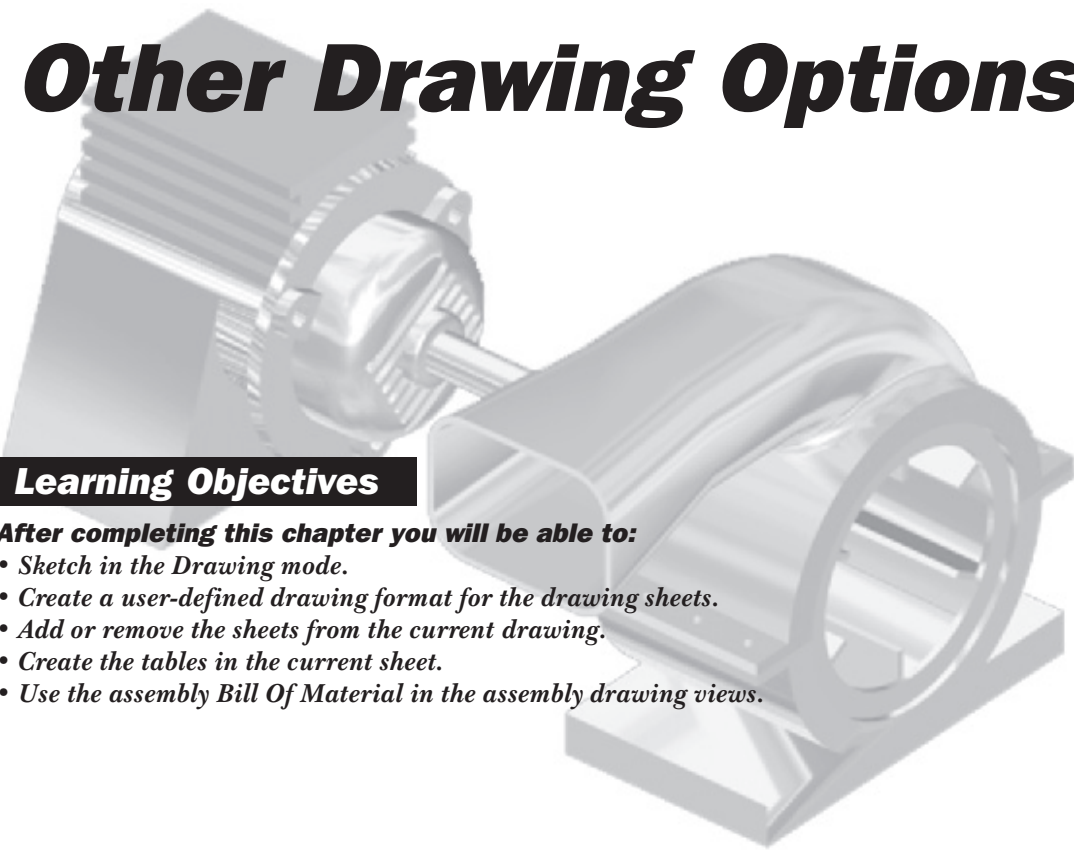


Chapter 12

Other Drawing Options



Learning Objectives

After completing this chapter you will be able to:

- Sketch in the Drawing mode.
- Create a user-defined drawing format for the drawing sheets.
- Add or remove the sheets from the current drawing.
- Create the tables in the current sheet.
- Use the assembly Bill Of Material in the assembly drawing views.

SKETCHING IN THE DRAWING MODE

Sketching is one of the most important tools available in the Drawing mode. Sketching in the Drawing mode is called Drafting. As discussed earlier, there are two types of drafting in Pro/ENGINEER: Generative drafting and Interactive drafting. Any item on the drawing sheet that is not generated from a model is called a draft entity or a draft item. Drafting is extensively used for creating user-defined formats, drawing tables, and also for drawing the title blocks in the formats. The sketching in the Drawing mode is almost similar to the sketching in the other modes of Pro/ENGINEER. The sketching can be done by using the Sketch menu in the menu bar or the tool buttons available in the **Drawing Sketcher Tools** toolbar from the **Right Toolchest**. The tools available in the **Drawing Sketcher Tools** toolbar and the options available in the **Sketch** menu that are used to sketch in the Drawing mode are discussed next.

Select Items



The **Select one item at a time - shift to gather more than one item.** button in the **Right Toolchest** is used to select drawing views, sketched entities, dimensions, notes, and so on from the drawing views. This button is used extensively to move drawing views or the items in the drawing view. If you select any option from the **Menu Manager** then this button is selected automatically.



Tip: To delete an entity or an item from a drawing view, you can select the item and then press the **DELETE** key. You can also select the entity and use the **Delete** option from the shortcut menu that is displayed when you hold down the right mouse button.

Line



The **Line** option in the **Sketch** menu in the menu bar or the **Create lines.** button from the **Right Toolchest** is used to create line segments. When you choose this option, the **Snapping References** dialog box is displayed as shown in Figure 12-1. The options in this dialog box are discussed next.

Select references button

When you choose the **Select references** button from the **Snapping References** dialog box, the **GET SELECT** menu is displayed and you are prompted to select an entity to which you want the line to be snapped.

Remove

The **Remove** button is used to remove a selected entity from the **Snapping References** dialog box.

Circle



The **Circle** option or the **Create circle.** button is used to create circles.

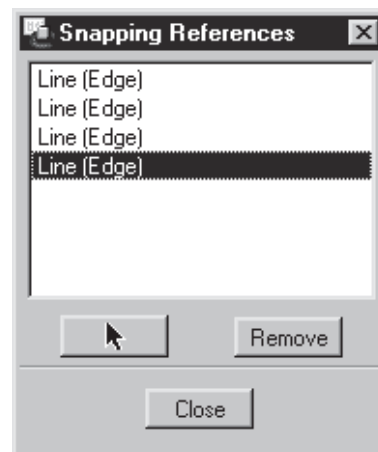


Figure 12-1 Snapping References dialog box

Arc



The **Arc** option or the **Create an arc by 3 points or tangent to an entity at its endpoint** button is used to create arcs.

The other geometric entities that you can draw using the tools available in the **Right Toolchest** are splines, construction circles and lines, ellipses, and points. You can also create a fillet and a chamfer between two entities.

Chain



The **Enable sketching chain** button is used when the object you want to draw consists of more than one entity. You can create the object as a chain by choosing this button and then drawing the entities by using the required buttons. The chain can be ended by pressing the middle mouse buttons.

MODIFYING THE SKETCHED ENTITIES

The sketched entities can be modified by using the options under the **TOOLS** submenu that is displayed when you choose **DRAWING > Tools** from the **Menu Manager**, see Figure 12-2. The options available under this submenu are discussed next.

Translate

The **Translate** option is used to move the selected entity from its actual location. The new location can be specified using the **GET VECTOR** and **GET POINT** submenus displayed upon selecting the entity to move.

Rotate

The **Rotate** option is used to rotate the selected entity from its actual location. You will be prompted to specify the center point for the rotation. The default direction of rotation will be in the counterclockwise direction. However, you can rotate the selected entity in the clockwise direction by specifying a negative rotation angle in the **Message Input Window**.

Rescale

The **Rescale** option is used to rescale the selected entity. You will be prompted to select the origin point for scaling when you select the entity. The new scale factor can be specified in the **Message Input Window**. A positive scale factor will increase the size of the selected entity and a negative scale factor will reduce the size of the selected entity.

Copy

The **Copy** option is used to copy the selected entity. When you select this option, you will be prompted to specify the method of copying the selected item. You can copy an entity by specifying the linear values or by specifying the angular values. You can also copy an entity



Figure 12-2 TOOLS submenu

from another drawing using this option.

Mirror

The **Mirror** option is used to mirror the selected entity using an existing sketched line.

Trim

The **Trim** option is used to trim two selected entities. The options of trimming the entities are the same as those in the **Sketch** mode.

Intersect

The **Intersect** option is used to break two sketched entities at their point of intersection.

Stretch

The **Stretch** option is used to stretch a sketched entity by selecting it using a window. The vertices of the sketched entity that will be included inside the window will be moved from their original location and stretched and the remaining vertices will be stationary.

Divide

The **Divide** option is used to divide the selected entity into a specified number of equal length segments. It automatically calculates the length of the selected entity and divides it into a specified number of segments.

Group

The **Group** option is used to create, suppress, resume, delete, or modify the groups.

Offset

The **Offset** option is used to offset the selected entity. You can offset a single entity or select the entire chain to offset. By default, the direction of offset will be in the direction of the arrow. However, you can also offset the selected entities in the other direction by entering a negative value in the **Message Input Window**.

Use Edge

The **Use Edge** option is selected to draw a copy of the selected edge in the drawing view. This option will be available only when you have at least one drawing view in the current drawing sheet.

Draft View

The **Draft View** option is used to add or remove the selected sketched entity to a view. If the sketched entity is added to the view and the view is deleted, the sketched entity will also be deleted along with the view. In case of more than one drawing view, you can set the selected view as the current view using this option.

Relate Obj

The **Relate Obj** option is used to add or remove notes, surface symbols, or gtols from the selected dimension.

USER-DEFINED DRAWING FORMATS

Pro/ENGINEER provides you with some standard drawing formats for generating the drawing views. These standard formats have standard sheet sizes, tables, and title blocks. However, sometimes you may need to create a user-defined drawing format that is specifically designed as per your requirements. This format can include the sheet size, tables, and title block specified as per your requirements. Choose the **Create a new object** button to display the **New** dialog box. In this dialog box, select the **Format** radio button from the **Type** area and then specify the name of the format in the **Name** edit box to proceed for creating the user-defined format. When you choose **OK**, the **New Format** dialog box will be displayed, see Figure 12-3.

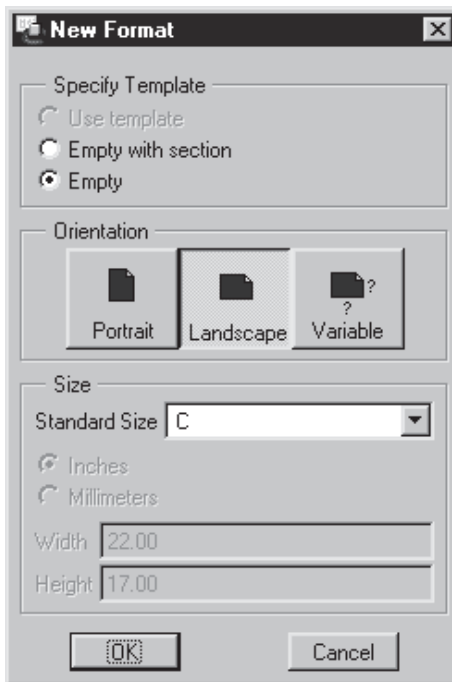


Figure 12-3 New Format dialog box

You can set the size and the orientation of the format sheet using this dialog box. Choose **OK** in this dialog box after setting the parameters for the format to proceed to the **Format** mode. You can create the desired entities in this format by using the **Drawing Sketcher** toolbar. You can also add the text material to the format by using the **Note** option from the **Insert** menu in the menu bar. Figure 12-4 shows a user-defined format created using the A4 size sheet. This format consists of the user-defined title block. Figure 12-5 shows the drawing views generated on the user-defined format.

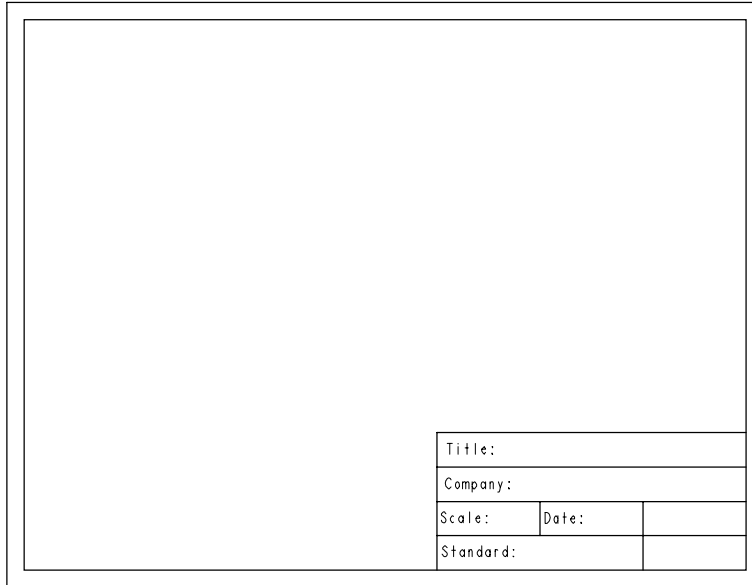


Figure 12-4 A user-defined format

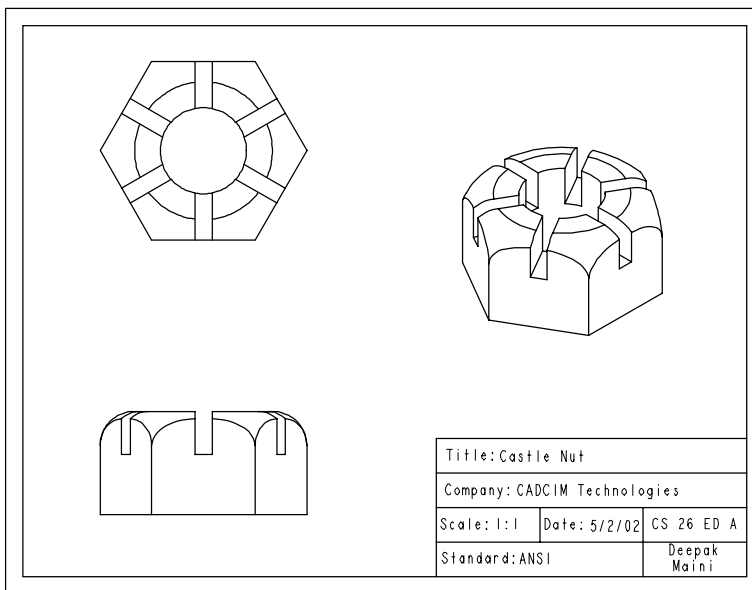


Figure 12-5 Drawing views generated using a user-defined format

RETRIEVING THE USER-DEFINED FORMATS IN THE DRAWINGS

For generating the drawing views you can also retrieve the user-defined format. Once you have created the user-defined formats, you can use them in the **Drawing** mode as sheets for generating the drawing views. To retrieve the user-defined format, select the **Empty with format** radio button and choose the **Browse** button from the **New Drawing** dialog box to invoke the **Open** dialog box. A folder named **System Formats** will be displayed in the **Open** dialog box. Some predefined formats are given in this folder. You can retrieve these predefined formats or browse the location where you have saved the user-defined format created earlier.



Tip: The user-defined format that you create is saved in the .frm format. The location of this format will be the working directory that was selected at the time when the format was created.

ADDING AND REMOVING SHEETS IN THE DRAWING

To add sheets in the current drawing, choose **Drawing > Sheets > Add** from the **Menu Manager**. A new sheet is displayed on the screen. At the bottom of the screen, the sheet number is displayed. You can generate different views of a model on multiple sheets. These all drawing views on different sheets are contained in a single drawing file.

To remove sheets from a drawing, choose **Drawing > Sheets > Remove** from the **Menu Manager**. The **Message Input Window** is displayed. Enter the sheet number that you want to remove.

The following points should be remembered while dealing with multisheets:

1. If you move a projected view to another sheet, the parent-child relationship of the two views is broken. That means, if you move the projected view on one sheet the source view on the other sheet does not move.
2. The scale of the drawing views on different sheets can be controlled independently.
3. If you erase a drawing view on one sheet, it can be resumed on the other sheet.



Tip: You can add or replace the formats in drawing by using the options available under the **SHEETS** submenu that is displayed when you choose **DRAWING > Sheets > Format** from the **Menu Manager**. You can add another sheet of the same format or some other format using these options.

CREATING TABLES

You can easily create any kind of tabular representation in the **Drawing** mode by using the options available under the **TABLE** submenu in the **Menu Manager**. To invoke this submenu, choose **DRAWING > Table**, see Figure 12-6.

TABLE submenu Options

The options that are available under this submenu are discussed next.

Create

The **Create** option is used to create a table. The table can be created in the ascending or descending order of the rows progress and in the right or the left direction. The length and width of the cells can be specified by picking the points on the screen or by specifying the value of their length and width. When you choose this option, the **TABLE CREATE** submenu is displayed and you can select the options for creating the table from this submenu.

Delete

The **Delete** option is used to delete the entire table. When you choose this option, you will be prompted to select the table to be deleted. As soon as you select the table, you will be prompted to confirm the deletion process using the message in the **Message Input Window**.

Move

The **Move** option is used to move the selected table from its default location. The table is moved using any of the four corners of the table.

Enter Text

The **Enter Text** option is used to enter text in the selected cell. When you invoke this option, you will be prompted to select the cell in which the text has to be placed. It is recommended that before entering the text in the cells you first define the justification for the rows or columns so that a uniformity is maintained in entering the text in the cells. The justification can be defined by using the **Mod Rows/Cols** option provided under the **TABLE** submenu. You can modify the text by simply overwriting it in the selected cell.

Copy

The **Copy** option is used to copy the text content of a cell or the entire table. If you want to copy the content of the cell then you have to first select the cell to copy and then specify the new cell in which this content has to be copied.

Modify Table

The **Modify Table** option is used to modify the selected table. When you choose this option, the **TABLE MODIFY** submenu will be displayed as shown in Figure 12-7.

TABLE MODIFY submenu options

Merge. This option is used to merge the selected rows, columns, or rows and

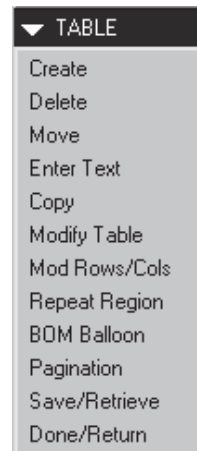


Figure 12-6 TABLE submenu

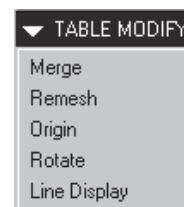


Figure 12-7 TABLE MODIFY submenu

columns. It has to be kept in mind that at the maximum, only one cell among the cells selected to merge can contain the text.

Remesh. This option is used to remesh the cells merged using the merge option. Here also the same rule applies, that is, at the maximum, only one cell can contain the text.

Origin. This option is used to locate the origin of the selected table. The origin can be defined at any of the four corners of the table. This origin will remain stationary during the addition of the rows or columns in the selected table. This origin is also used as the center point during the rotation of the table.

Rotate. This option is used to rotate the selected table by 90 degree in the counterclockwise direction about the origin of the table.

Line Display. This option is used to modify the display of the lines in the selected table. You can remove the display of the selected line and make it blank or unblank.



Tip: In Drawing mode, nearly all the modifications and editing can be done using the shortcut menu that is displayed when you select the item to modify and then hold down the right mouse button. The options in this menu vary and depend upon the item selected.

Mod Rows/Cols

The **Mod Rows/Cols** option is used to modify the rows and columns of the selected table. When you choose this option, the **ROW/COL OPTS** submenu is displayed as shown in Figure 12-8.

ROW/COL OPTS submenu Options

Insert. This option is used to insert a row or a column in the table. To insert a row or a column, pick on the border of the cell where it has to be added.

Remove. This option is used to remove the selected row or column. You will be prompted to confirm the deletion process when you choose this option.

Change Size. This option is used to change the size of the selected row or column. The new size can be specified in terms of the length of the cell or by picking two points on the screen.



Figure 12-8 ROW/COL OPTS submenu



Tip: It is advised that you use the options under the **TABLE** submenu to create the title blocks in the formats. The text can be easily added to the title block by using the **Enter Text** option from the **TABLE** submenu.

Justify. This option is used to specify the text justification for the columns. It has to be kept in mind that the justification can be specified only for the columns.

USING THE ASSEMBLY BILL OF MATERIAL IN THE DRAWINGS

The Bill Of Materials for the assembly created in the Assembly mode can be directly used in the Drawing mode if it was saved in the **.bom** format. To use the assembly BOM, choose **Insert > Note > File** from the menu bar. Select the other options for placing the BOM and then enter the name of the file that was saved in the **.bom** format in the Assembly mode. You can modify the BOM and you can also sketch a table around the BOM using the sketcher tools.

TUTORIALS

Tutorial 1

In this tutorial you will create a format of size A and add the title block in the format. Then you will retrieve the format in the Drawing mode and generate the exploded isometric view of the **Pedestal Bearing** assembly created in Tutorial 2 of Chapter 9. Also, add the assembly Bill of Material and the Balloons to the drawing view as shown in Figure 12-9.

(Estimated time: 45 min)

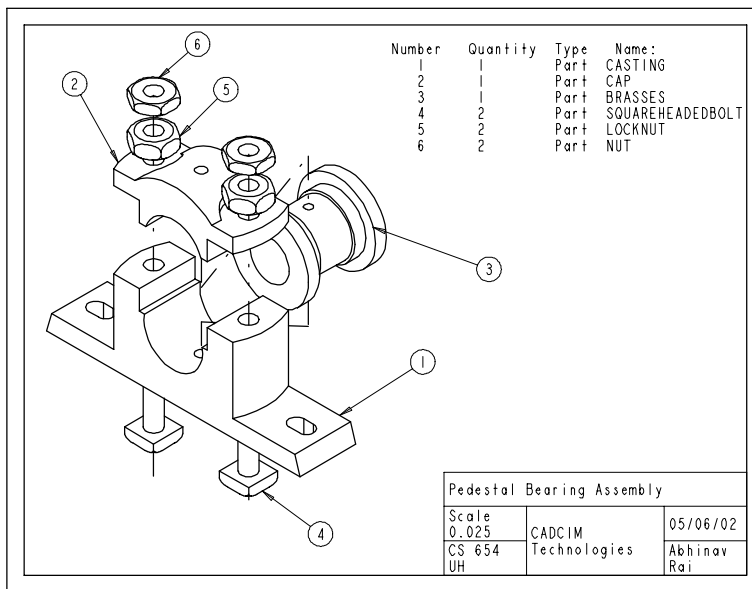


Figure 12-9 The drawing view of the assembly showing the BOM and balloons

The following steps outline the procedure to complete this tutorial:

- Open a new file in the Format mode, create the format of the drawing, and add the title block in the format.

- b. Save the format file and then close it.
- c. Open a new drawing file in the Drawing mode, select the **Pedestal Bearing** as the model, and retrieve the format that you had created.
- d. Create and save the BOM.
- e. Generate the exploded isometric view of the assembly, and add BOM and balloons to the drawing.

Before you start creating the drawing view, you need to set the working directory. Set the working directory to **C:/ProE/c12**. Since this is the first tutorial of this chapter, you need to create the **c12** folder inside the **ProE** folder.

Copy all the components of the Pedestal Bearing assembly and the assembly file named **pedestalbearing.asm** from the **c09** folder in the current working directory. This is because the assembly file with all the files of the components should lie in the folder in which the drawing file will be stored.

Creating the Format File

As evident from Figure 12-9, the exploded view of the Pedestal bearing is generated on a drawing sheet with a format. Hence, you need to create a format before generating the drawing view and then this format will be retrieved later.

1. Choose the **Create a new object** button from the **Top Toolchest**, the **New** dialog box is displayed.
2. Select the **Format** radio button from the **Type** area in the **New** dialog box and name the file as **Format1**. Choose **OK** from the **New** dialog box.

The **New Format** dialog box is displayed.

3. The **Empty** radio button in the **Specify Template** area and the **Landscape** button in the **Orientation** area of the **New Format** dialog box are selected by default. If they are not selected then you need to select them.
4. Select **A** from the **Standard Size** drop-down list in the **Size** area and then choose **OK** to proceed to the Format mode. Note, that a sheet of size **A** is displayed on the screen. This is evident from the text displayed below the sheet on the screen.

Creating Format

1. Choose **FORMAT > Tools > Offset > Ent Chain** from the **Menu Manager** and then select one by one all the four border lines of the format. Choose **Done Sel** from the **GET SELECT** submenu after you have made the selection.
2. An arrow is displayed pointing outwards. The arrow displays the direction of offset. Since you need to offset the lines in the opposite direction, therefore, enter **-0.25** in the **Message**

Input Window and press ENTER. Choose **Done/Return** from the **TOOLS** submenu.

3. Choose **FORMAT > Table > Create** from the **Menu Manager** to display the **TABLE CREATE** submenu.
4. Choose **Ascending > Leftward > By Length** from the **TABLE CREATE** submenu in the **Menu Manager** and then choose **Vertex** from the **GET POINT** submenu.
5. Select the lower right corner of the new rectangle created by offsetting the border lines. The corner of the new rectangle is selected because the title block will be created such that it is attached to the new rectangle. Now, you need to enter the dimensions for the title block.
6. Enter the width of the first column as **1.2** in the **Message Input Window** and press ENTER. Similarly, enter the width of the second column as **2** and for the third column as **1.2** in the **Message Input Window** and press ENTER. You are prompted to enter the width for next column. Press ENTER.

Now, you are prompted to enter the height for the first row.

7. Enter the height of the first row as **0.5**, the second as **0.5**, and the third as **0.5** in the **Message Input Window**. Press ENTER twice. Choose **Done/Return** from the **TABLE** submenu.

The title block created will be similar to the one shown in Figure 12-10. Note that the title

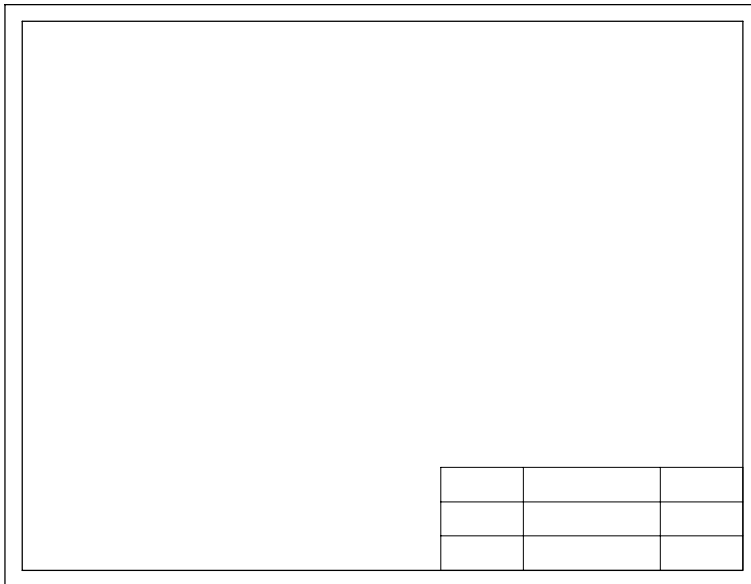


Figure 12-10 Format with the title block

block created is not as required. Therefore, you need to modify it.

8. Choose **FORMAT > Table > Modify Table** to display the **TABLE MODIFY** submenu.
9. Choose **Merge** from this submenu, the **Rows & Cols** option is selected by default in the **TABLE MERGE** submenu. Select the first column and the second column of the top row to merge. Similarly, merge the third column of the top row with the new bigger column.
10. Now, merge the second column of the second row with the second column of the third row. The table after merging the rows and columns should look similar to the one shown in Figure 12-11.

Figure 12-11 Modified title block

11. Choose **Mod Rows/Cols** option from the **TABLE** submenu in the **Menu Manager** to display the **ROW/COL OPTS** submenu. Choose the **Justify** option from the **ROW/COL OPTS** submenu. Now, you need to specify the alignment of the text that will be entered later in the title block.
12. Select **Left > Middle** from the **HOR JUST TXT** submenu and then one by one select all the three columns of the second row. Choose **Done/Return** from the **TABLE** submenu.

Saving the Format File

You need to save the format file that you have created so that you can use it as a template in the Drawing mode where you will generate the exploded drawing view of the Pedestal Bearing. The file will be stored in the .frm file format.

1. Choose the **Save the active object** button from the **Top Toolchest**. The **Message Input Window** is displayed with the name of the file that you had entered earlier. Press ENTER.
2. Close the current window by choosing **Window > Close** from the menu bar.

Creating a New Drawing File

You need to create a new drawing file to generate the exploded drawing view of the Pedestal Bearing.

1. Choose the **Create a new object** button to display the **New** dialog box. Select the **Drawing** radio button and specify the name of the drawing as **c12tut1** and then choose **OK**. The **New Drawing** dialog box is displayed.
2. If in the **Default Model** area the **PEDESTALBEARING.ASM** is not displayed then choose the **Browse** button and select the assembly file named **pedestalbearing.asm**.
3. Select the **Empty with format** radio button from the **Specify Template** area in the **New Drawing** dialog box.
4. Select **FORMAT1** from the **Format** drop-down list. If the **FORMAT1** is not available in the drop-down list, choose the **Browse** button to locate the **FORMAT1**. Choose **OK** to exit the **New Drawing** dialog box.

Generating the Exploded Drawing View

In Tutorial 2 of Chapter 9 you created the exploded view of the Pedestal Bearing in the Assembly mode. The name of the exploded view that you created was **EXP1**. Hence, the exploded view of the Pedestal Bearing is integrated with the assembly file that you copied from the c09 folder. Now, you will use this exploded state to generate the exploded drawing view.

1. Choose **DRAWING > Views** to display the **VIEW TYPE** submenu.
2. Choose **General > Full View > No Xsec > Exploded > Scale > Done** from the **VIEW TYPE** submenu. You are prompted to specify a center point.
3. Select a point on the left of the drawing sheet. The **SEL STATE** submenu is displayed. Select the **EXP1** check box if it is not selected by default and choose **Done**.

The **Message Input Window** is displayed and you are prompted to specify the scale for the drawing view.

4. Enter a value of **0.025** in the **Message Input Window** and press ENTER. The **Orientation** dialog box is displayed with the default view of the exploded assembly. But, you need the isometric view of the exploded assembly.
5. Select the **Preferences** option from the **Type** drop-down list in the **Orientation** dialog box.

6. Select the **Isometric** option from the **Default Orientation** drop-down list. The isometric view of the exploded assembly is displayed. Choose **OK** from the **Orientation** dialog box.
7. Turn off the display of the datums from the **Datum Display** toolbar. If the model is not properly placed inside the boundaries of the drawing sheet, double-click on the drawing view and place it as shown in Figure 12-12.

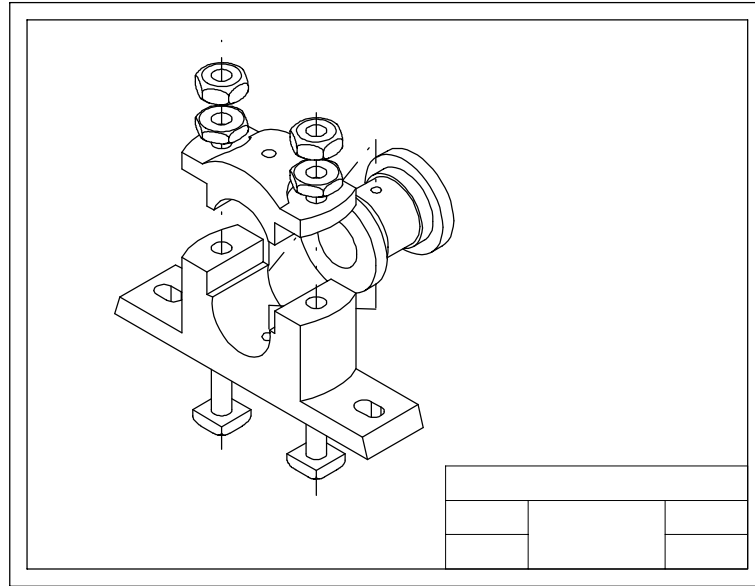


Figure 12-12 Exploded isometric view of the assembly

8. Erase the text **Scale 0.025** from the drawing sheet because it is not needed. You can erase the text by selecting it and then hold down the right mouse button to display the shortcut menu. From the shortcut menu choose the **Erase** option.

Creating the BOM

1. Choose **Info > Bill of Materials** from the menu bar. The **BOM** dialog box is displayed. In this dialog box, the **Top Level** radio button is selected by default. Choose **OK** to display the **INFORMATION WINDOW**.
2. In this window, choose **File > Save As** from the menu bar. The **Save a Copy** dialog box is displayed. The name of the BOM file is by default displayed in the **New Name** edit box.
3. Choose **OK** to exit the **Save a Copy** dialog box. Choose the **Close** button to close the **INFORMATION WINDOW**.



Tip: You can also create and save the BOM in the Assembly mode and then insert it in the drawing mode.

The BOM file named pedestalbearing.bom is saved in the current working directory.

Placing the BOM in the Drawing

1. Choose **Insert > Note** from the menu bar. The **NOTE TYPES** submenu is displayed.
2. Choose **No Leader > File > Horizontal > Standard > Default > Make Note**. Specify the placement point for the BOM on the drawing sheet close to the top right corner. The **Open** dialog box is displayed.
3. Select the **All files** option from the **Type** drop-down list.
4. Select the **pedestalbearing.bom** from the **Open** dialog box to open. The bom file is placed on the drawing sheet, but this information contained in the file is not what is needed.
5. Choose **Done/Return** from the **NOTE TYPES** submenu.

Editing the Text

You need to edit the text in the BOM. The BOM that you generated does not convey the required information of the drawing and the identification of the components that constitutes the assembly.

1. Select the text so that it turns red in color. Now, hold down the right mouse button to display the shortcut menu.
2. Choose the **Properties** option from the shortcut menu. The **Enter Text** dialog box is displayed.
3. Choose the **Editor** button from the **Enter Text** dialog box. The **Notepad** window is displayed. Delete the first two lines and all the text starting from point number 9 in this window.
4. In line number **2** in the notepad, change **Assembly** to **Number**, **PEDESTALBEARING** to **Quantity**, and **contains** to **Type**. After **Type**, type and add **Name**.
5. In line number **3**, after **3:**, type **1** such that the number 1 is middle aligned with **Number**. Similarly, below **1** type **2**, below **2** type **3**, and so on.
6. Using the SPACEBAR (longest key on the keyboard) middle align the numbers under **Quantity**.
7. Using the DELETE key, middle align **Part** with **Type**.
8. Similarly, middle align the **Name** list.
9. Choose **File > Exit** to exit the **Notepad** window. You are prompted to save the changes made. Choose the **Yes** button and choose **OK** from the **Enter Text** dialog box.

Move the BOM if it overlaps the boundary of the drawing sheet by double-clicking on it.

Entering Text in the Title Block and Adding Balloons

1. Choose **DRAWING > Table > Enter Text** from the **Menu Manager** and then select the top row to enter the text. The **Message Input Window** is displayed and you are prompted to enter text.
2. Enter **Pedestal Bearing Assembly** in the **Message Input Window** and press ENTER twice.
3. Select the cell that is in the first column and second row. The **Message Input Window** is displayed.
4. Enter **Scale:** and press ENTER. Now, enter **0.025** in the **Message Input Window** and press ENTER twice. The text **Scale: 0.025** is in two lines in the title block, therefore you entered them as two lines.

Similarly, enter the text in all the remaining cells one by one as shown in Figure 12-13.

5. Choose **Done/Return** from the **TABLE** submenu.



Tip: *If the text you need to enter is in two lines then press ENTER after entering the first line in the **Message Input Window** and then enter the second line. Now, after entering the second line press ENTER twice.*

6. Add the balloons to the assembly by choosing **Insert > Balloon** from the menu bar. The **NOTE TYPES** submenu is displayed.
7. Choose **Leader > Enter > Horizontal > Standard > Default > Make Note** from the **NOTE TYPES** submenu in the **Menu Manager**. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default.
8. Select an edge of the Brasses close to the back face and then press the middle mouse button on the screen where the balloon will be placed, see Figure 12-13. The **Message Input Window** is displayed. Enter **3** and press ENTER twice.
9. Choose the **Make Note** option from the **NOTE TYPES** submenu. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default. Select an edge of the Cap and press the middle mouse button on the screen where the note will be placed, see Figure 12-13. The **Message Input Window** is displayed. Enter **2** and press ENTER twice.
10. Choose the **Make Note** option from the **NOTE TYPES** submenu. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default. Select an edge of the Casting and press the middle mouse button on the screen where the note will be placed, see Figure 12-13. The **Message Input Window** is displayed. Enter **1** and press ENTER twice.

Similarly, place the remaining balloons. The sheet after placing the BOM and adding the balloons should look similar to the one shown in Figure 12-13.

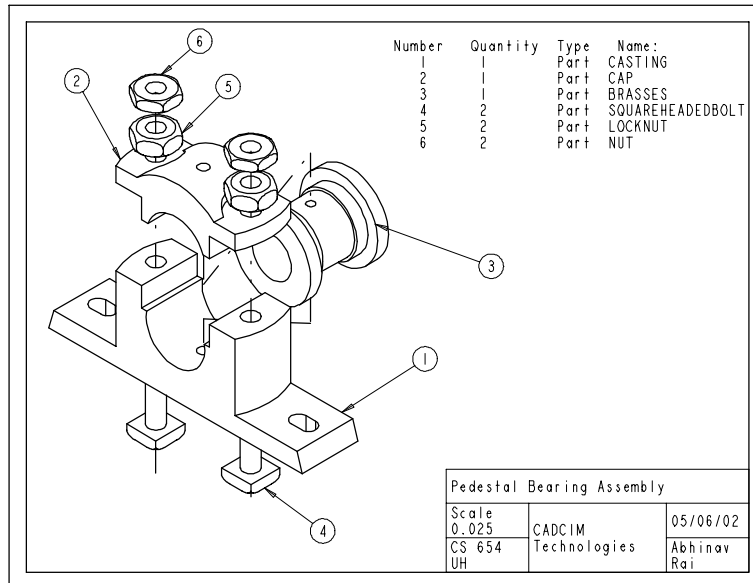


Figure 12-13 The drawing view of the assembly showing the BOM and balloons

Saving the Drawing File

1. Choose the **Save the active object** button from the **Top Toolchest** to save the drawing file.
2. The **Message Input Window** is displayed. Press ENTER.

Closing the Window

The drawing file is saved and now you can exit the Drawing Mode.

1. Choose the **Close** option from the **Window** menu in the menu bar.



Note

The occurrence of the components in BOM depends on the order in which they were assembled in the assembly. In other words, the component placed first will occur first in the BOM.

Tutorial 2

In this tutorial you will create a format of size A and add the title block in the format. You will retrieve the format in the Drawing mode and generate the exploded isometric view of the **Shock** assembly created in Tutorial 1 of Chapter 9. Also, add the assembly Bill of Material and the Balloons to the drawing view as shown in Figure 12-14. **(Estimated time: 45 min)**

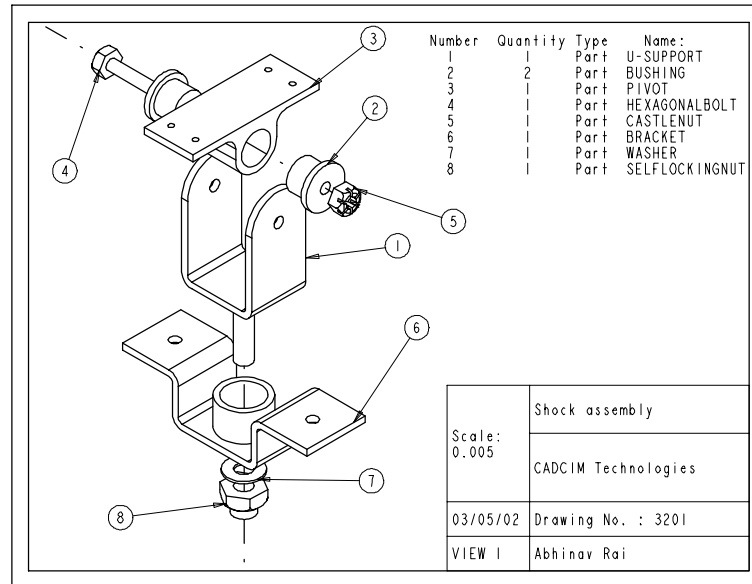


Figure 12-14 The drawing view of the exploded assembly with BOM

The following steps outline the procedure to complete this tutorial:

- Open a new file in the Format mode, create the format of the drawing, and add the title block in the format.
- Save the format file and then close it.
- Open a new drawing file in the Drawing mode, select the **shockassembly.asm** as the model, and retrieve the format that you had created.
- Create and save the BOM.
- Generate the exploded isometric view of the assembly and add BOM and balloons to the drawing.

The working directory was selected in Tutorial 1, and therefore there is no need to select the working directory again. But if you still want to select the working directory, choose the **Set Working Directory** option from the **File** menu. The **Select Working Directory** dialog box is displayed. Set the working directory to **C:/ProE/c12**.

Copy all the components of the Shock assembly and the assembly file named **shockassembly.asm** from the **c09** folder in the current working directory. This is because the assembly file with all the files of the components should lie in the folder in which the drawing file will be stored.

Creating the Format File

As mentioned in the tutorial description, you need to create a format of A size that will be retrieved later to create the drawing view of the shockassembly.

1. Choose the **Create a new object** button from the **Top Toolchest**, the **New** dialog box is displayed.
2. Select the **Format** radio button from the **Type** area in the **New** dialog box and name the file as **Format2**. Choose **OK** from the **New** dialog box.

The **New Format** dialog box is displayed.

3. The **Empty** radio button in the **Specify template** area and the **Landscape** button in the **Orientation** area of the **New Format** dialog box are selected by default. If they are not, you need to select them.
4. Select **A** from the **Standard Size** drop-down list in the **Size** area and then choose **OK** to proceed to the Format mode. Note, that a sheet of size **A** is displayed on the screen. This is evident from the text displayed below the sheet on the screen.

Creating Format

1. Choose **FORMAT > Tools > Offset > Ent Chain** from the **Menu Manager** and then one by one select all the four border lines of the sheet that is displayed in the format file. Choose **Done Sel** from the **GET SELECT** submenu after you have made the selection.
2. An arrow is displayed pointing outwards. The arrow displays the direction of offset. Since you need to offset the lines in the opposite direction, therefore, enter **-0.25** in the **Message Input Window** and press ENTER. Choose **Done/Return** from the **TOOLS** submenu.
3. Choose **FORMAT > Table > Create** from the **Menu Manager** to display the **TABLE CREATE** submenu.
4. Choose **Ascending > Leftward > By Length** from the **TABLE CREATE** submenu in the **Menu Manager** and then choose **Vertex** from the **GET POINT** submenu.
5. Select the lower right corner of the new rectangle created by offsetting the border lines. The corner of the new rectangle is selected because the title block will be created such that it is attached to the new rectangle. Next, you need to enter the dimensions for the title block.
6. Enter the width of the first column as **3.2** and press ENTER. Similarly, enter the width of the second column as **1.2** in the **Message Input Window** and press ENTER. You are prompted to enter the width for next column. Press ENTER.

Now, you are prompted to enter the height for the first row.

7. Enter the height of the first row as **0.5**, the second as **0.5**, the third as **1**, and the fourth as

0.7 in the **Message Input Window**. Press ENTER twice. Choose **Done/Return** from the **TABLE** submenu.

The title block created will be similar to the one shown in Figure 12-15. Note the difference between the title block shown in Figure 12-14 and that shown in Figure 12-15. You need to modify the title block that you have created so that it is similar to the one shown in Figure 12-14.

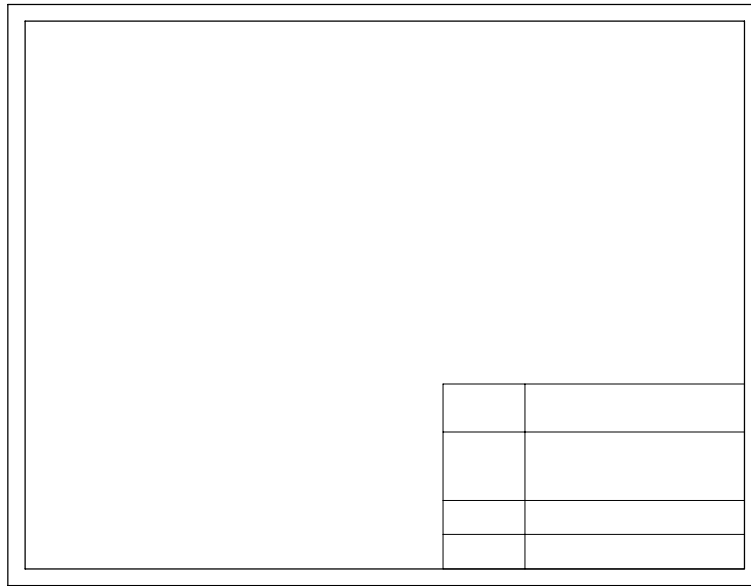


Figure 12-15 Format with the title block

8. Choose **FORMAT > Table > Modify Table** to display the **TABLE MODIFY** submenu.
9. Choose **Merge** from this submenu, the **Rows & Cols** option is selected by default in the **TABLE MERGE** submenu. Select the first row and the second row of the first column to merge.

The table after merging the rows and columns should look similar to the one shown in Figure 12-16.

10. Choose **Mod Rows/Cols** option from the **TABLE** submenu in the **Menu Manager** to display the **ROW/COL OPTS** submenu. Choose the **Justify** option from the **ROW/COL OPTS** submenu. Now, you need to specify the alignment of the text that will be entered later in the title block.
11. Select **Left > Middle** from the **HOR JUST TXT** submenu and then one by one select both the columns. Choose **Done/Return** from the **TABLE** submenu.

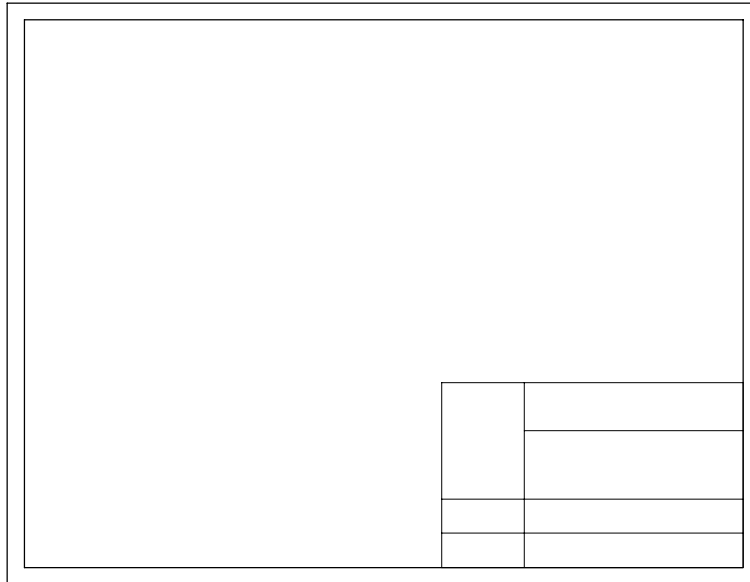


Figure 12-16 Modified title block

Saving the Format File

You need to save the format file that you have created so that you can use it as a template in the Drawing mode where you will generate the exploded drawing view of the Shock assembly. The file will be stored in the .frm file format.

1. Choose the **Save the active object** button from the **Top Toolchest**. The **Message Input Window** is displayed with the name of the file that you had entered earlier. Press ENTER.
2. Close the current window by choosing **Window > Close** from the menu bar.

Creating a New Drawing File

You need to create a new drawing file to generate the exploded drawing view of the Shock assembly.

1. Choose the **Create a new object** button to display the **New** dialog box. Select the **Drawing** radio button and specify the name of the drawing as **c12tut2** and then choose **OK**. The **New Drawing** dialog box is displayed.
2. If in the **Default Model** area **SHOCKASSEMBLY.ASM** is not displayed, choose the **Browse** button and select the assembly file named **shockassembly.asm**.
3. Select the **Empty with format** radio button from the **Specify Template** area in the **New Drawing** dialog box.
4. Select **FORMAT2** from the **Format** drop-down list. If the **FORMAT2** is not available in

the drop-down list, choose the **Browse** button to locate the **FORMAT2**. Choose **OK** to exit the **New Drawing** dialog box.

Generating the Exploded Drawing View

In Tutorial 1 of Chapter 9 you had created the exploded view of the Shock assembly in the Assembly mode. The name of the exploded view that you specified was **EXP1**. Hence, the exploded view of the Shock assembly is integrated with the assembly file that you copied from the c09 folder. Now, you will use this exploded state to generate the exploded drawing view.

1. Choose **DRAWING > Views** from the **Menu Manager** to display the **VIEW TYPE** submenu.
2. Choose **General > Full View > No Xsec > Exploded > Scale > Done** from the **VIEW TYPE** submenu. You are prompted to specify a centerpoint.
3. Select a point on the left side on the drawing sheet. The **SEL STATE** submenu is displayed. Select the **EXP1** check box if it is not selected by default and choose **Done**.

The **Message Input Window** is displayed and you are prompted to specify the scale for the drawing view.

4. Enter a value of **0.004** in the **Message Input Window** and press ENTER. The **Orientation** dialog box is displayed with the default view of the exploded assembly. But, you need the isometric view of the exploded assembly.
5. Select the **Preferences** option from the **Type** drop-down list in the **Orientation** dialog box.
6. Select the **Isometric** option from the **Default Orientation** drop-down list. The isometric view of the exploded assembly is displayed. Choose **OK** from the **Orientation** dialog box.
7. Turn off the display of the datums from the **Datum Display** toolbar. If the model is not placed properly inside the boundaries of the drawing sheet, double-click on the drawing view and place it as shown in Figure 12-17.
8. Erase the text **Scale 0.004** from the drawing sheet because it is not needed. You can erase the text by selecting it and then hold down the right mouse button to display the shortcut menu. From the shortcut menu choose the **Erase** option.

Creating the BOM

1. Choose **Info > Bill of Materials** from the menu bar. The **BOM** dialog box is displayed. In this dialog box, the **Top Level** radio button is selected by default. Choose **OK** to display the **INFORMATION WINDOW**.
2. In this window, choose the **File > Save As** option from the menu bar. The **Save a Copy** dialog box is displayed. The name of the BOM file is displayed by default in the **New Name** edit box.

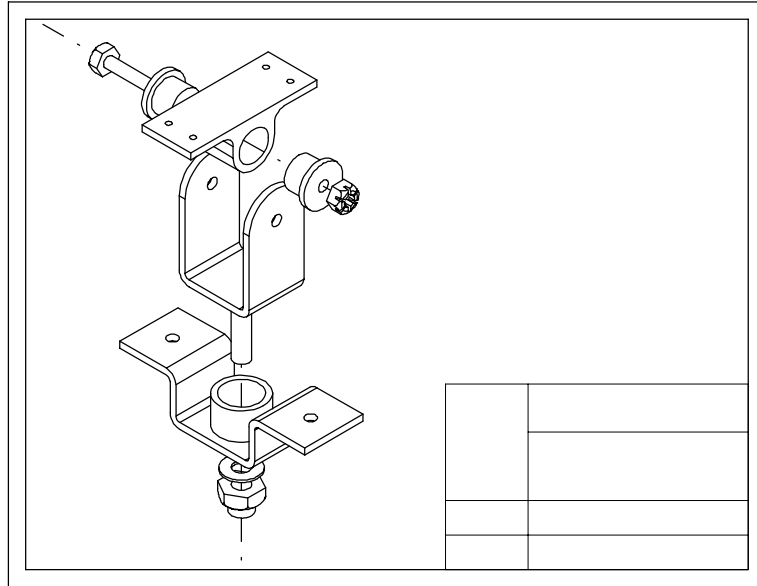


Figure 12-17 Exploded isometric view of the assembly

3. Choose **OK** to exit the **Save a Copy** dialog box. Choose the **Close** button to close the **INFORMATION WINDOW**.

The BOM file named shockassembly.bom is saved in the current working directory.

Placing the BOM in the Drawing

1. Choose **Insert > Note** from the menu bar. The **NOTE TYPES** submenu is displayed.
2. Choose **No Leader > File > Horizontal > Standard > Default > Make Note**. Specify the placement point for the BOM on the drawing sheet close to the top right corner.

The **Open** dialog box is displayed.

3. Select the **All files** option from the **Type** drop-down list.
4. Select **shockassembly.bom** from the **Open** dialog box to open. The bom file is placed on the drawing sheet, but this information contained in the file is not what is needed.
5. Choose **Done/Return** from the **NOTE TYPES** submenu.

Editing the Text

The BOM that you generated does not link the name of the parts with the drawing view. Hence, you need to edit the BOM according to your need.

1. Select the text so that it turns red in color. Now, hold down the right mouse button to display the shortcut menu.
2. Choose the **Properties** option from the shortcut menu. The **Enter Text** dialog box is displayed.
3. Choose the **Editor** button from the **Enter Text** dialog box. The **Notepad** window is displayed. Delete the first two lines and all the text starting from point number 11 from this window.
4. In line number **2** in the notepad, change **Assembly** to **Number**, **SHOCKASSEMBLY** to **Quantity**, and **contains** to **Type**. After **Type**, type and add **Name**.
5. In line number **3**, after **3:**, type **1** such that the number 1 is middle aligned with **Number**. Similarly, below **1** type **2**, below **2** type **3**, and so on.
6. Using the SPACEBAR (longest key on the keyboard) middle align the numbers under **Quantity**.
7. Using the DELETE key, middle align **Part** with **Type**.
8. Similarly, middle align the **Name** list.
9. Choose **File > Exit** to exit the **Notepad** window. You are prompted to save the changes made. Choose the **Yes** button and choose **OK** from the **Enter Text** dialog box.

Move the BOM if it overlaps the boundary of the drawing sheet by double-clicking on it.

Entering Text in the Title Block and Adding Balloons

1. Choose **DRAWING > Table > Enter Text** from the **Menu Manager** and then select the cell from the first row and the second column. The **Message Input Window** is displayed and you are prompted to enter the text.
2. Enter **Shock assembly** in the **Message Input Window** and press ENTER twice.
3. Now, select the cell from the second row and the second column. The **Message Input Window** is displayed. Enter **CADCIM Technologies** and press ENTER twice.
Similarly, enter the text in all the remaining cells one by one as shown in Figure 12-18.
4. Choose **Done/Return** from the **TABLE** submenu.
5. Add the balloons to the assembly by choosing **Insert > Balloon** from the menu bar. The **NOTE TYPES** submenu is displayed.
6. Choose **Leader > Enter > Horizontal > Standard > Default > Make Note** from the **NOTE TYPES** submenu in the **Menu Manager**. The **ATTACH TYPE** submenu is

displayed. The **On Entity > Arrow Head** option is selected by default.

7. Choose the **Make Note** option. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default.
8. Select an edge of the Pivot close to the back face and then press the middle mouse button on the screen where the note will be placed, see Figure 12-18. The **Message Input Window** is displayed. Enter **3** and press ENTER twice.
9. Choose the **Make Note** option. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default.
10. Select an edge of the Bushing and press the middle mouse button on the screen where the note will be placed, see Figure 12-18. The **Message Input Window** is displayed. Enter **2** and press ENTER twice.
11. Choose the **Make Note** option. The **ATTACH TYPE** submenu is displayed. The **On Entity > Arrow Head** option is selected by default.
12. Select an edge of the U-Support and press the middle mouse button on the screen where the note will be placed, see Figure 12-18. The **Message Input Window** is displayed. Enter **1** and press ENTER twice.

Similarly, place the remaining balloons. The sheet after placing the BOM and adding the balloons should look similar to the one shown in Figure 12-18.

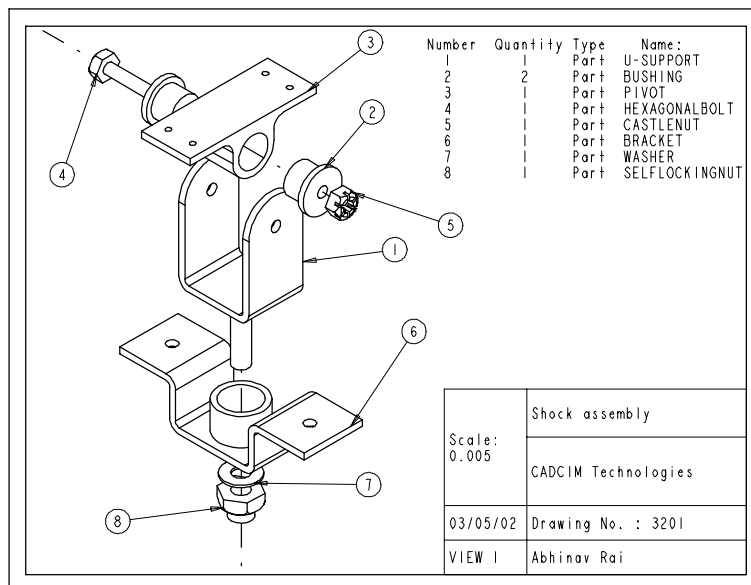


Figure 12-18 The drawing view of the assembly showing the BOM and title block

Saving the Drawing File

1. Choose the **Save the active object** button from the **Top Toolchest** to save the drawing file.
2. The **Message Input Window** is displayed. Press ENTER.

Closing the Window

The drawing file is saved and now you can exit the Drawing Mode.

1. Choose the **Close** option from the **Window** menu in the menu bar.

Self-Evaluation Test

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

1. The **Select one item at a time - shift to gather more than one item.** button in the **Right Toolchest** is used to select any item from the drawing view or the drawing view itself to modify. (T/F)
2. Pro/ENGINEER provides you with some standard drawing formats for generating the drawing views. (T/F)
3. Pro/ENGINEER allows you to create the user-defined formats. (T/F)
4. The assembly BOM can be directly used in the **Drawing** mode. (T/F)
5. Only one drawing sheet is available in one drawing file. (T/F)
6. The _____ option of the **TOOLS** submenu is used to move the selected draft entity.
7. The _____ option of the **TABLE** submenu is used to enter text material in the selected cell.
8. The _____ dialog box is used to edit the text.
9. The _____ key from the keyboard can also be used to delete any draft entity.
10. The _____ submenu is used to draw tables in the Drawing mode.

Review Questions

Answer the following questions.

- Which of the following options from the **TABLE** submenu is used to delete the entire table?
(a) **Delete** (b) **Create**
(c) **Copy** (d) **Pagination**
- Which of the following options in the **TOOLS** submenu is used to move a draft entity?
(a) **Rotate** (b) **Group**
(c) **Translate** (d) **Rescale**
- Which of the following buttons on the **Right Toolchest** enables to draw the geometric entities continuously?
(a) **Create lines.** (b) **Create circle.**
(c) **Enable sketching chain.** (d) **None**
- Which of the following radio buttons is selected to create a user-defined format for the drawing sheet?
(a) **Part** (b) **Format**
(c) **Drawing** (d) **None**
- Which of the following menus in the menu bar has the **Close** option to close the current window?
(a) **Info** (b) **Window**
(c) **View** (d) **None**
- You can open more than one windows in Pro/ENGINEER. (T/F)
- It is not possible to draw splines in the Drawing mode. (T/F)
- There are two types of drafting in Pro/ENGINEER: Generative drafting and Interactive drafting. (T/F)
- You can import a text file in the Drawing mode. (T/F)
- The **Relate Obj** option is used to add or remove notes, surface symbols, or gtols from the selected dimension. (T/F)

Exercise

Exercise 1

Create a format of size A and then retrieve it to place the top, front, and an exploded isometric view of the **Cross Head** assembly created in Exercise 1 of Chapter 9. Add the assembly BOM and the balloons to the drawing view as shown in Figure 12-19.

(Estimated time: 45 min)

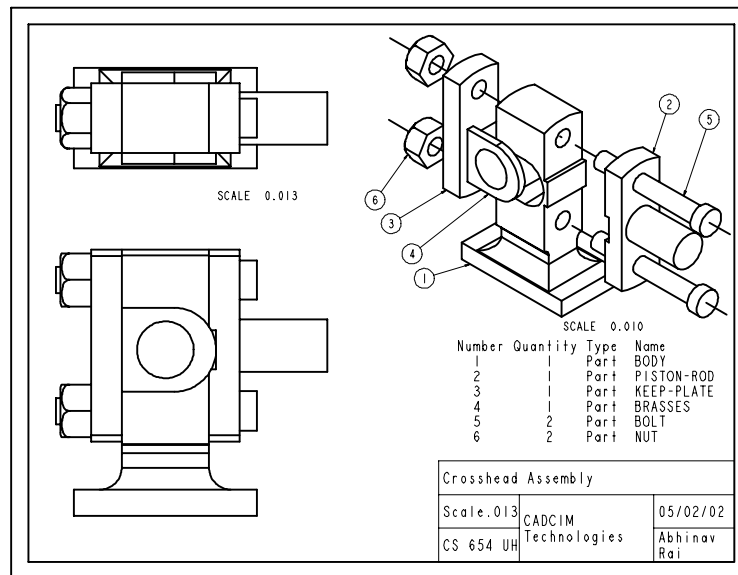


Figure 12-19 Drawing for Exercise 1

Answers to the Self-Evaluation Test

1 - T, 2 - T, 3 - T, 4 - T, 5 - F, 6 - Translate, 7- Enter Text, 8 - Enter Text, 9 - DELETE, 10 - TABLE CREATE.