

QCAD

An Introduction to Computer-
Aided Design (CAD)

Andrew Mustun

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Mustun, Andrew QCAD - An Introduction to Computer-Aided Design (CAD)

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Chapter 2

Introducing the QCAD Application

Objective

In this chapter, you will

- get familiar with the QCAD application window,
- learn how to use menus,
- customize the user interface to better suit the needs of subsequent chapters,
- find out how to launch the integrated help system.

The First Start

When starting QCAD for the first time, it asks you what default unit and user interface language you want to use (see Figure 2-1). Choose the unit 'Millimeters' for now. You can always change this setting later in the application preferences or drawing preferences.

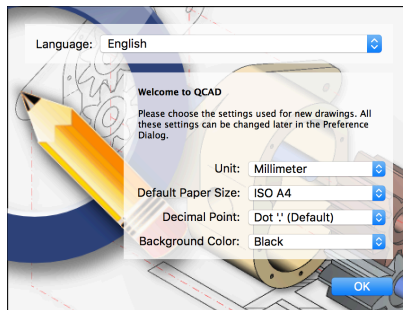


Figure 2-1: This dialog is only shown when starting QCAD for the first time.

The Application Window

After launching QCAD, the application window is shown (Figure 2-2).

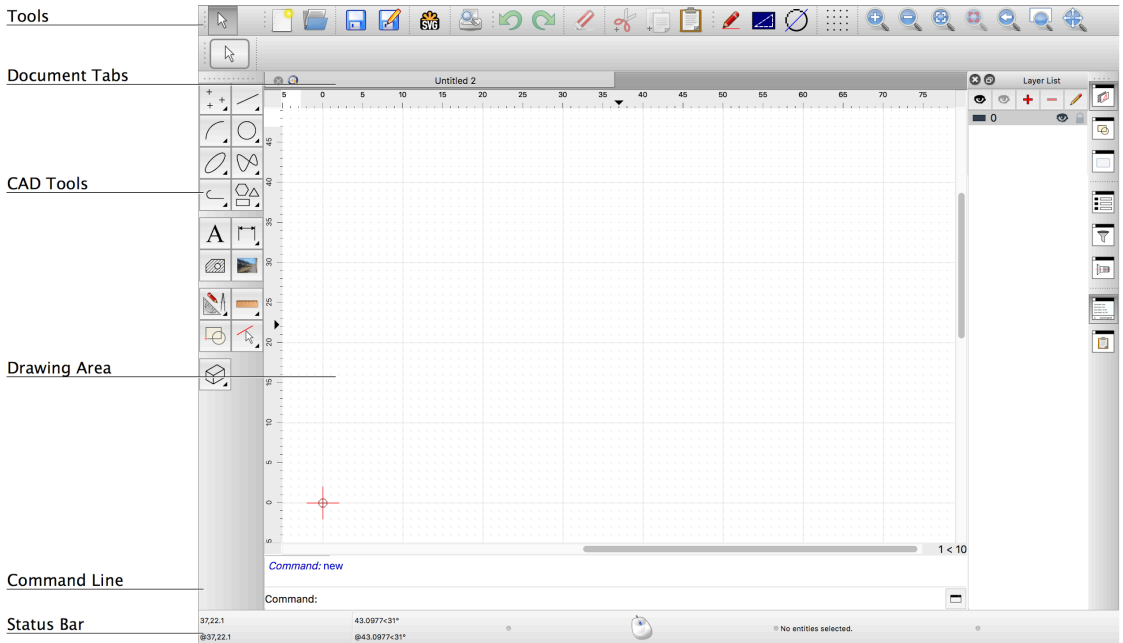


Figure 2-2: The QCAD application window.

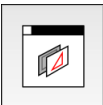
Most of the application window consists of the drawing area which is used to display your drawing. The drawing area contains many evenly arranged small dots. These are grid dots that have a similar function like the horizontal and vertical lines of squared paper (or graph paper). You will learn later in this book how to use this grid in QCAD.

The various toolbars, controls and other user interface components are arranged around the drawing area.

Hands-on: Customizing the Application Window

As you might have noticed, some of the user interface components are not labeled in Figure 2-2. We will not use them for the first steps with QCAD. For this reason, please follow these simple steps to hide them for now:

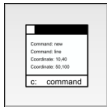
1. Click the menu *View > Layer List* to hide the layer list. Alternatively, you can also click the button to toggle the layer list in the toolbar at the right:



2. The property editor is also shown initially (as separate window). Hide it with menu *View > Property Editor* or by clicking this button:



3. Another component you will not need for the first few chapters is the command line. To switch it off choose the menu *View > Command Line* or click the button to toggle the command line:



4. The application window should now look as shown in Figure 2-3.

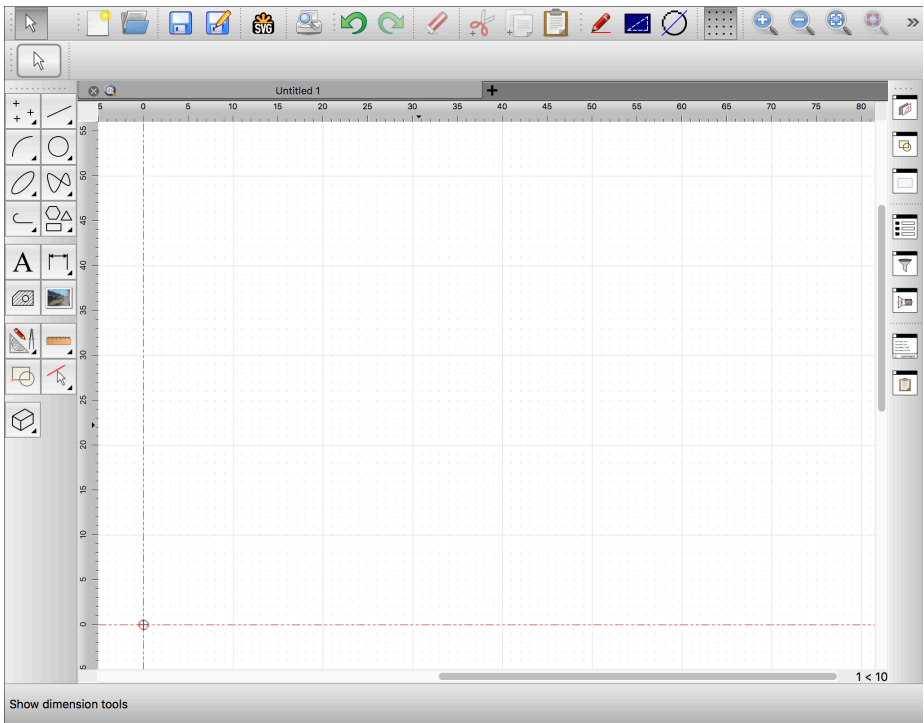


Figure 2-3: The QCAD application window after hiding the layer list, the property editor and the command line.

Chapter 3

Using CAD Tools

Objective

In this chapter, you will

- create your first simple drawing,
 - learn how to choose tools from the CAD toolbar,
 - get an idea of what it means to create precise drawings with QCAD,
 - learn how to save your drawings,
 - print your first drawing.
-

The CAD Toolbar

At the left side of the QCAD application window you can find the CAD toolbar. The CAD toolbar offers all CAD specific tools that you will use to draw and modify drawings. Unlike other toolbars, the CAD toolbar always shows only those tools which currently make most sense to be used. Right after the start of QCAD, the CAD toolbar shows the main tool set as shown in Figure 3-1.

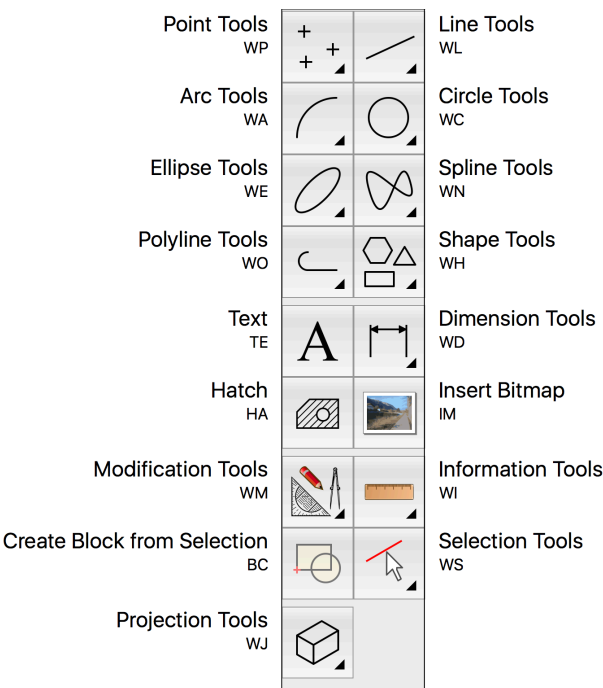


Figure 3-1: The CAD toolbar after starting QCAD.

Some of the buttons have a small black triangle arrow in the lower right corner. This indicates that the button represents a whole category of tools. For example the line tools button (the button at the top right in Figure 3-1) represents all tools for drawing lines.

Especially for beginners, the icons shown in the CAD toolbar might not be obvious to understand. For this reason, QCAD shows you a hint if you move the mouse cursor on top of the button without clicking a mouse button. These hints are often called *tooltips*. Figure 3-2 shows the tooltip for the previously mentioned line button. Tooltips appear not only for the buttons in the CAD toolbar but also for most other buttons in the QCAD application.

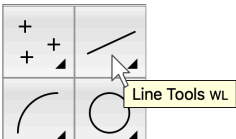


Figure 3-2: Tooltips provide useful information about the buttons in the CAD toolbar.

Starting Tools

As in most applications, there is more than one way to start a tool in QCAD. The application menus list most available tools and offer a convenient way to browse through tools by name if you are not sure what exactly you are looking for. Some frequently used tools can also be started from the toolbars at the top. For CAD specific tools, the CAD toolbar at the left can be used to browse and start the various tools. If you are planning to use QCAD on a regular basis, it might be

worth learning the keycodes for some tools you use a lot. Keycodes in QCAD are combinations of two or three letters you can type shortly after each other to launch a tool.

For example to start drawing lines, you can either choose the menu *Draw > Line > Line from 2 Points* or click the appropriate buttons in the CAD toolbar or you can press the *L* key on your keyboard followed by the *I* key. Entering this keycode *LI* is the quickest way to start the tool to draw lines. Figure 3-1 and similar figures in subsequent chapters list these keycodes below the tool name.

For the purpose of this book, we describe how to use the CAD toolbar on the left for CAD tools and menus for some other tools. Keycodes are mentioned in tool overviews for your reference.

Correcting Mistakes

When working through the step by step instructions of this book, you should always keep the following two tools in mind. There is no need to start from scratch if something goes wrong.

Undo

Menu: Edit > Undo
Keycodes: OO, Ctrl-Z (Mac: ⌘Z)



Whenever you are working with the CAD tools to draw, modify or delete objects you can use the *Undo* tool to correct a mistake you have just made. You might for example draw a line at the wrong position, accidentally delete something or drag a line to another location and then decide you want to move it back to its original location. Simply choose the menu *Edit > Undo* or click the undo button shown above to undo the mistake.

Redo

Menu: Edit > Redo
Keycodes: UU, Ctrl-Shift-Z (Mac: ⌘⇧Z)



Should you then decide that the modification was correct after all, there is also a *Redo* tool, the counterpart of undo.

The Neutral State of QCAD

Menu: Edit > Reset
Keycode: QQ



After the start, QCAD is in its neutral state. That means that no special tool is active and QCAD waits for you to click a menu or tool button to start a tool and start doing something.

If you get lost in a tool while working with QCAD, you can easily return to this neutral state by clicking the arrow tool button at the top left.

Alternatively, you can also click the right button of your mouse to return back to the neutral state step by step. Depending how far you have progressed with a tool, you might have to click the right mouse button more than once to fully return to the neutral state. The same can also be achieved by hitting the Escape key on your keyboard a multiple times.

Hands-on: Drawing a Rectangle

The following instructions guide you through the complete procedure of drawing a simple rectangle. You will probably not yet understand all steps involved but it is crucial that you successfully complete these steps since all CAD tools work in a similar way like the rectangle tool.

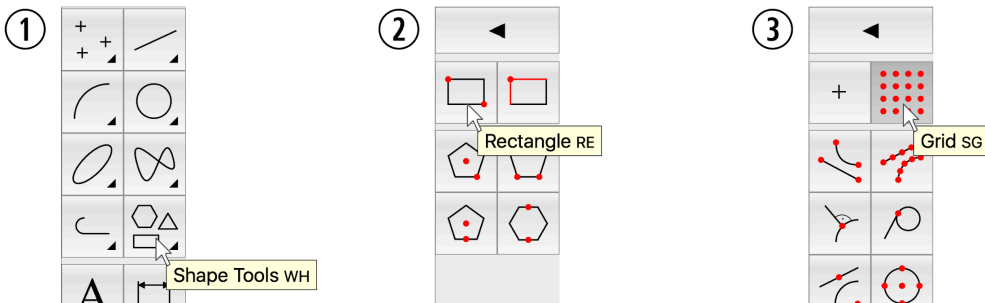
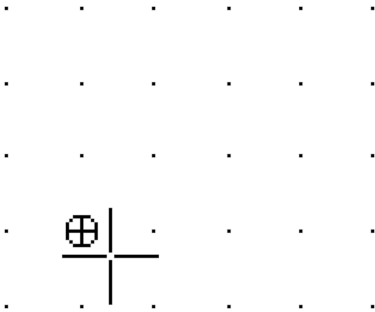


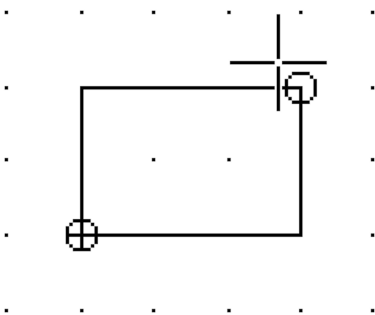
Figure 3-3: Choosing the CAD tools for drawing rectangles and activating the *Snap to grid* tool.

1. Launch QCAD if it is not already running. QCAD shows its application window and creates a new, empty drawing.
2. Before you start drawing anything, save this empty drawing to a file on your disk. To do so, choose the menu *File > Save As...*
The dialog for saving a drawing is shown. The dialog automatically suggests a location for your file. This location is usually not a bad place to start with. You might want to use a sub-folder *drawings* in this location instead, but to keep things simple the following steps assume that you use this default location for saving your drawing.
3. Type the filename *example* into the input field with the label *File name*, then click the *Save* button to save the empty drawing. The dialog window closes and you are now ready to start drawing.
Although it is not necessary to first save the empty drawing, it is good practice to do so as it forces you to think about where you want to store the file before you start drawing.
4. Move your mouse cursor to the shape button as shown in Figure 3-3 at the left (1). Click the left mouse button to show the shape tools (2).
5. Click the button with a rectangle on it as shown in Figure 3-3 (2). QCAD now knows that you intend to draw a rectangle and shows the CAD toolbar with the snap tools.
6. Click the button with a grid on it as shown in Figure 3-3 (3).
7. Move the mouse cursor around in the drawing area. There are two things to notice:
 - The mouse cursor has changed its shape and is now shown as a pair of cross hairs.

- There is a small yellow circle that follows the mouse cursor around whenever you move it. This circle is not positioned exactly under the mouse cursor. It ‘snaps’ always to the grid point in the drawing area that is the closest to the mouse cursor.
This yellow circle indicates what position QCAD is currently working with. The exact position of the crosshair mouse cursor is irrelevant to QCAD as long as the yellow circle is in the correct place. In the previous step you have chosen to use the grid for positioning (*Snap to grid*). QCAD is now automatically restricting the options for choosing a position to the grid points.
8. Click somewhere into the drawing area. A little red circle with a cross appears at the closest grid point as shown here:



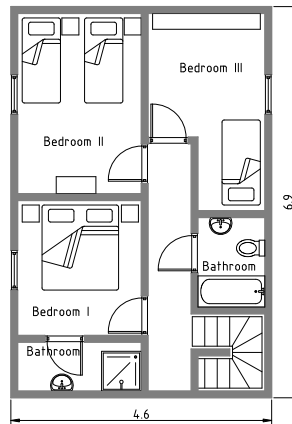
You have now set the first corner of the rectangle you are about to draw. If you move the mouse cursor around in the drawing area, you will see that QCAD draws a rectangle from the chosen position to the grid point that is closest to the mouse cursor as shown below:



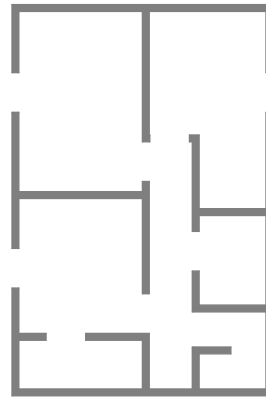
Note that this rectangle is not yet part of your drawing and keeps changing whenever you move the mouse. This is called a *preview*. QCAD uses these previews to show you what *would* be drawn if you would click the mouse button at this point.

9. Move the mouse cursor until the rectangle that is shown is three grid spacings wide and two grid spacings high. Your rectangle should look like that one in the figure above.

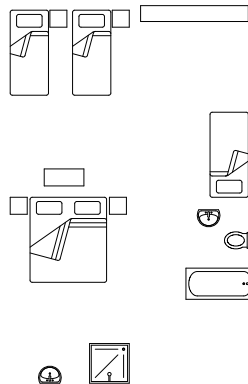
Complete Drawing:



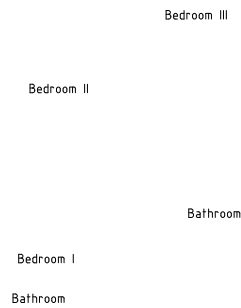
Layer "Walls":



Layer "Interiors":



Layer "Labels":

**Figure 5-1:** Organization of a drawing with layers.

Example Uses for Layers

Layers can be used in many different ways, depending on the type of drawing. The following list gives some common example uses for layers in different areas of application.

- **Mechanical drafting**
 - Visible / invisible edges
 - Center lines
 - Hatches
 - Dimensions
 - Text labels
 - Auxiliary constructions
 - ...
- **Architecture**
 - Exterior / interior walls

- Furniture
- Windows / doors
- Electrical, gas, water installations
- Dimensions
- Text labels
- Auxiliary constructions
- ...
- **Geographic Information System (GIS)**
- Roads, railway lines
- Houses
- Border lines
- Text labels representing ZIP codes
- ...
- **Schemas and Diagrams**
- Text labels
- Boxes
- Arrows, connectors
- Entity relationships
- ...

The Layer List

QCAD shows the layers of the current drawing in the layer list. To display the layer list, choose the menu *View > Layer List*. The layer list is displayed at the right as shown in Figure 5-2.



Figure 5-2: The layer list displays the layers of the current drawing.

After activation, the layer list shows only one layer with name *0*. The layer name is highlighted to indicate that this is the active layer, the layer onto which you are currently drawing. This initial layer is present in every drawing and can never be removed from a drawing or renamed.

The layer list also offers tools at the top to show, hide, add, remove and edit layers.

Layers and Linetypes

When creating a drawing, you will be using different linetypes to convey information about the specific role of an entity. For example there are lines that indicate visible edges, invisible edges or the center lines of a round part. To make clear to the reader of a drawing what the role of each line is, several linetype standards have been defined. It is advisable to apply these standards to your own drawings whenever possible.

Note that there are no universal conventions and these standards may be country-specific, industry-specific and even company-specific. Table 5-1 shows some established standards for linetypes in mechanical drawings.

Table 5-1		Established Linetypes	
Linetype	Example uses	Line thickness (mm)	
		A4/A3/A2	A1/A0
Continuous, thick 	Visible contours and edges.	0.5	0.7
Continuous, thin 	Dimensions, leaders, hatches and short center lines.	0.25	0.35
Dash, thin 	Hidden contours and edges.	0.25	0.35
Dash dot, thin 	Center lines and lines of symmetry.	0.25	0.35

Figure 5-3 shows an example drawing of a mechanical part with different linetypes.

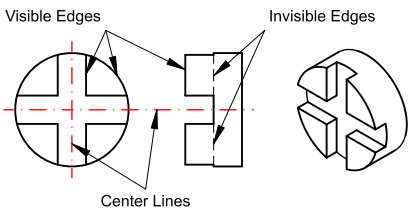


Figure 5-3: Different linetypes are used to indicate the meaning of a line.

Hands-on: Using Layers

These steps will guide you through the process of creating a simple drawing with layers. We start by adding some layers and then create the elements onto these layers.

Creating The Layers

Menu:

Layer > Add Layer

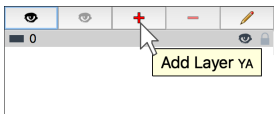
Keycode:

YA



Usage

1. In the layer list click the + button to add a new layer:



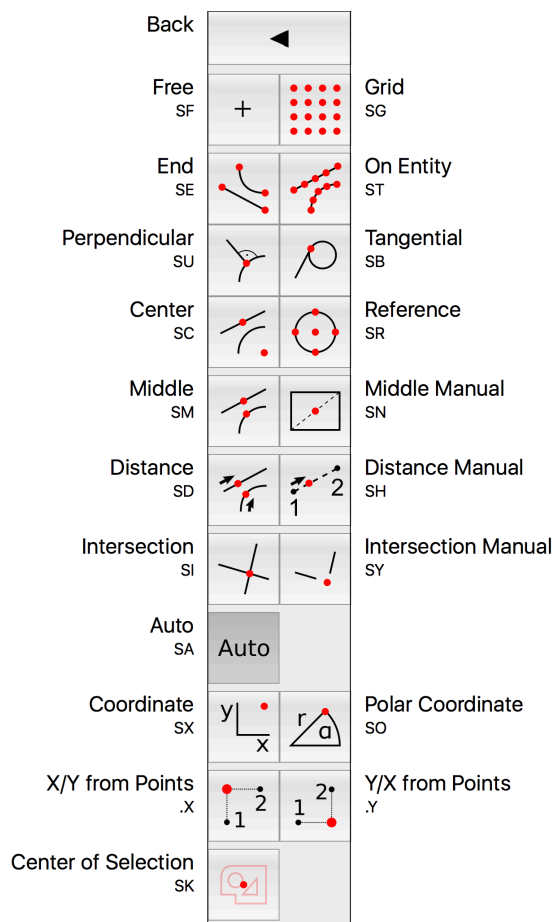


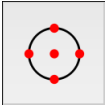
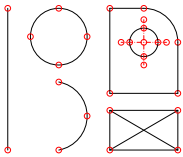
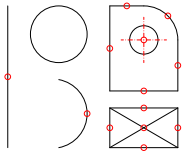
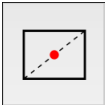
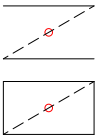
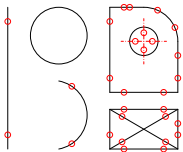
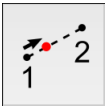
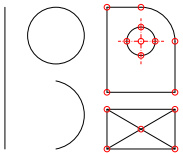
Figure 7-1: The toolbar with the snap tools is displayed automatically whenever you're expected to specify an exact position.

Snap tools can be used whenever you have to specify a point as part of a geometry or as part of an operation. For example when specifying the startpoint and endpoint of a line, when choosing a reference point for moving or copying entities, etc.

QCAD shows the complete palette of snap tools in the toolbar at the left whenever it expects you to specify a precise point. Figure 7-1 shows the snap tools. The first group of tools are snap tools that allow you to specify points based on existing geometry or the grid. Below are the coordinate snap tools that will be discussed in detail in a subsequent chapter.

The toolbar also shows advanced snap tools for snap restriction and the tools to move or lock the position of the relative zero point (not shown in Figure 7-1).

Table 7-1 shows an overview of the first group of snap tools. In most cases, the default snap tool *Auto Snap* will be the one you will be using but in some situations it is necessary or convenient to activate another, more specific snap tool.

Button	Snap tool	Key code	Effect and Examples
	Reference	SR	Snaps to reference points. Reference points are defining points of an entity. This is often used to snap to orthogonal points on circles as shown in the example figure. 
	Middle	SM	Snaps to middle points of arcs, ellipse arcs and lines. <i>Note that the middle point of an arc is on the arc and not at the center of the arc.</i> 
	Middle Manual	SN	Snaps to imaginary middle points. These are points in the middle between any two other points. This can for example be used to snap to the center of a rectangle, without having to create any auxiliary lines. This tool lets you pick two points to snap to the middle between them. 
	Distance	SD	Snaps to points with a given distance to the end point of an entity. 
	Distance Manual	SH	Snaps to points with a given distance along the imaginary line between two chosen points.
	Intersection	SI	Snaps to visible intersection points. 

Notes for Advanced Users

If you prefer to work with the command line of QCAD for entering coordinates, you can show the command line by choosing the menu *View > Command Line*. You can then enter coordinates in the command line instead of using the coordinate snap tool. The format how you have to enter coordinates is the same as described above, without any brackets: x,y for absolute Cartesian coordinates, @x,y for relative Cartesian coordinates, radius<angle for absolute polar coordinates and @radius<angle for relative polar coordinates.

If this format is not accepted, you might have configured an alternative format in the application preferences (see *Edit > Application Preferences > General > Coordinate Format*).

Hands-on: Drawing a Shape Using Relative Coordinates

The drawing shown in Figure 8-9 could be drawn with absolute coordinates, like the triangle in the previous example. To do so, you would of course have to calculate the absolute coordinates of each point first. It is easier in this case to draw the shape using mostly relative coordinates.

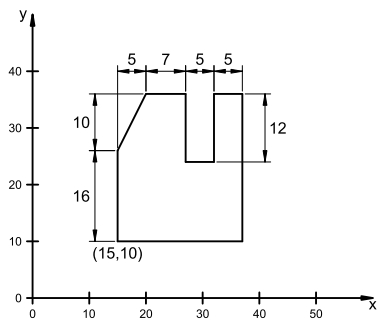


Figure 8-9: This shape can be easily constructed using mostly relative coordinates.

Before you start drawing a shape like this it is a good idea to write down the coordinates of every corner of the shape. Remember to use an AT symbol (@) to indicate which coordinates are relative. Figure 8-10 shows the shape again with all coordinates you will need to enter. The arrow shows in what direction the shape will be created. Unlike with absolute coordinates this is significant.

Tool options	Click point and constructed line
Angle: 60 Length: 20 Snap Point: End	

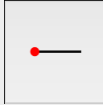
Horizontal Line / Vertical Line

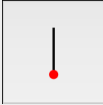
Menu:

Draw > Line > Horizontal Line / Vertical Line

Keycodes:

LH, LV





The tools for drawing horizontal and vertical lines work in the same way as the tool for lines with a given angle. The only difference is that the angle is restricted to 0 degrees for horizontal lines and 90 degrees for vertical lines.

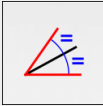
Bisector

Menu:

Draw > Line > Bisector

Keycode:

LB



A bisector line is a line that divides the angle between two lines into two equal halves (Figure 9-6).

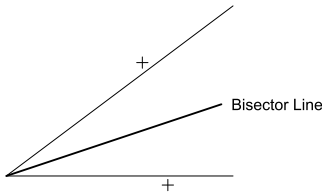
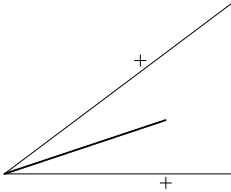
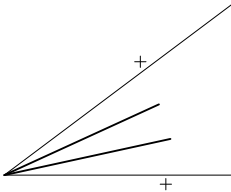
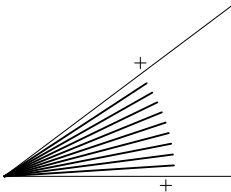


Figure 9-6: A bisector line divides the angle between two lines into two equal halves.

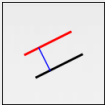
Usage

1. Enter the length of the line in the options toolbar.
2. Make sure that *Number* is set to 1.
3. Click the first line. This is one of the two lines that enclose the angle that will be divided into two equal parts by the bisector line.
4. Click the second line.
5. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

The bisector tool can also be used to divide the angle between two lines into more than two equal parts. This behavior is controlled by the *Number* option in the toolbar. Table 9-2 shows some examples with different tool options.

Table 9-2 Bisector Tool Examples	
Tool options	Click points and constructed lines
Length: 30 Number: 1	
Length: 30 Number: 2	
Length: 30 Number: 9	

Parallel

Menu:	Draw > Line > Parallel (with Distance)	
Keycode:	LP	

The tool for drawing parallels is one of the most important drawing tools. The main reason for this is that most man-made objects have many parallel edges and surfaces. Another reason is that you will often know the distance between two lines which makes it convenient to use this tool rather than having to indicate the start point and end point of a line.

Usage

1. Enter the distance of the parallel to the original, existing line in the options toolbar.
2. Make sure that *Number* is set to 1.
3. Move the mouse cursor close to an existing line in your drawing. QCAD shows a preview of the parallel line before you click the mouse button. Note that there are two possible

parallels to any line: one on each side of the line. If you move the mouse cursor somewhat to one side of an existing line, a parallel is shown at that side as shown in Figure 9-7.

- 4. Click the left mouse button when the preview shows the parallel you want to draw.
- 5. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

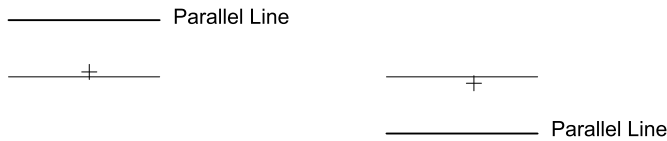


Figure 9-7: The parallel is created on the same side of the line on which the mouse cursor is positioned.

The parallel tool is sometimes also used to create multiple parallels to an existing line. Table 9-3 shows some examples with different tool options.

Table 9-3 Parallel Tool Examples	
Tool options	Click points and constructed lines
Distance: 5 Number: 1	A diagram showing a single line being drawn parallel to an existing line. The new line is below the existing line, and a small cross symbol indicates the cursor position.
Distance: 5 Number: 3	A diagram showing three lines being drawn parallel to an existing line. The new lines are below the existing line, and a small cross symbol indicates the cursor position.

Parallel through Point

Menu:

Draw > Line > Parallel (through Point)

Keycode:

LG

The icon shows a red line and a black line intersecting at a point. The red line is horizontal, and the black line is diagonal. A small red dot marks the intersection point.

This is an alternative tool for drawing parallels. Rather than creating a parallel with a distance this tool creates a parallel through a given point.

Usage

- 1. Make sure that Number is set to 1.
- 2. Choose the line for which you want to create a parallel by clicking it.

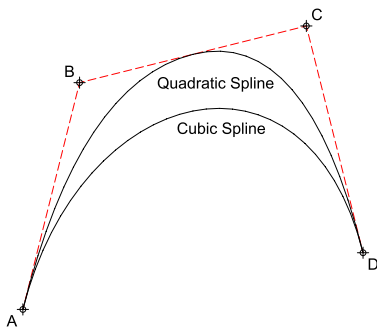


Figure 9-35: Quadratic and cubic splines.

Figure 9-36 shows the CAD toolbar with the spline tools.

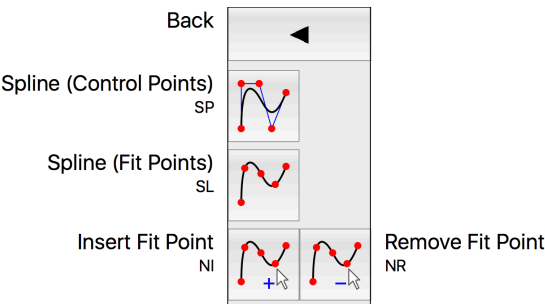
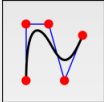


Figure 9-36: The CAD toolbar with the tools for drawing splines.

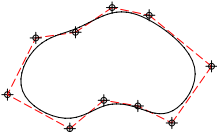
Spline from Control Points

Menu:	Draw > Spline > Spline (Control Points)	
Keycode:	SP	

Usage

1. Choose the spline degree in the options toolbar and tick the check box *Closed* if you want to create a closed spline.
2. Click the position of the first control point. For open splines this is also the start point of the spline.
3. Click any further control points. For quadratic splines you need to click at least three control points, for cubic splines at least four.
4. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

Table 9-8 shows some example uses of this spline tool. The control points are only shown for illustration purposes.

Table 9-8	Spline from Control Points
Tool options	Click points and constructed splines
Degree: 2 Open	
Degree: 3 Open	
Degree: 3 Closed	

Spline from Fit Points

Menu:

Draw > Spline > Spline (Fit Points)

Keycode:

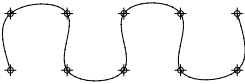
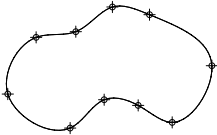
SL



Usage

1. Tick the check box *Closed* in the options toolbar if you want to create a closed spline.
2. Click the position of the first fit point. For open splines this is also the start point of the spline.
3. Click any further fit points.
4. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

Table 9-9 shows some example uses of this spline tool. The fit points are also shown for illustration purposes. Note that splines from fit points are always cubic (3rd degree).

Table 9-9	Spline from Fit Points
Tool options	Click points and constructed splines
Open	
Closed	

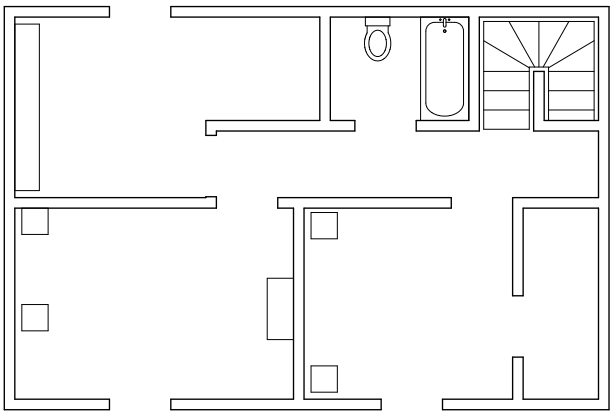
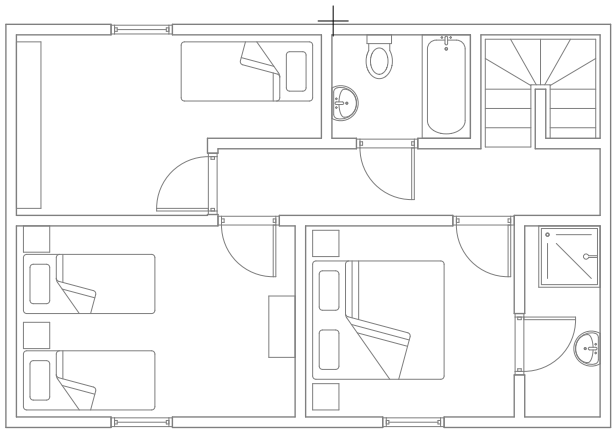


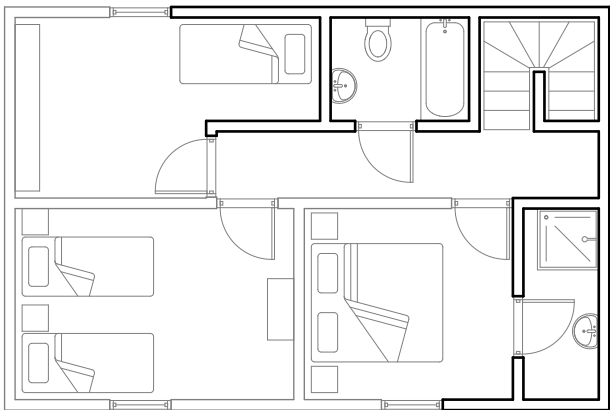
Figure 10-2: A floorplan with wall outlines and interiors. The tool to select contours can be used to easily select only the wall outlines.

Usage

- 1. Choose the desired selection mode (see Table 10-1).
- 2. Move the mouse cursor to an entity that is part of the contour you want to select. In this example, that is the wall outline at the top / right:



- 3. Click the left mouse button. All entities that are directly or indirectly connected to the entity you have picked are selected:



Selecting Intersected Entities

Menu: Select > (De-)Select Intersected Entities

Keycodes: UX, TX



This tool selects entities that are arranged in a straight line but cannot be selected using the window selection tool.

In the example drawing in Figure 10-3, all circles from the lower left diagonally to the upper right are to be selected. The window selection tool is not useful in this case since a rectangular area around those circles would also select all the other circles.

This tool lets you draw a line instead to select all entities that are intersected by this line.

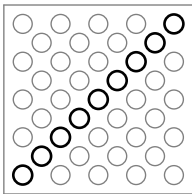


Figure 10-3: Entities that are in a row, but neither vertically nor horizontally can be easily selected with this tool.

Usage

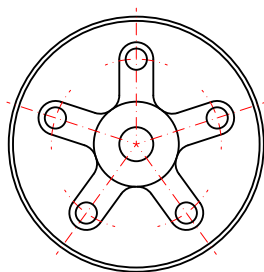
- 1. Choose the desired selection mode (see Table 10-1).

The choices for the number of copies, attribute usage and layer usage have the same meaning as for the move / copy tool.

Note that for a counter-clockwise rotation you have to enter a positive rotation angle and for a clockwise rotation a negative angle.

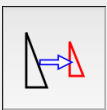
Since we want to have five evenly distributed copies of the selection around the center, we choose the option *Multiple copies* and enter 4 for the number of copies. For the angle we enter $360/5$. Of course we could also enter 72, but letting QCAD compute the value is usually more convenient and prevents mistakes.

5. Click *OK* to confirm the options.
6. QCAD creates the rotated copies of the selection:



Scaling Entities

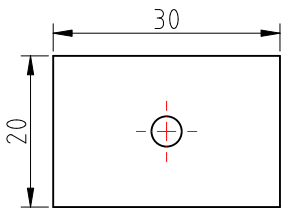
Menu: Modify > Scale
 Keycode: SZ



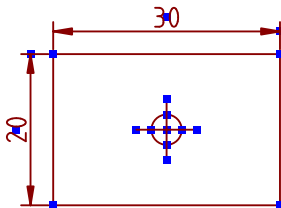
Scaling means to change the size of the selected entities, usually while maintaining proportions. The selection can be scaled up (making it larger) or down (making it smaller).

Usage

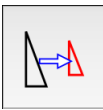
1. Select the entities you want to scale.
 The example for this tool contains dimension entities that indicate the part size:



We select the whole part for scaling in this example:

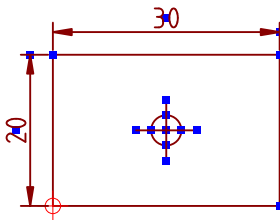


- Click the scale tool button:

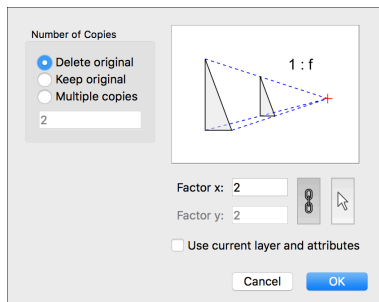


- Click the center for the scaling operation. The selection will be scaled towards or from this center.

In the example, we choose the lower left corner as the center:



- The dialog with the scaling options is shown:



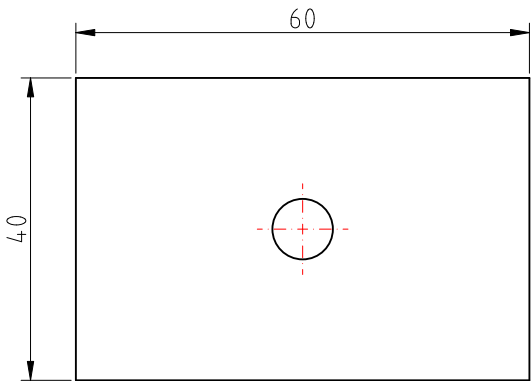
The choices for the number of copies, attribute usage and layer usage have the same meaning as for the move / copy tool or the rotation tool. Creating copies when scaling is very unusual though and *Delete original* is by far the most common choice.

At the right side of the scale factor, there is a button with a chain on it. It is usually checked to indicate that only one factor can be entered. In that case, the scale factor in Y direction is the same as the factor in X direction (proportional scaling). In most cases you will want to use proportional scaling. Non proportional scaling is only supported for line entities and polyline entities which only contain line segments.

We enter the number 2 for the factor to double the size of the part. If we would want to scale the part down to half its size, we would enter the number 0.5 or 1/2 instead.

- Click *OK* to confirm the options.

6. QCAD scales the part up and deletes the original:



Note that the dimension labels have been adjusted automatically to show the correct measurements.

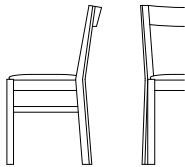
Mirroring Entities

Menu:	Modify > Mirror	
Keycode:	MI	

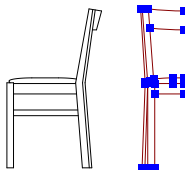
The mirroring tool is helpful to create symmetrical drawings. The symmetry is defined by an axis of symmetry. All objects on one side of the axis are mirrored (or *flipped*) to the other side of the axis.

Usage

1. Select the entities you want to mirror.
For this example, we assume that you have already created the side view and half of the front view of a chair. The other half of the front view can be created by mirroring the existing half along a vertical axis of symmetry.



In the first step, we select the existing half:



2. Click the mirror tool button:



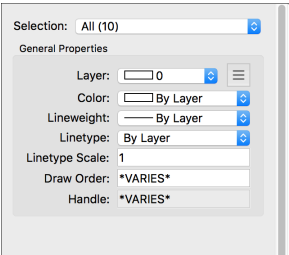
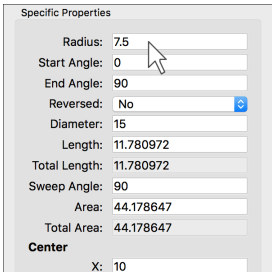


Figure 11-4: When multiple entities of different types are selected, only those properties are shown that are available for all selected entities.

Usage

- 1. Select the entity or the entities for which you want to browse or change properties.
- 2. Click into the value field of the property you want to change.
For example, to change the radius of a selected arc, click into the field where the radius property is shown at the right:



- 3. The moment you click into the field it becomes active and lets you change the property.
- 4. Change the property the way you want to.
For input boxes like the one shown for the arc radius, you have to press the Enter key or the Tab key or click somewhere outside the input box when you are done to confirm your changes. If you press the Tab key, you can directly edit the next property.
Whenever a numeric value is expected, you can either type a value or a mathematical expression (e.g. 15/2 instead of 7.5).

Filtering Entity Types

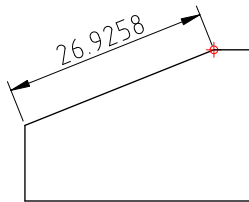
The combo box at the top of the property editor shows the total number of selected entities. It can also be used to show and edit only properties of a certain entity type. This can be convenient when entities of multiple types are selected but you are only interested in one entity type.

You might for example want to change the font of all texts in a drawing. Instead of going through the procedure of selecting each individual text entity you can simply select all entities in your drawing and then activate the filter for text entities. Once the filter is set, the properties of the text entities are shown and the font can be changed.

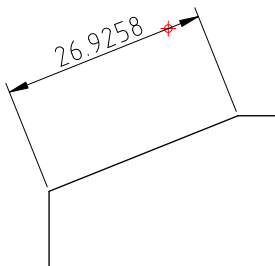
Usage:

- 1. Select some entities of mixed types.

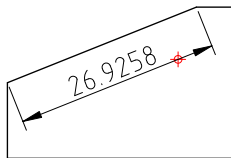
2. Click the position of the second point.
Note that while you are moving the mouse, QCAD already previews the aligned dimension and its measurement.
For the example, we click the other end point of the tilted line:



3. Click the position of the dimension line. This step allows you to place the dimension line closer or further away from the object.



Alternatively, you could also place the dimension line below the tilted line by clicking in that area:



4. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

Rotated (Linear) Dimension

Menu: Dimension > Rotated (Linear)
Keycode: DL



This tool creates dimensions with a fixed angle for the dimension line. The most common angles for linear dimensions are 0 degrees (horizontal) and 90 degrees (vertical). Because they are so common, QCAD offers two separate tools for those dimension types.

Sometimes it is necessary to create a dimension at a different fixed angle. This is often the case if a whole part of a drawing is rotated by a fixed angle and the dimensions are also given at that angle. Figure 14-7 shows an example for such a drawing. The center axis of the right part of the drawing has an angle of 60 degrees. The dimension lines of the two dimensions also need to have an angle of 60 degrees in order to automatically show the correct measurement.

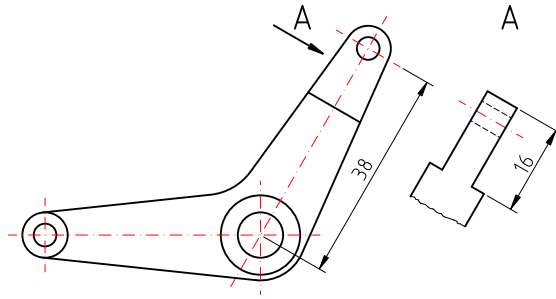


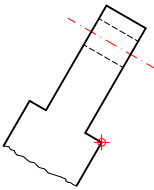
Figure 14-7: The two linear dimension have a fixed angle of 60 degrees, parallel to the distance they measure.

Usage

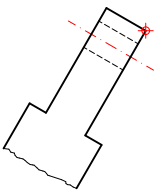
1. Enter the angle of the dimension line in the right part of the options toolbar. For this example, we enter 60 to produce a linear dimension with the dimension line at an angle of 60 degrees:



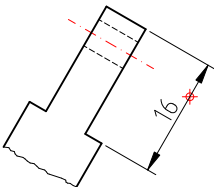
2. Click the position of the first of the two points between which the distance should be dimensioned.



3. Click the position of the second point.



4. Click the position of the dimension line. Note that you can still adjust the angle of the dimension line in the options toolbar at this point if you can see from the preview that you have entered a wrong angle.



5. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

Table 16-2	Updated Block Definition
Workstation	
	

As soon as the block is modified, all block references in the drawing are automatically updated by QCAD to reflect the changes. Figure 16-2 shows the updated drawing.

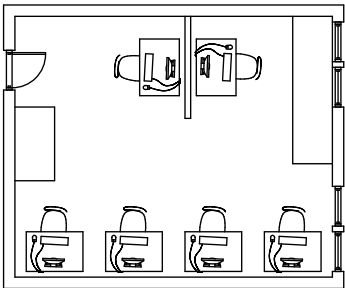


Figure 16-2: Example drawing after changing the monitors in block *Workstation* from monitors to flat screens.

The Block List

Before starting to work with blocks, please make sure that the QCAD application window displays the block list. The block list is a user interface component that can be permanently shown or hidden like the layer list. The block list shows the names of all blocks in your drawing. It also allows you to manage blocks and to insert blocks into your drawing.

To show the block list, choose the menu *View > Block List* or click the appropriate button at the right.

Menu:

View > Block List

Keycode:

GB



The block list is displayed at the right in the QCAD application window as shown in Figure 16-3.

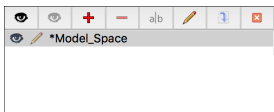


Figure 16-3: Make sure that the block list is visible when you are working with blocks.

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