

Drawing

From this point on, when we refer to a command, we will separate the main and submenu selections by a forward slash (/) and the text will be in *italics*. For example, the line drawing command would be referred to as *Draw/Line*. To select it, you would move the mouse pointer so it is over the text *Draw* on the pull down menu and click the left mouse button. Then you would move the mouse pointer over the text *Line* and click the left mouse button. The line drawing command is then started.

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This tutorial describes how to create drawing elements (e.g. lines, arcs, text, etc.). Select the *File/New* command to create a new drawing.

To draw elements within the drawing area, select *Draw* from the pull down menu at the top of the window. Then choose one of the options on this menu. For example, to draw a line select *Line* from the *Draw* menu.

Draw	View	Snaps	Options
Line			Ctrl+L
Arc			Ctrl+A
Circle			Ctrl+I
Point			Ctrl+P
Text			Ctrl+T
Insert symbol...			
Load symbol library...			Ctrl+O
Dimension			▶
Polyline			▶
Rectangle			
Fill / Hatch			
Stop			Esc

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Choose the *Draw/Line* command. The cursor changes from the arrow cursor to a plus cursor when it is in the drawing area. The prompt area at the bottom of the window will display:

Select first end point of line

The prompt area tells you what the program is currently looking for. In this case, it is asking you to enter the first point of a line.

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Move the mouse pointer over the edit box to the right of the X field in the coordinate input dialog and press the left mouse button. Remove the current contents of the field and type the number one (1) in the X field. From this point on, when we indicate text which you must enter into a field, it will be enclosed in double quotes ("). Type the text without the quote marks. Switch to the Y field by clicking on it, clear it, and type "1" into the Y field. Click on the *OK* button or press the Enter key. You have now designated the position of the first end of the line as (1,1).

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After specifying the first end of the line, as you move the cursor, a line is drawn from the first point to the current cursor position. This is sometimes called a rubber band line. It shows you what the line would look like at the current cursor position. The prompt area has changed to:

Select second end point of line

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From this point on, when you need to enter a coordinate, we will simply give you the xy coordinates in parentheses (x,y). To enter the coordinate, you must press the F2 key or click the right mouse button to bring up the coordinate input dialog. Then type the x value in the *X* field and the y value in the *Y* field and press Enter or click on *OK*.

Enter the coordinate (5,1) for the second end of the line. You should now have a horizontal line near the bottom of the drawing area.

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We will now check and make sure you have drawn the line correctly. Choose the *Edit/Select/Select one* command. The prompt area displays: *Select a single element*. The cursor should be a box when in the drawing area. Move the cursor so the line you just drew passes through it and click the left mouse button or press Enter.




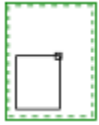
The line will be drawn in a light gray color. This means it is selected. Choose the *Edit/List* command. A dialog box will be displayed which gives the properties associated with the selected line.

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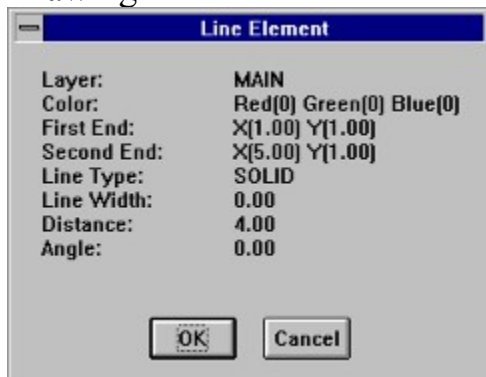
We will now draw an arc on top of the rectangle we just drew. Choose the *Draw/Arc* command.

The prompt area will display: *Select first end point of arc.* Click on the *End* snap button  and move the box cursor so the top right corner of the rectangle passes through it. Click the left mouse button.



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The first end should be (1,1) and the second end should be (5,1). Press the *OK* button. If the line is incorrect, press the Delete key and it will be erased. Go back and redraw the line. When the line is correct, move onto the next step.

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We are now going to draw a vertical line which starts at the right end of the line you just drew.

A quicker way to initiate the line command is to click on the line button  on the command tool bar. The command tool bar is just below the menu.

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Choose the line command. To start this new line at the end of the last line, we are going to use the *End* element snap. Click on the end snap button  on the snaps tool bar. The snaps tool bar runs down the left side of the drawing area. You can also set the end snap mode by choosing *End* from the *Snaps* menu. The cursor should be a box when it is in the drawing area. Move the cursor so the line you drew previously passes through it. The cursor should be nearest the right end of the line. Click the left mouse button or press Enter.



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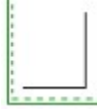
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The program will snap to the right end of the line. This means the first point of the new line will be set to exactly the same coordinates as the right end of the previous line. Snaps are used to specify a position based on an existing drawing element. As you move the cursor, a line is drawn from the end of the previous line to the current cursor position. If the first point is not at the end of the previous line, press the Esc key and try the command again.

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We are going to use x and y offsets (dx,dy) to specify the second point of the vertical line. Bring up the coordinate input dialog. Click on the *XY Offset Input* radio button. The prompts on the 2 fields on the right will change to *DX* and *DY*. *DX* is the offset of the second end point of the line from the first in the x direction and *DY* is the offset in the y direction. Type "0" for dx and "5" for dy. Press the *OK* button. A vertical line is added to the drawing, beginning at the right end of the first and 5 units high.




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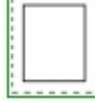
The line command automatically continues from the end of the last line drawn, unless you end the command. As you move the cursor, a line is drawn from the top end of the vertical line to the current cursor position. We will now draw a horizontal line using polar input. When using polar input, you specify the distance of the line and the angle from the start point. Bring up the coordinate input dialog and select the *Polar Input* radio button. The prompts next to the 2 fields on the right will change to *Distance* and *Angle*. Type "4" in the distance field and "180" in the angle field and press the *OK* button. A line will be added with a length of 4 and at a 180 degree angle. The angles are measured counterclockwise from the x axis (i.e. 0 to the right, 90 up, 180 left and 270 down).

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We will now close the box you have been drawing. Make sure the *End* element snap button is still pressed on the snap tool bar. Move the box cursor so the left end of the first line you drew passes through it. Click the left mouse button or press Enter. Click on the stop button  on the command tool bar (this button may not appear due to the current size of the program window) or press the Esc key to end the line command. The prompt area is blanked and the cursor returns to the standard arrow cursor.

Your drawing should look like this:



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As you may have noticed by now, moving the mouse cursor over the command and snap tool bars displays a description of the button the mouse is over. If you are unsure what a tool bar button does or which button to choose, you can move the mouse cursor over the tool bar to find out. This descriptive text is also displayed when you use the arrow keys to highlight different commands on the pull down menu.

GammaCAD needs to be active before it can accept keyboard input. Click on its title bar to activate it before entering key strokes.


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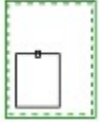
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You may have also noticed, the cursor gives an indication of what type of input the program is looking for. The cursor is an arrow in the drawing area when no command is active. The cursor is a plus or a box in the drawing area when a command is active. The arrow cursor is always displayed outside of the drawing area. The plus cursor is displayed when no snap mode is active. The box cursor indicates either the program is prompting you to select a drawing element, or enter a position and a snap mode is active. In either case, you need to move the box cursor so the desired drawing element passes through it.

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The prompt area now displays: *Select center point of arc.* Click on the *Midpoint* snap button .
. Move the box cursor so the top line of the rectangle passes through it and click the left mouse button.



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The prompt area now displays: *Select second end point of arc.* As you move the cursor, an arc is drawn using the current cursor position as the second end point. Click on the *End* snap button and move the box cursor so the top left corner of the rectangle passes through it. Click the left mouse button. Your drawing should now look like this.



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We will now save this drawing. Click on the save button  on the command tool bar. The *Save As* dialog will be displayed. Type the name "tutor1" into the *File Name* box and click on *OK*. The name in the title bar will change from *Untitled* to *TUTOR1.GCI*.

If you leave this tutorial, you should first save the drawing. When you come back to the tutorial, re-open the drawing by clicking on the open button  and double clicking on the *tutor1.gci* file name.

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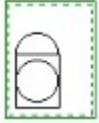
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Now we will draw a circle inside the rectangle. Choose the *Draw/Circle* command. The prompt area will display: *Select center point of circle*. Bring up the coordinate input dialog (press the right mouse button while the cursor is in the drawing area). Make sure the *XY Input* radio button is selected and enter the coordinates (3,3.5).

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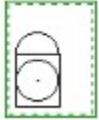
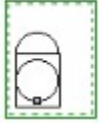
The prompt area now displays: *Select point on the circle.* As you move the mouse cursor, a circle is drawn using the current position as the point on the circle. Bring up the coordinate input dialog and select the *Polar Input* radio button. Type in a distance of "2" and an angle of "0" and click on *OK*. You have now created a circle of radius 2. This makes it tangent to the right and left edges of the rectangle. Your drawing should now look like this.



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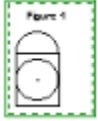
Now we will draw a point in the center of the circle. Choose the *Draw/Point* command. The prompt area will display: *Select point*. Click on the *Center* snap button  and move the box cursor so the circle and only the circle passes through it (as in the first figure below). Click the left mouse button. A point is added to the drawing at the center of the circle. Your drawing should look like the second figure below.



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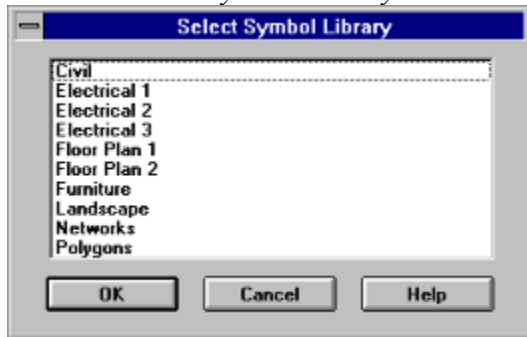
The *Draw/Text* command allows you to add text strings to a drawing. Choose this command and bring up the coordinate input dialog. Make sure the *XY Input* radio button is selected and enter the coordinates (2,10) for the text alignment point. The program will then display a dialog in which you enter the desired text string. Type in "Figure 1" and click on *OK*. Your drawing will now look like the one below.



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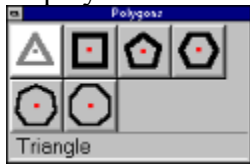
GammaCAD comes with many pre-drawn symbols which you can place in any drawing. Choose the *Draw/Load symbol library* command. The following dialog will be displayed.




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Click on the *Polygons* library in the list box and click on *OK*. The following dialog will be displayed.




Click on the triangle button. The prompt area will display: *Select symbol insertion point*. The insertion points correspond to the red dots in the dialog above. Click on the *Point* snap button  and move the box cursor so it contains the point in the center of the circle. Click the left mouse button.

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Next, the program will prompt you for a rotation angle. Click on *OK* to use the default of 0. The scale dialog is then displayed. Enter a scale factor of "2." Your drawing will now look like the one below.



You can move the symbol dialog like any window. Click on its title bar and drag it to the desired location. Double click on the system menu  in the upper left corner of the dialog to close the library.

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Polylines are similar to lines, except they can contain multiple segments. Choose the *Draw/Polyline/Closed polyline* command. The prompt area will display: *Select first end point of closed polyline*. Enter the coordinates (6,1). The prompt area changes to: *Select next end point of polyline*. As you move the cursor, a line is drawn from the first end to the current cursor position. Enter the coordinates (8,1) for the second point and the coordinates (8,3) for the third point. Press the Esc key to end the command. A closing line is automatically added to the polyline.

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Your drawing should now look like the one below.

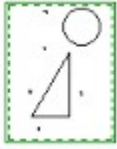


All the segments of a polyline are treated as one drawing element. A figure drawn with polylines can be filled and hatched, whereas a figure drawn with lines cannot.

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Choose the *File/Open* command or click on the open button  on the command tool bar. Double click on the file *tutor3.gc1*. It should look like this.



We will use this drawing to demonstrate the dimensioning commands in GammaCAD.


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Choose the *Draw/Fill / Hatch* command. The prompt area will display: *Select a polyline to fill / hatch*. Move the box cursor so a segment of the polyline you just drew passes through it.


Click the left mouse button. The interior of the polyline is then filled as in the figure below.

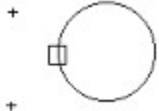


Choose the *File/Save* command or click on the save button  to save the drawing.

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Choose the *Draw/Dimension/Leader* command. The prompt area will display: *Select arrow head point of leader*. Click on the *Nearest* element snap button  and move the cursor so the left edge of the circle passes through it.



Click the left mouse button. The program will snap to the point on the circle which is nearest the cursor position.

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The prompt area will display: *Select second end point of leader*. As you move the cursor, a line is drawn from the first point selected to the current cursor position. This represents the leader line. Click on the *Point* element snap button  and move the box cursor so the point to the left of the circle falls within it.

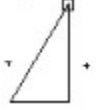


Click the left mouse button. Enter the text "Leader" in the dialog which is displayed next and press *OK*. The leader is then added to the drawing.

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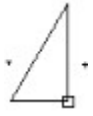
Choose the *Draw/Dimension/Vertical* command. The prompt area displays: *Select first end point of vertical dimension*. Click on the *End* element snap button  and move the box cursor over the top corner of the triangle. Click the left mouse button.



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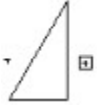
The prompt area will display: *Select second end point of vertical dimension.* Move the box cursor over the lower-right corner of the triangle and click the left mouse button.



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The prompt area will display: *Select position of dimension line.* Choose the *Point* element snap  and move the box cursor over the point to the right of the triangle. Click the left mouse button.




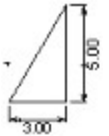
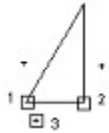
A dialog is displayed with the vertical dimension value calculated from the first 2 points you input. You could make changes to this value (e.g. add units like inches). Click on the *OK* button to accept the default value. The vertical dimension is added to the drawing.

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Creating a horizontal dimension is very similar to creating a vertical one. Choose the *Draw/Dimension/Horizontal* command. Select the *End*  element snap and pick points 1 and 2 in the figure below, left. Select the *Point*


 element snap and pick point 3. Press *OK* to accept the default dimension text. The dimension should look like the figure on the right, below.

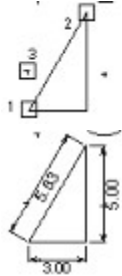


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Creating a parallel dimension is very similar to creating a vertical and horizontal one. Choose the *Draw/Dimension/Parallel* command. Select the *End*  element snap and pick points 1 and 2 in the figure below, left. Select the *Point*

 element snap and pick point 3. Press *OK* to accept the default dimension text. The dimension should look like the figure on the right, below.



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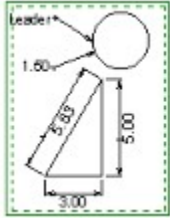
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Choose the *Draw/Dimension/Radial* command. The prompt area displays: *Select arc or circle to radially dimension.* Move the box cursor so the circle at the top of the drawing pass through it. Click the left mouse button. Now the prompt area displays: *Select the position of the radial dimension.* Select the *Point*  element snap and move the box cursor over the point below and to the left of the circle. Click the left mouse button. A dialog displays the radius of the circle. Press *OK* to accept the default text. The radial dimension is then added to the drawing.

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The drawing should now look like the one below.



This ends this tutorial.

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With GammaCAD you can draw using exact coordinates. To do this, we will use the coordinate input dialog. But first, a quick note about switching between this tutorial and GammaCAD. When you bring up a dialog in GammaCAD, like the coordinate input dialog, it may cover up this tutorial text. To switch back to the tutorial, move the mouse cursor over the tutorial text or title bar and click the left mouse button. To move back to the dialog, click on it. Be careful where you click on a dialog, because if you click on a control, not only will the dialog be re-activated, but the function of that control will also be performed. For example, if you click on the *OK* button to re-activate a dialog, it will be re-activated, but it will also exit the dialog using your current input.

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We will now bring up the coordinate input dialog in GammaCAD. After doing so, switch back to this tutorial by clicking on this text area or on the tutorial title bar. Move the mouse cursor into the drawing area and click the right mouse button. The coordinate input dialog will be displayed. Click on the tutorial to re-activate it.

You can also move the dialog so the tutorial text is not blocked by it. To do this, click on the dialog title bar and drag it to a new position.

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