

TotalCAD Tutorial Topics



How to Use the Tutorial



Quick Tour of TotalCAD



Tips for Users



TotalCAD Basics



TotalCAD Features



Drawing an Office

TotalCAD Features



Snap Modes and Coordinate Systems



Formatting Objects



Node Edit



Dimensions



How to Use the Tutorial

This electronic tutorial is designed to help you start using TotalCAD quickly and easily.

Keep TotalCAD running while you go through the steps of this tutorial. You can use the Show Me button, shown below, to see examples of the work in progress:



The sample drawings that you will need for the tutorial can be found in the TotalCAD Tutorial folder. The tutorial folder is located inside your TotalCAD folder. If you used the default settings when you installed TotalCAD, the path of the Tutorial folder will be C:\IMS\TotalCAD\Tutorial.

You can come back to the tutorial at any time by selecting Help|Tutorial from the TotalCAD menu.



Quick Tour of TotalCAD

Using a CAD (Computer-Aided Design) program can seem daunting until you realize the simplicity of the CAD approach to drawing. Unlike "freehand" drawing programs, a CAD program provides tools to create *dimensionally accurate* drawings, which are precise models of the objects that they represent.

The best approach to TotalCAD is to start with simple drawings (as shown in this tutorial) and expand into more complex designs as you learn how to use the tools.

This tutorial topic will take you through a quick tour of the basic elements of the TotalCAD interface.



→ Starting a New Drawing

To start a new drawing, choose New from the File menu. If you haven't switched off the Prompt for Template Name option in the Tools|Program Setup menu, the Create from Template dialog will appear.



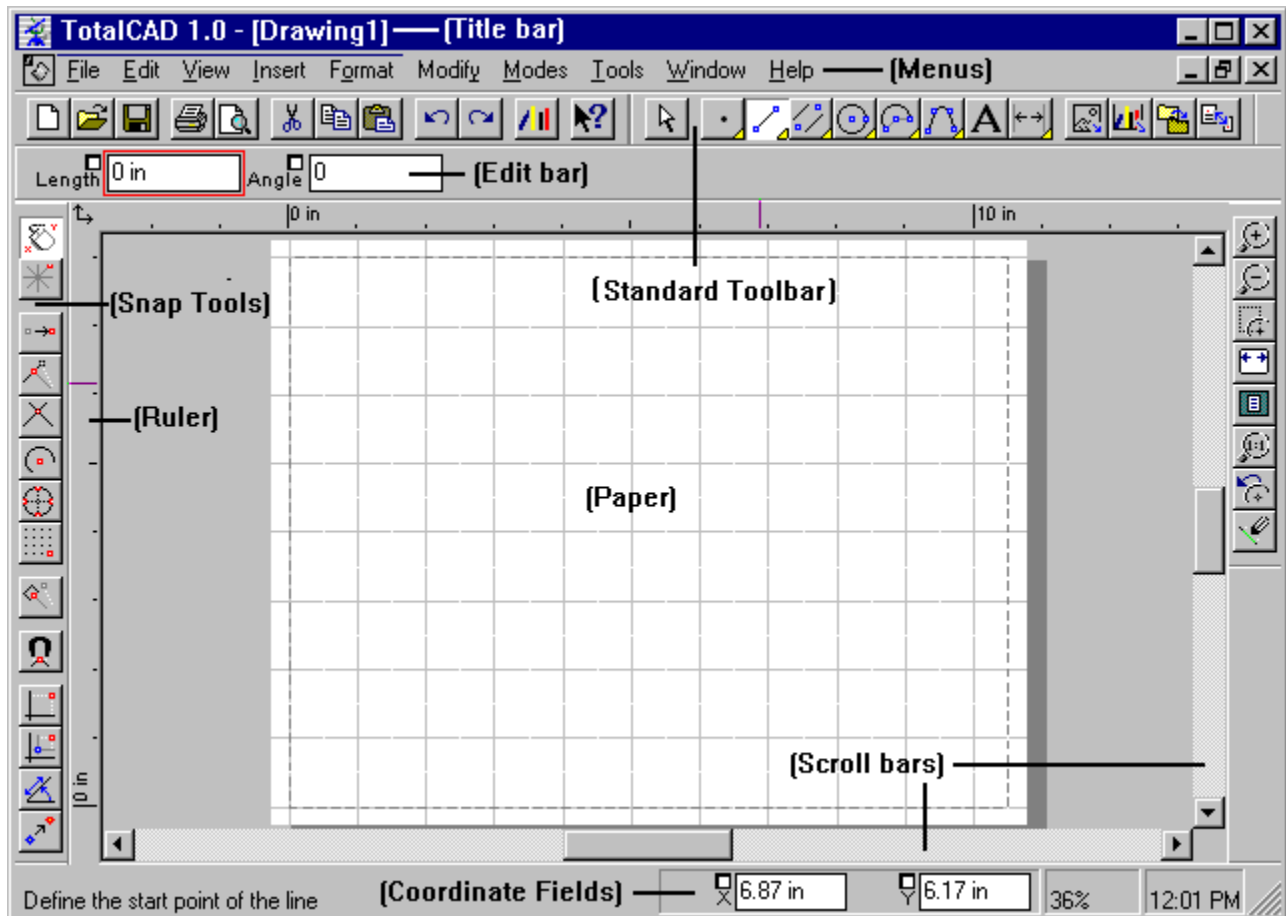
For this tutorial, choose the Quiktour template.

Templates do most of the work of setting up your drawing. They contain its system of measurement, grid options, page setup, and much more. As you gain expertise in TotalCAD, you will probably want to modify the settings in templates and create drawing templates of your own.



TotalCAD Screen

Your screen should now look something like this. The components of the TotalCAD screen are listed below.



Title Bar

The title bar at the top of the window displays the name of the program and the name of the currently active drawing.

Menus

These are standard Windows menus. To choose a menu command, click on the menu name, then click on the name of command in the menu that appears. Some commands display submenus from which you can again choose a command.

Paper

The rectangle in the middle of the drawing area is the Paper. This is the area of the drawing that will be printed.

Standard Toolbar

The Standard toolbar contains tools for opening, saving, and printing drawings, as well as other standard functions. By default, this toolbar is docked horizontally at the top of the screen. You can reposition it by dragging it to the side of the screen or to the drawing area.

Edit Bar

The Edit Bar lets you draw and edit by typing numeric values into the Edit Bar fields. You can enter the Edit Bar from the Coordinate Fields or drawing area by pressing <Ctrl>+<E>.

Snap Tools

The tools on the Snap toolbar let you activate snap and coordinate systems modes. By default, this toolbar is docked vertically. You can reposition it by dragging it to the top of the screen or the drawing area.

Coordinate Fields

The Coordinate Fields let you specify locations in your drawing by typing their coordinates. You can use absolute, polar, or relative coordinates, depending on the current coordinate system. You can enter the Coordinate Fields from the Edit Bar or drawing area by pressing <Ctrl>+<R>.

Ruler

The vertical and horizontal rulers provide feedback about the location and size of objects in your drawing. You can change the rulers between World and Paper coordinates by clicking on the World/Paper button in the upper-left corner of the rulers. You can hide the rulers to increase the size of your drawing area using the View|Rulers command.

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Scroll bar

Click on this scroll bar to "scroll" through your drawing. You can drag the "thumb" of the scroll bar to move rapidly through the drawing. Clicking on the arrow at either end scrolls a small amount. Click in the area between the thumb and the arrow to scroll a larger distance.



→ **Help**

To get help on any screen element or command select the context-sensitive help button on the Standard toolbar, then click on the tool, menu item, or screen element that you want help on.



You can also press <F1> at any time to get information about a command, dialog, or process. To get help on an item in the local menu, just highlight the item, then press <F1>.

To practice, try getting help on one screen element, one menu item, one tool, and one local menu item using the context-sensitive help button and the <F1> key.



→ Accessing Commands

TotalCAD lets you access commands in three ways: through menus, toolbars, and local menus.



Menus

Menus are shown at the top of the program window and operate just like menus in any standard Windows program. Click on the menu name to display the menu, then click again to choose a command or submenu.



→ Toolbars

Toolbars are bars containing command buttons. TotalCAD contains many toolbars, only a few of which are displayed on your screen when you first start the program. You can display and hide toolbars using the View|Toolbars command.

The Insert toolbar, which includes the basic drawing tools, contains *flyout toolbars*. Tools containing flyouts are marked with a yellow triangle in their lower-right corner. To view a flyout toolbar, position the mouse cursor over a button, then hold the left mouse button down. The flyout will appear below the tool. You can now click on any tool on the flyout to select it.





Local Menus

Local menus are accessed by clicking the right mouse button in the drawing area. These menus are context-sensitive, meaning the menu choices shown will depend on what you are doing at the time you display the local menu. You can use the local menus to cancel or finish a drawing operation, to change snap modes, and to access commands that are specific to a drawing or editing tool.





→ **Controls**

In addition to the command interface, TotalCAD has two other important *controls* that you can use to draw and edit. These are the *Edit Bar* and the *Coordinate Fields*.



Edit Bar

TotalCAD's Edit Bar is normally displayed just above the drawing area, although you can drag it to another screen position if you prefer.

The Edit Bar changes as you work with the program to show you lengths, angles, and other important information. You can enter information directly into the Edit Bar to draw entities using numeric values.



Using the Edit Bar

For example, you might want to draw a line exactly two feet long (in World units) at exactly 45 degrees. To do this:

1. Click on the Single Line tool on the Insert toolbar.



2. Click on the No Snap tool on the Snaps toolbar to make sure that snap modes are turned off.



3. Click anywhere in the drawing area to define the first point of the line.
4. Press <Tab> to enter the edit bar, then type **2** into the Length field.
5. Press <Tab> again to move to the Angle field, then type **45**.
6. Press <Enter> to complete the line.

You have now drawn a line two feet long (in World units) at exactly 45 degrees, using the Edit Bar.





Coordinate Fields

The Coordinate Fields, located in the lower-right corner of the TotalCAD program window, allow you to specify locations in the drawing by typing numeric coordinates.



Using the Coordinate Fields

Let's draw a circle using a combination of the Coordinate Fields and Edit Bar.

1. Activate the Center and Point circle tool.
2. Press <Ctrl>+<R> to go to the Coordinate Fields. (You could also simply click in the first field.)
3. Type **3** into the X field.
4. Press <Tab> to go to the Y field, then type **3**.
5. Press <Enter> to accept the values you have typed. TotalCAD draws a marker at a point three feet across on the X axis and three feet up on the Y axis.
6. Press <Ctrl>+<E> to go to the Edit Bar. Type **1** into the Radius field.
7. Press <Enter> to accept the value. TotalCAD draws a circle with its center at the point you defined, and with a radius of one foot.





Moving between Controls

Here are some tips for moving between the Edit Bar, the drawing area, and the Coordinate Fields.

- Press <Ctrl>+<E> to move to the drawing area.
- Press <Ctrl>+<D> to move to the drawing area.
- Press <Ctrl>+<R> to move to the Coordinate Fields.

If you are in the drawing area and you begin typing, the Edit Bar will be activated and the characters you type will be placed in the first field of the Edit Bar. (This won't work if you are using the Text tool, however.)

You can also press <Tab> and <Shift>+<Tab> to move between fields on the Edit Bar or Coordinate Fields. Pressing <Tab> from the drawing area will move you into the first Edit Bar field.

And, of course, you can always move from one area to another by clicking.



Conclusion

This concludes the Quick Tour of the TotalCAD interface. For an introduction to basic drawing and editing techniques using TotalCAD's tools, run the TotalCAD Basics tutorial.



Tips for Users

You will see most of your favorite tools and commands in the TotalCAD 1, and you'll also find many features and capabilities. This topic discusses some of the more important features and capabilities.



Toolbars

The toolbars in TotalCAD have been designed and organized for convenient use. Here are the two most important features:

- The toolbars are floating and dockable. To see this capability, grab any toolbar with your mouse (in an area next to a button) and drag it into the drawing area. Drag the toolbar to a location on the edge of the drawing area to dock it permanently.
- The Insert toolbar (the one with the basic line, circle, text, etc. drawing tools) has "flyout toolbars".



Hold the left mouse button down on one of the buttons that has a yellow rectangle on the lower-right corner: a flyout toolbar will be displayed. This toolbar is similar to a pull-down menu, except with buttons.



Property Sheets

Object properties (*entity attributes*) are set via *property sheets*. Property sheets are dialog boxes with tabs for different pages of settings. To access properties for any entity, you can select the entity, then select Format|Properties to see the associated property sheets. To set the default properties for an entity type (that is, all lines to be drawn next), you can select Format|Properties when the desired drawing tool is selected. Alternately, you can right-click on the drawing tool to access the default property sheets for that type of entity.

For example, right-click now on the single-line segment tool



You will see this dialog, containing three property sheets.





Local Menu

Another powerful interface feature of TotalCAD 1.0 is the *local menu*. Many commands are available through local menus, accessed with a click of the right mouse button in the drawing area. Use the local menu to set snap modes while drawing, to finish or cancel an operation, and to perform actions specific to the drawing tool that you are using.

1. To see an example, first click on the single-line tool to activate it.



2. Now right-click in the drawing area to display the local menu.
3. Choose Local Snap to display the Local Snap submenu.





Snap Modes

The snap modes in TotalCAD can be combined, allowing you to use multiple snap types at the same time. Some of the snap modes have been combined to simplify the operation of the program.



Symbol Treatment

A great deal of work has been done to simplify the use of symbols in TotalCAD. Symbols in TotalCAD 1.0 are simply folders containing any CAD files that TotalCAD can read (including .DXF). Once you have attached a symbol library to your drawing (using Tools|Symbol Library), you can drag and drop the symbols into your drawing using the Symbol Library palette (View|Libraries).



You can also drag objects from your drawing into a symbol library to convert them into symbols.



Conclusion

This is the conclusion of the Tips for Users section. Check your Online User Guide for more tips on how to use TotalCAD.



TotalCAD Basics

The next few topics introduce some basic concepts that will help you get started with TotalCAD.



Starting a New Drawing

When you start a TotalCAD, the program asks you to select a template, on the basis of which it automatically creates a new, blank drawing. In some cases, you will want to make some changes to the template's settings to suit the drawing task you have in mind, but for this introductory example, you will use the settings of the Normal template.

To start a new drawing:

1. Choose New from the File menu. The Create from Template dialog appears.



2. In the Create from Template dialog, choose the Normal template, then click OK. A new blank drawing is created, with pre-set grid settings, unit and scale settings, page layout settings, and so on.





Drawing Basic Entities

The next set of topics will show you how to use some basic drawing tools in combination with Snap to Grid.



Using the Grid

Notice that pattern of intersecting lines displayed on the Paper, which occupies much of the drawing area. These lines are the *grid*. You can turn the grid display on and off using the View|Grid command. When snap to grid mode is on, points that you define by clicking in the drawing space will "snap" to the grid points (that is, to the points where the lines on screen intersect).

Turn Snap to Grid on now by clicking on the Snap to Grid button on the Snap toolbar.





Magnetic Point and Snap Aperture

Magnetic Point is a tool you can activate to see where your next mouse click to snap to before you actually click the mouse. This tool will help you avoid mistakes. Turn on this feature now by clicking on the Magnetic Point tool, located on the Snap toolbar.



Snap Aperture puts a circle around your drawing cursor to show you how close you must be to an object before TotalCAD will snap to a point on the object. Turn on this feature now by choosing Aperture from the View menu.



Drawing a Line

Let's practice drawing some single lines using the grid, Magnetic Point, and snap aperture.

1. Select the Insert|Line|Single menu command, or click on the single line drawing tool in the Insert toolbar.



2. Move the cursor near any grid intersection on the drawing. When the grid point is inside the snap aperture circle, click the mouse to define the first point of the line.
3. Move near another grid intersection. As you move the mouse a *rubber band* line stretches between your mouse position and the first point. Notice that because Magnetic Point is on, then rubber band line jumps to any grid intersection that falls within the snap aperture.
4. Click to define the second point of the line.

Draw a few more lines using this method.





Drawing a Rectangle

Let's select the rectangle tool using a flyout toolbar.

1. Hold the left mouse button down on the single-line drawing button to see other types of line drawing tools and select the Rectangle drawing tool.



Notice that the Insert toolbar now displays the Rectangle tool in place of the Single line tool, so that you can re-select the Rectangle tool without using the flyout toolbar.

2. Move the cursor near any grid intersection on the drawing and click the left mouse button to "anchor" the starting point for a rectangle.
3. Move near another grid intersection and click the mouse to define the opposite corner of the rectangle.





Drawing a Multisegment Line

Select the Multiline drawing tool in the Insert toolbar



1. Move the cursor near any grid intersection on the drawing and click the left mouse button to anchor the starting point for the multiline.
2. Move near another grid intersection and click the mouse again to complete the first line.
3. Move near another grid intersection and click the mouse again to add another line segment.
4. Repeat step 3, drawing a few more line segments, then click the right mouse button to display the local menu, then select Finish. (You could also double-click the left mouse button to finish your multiline.)
5. Try another multiline with at least three line segments, then use the Close command on the local menu to see another useful way to finish the multiline drawing.





Undo and Redo

If you make a mistake, you can use the Undo button, located on the Standard toolbar, to undo the previous drawing, editing, or formatting operation.



You can also use the local menu or choose Edit|Undo in the main menu to access the Undo command. If you mistakenly undo an operation and then decide that you want to restore it, use the Redo tool.



To see these tools in action, click the Undo button three times to undo the last three entities that you drew. Then click the Redo button three times to restore the entities.



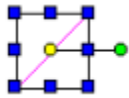
Select Editing

Select editing, otherwise known as handle-based editing, allows you to move, copy, resize, rotate, and format objects in an easy and intuitive way. To use select edit, you must first select some objects you have drawn.

1. Click on the Select tool, located on the Insert toolbar, to put TotalCAD into Select mode.



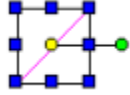
2. Now click on one of the lines you have drawn. You will see selection handles similar to this:



You can select multiple objects by holding the <Shift> key while clicking on objects or by holding the mouse down in an open area of the drawing and dragging a selection window around the objects.



Resizing



The blue squares, which define the selection rectangle, are *resizing handles*. You can drag the resizing handles with the mouse to resize the selected object. Hold down the <Shift> key to retain the proportions of the object while resizing. Try it now:

1. Turn off all snap modes by clicking the No Snap icon.

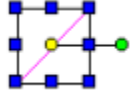


2. Place the mouse cursor over any selection handle, then drag the handle while holding down the <Shift> key. The line will be rescaled proportionally.

Note that you can also use snap modes when you are resizing an object.



Rotating



The green circle is the *rotation handle*. You can rotate the object by dragging the rotation handle. Note how the object is rotated around the center denoted by the yellow circle. You can constrain the angle of rotation using Ortho mode. To do this, click on the Ortho button, then rotate the object.



Moving

There are three basic methods of moving an object: simple dragging, "picking" the reference point, and OLE drag and drop.

To use *simple dragging*, just drag the object by any point inside the selection rectangle, except by the yellow dot. Simple dragging is convenient if you don't care about precise placement, since it doesn't permit you to use snap modes.

To *pick the reference point*, click on the yellow circle. The object will be locked to the mouse, allowing you to snap the object to grid points or other snap locations. You can also use the Coordinate Fields to specify a coordinate location at which to place the object.

To use *OLE drag and drop*, drag the object by its the yellow circle. This option allows you to drag objects from one drawing into another.

Try moving the object you have selected using all three of these methods.



Deleting

You can delete selected objects by pressing the <Delete> key, or by choosing Clear from the Edit menu.



Making Copies

TotalCAD provides a lot of options for copying objects. Let's look at some of the most common methods:



Simple copying



Rubber Stamp



Copy and paste



Copying with the Control Key

You can copy an object easily by holding down the <Control> key. Try this:

1. Turn on Snap to Grid.



2. Click on the yellow circle inside the selection rectangle.
3. Hold down the <Ctrl> key and click on the grid point where you would like to make a copy of the object.



Rubber Stamp

Rubber Stamp provides any easy way to make multiple copies:

1. Select an object.
2. Click the right mouse button in the drawing area to display the local menu, then select Rubber Stamp.
3. Click the left mouse button to make copies of the selected object anywhere in your drawing.
4. Select "Cancel" from the local menu (or press <Esc>) when you have made all the copies you want.



Copy and Paste

Another method to make a copy is through the Windows Clipboard:

1. Select an object.
2. Select the Edit|Copy command to make a copy of the selection on the Windows clipboard.
3. Select the Edit|Paste command to copy the clipboard back to your drawing.

The new copy will always be inserted in your drawing in the center of the view.



Setting Properties

Properties are the settings that control the appearance and other attributes of objects in your drawings. Common properties include colors, line thickness, and line styles. Other, more advanced properties are available for some types of objects.

You can set properties several different ways, but the basic approach is quite easy.



Tool and Object Properties

You will be interested in changing properties under two conditions:



Setting properties for the objects you are about to draw (defaults). To set the property defaults for new objects, select the type of object you want to draw from the Insert menu or the Insert toolbar. Select Format|Properties or right-click on the tool to display the Properties dialog box for the object type.



Setting properties for objects you have already drawn. If you have already drawn an object and you want to change its properties, select the object(s), then select the Format|Properties command or right-click to display the local menu, then select Properties. The same dialog will appear as the default properties, but changes you make will only affect the selected objects.

Run the [Formatting](#) tutorial for more exercises related to setting properties.



Moving Around Your Drawing

If you are working on a detailed drawing, you will need to get a closer look at some parts of the drawing. TotalCAD provides several ways to move around in your drawing.



Zooming In and Out

Press the + (plus) or - (minus) keys on your numeric keypad to zoom in or zoom out the view of the drawing. You can also use the Zoom In or Zoom Out buttons, or choose View|Zoom|Zoom In



or View|Zoom|Zoom Out.



Note this important distinction. When you use the keyboard, TotalCAD zoom to a view centered on the mouse cursor. This can be very useful, because it allows you to point at the exact location you want to zoom to. If you use the View toolbar or the menu, TotalCAD zooms to or from the center of the current view.



To see the entire drawing, use the View|Zoom|Zoom Full command.



To move the drawing within the viewing window, use the scroll bars (this is one form of panning).



Zoom Window

The *Zoom Window* lets you draw a selection rectangle around the area that you want to zoom in on.

1. Click the Zoom Window tool on the Views toolbar.



2. Place the mouse at one corner of the area that you want to zoom to.
3. Drag the mouse over the area, then release the mouse button.

Of course, the rectangle you draw probably isn't exactly the same proportions as the screen, so TotalCAD will calculate the closest possible rectangle.



Extents and Full View

Two other commonly used zooms are *Full View* and *Extents*. Try these now:

Click on the Full View button to center the entire Paper in the view.



Click on the Extents button to get the largest possible view of all your drawing objects.



TotalCAD has several other powerful options for zooming and panning in your drawing. For details, search for "zoom" and "pan" in the help system.



Printing Your Drawing

TotalCAD provides many printing options. The simplest and most common need is to quickly print the entire drawing on a single sheet of paper. You can print quickly by selecting the print tool on the main toolbar.



This displays the Print dialog.





Center and Fit

Click the Page Setup button in the Print dialog to display the Page Setup dialog. Click the Fit button in the dialog to fit and center your drawing on the page. Click OK to return to the Print dialog, then click OK again to print your drawing.





Saving Your Drawing

You can save your work to disk by selecting the File|Save command or selecting the save tool on the main toolbar.





Conclusion

This concludes the TotalCAD Basics tutorial. If you would like to practice using the tools to create an actual drawing, run the [Drawing an Office](#) tutorial. If you would like to find out more details about TotalCAD features, choose one of the [TotalCAD Features](#) tutorial topics.



Drawing an Office

In this lesson you will create a simple drawing of an office. It will show you how to use basic drawing tools in combination with editing tools.



Preparing to Draw

The first step in creating a drawing is to set up your drawing environment. For this tutorial, we'll use a template with predefined settings.

1. Close all open drawings using the File|Close command
2. Select the File|New command. The Create from Template dialog appears.
3. Click on the Tutor1 template, then click OK to close the dialog.

When you venture into your own drawings, you will probably want to start with one of the predefined templates and modify the settings using the File|Page Setup and Tools|Drawing Setup dialogs.



Turn on Magnetic Point

To make drawing easier during this exercise, make sure that Magnetic Point is turned on. Click on the Magnetic Point tool on the Snap toolbar if it isn't already active.





Drawing Walls

Let's start by drawing the outer office walls. The office that we are going to draw is 25 feet by 30 feet. We can draw the wall by specifying there length using the Double Line Multiline tool.

1. Select the Insert|Double Line|Multiline tool, using either the menu or the Insert toolbar flyout.



2. Select Snap to Grid from the Snap toolbar.



3. Anchor the rectangle by clicking on a grid point at 5 feet across (X) by 30 feet (Y). We can represent this point conventionally as (5,30). To locate the grid point, use the Rulers and Coordinate Fields. Because Magnetic Point is on, the Coordinate Fields will also show the location of the grid point when the mouse cursor gets close to it.
4. Click again at (35,30), (35,5), and (5,5) using the Rulers and Coordinate Fields to help you locate the points.
5. Right-click to open the local menu, then click Close. This draws a complete double-line rectangle.





Adding an Inner Wall

Now let's divide the office into rooms by adding inner walls.

1. Select Insert|Double Line|Single (from the menu or the toolbar).



2. Draw a double line from the top wall to the bottom wall, from point (20,5) to point (20,30).
3. Draw another wall by clicking on points (20,15) and (35,15).





Cleaning Up the Walls

Note: You can start at this point by loading TUTOR101.TCW from the TUTORIAL folder. If you do, immediately use File|Save As to rename the drawing so you don't overwrite the sample files with your changes!

Notice how your inner walls don't intersect cleanly with the outside walls? TotalCAD provides an easy way to clean up these intersections.

1. Select the T-Meet 2 Double Lines from the Modify menu.
2. In turn, click each of the 2 double lines close to where they intersect. The result is a clean intersections of the walls.





Adding Doors

Note: You can start at this point by loading TUTOR102.TCW from the TUTORIAL folder. If you do, immediately use File|Save As to rename the drawing so you don't overwrite the sample files with your changes!

Our office would not be very accessible without doors, so let's add some now. First, use Node Edit to add an outside door.

1. Click the Select tool on the Insert toolbar to go into Select Mode.



2. Click on the outer wall of the office to select it.
3. Right-click to open the local menu, then choose Node Edit. The *nodes* of the outer wall are now visible.
4. Place the mouse cursor over the node in the upper-left corner of the office. The cursor will change into a four-directional pointer when it is in the right position.
5. Right-click to open the local menu, then choose Break. This "breaks" the rectangle at the corner.
6. Click on the node, then click on the next lower grid intersection. This moves the node down, opening a space in the wall.
7. Right-click on one node of one opening and select Close Double End to clean up the door opening. Repeat this with the other opening.
8. Click in the drawing area anywhere other than the wall to deselect the wall.





Add Some More Doors...

You can add doors to the inside rooms by using the Line Length tool.

1. Choose the Modify|Line Length command from the menu.
2. Click near the end of the vertical interior wall at the top. Notice how the end of the wall is now attached to your mouse cursor?
3. Click on the next grid intersection down from the top wall.
4. Repeat steps 2 and 3 to create an opening at the bottom wall.





Cleaning Up Intersections

A little cleanup is now required to make the walls look finished.

1. Click the Select tool, then select the outer wall. Right-click to open the local menu, then choose Node Edit.



2. Right-click to open the local menu again, then choose Drop Gaps. This closes all the gaps in the double lines that were created when you edited the line lengths.
3. Now select the vertical interior wall, use the local menu to first go into Node Edit mode, then right-click on each end in turn to use the Close Double End command.
4. Save your work. (This stage of the tutorial has been pre-saved as TUTOR103.TCW.)





Changing Grid Spacing

Note: You can start at this point by loading TUTOR103.TCW from the TUTORIAL folder. If you do, immediately use File|Save As to rename the drawing so you don't overwrite the sample files with your changes!

For this stage of the tutorial, you will use a smaller grid spacing (2.5 feet):

1. Select Tools|Grid.
2. Change both the Spacing fields to 2.5 feet.
3. Click OK to close the dialog. Notice that your grid points are half as far apart now as they were before.



Drawing Cabinets

To draw a storage cabinet, you will use a new snap mode called *Vertex Snap*, which snaps your drawing cursor to the vertices of objects. Vertices are the endpoints of line segments, the corners of polygons and rectangle, and the endpoints of arcs.

1. Click on the Vertex snap tool (or select Modes|Snaps|Vertex).



2. Select the single line rectangle drawing tool.



3. Click the upper-right corner of the vertical wall on the left side of the drawing
4. Click on the next grid intersection to the right, and down seven spaces.
5. Turn off Vertex snap by clicking once again on the Vertex Snap tool.





Drawing Desks

1. Using grid snap and the single line rectangle tool, draw a desk anywhere on your drawing 2.5 feet by 5 feet (one grid square by two grid squares).
2. Draw another desk the same size rotated 90 degrees.
3. Use the Select tool to select and drag the 2 desks to convenient locations.





Drawing a Conference Table

Let's add one more piece of furniture to the office: a circular conference table.

1. Select the Center and Point circle tool.



2. Click a grid intersection in the center of the upper right room, then click the the next grid intersection to the right to complete the circle.
3. Click on the Select tool, then click on the circle to select it.



4. Right-click to open the local menu, and choose Properties.
5. Click on the Brush tab to bring the Brush Property sheet to the front. Select the PARQ1 pattern and set the Hatch Scale to 0.1, then click OK.



6. Save your work. (This stage of the tutorial has been saved as TUTOR104.TCW.)





Adding Text

Note: You can start at this point by loading TUTOR104.TCW from the TUTORIAL folder. If you do, immediately use File|Save As to rename the drawing so you don't overwrite the sample files with your changes!

Your office layout is almost complete. You need to add some text to your drawing to identify the rooms.

1. First, click the Text tool on the Insert toolbar.



2. Turn off the Snap to Grid tool by clicking on it.



3. Click near the left edge of the conference room, under the table, type **Conference Room**, then press <Enter> to finish the text entry.
4. Follow the same procedure to add the text **Work Room** to the lower-right room.
5. Now click in the center of the third room and type **Storage Room**, BUT DON'T PRESS ENTER YET!
6. Click in the Angle field of the Edit Bar (located just above the drawing) and enter **90**, then press <Enter>. Notice how your text has been rotated 90 degrees.



Fine Tuning the Text Position

1. Click the Select tool, then drag your text to the exact position where you want it..
2. Save your work. (This stage of the tutorial has been pre-saved as TUTOR105.TCW.)





Adding a Horizontal Dimension

Note: You can start at this point by loading TUTOR105.TCW from the TUTORIAL folder. If you do, immediately use File|Save As to rename the drawing so you don't overwrite the sample files with your changes!

The final step of this tutorial is to add dimensions to your drawing. These dimensions will display the lengths of some of the walls represented in the drawing. While using these tools, you'll make Vertex Snap the default snap mode, but switch to No Snap in the local menu for defining certain points.

1. Click on the Vertex Snap tool to activate Vertex Snap.



2. Select the Horizontal Dimension tool using the Insert toolbar.



3. Click on the outer-left corner of the lower wall, then click on the outer-right corner of the lower wall.
4. Right-click to open the local menu, click on Local Snap, then click on No Snap. This overrides Vertex snap until after you have defined the next point. (This is called a "one-shot" snap mode.)
5. Click outside the wall to finish the dimension.



Adding a Vertical Dimension

Let's repeat the previous exercise, but this time we'll dimension a vertical wall.

1. Choose the Vertical Dimension tool using the Insert toolbar flyout.



2. Click on the the outer-top corner of the right wall, then the outer-bottom corner.
3. Right-click to open the local menu, click on Local Snap, then click on No Snap. (Again, this temporarily overrides the snap mode you set in the Snap toolbar.)
4. Click outside the wall to finish the dimension.
5. Save your work. (This stage of the tutorial has been saved as TUTOR106.TCW.)





Conclusion

Congratulations, you have now completed your first TotalCAD drawing! For more exercises on TotalCAD node editing, dimensioning, and formatting tools, run the TotalCAD Features tutorials.

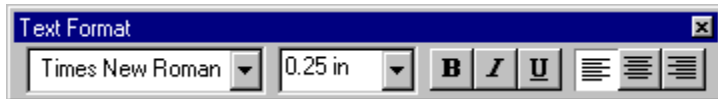


Formatting Objects

The next few topics will introduce you to the basics of how to set and change the appearance of objects, such as their layer, color, line style and brush style (fill patterns for closed objects). For this section of the tutorial, you will need the Property and Text Format toolbars visible on your screen. They look like the ones below, unless they are docked above of the drawing area, in which case the title disappears.



Property toolbar



Text Format toolbar

If you don't see them on your screen, choose the View pulldown menu, then choose Toolbars. When the dialog opens, verify that Property and Text Format have check marks next to them. If they do not, click on them, then click OK to close the dialog.





Object Properties

TotalCAD drawing entities share a common set of properties. (The exceptions to the rule are text entities, which have only general and text properties.)

Some entities, such as double lines, have additional properties specific to them alone. To view the properties of an object, select the object, then right-click your mouse to bring up the local menu, then click Properties.





General, Pen, and Brush Properties

Let's now select an object and examine its property sheets.

1. Choose File|Open from the menu, then select prop1.tcw from the Tutorial folder.
2. Choose the Select tool, then click on the filled circle to select it.



3. Right-click to bring up the local menu, and choose Properties. Notice the three tabs on the Properties sheet: General, Pen, and Brush. These are the properties common to all objects except text, bitmaps, and OLE objects.



Format the Circle

Let's now explore the properties in these sheets and change some of the properties.

1. Click on the General tab and notice that it shows you the layer the object is on, and also allows you to enter attributes (text information) about the object. If the object is a named object, such as a group, this field will contain the name. If the object is a text object, this field will contain the text, which can be edited here.
2. Click on the Pen tab, then select a different color for the circle by clicking on the down-facing arrow next to the color. Select Red.
3. Click on the Brush tab, and in the Color box select By Pen. By Pen means that the fill color will match the pen color. Make sure that the solid black rectangle is selected in the Pattern box. Click below to compare our dialog to yours.



Click OK., and when the dialog closes, click away from the circle to deselect it. It's red!



Format the Double Line

Now change a property of the double line.

1. Select the double line, then right-click your mouse and choose Properties. Notice that this Property sheet has a new property sheet called Double Line. Click on the Double line tab.
2. Change the Separation to **1"**, click OK to close the dialog, then deselect the double line by clicking away from it.

Notice that the distance between the two lines composing the double line has changed.

Hint: If you need to refresh your screen, press the F5 key.





Changing Default Properties

With no object selected, you can right-click the buttons on the Insert Entity toolbar and change properties for any future objects of that kind that you draw.



Let's try this now with the Text tool.

1. Right-click the Text tool. The Properties dialog will appear.



2. Click on the Text tab. Change the font to Arial, the height to 1 in, and the color to blue. Click OK to close the dialog.
3. Left-click on the Text tool to activate it, then click below and to the left of the red circle on your screen. The blinking cursor is the upper left hand starting point of your text. Notice that the command line at the lower left corner of your screen says "Type text, click to finish or use local menu".

Type **Text Properties**, then press <Enter>. Until you change the property settings again, all text will be 1 inch tall blue Arial.





Property Toolbar

In the next exercise you'll be using the Property Toolbar. Click on the controls on the following picture of the Property Toolbar to find out what they do.



Note: In TotalCAD 1.0 you cannot apply a style to a selection. The Style pulldown box is accessible only when you are setting the default properties of a drawing tool.



Open the Sample Drawing

Open a drawing in the Tutorial folder called prop-tb.tcw. In this drawing you have a closed multiline (i.e. an irregular polygon), a spline curve, and a cyan-filled circle with a black pen outline.

Use the Save As command to save this drawing under a different name to avoid overwriting the sample drawing.



Circle Properties

1. Select the cyan circle. Look at the Property toolbar. No style was chosen as a default circle property, so this field is grayed out on your screen. The circle resides on layer 0. The pen color is black. (The outline of the circle.) The pen style is a continuous line. The brush style is solid color. (The fill color is not displayed on the Property toolbar.)
2. On the Property toolbar, click on the Brush Style down arrow, scroll down about half way, select the little rows of crosses, then click away from the circle.
3. Select the circle again. On the Property toolbar click on the Pen Color down arrow and choose Red. Then click away from the circle. Click the Show Me icon below and compare your drawing to the example.





Curve and Polygon Properties

1. Select the spline curve. Click on the Pen Style down arrow on the Property toolbar and select the dashed line just above the words "By Layer". Then click on the Pen Color down arrow and select Brown.
2. Select the closed multiline, choose the down arrow next to Brush Pattern on the toolbar, then select the solid black pattern at the top of the menu. Click away from the polygon.

Now click the Show Me icon and see if your changes match ours.

Hint: If you need to refresh your screen, press the F5 key.





Layer Property

The layer property of an object determines the layer of the drawing on which the object resides. Let's start by adding a new layer to your drawing, then we'll move an object to that layer by resetting its Layer property.

1. Click on the Tools pulldown menu and select Layer. The Layers property sheet appears.
2. Notice that the cursor is blinking in the Name box. Type **polygon** and click on New. In the Color box, click on the down arrow and choose Magenta. Click on the Show Me button to see what the dialog should now look like.



3. Click OK to close the dialog and return to your drawing.



Color by Layer

You have just created a new layer called **polygon**, which has a default color of magenta. Now lets create an object on this layer and color it the default color of the layer.

1. Select the irregular black polygon, click on the down arrow for Layer in the Property toolbar, and choose polygon. Your irregular polygon is now on layer **polygon**, but it is still black.
2. Click on the Color down arrow of the polygon and choose By Layer. The pen color of the polygon is now magenta, matching the default color of the polygon layer. Click away from the polygon to see the results.



Add New Objects

Now we'll set the color of the line tools to By Layer, so that all objects drawn with these tools will match the layer color.

1. Click the Single Line tool to activate it. You are now going to change the default properties of this tool. These changes will affect all of the tools on the Single Line flyout.
2. In the Color field of the Properties toolbar choose By Layer. In the Layer field, choose **polygon**.
3. Select the Rectangle tool.



4. Draw a rectangle at the bottom center of your drawing. Because you globally changed the color of the Single Line tools to By Layer and their layer to Polygon, the objects that you draw with these tools will have the default color of the **polygon** layer (magenta).
5. Choose the Polygon tool from the Single Line flyout.



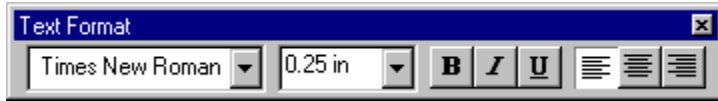
6. Select Layer 0 on the Property toolbar. Now draw a polygon to the right of your rectangle. The polygon will be black, the default color of Layer 0. Click the Show Me icon below to compare your drawing to ours.





Formatting Text

If the Text Format toolbar is not displaying on your screen, display it now using the View|Toolbars command. Click on the parts of the image below to find out what they do.



Open a drawing in your Tutorial folder called text1.tcw. Click on the Show Me icon below to see what your screen should look like.





Changing Text Properties

1. Select the top sentence on the page. Click on the font box on the Text Format toolbar where it says Times New Roman, then scroll up and click on Arial.
2. Select the second text object. Click on the Text Size box on the Text Format toolbar and either type **.25** or select 0.25 in on the drop-down list.
3. Notice that the height of the text changes, but it remains the same length. Click in the Xscale field of the Edit Bar and change it to **.5**, then press <Enter>. This adjusts the width of the text so that it is proportional to the height.
4. Select the third text object from the top of the drawing, then click the Center Justify button on the Text Format toolbar.
5. Use the text format toolbar to make the next text object bold, and the bottom text object underlined and italic.
6. Select the bottom text, right-click to pick Properties from the local menu, pick the Text tab, then select Red from the Color dropdown box. Click OK to close the Properties dialog.

Click on the Show Me icon to compare your drawing.





Conclusion

This concludes the tutorial on formatting. For more detailed information, search for "format" or "property".



Snap Modes and Coordinate Systems

Snap modes are one of the most useful features of TotalCAD. If you completed the TotalCAD Basics tutorial, you have used two snap modes, Snap to Grid and Snap to Vertex. Snap modes are the intelligent drawing tools inside TotalCAD that force your mouse click to a specific position. This tutorial is designed to acquaint you with all the snap modes so that you will be able to use them in your own drawings.

To prepare for this tutorial, close all open drawings, then load the TUTOR2.TCW drawing found in the TUTORIAL folder.

Use the File|Save As command to save the drawing as MYTUTOR2.TCW so that you don't overwrite the sample drawing.



Snap Modes

In the following exercises, you will use the single line drawing tool. You can activate this tool by selecting the Insert|Line command or selecting the single line tool from the Insert Toolbar.



As you work through this tutorial, you will be instructed to draw lines between numbered points on the sample drawing. If TotalCAD beeps at you, then you have not clicked close enough to the cross marking the point.



No Snap

This snap mode is really the absence of any snap. When using this mode, your mouse click will stay exactly where you click.





Ortho Mode

Ortho mode constrains the angle of lines. In the default TotalCAD setup, Ortho constrains angles to 0 degrees and 90 degrees (that is, the lines are *orthogonal*). You can customize Ortho if you like using the Tools|Angle property sheet.



Select Ortho mode now, and begin to draw a line below the rectangle on the sample drawing (only click once!). Move your cursor to the right and see how the line remains horizontal. Move the cursor above the first point you defined and see how the line jumps to vertical. Complete the line at any time with a second click.



Vertex Snap

Vertex snap mode forces your mouse click to a defining point of an existing object (the end of a line, etc.).



Make sure only Vertex snap is selected, then draw a line from point 1 to point 2 on the sample drawing. Click on the Show Me icon to see what your drawing should look like.





Middle Point Snap

Middle Point snap forces your mouse clicks to the center of objects like line segments.



To see the use of this command, make sure only Middle Point snap is selected, then draw a line from point 3 to point 4 on the sample drawing. Click on the Show Me icon to see what your drawing should now look like.





Intersection

Intersection snap is used to force your mouse clicks to the intersection of 2 objects.



The most obvious example is the intersection of 2 lines, but TotalCAD also allows you to snap to the intersection of 2 circles, a circle and a line, or other types of intersections. To see the use of this command, make sure only Intersection snap is selected, then draw a line from point 5 to point 6 on the sample drawing. Click on the Show Me icon to see what your drawing should now look like.





Arc Center and Quadrant Snaps

For this example, you will use 2 different snap modes to draw a line from the center of the circle in the sample drawing to a point on the circumference. Arc Center snap is used to snap to the center of the circle, but you must remember that your mouse cursor must be close to the edge of the circle you want to find the center of. Quadrant snap mode forces your mouse cursor to snap to the circumference of a circle at specific increments of degrees (0, 90, 180, and 270 by default). Let's try the example now:

1. Select No Snap to turn off all other snap modes.
2. Select Arc Center Snap.



3. Click near point 7 on the sample drawing to snap to the center of the circle.
4. Select Quadrant Snap.



5. Click near point 8 on the sample drawing to finish the line.

Click on the Show Me icon to see what your drawing should now look like.





Grid

Grid snap forces your mouse clicks to snap to the nearest grid point. Grid snap was covered in an earlier tutorial and is not demonstrated again here. It is important to note, however, that grid snap works even if the grid is not visible.



Nearest Entity

Nearest Entity snap to forces your mouse clicks to the nearest object inside the snap aperture. To see the use of this command, make sure only Nearest Entity snap is selected, then draw a line from point 9 to point 10 on the sample drawing. Click on the Show Me icon to see what your drawing should now look like.





Local Menu Snaps

Many times you will want to use a particular snap mode only once, then revert to the snap modes set previously. TotalCAD provides local menu (right-click) snaps, also called "one shot snaps." Try drawing a line using any of the snaps previous discussed for the first point, then right-click the mouse and select a new snap mode from the local menu.



Coordinate Systems

Another method of defining points in TotalCAD is to directly enter their coordinates (the locations on the drawing). Coordinate systems determine the method TotalCAD uses to evaluate your coordinate entries.

Coordinates can be entered in the Coordinate Fields (in the lower-right area of the screen) any time that TotalCAD is waiting for you to click the mouse. To enter coordinates, click the mouse in the X or Y fields and enter the value you want (these fields normally show you where the mouse cursor is currently located in your drawing). You can lock the value by clicking the small square next to the edit field, although this is optional. After entering one value, use the Tab key to move to the next field, or press <Enter> if both values are acceptable.

You can also enter the Coordinate Fields by pressing <Ctrl>+<R>.



Absolute Coordinate Mode

Absolute Coordinate mode is the default mode. Absolute the coordinates are based on a horizontal (X) or vertical (Y) distance from the absolute origin of the drawing. Look at the rulers to see where the origin is located on the sample drawing. X values increase to the right, and Y values increase toward the top. Make sure Absolute Coordinate mode and No Snap are selected, then try typing some coordinate values in the coordinate fields to draw some lines. For each point in the line, define a point in the Coordinate Fields, and press <Enter>.



Relative Coordinate Mode

Relative Coordinate mode is used to define points relative to another point called the *Relative Origin*. To see the use of relative mode, you will draw a 6 inch line as follows:

1. Make sure Absolute Coordinate mode and No Snap are selected.
2. Start a line at coordinate 1,1 by typing these values in the Coordinate Fields and then pressing <Enter>.
3. Click on Relative Coordinates tool to change to Relative Coordinate mode. Note the red square marking the Relative Origin, which is the position from which the relative coordinates are measured.



4. Make a horizontal line 6 inches long by entering **6** and **0** in the coordinate fields followed by Enter. Note that the relative origin is now at the end of the new line. Click on the Show Me icon to see what your drawing should now look like.





Polar Coordinate Mode

Polar Coordinate mode is used to define points at a specified distance and angle from the Relative Origin. To see the use of Polar mode, you will draw a line 1 inch long and 45 degrees up from the line you drew in the previous example:

1. Make sure Absolute Coordinate mode and Vertex Snap are selected
2. Start a line at the left end of the line drawn in the previous example
3. Change to Polar Coordinate mode and No Snap. Note the red square marking the relative origin, which is the position from which the relative coordinates are measured.



4. Make a slanted line 1 inch long by entering, type **1** and **45** into the Dist and Ang Coordinate Fields, then press <Enter>. Note how the polar origin is now at the end of the new line.

Click on the Show Me icon to see what your drawing should now look like.





Summary

This tutorial has presented an overview of snap modes and coordinate systems. Remember, snap modes can be used in combinations and provide a powerful method to make precision drawings.



Dimensions

TotalCAD provides a variety of dimension tools that you can use to display the measurements of lines and angles. To prepare for this lesson:

1. Close all open drawings.
2. Open the TUTOR3.TCW drawing located in the TUTORIAL folder. Notice that there are five separate areas on the drawing, each labelled with the name of an exercise.
3. Immediately use File|Save As to save the drawing as MYTUTOR3.TCW. This will prevent you from accidentally overwriting the sample files.



Using the Zoom Window

During the dimension exercises, you will need to zoom in to view a particular area of the drawing. The best way to do this is to use the Zoom Window tool, located on the View toolbar.



To zoom into an area, click the Zoom Window tool, then drag the mouse diagonally across the area that you want to zoom to. Try to leave enough space in the zoom rectangle so that you can add dimensions. If you need to zoom out, press the minus key on the numeric keypad, or click the Zoom Out tool.



To return to a view of the entire drawing, press <Shift>+<Backspace>, or click the Full View tool.





Basic Dimensions

Referring to the TUTOR3 drawing, you will first draw some simple dimensions on the shape marked Basic Dimensions. Use the Zoom Window tool to zoom into this area of the drawing, leaving enough space around the rectangle to add dimensions.

1. Turn off all snap modes by clicking on the No Snap tool.



2. Choose Insert|Dimension|Horizontal from the menu, or choose this tool using the Dimension flyout on the Insert toolbar.



3. Right-click on the drawing to open the local menu, then select Segment Dimensioning. Segment Dimensioning means that TotalCAD will automatically dimension a line segment that you select.
4. Click on line #1 in the Basic Dimensions shape. TotalCAD automatically selects the line and displays a dimension.
5. Move your mouse down from the line until the dimension is positioned at approximately 1 grid line below line #1, then click again to lock the dimension into position.

You just created a simple horizontal dimension, using Segment Dimensioning.



More Dimensions...

Now you can add some additional dimensions.

1. Choose the Vertical Dimension tool by holding the mouse button down over the Horizontal dimension tool and selecting the Vertical Dimension tool from the flyout.



2. Click on line #2 and then position the dimension just to the left of the line.
3. Choose the Parallel Dimension tool using the Insert toolbar flyout.



4. Click near line #3 and position the dimension just to the right of the line

Congratulations! You've now added the basic dimensions to the drawing!





Angular Dimensions

Angular dimensions draw a dimension showing the angle formed by two lines, by points on the perimeter of a circle or arc, or the angle formed by any two points relative to a defined vertex. The simplest case is dimensioning the angle between 2 lines, which this example will demonstrate.

To get a good view of the Angular Dimensions area of your drawing, first press <Shift>+<Backspace> to view the entire drawing, then use the Zoom Window tool to zoom into the area labelled Angular Dimension.

1. Select the Angular Dimension tool.



2. Select No Snap mode.



3. Click near the right end of line #1 in the Angular Dimension set of lines in your drawing.
4. Click near the right end of line #2. If you now move the mouse cursor between the two lines, TotalCAD forms a dimension showing the inner angle. If you move the mouse cursor outside the two lines, a dimension will be formed showing the outer angle.
5. Click to specify the position of the dimension arc (near point #3).
6. Click to specify the position of the dimension text (near point #4).

Tip: If you had clicked near the left end of the lines instead of the right ends, you would have dimensioned the exterior rather than the interior angle formed by the lines.

Click on the Show Me icon to see what your drawing should look like now.





Diameter and Radius Dimensions

Two more simple dimension types demonstrate the ease of use of TotalCAD dimensions. Radius and Diameter dimensions measure the radius or diameter of arcs and circles.

To get a good view of the Diameter and Radius Dimensions area of your drawing, first press <Shift>+<Backspace> to view the entire drawing, then use the Zoom Window tool to zoom into the area labelled Diameter and Radius Dimensions. Make sure to leave room to maneuver.

1. Choose the Radius Dimension tool.



2. Switch on Ortho mode.



3. Click near point #1 on the circle titled "Diameter and Radius Dimensions."
4. Click near point #2 to finish the radius dimension.
5. Select the Diameter Dimension tool.



6. Click near point #3 on the circle.
7. Click near point #4 to finish the diameter dimension.
8. Click on Ortho again to switch it off.

Click on the Show Me icon to see what your drawing should look like now.





Leader Dimensions

Leader dimensions don't generally measure anything: They are used to label items on your drawing. Let's try marking the circle from the last example:

1. Choose the Leader Dimension tool.



2. Press <Tab> to edit the Text field in the Edit Bar and change it to read "Circle", then press <Enter>.
3. Click on the lower-left side of the circle in your drawing.
4. Move your mouse slightly down and to the left, then click again.
5. Right-click to open the local menu, then select Finish to complete the leader dimension.

The final dimension tutorial drawing has been saved as TUTOR301.TCW. Click on the Show Me icon to see what your drawing should look like now.





Dimension Options

TotalCAD includes a tremendous number of dimension options to suit almost every dimensioning need imaginable. These options are available by either:

1. Right-clicking on one of the dimension tools to set the defaults for any dimensions that are subsequently drawn.
2. Selecting an existing dimension and then Format|Properties to change the settings of an existing dimension.

The best way to learn about dimension options is to experiment. Try selecting dimensions in the dimension tutorial drawing you just created and changing the properties to see the effects. Refer to your TotalCAD help system or to the Complete Guide to Learning and Using CAD for further explanation of dimension properties.



Node Edit

In Node Edit mode you can reshape objects by directly manipulating points called *nodes*. Open a drawing called node-ed1.tcw which you will find in your TotalCAD Tutorial folder. This drawing includes a single line, circle, filled irregular polygon (a triangular closed multiline), a spline curve and a double line. Click on the Show Me icon to verify that you have the correct drawing.



Use File|Save as to save the file under a new name. This will prevent you from overwriting the sample file.



Starting Node Edit Mode

To start Node Edit mode, you first need to select an object, then use the local menu to change modes. Try it with the single line.

1. Click on the Select tool, then click on the single line to select it.
2. Right-click to open the menu, then choose Node Edit.

Notice that small square boxes show at the ends of the single line.



Adding a Node

Let's add a node to the line:

1. Turn on Snap to Grid.



2. Put your selection cursor on the line near its center, right-click your mouse, then choose Add Node from the local menu. A new node appears near the center of the line.
3. Place your cursor over the node, and it becomes a four-way arrow. Click at this point, then click at a position two grid points down to move the node. Deselect the line by clicking away from it. (If a ghost of the line remains on the screen, simply press the F5 key to refresh the screen.)
4. Now select the spline curve, again right-click to bring up the local menu and choose Node Edit. Notice that this curved line has nine nodes.



Deleting and Moving a Node

Let's continue to edit the spline curve:

1. Put your cursor over the upper left end node, and access the local menu again. Select Delete Node.
2. Use snap to grid to move the node just to the right of the new end to a location two inches below where it is now. The grid points are half an inch apart.
3. Click the Show Me icon and see if your drawing is close to ours.





Edit the Circle

Now select the circle, and bring up the local menu again, choosing Node Edit. Notice that instead of just blue squares, there are also little green and red squares. If you move the small blue square, you will change the diameter of the circle. The other two squares control the ends of the circle. (Think of it as a 360 degree arc with connected ends).

1. Move the blue square in 1/2 inch.
2. Click on No Snap to turn off all snap modes.



3. Thinking of the circle as a clock, move the little green square up to approximately two o'clock.
4. Move the little red square down to five o'clock.
5. Deselect the circle by clicking away, and if necessary refresh your screen by pressing <F5>.
6. Click the Show Me icon and compare drawings.





Editing a Polygon

1. Select the triangle. (It was made with the multiline tool.)
2. Right-click to bring up the local menu, and choose Node Edit. Three nodes appear on the polygon.
3. Put your cursor on the top node so that it turns into a four-way arrow, click, and drag it up and to the right about two inches.
4. Put your cursor approximately between the two lower nodes on the bottom edge of the polygon. Right-click to access the local menu, and choose Add Node. A new node appears on the bottom of the polygon.
5. Drag this node down about .5 inch, and click away from the polygon.
6. Click the Show Me icon and check your drawing with ours.





Edit the Double Line

1. Select the double line, right-click to bring up the local menu, and choose Node Edit. Six nodes appear on the double line.
2. Put your cursor over the bottom center of the double line. Hold down your <Control> key and notice that the cursor becomes a plus sign, with a node and an arrow. This is a quick key command to add a node without bringing up the local menu. Left-click to place a node.
3. Drag this new node up about 1 inch.
4. Now place your cursor over the left end node. Hold down both the <Control> and <Shift> keys at the same time. The cursor becomes a gray node with an arrow to a waste basket. Left-click to remove that node. (If necessary refresh your screen by pressing the <F5> key.)
5. Click on the Show Me Icon and compare your work in progress.





Closing a Multiline into a Polygon

Open a drawing called node-ed2.tcw, which you will find in your Tutorial folder. This drawing contains a single multiline, a double multiline, and a spline curve. You can close any of these open figures into a closed figure so that you can fill it with a color or pattern.

Note: If you simply move the end nodes so they are on top of one another, it may look exactly like a polygon, but to TotalCAD it is still a multiline and so it will not fill properly.

1. Select the double multiline, right-click to bring up the local menu, then choose Node Edit.
2. Place your cursor over either end node of the double line, so that the four-way arrow appears.
3. Right-click to bring up the local menu, and choose Close.
4. Click away from the object. Notice that it closes the polygon and cleans up the double line intersections as well.





Closing Another Multiline

1. Now select the red multiline.
2. Right-click to bring up the local menu and choose Node Edit.
3. Place your cursor over either end node of the red multiline so you can see the four-way arrow, right-click, then choose Close.



Adding a Fill to the Polygon

1. Right-click again and choose Select Edit. The handle-based editing box appears again.
2. Right-click and select Properties from the local menu. Click on the Brush tab, and in the Pattern box, click on the filled black rectangle. Click OK to close the dialog.
3. Click away from the filled polygon and, if necessary, refresh your screen with the F5 key.

Check your drawing by clicking on the Show Me icon.





Filling a Wall

1. Select the double line polygon again.
2. Right-click and choose Properties. Click on the Brush tab.
3. In the Pattern box, select the L45 diagonal line pattern just below the solid black.
4. In the Hatch box, change the scale to **.1**, then press <Enter> to close the dialog.
5. Click away from the object and notice how smart TotalCAD is! With a double line, the program assumes that you want to show a fill pattern like insulation, for walls.





Close the Spline Curve

1. Select the spline curve.
2. Right-click and choose Node Edit.
3. Place your cursor over either end node, and right-click again. Choose Close.
4. If necessary, refresh your screen by pressing the <F5> key.



Add a Pattern

Right-click and choose Select Edit.

1. Right-click again and select Properties.
2. Click on the Brush tab, and in the Pattern box, scroll down slightly less than half way until you see a little diagonal line pattern called EARTH. Click on it.
3. Click OK to close the dialog and the closed curve fills with the EARTH pattern.





Breaking a Polygon

You can break a polygon down into multiline segments. Open a drawing called node-brk.tcw that you will find in your Tutorial folder.

1. Select the blue polygon.
2. Right-click to choose Node Edit from the local menu.
3. Place your cursor on the node on the upper-left point of the polygon, so that your cursor becomes a four-way arrow.
4. Right-click to access the local menu, and choose Break. The polygon appears the same, as though nothing happened.
5. Place your cursor over that same node, and move it down a couple of inches, and to the left a half inch.
6. If necessary, refresh your screen with the <F5> key. In fact, you did break the polygon, but the two nodes were resting directly on top of one another.

Click the Show Me icon to check your work in progress.





Dividing a Double Line

1. Now select the double line.
2. Right-click and choose Node Edit from the local menu.
3. Place your arrow cursor over the upper line between the nodes, right-click and select Segment Divide.
4. When the small dialog appears, enter 3 for Number of Parts, and click OK.
5. The upper segment of the double line is now composed of three equal segments.

Click the Show Me icon to verify your work.





Dividing arcs and circles

Open an drawing called divide.tcw which you will find in your Tutorial folder. Arcs and circles can be divided into equal arc segments just like a multiline.

1. Select the black innermost circle.
2. Right-click for the local menu and choose Node Edit.
3. Right-click again and select Arc Divide.
4. Enter 15 for the Number of Parts, and click OK to close the dialog.
5. Deselect the circle by clicking away. It appears as though nothing has happened.
6. Left-click at approximately 7 o'clock on that circle, and one of the fifteen segments will be selected.

Click the Show Me icon to verify your results.





Dividing an Arc

1. Delete the segment of the circle that is selected by pressing your delete key.
2. Select the magenta circle, the next circle moving outward. Right-click to bring up the local menu, and choose Node Edit.
3. Right-click again, choose Arc Divide from the local menu, then accept the default 2 by clicking OK to close the dialog.
4. Click away from the circle, then select the bottom half and delete it.
5. Then select the top half, right-click to access the local menu, and choose Node edit.
6. Drag each end down until it lines up with the ends of the smaller arc inside. With Node Edit mode still active, click on the Show Me icon and compare your work.



If you want more practice, divide the circles and manipulate the nodes of the other two circles or arcs that remain.



Dimension Node Editing

Dimensions have nodes that let you change the position of the dimension line, extension lines or dimension text. Open a drawing called dim-node.tcw which you will find in your Tutorial folder.

1. Select the 7.1 inch dimension. Right-click, and choose Node Edit from the local menu. Notice that four nodes appear.

Tip: Place the mouse cursor inside the selection and press the plus key on the numeric keypad to get a better view.

2. Left-click on the node at the right arrow, drag it up about one quarter of an inch, and release to place the dimension line.
3. Let's edit the dimension so that it only measures half of the line. Place your cursor on the left node at the corner of the filled double line and click. Right-click, then choose Local Snap|Middle Point from the local menu. Click near the middle point of the horizontal double line to move the node. Notice that the dimension text changes to reflect the new measurement.
4. Place your cursor on the node on the dimension text. Drag the four-way arrow into the middle of that dimension line. Release to place the text, and notice that the line splits apart to make room for the text. Click away to see your work. Select the Show Me icon below to compare your work in progress.





Editing an Angular Dimension

1. Select the circle dimension by clicking on the arrows or the text. Right-click to choose Node Edit from the local menu.
2. Click on the node that is on the extension line outside of the circle, and drag it up to approximately 11 o'clock.
3. Click the node on the text and move the text inside the circle. Right-click and choose Select Edit.
4. See the line that remains outside the circle? Right-click, and choose Properties. Select the Format tab. At the bottom of the dialog, select both Force Text Horizontal and Force Dimension Text Inside Extension Lines. Click the Show Me icon below to make sure your settings agree with ours.



5. Click OK to close the dialog. Click the Show Me icon to compare your results.

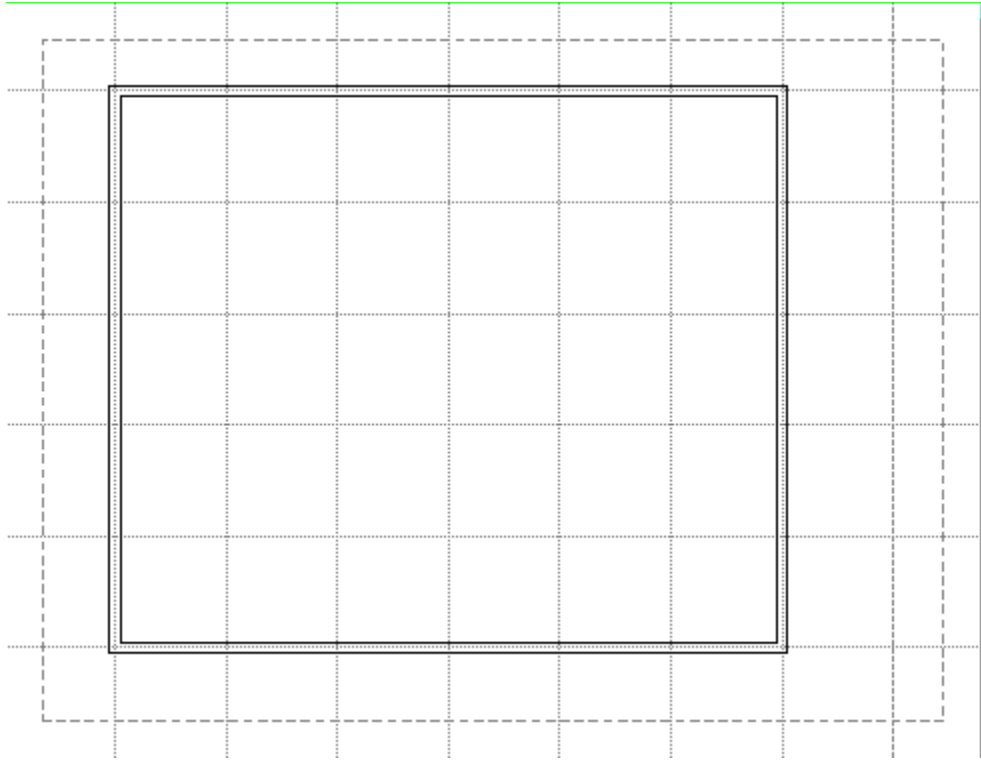


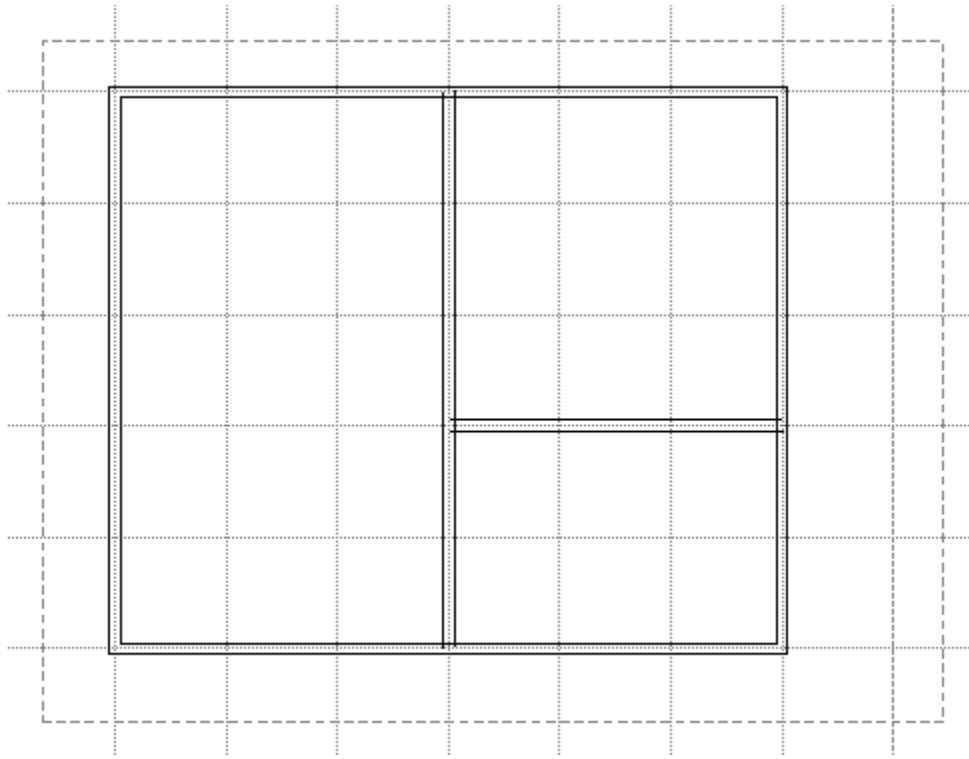


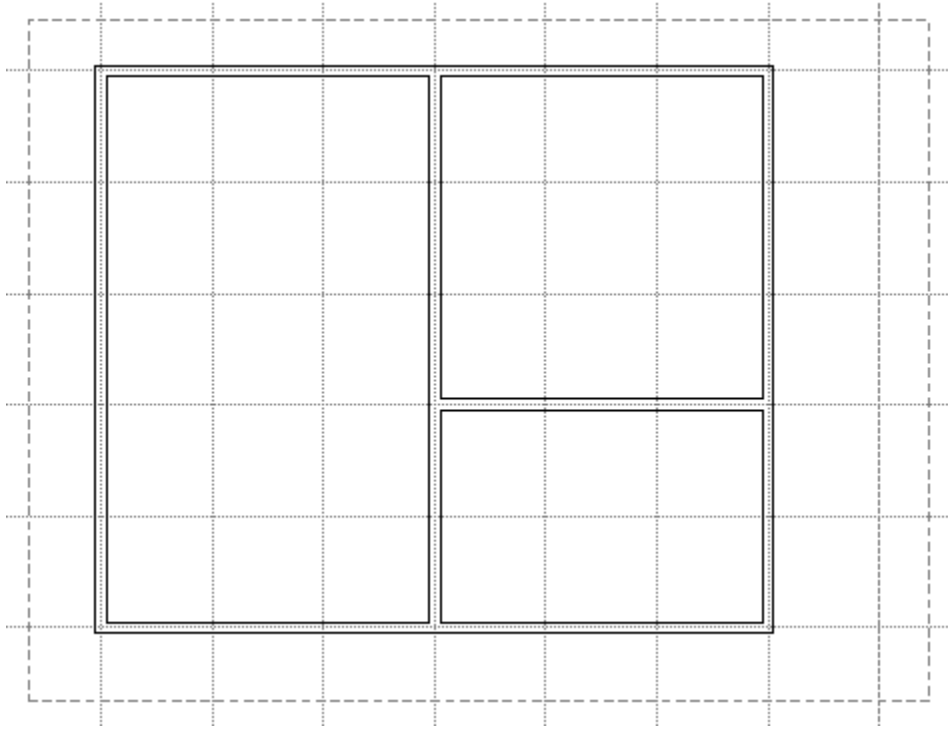
Conclusion

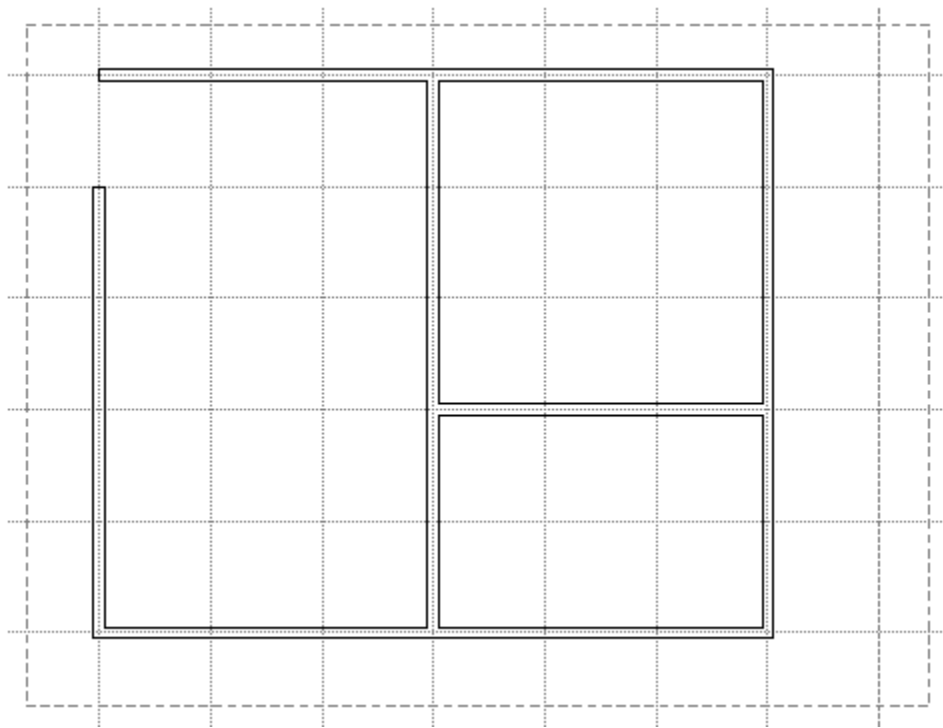
This concludes the tutorial on Node Edit mode.

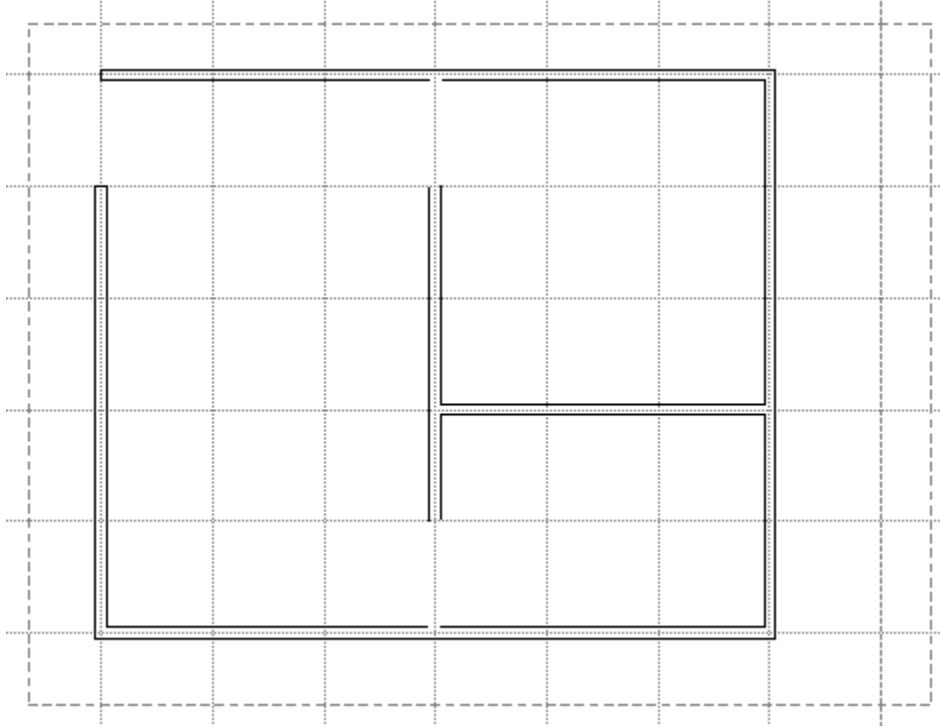


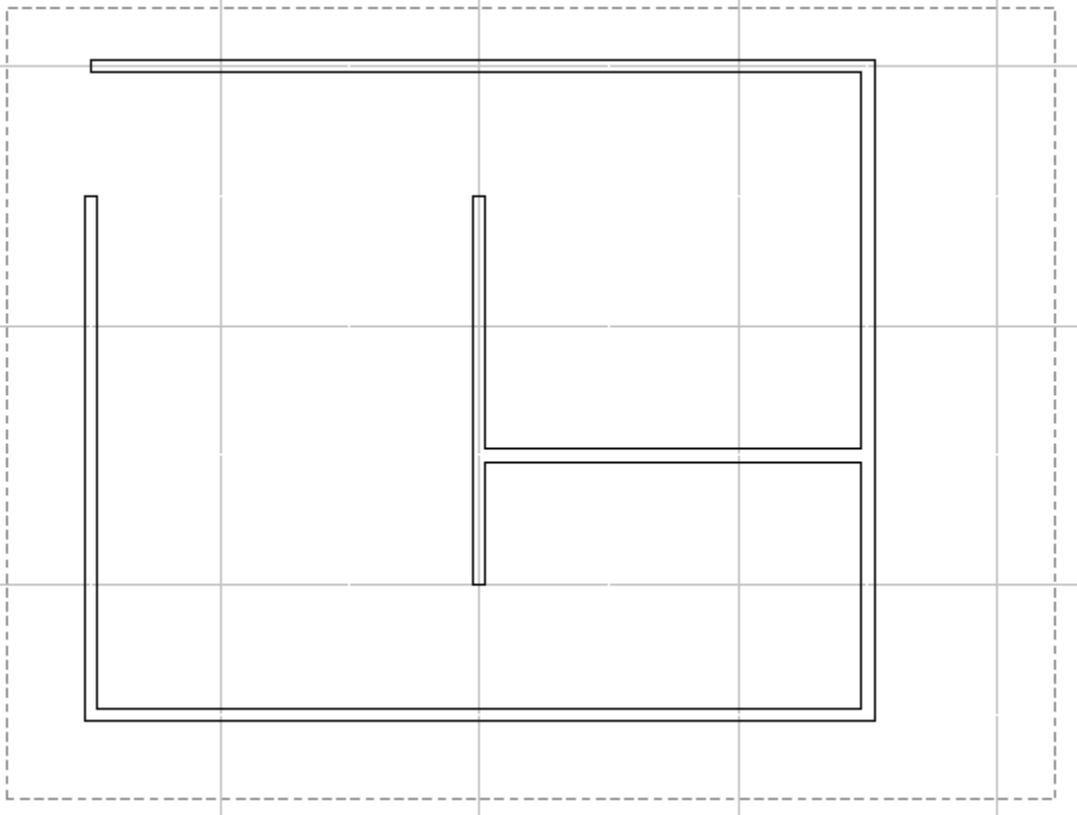


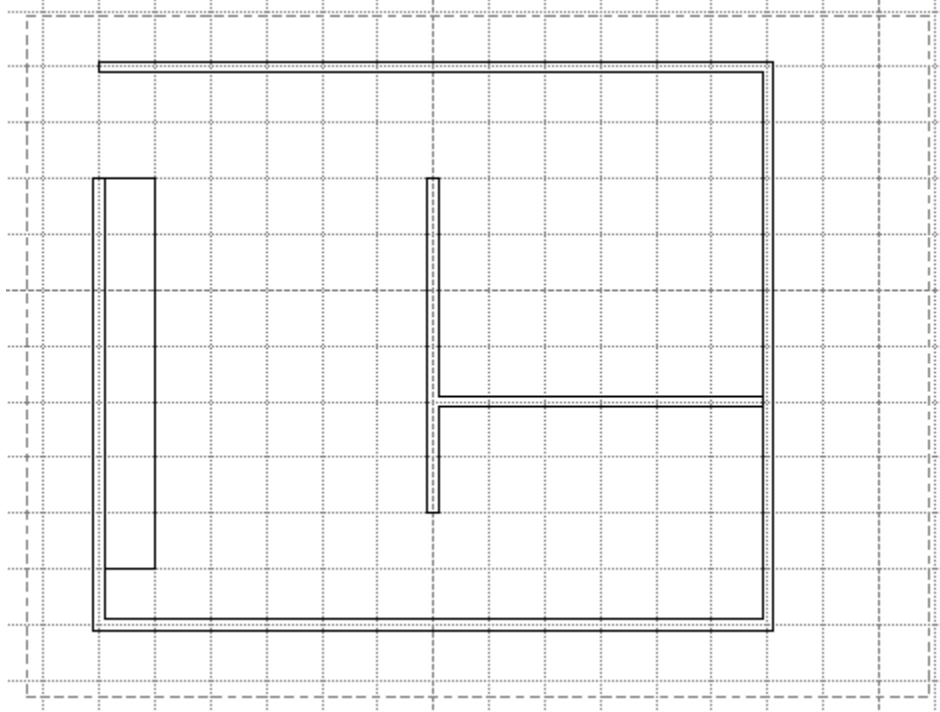


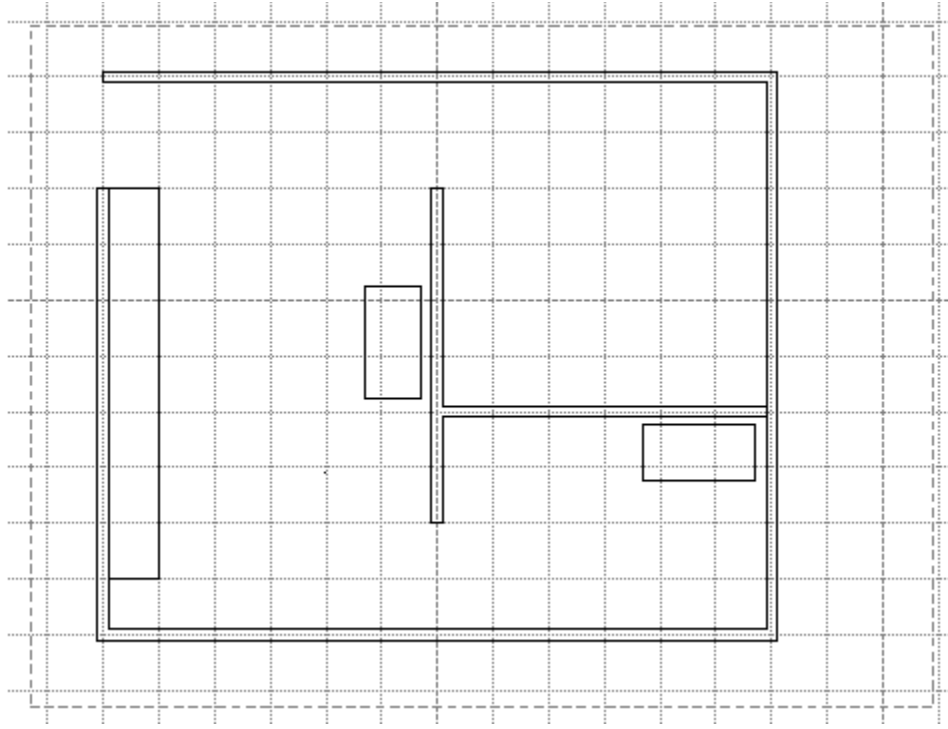


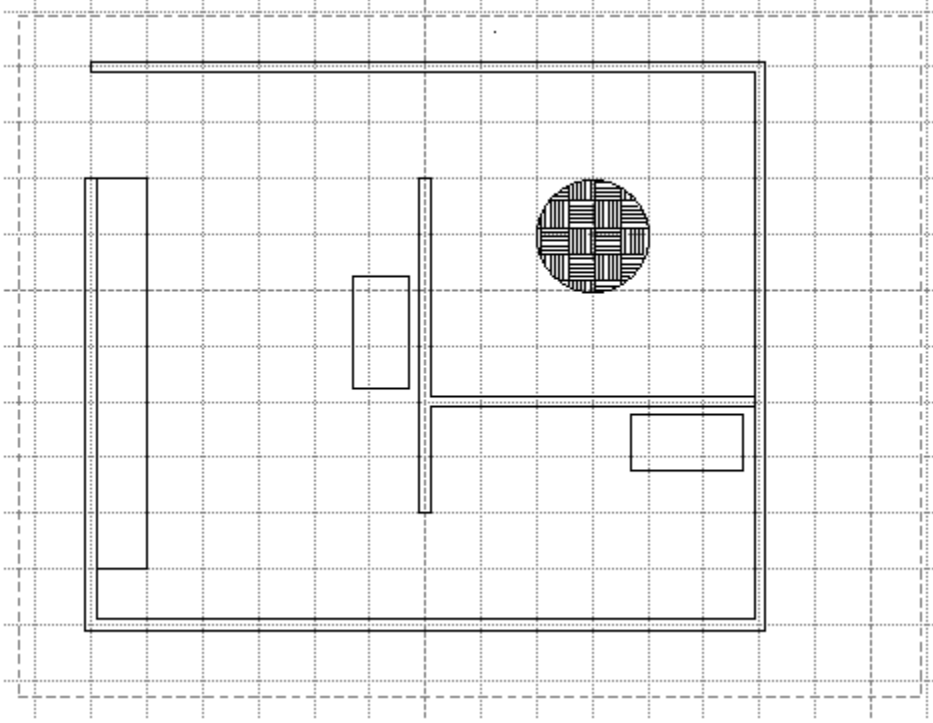


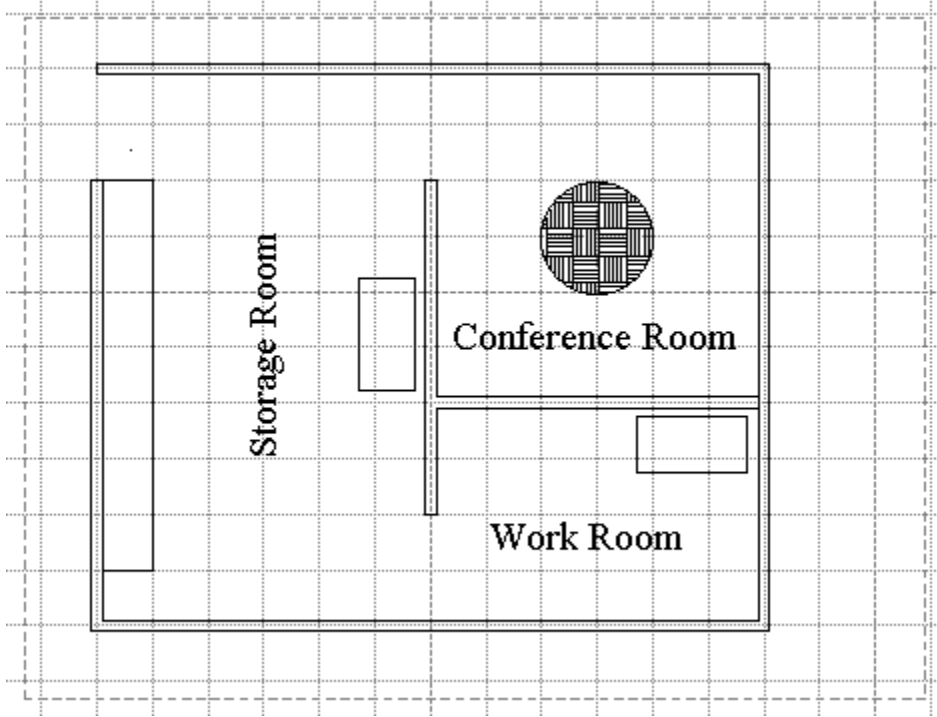


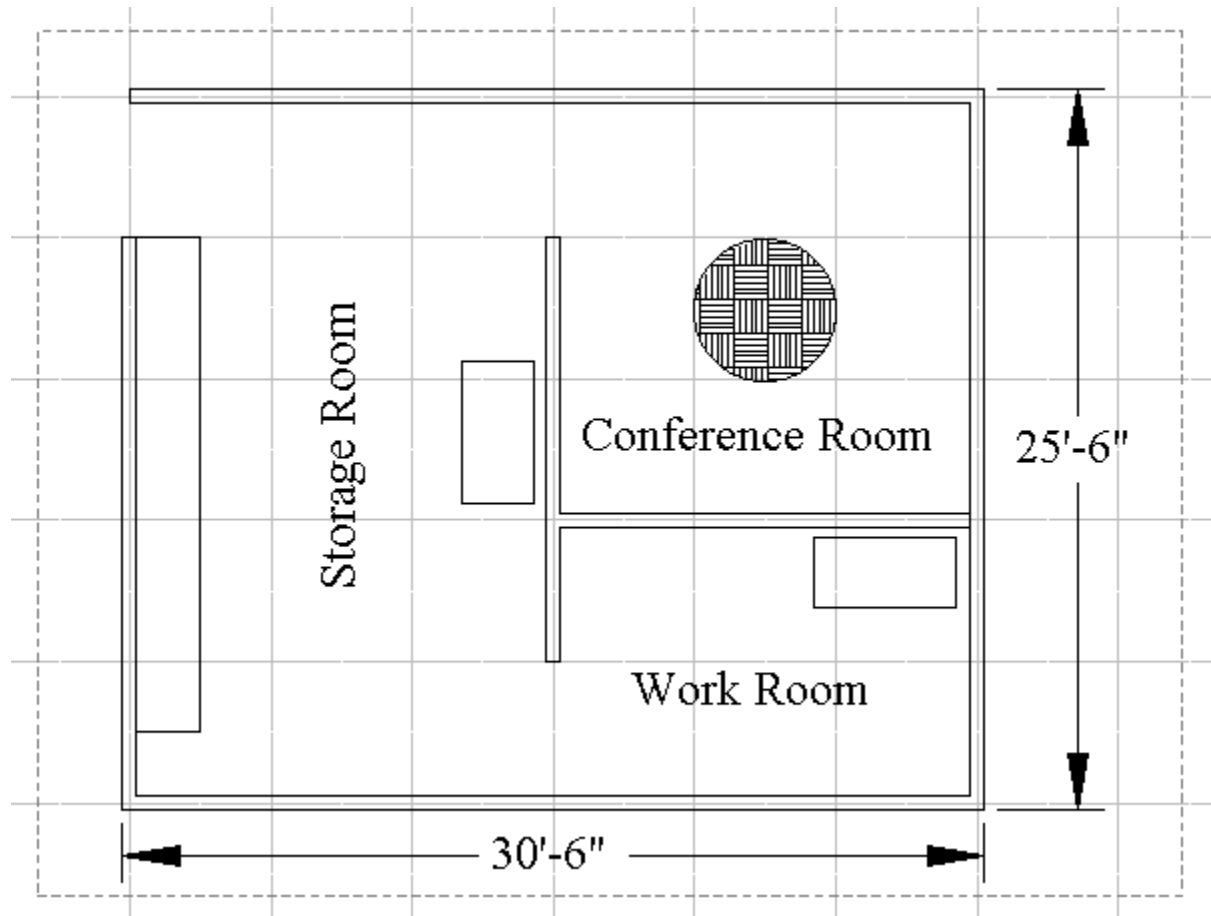




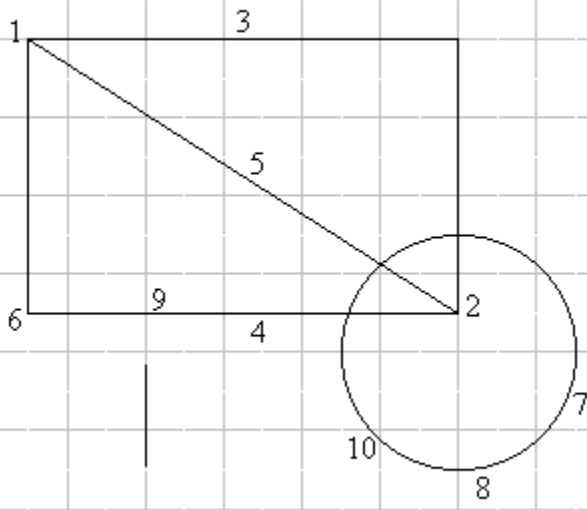




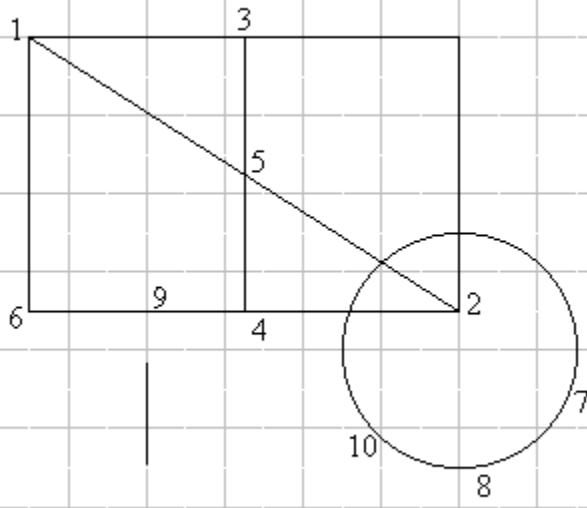




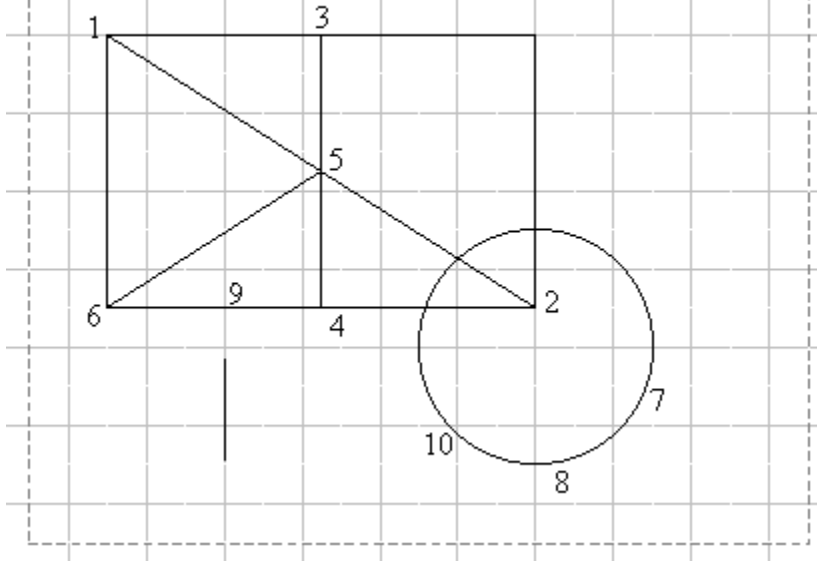
Snap Modes Tutorial



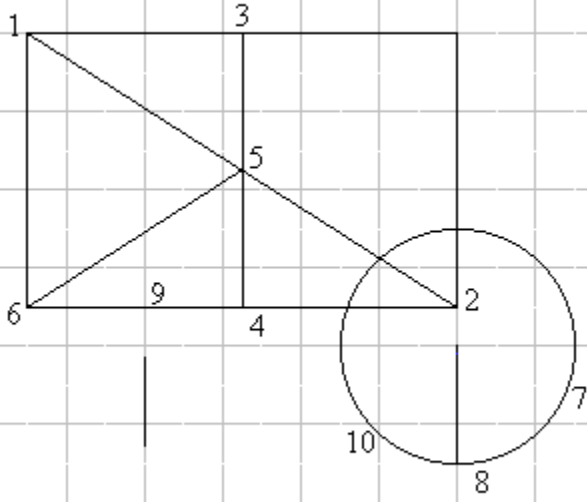
Snap Modes Tutorial



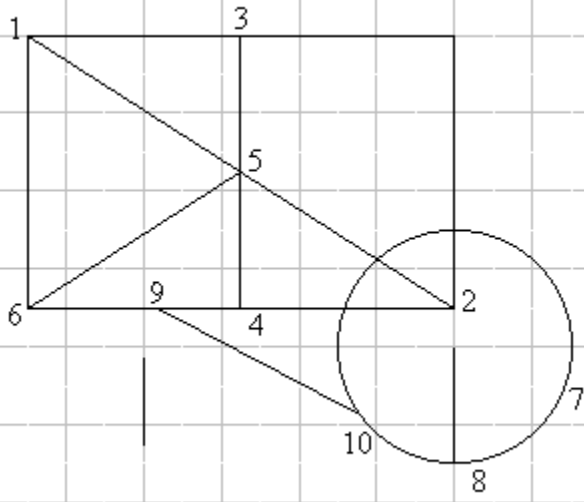
Snap Modes Tutorial



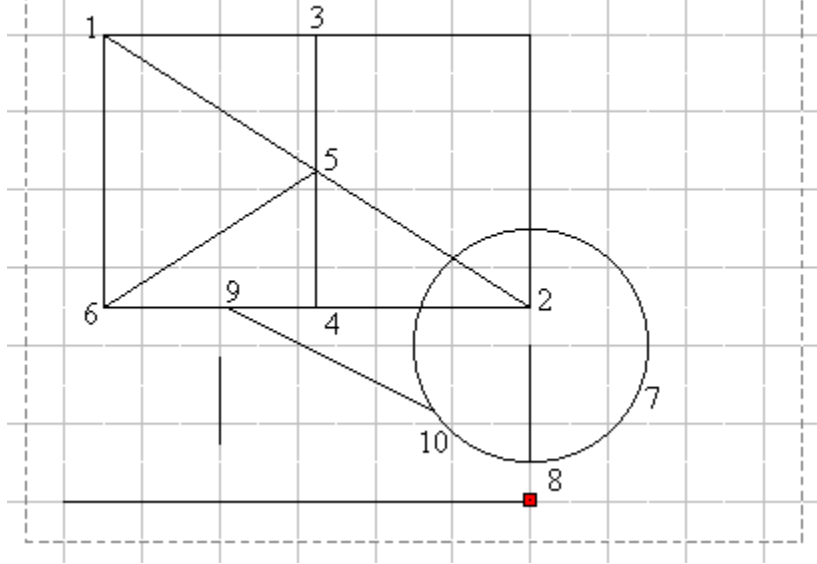
Snap Modes Tutorial



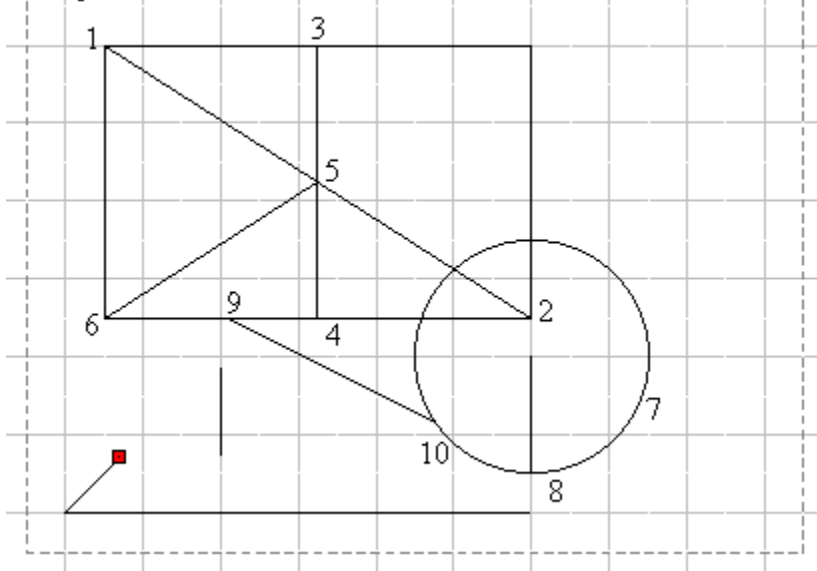
Snap Modes Tutorial

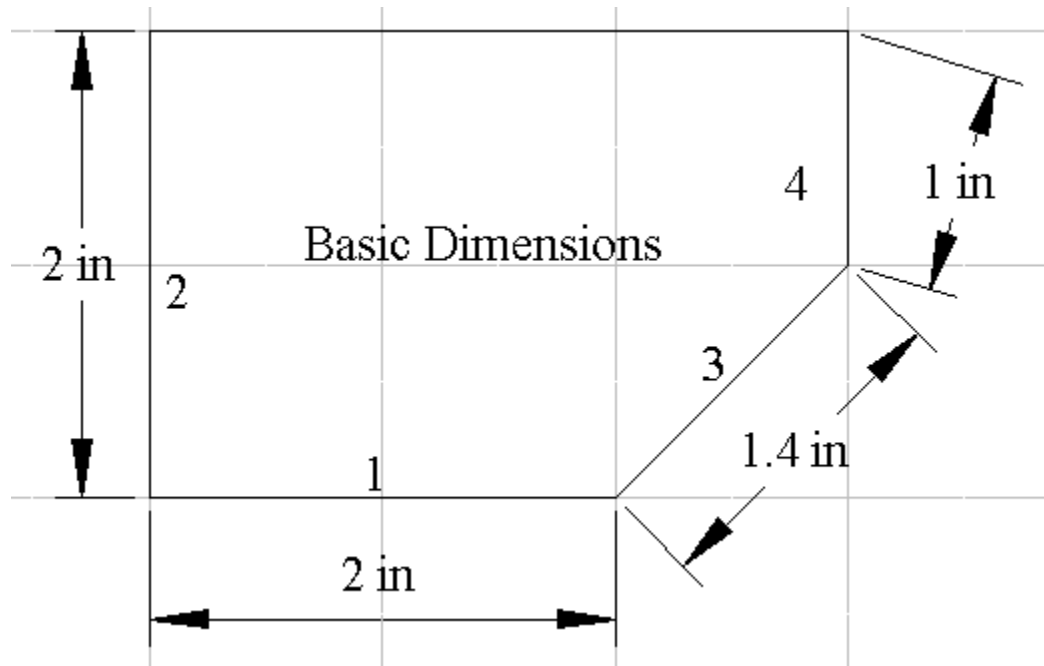


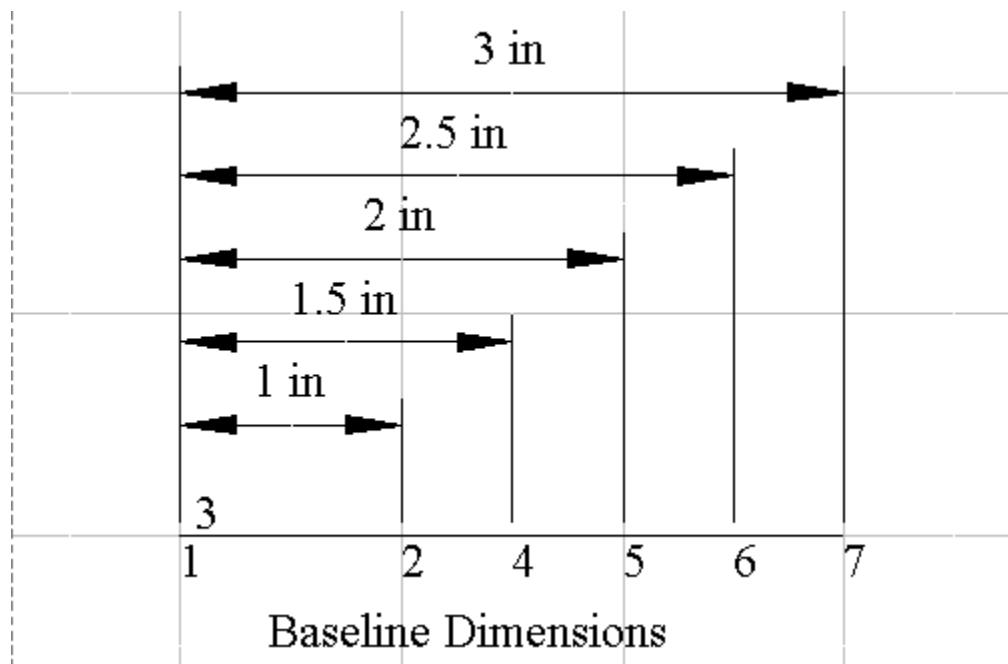
Snap Modes Tutorial

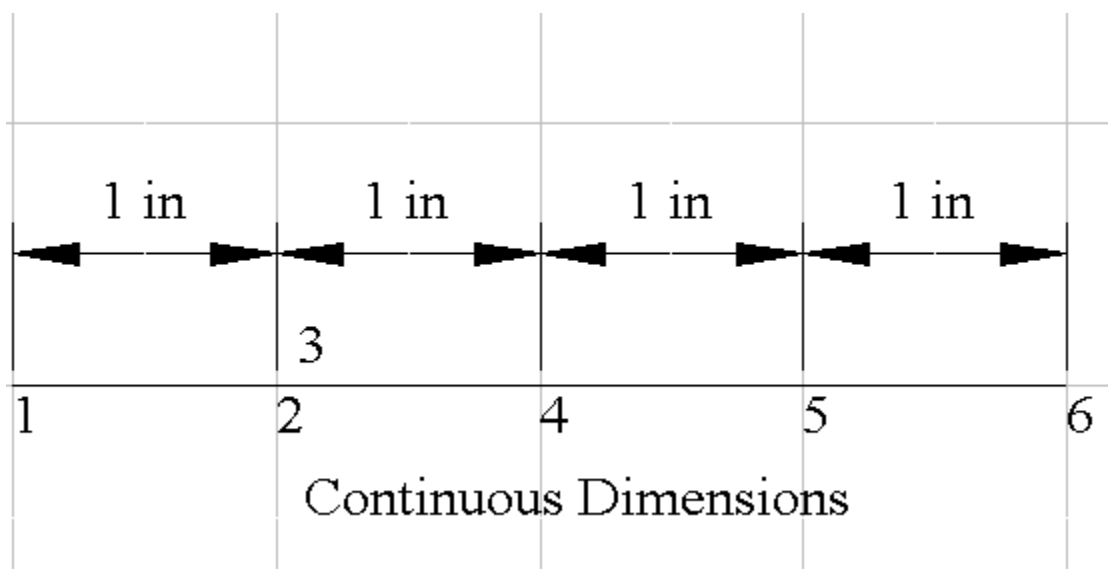


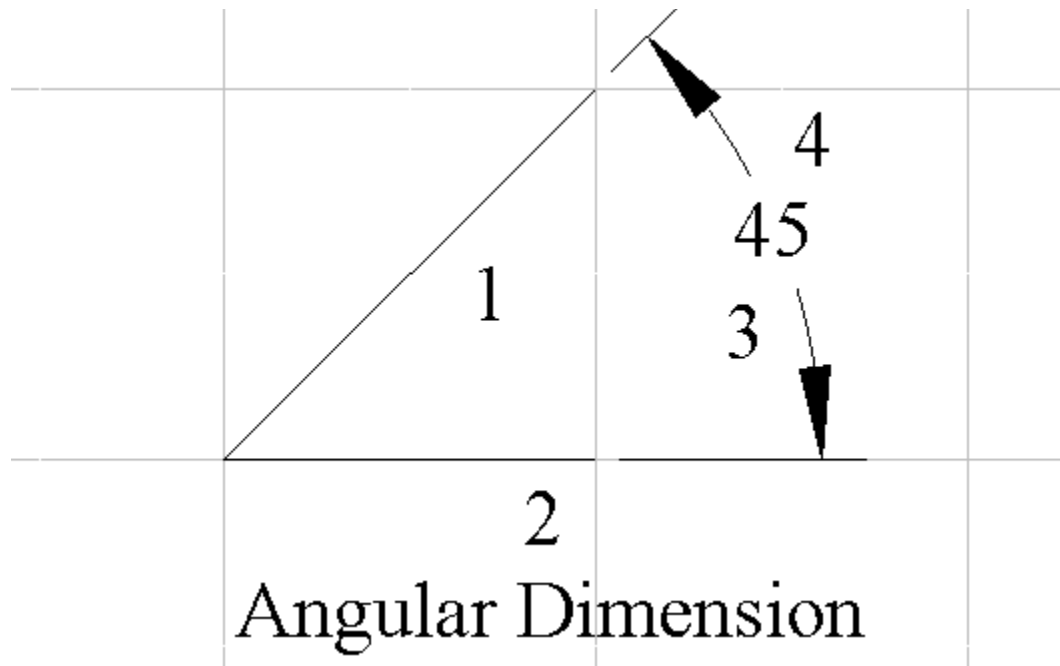
Snap Modes Tutorial

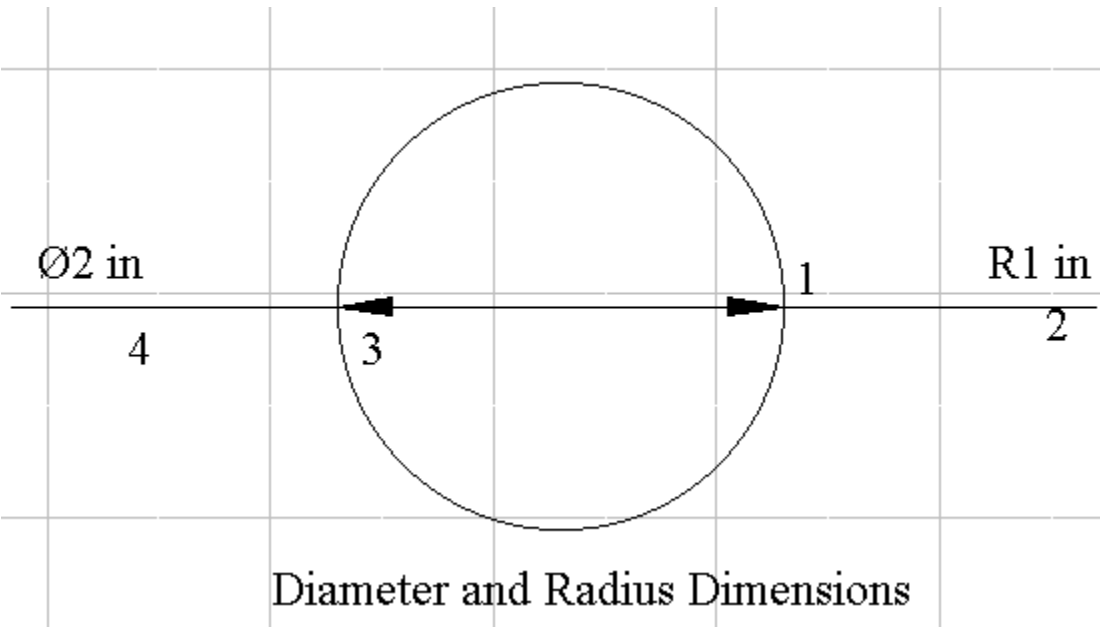


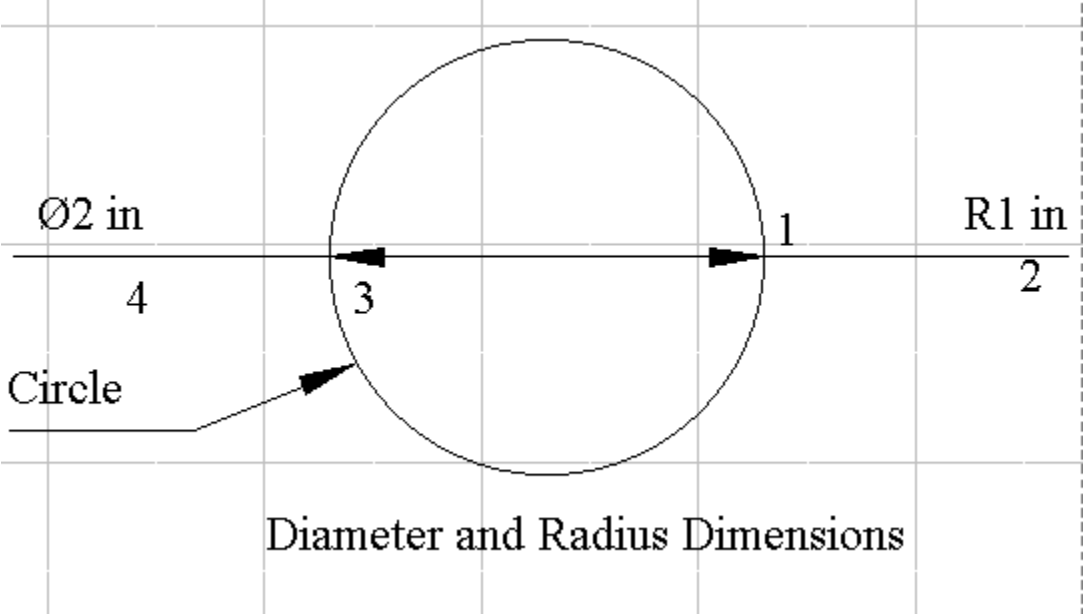


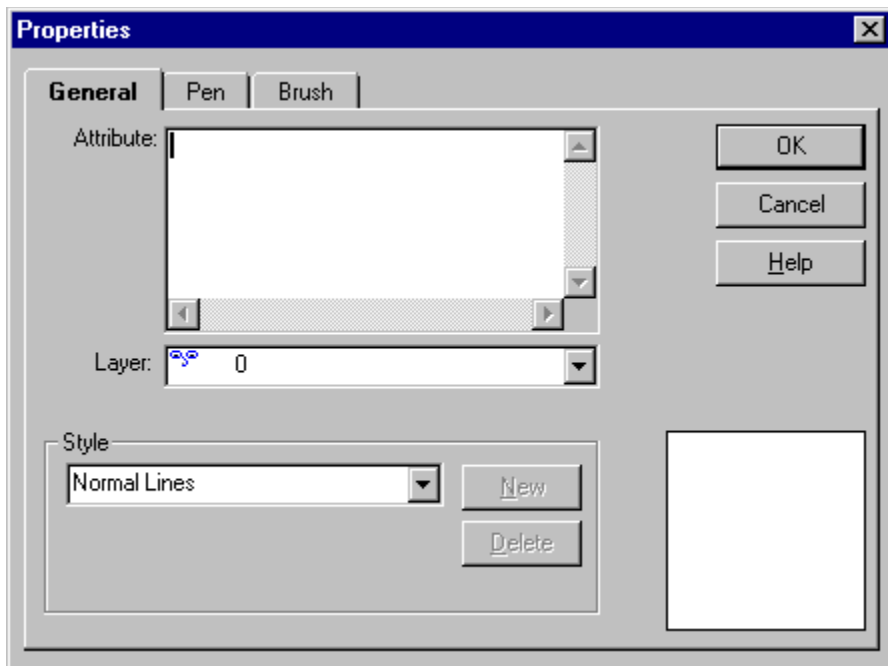




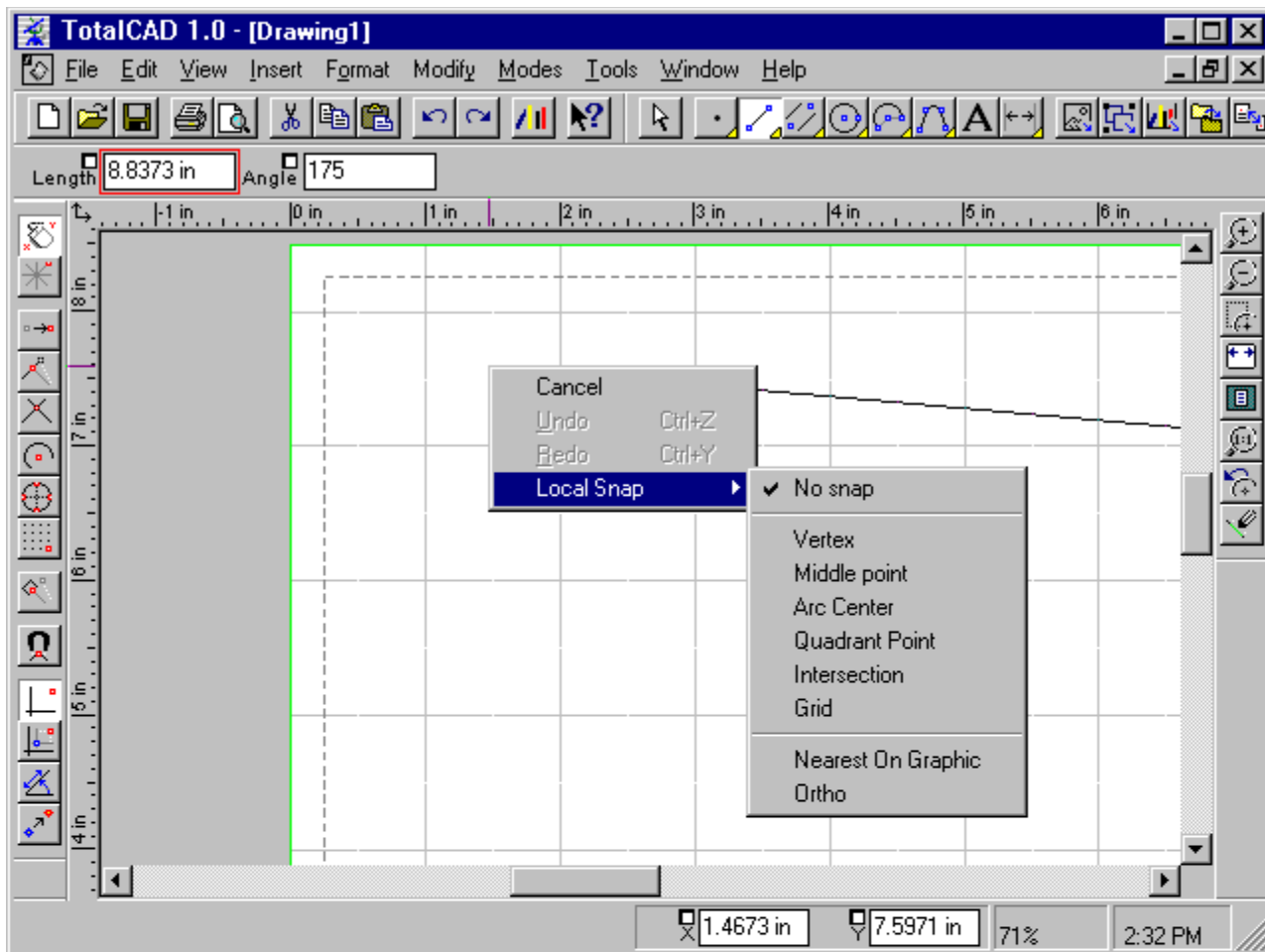


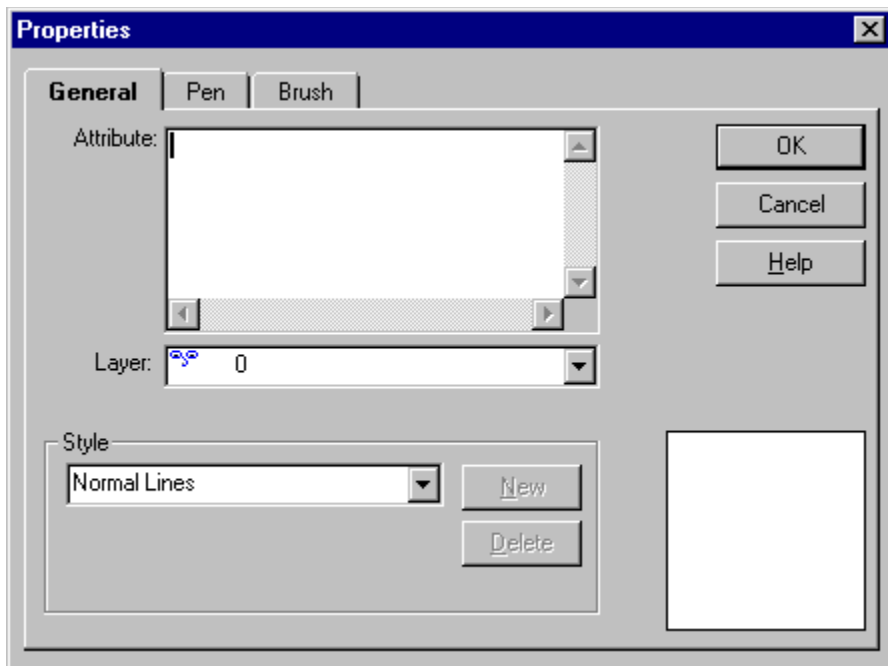


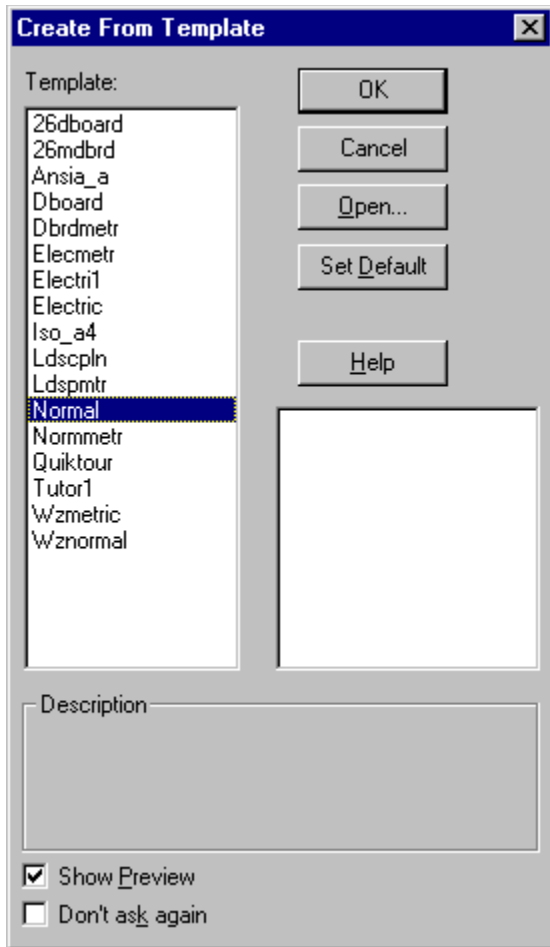




Click on this button to see how your drawing should look after completing a stage of the tutorial.







Snap Tools

The tools on the Snap toolbar let you activate snap and coordinate systems modes. By default, this toolbar is docked vertically. You can reposition it by dragging it to the top of the screen or the drawing area.

Edit Bar

The Edit Bar lets you draw and edit by typing numeric values into the Edit Bar fields. You can enter the Edit Bar from the Coordinate Fields or drawing area by pressing <Ctrl>+<E>.

Paper

The rectangle in the middle of the drawing area is the Paper. This is the area of the drawing that will be printed.

View Toolbar

The View toolbar contains tools that let you control your view of the drawing. By default, this toolbar is docked vertically. You can reposition it by dragging it to the top of the screen or the drawing area.

Scrollbar

Click on this scrollbar to "scroll" through your drawing. You can drag the "thumb" of the scrollbar to move rapidly through the drawing. Clicking on the arrow at either end scrolls a small amount. Click in the area between the thumb and the arrow to scroll a larger distance.

World Paper Button

The World/Paper button changes between World and Paper units. World units are used to measure the objects that are modeled by your drawing. Paper units measure the drawing itself at the size it will be when printed. For example, you might make a picture of a table four inches long in Paper units that represents a "real" table four feet long in World units.

Insert Toolbar

The Insert toolbar contains all the basic drawing tools. Tools that have a yellow rectangle in the lower-right corner contain flyout toolbars. To display a flyout toolbar, position the mouse cursor over the button and hold down the left mouse button.

Menus

These are standard Windows menus. To choose a menu command, click on the menu name, then click on the name of command in the menu that appears. Some commands display submenus from which you can again choose a command.

Standard Toolbar

The Standard toolbar contains tools for opening, saving, and printing drawings, as well as other standard functions. By default, this toolbar is docked horizontally at the top of the screen. You can reposition it by dragging it to the side of the screen or to the drawing area.

Coordinate Fields

The Coordinate Fields let you specify locations in your drawing by typing their coordinates. You can use absolute, polar, or relative coordinates, depending on the current coordinate system. You can enter the Coordinate Fields from the Edit Bar or drawing area by pressing <Ctrl>+<R>.

Zoom Percentage

The Zoom percentage box tells you how far you are zoomed into your drawing. A zoom percentage of 100% means that you are viewing the drawing at its printed size.

Ruler

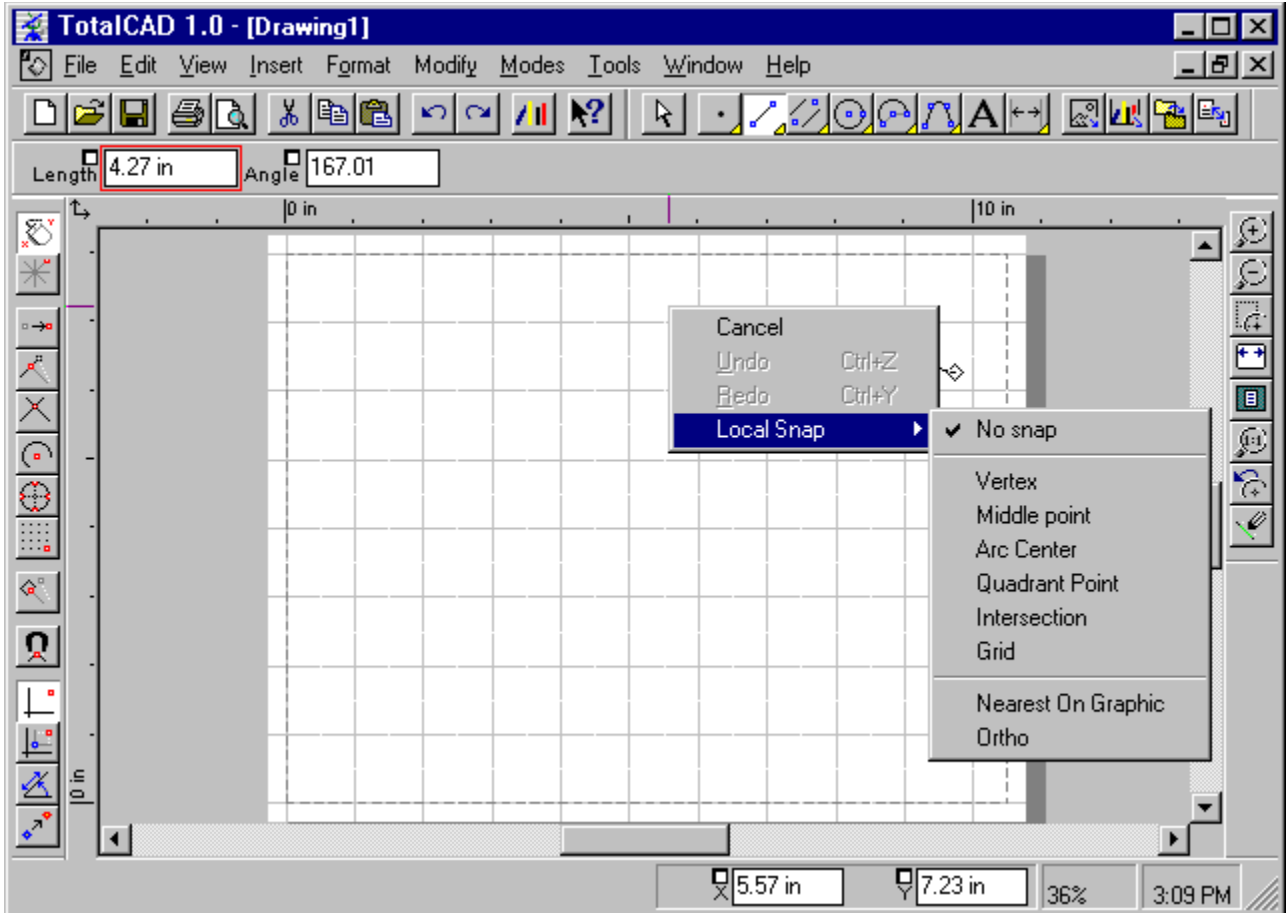
The vertical and horizontal rulers provide feedback about the location and size of objects in your drawing. You can change the rulers between World and Paper coordinates by clicking on the World/Paper button in the upper-left corner of the rulers. You can hide the rulers to increase the size of your drawing area using the View|Rulers command.

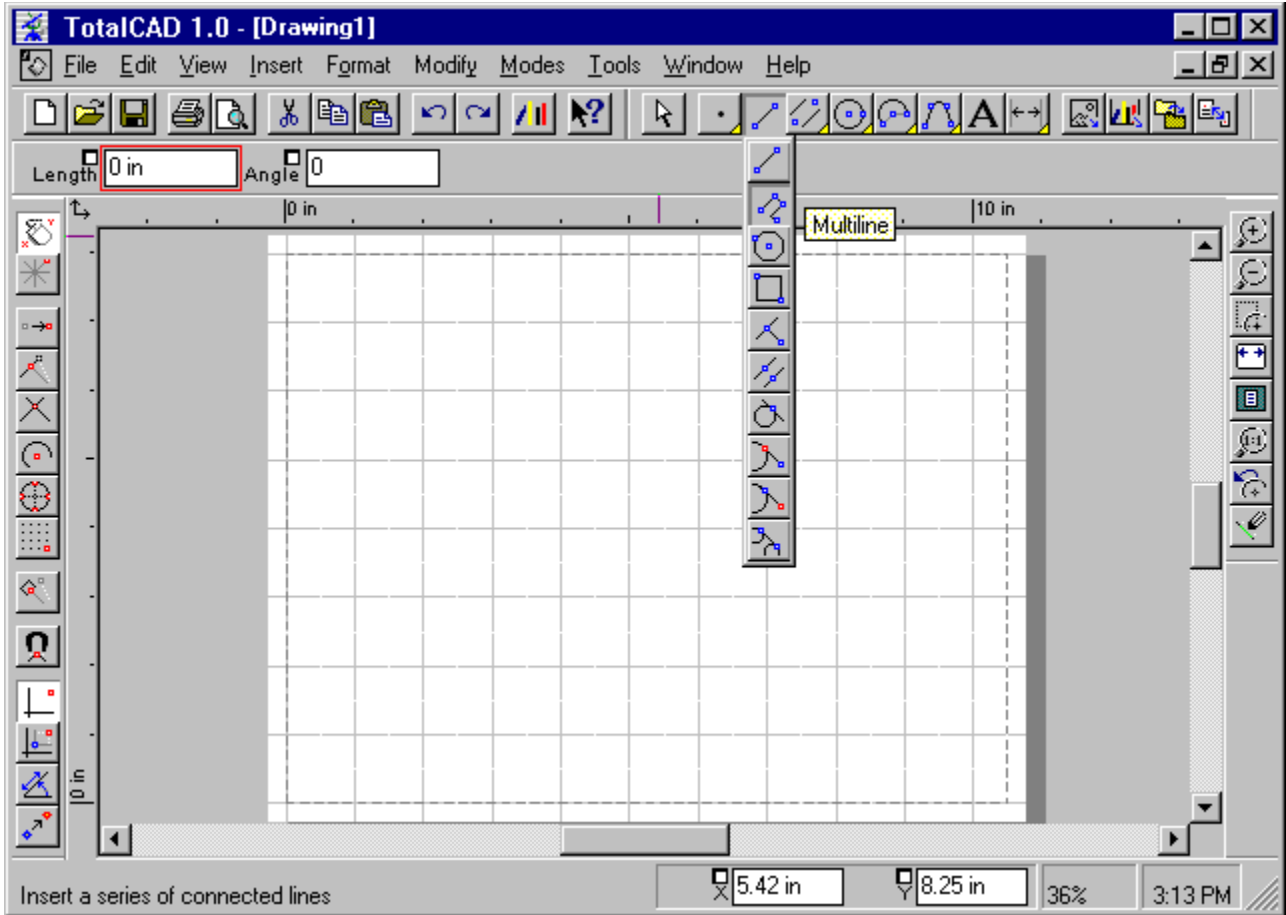
Drawing Area

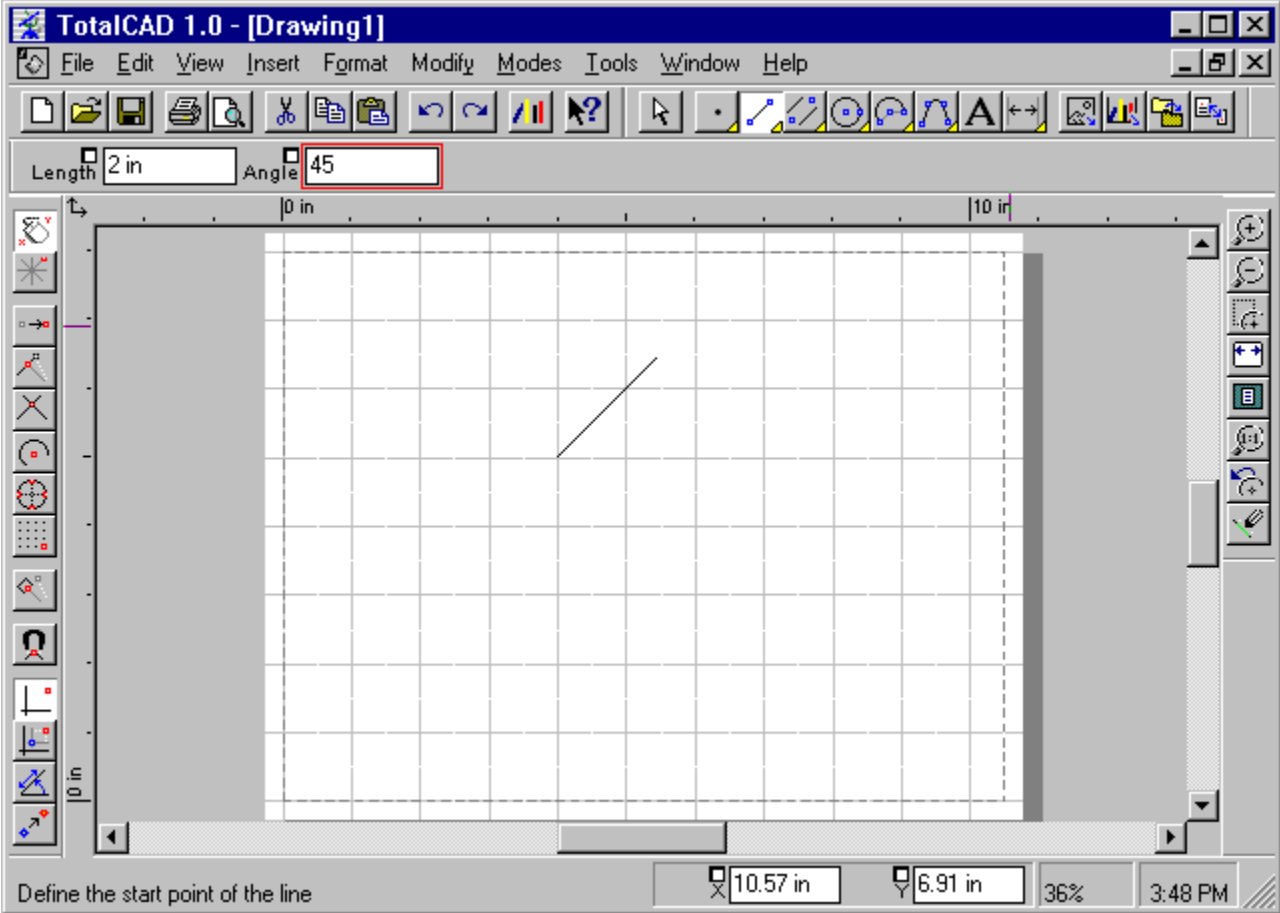
The drawing area is where you actually create your drawing. You can display the local menu by right-clicking in this area. You can enter the drawing area from the Coordinate Fields or from the Edit Bar by pressing <Ctrl>+<D>.

Status Bar Message

The Status Bar message gives you information about TotalCAD's current state or the next step in a procedure.





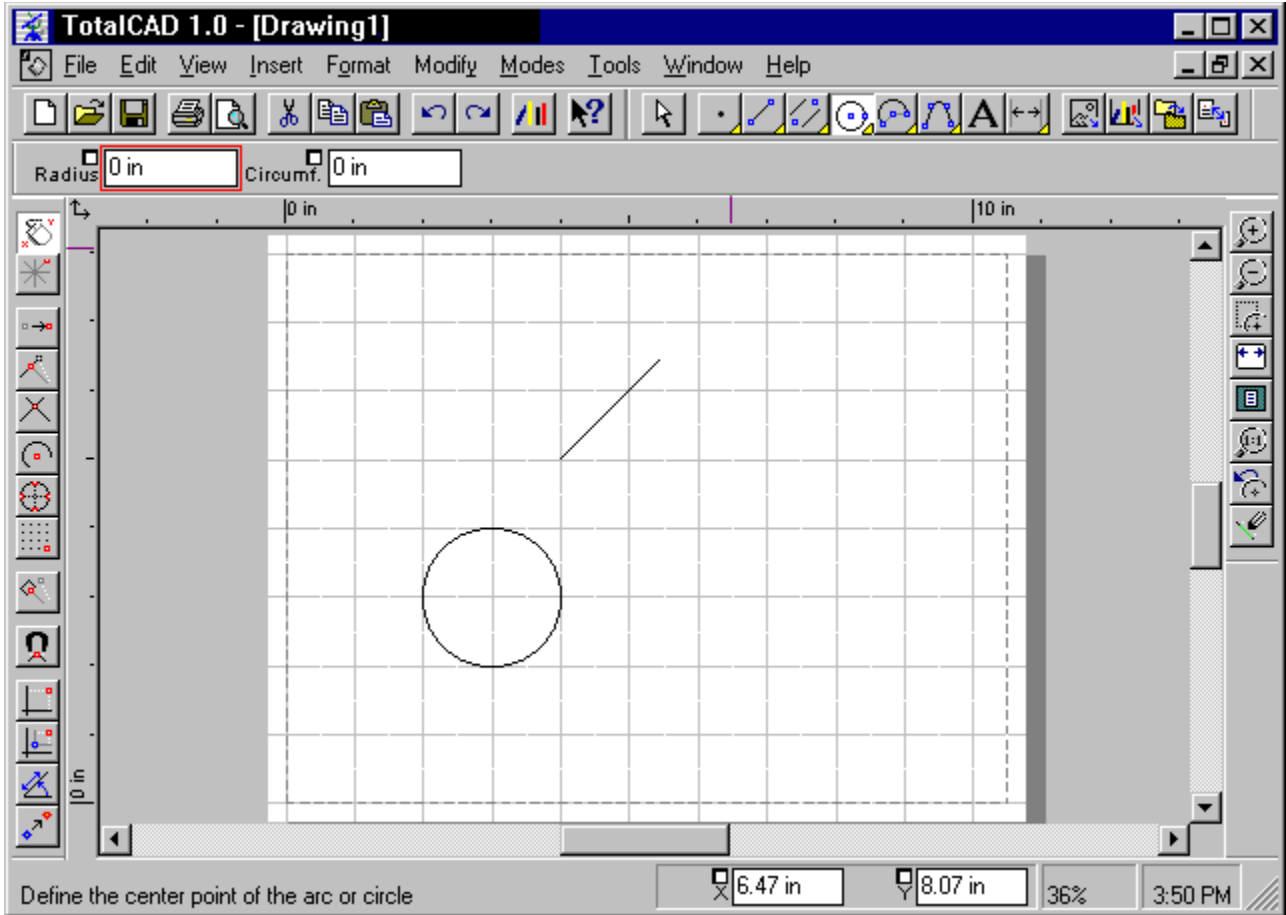


Windows Buttons

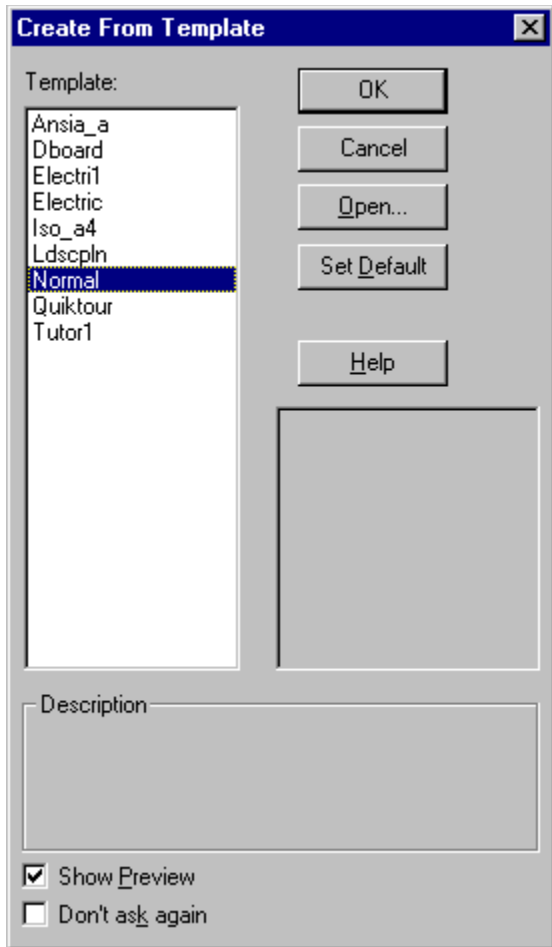
These standard Windows buttons allow you to minimize, maximize, restore, or close a window. See your Microsoft Windows documentation for details.

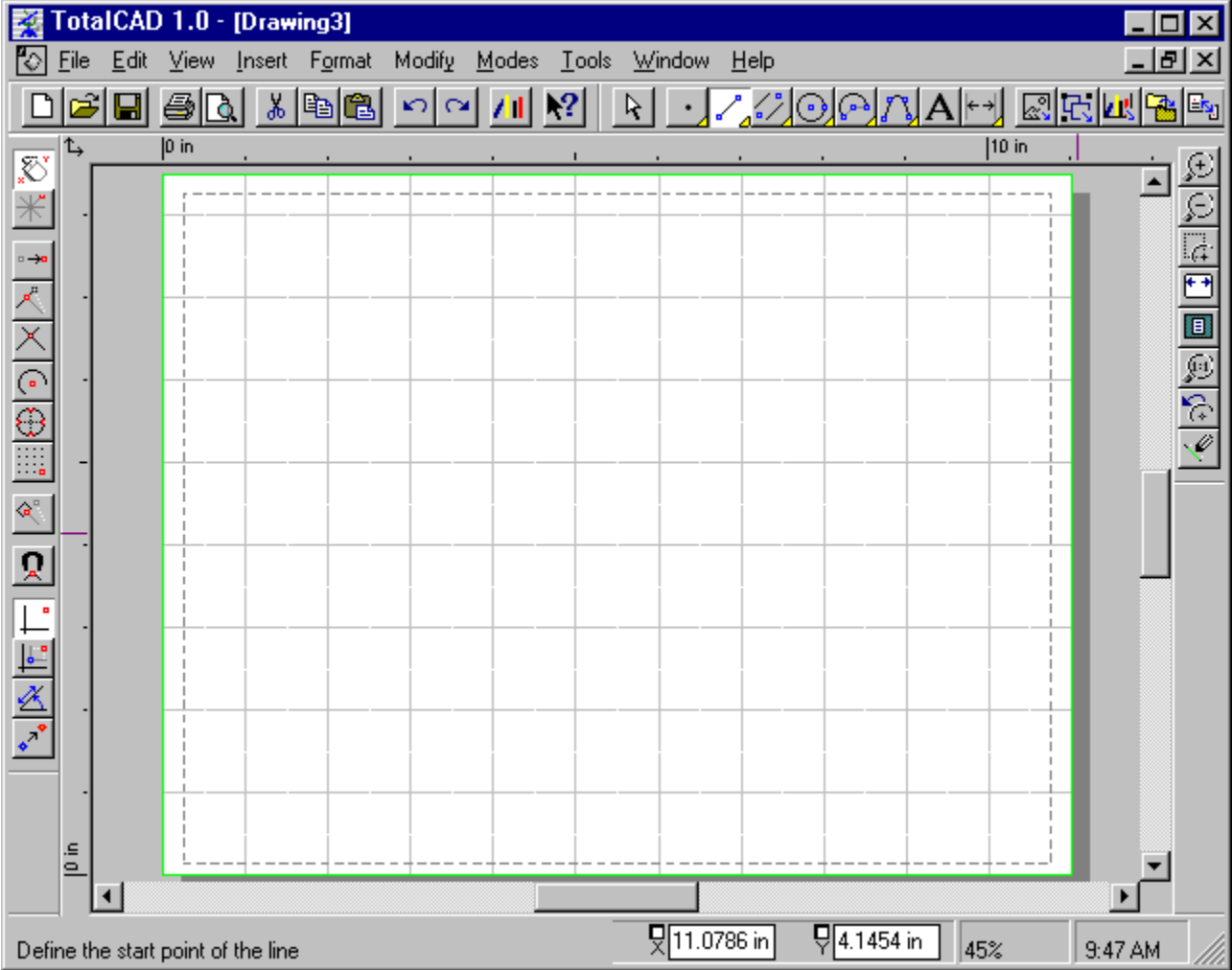
Title Bar

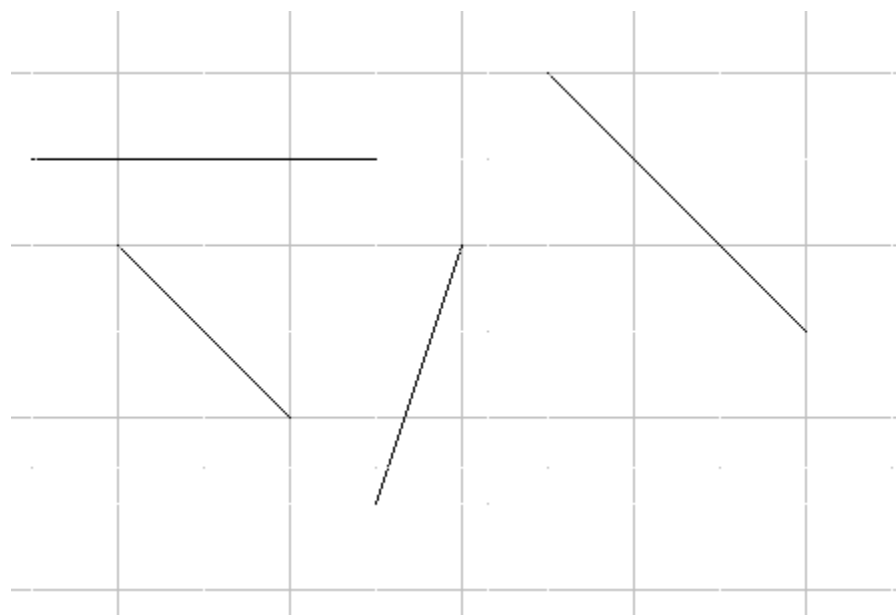
The title bar at the top of the window displays the name of the program and the name of the currently active drawing.

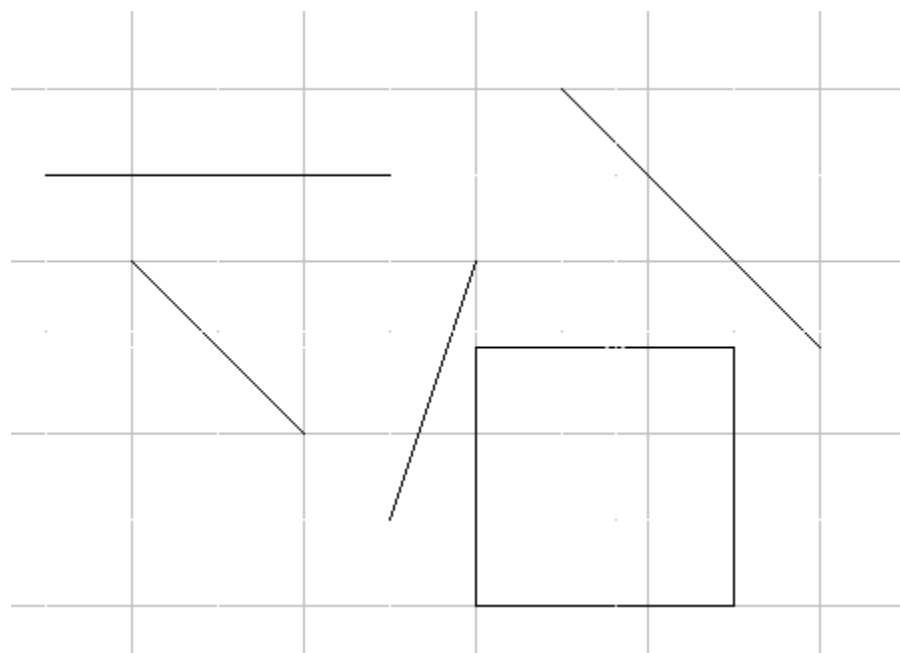












Print [X]

Printer: System Printer (Linotronic 530)

Print Source

Drawing

Selection

Print Range

All

View:

* Current View

Pages

From: 1 To: 1

Fit on 1 Page

Print Quality: 1693 dpi

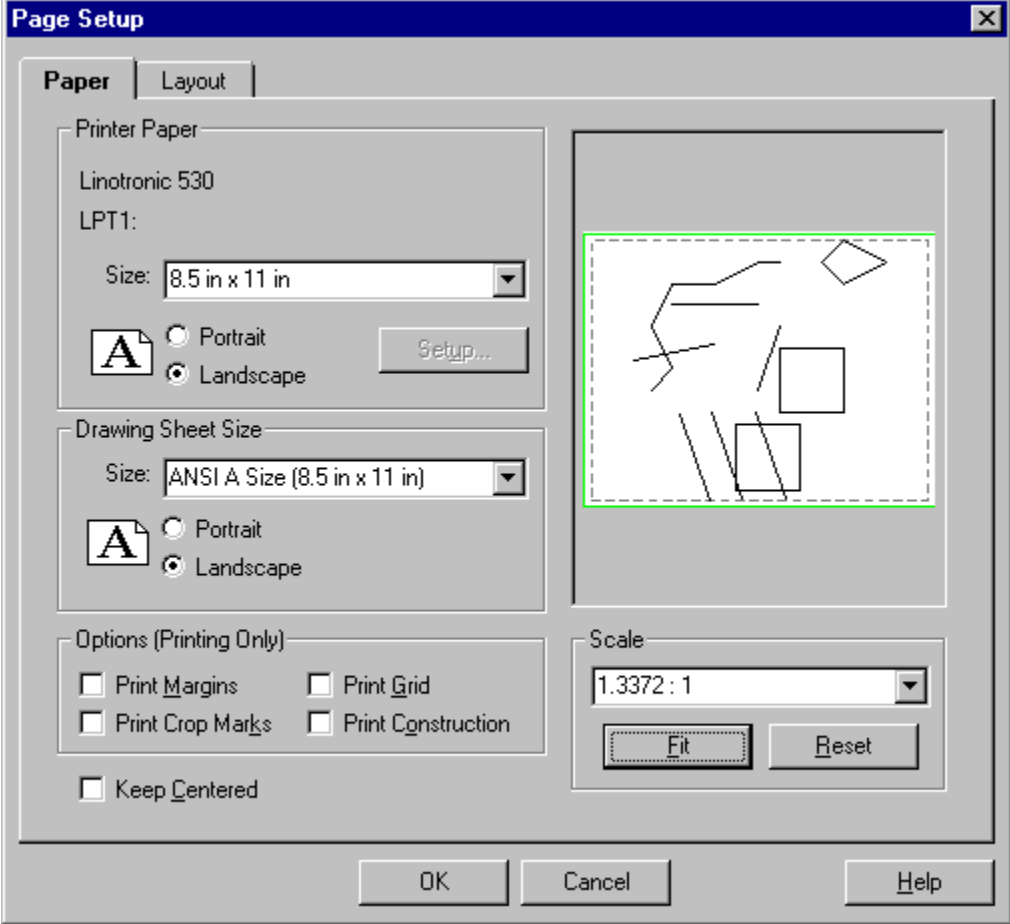
Copies: 1

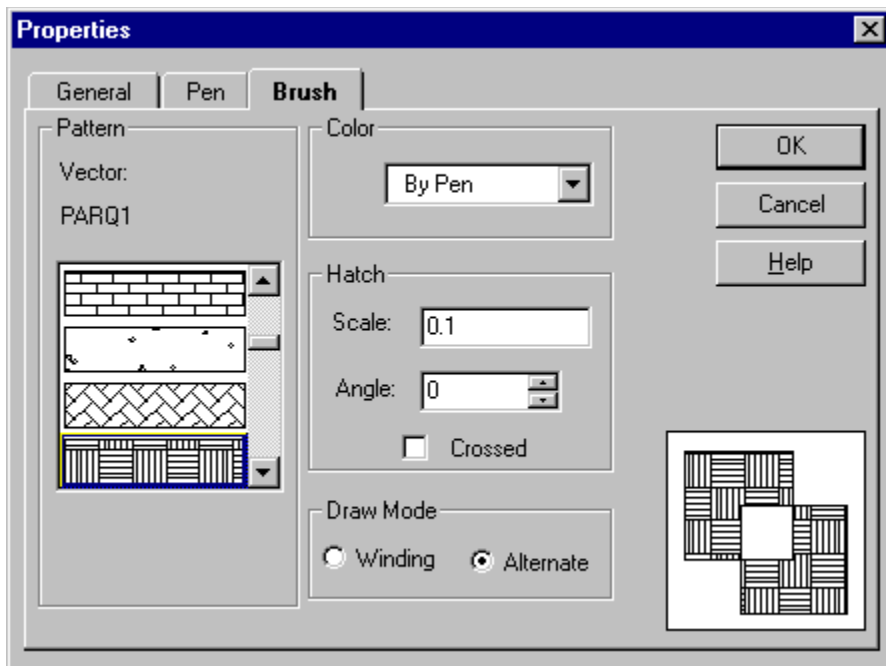
OK

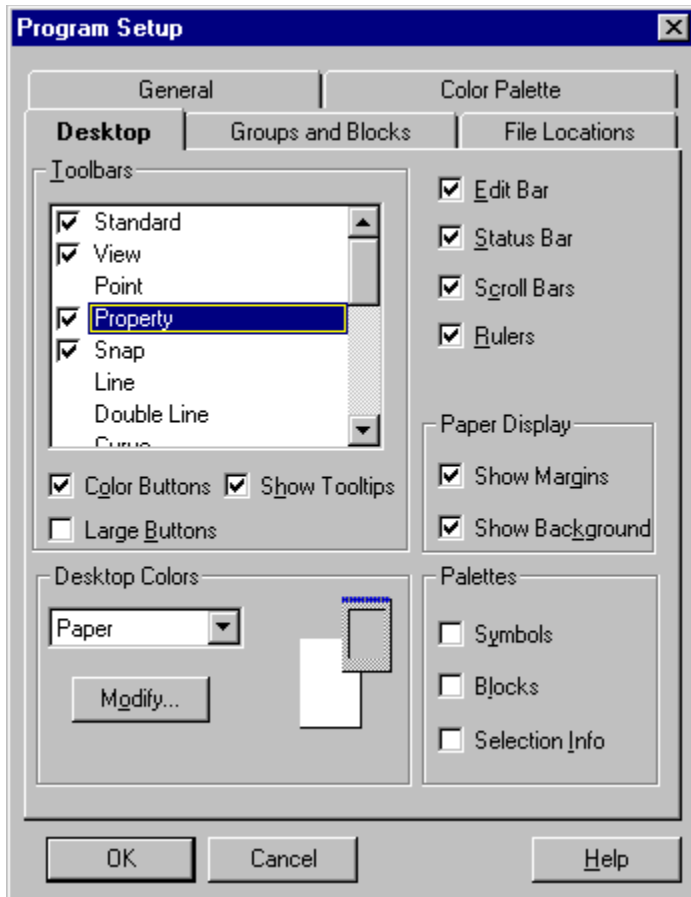
Cancel

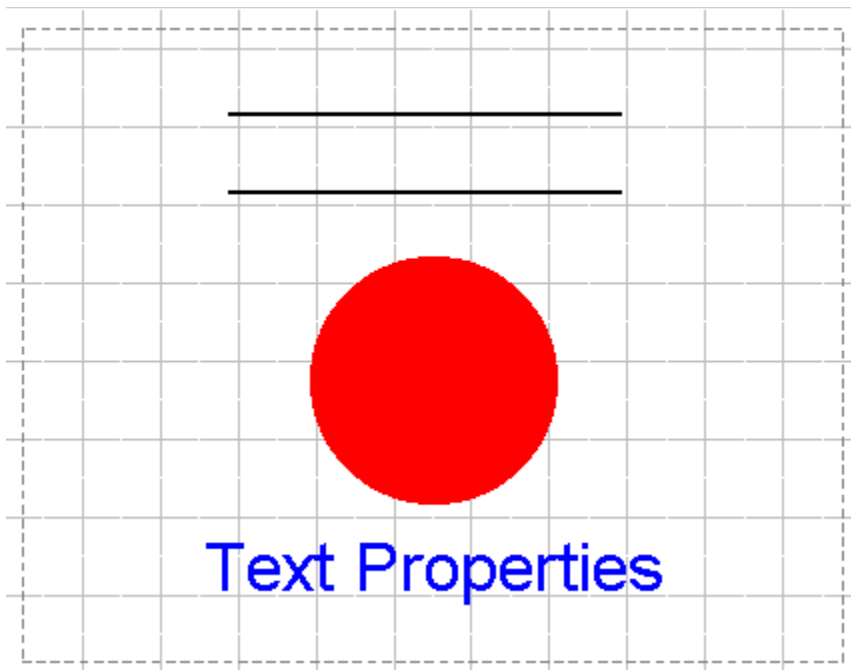
Page Setup...

Print Setup...

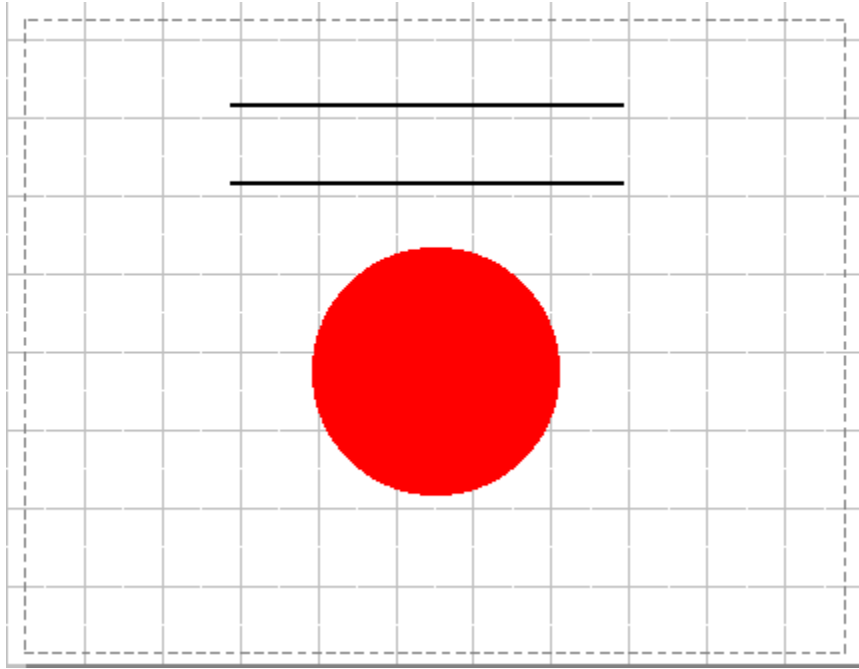








Text Properties



Style

Use this list box to apply predefined property setting to a drawing tool. You can define and save styles in the Properties dialog. This control can be used only to set the properties of a drawing tool. It is unavailable if you are setting the properties of selected objects.

Layer

Use this list box to set the layer on which objects are drawn.

Color

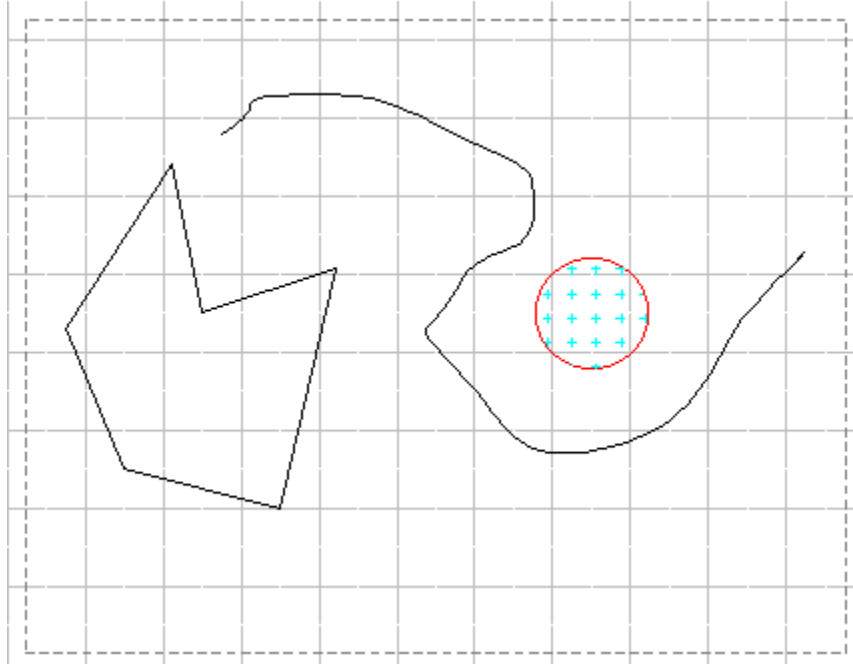
Use this list box to set the pen color.

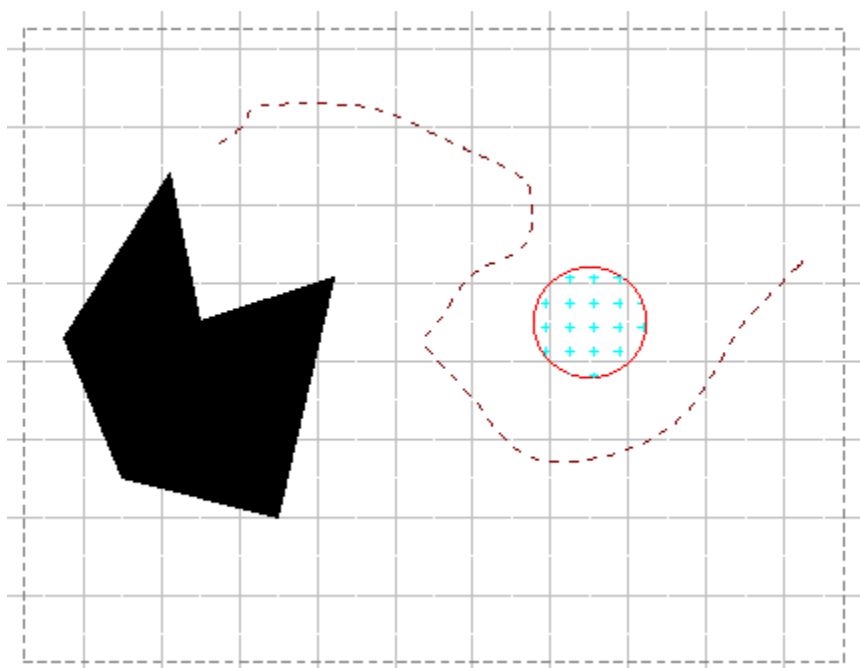
Line Style

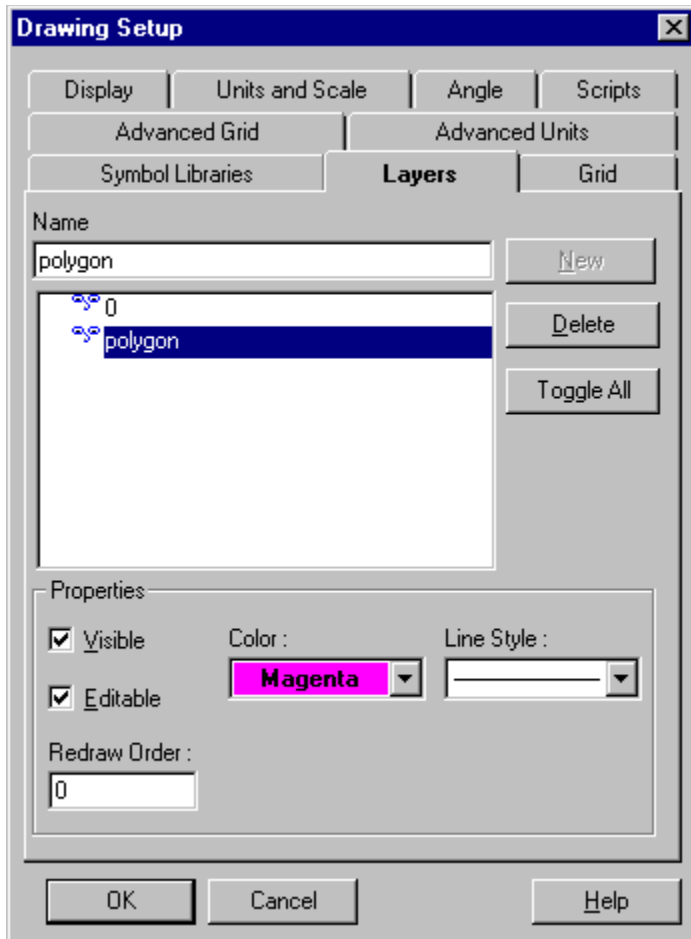
Use this list box to set the style of the line. You can use a continuous line, or choose from several types of dotted or dashed lines.

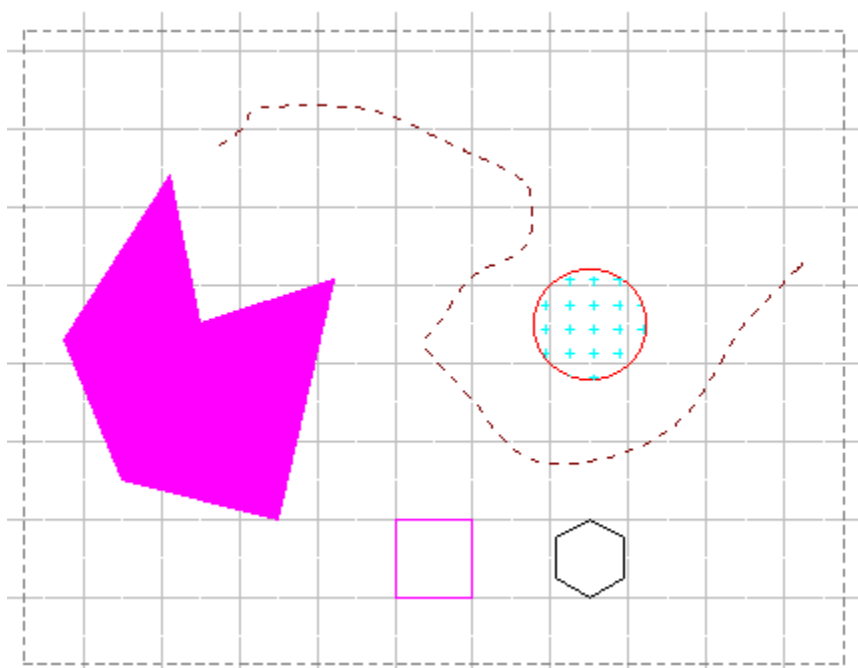
Brush Style

Use this list box to choose a fill pattern for closed entities.









Font

Pull down this list box to set the font. You can use any TrueType font that is installed on your system.

Size

Pull down this list box to set the text size.

Attribute

Click the Bold, Italic, and Underline buttons to turn these text attributes on and off.

Justification

Click these buttons to choose whether the text is left, center, or right-justified in its frame.

You will change the font of this text from Times New Roman to Arial.

You will change the height of this text to .25 inches.

You will change this text to center justification so that it could be a heading at the top of a drawing.

You will make this text bold.

You will make this text underlined italic.

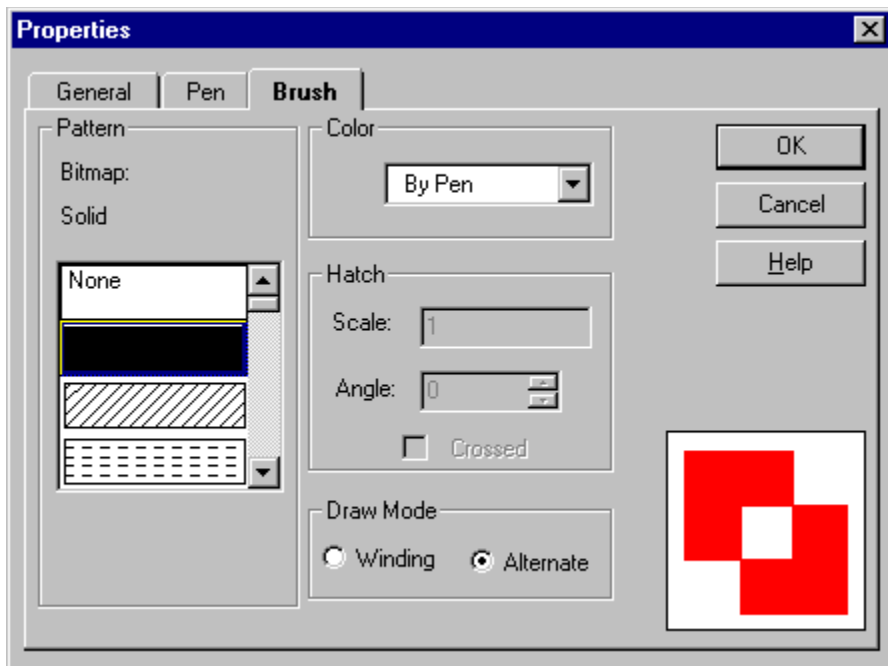
You will change the font of this text from Times New Roman to Arial.

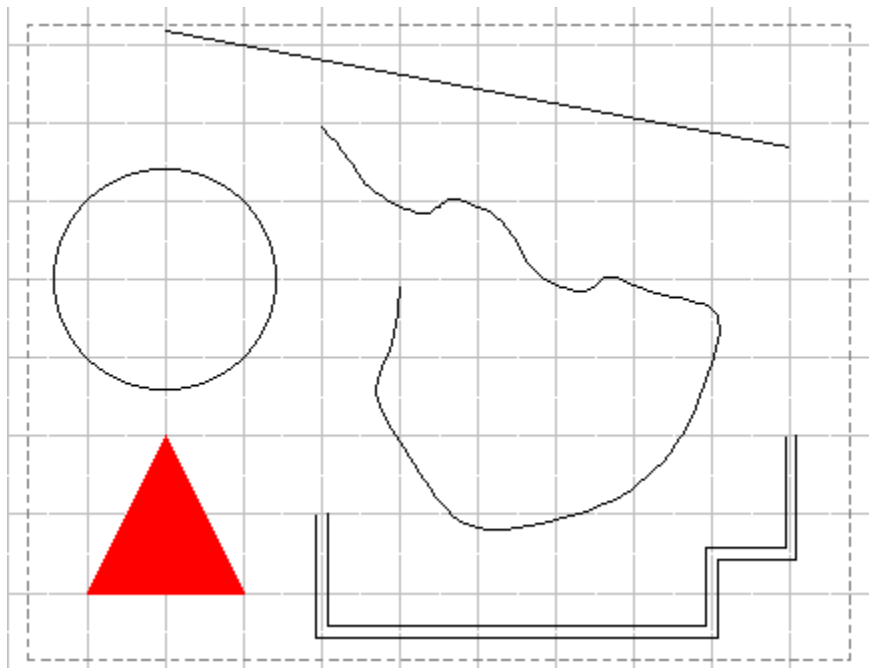
You will change the font of this text to Arial.

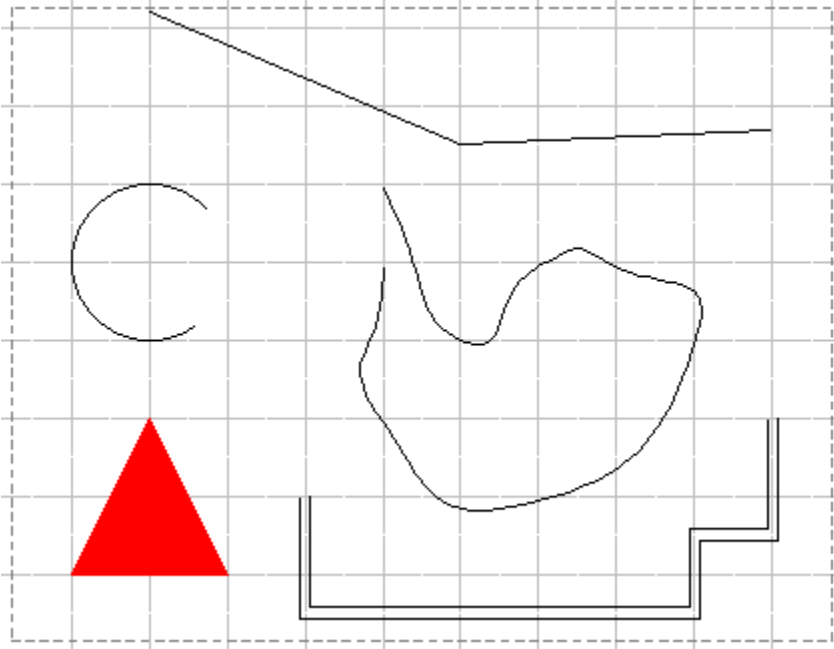
You will change this text to center justification so that it could be a heading at the top of a drawing.

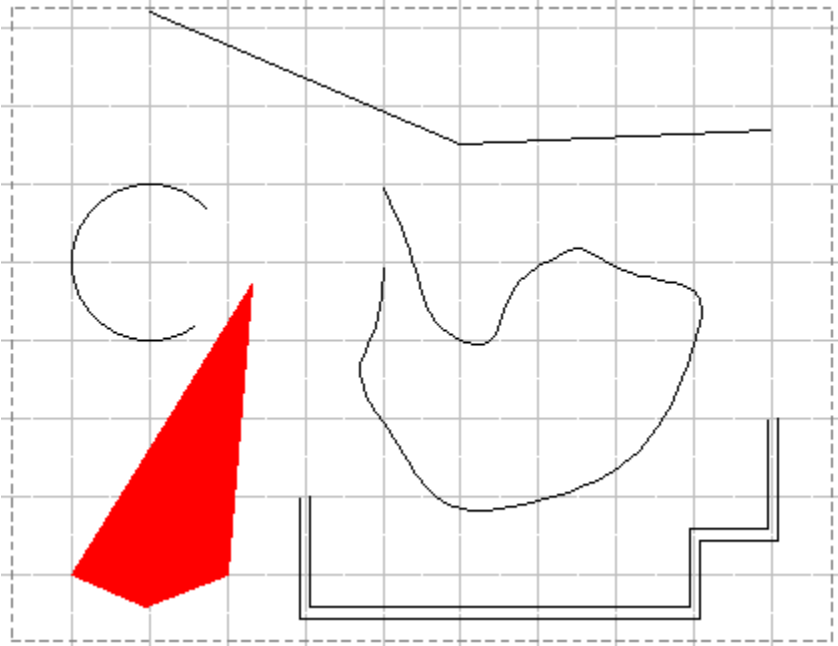
You will make this text bold.

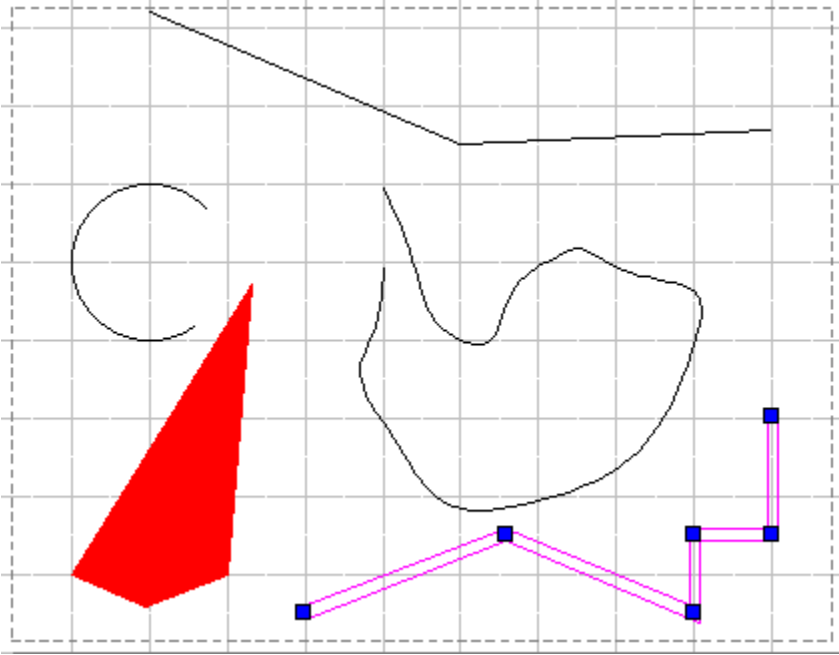
You will make this text underlined italic.

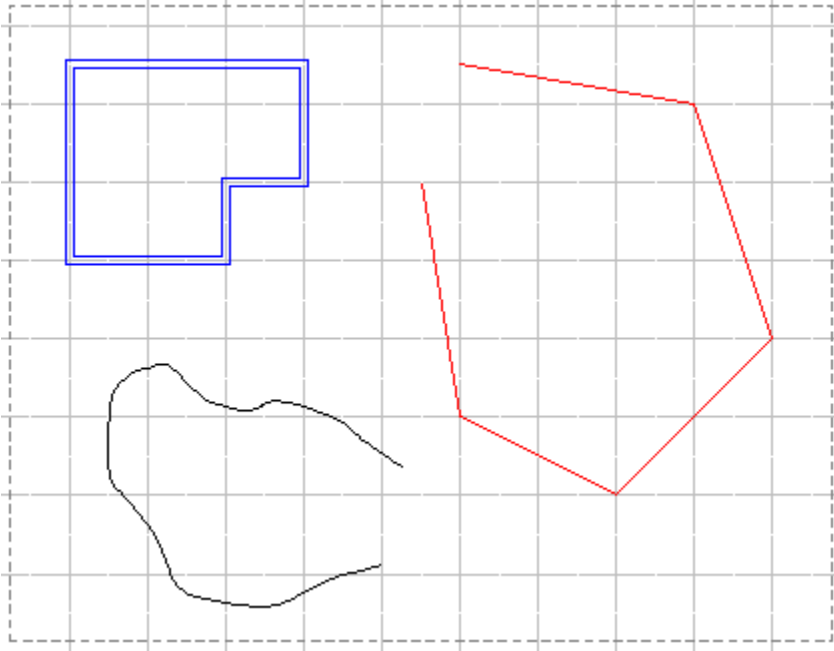


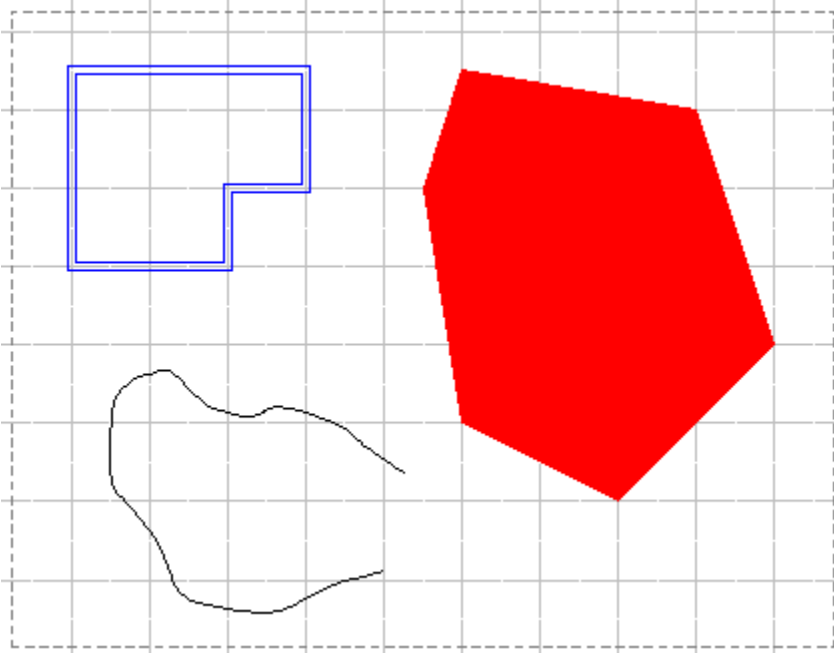


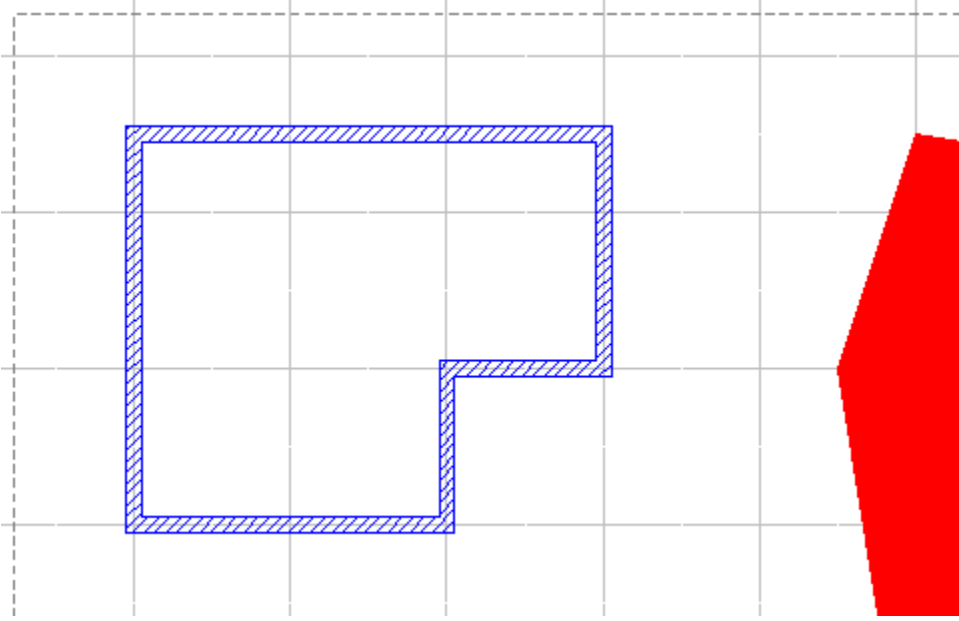


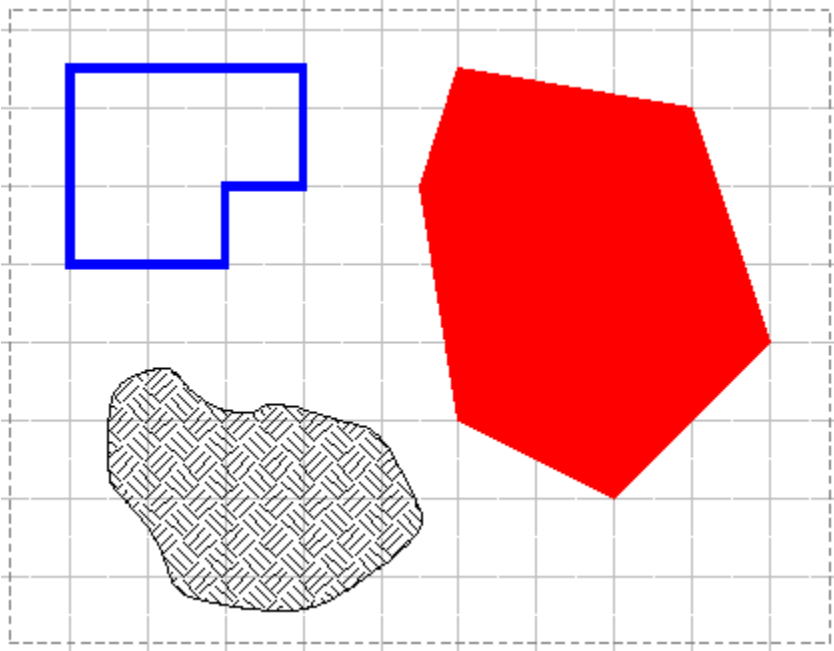


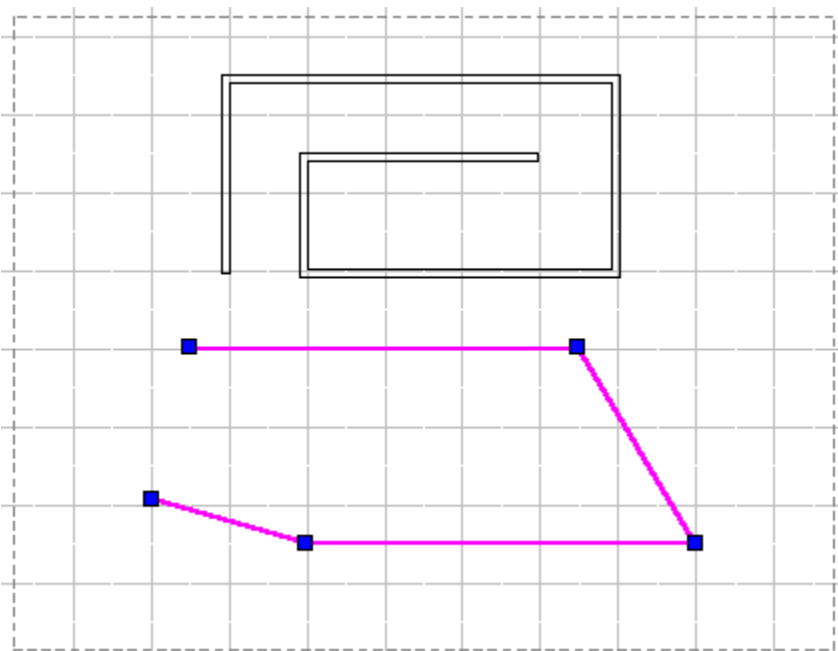


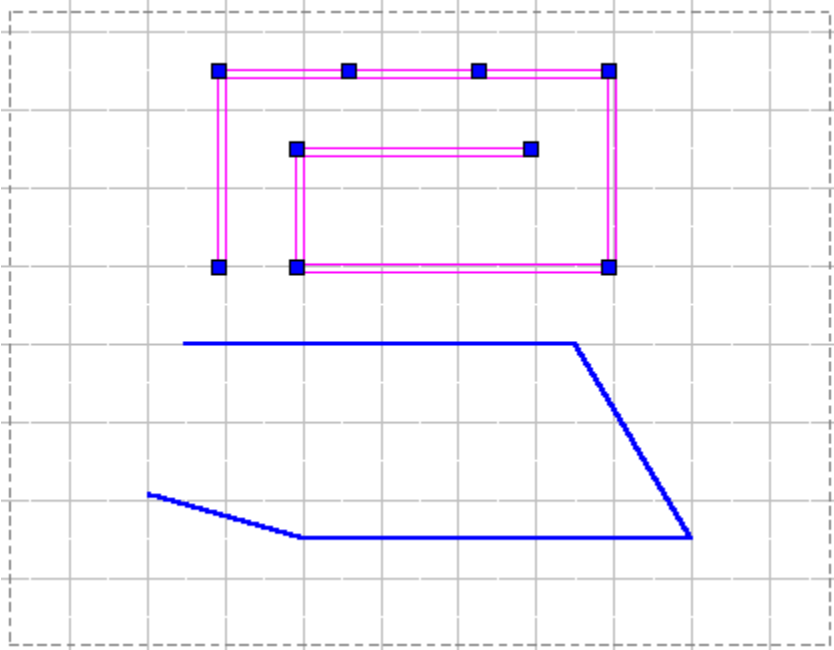


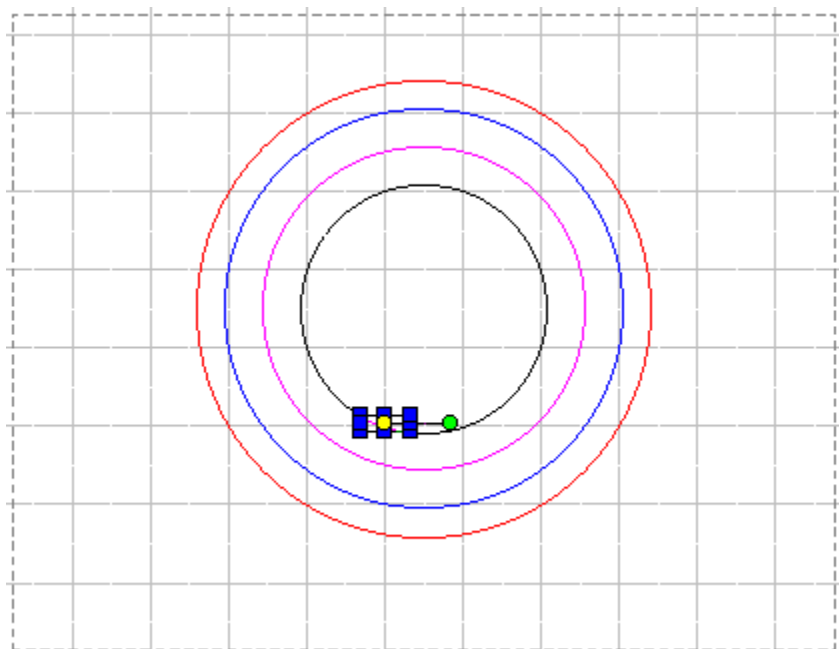


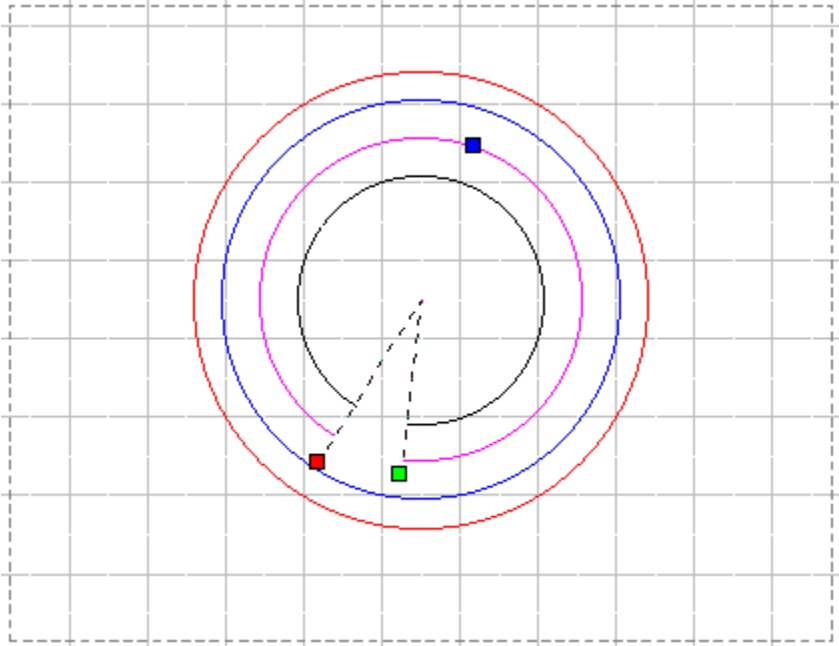


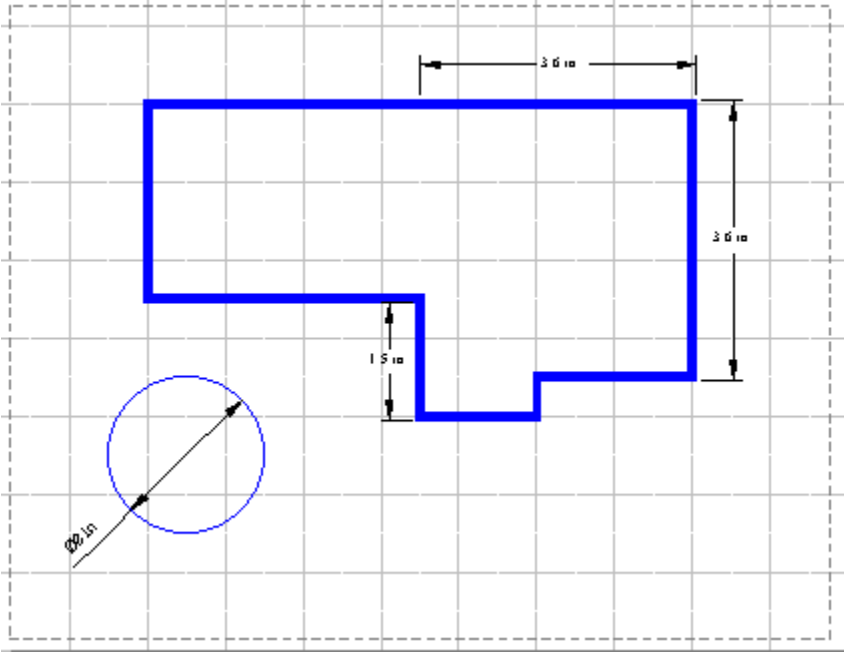












Properties [X]

General **Format**

Arrowheads

1 st: [Blue Arrowhead] Size: 0.25 in

2 nd: [White Arrowhead] Tick Extension: 0 in

Text

Font: Times New Roman Height: 0.25 in

Position

Above On Line

Outside On Line Adjustment: 0

Force Text Horizontal

Force Dimension Text Inside Extension Lines

Associative Dimension

OK

Cancel

Help

