

Chapter 1

Template Drawings

CHAPTER OBJECTIVES

After completing this chapter, you will be able to:

- Create template drawings.
- Load template drawings using dialog boxes and the Command line.
- Do an initial drawing setup.
- Customize drawings with layers and dimensioning specifications.
- Customize drawings with layouts, viewports, and paper space.

KEY TERMS

- *Template*
- *Plot Size*
- *Layout*
- *Template Files (*.dwt)*
- *LTSCALE*
- *Viewport*
- *Drawing Scale*
- *DIMSCALE*
- *STARTUP*
- *TEXTSIZE*

CREATING TEMPLATE DRAWINGS

One way to customize AutoCAD is to create template drawings that contain initial drawing setup information and if desired, visible objects and text. When the user starts a new drawing, the settings associated with the template drawing are automatically loaded. If you start a new drawing from the scratch, AutoCAD loads default setup values. For example, the default limits are (0.0,0.0), (12.0,9.0) and the default layer is 0 with white color and a continuous linetype. Generally, these default parameters need to be reset before generating a drawing on the computer using AutoCAD. A considerable amount of time is required to set up the layers, colors, linetypes, lineweights, limits, snaps, units, text height, dimensioning variables, and other parameters. Sometimes, border lines and a title block may also be needed.

In production drawings, most of the drawing setup values remain the same. For example, the company title block, border, layers, linetypes, dimension variables, text height, LTSCALE, and other drawing setup values do not change. You will save considerable time if you save these values and reload them when starting a new drawing. You can do this by creating template drawings that contain the initial drawing setup information configured according to the company specifications. They can also contain a border, title block, tolerance table, block definitions, floating viewports in the paper space, and perhaps some notes and instructions that are common to all drawings.

STANDARD TEMPLATE DRAWINGS

AutoCAD comes with standard template drawings like *Acad.dwt*, *Acadiso.dwt*, *Acad3D.dwt*, *Acadiso3D.dwt*, *Acad-named plot styles.dwt*, *AcadISO-named plot styles.dwt*, and so on. The iso template drawings are based on the drawing standards developed by ISO (International Organization for Standardization). When you start a new drawing with **STARTUP** system variable set to 1, the **Create New Drawing** dialog box will be displayed. To load the template drawing, choose the **Use a Template** button; the list of standard template drawings is displayed. From this list, you can select any template drawing according to your requirements. If you want to start a drawing with the default settings, select the **Start from Scratch** button in the **Create New Drawing** dialog box. The following are some of the system variables, with the default values that are assigned to the new drawing.

<u>System Variable Name</u>	<u>Default Value</u>
CHAMFERA	0.0000
CHAMFERB	0.0000
COLOR	Bylayer
DIMALT	OFF
DIMALTD	2
DIMALTF	25.4000
DIMPOST	None
DIMASO	ON
DIMASZ	0.1800
FILLETRAD	0.0000
GRID	0.5000
GRIDMODE	0
ISOPLANE	Top
LIMMIN	0.0000,0.0000
LIMMAX	12.0000,9.0000
LTSCALE	1.0
MIRRTEXT	0 (Text not mirrored like other objects)
TILEMODE	1 (OFF)

EXAMPLE 1

Advance Setup Wizard

Create a drawing template using the **Advanced Setup** wizard displayed on invoking the **Use a Wizard** button. You can use the following specifications and save it with the name *proto1.dwt*.

Units	Engineering with precision 0'-0.00"
Angle	Decimal degrees with precision 0
Angle Direction	Counterclockwise
Area	144'x96'

Step 1: Setting the STARTUP system variable

Set the value of the **STARTUP** system variable to 1 to show the **Startup** dialog box. To do so, start AutoCAD using the default template. Enter **STARTUP** at the Command prompt; you will be prompted to enter the new value for startup. Enter **1** in the command prompt; the **STARTUP** system variable is set to **1**. Next, choose the **New** tool from the **Quick Access** toolbar to display the **Create New Drawing** dialog box. Choose the **Use a Wizard** button and choose the **Advanced Setup** option, as shown in Figure 1-1. Choose **OK**. The **Units** page of the **Advanced Setup** dialog box is displayed, as shown in Figure 1-2.

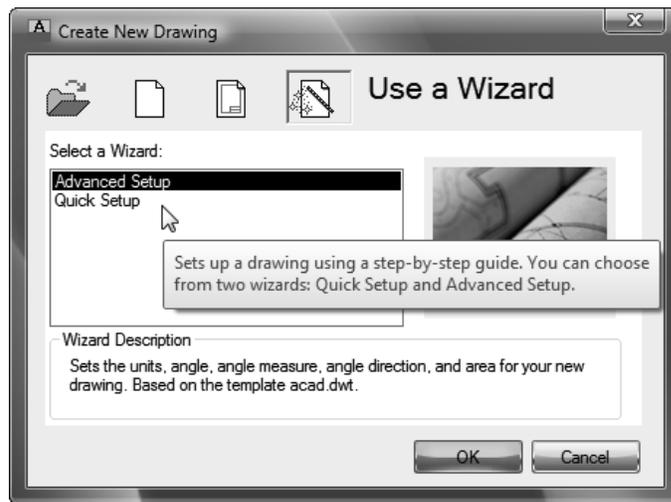


Figure 1-1 The **Advanced Setup** wizard in the **Create New Drawing** dialog box

Step 2: Setting the units of the drawing file

Select the **Engineering** radio button. Select **0'-0.00"** precision from the **Precision** drop-down list if it is not selected by default, refer to Figure 1-2, and then choose the **Next** button; the **Angle** page of the **Advanced Setup** dialog box is displayed.

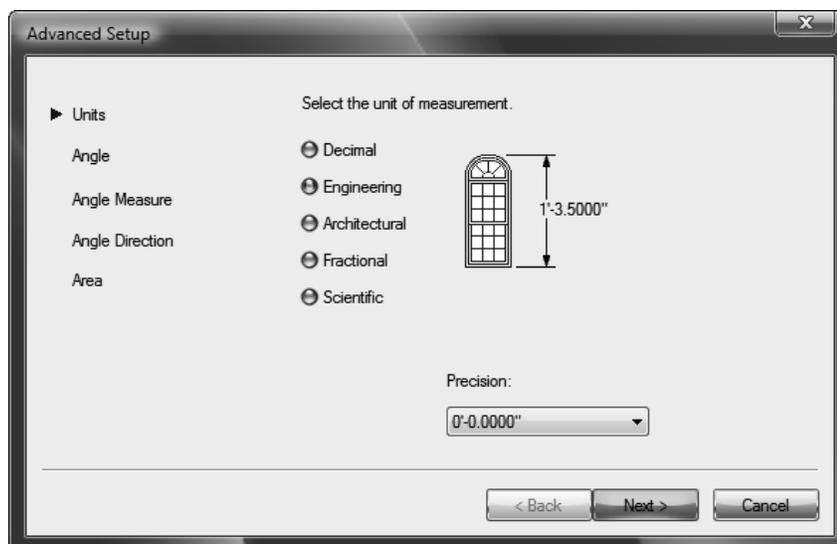


Figure 1-2 The **Units** page of the **Advanced Setup** dialog box

Step 3: Setting the angle measurement system

In the **Angle** page, select the **Decimal Degrees** radio button and select **0** from the **Precision** drop-down list, as shown in Figure 1-3. Choose the **Next** button; the **Angle Measure** page of the **Advanced Setup** dialog box is displayed.

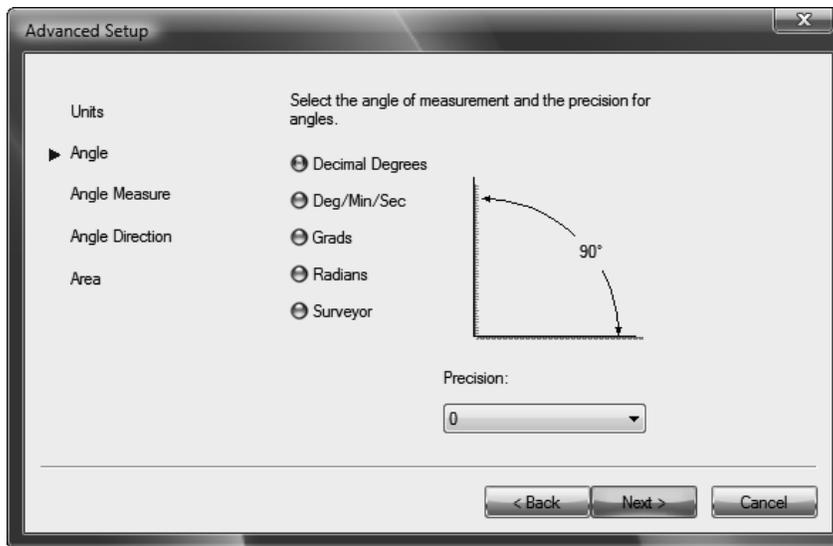


Figure 1-3 The Angle page of the Advanced Setup dialog box

Step 4: Setting the horizontal axis for angle measurement

In the **Angle Measure** page, select the **East** radio button. Choose the **Next** button to display the **Angle Direction** page.

Step 5: Setting the angle measurement direction and drawing area

Select the **Counter-Clockwise** radio button and then choose the **Next** button. The **Area** page is displayed. Specify the area as 144' and 96' by entering the value of the width and length as **144'** and **96'** in the **Width** and **Length** edit boxes and then choose the **Finish** button. Use the **All** option of the **ZOOM** command to display new limits on the screen.

Step 6: Saving the drawing as template file

Now, save the drawing as *proto1.dwt* using the **Save** tool from the **Quick Access** toolbar. You need to select **AutoCAD Drawing Template (*.dwt)** from the **Files of type** drop-down list and enter **proto1** in the **File name** edit box in the **Save Drawing As** dialog box. Next, choose the **Save** button; the **Template Options** dialog box will be displayed on the screen, as shown in Figure 1-4. Enter the description about the template in the **Description** edit box and choose the **OK** button. Now, the drawing will be saved as *proto1.dwt* on the default drive.

**Note**

To customize only the units and area, you can use the **Quick Setup** option in the **Create New Drawing** dialog box.

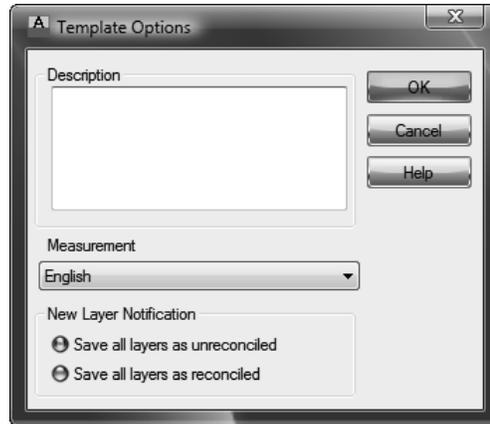


Figure 1-4 The Template Options dialog box

EXAMPLE 2 Start from Scratch Option

Create a drawing template using the following specifications. The template should be saved with the name *proto2.dwt*.

- Limits 18.0,12.0
- Snap 0.25
- Grid 0.50
- Text height 0.125
- Units 3 digits to the right of decimal point
 Decimal degrees
 2 digits to the right of decimal point
 0 angle along positive X axis (east)
 Angle positive if measured counterclockwise

Step 1: Starting a new drawing

Start AutoCAD and choose **Start from Scratch** button from the **Create New Drawing** dialog box. From the **Default Settings** area, select the **Imperial (feet and inches)** radio button, as shown in Figure 1-5. Choose **OK** to open a new file.

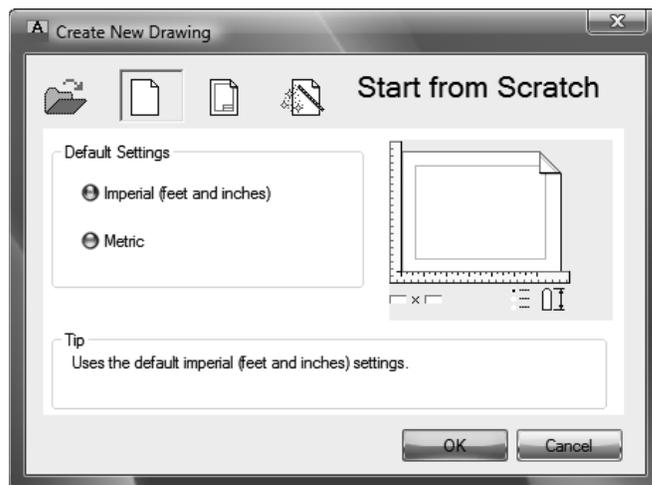


Figure 1-5 The Start from Scratch button of the Create New Drawing dialog box

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Step 2: Setting limits, snap, grid, and text size

The **LIMITS** command can be invoked by entering **LIMITS** at the Command prompt.

Command: **LIMITS**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>:

Specify upper right corner <12.0000,9.0000>: **18,12**

After setting the limits, the next step is to expand the drawing display area. Use the **ZOOM** command with the **All** option to display new limits on the screen.

Now, right-click on the **Snap Mode** or **Grid Display** button in the Status Bar to display a shortcut menu. Choose the **Settings** option from the shortcut menu to display the **Drafting Settings** dialog box. Choose the **Snap and Grid** tab. Enter **0.25** and **0.25** in the **Snap X spacing** and **Snap Y spacing** edit boxes in **Snap Spacing** area, respectively. Enter **0.5** and **0.5** in the **Grid X spacing** and **Grid Y spacing** edit boxes, respectively. Then, choose **OK**.



Note

You can also use **SNAP** and **GRID** commands to set these values.

The size of the text can be changed by entering **TEXTSIZE** at the Command prompt.

Command: **TEXTSIZE**

Enter new value for TEXTSIZE <0.2000>: **0.125**

Step 3: Setting units

Choose the **Units** tool from **Application Menu > Drawing Utilities > Units** or enter **UNITS** at the Command prompt to invoke the **Drawing Units** dialog box, as shown in Figure 1-6. In the **Length** area, select **Decimal** from the **Type** drop-down list and select **0.0000** from the **Precision** drop-down list. In the **Angle** area, select **Decimal Degrees** from the **Type** drop-down list and **0.00** from the **Precision** drop-down list. Also make sure the **Clockwise** check box in the **Angle** area is not selected.

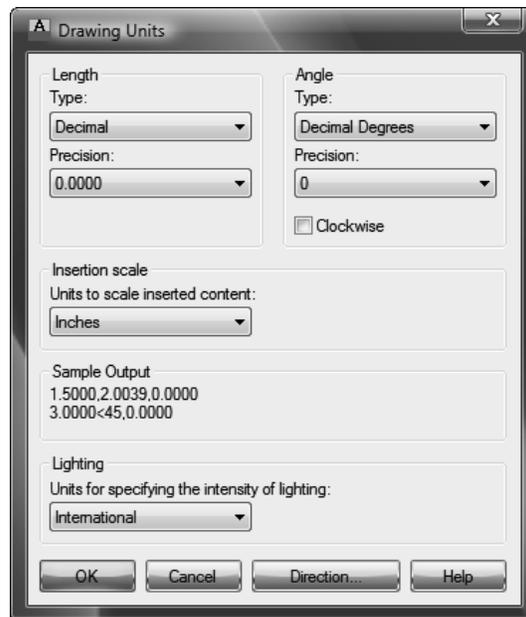


Figure 1-6 The Drawing Units dialog box

Choose the **Direction** button from the **Drawing Units** dialog box to display the **Direction Control** dialog box (Figure 1-7) and then select the **East** radio button. Exit both the dialog boxes.



Figure 1-7 The **Direction Control** dialog box

Step 4: Saving the drawing as template file

Now, save the drawing as *proto2.dwt* using the **Save** tool from the **Quick Access** toolbar. You need to select **AutoCAD Drawing Template (*.dwt)** from the **Files of type** drop-down list and enter **proto2** in the **File name** edit box in the **Save Drawing As** dialog box. Next, choose the **Save** button; the **Template Options** dialog box will be displayed on the screen. Enter the description about the template in the **Description** edit box and choose the **OK** button. The drawing will be saved as *proto2.dwt* on the default drive. You can also save this drawing to some other location by specifying other location from the **Save in** drop-down list of the **Save Drawing As** dialog box.

LOADING A TEMPLATE DRAWING

You can use the template drawing to start a new drawing file. To use the preset values of the template drawing, start AutoCAD or choose the **New** tool from the **Quick Access** toolbar. The dialog box that appears will depend on whether you have set the **STARTUP** system variable to **1** or **0**. If you have set this value as **1**, the **Create New Drawing** dialog box will appear. Choose the **Use a Template** option. All templates that are saved in the default **Template** directory will be shown in the **Select a Template** list box, see Figure 1-8. If you have saved the template in any other location, choose the **Browse** button. The **Select a template file** dialog box will be displayed. You can use this dialog box to browse the directory in which the template file is saved.

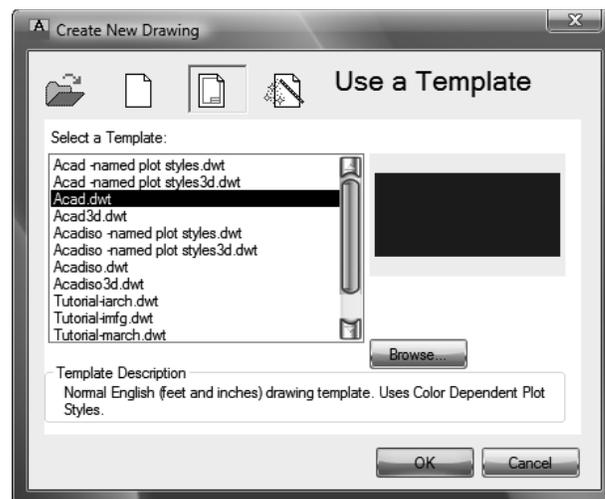


Figure 1-8 Default templates in the **Create New Drawing** dialog box

If you have set the **STARTUP** system variable to 0, the **Select template** dialog box appears when you choose the **New** tool. This dialog box also displays the default **Template** folder and all template files saved in it, see Figure 1-9. You can use this dialog box to select the template file that you want to open.

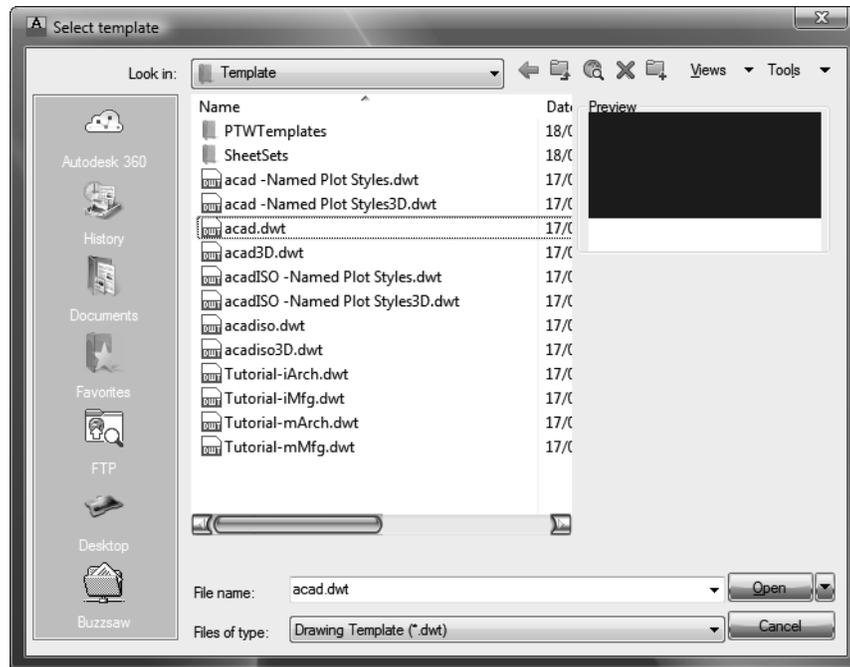


Figure 1-9 The **Select template** dialog box that appears while starting a new drawing file, when the value of **STARTUP** variable is set to 0

Using any of the previously mentioned dialog boxes, select the *proto1.dwt* template drawing. AutoCAD will start a new drawing that will have the same setup as that of the template drawing *proto1.dwt*.

You can have several template drawings, each with a different setup. For example, **PROTOB** for a 18" X 12" drawing, **PROTOC** for a 24" X 18" drawing. Each template drawing can be created according to user-defined specifications. You can then load any of these template drawings, as discussed previously.

CUSTOMIZING DRAWINGS WITH LAYERS AND DIMENSIONING SPECIFICATIONS

Most production drawings need multiple layers for different groups of objects. In addition to layers, it is a good practice to assign different colors to different layers to control the line width at the time of plotting. You can generate a template drawing that contains the desired number of layers with linetypes and colors according to your company specifications. You can then use this template drawing to make a new drawing. The next example illustrates the procedure used for customizing a drawing with layers, linetypes, and colors.

EXAMPLE 3

Template with Title Block & Layers

Create a template drawing *proto3.dwt* that has a border and the company's title block, as shown in Figure 1-10.

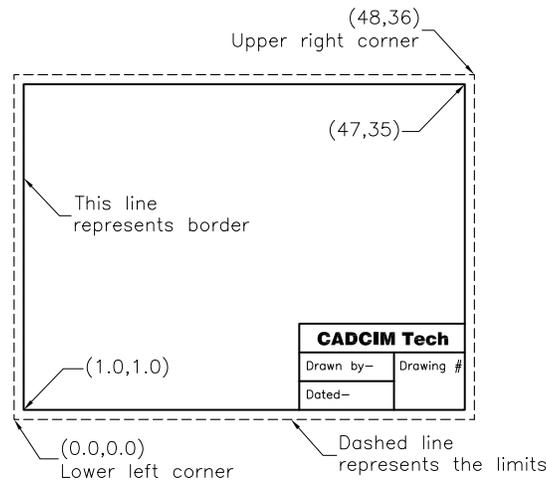


Figure 1-10 The template drawing for Example 3

This template drawing will have the following initial drawing setup:

Limits	48.0,36.0
Text height	0.25
Border line lineweight	0.012"
Ltscale	4.0

DIMENSIONS

- Overall dimension scale factor 4.0
- Dimension text above the extension line
- Dimension text aligned with dimension line

LAYERS

<u>Layer Name</u>	<u>Line Type</u>	<u>Color</u>
0	Continuous	White
OBJ	Continuous	Red
CEN	Center	Yellow
HID	Hidden	Blue
DIM	Continuous	Green
BOR	Continuous	Magenta

Step 1: Setting limits, text size, polyline width, polyline, and linetype scaling

Start a new drawing with default parameters by selecting the **Start from Scratch** option in the **Create New Drawings** dialog box. In the new drawing file, use the AutoCAD commands to set up the values as given for this example. Also, draw a border and a title block, as shown in Figure 1-10. In this figure, the hidden lines indicate drawing limits. The border lines are 1.0 units inside the drawing limits. For border lines, increase the lineweight to a value of 0.012".

Use the following procedure to produce the prototype drawing for Example 3.

1. Invoke the **LIMITS** command by entering **LIMITS** at the Command prompt. The prompt sequence is given next.

Command: **LIMITS**
 Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>:
 Specify upper right corner <12.0000,9.0000>: **48,36**

- Increase the drawing display area by choosing the **All** option of the **ZOOM** command.
- Enter **TEXTSIZE** at the Command prompt to change the text size.

Command: **TEXTSIZE**
 Enter new value for TEXTSIZE <0.2000>: **0.25**

- Next, draw the border using the **RECTANGLE** tool. The prompt sequence to draw the rectangle is:

Command: Choose the **RECTANGLE** tool from the **Draw** panel
 Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **1.0,1.0**
 Specify other corner point or [Area/Dimensions/Rotation]: **47.0,35.0**

- Now, select the rectangle and change its lineweight to **0.012"**. Make sure that the **Show/Hide Lineweight** button is chosen in the Status Bar.
- Enter **LTSCALE** at the Command prompt to change the linetype scale.

Command: **LTSCALE**
 Enter new linetype scale factor<Current>: **4.0**

Step 2: Setting dimensioning parameters

You can use the **Dimension Style Manager** dialog box to set the dimension variables. Click the inclined arrow displayed on the **Dimensions** panel title bar in the **Annotate** tab; the **Dimension Style Manager** dialog box will be displayed, as shown in Figure 1-11.

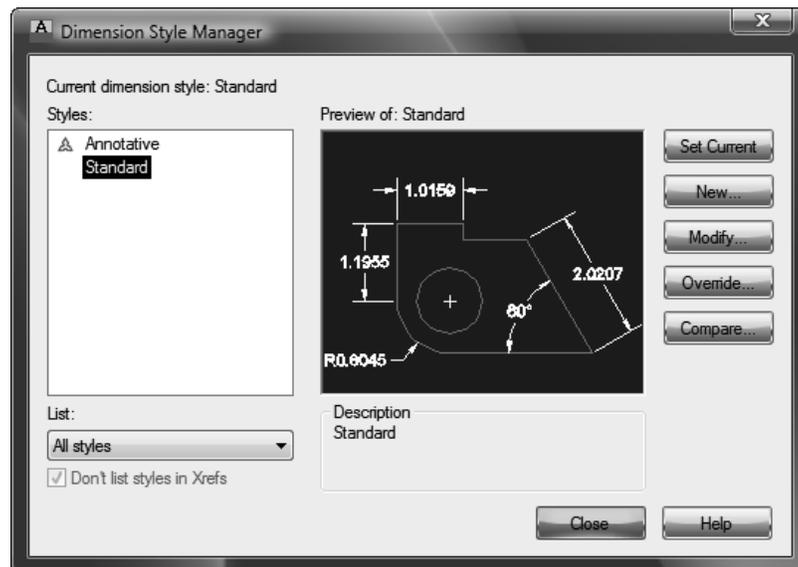


Figure 1-11 The Dimension Style Manager dialog box

You can also invoke this dialog box by entering **DIMSTYLE** at the Command prompt. Choose the **New** button from the **Dimension Style Manager** dialog box; the **Create New Dimension**

Style dialog box will be displayed. Specify the new style name as **MYDIM1** in the **New Style Name** edit box, as shown in the Figure 1-12 and then choose the **Continue** button. The **New Dimension Style:MYDIM1** dialog box is displayed.

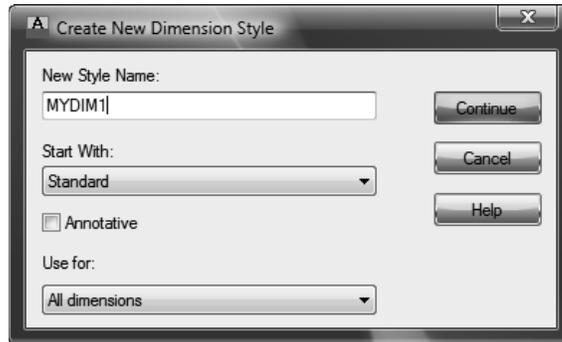


Figure 1-12 The Create New Dimension Style dialog box

Specifying Overall dimension scale factor

To specify a dimension scale factor, choose the **Fit** tab of the **New Dimension Style: MYDIM1** dialog box. Set the value in the **Use overall scale of** as **4** in the **Scale for dimension features** area, as shown in Figure 1-13.

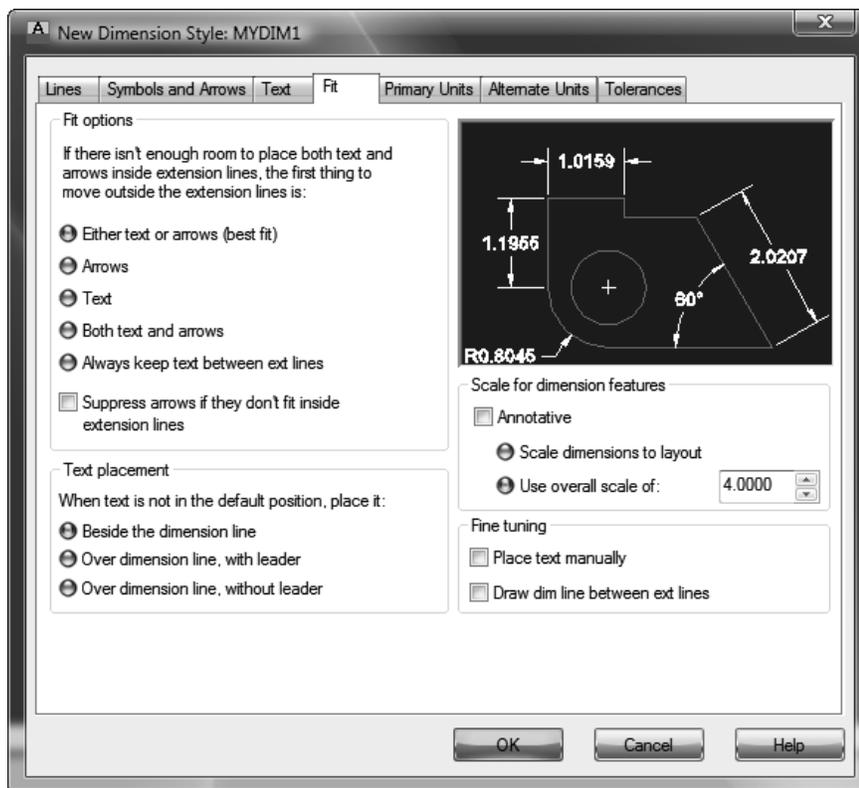


Figure 1-13 The Fit tab of the New Dimension Style: MYDIM1 dialog box

Placing the dimension text over the dimension line

In the **Fit** tab of the **New Dimension Style: MYDIM1** dialog box. Select the **Over dimension line, with leader** radio button from the **Text placement** area.

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Dimensioning text aligned with the dimension line

In the **Text Alignment** area of the **Text** tab of the **New Dimension Style: MYDIM1** dialog box, select the **Aligned with dimension line** radio button and then choose the **OK** button.

Setting the symbols and arrows

From the **Arrowheads** area of the **Symbols and Arrows** tab in the **New Dimension Style: MYDIM1** dialog box, you can select different types of arrows from the **First**, **Second**, and **Leader** drop-down lists. You can also set the size of the arrows in the **Arrow size** spinner.

Setting the dimension line type

The options in the **Dimension lines** area and **Extension lines** area of the **Lines** tab in the **New Dimension Style: MYDIM1** dialog box are used for selecting different types of dimension lines.

Setting the new dimension style to current

A new dimension style with the name **MYDIM1** is shown in the **Styles** area of the **Dimension Style Manager** dialog box. Select this dimension style and then choose the **Set Current** button to make it the current dimension style. Choose the **Close** button to exit this dialog box.

Step 3: Setting layers

Choose the **Layer Properties** tool from the **Layers** panel or choose the **Layer** tool from the **Format** menu bar to invoke the **Layer Properties Manager** dialog box. Choose the **New Layer** button in the **Layer Properties Manager** dialog box and rename Layer1 as **OBJ**. Choose the color swatch of the **OBJ** layer to display the **Select Color** dialog box. Select the **Red** color and choose **OK**; the red color is assigned to the **OBJ** layer. Again, choose the **New Layer** button in the **Layer Properties Manager** dialog box and rename the Layer1 as **CEN**. Choose the linetype swatch to display the **Select Linetype** dialog box.

If different linetypes are not already loaded, choose the **Load** button to display the **Load or Reload Linetypes** dialog box. Select the **CENTER** linetype from the **Available Linetypes** area and choose **OK**; the **Select Linetype** dialog box will reappear. Select the **CENTER** linetype from the **Loaded linetypes** area and choose **OK**. Choose the color swatch to display the **Select Color** dialog box. Select the **Yellow** color and choose **OK**; the color yellow and linetype center will be assigned to the layer **CEN**.

Similarly, different linetypes and different colors can be set for different layers mentioned in the statement of this example, as shown in Figure 1-14.

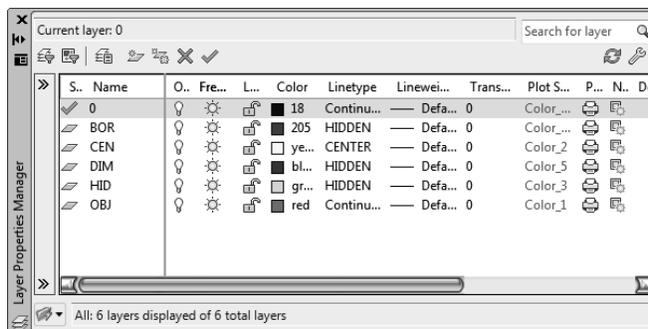


Figure 1-14 The **Layer Properties Manager** dialog box

Step 4: Adding title block

Next, add the title block and the text, as shown in Figure 1-10. After completing the drawing, save it as *proto3.dwt*. You have created a template drawing (proto3) that contains all the information given in Example 3.

CUSTOMIZING A DRAWING WITH LAYOUT

The Layout (paper space) provides a convenient way to plot multiple views of a three-dimensional (3D) drawing or multiple views of a regular two-dimensional (2D) drawing. It takes quite some time to set up the viewports in the model space with different vpoints and scale factors. You can create prototype drawings that contain predefined viewport settings, with vpoint and the other desired information. If you create a new drawing or insert a drawing, the views are automatically generated. The following example illustrates the procedure for generating a prototype drawing with paper space and model space viewports:

EXAMPLE 4

Template with Layouts

Create a drawing template, as shown in Figure 1-15, with four views in Layout3 (Paper space) that display front, top, side, and 3D views of the object. The plot size is 10.5 by 8 inches. The plot scale is 0.5 or 1/2" = 1". The paper space viewports should have the following vpoint settings:

<u>Viewports</u>	<u>Vpoint</u>	<u>View</u>
Top right	1,-1,1	3D view
Top left	0,0,1	Top view
Lower right	1,0,0	Right side view
Lower left	0,-1,0	Front view

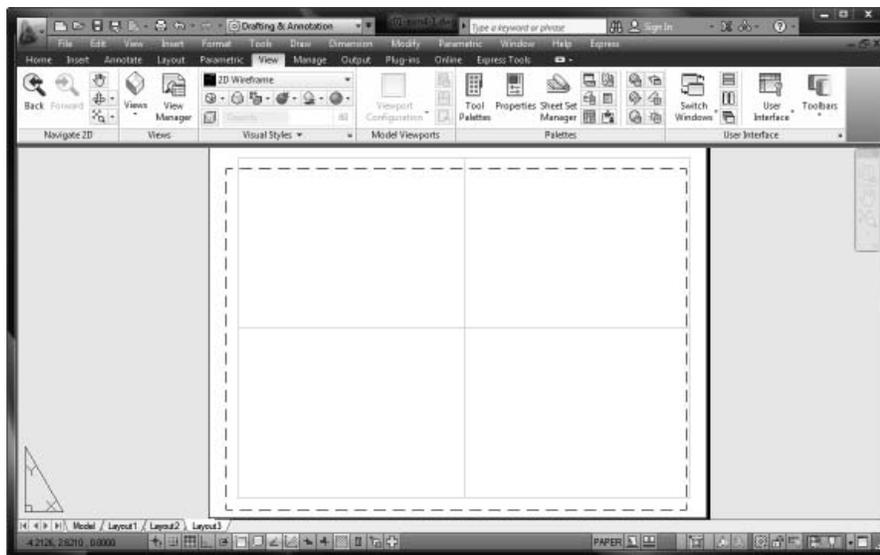


Figure 1-15 Paper space with four viewports

Start AutoCAD and create a new drawing. Use the following commands and options to set various parameters:

Step 1: Creating a new layout

First, you need to create a new layout using the **LAYOUT** command. You can also choose **New Layout** tool from the **Quick View Layouts** panel which is available on choosing the **Quick View Layouts** button in the Status Bar. A new layout is automatically created with the default name. Alternatively, you can also use the **LAYOUT** command to create a new layout. The Command prompt that is displayed is given next.

Command: **LAYOUT**

Enter layout option [Copy/Delete/New/Template/Rename/Saveas/Set/?] <set>: **N**

Enter new Layout name <Layout3>:

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Step 2: Specifying the page setup for the new layout

Next, choose the new layout (**Layout3**) tab. The new layout (**Layout3**) will be displayed with the default viewport. Delete this existing viewport. Invoke the **Page Setup Manager** dialog box from the shortcut menu that will be displayed when you right click on the **Quick View Layouts** button in the Status Bar and choose the **Modify** button to modify the default page setup. In the **Page Setup - Layout3** dialog box, select the required printer from the **Printer/plotter** area. In this example, HP LaserJet 4000 has been used.

Now, from the **Paper size** area, select the paper size that is supported by the selected plotting device. In this example, the paper size is A4. Choose the **OK** button to accept the settings and return to the **Page Setup Manager** dialog box. Choose the **Close** button to close the **Page Setup Manager** dialog box.

Step 3: Creating new layer for the viewports object

Next, you need to set up a layer with the name VIEW for viewports object and assign it green color. Invoke the **Layer Properties Manager** dialog box. Choose the **New Layer** button and rename the Layer1 as VIEW. Choose the color swatch of the VIEW layer to display the **Select Color** dialog box. Select the color **Green** and choose the **OK** button. This color will be assigned to **View** layer. Also, make the VIEW layer current and then exit the **Layer Properties Manager** dialog box.

Step 4: Creating viewports

To create four viewports, choose the **4 Viewports** tool from **View > Viewports** in the menu bar or enter **MVIEW** command at the Command prompt. The following is the prompt sequence when you invoke the **MVIEW** command.

```
Command: Choose the 4Viewports tool from View > Viewports in the menu bar
Specify corner of viewport or
[ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/LAyer/2/3/4] <Fit>: _4
Specify first corner or [Fit] <Fit>: 0.25,0.25
Specify opposite corner: 10.25,7.75
```

Step 5: Setting the required viewpoint in all layers

Choose the **PAPER** button in the Status Bar to activate the model space or enter **MSPACE** at the Command prompt.

```
Command: MSPACE (or MS)
```

Make the lower left viewport active by clicking on it. Next, you need to change the viewpoints of different paper space viewports using the **VPOINT** command. To invoke this command, choose the **Viewpoint** tool from **View > 3D Views** in the menu bar or enter **VPOINT** at the Command prompt. The viewpoint values for different viewports are given in the statement of Example 4. To set the view point for the lower left viewport the Command prompt sequence is given next.

```
Command: Choose the Viewpoint tool from View > 3D Views in the menu bar
Current view direction: VIEWDIR= 0.0000,0.0000,1.0000
Specify a view point or [Rotate] <display compass and tripod>: 0,-1,0
```

Similarly, use the **VPOINT** command to set the viewpoint of the other viewports.

**Note**

You can also use the ViewCube or the SteeringWheel to orient the view in any viewport.

Make the top left viewport active by selecting a point in the viewport and then use the **ZOOM** command to specify the paper space scale factor to 0.5. The **ZOOM** command can be invoked by choosing **View > Zoom > Scale** from the **Application Menu** or by entering **ZOOM** at the Command prompt. The Command prompt sequence is given next.

Command: **ZOOM**

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **0.5XP**

Now, make the next viewport active and specify the zoom scale factor. Do the same for the remaining viewports.

Step 6: Creating border line

Use the **MODEL** button in the Status Bar to change to paper space and then set a new layer **PBORDER** with yellow color. Make the **PBORDER** layer current, draw a border, and if needed, a title block using the **PLINE** command. You can also change to paper space by entering **PSPACE** at the Command prompt.

The **PLINE** command can be invoked by choosing the **Polyline** tool from the **Draw** menu or by choosing the **Polyline** tool from the **Draw** panel. The **PLINE** command can also be invoked by entering **PLINE** at the Command prompt. While specifying the coordinate values for the **PLINE** command, make sure that the **Dynamic Input** button is turned off in the Status Bar.

Command: *Choose the **Polyline** tool from the **Draw** panel*

Specify start point: **0,0**

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **0,8.0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **10.5,8.0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **10.5,0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **C**

Step 7: Saving and testing the template file

The last step is to choose the **Model** tab (or change the **TILEMODE** to 1), if not already active and save the prototype drawing as template. To test the layout that you just created, make the 3D drawing, as shown in Figure 1-19 or make any 3D object. Switch to **Layout 3** tab; you will find four different views of the object (Figure 1-16). If the object views do not appear in the viewports, use the **PAN** commands to position the views in the viewports. You can freeze the **VIEW** layer so that the viewports do not appear on the drawing. You can plot this drawing from the **Layout3** with a plot scale factor of 1:1 and the size of the plot will be exactly as specified.

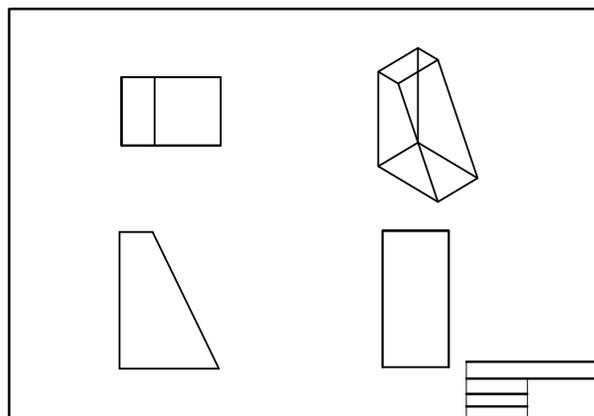


Figure 1-16 Four views of a 3D object in paper space

CUSTOMIZING DRAWINGS WITH VIEWPORTS

In certain applications, you may need multiple model space viewport configurations to display different views of an object. This involves setting up the desired viewports and then changing the viewpoint for different viewports. You can create a prototype drawing that contains a required number of viewports and the viewpoint information. If you insert a 3D object in one of the viewports of the prototype drawing, you will automatically get different views of the object without setting viewports or viewpoints. The following example illustrates the procedure for creating a prototype drawing with a standard number (four) of viewports and viewpoints.

EXAMPLE 5

Template with Viewports

Create a prototype drawing with four viewports, as shown in Figure 1-17. The viewports should have the following viewpoints:

<u>Viewports</u>	<u>Vpoint</u>	<u>View</u>
Top right	1,-1,1	3D view
Top left	0,0,1	Top view
Lower right	1,0,0	Right side view
Lower left	0,-1,0	Front view

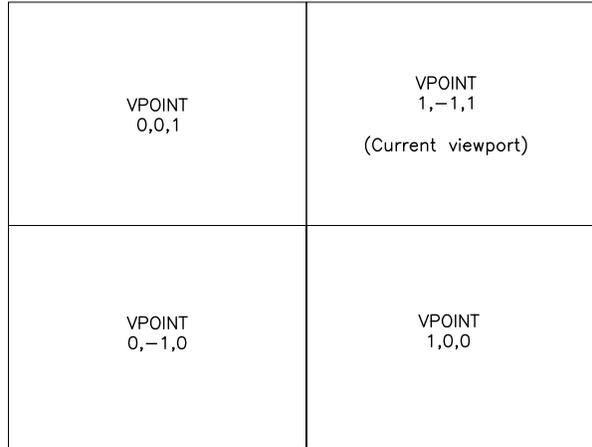


Figure 1-17 Viewports with different viewpoints

Step 1

Start AutoCAD and create a new drawing from scratch.

Step 2

Setting viewports in Model tab

Viewports and corresponding viewpoints can be set with the **VPORTS** command. You can also choose the **New Viewports** tool from **View > Viewports** in the menu bar or choose the **Named** tool from **View > Model Viewports** panel to display the **Viewports** dialog box, as shown in Figure 1-18. Select **Four: Equal** from the **Standard viewports** area. In the **Preview** area four equal viewports are displayed. Select **3D** from the **Setup** drop-down list. The four viewports with the different viewpoints will be displayed in the **Preview** area as Top, Front, Right and SE Isometric. **Top** represents the viewpoints as (0,0,1), **Front** represents the viewpoints as (0,-1,0), **Right** represents the viewpoints as (1,0,0) and **SE Isometric**, represents the viewpoints as (1,-1,1) respectively. Choose the **OK** button. Save the drawing as *proto5.dwt*. You can also set the type of visible style of the model in each individual viewports by selecting each viewport and setting the visible style in the **Visible Style** drop-down list of the **Preview** area.

Viewports and viewpoints can also be set by entering **-VPOR**T and **VPOINT** at the Command prompt, respectively.



Figure 1-18 The Viewports dialog box

Step 3

Start a new drawing and draw the 3D tapered block, as shown in Figure 1-19.

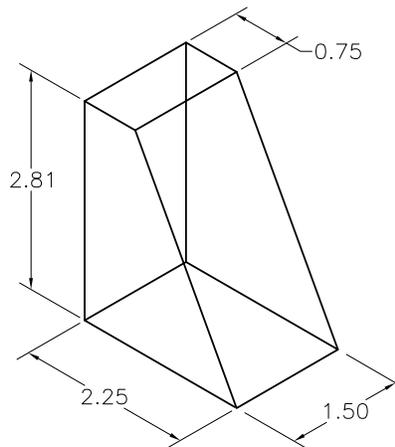


Figure 1-19 3D tapered block

Step 4

Again, start a new drawing, TEST, using the prototype drawing *proto5.dwt*. Make the top right viewport current and then insert or create a drawing, refer to Figure 1-19. Four different views will be automatically displayed on the screen, as shown in Figure 1-20.



Note

The method of creating 3D models is discussed in Chapter 25 (Getting Started with 3D) of *AutoCAD: A Problem-Solving Approach: 2013 and beyond* textbook by Prof. Sham Tickoo.

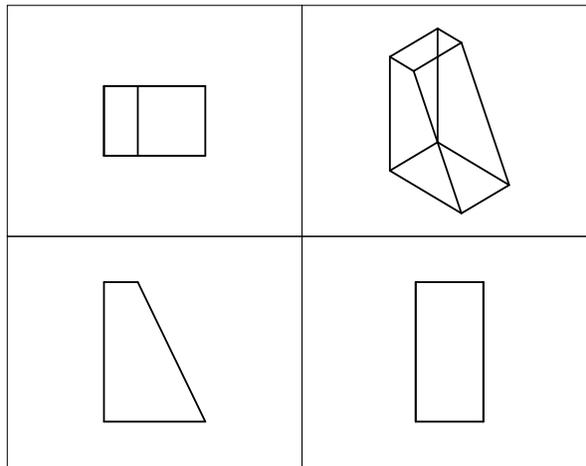


Figure 1-20 Different views of a 3D tapered block

CUSTOMIZING DRAWINGS ACCORDING TO PLOT SIZE AND DRAWING SCALE

For controlling the plot area, it is recommended to use layouts. You can make the drawing of any size, use the layout to specify the sheet size, and then draw the border and title block. However, you can also plot a drawing in the model space and set up the system variables so that the plotted drawing is according to your specifications. You can generate a template drawing according to plot size and scale. For example, if the scale is $1/16'' = 1'$ and the drawing is to be plotted on a 36" by 24" area, you can calculate drawing parameters like limits, **DIMSCALE**, and **LTSCALE** and save them in a template drawing. This will save considerable time in the initial drawing setup and provide uniformity in the drawings. The next example explains the procedure involved in customizing a drawing according to a certain plot size and scale.



Note

You can also use the paper space to specify the paper size and scale.

EXAMPLE 6 Template with Plot Size and Drawing Scale

Create a drawing template (PROTO6) with the following specifications:

Plotted sheet size	36" by 24" (Figure 1-21)
Scale	$1/8'' = 1.0'$
Snap	3'
Grid	6'
Text height	1/4" on plotted drawing
Linetype scale	Calculate
Dimscale factor	Calculate
Units	Architectural
	Precision, 16-denominator of the smallest fraction
	Angle in degrees/minutes/seconds
	Precision, 0d00'
	Direction control, base angle, east
	Angle positive, if measured counterclockwise
Border	Border should be 1" inside the edges of the plotted drawing sheet, using PLINE 1/32" wide when plotted (Figure 1-21)

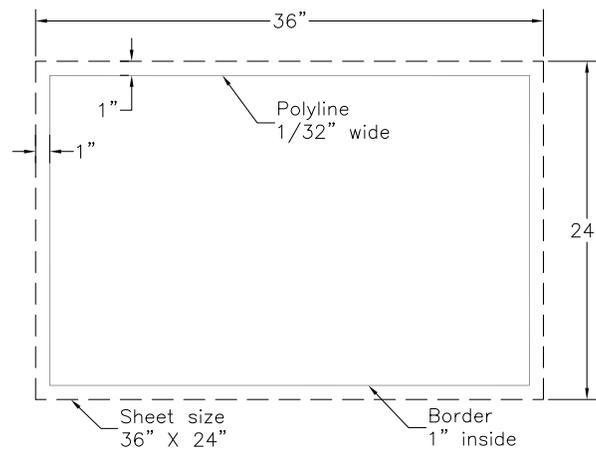


Figure 1-21 Border of the template drawing

Step 1: Calculating limits, text height, linetype scale, dimension scale, and polyline width

In this example, you need to calculate some values before you set the parameters. The limits of the drawing depend on the plotted size of the drawing and its scale. Similarly, **LTSCALE** and **DIMSCALE** depend on the plot scale of the drawing. The following calculations explain the procedure for finding the values of limits, ltscale, dimscale, and text height.

Limits

Given:

Sheet size 36" x 24"
 Scale 1/8" = 1'
 or 1" = 8'

Calculate:

X Limit

Y Limit

Since sheet size is 36" x 24" and scale is 1/8" = 1'

Therefore, X Limit = 36 x 8' = 288'
 Y Limit = 24 x 8' = 192'

Text height

Given:

Text height when plotted = 1/4"
 Scale 1/8" = 1'

Calculate:

Text height

Since scale is 1/8" = 1'

or 1/8" = 12"

or 1" = 96"

Therefore, scale factor = 96

Text height = 1/4" x 96
 = 24" = 2'

Linetype scale and dimension scaleKnown:

Since scale is $1/8'' = 1'$
 or $1/8'' = 12''$
 or $1'' = 96''$

Calculate:

LTSCALE and DIMSCALE

Since scale factor = 96

Therefore, LTSCALE = Scale factor = 96

Similarly, DIMSCALE = 96

(All dimension variables, like DIMTXT and DIMASZ, will be multiplied by 96.)

Polyline WidthGiven:

Scale is $1/8'' = 1'$

Calculate:

PLINE width

Since scale is $1/8'' = 1'$
 or $1'' = 8'$
 or $1'' = 96''$

Therefore,

PLINE width = $1/32 \times 96$
 = 3''

After calculating the parameters, use the following AutoCAD commands to set up the drawing and save the drawing as *proto6.dwt*.

Step 2: Setting units

Start a new drawing and enter **UNITS** at the Command prompt to display the **Drawing Units** dialog box. Select **Architectural** from the **Type** drop-down list in the **Length** area. Select **0'-01/16''** from the **Precision** drop-down list. Make sure the **Clockwise** check box in the **Angle** area is not selected. Select **Deg/Min/Sec** from the **Type** drop-down list and select **0d00'** from the **Precision** drop-down list in the **Angle** area. Now, choose the **Direction** button to display the **Direction Control** dialog box. Select the **East** radio button, if it is not selected, in the **Base Angle** area and then choose the **OK** button twice to close both the dialog boxes.

Step 3: Setting limits, snap and grid, textsize, linetype scale, dimension scale, dimension style and pline

To set limits, enter **LIMITS** at the Command prompt.

Command: **LIMITS**

Specify lower left corner or [ON/OFF] <0'-0",0'-0">: **0,0**

Specify upper right corner <1'-0",0'-9">: **288',192'**

Invoke the **All** option of the **ZOOM** command to increase the drawing display area.

Right-click on the **Snap Mode** or **Grid Display** button in the Status Bar to invoke the shortcut menu. In the shortcut menu, choose **Settings** to display the **Drafting Settings** dialog box. In this dialog box, enter **3'** in both the **Snap X spacing** and **Snap Y spacing** edit boxes. Similarly, enter **6'** in both the **Grid X spacing** and **Grid Y spacing** edit boxes. Then, choose **OK**.

You can also set these values by entering **SNAP** and **GRID** at the Command prompt.

The size of the text can be changed by entering **TEXTSIZE** at the Command prompt.

Command: **TEXTSIZE**

Enter new value for TEXTSIZE <current>: **2'**

To set the **LTSCALE**, choose the **Other** option from **Home > Properties > Linetype** drop-down list, or choose the **Linetype** option from the **Format** menu bar, or enter **LINETYPE** at the Command prompt; the **Linetype Manager** dialog box will be invoked. Choose the **Show details** button; the **Linetype Manager** dialog box will be expanded. Now, specify the **Global scale factor** as **96** in the **Global scale factor** edit box. Choose the **OK** button to accept the changes and exit the **Linetype Manager** dialog box.

You can also change the scale of the linetype by entering **LTSCALE** at the Command prompt.

Choose the inclined arrow displayed on the **Dimensions** panel title bar from the **Annotate** tab to invoke the **Dimension Style Manager** dialog box, refer to Figure 1-11. Choose the **New** button from the **Dimension Style Manager** dialog box to invoke the **Create New Dimension Style** dialog box. Specify the new style name as **MYDIM2** in the **New Style Name** edit box and then choose the **Continue** button. The **New Dimension Style: MYDIM2** dialog box will be displayed. Choose the **Fit** tab and set the value in the **Use overall scale of** spinner to **96** in the **Scale for dimension features** area. Now, choose the **OK** button to display the **Dimension Style Manager** dialog box. Choose **Close** button to exit the dialog box.

You can invoke the **PLINE** command by choosing the **Polyline** tool from the **Draw** panel or entering **PLINE** at the Command prompt. While specifying the coordinate values for the **PLINE** command, make sure that the **Dynamic Input** button is turned off in the Status Bar.

Command: *Choose the **Polyline** tool from the **Draw** panel*

Specify start point: **8',8'**

Current line-width is 0'-0"

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **W**

Specify starting width <0'-0">: **3**

Specify ending width <0'-3">:

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **280,0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **0,184**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **-280,0**

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **C**

Now, save the drawing as *proto6.dwt*.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The template drawings are saved in the _____ format.
2. To use a template file, select the _____ option in the **Create New Drawing** dialog box.
3. To start a drawing with the default setup, select the _____ option in the **Create New Drawing** dialog box.

4. If the plot size is 36" x 24", and the scale is 1/2" = 1', then X Limit = _____ and Y Limit = _____.
5. _____ command is used to set up a viewport in the paper space.

Review Questions

Answer the following questions:

1. The default value of **DIMSCALE** is _____.
2. The default value of **DIMTXT** is _____.
3. The default value of **SNAP** is _____.
4. Architectural units can be selected by using the _____ command or the _____ command.
5. The three standard template drawings that come with AutoCAD software are _____, _____, and _____.
6. If the plot size is 24" x 18", and the scale is 1:20, the X Limit = _____ and Y Limit = _____.
7. If the plot size is 200 x 150 and limits are (0.00,0.00) and (600.00,450.00), the **LTSCALE** factor = _____.
8. _____ provides a convenient way to plot multiple views of a 3D drawing or multiple views of a regular 2D drawing.
9. You can use the _____ command to change to paper space.
10. You can use AutoCAD's _____ command to change to model space.
11. The values that can be assigned to **TILEMODE** are _____ and _____.
12. In the model space, if you want to reduce the display size by half, the scale factor you enter in the **ZOOM > Scale** option is _____.

EXERCISE 1

Template with Limits

Create a drawing template (*protoe1.dwt*) with the following specifications:

Units	Architectural with precision 0'-0 1/16
Angle	Decimal Degrees with precision 0
Base angle	East
Angle direction	Counterclockwise
Limits	48' x 36'

EXERCISE 2

Template with Limits, Text Height, & Units

Create a drawing template (*protoe2.dwt*) with the following specifications:

Limits	36.0,24.0
Snap	0.5
Grid	1.0
Text height	0.25
Units	Decimal
	Precision 0.00
	Decimal degrees
	Precision 0
	Base angle, East
	Angle positive if measured counterclockwise

EXERCISE 3 *Template with Layers, LTSCALE, & DIMSCALE*

Create a drawing template (*protoe3.dwt*) with the following specifications:

Limits	48.0,36.0
Text height	0.25
PLINE width	0.03
LTSCALE	4.0
DIMSCALE	4.0
Plot size	10.5 x 8

LAYERS

<u>Layer Name</u>	<u>Line Type</u>	<u>Color</u>
0	Continuous	White
OBJECT	Continuous	Green
CENTER	Center	Magenta
HIDDEN	Hidden	Blue
DIM	Continuous	Red
BORDER	Continuous	Cyan

EXERCISE 4 *Relative Rectangular and Absolute Coordinates*

Create a prototype drawing (*protoe4.dwt*) with the following specifications:

Limits	36.0,24.0
Border	35.0,23.0
Grid	1.0
Snap	0.5
Text height	0.15
Units	Decimal (up to 2 places)
LTSCALE	1
Current layer	Object

LAYERS

<u>Layer Name</u>	<u>Linetype</u>	<u>Color</u>
0	Continuous	White
Object	Continuous	Red
Hidden	Hidden	Yellow
Center	Center	Green
Dim	Continuous	Blue
Border	Continuous	Magenta
Notes	Continuous	White

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This prototype drawing should have a border line and a title block, as shown in Figure 1-22.

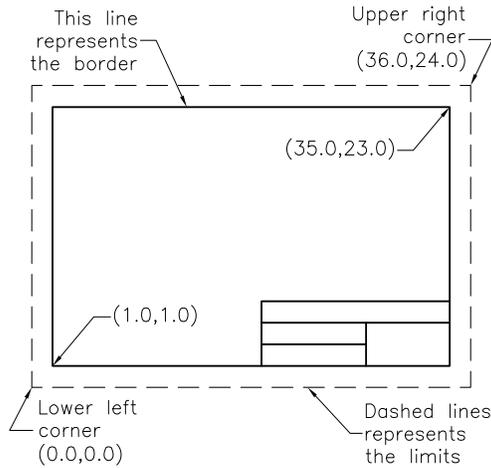


Figure 1-22 Prototype drawing

EXERCISE 5 Template with Plot Sheet Size & Title Block

Create the template drawing shown in Figure 1-23 with the following specifications and save it with the name *protoe5.dwt*:

- Plotted sheet size 36" x 24" (Figure 1-23)
- Scale 1/2" = 1.0'
- Text height 1/4" on plotted drawing
- LTSCALE 24
- DIMSCALE 24
- Units Architectural
- 32-denominator of smallest fraction to display
- Angle in degrees/minutes/seconds
- Precision 0d00"00"
- Angle positive if measured counterclockwise
- Border Border is 1-1/2" inside the edges of the plotted drawing sheet, using the PLINE 1/32" wide when plotted.

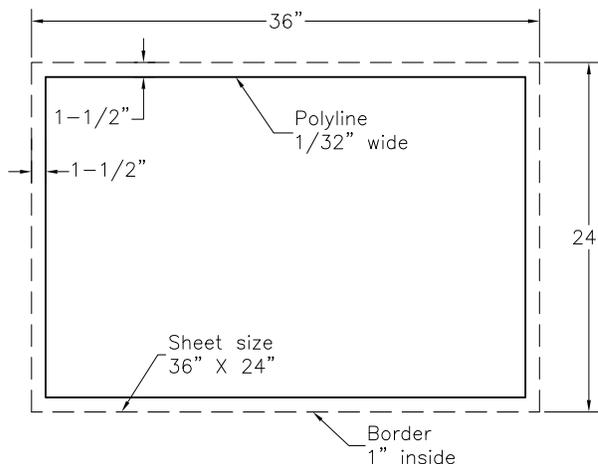


Figure 1-23 Drawing for Exercise 5

EXERCISE 6 Template with Title Block & Dimension Style

Create the prototype drawing, as shown in Figure 1-24 with the following specifications (the name of the drawing is *protoe6.dwt*):

- Plotted sheet size 24" x 18" (Figure 1-24)
- Scale 1/2" = 1.0'
- Border The border is 1" inside the edges of the plotted drawing sheet, using the PLINE 0.05" wide when plotted (Figure 1-24)
- Dimension text over the dimension line
- Dimensions aligned with the dimension line
- Calculate overall dimension scale factor
- Enable the display of alternate units
- Dimensions to be associative.

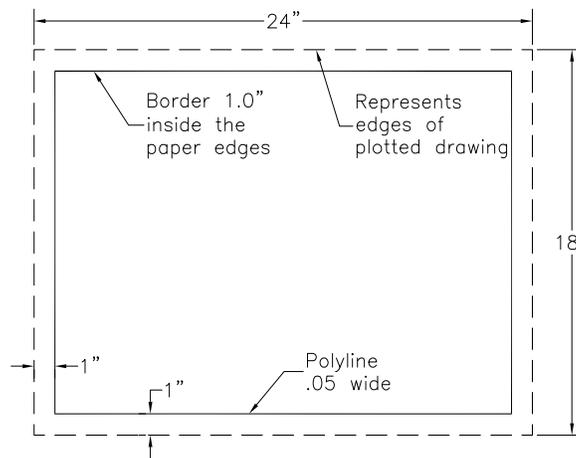


Figure 1-24 Prototype drawing

EXERCISE 7 Creating & using Template for Generating Drawing

Create the template with the necessary specifications to draw the Articulated Rod shown in Figure 1-25, with the following specifications.

- Limits 36.0,24.0
- Border 35.0,23.0
- Grid 1.0
- Snap 0.5
- Text height 0.15
- Units Decimal (up to 2 places)
- LTSCALE 1
- Current layer Object

LAYERS

Layer Name

- 0
- Object
- Hidden

Linetype

- Continuous
- Continuous
- Hidden

Color

- White
- Red
- Yellow

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Center	Center	Green
Dim	Continuous	Blue
Border	Continuous	Magenta
Notes	Continuous	White

Also, use this template and the layers created in it to draw the views of the Articulated Rod.

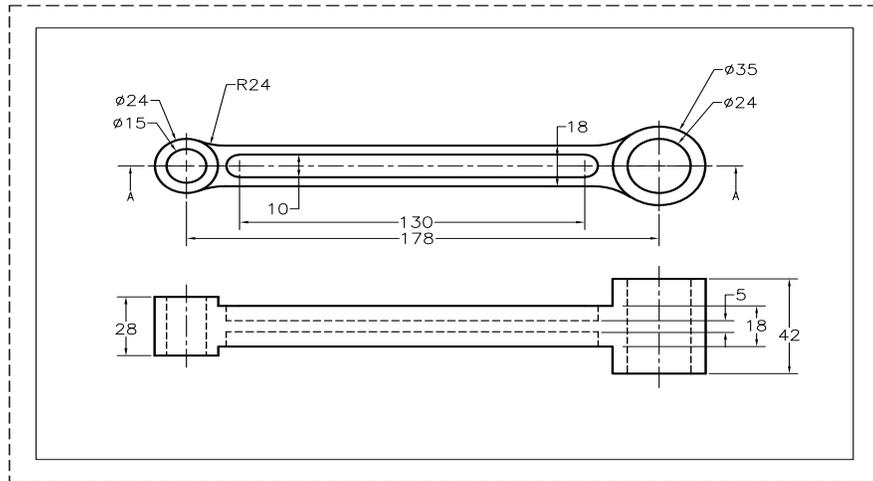


Figure 1-25 Views and dimensions of the Articulated Rod

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Answers to Self-Evaluation Test

1. *.dwt*, 2. Use a Template, 3. Start from Scratch, 4. 72", 48", 5. MVIEW.