

QCAD

An Introduction to Computer- Aided Design (CAD)

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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.

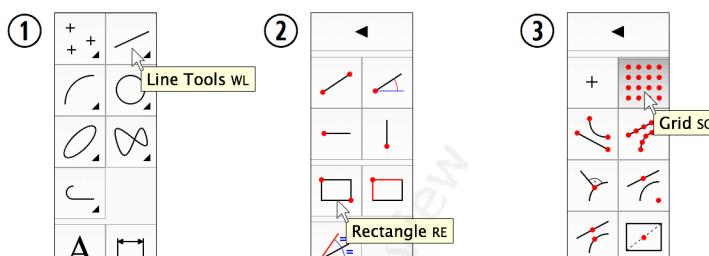


Figure 2-3: Choosing the CAD tools for drawing rectangles and activating the *Snap to grid* tool.

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 line button as shown in Figure 2-3 at the left (1). Click the left mouse button to show the line tools (2).

5. Click the button with a rectangle on it as shown in Figure 2-3 (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 2-3 (3).

7. Move the mouse cursor around in the drawing area. There are two things to notice:
 - o The mouse cursor has changed its shape and is now shown as a pair of cross hairs.
 - o 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.

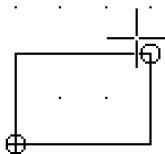
This yellow circle indicates what position QCAD is currently working with. The exact position of the crosshair mouse cursor is irrelevant to QCAD as long as the

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:

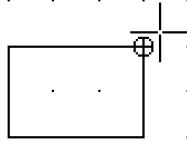


You have now set the first corner of the rectangle you are about to draw. If you move the mouse cursor around in the drawing area, you will see that QCAD draws a rectangle from the chosen position to the grid point that is closest to the mouse cursor as shown below:



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.
10. Click the left mouse button to set the second corner of the rectangle. This leaves you with a drawing that looks like this:



The rectangle that is shown now, is a part of your drawing.

11. QCAD is ready to draw the next rectangle and waits for the first corner of the next rectangle. Since we don't want to draw more rectangles, we will terminate this tool now. To do so, click the right mouse button twice. If you don't have a right mouse button, press the *Escape* or *Esc* key on your keyboard twice. The mouse cursor is back to normal and the CAD toolbar shows the same tools as it did after starting QCAD. Your rectangle should still be visible. If that is not the case, you did something wrong and you need to carefully repeat the steps 4 to 10.
12. Save your drawing by choosing the menu *File - Save*.

In the example you have just completed, you have used a tool called *Snap to grid*. As a result, the corners of the rectangle are exactly aligned to the grid points. Snap tools are a central concept of any CAD system and there are many other snap tools you will get to know later in this book.

Hands-on: A Line through the Middle

To emphasize the importance of snap tools, we will now extend our drawing with a vertical line that separates the rectangle in two equal halves.

Vertical means that the line extends from a first point to another point directly under or above it. In our case, the line starts in the middle of the top line of the rectangle and ends in the middle of the bottom line. The top and bottom lines of the rectangle are *horizontal*, that means they extend from left to right. You can easily remember what *horizontal* means by thinking that the *horizon at the seaside looks horizontal*.

Note that there are no grid dots at the center of the top and the bottom line of the rectangle. For this line we will have to use a different snap tool.

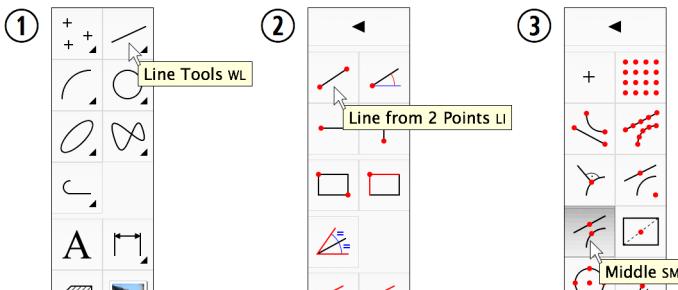
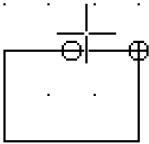


Figure 2-4: Choosing the CAD tool for drawing lines with two points and changing the snap tool to *Snap to middle points*.

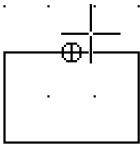
1. Choose the *Line Tools* button again from the CAD toolbar as shown in Figure 2-4 (1).
2. This time, select the tool *Line from 2 Points* (2).
3. Click the button *Middle* (3). This activates the snap tool to snap to middle points of lines and arcs. Note that only one snap tool can be active at any time.
4. Move the mouse cursor around in the drawing area like we did before with the grid snap tool. As you can see, the yellow circle no longer jumps from grid point to grid point. Instead it now only shows up in four different positions which are the middle points of the four lines that form the rectangle. One such possibility is shown here:



Try also to find the other three by moving the mouse cursor around.

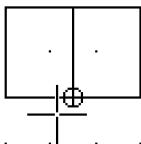
5. Click the left mouse button while the mouse cursor is located somewhere close to the middle of the top line of the rectangle. It doesn't really matter where exactly the mouse cursor is, as long as the yellow circle is located in the middle of the top line as shown above.

After clicking the left mouse button, the drawing should look like shown below. The start point of the line is now set:



6. Move the mouse cursor approximately to the middle of the bottom line of the rectangle. You can see a preview of the vertical line we are about to draw.

- Click the left mouse button to set the end point of the line. The drawing should now look like this:

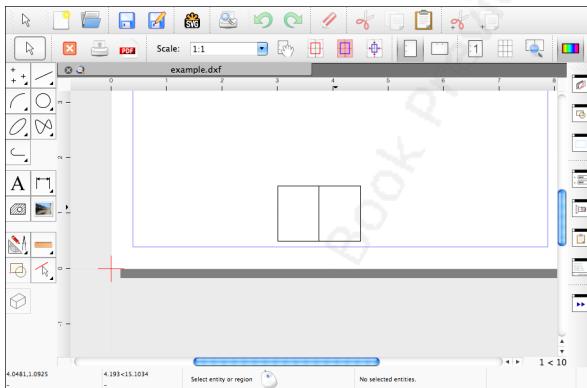


- QCAD now waits for you to draw the next line or to terminate the tool. Click the right mouse button twice or press the Escape key on your keyboard twice to make sure the tool is terminated.
- Save your drawing again by choosing the menu *File - Save*.

Hands-on: Printing a Drawing

Once your drawing is finished, you will most likely want to print it on paper. In the following steps we will print your drawing on an *A4* or *Letter* size paper.

- Activate the print preview by choosing the menu *File - Print Preview*. QCAD shows your drawing as it will be printed. Hit the minus key on your keyboard a couple of times to zoom out until you can see the paper border:



The white area shows the size and location of the paper.

The toolbar at the top shows some tools and options for the print preview.

- Print the drawing by choosing the menu *File - Print*. The printer dialog is shown. If your printer is set up correctly, it should not be necessary to make any adjustments here. Click *OK* to print your drawing.
- Close the print preview by clicking the close button at the top left in the options toolbar:



- Save your drawing by choosing the menu *File - Save* (QCAD will save the paper settings with your drawing).
- You can close your drawing now. To do this, choose the menu *File - Close*.

Closing QCAD

If you want to continue right away with the next chapter you can keep QCAD running. If you want to finish for now, you might want to close the QCAD application. You can do this by choosing the menu *File - Quit*.

4. QCAD adds the new layer to the layer list. Later we will use this layer for all visible edges of the drawing.
5. In the same way, add the following layers with these names and attributes to the drawing:
 - This layer will later be used for hidden edges:
Layer name: *hidden*
 1. Color: *Black*
 2. Width: *0.25mm*
 3. Line type: *Dash*
 - All center lines and symmetry lines will be placed on this layer:
Layer name: *center*
 1. Color: *Red*
 2. Width: *0.13mm*
 3. Line type: *Dash Dot*

Drawing onto Layers

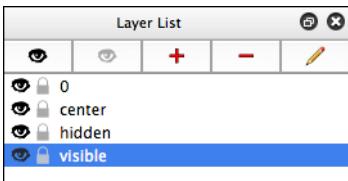
The layer list now shows the layers you have just added in addition to the layer *0*. Before you are drawing something, you have to decide on which layer you want to draw. In the following steps you will draw some elements onto each layer.

The drawing we will produce is shown in Figure 4-4. It is the front view of a simple mechanical part. The instructions below will guide you through the complete process of creating the drawing in Figure 4-4. You will use the rectangle tool again, get to know the tool for drawing parallel lines and use two simple modification tools to finish the drawing.



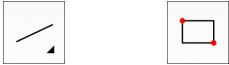
Figure 4-4: The final drawing of this exercise.

1. Click on the layer name *visible* in the layer list. Make sure that you click on the name and not one of the icons beside it. The layer name is now highlighted:



This indicates that layer *visible* is now the active layer. The active layer is the layer onto which you are currently drawing.

2. Choose the drawing tool for drawing a rectangle:

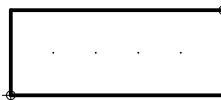


3. Activate the grid snap:

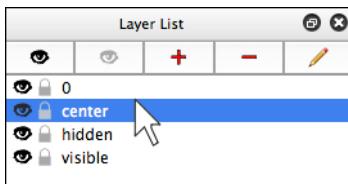


4. Adjust the scale of the drawing area so that you can see a space of about 100 units horizontally. To do this, keep an eye on the rulers that QCAD displays at the top and at the left. Check the bottom right corner of the drawing area to make sure that the grid is currently shown with a spacing of 10 units (the text 10 / 100 is shown).

5. Draw a rectangle that is 50 units wide and 20 units high using the grid. Set the first corner at any grid point in the drawing and set the second corner 5 grid points to the right and 2 grid points to the top of the first corner:



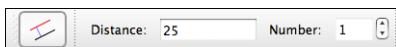
6. We will now create the vertical center line in the middle of the rectangle. We can use the parallel tool for that, but first we need to switch the active layer to *center*. Click on the layer *center* in the layer list to activate it:



7. Choose the drawing tool for drawing parallel lines:



8. The options toolbar for parallels is shown at the top. Enter 25 for Distance and make sure that Number is set to 1:



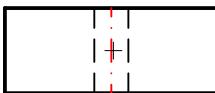
9. Move the mouse pointer close to the left side of the rectangle, just slightly to the right of it. QCAD gives you an immediate preview of where the parallel will be placed. This should look like this:



As soon as the parallel is shown at the right place, click the left mouse button to create it.

Note that the center line you have just created has the attributes of layer *center*. It should be displayed in red and with a dash-dot line pattern.

10. Activate the layer with name *hidden* and create the two hidden lines. The distance from the center line to each of the hidden lines is 4, so you need to change the distance in the options toolbar of the parallel tool to 4 and then create the lines as shown here:



11. The drawing is now almost finished. Center lines and symmetry lines are usually slightly extended to clearly separate them from the edges of the object. QCAD offers a modification tool to extend a line by a given amount. Start this tool by clicking the

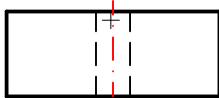
button to show the modification tools, followed by the button for the tool to lengthen entities:



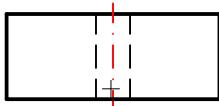
12. Enter the amount 2.5 in the options toolbar:



QCAD will now lengthen every entity you click by an amount of 2.5 units. The element is extended at the end which is closer to the mouse cursor when you click the entity. To extend the center line by 2.5 units at the top end, click the center line close to its top:

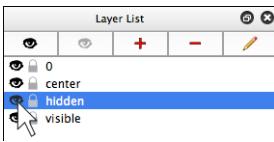


13. Click the center line close to its bottom end to extend it in that direction as well:

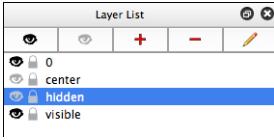


Changing the Visibility of Layers

1. You can now easily view your drawing without invisible lines and without the center line by hiding the layers *hidden* and *center*. To change the visibility of layer *hidden*, click on the eye symbol next to its name in the layer list:



2. It is not necessary to activate a layer in order to change its visibility, so you can simply click the eye icon of layer *center* to hide it as well.
The layer list indicates hidden layers with a gray eye icon:



3. After hiding the layers *hidden* and *center*, your drawing now only shows the layers *0* (which is empty) and layer *visible*:



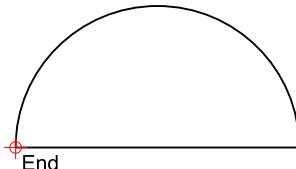
4. Make sure that all layers are visible again by clicking the button to show all layers:



10. Make sure that the direction of the arc is set to counter-clockwise in the options toolbar:



11. Move the mouse cursor to the other end of the line at the left and click that endpoint to define the end angle of the arc:

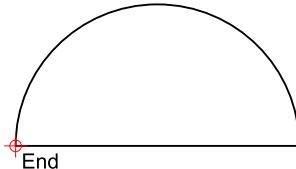


The arc is now completely defined.

12. Terminate the arc tool by clicking the right mouse button twice or by clicking the selection pointer button in the toolbar:



13. Start the line tool as previously described and set the start point at the left endpoint of the horizontal line:



14. For this example, we want to place the end point at a distance of exactly 20 units along the arc line.

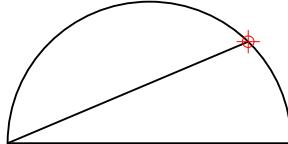
There are some snap points that cannot be set with the auto snap tool. At the moment, we would like to snap to a point on the arc line with a distance of 20 units from the end point. For this, we use the snap tool called *Distance from end point*. This tool snaps to a point with a given distance from an end point. Click the *Distance from end point* snap tool in the CAD toolbar at the left:



The options toolbar at the top now shows a text field. Enter the distance from the end point to which you want to snap. For this example, enter 20:



15. Move the mouse cursor along the arc again. The mouse cursor now only snaps to two positions. One of them is at the right, 20 units away from the right endpoint of the arc:



The other one is at the left, 20 units away from the left endpoint of the arc:

Chapter 7

Coordinates

Objective

In this chapter, you will

- learn what coordinates are,
- get to know the different types of coordinates QCAD supports,
- learn how to define positions by entering coordinates.

The Cartesian Coordinate System

In the previous chapters you have already seen and used the drawing area of QCAD. Like a sheet of paper, the drawing area is a flat area onto which you can draw something.

When working with a CAD system, you will often be confronted with the coordinate system of the drawing area. A coordinate system uniquely defines each point in the drawing area and in your drawing. If you point with a pen to any position in the drawing area, that position has a unique coordinate that defines where this point is in the drawing.

By far the most commonly used coordinate system is the *Cartesian coordinate system*. A coordinate system is not something that is given by nature. Coordinate systems were defined once by someone (in this case René Descartes in 1637) to define a standard for specifying the position of a point on a two dimensional surface. The Cartesian coordinate system is not only used in CAD applications but in many areas of mathematics, physics and engineering.

The Cartesian coordinate system is based on two axes that are at right angles (orthogonal) to each other. The horizontal axis is commonly called the *X-axis* while the vertical one is called the *Y-axis* as shown in Figure 7-1.

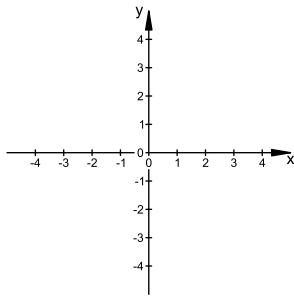


Figure 7-1: The coordinate axes of the Cartesian coordinate system.

The *origin* of the coordinate system is the point where the X and the Y axes cross each other. This point is also referred to as the *absolute zero point* or just *absolute zero*.

Both axes have a direction. The X-axis is directed to the right and the Y-axis upwards. This is not necessarily a logical choice, it was simply defined this way. As you can see in Figure 7-1, the axes are divided into smaller sections, each one unit long.

Any particular position can be described by its distance from the origin in X-direction and in Y-direction. For example the position of the point P in Figure 7-2 is 3 units away from the origin in X-direction and 2 units away from the origin in Y-direction. Or, to use the correct notation, the point P is located at $(3,2)$. This notation in brackets indicates the location of a point as a pair of an X-distance and a Y-distance (X,Y) .

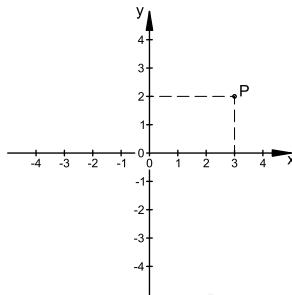


Figure 7-2: The location of the point P can be noted as $(3,2)$ where 3 is the distance to the origin in X-direction and 2 is the distance to the origin in Y-direction.

If a point is located left of the origin, its X-coordinate turns negative. If it is located below the origin, its Y-coordinate turns negative. Figure 7-3 shows some points in the Cartesian coordinate system and their (X,Y) notation. The (X,Y) notation for the origin is $(0,0)$.

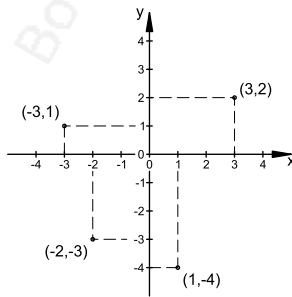


Figure 7-3: Some example coordinates in the Cartesian coordinate system.

There are different ways to specify the location of a point in the Cartesian coordinate system. The most common ones are: absolute, relative and polar coordinates.

Absolute Cartesian Coordinates

Absolute Cartesian coordinates indicate the position of a point by its distance to the origin along the X and Y axes. The coordinates used in previous examples are all absolute Cartesian coordinates.

Absolute Cartesian coordinates are usually noted as (X,Y) , for example $(6,4)$. Figure 7-4 shows an example for an absolute Cartesian coordinate.

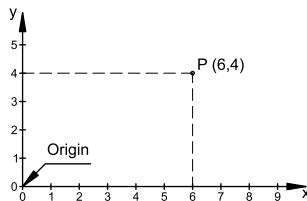


Figure 7-4: The absolute Cartesian coordinate of point P is (6,4).

Relative Cartesian Coordinates

Relative Cartesian coordinates relate to the last used position and not to the origin of the drawing.

There is no standard notation for relative coordinates. However, a common notation in the CAD industry is to prepend an AT sign (@) to a relative coordinate.

A relative Cartesian coordinate may for example be used to position the end point of a line relatively to its start point. Let's assume you have just set the start point of a line at the absolute coordinate (2,2). You can now set the end point of the line at the relative coordinate (@@5,3). The absolute coordinate of the end point will be at (7,5). In this example, the relative coordinate relates to the start point of the line (2,2). In other words, the values of the relative coordinate of the end point are added to the absolute coordinate of the start point: $(2,2) + (@@5,3) = (2+5,2+3) = (7,5)$.

In previous exercises you might have already noticed a small red circle that moves always to the previously clicked point when you draw lines. This red circle visualizes the position of the relative zero point of your drawing. When entering relative coordinates, they relate to the current position of this red circle. Figure 7-5 shows an example for a relative coordinate.

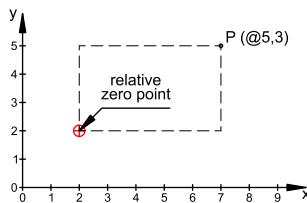


Figure 7-5: The relative Cartesian coordinate of point P is (@@5,3).

Absolute Polar Coordinates

Polar coordinates specify the position of a point by an angle and the distance to the origin (often called radius).

A common notation for absolute polar coordinates is (distance<angle), for example (8<30) for a point with a distance of 8 units from the origin at an angle of 30 degrees. Angles are always measured from the positive part of the X-axis. In other words, 0 degrees is east or 3 o'clock on your watch. Angles are measured counter-clockwise which is the mathematical and technical standard for indicating angles. Negative angles may be used for clockwise angles. Figure 7-6 shows an example for an absolute polar coordinate.

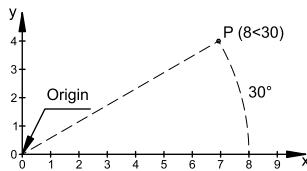


Figure 7-6: The absolute polar coordinate of point P is $(8<30)$.

Relative Polar Coordinates

Just like Cartesian coordinates, polar coordinates can also refer to the relative zero point instead of the origin. In this case we talk about relative polar coordinates.

We use the notation (@distance<angle) for relative polar coordinates. Figure 7-7 shows an example for a relative polar coordinate.

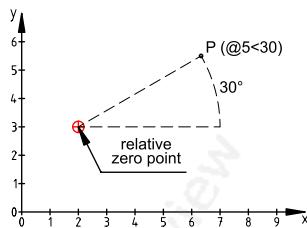


Figure 7-7: The relative polar coordinate of point P is $(@5<30)$.

Notes

Note that QCAD stores all coordinates as absolute Cartesian coordinates internally. So there is no difference in the end result if you use absolute or relative and Cartesian or polar coordinates. The different ways for specifying a position are only a help for you as a user to avoid having to calculate positions.

Hands-on: Drawing a Triangle from Three Absolute Coordinates

The following steps guide you through the process of drawing a triangle from the three coordinates of its corners. The goal is to draw the triangle shown in Figure 7-8.

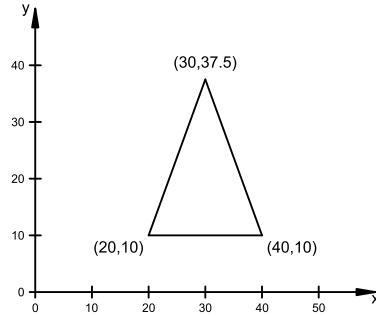


Figure 7-8: A triangle with known corner coordinates.

1. Launch QCAD with a new, empty drawing and save it under the name *coordinates01.dxf*.

2. Click the position of the corner diagonally opposite of the first corner.
3. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

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 8-7).

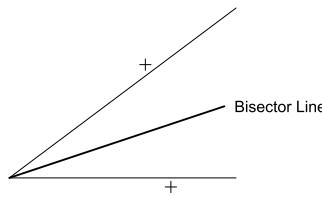


Figure 8-7: A bisector line divides the angle between two lines into two equal halves.

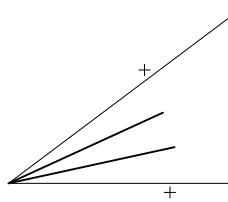
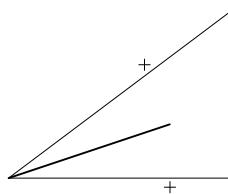
Usage

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 8-2 shows some examples with different tool options.

Table 8-2 **Bisector Tool Examples**

Tool options	Click points and constructed lines
Length: 30 Number: 1	
Length: 30 Number: 2	



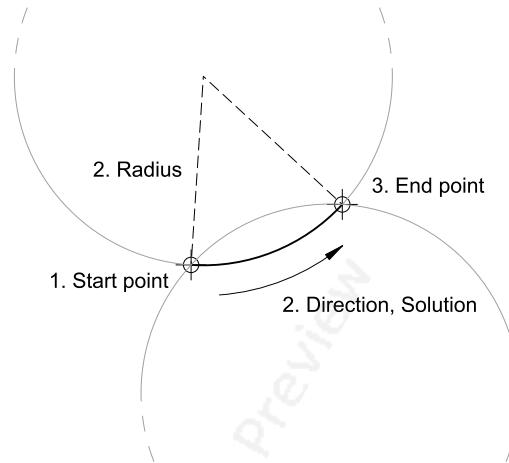
Arc from Two Points and Radius

Menu: Draw - Arc - 2 Points and Radius

Keycode: AD



This is the first of two tools which can be used to draw an arc from its start point and end point. This tool also requires a radius to be input. Since there are four arcs possible, you also have to define the direction and choose between the shorter or longer of the possible arcs. Figure 8-16 shows the two points with the four possible solutions in gray and the chosen solution in black.

**Figure 8-16:** Arc from two points and a radius.**Usage**

1. Click the start point of the arc.
2. Enter the arc radius and choose the arc direction (counterclockwise or clockwise) and which solution you want (shorter or larger arc):



For the example in Figure 8-16 the direction was chosen as counterclockwise and the solution.

3. Click the end point of the arc.
If the end point is two times the radius away from the start point or more, a half circle is drawn with the given radius.
4. Terminate the tool by clicking the right mouse button or by hitting the Escape key on your keyboard.

Arc from Two Points and Angle

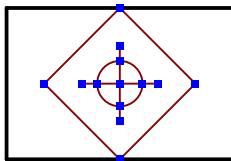
Menu: Draw - Arc - 2 Points and Angle

Keycode: A2



This is the second tool to draw an arc from its start point and end point. With this tool you can specify the angle that is covered by the arc and the arc direction. Figure 8-17 shows the two points with the four possible solutions in gray and the chosen solution in black.

3. Let go of the left mouse button. QCAD selects the entities completely inside the chosen area as well as those that are only partly inside the area:



The Shift key has the same effect for this selection tool as when picking single entities. If you press the Shift key while doing a rectangular selection, the selection is added to the current selection. Otherwise, the previous selection is replaced with the new one.

Advanced Selection Tools

In addition to the simple selection tools described so far, QCAD offers some more selection tools for advanced selection needs. Those more advanced selection tools can only be accessed through menu *Select* or through the selection tools button in the CAD toolbar at the left:

Menu: Select
Keycode: WS



Figure 9-1 shows the CAD toolbar with the complete palette of selection tools.

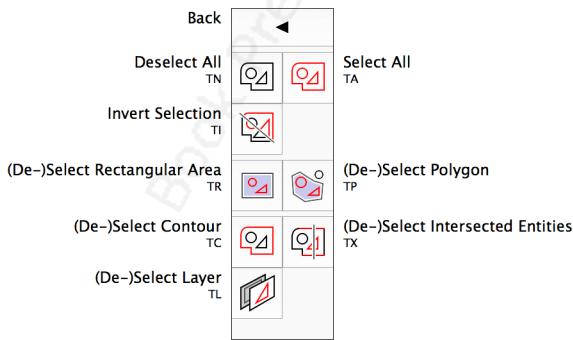


Figure 9-1: The CAD toolbar showing the advanced selection tools.

Note that some of these tools are also available in the neutral mode of QCAD as basic selection tools as previously described.

Selection Modes

Some of the selection tools offer different selection modes. The selection mode that is chosen defines how the newly chosen selection affects the current selection. The default selection mode replaces the current selection with the new selection.

Table 9-1 **Selection modes for the rectangle selection tool and other selection tools.**

Selection Mode	Description
Replace	Replaces the current selection with the new selection. If used for example with the rectangle selection tool, all entities inside the rectangle are selected and everything else is deselected.

Selection Mode	Description
Add 	Adds all matching entities to the current selection. For the rectangle selection tool, all entities inside the rectangle are selected in addition to the entities that were already selected before using the tool.
Subtract 	Subtracts all matching entities from the current selection. This turns for example the rectangle selection tool into a rectangle deselection tool. All entities inside the rectangular area are deselected. Entities outside the rectangle that were previously selected remain selected.
Intersect 	Intersects the current selection with the new selection. After using the rectangle selection tool with this selection mode, only entities that were already selected and that are inside the rectangle are selected.

Deselecting Everything

Menu: Select - Deselect All



Keycode: TN

Click this button to clear any selections. Alternatively, you may want to simply click into an empty area of your drawing to deselect everything.

Selecting Everything

Menu: Select - Select All



Keycode: TA

Click this button to select all visible, editable entities of your drawing. Entities on layers that are hidden or locked are not selected by this tool. This prevents you from accidentally selecting and later modifying or deleting something you are not aware of because it is hidden.

Note that those entities which are on a visible, unlocked layer but are not within the currently visible area of the screen are also selected with this tool.

Inverting Selection

Menu: Select - Invert Selection



Keycode: TI

With this tool you can quickly invert the current selection. The tool selects all not selected entities and deselects all selected entities. This is especially useful if you need to modify almost your entire drawing, except a few entities. In this case you might want to select the entities which should remain the same and then invert the selection with this tool.

Selecting Rectangular Areas

Menu: Select - (De-)Select Rectangular Area



Keycode: TR

This is an alternative tool for selecting and deselecting entities inside a rectangular area. This tool is similar like the basic window selection tool described in the previous section. The advantage of this tool is that you may choose a selection mode in the options tool bar. Table 9-1 shows what the different selection modes do.

Cut and Copy with Reference Point

Menu:	Edit - Cut with Reference	
Keycodes:	RT, Ctrl-Shift-X (Mac: ⌘⇧X)	
Menu:	Edit - Copy with Reference	
Keycodes:	RC, Ctrl-Shift-C (Mac: ⌘⇧C)	

Figure 9-5 shows the side view of the previous example drawing with the chair and the table. Again, we want to move the chair closer to the table using cut and paste. This time it is important that the pasted chair is aligned at the bottom with the table. Using the center of the chair as reference point might not be convenient in this situation.

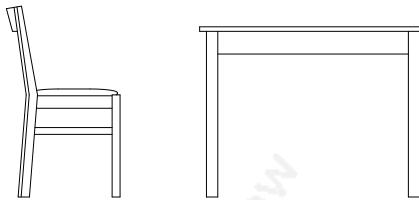
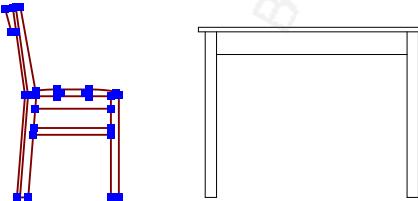


Figure 9-5: To place the chair at an exact position, the tool *Cut with Reference* can be used.

Usage

1. Select the entities you want to cut or copy.
In the example, that is the side view of the chair:



2. Choose the tool "Copy with reference":



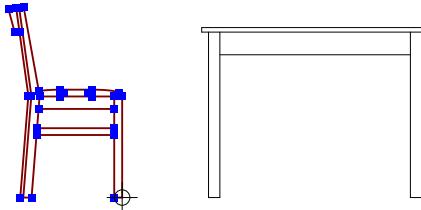
or the tool *Cut with Reference*:



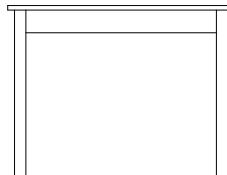
Since we want to remove the original chair in this example, we choose the tool *Cut with Reference*.

3. QCAD now asks you to specify the reference point. Click a point in the drawing to use as reference point. This can be any point but is typically a significant point within the selection or a grid point.

For our example, we choose the corner point at the lowest right edge of the chair:



4. As soon as you have chosen the reference point, the original entities are removed from the drawing and stored on the clipboard:



5. You can now use the paste tool to paste the clipboard contents. Note that there is only one paste tool which always requires you to position the pasted entities:

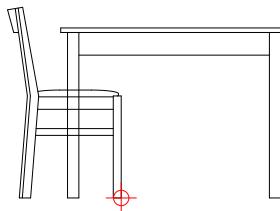


6. Make sure that the options in the options tool bar are reset to a rotation angle of 0, a scale of 1 and that the mirror buttons are not activated. To quickly reset all values, click the reset button at the right of the options toolbar:



7. Click the left mouse button to place the entities. The entities can now be positioned by the previously chosen reference point.

For this example, this means that you can position the chair to be exactly aligned with the bottom of the table by clicking one of the grid points on the same height as the ground level:

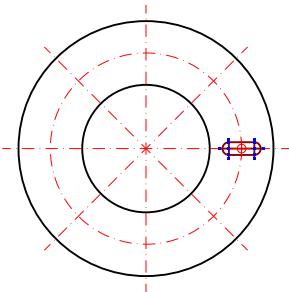


8. We can continue to place another copy of the chair at the other side of the table. Click the vertical flip button in the options toolbar to mirror the chair horizontally:

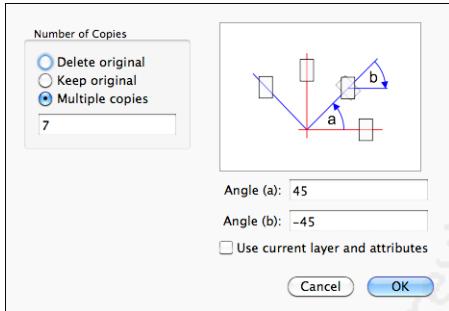


4. Click the center for the secondary rotation. This is usually at the center of the small part that is rotated around the center point for the primary rotation.

In this example, we choose the center of the elongated hole:



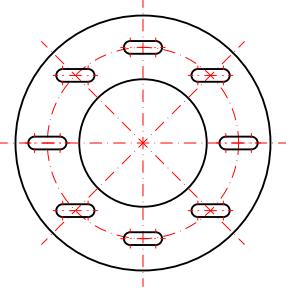
5. QCAD shows the dialog with the options for this tool:



For our example we want to create 7 copies. The main rotation angle a is 45 degrees and the secondary rotation angle b is -45 degrees. The secondary angle b usually has the same value as the main rotation angle a , but with the opposite sign. This keeps the rotated selection exactly straight.

6. Click *OK*.
7. QCAD creates the rotated copies with the specified rotation angles.

The finished example looks like this:



Trimming Entities

Menu: Modify - Trim

Keycode: RM



The trim tool provides a way to trim an entity to meet another entity. The entity is shortened or extended in such a way that the end point exactly touches the other entity.

For example in the drawing shown in Figure 9-10 we can trim the upper one of the horizontal lines to the skewed line to form an L-shape.



Figure 9-10: With the trim tool, the upper one of the horizontal lines at the left can be shortened in a way that the end point is exactly on the skewed line as shown at the right.

Note that this tool does not operate on a previously made selection. Any existing selection is ignored.

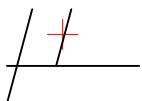
Usage

1. Start the trim tool:



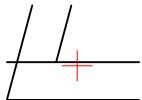
2. Pick first the limiting entity. This entity will not be changed in any way but it defines the position to which the other entity should be trimmed.

For the example drawing, we choose the skewed line to which we want to trim the horizontal line:



3. Click the entity you want to trim. Note that it is significant where you click the entity. Click the entity on that part which you want to keep, not the part you want to trim away when shortening an entity.

In this example, we choose the horizontal line somewhere at the right of the intersection point between the limiting line and the line that has to be trimmed:



4. QCAD trims the horizontal line to meet exactly with the skewed line:

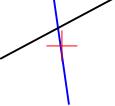
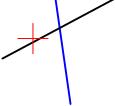
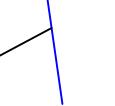
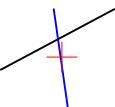
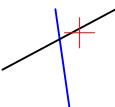
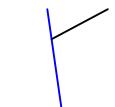
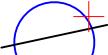
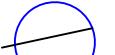
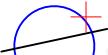
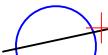
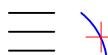
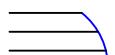


Table 9-3 shows more examples for trim operations. Pay special attention to the relevance of the click points.

Table 9-3

Trimming

Choosing the limiting entity	Choosing the trim entity(ies)	Result after trimming
Extending a line to another line.		

	Choosing the limiting entity	Choosing the trim entity(ies)	Result after trimming
Shortening a line to another line. Note how the click point used when choosing the entity to trim defines which part of the line is kept and which one is trimmed. The part you click on is always the part you want to keep. Here that is the left part.			
Here the right part is clicked and kept.			
Trimming a line to an arc.			
The click point when choosing the limiting entity can also be relevant since there are two possibilities how an arc can limit the trimming of a line.			
Trimming a circle to a line. Because a circle has no end points, the circle is changed into an arc and both arc end points are trimmed to the limiting entity. Here, the top part of the circle is meant to stay and the bottom part to be trimmed (removed).			
In this example, an arc is created from the bottom part of the circle. The top part is removed.			
This example shows how multiple entities can be trimmed to the same limiting entity. Simply click the limiting entity and then click all entities to trim to it.			

2. Start the alignment tool:



3. In the options toolbar, choose if you want to align the selection to the document boundaries or to a picked entity.

You can also choose the vertical and horizontal alignment and if you want to treat the selected entities as one group or as individual entities to be aligned:



Table 9-5 shows the effect of the alignment tool if *Selection as group* is on or off. The selected entities are treated as one object that is aligned to the rectangular polyline if the option is on. Without treating the selection as group, each individual of the selected entities is aligned to the rectangular polyline.

4. If you chose to align the selection to the document boundaries, click the green tick to confirm the alignment operation.
If you chose to pick an entity to align to, click that entity now (the rectangular polyline in our example).

Table 9-5

Alignment

Horizontal Alignment	Vertical Alignment	Selection as group: <i>on</i>	Selection as group: <i>off</i>
Left	None		
None	Top		
Center	Center		
Right	Bottom		

Creating Text Entities

Menu: Draw - Text

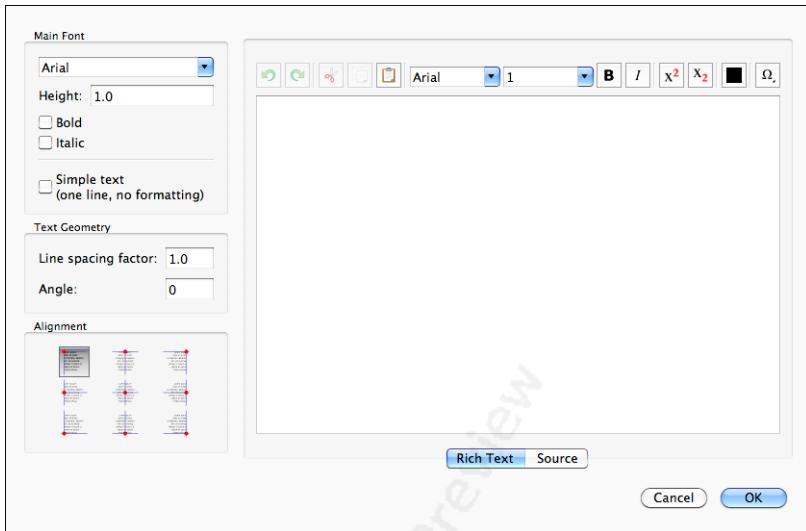
Keycode: TE

A

To create a new text entity, click the text tool button in the CAD toolbar.

Usage

1. The text dialog of QCAD is the starting point to create new text entities:



2. Choose a font for the text entity.

There are two types of fonts available in QCAD:

CAD fonts with letters that consist of lines and arcs. They are shown at the top of the font list. If you are creating a text using a CAD font, the font *standard* is usually a good choice:

Sample text in CAD font 'Standard'

System wide fonts are shown below the CAD fonts. These fonts create texts in which the letters are displayed as filled areas rather than lines and arcs. If you are planning to share your drawing, keep in mind that other users might not have the same fonts installed on their systems as you. The font 'Arial' is usually a safe choice for a system font as it is available on most systems:

Sample text in font 'Arial'

3. Enter the height of your text by following the notes about drawing scale above. Texts that are too small on paper (smaller than about 2.5mm) are hardly readable while large texts might get in the way of the actual drawing elements.
The text height is measured from the bottom line of the text to the top of a capital letter. The height that is chosen is the initial height for your text entity. You may also change the text height anywhere in the text using the text editor at the right.
4. Check the *Bold* or *Italic* choices to make your text bold or italic. These settings are default settings for your text. You may also change the style inline using the text editor at the right.
Bold and italic styles cannot be applied to CAD fonts.
5. Make sure that the *Line spacing factor* is set to 1. The line spacing factor does not need to be changed in most cases.

Angle Dimension

Menu: Dimension - Angular
 Keycode: DN



The angle dimension tool dimensions an angle in degrees or an alternative angle unit. The angle is defined by either two lines in the drawing or an arc.

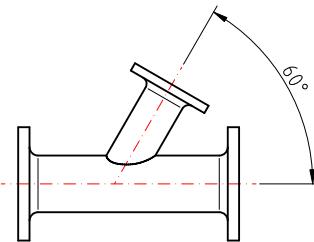
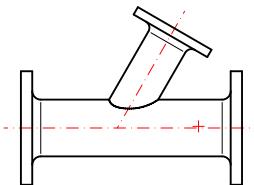


Figure 13-13: Angle dimensions indicate the angle between two lines or the angle of an arc.

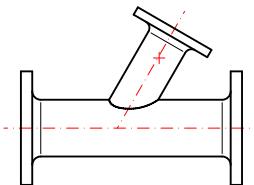
Usage

1. Click the first line that limits the angle or click an arc for which you want to dimension the angle.

In this example, we click the horizontal center line of the drawing:

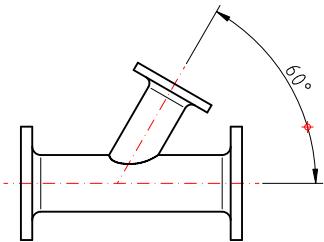


2. If you have clicked a line in the previous step, you can now pick another line. We pick the other center line for this case:



If you have picked an arc in the first step, this step does not apply since an arc defines the start angle and the end angle for the angle dimension.

3. Click the position of the dimension line.



Text string for dimension label and explanation	Example
<p>Text string: $<> G7(5+0.028\)+0.007;)$ To have a text after a subscript or superscript (here a closing bracket), the text sequence for subscripts and superscripts can be used instead of the tolerance input fields.</p>	
<p>Text string: $<>\pm 0^{\circ}0'30"$ Custom label example with an angle dimension.</p>	

Moving the Text Label

QCAD automatically positions the label of dimensions. Sometimes QCAD places the label in a position where it overlaps with other parts of the drawing (for example as shown in Figure 13-17). In such cases, you can manually move the label to another position.

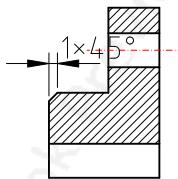
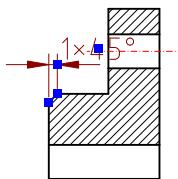
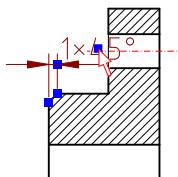


Figure 13-17: In situations like this, the dimension text label can be repositioned manually.

1. Make sure that no tool is active and QCAD is in its neutral state.
2. Select the dimension entity for which you want to move the label by clicking on it.
QCAD highlights that entity and shows its reference points as blue spots:



3. Move the mouse cursor to the blue spot that is at the center of the text label:



4. Press the left mouse button and hold it down.
5. Move the mouse cursor until it turns into a crosshair.
6. Let go of the left mouse button.

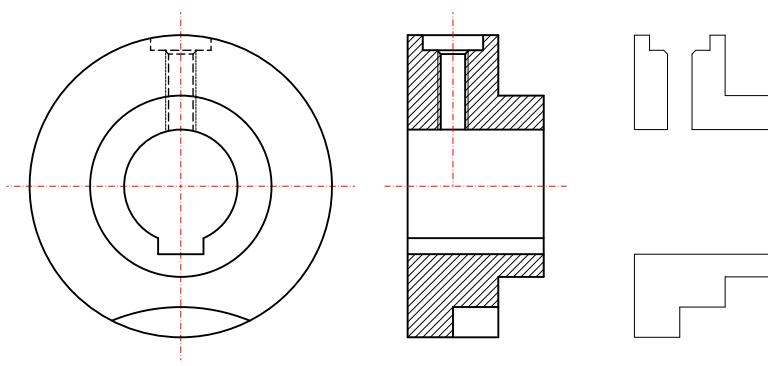


Figure 14-4: Before an area can be filled with a hatch or solid fill, the boundary of the area has to be isolated as shown at the right.

For better understanding of the drawing in Figure 14-4, Figure 14-5 shows an isometric drawing of the same mechanical part.

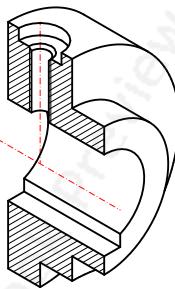
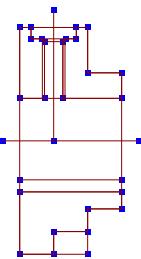
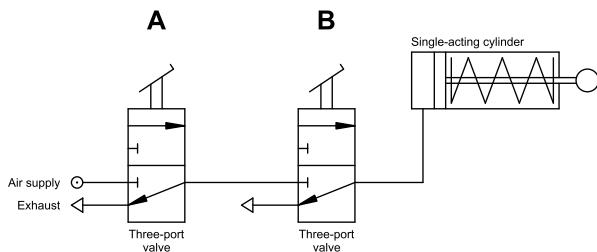


Figure 14-5: Isometric drawing of the part in Figure 14-4.

The following instructions guide you through the process of isolating the hatch boundary. This is not the only way how this can be done. The goal is to have the complete, closed hatch boundary on a separate layer, so it can be easily selected and later hidden or deleted when it is not needed anymore.

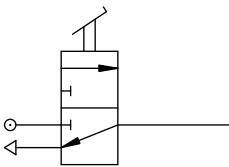
1. Create a new layer for the hatch boundary.
For this example, we call the layer *hatch boundary*. The layer attributes don't matter, you can just leave the default attributes on.
2. Select all entities that are or might be part of the hatch boundary. At this point it does not matter if you select too many entities. If your drawing is not very large, you can also simply select the whole drawing.
For our example, we select the complete cross section:



**Figure 15-4:** Blocks are very useful for symbols in schematics like this.

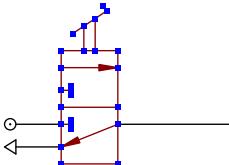
- First, the symbol can be constructed with the normal drawing and modification tools of QCAD.

For this example, we assume that the drawing has been started with the entities shown here:



- Select the entities that make up the new block.

In the example, we select all entities of the symbol that represents a pneumatic three-port valve:

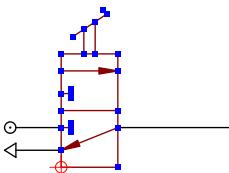


- Click the tool button in the CAD toolbar to create a block from selected entities:



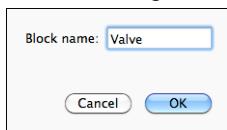
- Click a reference point for the block. This point should be a significant point of the block. It will be used later when positioning references of that block.

In our example, we use the left bottom corner as reference point:



- QCAD shows a dialog to enter a name for the block. Enter a unique name that helps you to later identify the block.

For this example block, we enter *Valve*:



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