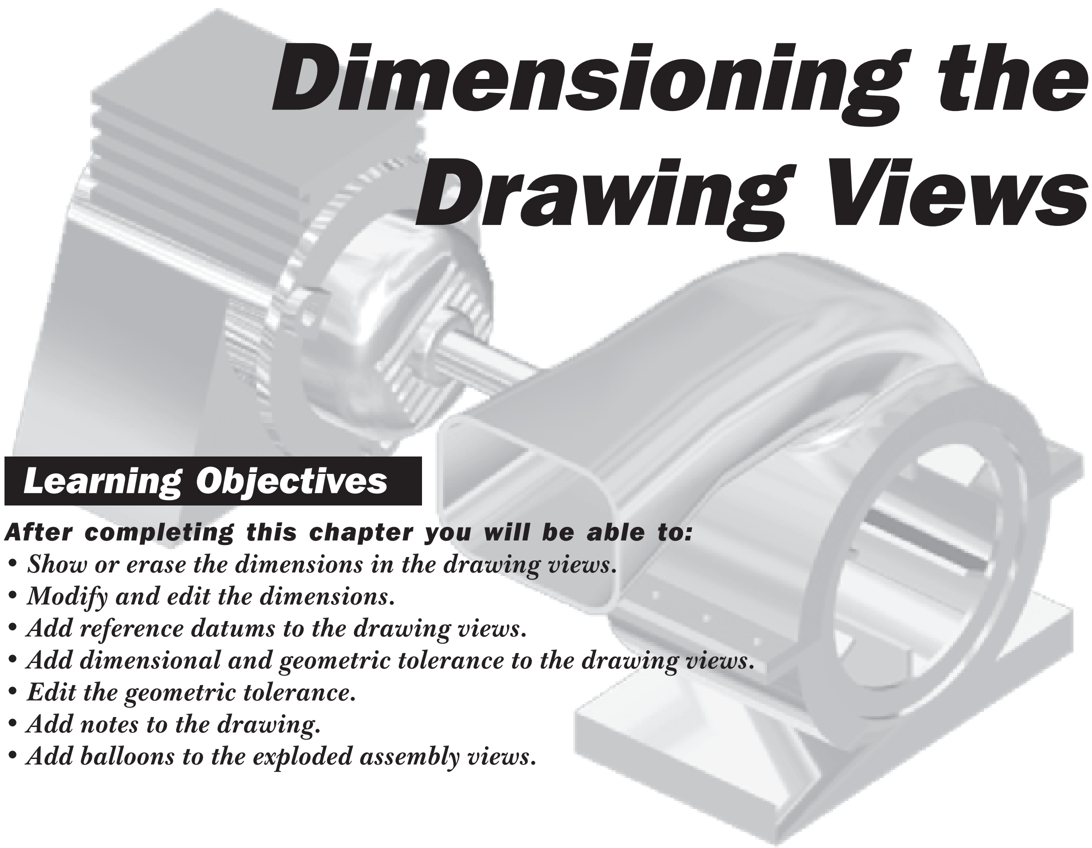
Chapter 11

Dimensioning the Drawing Views



Learning Objectives

After completing this chapter you will be able to:
- *Show or erase the dimensions in the drawing views.*
- *Modify and edit the dimensions.*
- *Add reference datums to the drawing views.*
- *Add dimensional and geometric tolerance to the drawing views.*
- *Edit the geometric tolerance.*
- *Add notes to the drawing.*
- *Add balloons to the exploded assembly views.*

DIMENSIONING THE DRAWING VIEWS

Once you have generated the drawing views, you need to generate the dimensions in the drawing views, and add notes, symbols, balloons, and so on. These dimensions are the dimensions assigned to each of the entities that created the model or to the features associated with the model. This is done by using the **Show / Erase** dialog box shown in Figure 11-1. The **Show / Erase** dialog box is displayed when you choose the **Show and Erase** option from the **VIEW** menu in the menu bar.

Show / Erase Dialog Box Options

Show

The **Show** button is chosen to select the options to generate the various parameters related to the model in the drawing views. The options that are displayed when you choose this button are discussed next.

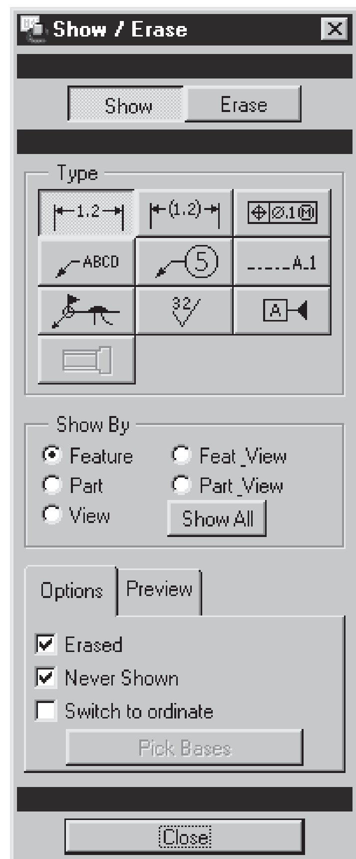
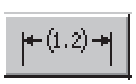


*Figure 11-1 Show / Erase dialog box with the **Show** tab*

Type Area

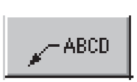
The options to display the different parameters will be available only when you select at least one button from this area.

Dimension. The **Dimension** button is chosen to generate the parametric dimensions in the drawing views. The bidirectional associative nature of this software package is valid when you change the dimension value generated using this option.

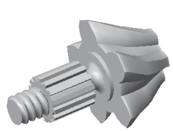


Reference Dimension. The **Reference Dimension** button is chosen to generate the reference dimensions in the drawing views.

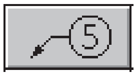
Geometric Tolerance. The **Geometric Tolerance** button is chosen to display the geometric tolerances in the dimensions.



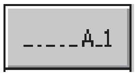
Note. The **Note** button is chosen to display the notes that were created in the Part mode. If any change is made to a note in the Drawing mode, the note in the Part mode is automatically modified. Similarly, if you modify this note in the Part mode, the modifications are reflected automatically in the Drawing mode.



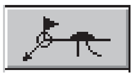
Tip: *The dimensional tolerance will be displayed only when the **tol_display** is set to **YES** in the configuration file. This file is displayed when you choose **DRAWING > Advanced > Draw Setup** from the **Menu Manager**.*



Balloon. The **Balloon** button is chosen to display the balloons associated with an assembly in the drawing views.

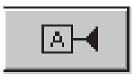


Axis. The **Axis** button is chosen to display the axis of the selected hole or cylindrical feature in the drawing views.

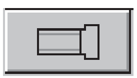


Symbol. The **Symbol** button is chosen to display any symbol associated with the model.

Surface Finish. The **Surface Finish** button is chosen to display the surface finish symbols associated with the part in the drawing views.



Datum Plane. The **Datum Plane** button is chosen to display the reference datum planes in the drawing views.



Cosmetic Feature. The **Cosmetic Feature** button is chosen to display the cosmetic features in the drawing views that are associated with the model.

Show By Area

The options in this area are discussed next.

Feature. The **Feature** radio button is chosen to display the required annotation of the selected feature. If the required annotation of the selected feature cannot be displayed in the view in which the feature was selected then the required annotation will be displayed in the appropriate views.

Part. The **Part** radio button is chosen to display all the required annotations of the selected part. The required annotations are displayed in all the views irrespective of the view in which the part was selected.

View. The **View** radio button is chosen to display all the required annotations that can be displayed in the selected view.

Feat_View. The **Feat_View** radio button is chosen to display the required annotations of the selected feature that can be displayed in the selected view.

Part_View. The **Part_View** radio button is chosen to display all the required annotations of the selected part that can be displayed in the selected view.

Show All. The **Show All** button is chosen to show all the required annotations in the drawing views. When you choose this button all the required annotations that were used in creating the model are displayed in the drawing views.

Options Tab

The options under this tab of the **Show / Erase** dialog box are discussed next.

Erased. The **Erased** check box is selected to display the annotations that were once erased. If this check box is cleared, the annotations that were once erased using the **Show / Erase** dialog box will not be displayed the next time you wish to display them.

Never Shown. The **Never Shown** check box is selected to display the annotations that are never displayed in the drawing views. At least one of the **Erased** or the **Never Shown** check box remains selected while specifying the annotation.

Switch to ordinate. The **Switch to ordinate** check box is selected to switch to the ordinate mode of dimensioning. When you select this check box, you will be prompted to select a dimension that will be the baseline dimension for the other ordinate dimensions.

Preview Tab

The options under this tab of the **Show / Erase** dialog box are discussed next.

With Preview. The **With Preview** check box is selected to allow you to confirm the placement of the annotation. If this check box is selected then when you show the annotations, you will be prompted to confirm the annotations that you want to keep or delete. You can select the annotations to keep, to delete, accept all, or delete all using the buttons provided under this area. The remaining options under this area will be available only after you place the annotations. If this check box is cleared then you will not be prompted to confirm the placement of the annotation. All the displayed annotations will be automatically placed in the drawing views.

Erase

The options under the **Erase** button are shown in Figure 11-2. These options are used to erase the annotations placed using the options under the **Show** button. The annotations once placed in the drawing view can also be erased by selecting it. When the selected annotation turns red in color, right-click and choose the **Erase** option from the shortcut menu that appears. The selected annotation gets erased from the drawing view.

Note

*The annotations erased using the **Erase** options are not deleted from the model.*

Type Area

The buttons provided under this tab are similar to those under the **Type** area of the **Show** button. The only difference is that here these buttons are used to erase the items.

Erase By Area

The options in this area are discussed next.

Selected Items. If the **Selected Items** radio button is selected then you can select the annotations to be erased using the left mouse button.

Feature. If the **Feature** radio button is selected then all the annotations of the feature that you select will be erased from all the views.

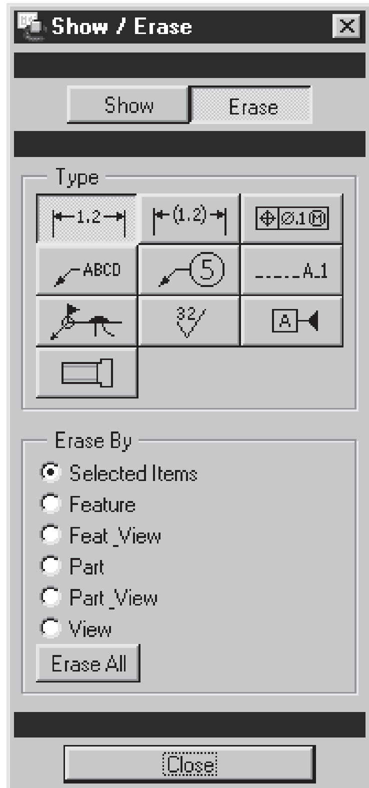


Figure 11-2 ***Show / Erase*** *dialog box with the* ***Erase*** *tab*

Feat_View. If the **Feat_View** radio button is selected then all the annotations related to the selected feature will be erased from the selected view.

Part. If the **Part** radio button is selected then all the annotations related to the selected part will be erased from all the drawing views.

Part_View. If the **Part_View** radio button is selected then all the annotations related to the selected part will be erased from the selected view.

View. The **View** radio button is selected to erase all the annotations displayed in the selected view.

Erase All. The **Erase All** button is chosen to erase all the annotations displayed in all the views of the current sheet. When you choose this button, you will be prompted to confirm that all the annotations can be erased.

MODIFYING AND EDITING THE DIMENSIONS

Pro/ENGINEER allows you to edit and modify the dimensions assigned to the drawing views. There are various methods of editing and modifying these dimensions. Figure 11-3 shows the

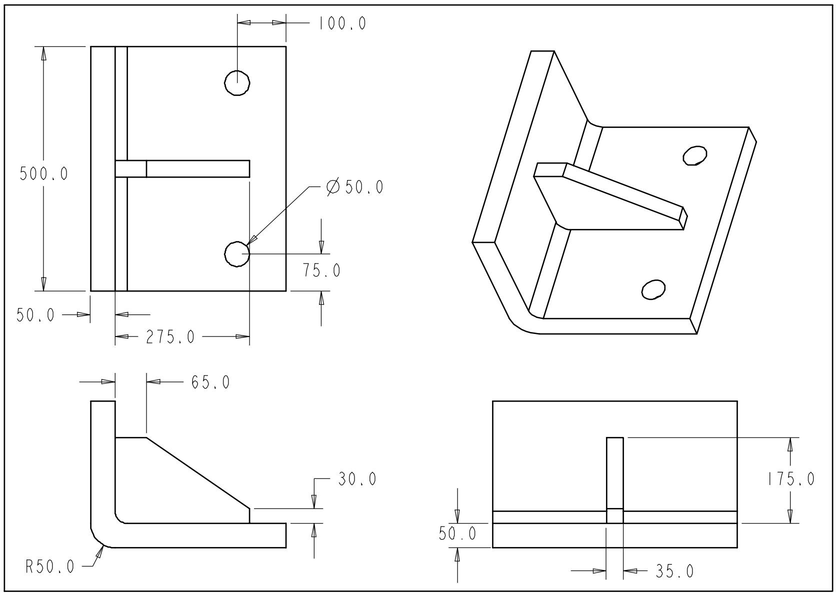
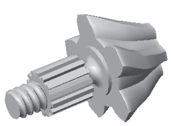


Figure 11-3 *Dimensioned drawing views*

dimensioned drawing views. But before these methods are discussed, it is important for you to learn the terminology used in these methods. Figure 11-4 shows the terminology that is used in the Drawing mode of Pro/ENGINEER.



Tip: *To display the dimensions in a 3D view, first you will have to set the value of **allow_3d_dimensions** to **YES** in the configuration file. If this value is set to **NO** then the dimensions will not be displayed in 3D views.*

Modifying the Dimensions using the Dimension Properties dialog box

The **Properties** option is available only when the dimension to be modified is selected on the drawing sheet. The **Dimension Properties** dialog box is displayed when you choose the **Properties** option from the **Edit** menu in the menu bar. The **Dimension Properties** dialog box can also be displayed when you select the dimension and hold down the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu. The **Dimension Properties** dialog box is displayed as shown in Figure 11-5. The options in this dialog box are discussed next.

Properties Tab

Value and Tolerance Area

The options provided under this area are related to the tolerance in the drawing.

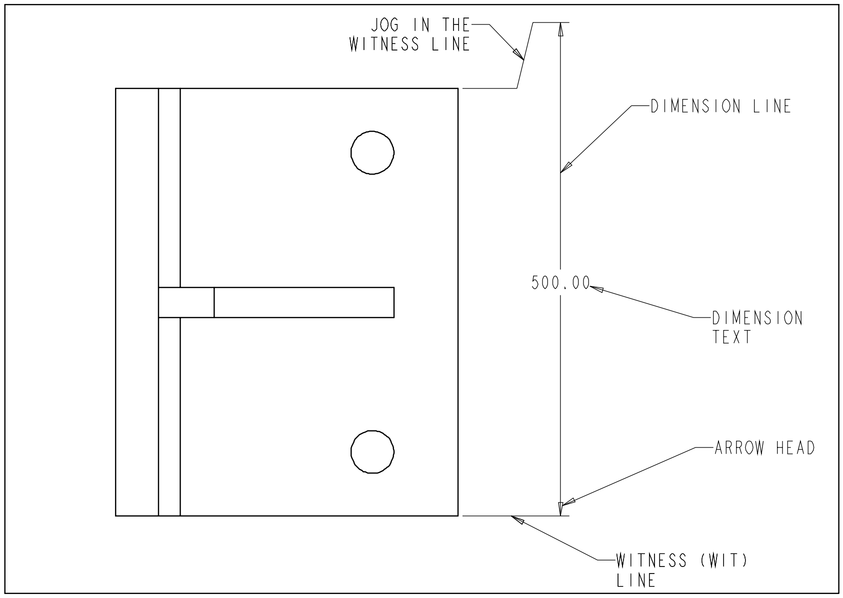


Figure 11-4 *Parameters associated with the dimensions*

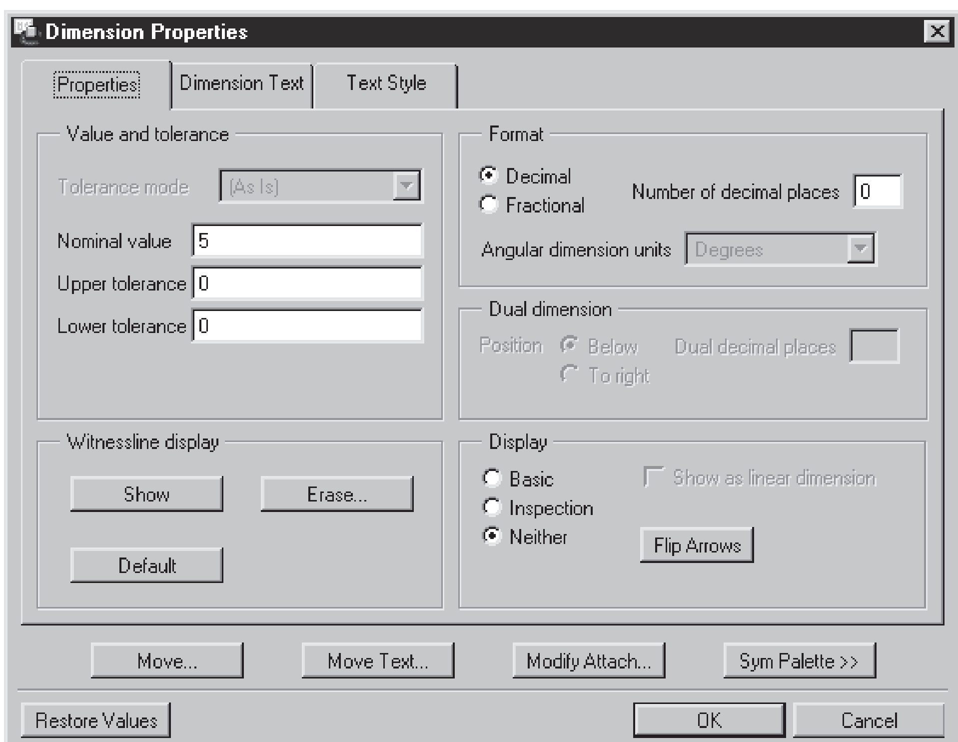


Figure 11-5 *The* ***Properties*** *tab of the* ***Dimension Properties*** *dialog box*

Tolerance Mode. This drop-down list will be available only when the value of **tol_display** is set to **YES** in the configuration file. The options under this drop-down list are used to select the type of tolerance mode. Depending upon the option selected from this drop-down list, the other options in this area change.

> **Tip**: *If you modify the nominal value of a dimension in the* ***Value and Tolerance*** *area, you will have to regenerate the model using the* ***DRAWING > Regenerate > Model*** *from the* ***Menu Manager*** *to view the effect of modifications.*

Format Area

Decimal. The **Decimal** radio button is chosen to display the dimensions in the decimal format. The number of decimal values can be set in the **Number of decimal places** edit box.

Fractional. The **Fractional** radio button is chosen to display the dimensions in the fractional format. The largest denominator can be set using the **Largest denominator** edit box that is displayed when you select the **Fractional** radio button.

Dual dimension Area

The options under the **Dual dimension** area will be available only when the value of **dual_dimensioning** is set to **YES** in the configuration file. The options under this area are used to specify the placement point for the dual dimensioning and the number of the decimal places.

Display Area

Basic. If the **Basic** check box is selected then the selected dimension values will be displayed inside a rectangular box.

Inspection. If the **Inspection** check box is selected then the selected dimension will be displayed inside an elliptical box.

Dimension Text Tab

The **Dimension Properties** dialog box with the options under the **Dimension Text** tab is shown in Figure 11-6.

Dimension Text Area

The **Dimension text** box displays the text of the dimension. You can edit the text using this box.

Name

The **Name** edit box display the dimension symbol. You can change the dimension symbol using this box.

Prefix

The **Prefix** edit box is used to add some additional text as prefix to the default dimension text.

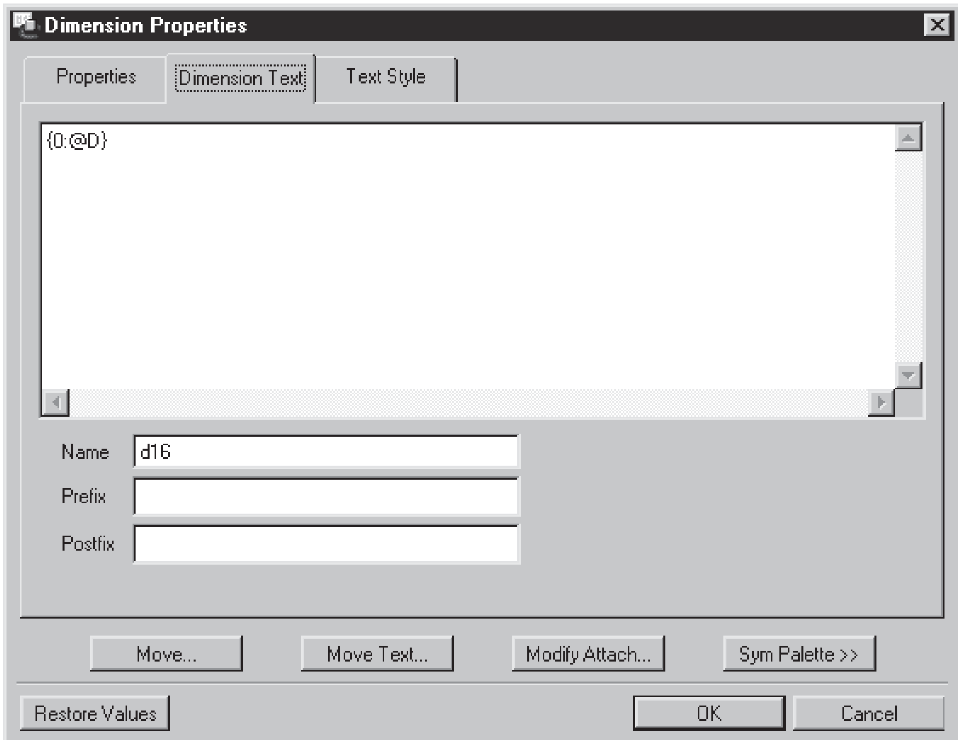


***Figure 11-6** The **Dimension Text** tab of the **Dimension Properties** dialog box*

Postfix

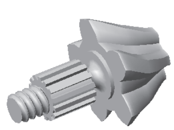
The **Postfix** edit box is used to add some additional text as suffix to the default dimension text.

Move

The **Move** button is chosen to move the dimension to a new location on the current sheet. You can also move the dimension by selecting it and then moving it by moving the mouse.

Move Text

The **Move Text** button is chosen to move the dimension text to a new location on the screen. Remember that only the text will be moved. The witness lines and the dimension lines do not move from their original location.



Tip: *The modifications made using the **Dimension Properties** dialog box are applied dynamically on the dimensions. You can view this in the background by moving the **Dimension Properties** dialog box on the side of the screen.*

Modify Attach

The **Modify Attach** button is used to modify the attachment point of dimensions of some features like draft, round, and chamfer.

Sym Palette

The **Sym Palette** button is chosen to display the **Symbol Palette** dialog box using which you can add special symbols to the default dimension text.

Modifying the drawing items using the Shortcut Menu

When you select an item from a drawing view and hold down the right mouse button to invoke the shortcut menu, various options are displayed in the shortcut menu. The options in the shortcut menu depends upon the item selected to modify and varies from item to item. Some of the items related to the drawing that can be modified using the shortcut menu are discussed next.

Value

You can modify the dimension value. However, to view the effect of the modification, you will have to regenerate the model using **DRAWING** > **Regenerate** > **Model** from the **Menu Manager**. This option is also used to modify the scale of the drawing view and tolerance values.

Number of Decimal Places

To modify the number of decimal values for the dimension text, choose the **Decimal Places** option from the **Format** menu in the menu bar. You will be prompted to specify the number of decimal places in the **Message Input Window**. After you have specified the value, you will be prompted to select the dimensions to be modified.

Miscellaneous Modifications

The modifications that can be made using the shortcut menu are: move a dimension, flip the arrows of the dimensions, switch the selected dimension from one view to the other, make a jog in the witness line, modify the values or text of the dimension, modify the arrow style, break the witness line, remove the break from the witness line, erase the witness line, show the erased witness line, or erase the selected dimension from the drawing view.

Text

The parameters that can be modified are the selected text line, the entire text material, text height, text style, style library for the text, and the current style.

Cleaning Up the Dimensions

Generally, the dimensions displayed in the drawing views are scattered and improperly placed. This in turn makes the drawing views look very untidy. Pro/ENGINEER allows you to place these dimensions in proper order using just one single step. The **Clean Dimensions** dialog box allows the user to set a standard for displaying the dimensions in the drawing views. When you choose the **Cleanup dimensions** option from the **Edit** menu in the menu bar, the **Clean Dimensions** dialog box will be displayed as shown in Figure 11-7 and you will be prompted to select the dimensions to be cleaned. The options in this dialog box are discussed next.

Placement Tab

The options under this tab of the **Clean Dimensions** dialog box are used for placing the dimensions. The distance between the two dimensions, the distance between the view boundary and the dimension, are set under this tab.

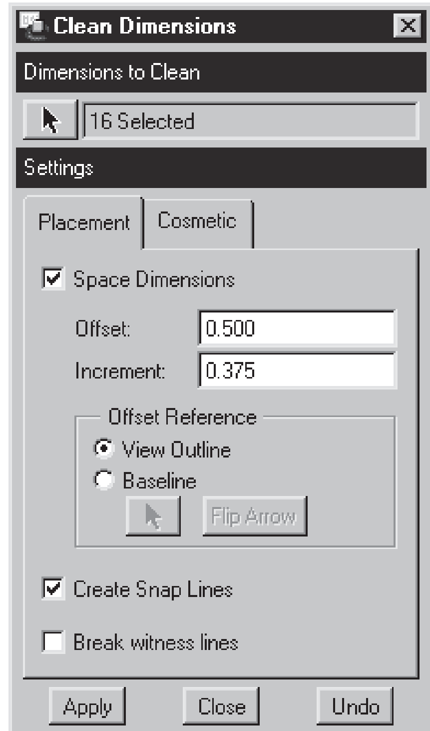


***Figure 11-7** The **Placement** tab of the **Clean Dimensions** dialog box*

Space Dimensions
The **Space Dimensions** option is used to maintain the distance between two dimensions.

Offset
The **Offset** edit box is used to specify the offset distance between the first dimension and the view boundary.

Increment
The **Increment** edit box is used to specify the offset distance between the two dimension lines.

Offset Reference Area
View Outline. The **View Outline** radio button is selected to specify the offset distances taking the outline of the view as reference.

Baseline. The **Baseline** radio button is used to specify the offset distance taking a selected flat edge, datum plane, snap line, detail axis line, or view border as the reference. When you invoke this option, you will be prompted to select either of the above-mentioned entities as the reference.

Create Snap Lines
The **Create Snap Lines** check box is selected to create the snap lines using which the dimensions will be cleaned. The snap lines will be created at the distances specified using the **Offset** and the **Increment** edit boxes. If this check box is cleared then the snap lines will not be created.

Break witness lines
The **Break witness lines** check box is selected to break the witness lines if some dimensions are overlapping each other.

Cosmetic Tab
The options under the **Cosmetic** tab of the **Clean Dimensions** dialog box shown in Figure 11-8 are used for determining a dimension value's location with the dimension line. The arrows and the orientation of the dimension text can also be controlled using this tab.

Flip Arrows
The **Flip Arrows** check box is selected to flip the arrows of the dimensions that are not easily adjusted inside the dimension lines.

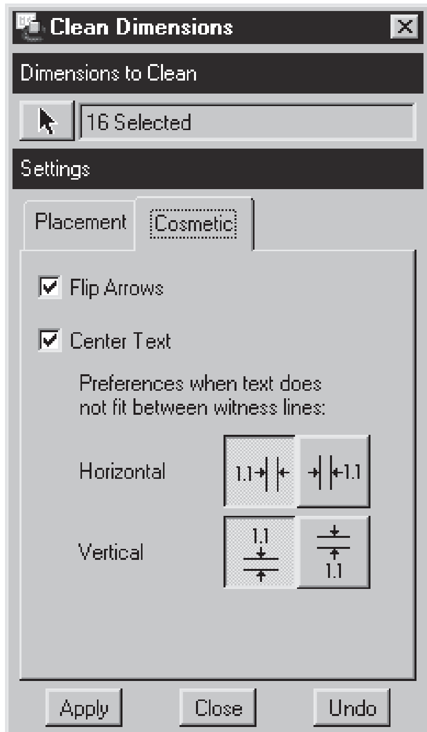


Figure 11-8 *The* ***Cosmetic*** *tab of the* ***Clean Dimensions*** *dialog box*

Center Text

The **Center Text** check box is selected to place the dimensions at the center of the dimension lines.

Horizontal/Vertical

These buttons are used to specify the placement of the horizontal or vertical dimension text outside the dimension line if they do not fit.

Figure 11-9 and Figure 11-10 explain the use of **Clean Dimensions** dialog box.

ADDING REFERENCE DATUMS TO THE DRAWING VIEWS

Reference datums are used as references for geometric tolerance. Datums are to be set before adding the geometric tolerance.

To create a reference datum, choose **Insert** > **Datum** > **Plane** from the menu bar. The **Datum** dialog box will be displayed, as shown in Figure 11-11. You can rename the selected datum and then choose the second button from the **Type** area to enclose the datum inside a feature control frame. You can place these reference datums freely in the drawing views or in a selected dimension. Figure 11-12 shows the drawing views with reference datums.

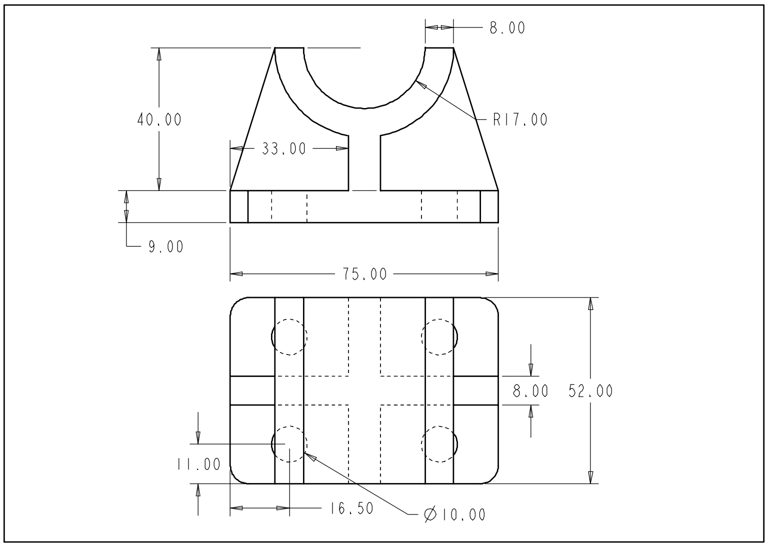


Figure 11-9 *Figure showing the drawing views with dimensions before cleaning*

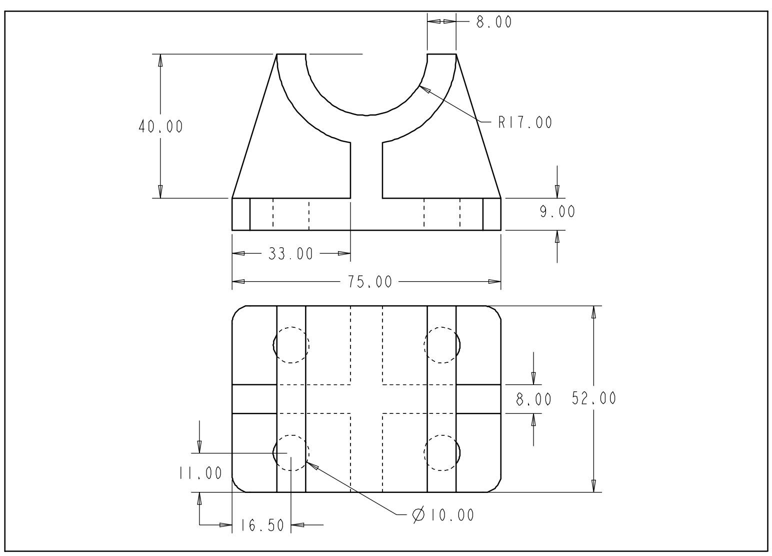


Figure 11-10 *The drawing views with dimensions after cleaning*

Note

The system does not show a reference datum plane in a drawing view unless it is perpendicular to the graphics screen.

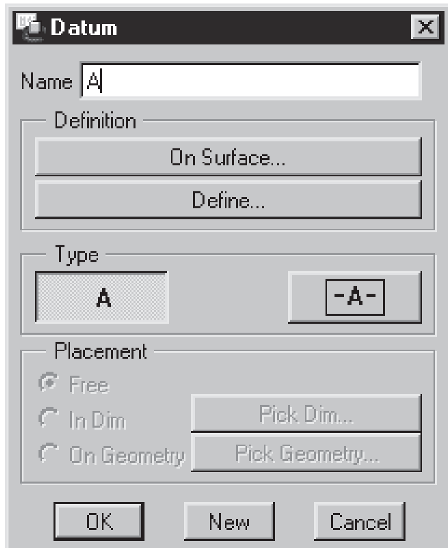


Figure 11-11 ***Datum*** *dialog box*

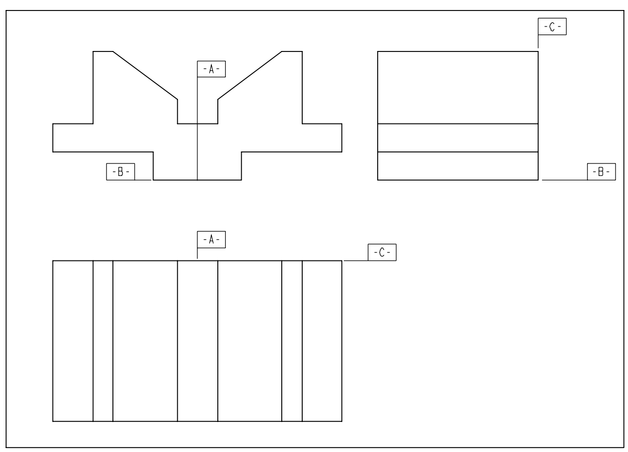


Figure 11-12 *Drawing views with reference datums*

ADDING TOLERANCES IN THE DRAWING VIEWS

Tolerance is defined as the difference between the maximum and minimum variations in the dimensions of the selected component. It is almost impossible to manufacture a component to the exact dimensions that have a very large precision. In such cases, the tolerance value is added to dimensions to make sure that some variation that occurs during manufacturing can be taken care of. However, when you actually send a part for manufacturing, there are other parameters along with the dimension tolerances that vary and that need some tolerances. Depending upon these factors, the tolerances are divided into two types: the dimensional tolerances and the geometric tolerances.

Dimensional Tolerances

These are the variation in the standard dimensional values. The dimensional tolerances can be easily displayed along with the dimensions in the drawing views by simply setting the value of **tol_display** to **YES** in the configuration file. You can select and modify the type of dimensional tolerances from the **Properties** tab of the **Dimension Properties** dialog box. Figure 11-13 shows the drawing views with the tolerance added to the dimensions.

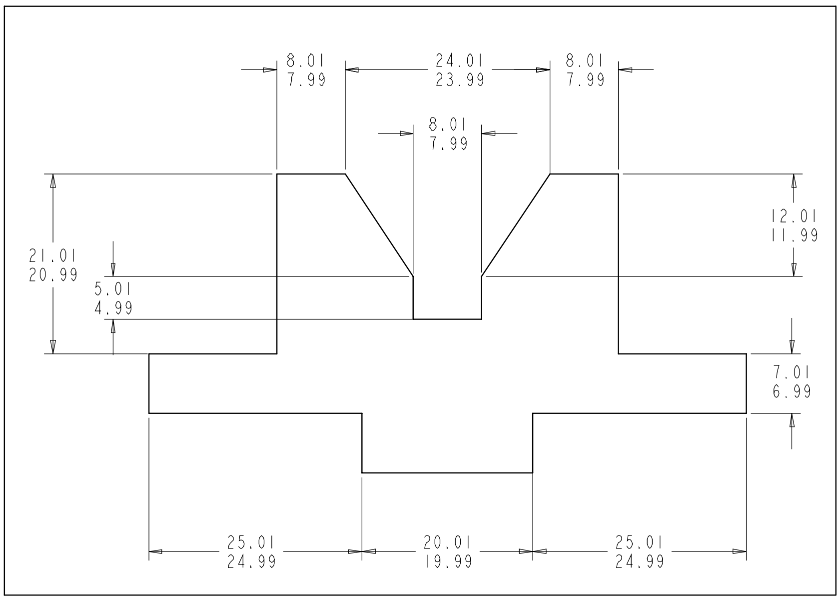


Figure 11-13 *Figure showing dimensions with tolerances*

Geometric Tolerances

When you send a component for manufacturing then you have to provide various other parameters along with the dimensions and dimensional tolerances. These parameters can be the geometric condition, the surface profile, the material condition, and so on. All these parameters are defined using the geometric tolerances. The geometric tolerances (will be called gtol henceforth) can be added to the drawing using the **Geometric Tolerance** dialog box, shown in Figure 11-14. The **Geometric Tolerance** dialog box is displayed when you choose **Insert > Geometric Tolerance** from the menu bar.

The options in the **Geometric Tolerance** dialog box are discussed next.

Model Refs Tab

The options under this tab are used to select the model or drawing where the gtol have to be placed and to specify the model references.

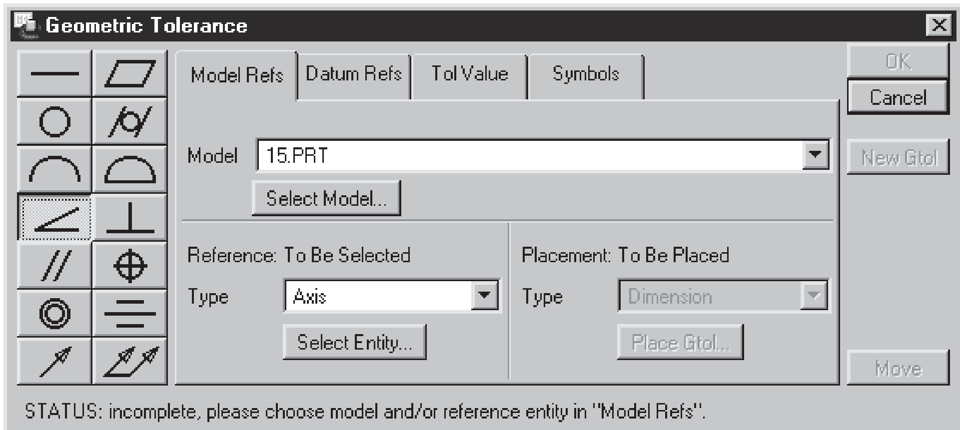


***Figure 11-14** The **Model Refs** tab of the **Geometric Tolerance** dialog box*

Model

This drop-down list is used to select the model on which you need to display the gtol. You can select the model from this drop-down list or select it by using the **Select Model** button.

Reference

The options under this area are used to select the reference for adding the gtol. The reference can be an edge, axis, surface, feature, datum, or an entity. You can select the reference type from the **Type** drop-down list or select it using the **Select Entity** button.

Placement

The options under this area will be available only when you select a reference type for placing the gtol. These options are used to specify the placement type for the gtol. The placement type can be selected from the **Type** drop-down list.

Datum Refs Tab

The options under the **Datum Refs** tab, shown in Figure 11-15, are used to select the primary, secondary, or tertiary datum references for the gtol. You can also specify the composite tolerances to the primary, secondary, or tertiary datum references by using the **Composite Reference** check box provided in this area. The **Composite Tolerance** check box is available only when the **Position** or the **Surface Profile** buttons from the types of gtol buttons available on the left side of the **Geometric Tolerance** dialog box are selected. The value of the composite reference can be specified in the **Value** edit box that is available only when you select the **Composite Reference** check box. The datum reference for the composite tolerance can be selected using the **Datum Reference** drop-down list that is available only when you select the **Composite Reference** check box.

Tol Value Tab

The options under the **Tol Value** tab, shown in Figure 11-16, are used to specify the tolerance value and the material condition for the gtol. You can specify the overall tolerance value or the per units tolerance value using the options under this tab. You can also specify the material condition using the **Material Condition** drop-down list.

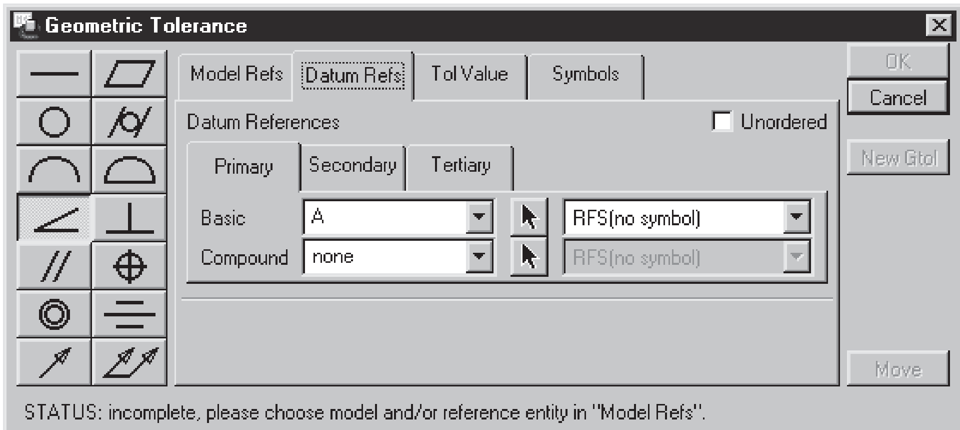


Figure 11-15 *The **Datum Refs** tab of the **Geometric Tolerance** dialog box*

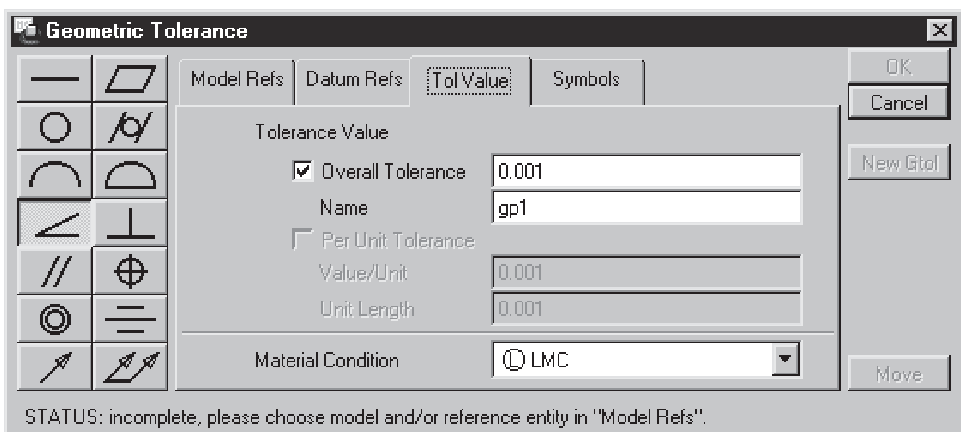


Figure 11-16 *The **Tol Value** tab of the **Geometric Tolerance** dialog box*

Symbols Tab

The options under the **Symbols** tab, shown in Figure 11-17, are used to specify the symbols for the gtol. The symbols and the modifiers for the gtol can be selected from the **Symbols and Modifiers** area and the projected tolerance zone symbol can be selected from the **Projected Tolerance Zone** area. The location of the projected tolerance zone symbol and its height can be specified using the options provided under this area. You can also specify the profile boundary symbol for the profile gtol from the **Profile Boundary** area that appears in the place to **Projected Tolerance Zone** area. **Profile Boundary** area is only available if you choose **Surface Profile or Line Profile** button.

New Gtol

The **New Gtol** button is chosen to accept the current gtol and place a new gtol.

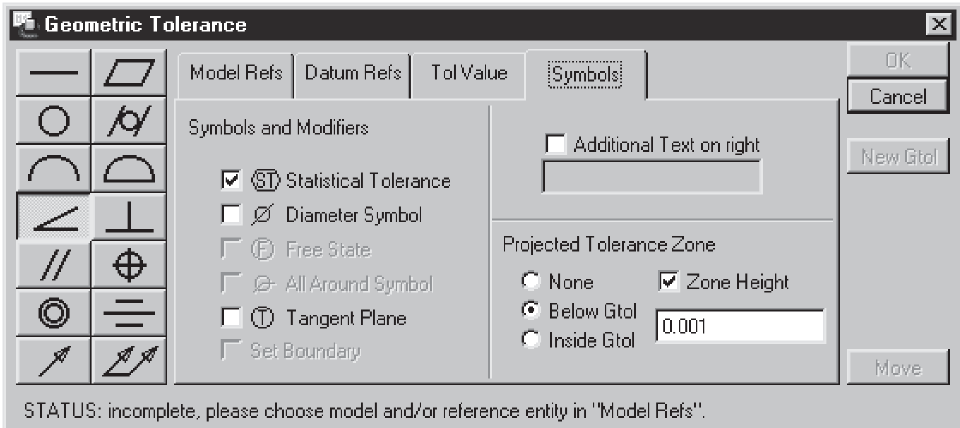


Figure 11-17 *The* ***Symbols*** *tab of the* ***Geometric Tolerance*** *dialog box*

Move

The **Move** button is chosen to change the location of the current gtol.

Figure 11-18 shows the drawing view with reference datums and gtol.

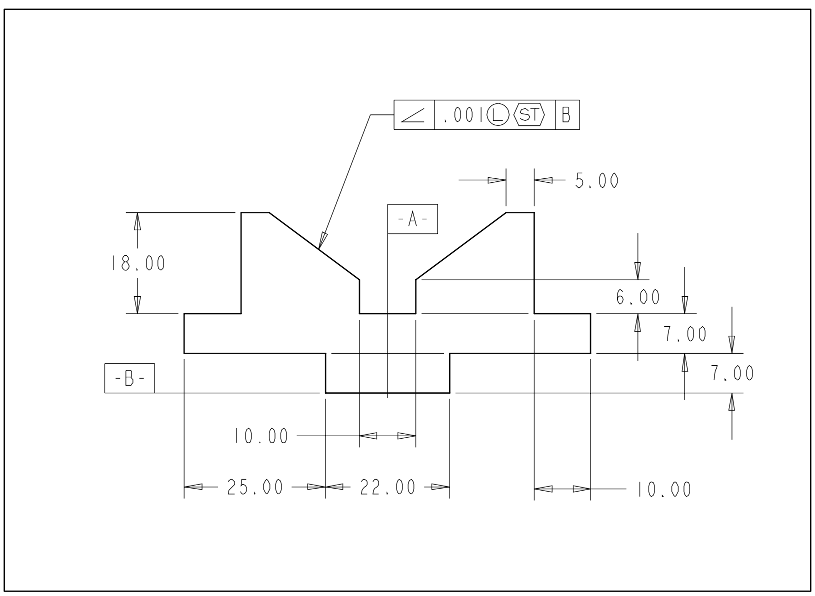


Figure 11-18 *The drawing view with reference datums and gtol*

EDITING THE GEOMETRIC TOLERANCES

The gtol added to the drawing views can be edited using the **Geometric Tolerance** dialog box. This dialog box is displayed when you select the text of the tolerance from the drawing view and then right-click to display the shortcut menu. Choose the **Properties** option from the shortcut menu to display the **Geometric Tolerance** dialog box. All the options under the different tabs of the **Geometric Tolerance** dialog box are similar to those discussed in the **Geometric Tolerance** dialog box for adding the gtol.

ADDING NOTES TO THE DRAWING

The notes added to the model in the Part mode can be easily displayed in the drawing views using the **Show / Erase** dialog box. However, when you have not added notes to the model in the Part mode and still want to add them in the drawing views then you will have to create them. The notes can be created by choosing **Insert > Note** from the menu bar. When you choose this option, the **NOTE TYPES** submenu will be displayed, see Figure 11-19. In the **NOTE TYPES** submenu, choose the type of leader to be used if required. Choose the **Make Note** option from this submenu and you are prompted to select the location for the note placement, choose a point on the screen and enter the text in the **Message Input Window** and press ENTER. The note will be placed on the point selected.

***Figure 11-19 NOTE TYPES** submenu*

Note

*You can also browse the text file saved in a .txt format using the **NOTE TYPES** submenu.*

ADDING BALLOONS TO THE ASSEMBLY VIEWS

The balloons can be added to the assembly drawing views by choosing **Insert** > **Balloon** from the menu bar. The procedure to place balloons is similar to that used while adding notes to the drawing. See Figure 11-20.

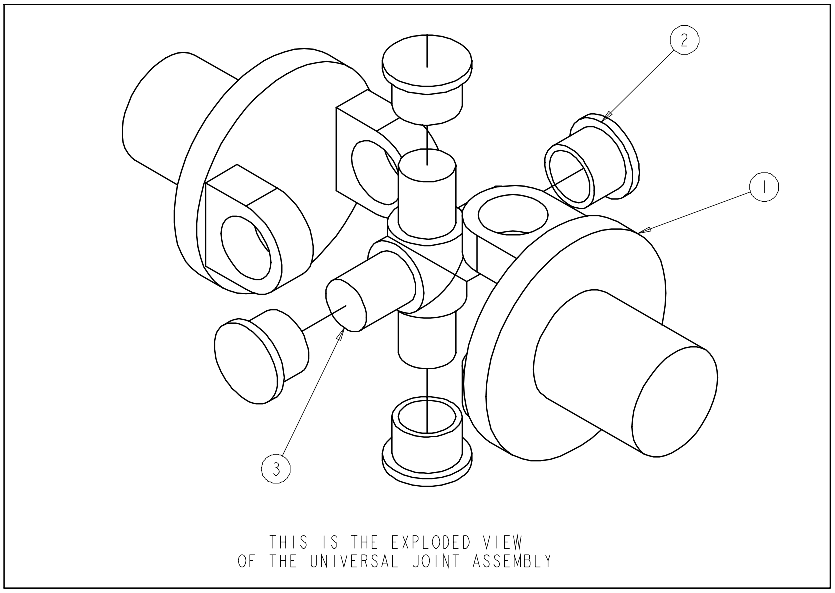
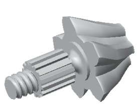


Figure 11-20 *Drawing view with balloons and note*



Tip: *The notes or balloons can be modified and edited using the options in the shortcut menu that is displayed when you select them using the left mouse button and hold down the right mouse button.*

TUTORIALS

Tutorial 1

In this tutorial you will generate the drawing views of the part shown in Figure 11-21. Generate the dimensions in the drawing views. The views to be generated are shown in Figure 11-22.

(Estimated time: 45 min)

The following steps outline the procedure to create the given drawing:

a. Create the model in the Part mode using the given dimensions and save the file.

b. Open a new drawing file in the Drawing mode and generate the top view and the front view of the model on the drawing sheet.

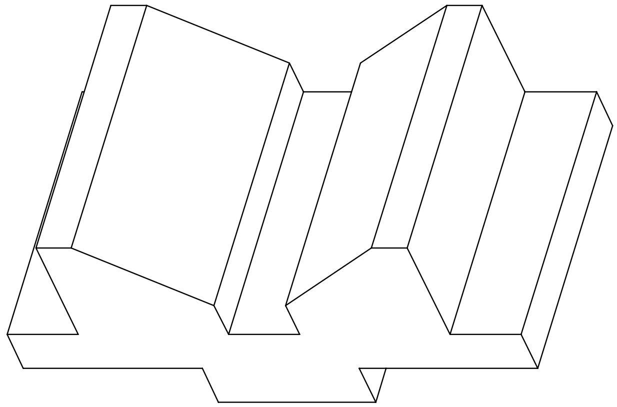


Figure 11-21 *Model for generating the drawing views*

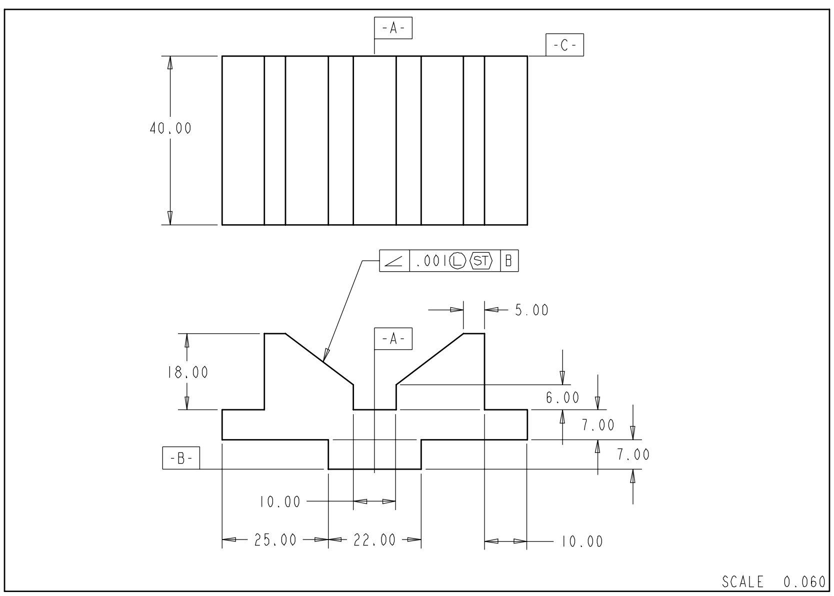


Figure 11-22 *Drawing views to be created along with the dimensions*

c. Generate the dimension to the views and clean them using the **Clean Dimensions** dialog box.

d. Add geometric tolerance to the required entities.

Before you start creating the model and the drawing views, set the working directory to

C:\ProE\c11. Make sure that the c11 directory exists inside the ProE directory. If these directories do not exist, create them using the **Select Working Directory** dialog box.

Creating the Model

First, create the given model in the part mode whose views are to be generated later.

1. Open a new object file in the Part mode and name it **c11tut1**.

2. Create the model in the Part mode and save this model.

Creating a New Drawing File

Create a new drawing file to generate the required views.

1. Choose the **Create a new object** button from the **File** toolbar. The **New** dialog box is displayed.

2. Select the **Drawing** radio button from the **Type** area and enter the name of the drawing as **c11tut1** in the **Name** edit box.

3. Choose **OK** from the **New** dialog box to proceed further. The **New Drawing** dialog box is displayed.

4. If the name of the model is not displayed in the **Default Model** area, choose the **Browse** button and select the model. Select the **Empty** radio button from the **Specify Template** area, **Landscape** button from the **Orientation** area, and the **A4** option from the **Standard Size** drop-down list.

5. Choose **OK** from the **New Drawing** dialog box to enter the Drawing mode.

   You enter the Drawing mode and a new A4 size drawing sheet is displayed on the screen.

Generating the Drawing Views

You need to generate the top view and the front view of the model. Any of the two views can be generated first. However, you will generate the top view first.

1. Choose the **Views** option from the **DRAWING** menu in the **Menu Manager** to display the **VIEW TYPE** submenu.

2. Choose **General** > **Full View** > **No Xsec** > **Scale** > **Done**.

3. Specify the placement point for the top view as shown in Figure 11-23. As soon as you specify the point, the **Message Input Window** is displayed. Enter the scale factor for the view as **0.06** and press ENTER. The default view is placed and the **Orientation** dialog box is displayed.

4. Choose the **TOP** option from the **Saved Views** drop-down list in the **Orientation** dialog box.

5. Choose the **Set** button. The default view is oriented as the top view.

6. Choose **OK** to exit the **Orientation** dialog box.

   The text for scale is placed at the corner of the sheet so that it does not interfere with the other text in the drawing sheet. To move the text perform the next step.

   Choose the **No Hidden** from the **Model Display** toolbar and repaint the screen using the **Redraw the current view** button from the **View** toolbar.

7. Double-click on the text; the text attaches to the cursor. Move the mouse and place the text at the bottom right corner of the drawing sheet using the left mouse button.

8. Choose the **Add View** option from the **VIEWS** submenu in the **Menu Manager** to display the **VIEW TYPE** submenu.

9. Choose **Projection** > **Full View** > **No Xsec** > **No Scale** > **Done**.

10. Specify the placement point for the front view below the top view as shown in Figure 11-23.

    The two views are shown in Figure 11-23.

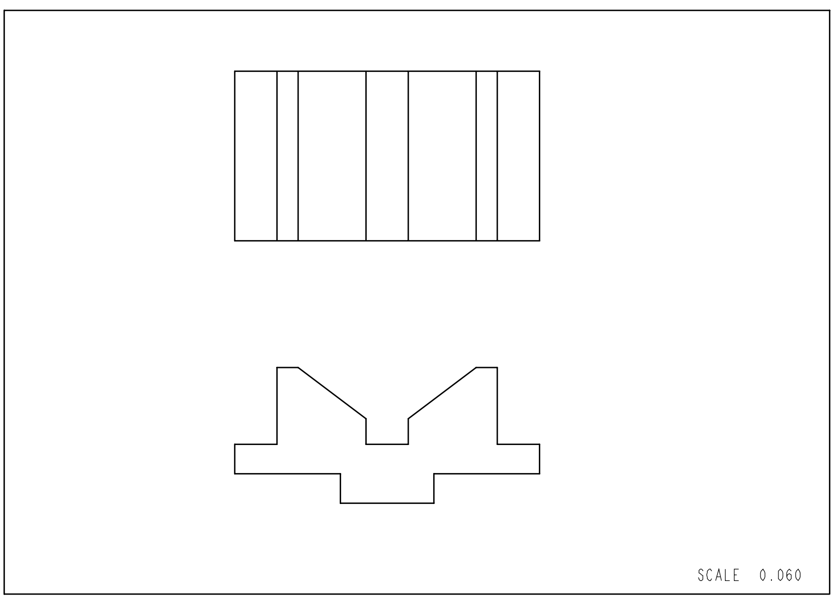


Figure 11-23 *Top view and the front view of the model*

Renaming the Datum Planes

The datum planes are renamed because the gtol values to the inclined surface will be applied with their reference. This is evident from Figure 11-22.

1. Choose the **Datum plane on/off** button from the **Datum Display** toolbar to display the datum planes and then repaint the screen.

2. Select the **RIGHT** datum plane from the top view and when it turns red in color hold down the right mouse button to display the shortcut menu.

3. Choose the **Properties** option from the shortcut menu. The **Datum** dialog box is displayed.

4. Enter **A** in the **Name** edit box and then select the second button from the **Type** area to enclose the datum in the reference box. Make sure the **Free** radio button is selected in the **Placement** area.

5. Choose **OK** to exit the **Datum** dialog box.

6. Repeat steps 3, 4, and 5 to rename **TOP** and **FRONT** datum planes to **B** and **C** respectively.

Dimensioning the Drawing Views

1. Choose **View > Show and Erase** from the menu bar. The **Show / Erase** dialog box is displayed.

2. Choose the **Show** button if it is not chosen by default and select the **Dimension** button from the **Type** area in the **Show / Erase** dialog box.

3. Choose the **View** radio button from the **Show By** area in the **Show / Erase** dialog box.

4. Select the front view (the view at the bottom) to display all the dimensions of this view.

5. Choose **Done Sel** from the **GET SELECT** menu.

6. Now, select the top view and then select **Done Sel** from the **GET SELECT** menu. All the dimensions and the dimensions for this view are displayed in the selected view.

Placing the Dimensions in Order

The dimensions displayed in the drawing views are scattered and improperly placed. This in turn makes the drawing look very untidy. You need to place the dimensions in order using the **Clean Dimensions** dialog box.

1. Choose **Edit > Cleanup Dimensions** from the menu bar to display the **Clean Dimensions** dialog box. You are prompted to select the dimensions to clean.

2. Choose the **Pick Many** option from the **GET SELECT 1** menu. Select all the dimensions from both the views by drawing a rectangle around them. Choose **Done Sel** from the **GET SELECT 1** menu so that the options in the **Clean Dimensions** dialog box are available.

3. Enter **0.5** in the **Offset** edit box and **0.5** in the **Increment** edit box.

4. Clear the **Create Snap Lines** check box. Choose **Apply** and then choose **Close** to exit the **Clean Dimensions** dialog box.

5. After cleaning the dimensions, the drawing views should look similar to the one shown in Figure 11-24.

   Select and move the dimensions if necessary.

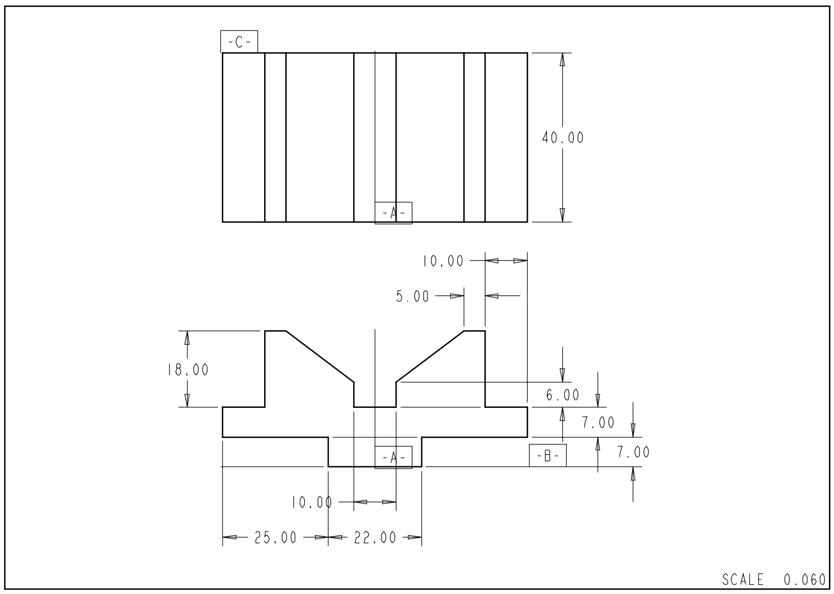


Figure 11-24 *Top and the front views of the model with the dimensions and datum planes*

Displaying the Geometric Tolerance

Actually the tolerance is created in the Part mode. However, in this chapter you will learn how the tolerances are represented in the Drawing mode.

1. Choose **Insert > Geometric Tolerance** from the menu bar to display the **Geometric Tolerance** dialog box. You will notice that the name of the model c11tut1.prt is displayed in the **Model** drop-down list.

2. Choose the geometric condition as angularity using **Angularity** button from the **Geometric Tolerance** dialog box.

3. Select the **Surface** option from the **Type** drop-down list in the **Reference** area and select the left inclined edge from the front view.

4. Select **Leaders** from the **Type** drop-down list in the **Placement** area. The **ATTACH TYPE** menu is displayed and you are prompted to select an edge for attaching the leader.

5. Select the left inclined edge from the front view and then middle-click at a position on the graphics screen where you want the note to be placed. Even if the note is placed at an improper position on the graphics screen, you can move its location later.

6. Choose the **Datum Refs** tab from the **Geometric Tolerance** dialog box.

7. Select **B** from the **Basic** drop-down list under the **Primary** tab of the **Datum References** area.

8. Select the **Tol Value** tab from the **Geometric Tolerance** dialog box.

9. Enter **0.001** in the **Overall Tolerance** edit box.

10. Select **LMC** from the **Material Condition** drop-down list.

11. Select the **Symbols** tab and then select the **Statistical Tolerance** check box from the **Symbols and Modifiers** area in the **Geometric Tolerance** dialog box.

12. Select the **Below Gtol** radio button from the **Projected Tolerance Zone** area and then select the **Zone Height** check box. Enter **0.001** in this edit box. Choose **OK** to exit the **Geometric Tolerance** dialog box.

13. The drawing views after adding the gtol should look similar to the one shown in Figure 11-25.

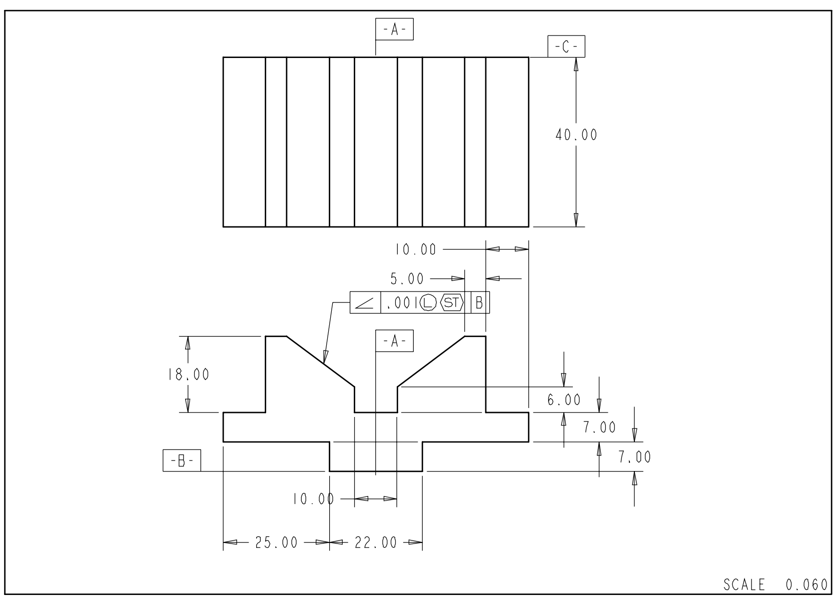


Figure 11-25 *Top and the front views of the model with the dimensions, datum planes, and the geometric tolerances*

You need to move the note for gtol to a location shown in Figure 11-26 by double-clicking on it. Similarly, arrange all the dimensions and datums on the drawing sheet as shown in Figure 11-26.

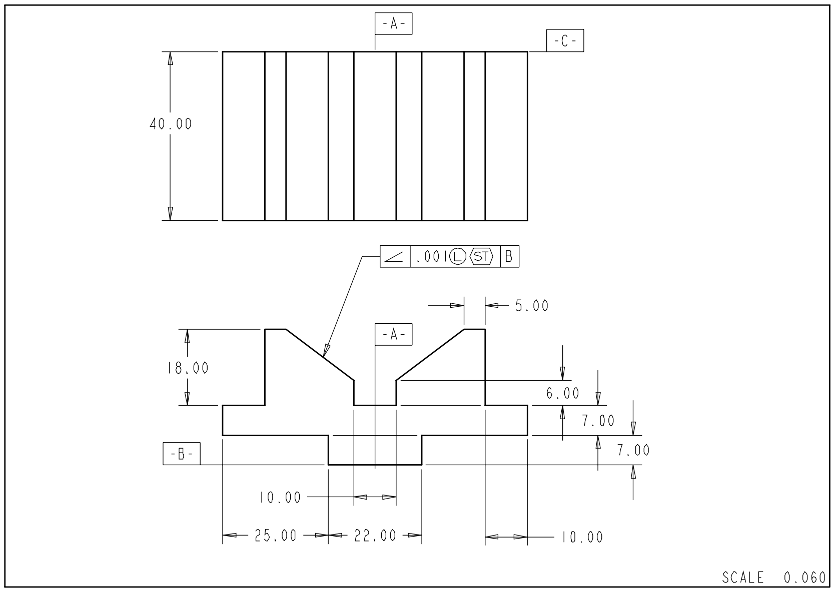


Figure 11-26 *The required drawing*

Saving the Drawing File

You need to save the drawing file that you have created as you might need it later.

1. Choose the **Save the active object** button from the **Top Toolchest**. The **Message Input Window** is displayed with the name of the drawing file that you had entered earlier.

2. Press ENTER to confirm the saving of the file.

Tutorial 2

In this tutorial you will generate the drawing views of the model shown in Figure 11-27. The dimensioned drawing views are shown in Figure 11-28. **(Estimated time: 40 min)**

The following steps outline the procedure to create the given drawing:

a. Create the model in the Part mode using the given dimensions and save the file.

b. Open a new drawing file in the Drawing mode and generate the top view, front view, right-side view, and the detailed view of the model on the drawing sheet.

c. Dimension the views and clean the dimensions using the **Clean Dimensions** dialog box.

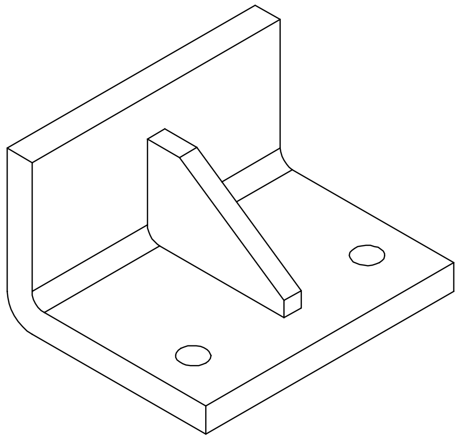


Figure 11-27 *Model for generating the drawing views*

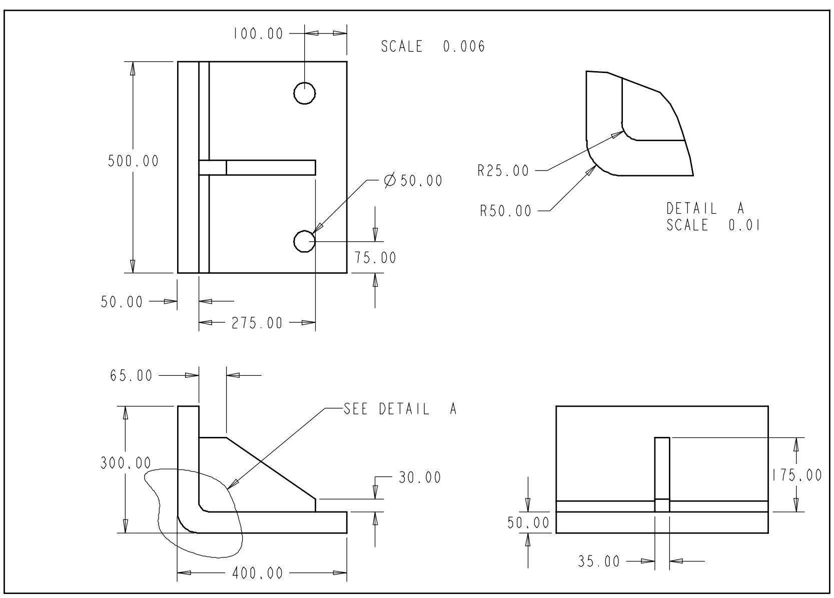


Figure 11-28 *The drawing views to be generated in Tutorial 2*

The working directory was selected in tutorial 1 of this chapter. But, if you still want to change the working directory set it to **C:\ProE\c11**.

Creating the Model

First create model whose drawing views are to be generated in the Part mode.

1. Create the model in the Part mode using the given dimensions.

2. Save this model created in the Part mode as **c11tut2**.

Creating a New Drawing File

Create a new drawing file to generate the required drawing views.

1. Choose the **Create a new object** button from the **File** toolbar. The **New** dialog box is displayed.

2. Select the **Drawing** radio button from the **Type** area and enter the name of the drawing as **c11tut2** in the **Name** edit box.

3. Choose **OK** from the **New** dialog box to display the **New Drawing** dialog box. The **New Drawing** dialog box is displayed with the name of the model in the **Default Model** area.

4. If the name of the model is not displayed in the **Default Model** area, choose the **Browse** button and select the model.

5. Select the **Empty** radio button from the **Specify Template**, **Landscape** button from the **Orientation** area, and the **A4** option from the **Standard Size** drop-down list.

6. Choose **OK** from the **New Drawing** dialog box to enter the Drawing mode. A new drawing sheet of A4 size is displayed on the screen.

Creating the Drawing Views

First, the top view of the model will be generated and then the front view will be generated by projecting it from the top view. The right-side view will be projected from the front view. The detail view will be placed on the drawing sheet by specifying a scale for it.

1. Choose the **Views** option from the **DRAWING** menu in the **Menu Manager** to display the **VIEW TYPE** submenu.

2. Choose **General** > **Full View** > **No Xsec** > **Scale** > **Done**.

3. Specify the placement point for the top view close to the top left corner of the sheet as shown in Figure 11-29. As soon as you specify the point, the **Message Input Window** is displayed. Enter the scale factor for the view as **0.006** and press ENTER. The **Orientation** dialog box is displayed. The default view is placed on the point selected.

4. Choose the **TOP** option from the **Saved Views** drop-down list in the **Orientation** dialog box.

5. Choose the **Set** button. The default view is oriented as top view on the drawing sheet.

6. Choose **OK** to exit the **Orientation** dialog box. Repaint and move the view if necessary.

   The front view of the model is generated from the top view.

7. Choose **Add View** from the **VIEWS** submenu in the **Menu Manager** to display the **VIEW TYPE** submenu.

8. Choose **Projection** > **Full View** > **No Xsec** > **No Scale** > **Done**.

9. Specify the placement point for the front view below the top view as shown in Figure 11-29.

10. Similarly, place the right-side view and the detailed view on the drawing sheet as shown in Figure 11-29.

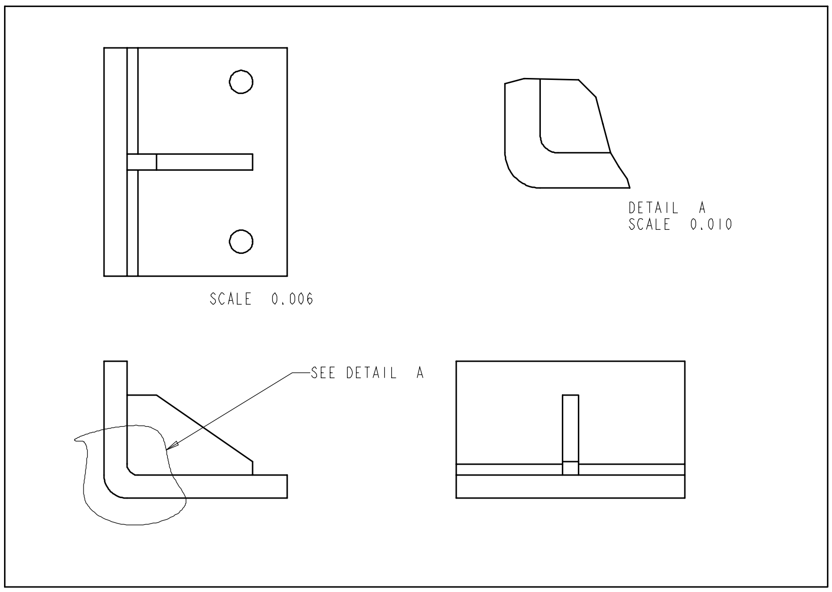


Figure 11-29 *The drawing views before adding the dimensions*

Dimensioning the Drawing Views

1. Choose **View** > **Show and Erase** from the menu bar.

2. Select the **Dimension** button from the **Type** area and then select the **Feature** radio button from the **Show By** area to display the **GET SELECT** menu. Select the feature in the detailed view. All the dimensions related to that feature (the base feature) will be displayed in all the views. Choose **Done Sel** from the **GET SELECT** menu. Another **GET SELECT** menu is displayed. Choose the **Done Sel** option again.

3. Select the **View** radio button from the **Show By** area and then select all the three views one by one to display all the dimensions in the views. Choose **Done Sel** from the **GET SELECT** menu and then close the **Show / Erase** dialog box by choosing the **Close** button.

Placing the Dimensions in Order

The dimensions displayed in the drawing views are scattered and improperly placed. Use the

Clean Dimensions dialog box to place the dimensions in order.

1. Choose **Edit** > **Cleanup Dimensions** from the menu bar to display the **Clean Dimensions** dialog box. You are prompted to select the dimensions to clean.

2. Choose the **Pick Many** option from the **GET SELECT 1** menu. Select all the dimensions from both the views by drawing a rectangle around them. Choose **Done Sel** from the **GET SELECT** menu so that the options in the **Clean Dimensions** dialog box are available.

   When you select the dimensions, the radius dimensions in the drawing views are not selected. This means that the radius dimensions will remain unaffected by the values that you will enter in the **Clean Dimensions** dialog box.

3. Enter **0.4** in the **Offset** edit box and **0.3** in the **Increment** edit box.

4. Clear the **Create Snap Lines** check box. Choose **Apply** and then choose **Close** to exit the **Clean Dimensions** dialog box.

5. After cleaning the dimensions, notice that the dimensions are not placed in the order that is required. Now, you need to manually place the dimensions in order.

   To move a dimension, double-click on the dimension and place it at the desired location on the drawing sheet. Some dimensions are repeated, for example, the diameter of the hole feature is displayed twice in the drawing view. So, you need to erase the dimensions that are repeated and that are not needed. To erase the dimensions, select the dimension and hold down the right mouse button to display the shortcut menu. Choose the **Erase** option from the shortcut menu. In some cases the dimensions are displayed in the views in which you do not want them to be displayed. In such cases you can switch that dimension to the other views. The **Switch View** option is available in the shortcut menu that is displayed when you right-click on a dimension.

   After manually placing the dimensions the drawing views should look similar to views shown in Figure 11-30.

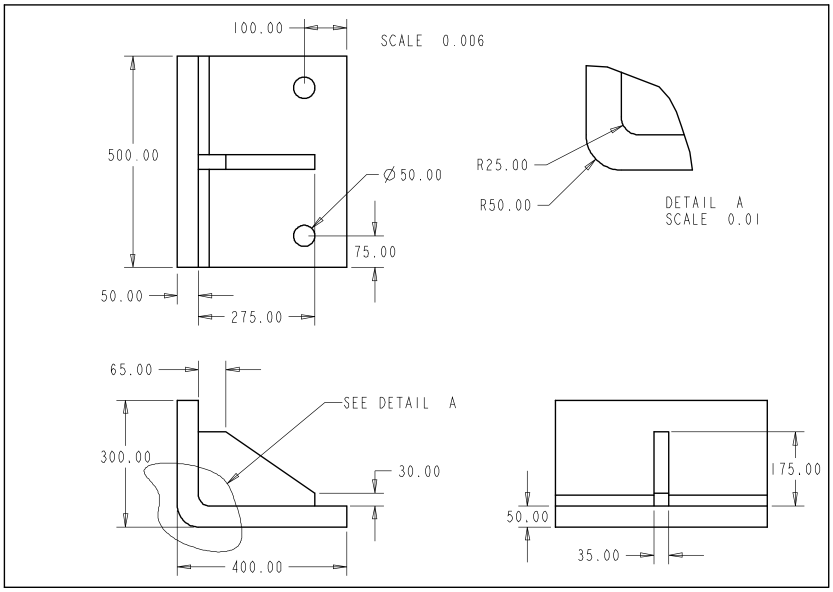


Figure 11-30 *The required drawing views*

Self-Evaluation Test

Answer the following questions and then compare your answers to the answers given at the end of this chapter.

1. The reference datum planes are used to display the geometric tolerances in a drawing view. (T/F)

2. If you modify a dimension value in the Drawing mode and then regenerate the drawing view, the modification is applied to the model in the Part mode also. (T/F)

3. The notes created in the Part mode on a model can also be displayed in a drawing view in the Drawing mode. (T/F)

4. When you select a drawing view to display dimensions, all the dimensions are placed in proper order in the drawing view. (T/F)

5. Most of the editing operations in a drawing view can be done using the shortcut menu that is displayed when you right-click on an item to be modified. (T/F)

6. The __________ dialog box is used to display the dimensions in the drawing views.

7. The dimension values can be modified using the __________ option from the **Edit** menu

in the menu bar.

8. The __________ dialog box is used to clean the dimensions in the drawing views.

9. You have to set the value of __________ to **YES** in the configuration file to display the dimensional tolerance in the drawing views.

10. The configuration file can be opened from __________.

Review Questions

Answer the following questions:

1. Which of the following dialog boxes is used to display dimensions in a drawing view?

   (a) **Dimension Properties** (b) **Clean Dimensions**
   (c) **Show / Erase** (d) None

2. Which of the following dialog boxes is used to modify properties of text in a drawing view?

   (a) **Dimension Properties** (b) **Clean Dimensions**
   (c) **Show / Erase** (d) None

3. Which of the following dialog boxes is used to display the geometric tolerances in a drawing view?

   (a) **Geometric Tolerance** (b) **Clean Dimensions**
   (c) **Show / Erase** (d) None

4. Which of the following options available in the shortcut menu is used to switch a dimension from one view to another?

   (a) **Erase** (b) **Flip Arrows**
   (c) **Switch View** (d) None

5. Which of the following radio button in the **Type** area is selected in the **New** dialog box to open a new drawing file?

   (a) **Assembly** (b) **Sketch**
   (c) **Part** (d) None

6. The dimensions erased using the **Erase** option from the shortcut menu are deleted from the model in the Part mode also. (T/F)

7. The notes or balloons can be modified and edited using the options in the shortcut menu. (T/F)

8. The system does not show a reference datum plane in a drawing view unless it is perpendicular to the graphics screen. (T/F)

9. Generally, the dimensions displayed in the drawing views are scattered and improperly placed. (T/F)

10. You cannot move text dynamically on the drawing sheet. (T/F)

Exercise

Exercise 1

In this exercise you will generate the drawing views of the model created in Exercise 2 of Chapter 6 shown in Figure 11-31 and add dimensions to the views as shown in the figure.

(Estimated time: 40 min)

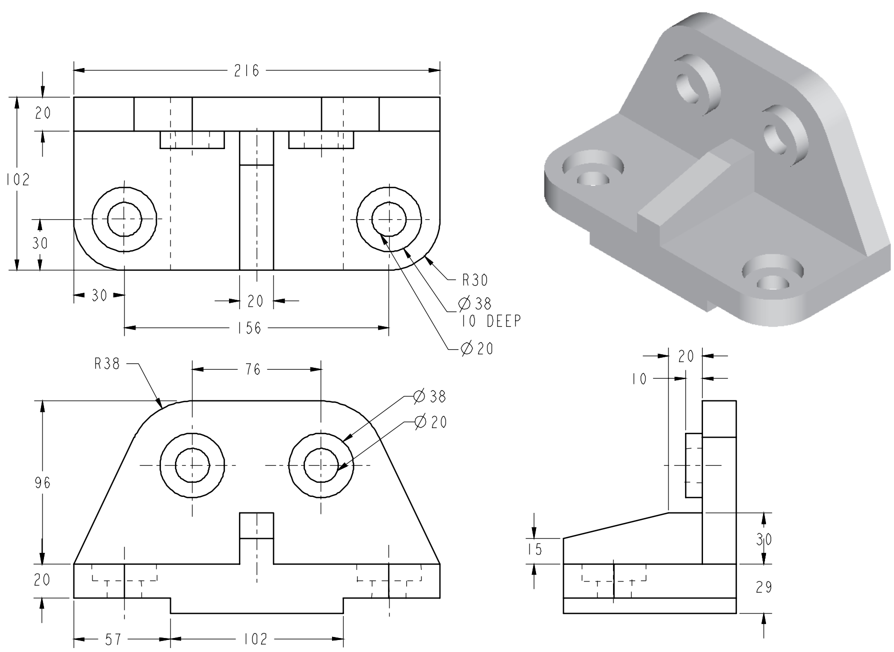


Figure 11-31 *The required drawing views with the solid model*

Note

In the above drawing views, the center lines are not generated but are created manually. After creating the center lines you need to define a linestyle and then apply it on the center lines.

Answers to the Self-Evaluation Test

1 - T, **2** - T, **3** - T, **4** - F, **5** - T, **6** - **Show / Erase**, **7** - **Value**, **8** - **Clean Dimensions**, **9** - **tol_display**, **10** - **DRAWING > Advanced > Draw Setup**.