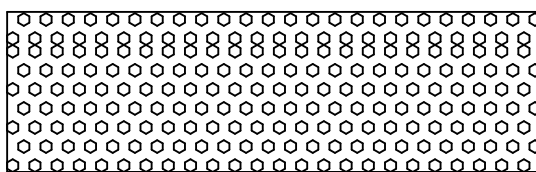
Figure 132: Hex
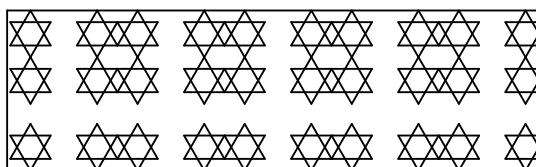
Figure 133: Hexagon_a
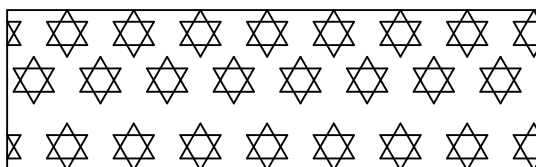
Figure 134: Hexagon_b
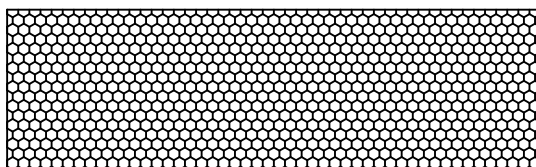
Figure 135: Honeycomb
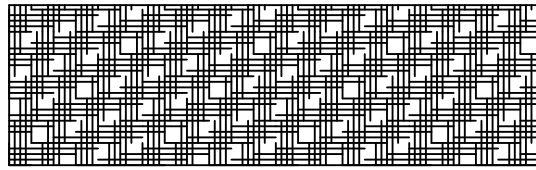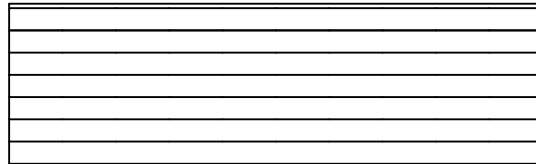
Figure 136: Hound

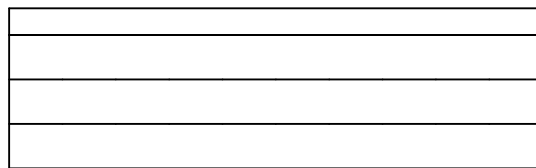

Figure 137: iso03w100

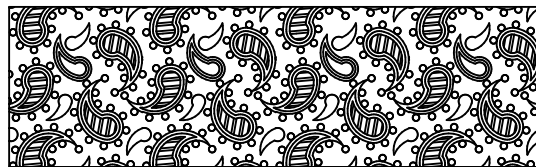
Figure 138: Iso03w100a

Figure 139: Paisley

Figure 140: Pantagon_a

Figure 141: Pantagon_b
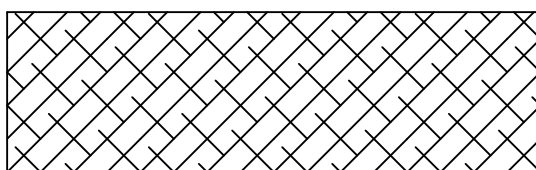

Figure 142: Plastic

Figure 143: Sacncr
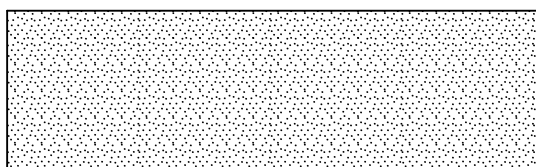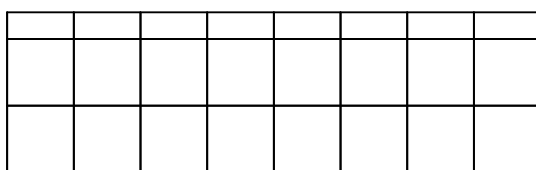

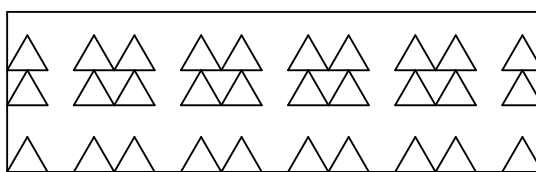**Figure 144:** Sand

Figure 145: Square
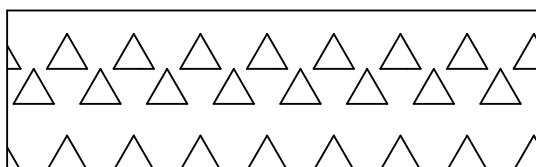

Figure 146: Triangle_a

Figure 147: Triangle_b

37. Mathematical Expressions

QCad comes with a built in mathematical expression parser called 'fparser'. The parser was written by Juha Nieminen 'Warp'. 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.

37.1. Supported Constants

pi	3.14159265

37.2. 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'.

abs(A)	Absolute value of A.
acos(A)	Arc-cosine of A. Returns the angle, measured in degrees, whose cosine is A.
acosh(A)	Same as acos() but for hyperbolic cosine.
asin(A)	Arc-sine of A. Returns the angle, measured in degrees, whose sine is A.
asinh(A)	Same as asin() but for hyperbolic sine.
atan(A)	Arc-tangent of (A). Returns the angle, measured in degrees, whose tangent is (A).
atan2(A,B)	Arc-tangent of A/B. The two main differences to atan() is that it will return the right angle depending on the signs of A and B (atan() can only return values between -pi/2 and pi/2), and that the return value of pi/2 and -pi/2 are possible.
atanh(A)	Same as atan() but for hyperbolic tangent.
ceil(A)	Ceiling of A. Returns the smallest integer greater than A. Rounds up to the next higher integer.
cos(A)	Cosine of A. Returns the cosine of the angle A, where A is measured in degrees.
cosh(A)	Same as cos() but for hyperbolic cosine.
cot(A)	Cotangent of A (equivalent to 1/tan(A)).
csc(A)	Cosecant of A (equivalent to 1/sin(A)).
eval(...)	This is 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 if() to avoid infinite recursion.
exp(A)	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.
floor(A)	Floor of A. Returns the largest integer less than A. Rounds down to the next lower integer.
if(A,B,C)	If int(A) 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 eval() in them.
int(A)	Rounds A to the closest integer. 0.5 is rounded to 1.
log(A)	Natural (base e) logarithm of A.
log10(A)	Base 10 logarithm of A.
max(A,B)	If A>B, the result is A, else B.
min(A,B)	If A<B, the result is A, else B.
sec(A)	Secant of A (equivalent to 1/cos(A)).
sin(A)	Sine of A. Returns the sine of the angle A, where A is measured in degrees.
sinh(A)	Same as sin() but for hyperbolic sine.
sqrt(A)	Square root of A. Returns the value whose square is A.
tan(A)	Tangent of A. Returns the tangent of the angle A, where A is measured in degrees.
tanh(A)	Same as tan() but for hyperbolic tangent.

38. Drawing Units

QCad supports various units for lengths and angles. This is because 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.

38.1. 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)

38.2. 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")

38.3. Supported Angle Units

Degrees (full circle = 360°)

Radians (full circle = 2 ¶)
Gradians (full circle = 400g)

38.4. 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)

39. 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.

39.1. 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

39.2. Other Formats

Letter: 215.9 x 279.4
Legal: 215.9 x 355.6
Executive: 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

40. 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.

40.1. 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 148.
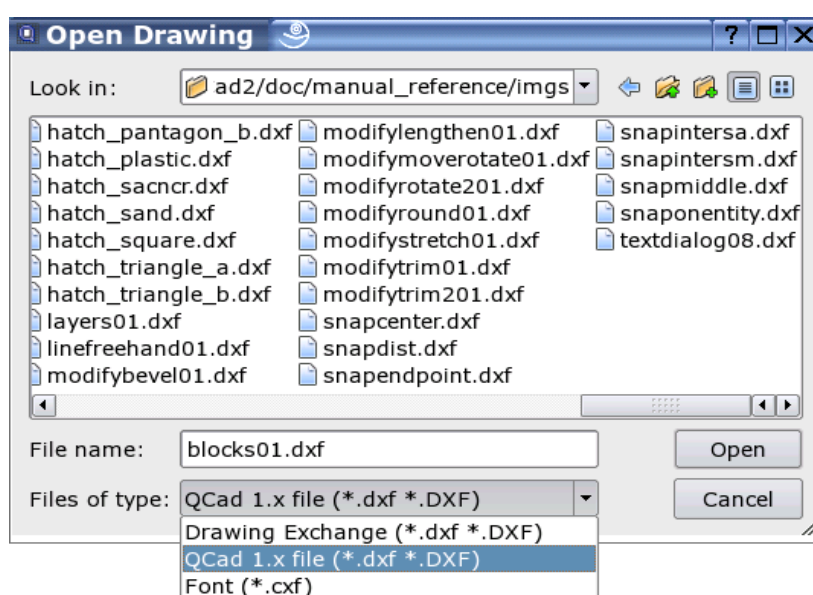

Figure 148: Opening QCad 1 drawings.

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.

40.2. Layer Handling

Changeing 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, tou might find that the attributes of the entities in old drawings are no longer appropriate. Usually it is 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 149.
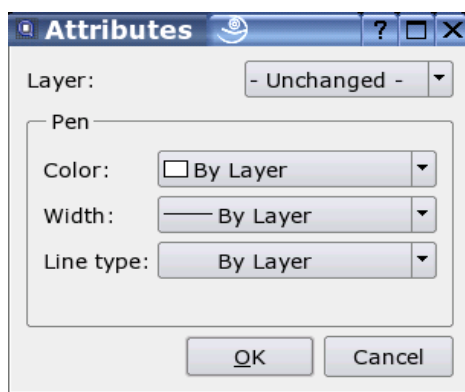
Figure 149: Changing all attributes to 'By Layer'.

41. Bibliography

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