

A 3D CAD model of a mechanical part, possibly a bracket or a connector, rendered in a light gray color. The part has a complex shape with multiple holes and a long, angled arm. It is positioned in the background, behind the chapter title.

Chapter 13

Working with the Drafting Workbench-II

Learning Objectives

After completing this chapter, you will be able to:

- *Insert additional sheets in the current drawing file.*
- *Insert frames and title blocks.*
- *Add annotations to the drawing views.*
- *Edit the annotations.*
- *Generate the Bill of Material (BOM).*
- *Generate balloons.*

INSERTING SHEETS IN THE CURRENT FILE

Menu: Insert > Drawing > Sheets > New Sheet

You can insert additional sheets to the current drafting file. This is a good practice when you need to generate the drawing views of all components of an assembly and also its other views such as the isometric view or a view with the Bill of Material (BOM), and balloons in a single drawing file. You will learn more about BOM and Balloons later in this chapter.

It is easier to manage all the drawings at the same time because a multisheet drawing file will act as a single storage space for the drawings of all the components of that assembly.



Note

The size of the multisheet document, with all the drawing views of components of the assembly, is less than the combined size of the individual drawing sheets of the drawing views of all components of the assembly.

To insert a new drawing sheet, choose **Insert > Drawing > Sheets > New Sheet** from the menu bar. A new sheet is added to the current drawing file, as shown in Figure 13-1.

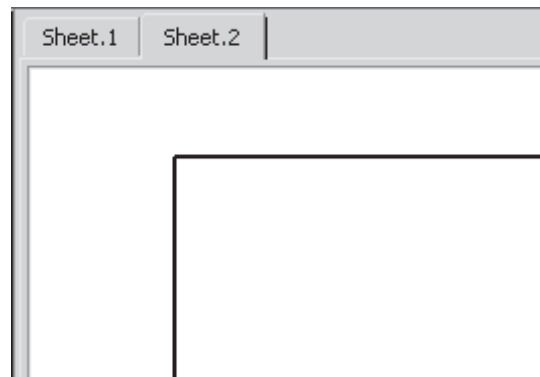


Figure 13-1 Partial view of the drawing with a new sheet added to the current file

The newly inserted sheet is active. To activate the first sheet, choose the **Sheet. 1** tab from the top of the drawing sheet or double-click on **Sheet. 1** in the specification tree. You can also choose the **Activate Sheet** option from the contextual menu. You can also activate the sheet by double-clicking on it in the specification tree.

You can also rename the sheet by selecting it from the specification tree and then choosing the **Properties** option from the contextual menu. Change the name of the sheet in the **Name** edit box and then edit the **Properties** dialog box.

To delete a drawing sheet, select the drawing sheet to be removed from the specification tree and invoke the contextual menu. Choose the **Delete** option. The selected sheet will be deleted.

INSERTING THE FRAME AND TITLE BLOCK

To insert the frame and the title block of the drawing sheet, you first need to set the drawing sheet to the edit background mode. To set this mode, choose **Edit > Background** from the menu bar. The background editing mode is invoked and the color of the sheet is automatically changed to grey. There are two options of inserting the frame and the title block. Both these methods are discussed next.

Automatic Insertion of the Frame and Title Block

Menu: Insert > Drawing > Frame and Title Block
Toolbar: Drawing > Frame Creation



This method is used to automatically insert the frame and the title block from those available in the system by default. For this, choose the **Frame Creation** button from the **Drawing** toolbar. The **Insert Frame and Title Block** dialog box is displayed, as shown in Figure 13-2.

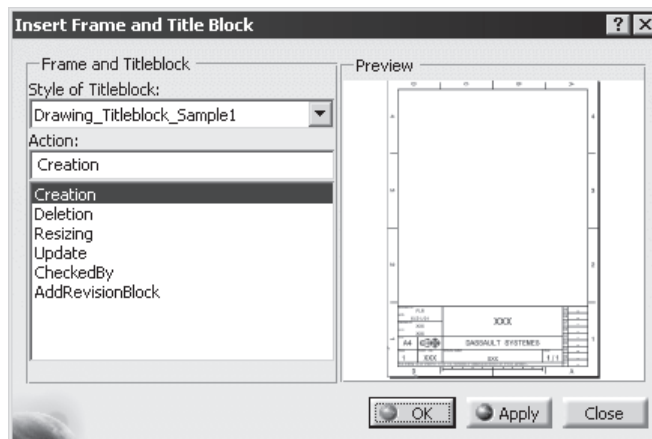


Figure 13-2 The Insert Frame and Title Block dialog box

The options available in this dialog box are discussed next.

Style of Titleblock

The **Style of Titleblock** drop-down in the **Frame and Titleblock** area is used to set the style of frame and titleblock. By default, the **Drawing_Titleblock_Sample1** title block is selected. The other two styles in this drop-down list are **Drawing_Titleblock_Sample2** and **Drawing_Titleblock_Sample_Enovia1**. Select the type of style from this drop-down list.

Action

The options in the **Action** list box are used to define the type of action that you need to execute. The options in this list are discussed next.

Creation

The **Creation** option is selected by default in the **Action** list box. This option is selected

to create the frame and the title block on the current drawing sheet. The preview of the frame and the title block to be created is shown in the preview area of the **Insert Frame and Title Block** dialog box. Figure 13-3 shows an **A2 ISO** sheet with the frame and title block inserted, using this option.

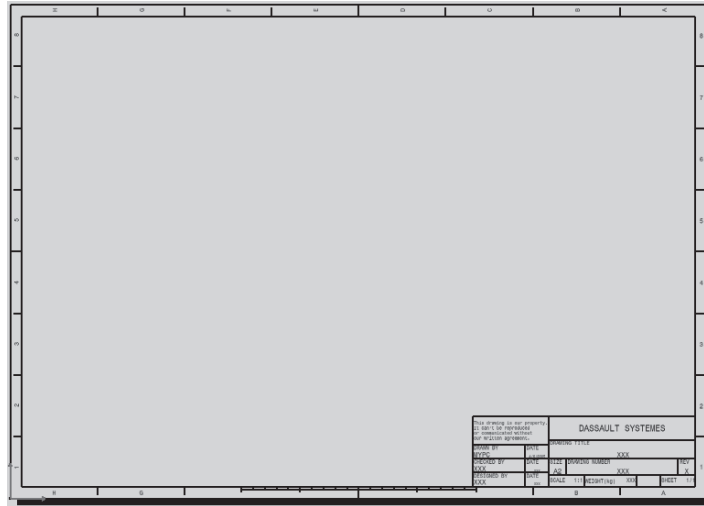


Figure 13-3 The drawing sheet, after adding the frame and title block to the drawing sheet

Deletion

The **Deletion** option in the **Action** list box is used to delete the existing frame and title block.

Resizing

The **Resizing** option is used to resize the frame and title block in case you have modified the size of the sheet after placing them. After modifying the size of the sheet, the frame and title block will not automatically adjust to the sheet boundaries. Therefore, to resize them, choose **Resize** from the **Action** list box and choose the **OK** button from the **Insert Frame and Title Block** dialog box. The frame and title block are now adjusted to the new sheet size.

Update

The **Update** option in the **Action** list box is used to update the frame and title block. This is done, in case you have modified some of the parameters of the sheet, such as the projection standard or sheet scale, or you have inserted new sheets. After performing the modifications, you need to update the frame and title block using this option.

Checked By

The **Checked By** option is used to specify the name of the person who checked the drawing sheet. To specify the name, select this option and choose the **OK** button from

the **Insert Frame and Title Block** dialog box; the **Controller's name** dialog box is displayed, as shown in Figure 13-4.

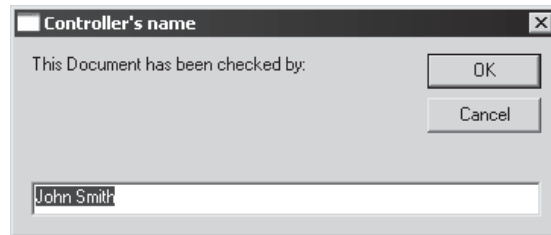


Figure 13-4 The Controller's name dialog box

Specify the name of the controller in the edit box and choose the **OK** button. The name specified in this dialog box will be displayed in the **CHECKED BY** column of the title block in the drawing sheet.

AddRevisionBlock

The **AddRevisionBlock** option is used to add revision blocks to the drawing sheets. To add a revision block, select this option and choose the **OK** button from the **Insert Frame and Title Block** dialog box. The preview of the revision block is displayed on the top right of the drawing sheet and the **Reviewer's name** dialog box is displayed. Specify the name of the reviewer, and choose the **OK** button; the **Description** dialog box is displayed. Specify the description in the edit box in this dialog box and choose the **OK** button. The information specified in these dialog boxes is displayed in the revision block. Similarly, you can also add other revision blocks to form a complete revision table.

After inserting the frame and the title block, you can specify parameters in the columns of the title block by double-clicking on the default parameter. The **Text Editor** dialog box is displayed. Specify the parameter and choose the **OK** button from the **Text Editor** dialog box.

Next, you need to exit the background editing mode. Choose **Edit > Working Views** from the menu bar.

Creating the Frame and Title Block Manually

You can also create frames and title blocks manually by invoking the background editing mode by choosing **Edit > Background** from the menu bar. Now, using the sketching tool in this mode, draw the sketch of the frame and title block. A drawing sheet, after drawing the sketch for the frame and title block using the sketching tools, is shown in Figure 13-5.

Adding Text in the Title Block

Menu:	Insert > Annotations > Text > Text
Toolbar:	Annotations > Text

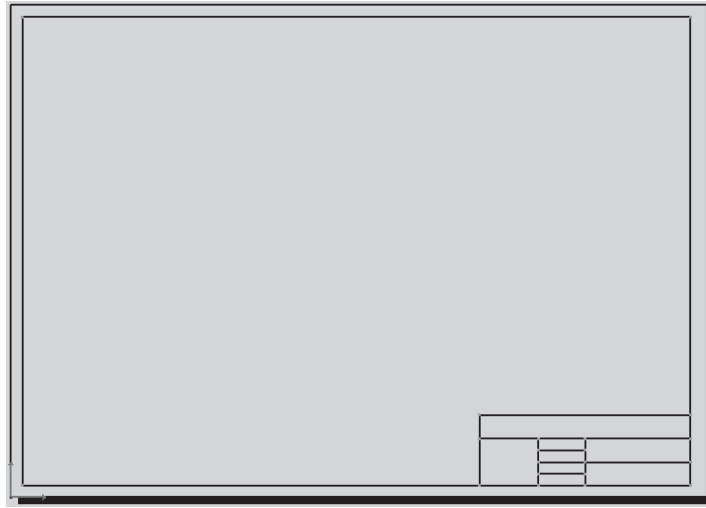


Figure 13-5 The drawing sheet, after drawing the frame and title block



After drawing the frame and title block, add text to the title block by invoking the background editing mode and then choosing the **Text** button from the **Annotations** toolbar; you are prompted to indicate the text anchor point. Select a point on the drawing sheet, to place the text. The **Text Editor** dialog box is displayed, as shown in Figure 13-6.

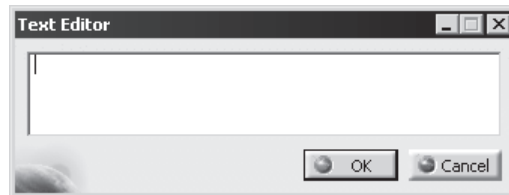


Figure 13-6 The Text Editor dialog box

Enter the text in the text box in the **Text Editor** dialog box and choose the **OK** button; the text will be displayed on the selected point on the drawing sheet. To relocate the text, select and drag it to a new location.

To modify the font, style, text height, and so on, select the text and choose the **Properties** option from the contextual menu; the **Properties** dialog box is displayed. Using the options available in this dialog box, you can modify the font, style, text height, and other properties of the text. After setting the parameters, choose the **OK** button from the **Properties** dialog box.



Note

You can also add text to the drawing views using this tool.

Inserting the Logo

You can also insert your company's logo in the title block by invoking the background editing mode and then choosing **Insert > Object** from the menu bar; the **Insert Object** dialog box is displayed. Select the **Create from File** radio button to insert a graphic created earlier. Choose the **Browse** button from the **Insert Object** dialog box; the **Browse** dialog box is displayed. Browse to the graphic file and double-click on it. The complete path name of the selected file is displayed in the **File** edit box of the **Insert Object** dialog box. Choose the **OK** button from the **Insert Object** dialog box; the graphic will be placed aligned to the lower left corner of the drawing sheet. You can move it by dragging. You can also resize it by holding it from one of the corners and dragging. Note that the aspect ratio of the image will be changed by resizing.

After adding the text and logo, choose **Edit > Working Views** from the menu bar to exit the sheet editing mode. On doing so, the color the sheet automatically changes to white. A drawing sheet, after completing the frame and title block, is displayed in Figure 13-7.

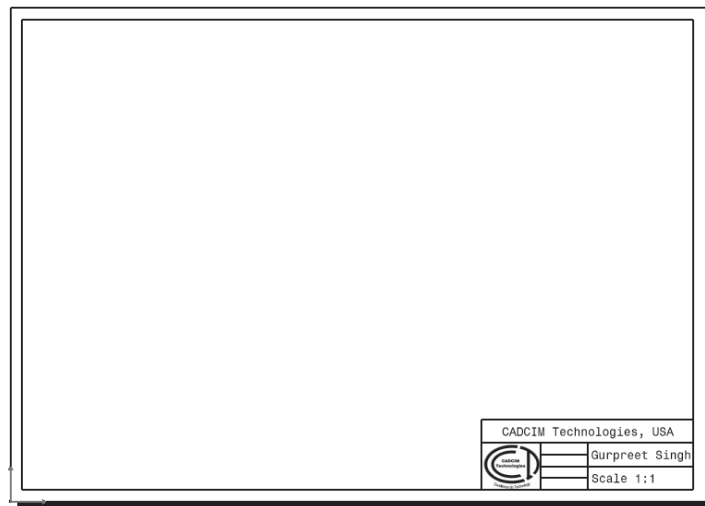


Figure 13-7 The drawing sheet, after completing the frame and title block

ADDING ANNOTATIONS TO THE DRAWING VIEWS

After generating the drawing views, you need to generate the dimensions in the drawing views and add other annotations, such as notes, surface finish symbols, geometric tolerance, and so on. Two types of annotations can be displayed in the drawing views, first is the generative annotation, in which you can generate the dimensions that were added, while creating the part in the **Part Design** workbench. The second is added manually to the geometry of drawing views, such as reference dimensions, notes, surface finish symbols, and so on. Both these annotations are discussed next.

Generating Dimensions

The dimensions applied to the sketches and features of the part can be generated and displayed on the drawing views using the following two tools.

Generating all Dimensions Together

Menu: Insert > Generation > Generate Dimensions
Toolbar: Generation > Dimension Generation > Generate Dimensions



The **Generate Dimensions** tool is used to generate all the dimensions of the drawing views in a single click. To do so, first invoke the **Options** dialog box and select **Drafting** from the **Mechanical Design** option on the left side of this dialog box. Now, choose the **Generation** tab and then select the **Filters before generating** option. Exit the **Options** dialog box and then choose the **Generate Dimensions** button from the **Dimension Generation** toolbar; after invoking it from the **Generation** toolbar. The **Dimension Generation Filters** dialog box is displayed, as shown in Figure 13-8.

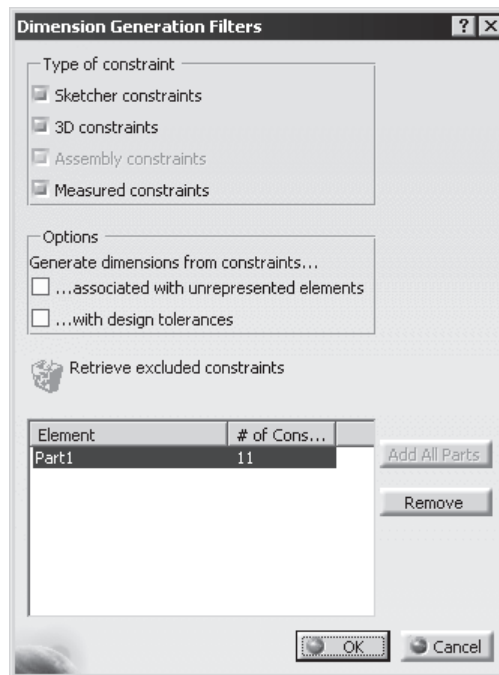


Figure 13-8 The *Dimension Generation Filter* dialog box

The options in this dialog box are discussed next.

Type of constraint Area

The options in the **Type of constraint** area are used to define the type of constraint that you need to generate. By default, all check boxes are selected in this area. While generating the dimensions of the assembly drawing views, the **Assembly constraint** check box is also invoked. Clear the check box of the type of constraint that you do not need to generate.

Options Area

The **associated with unrepresentable element** check box provided in the **Options** area is used to generate also those dimensions whose references are not displayed in the

drawing view. These include the dimensions that are referenced to a plane, axis, reference sketch and so on.

The **with design tolerances** check box is selected to generate the dimensions with design tolerances. Tolerance is also applied to the generated dimensions.

Retrieve excluded constraints

After deleting some of the generated dimensions, if you need to restore them, invoke the **Dimensions Generation Filters** dialog box and choose the **Retrieve excluded constraints** button. Now, choose the **OK** button from the **Dimensions Generation Filters** dialog box. The dimensions, that were deleted earlier, will be restored in their respective views.

Add All Parts

The **Add All Parts** button is available only while generating dimensions of the drawing views of an assembly. By default, only the name of the assembly is displayed in the **Element** column provided on the left of this button. If you choose this button, then the name of all the parts of the assembly are displayed in this column. Also, the dimensions of all components will be generated in the assembly drawing views. To remove a part from the selection set of the **Element** column, select it and choose the **Remove** button provided below the **Add All Parts** button.

After setting all parameters, choose the **OK** button from the **Dimension Generation Filter** dialog box. If the **Generated Dimension Analysis** dialog box is displayed, choose the **OK** button from it.



Tip. By default, dimensions are generated in all the views. To generate the dimensions in a particular view, select the view and choose the **Generate Dimensions** button.

Generating Dimensions Step by Step

Menu:	Insert > Generation > Generate Dimensions Step By Step
Toolbar:	Generation > Dimension Generation > Generate Dimensions Step By Step



To generate the dimensions using this tool, invoke it from the **Dimension Generation** toolbar. If the **Dimension Generation Filter** dialog box is displayed, set the option in this dialog box and choose the **OK** button to exit it. The **Step By Step Generation** dialog box is displayed, as shown in Figure 13-9.

Choose the **Next Dimension Generation** button from the **Step By Step Generation** dialog box. It will start generating dimensions one after the other. To generate all the dimensions at once, choose the **Dimension Generation Up To End** button from the **Step By Step Generation** dialog box. If you need to abort the dimension generation, choose the **Abortion in Dimension Generation** button from the **Step By Step Generation** dialog box.

If you do not generate a particular dimension, then choose the **Pause in Dimension Generation** button immediately after it is displayed on the drawing sheet. Now, choose the

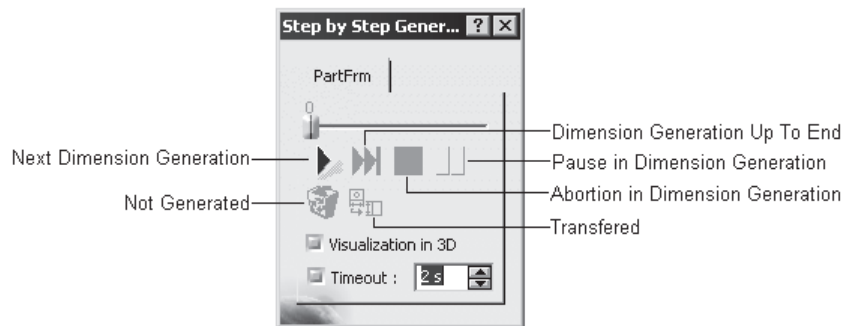


Figure 13-9 The Step by Step Generation dialog box

Click the **Not Generated** button from the **Step By Step Generation** dialog box. Choose the **Next Dimension Generation** button to resume generating the remaining dimensions.

To switch the dimensions from one view to another, pause the dimension generation process immediately after the dimension to be transferred is displayed on a drawing sheet. Now, choose the **Transfer** button and select the view on which you need to place the transferred dimension. After transferring it, continue generating the dimensions by choosing the **Next Dimension Generation** button.

The **Visualization in 3D** check box, selected by default, is used to display the dimensions in the solid model as they are generated in the drawing views. You can visualize the dimensions in the solid model also, if you have tiled the windows of the part and drawing.

The **Timeout** check box is selected to automatically generate the dimensions, after selecting the **Next Dimension Generation** button. You can also specify the time gap between the consecutive generated dimensions using the spinner on the right of this check box. If you clear this check box, only a single dimension will be generated on choosing the **Next Dimension Generation** button. Choose this button again to generate the next dimension.

Creating Reference Dimensions

Menu: Insert > Dimensioning > Dimensions > Dimensions
Toolbar: Dimensioning > Dimensions



In the **Drafting** workbench of CATIA V5, you can use the **Dimensions** tool to create reference dimensions. Before doing so, invoke the **Options** dialog box by choosing **Tools > Options** from the menu bar. Select **Drafting** from the **Mechanical Design** option on the left side of the **Options** dialog box, if it is not selected. Now, choose the **Dimension** tab from the right side of this dialog box and then select the **Dimension following the cursor (CTRL) toggle** check box. Now, to create reference dimensions, choose the **Dimensions** button from the **Dimensioning** toolbar; the **Tools Palette** toolbar will be displayed, as shown in Figure 13-10.

You can specify the type of dimension that you need to add using this toolbar. Now, select the geometrical element or elements to which you need to add the dimension; the dimension

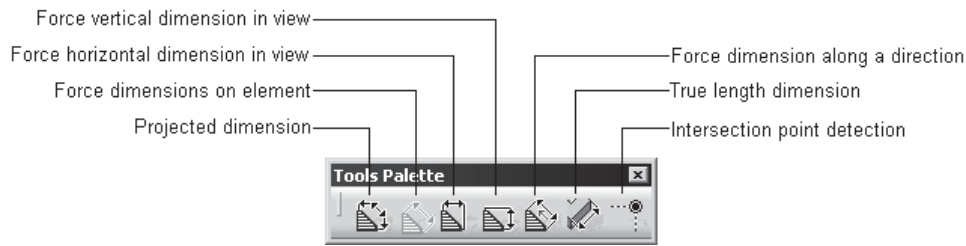


Figure 13-10 The **Tools Palette** toolbar

will be attached to the cursor. Move the cursor and specify a point on the drawing sheet to place the dimension.

Creating dimensions using the **Tools Palette** is the same as discussed, while dimensioning the entities in the **Sketcher** workbench. The remaining dimensioning tools are discussed next.

Creating Chamfer Dimensions

Menu: Insert > Dimensioning > Dimensions > Chamfer Dimension
Toolbar: Dimensioning > Dimensions > Chamfer Dimension



To create a chamfer dimension, choose the **Chamfer Dimension** button from the **Dimensions** toolbar, after invoking it from the **Dimensioning** toolbar. The **Tools Palette** toolbar is displayed, as shown in Figure 13-11.

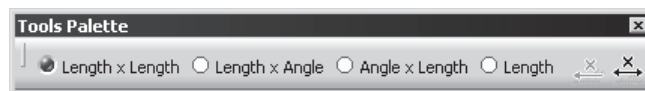


Figure 13-11 The **Tools Palette** toolbar

The options in this toolbar are used to define the form of the chamfer dimension. The **Length x Length** radio button is used to create the chamfer dimension using the chamfer lengths in the X and Y directions. The **Length x Angle** radio button is used to create the chamfer dimension by defining the length and angle of the chamfer. The **Angle x Length** radio button is used to create the chamfer by defining the angle first and then the length. The **Length** radio button is used to create the chamfer by defining the length of one side of the chamfer. After selecting the form of chamfer from the **Tools Palette** toolbar, move the cursor to the chamfered edge. You will notice that **2 1 3** and **3 1 2** digits swap their position, as you move the cursor to the upper or lower portion of the chamfer. When you move the cursor close to the upper portion of the chamfered edge, **2 1 3** will be displayed. This implies that the chamfered edge will be selected first and then it will select the upper edge as the dimensional reference. The chamfer dimension will be created with reference to the upper edge.

If you move the cursor close to the lower portion of the chamfered edge, then **3 1 2** will be displayed. This implies that the chamfered edge will be selected first and then it will select the edge close to the digit **2** as the dimensional reference. The resulting chamfered dimension

will be created with reference to that edge. Figures 13-12 through 13-15 show the selection sequences and the resulting chamfer dimensions.

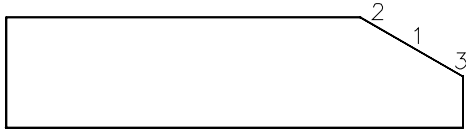


Figure 13-12 Selection sequence

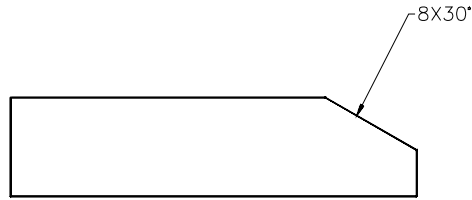


Figure 13-13 Resulting chamfer dimension

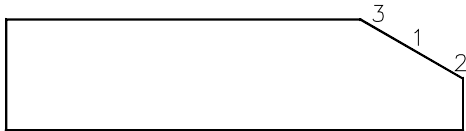


Figure 13-14 Selection sequence

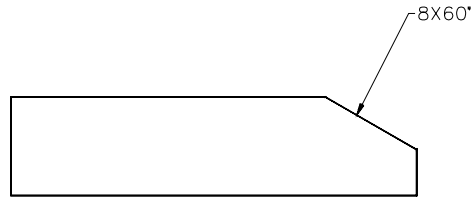


Figure 13-15 Resulting chamfer dimension

Select the chamfered edge and move the cursor to an appropriate location. Click a point on the drawing sheet to place the chamfer dimension.

Adding Datum Features

Menu: Insert > Dimensioning > Tolerancing > Datum Feature
Toolbar: Dimensioning > Tolerancing > Datum Feature



You can add the datum feature symbol to the drawing views, using the **Datum Feature** tool. The datum feature symbols are used as the datum references, while adding geometric tolerances to the drawing view. To add the datum feature symbol, choose the **Datum Feature** button from the **Tolerancing** toolbar, after invoking it from the **Dimensioning** toolbar; you are prompted to select an element or click the leader anchor point. Select the reference element on which you need to attach the datum feature. Its preview is attached to the cursor. Move the cursor to the desired location and click a point on the drawing sheet to place the datum feature; the **Datum Feature Creation** dialog box is displayed, as shown in Figure 13-16.



Figure 13-16 The *Datum Feature Creation* dialog box

Choose the **OK** button from this dialog box. The datum feature will be displayed attached to selected edge and placed at the specified location.

Adding Geometric Tolerance to Drawing Views

Menu: Insert > Dimensioning > Tolerancing > Geometrical Tolerance
Toolbar: Dimensioning > Tolerancing > Geometrical Tolerance



In shop floor drawings, you need to provide various other parameters, along with the dimensions and dimensional tolerance. These parameters can be a geometric condition, material condition, and so on. All these type parameters are defined using the geometric tolerance. To add the geometric tolerance to the drawing views, choose the **Geometric Tolerance** button from the **Tolerancing** toolbar. Select the element on which you need to add the geometric tolerance; the preview of the geometric tolerance is attached to the cursor. Move the cursor to an appropriate location and click on the drawing sheet to place the tolerance on that location; the **Geometrical Tolerance** dialog box is displayed, as shown in Figure 13-17.

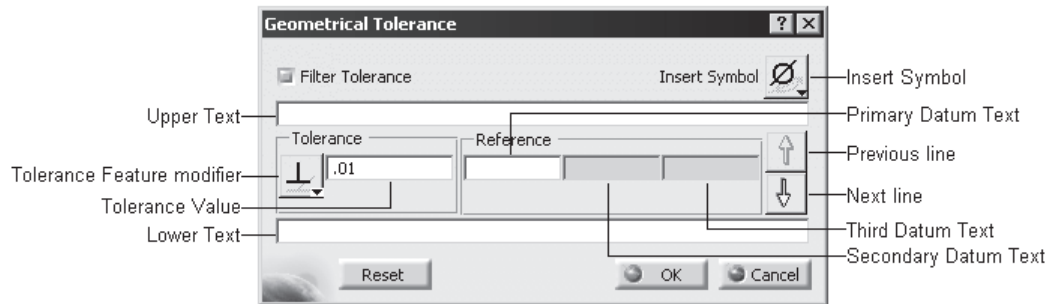


Figure 13-17 The *Geometrical Tolerance* dialog box

The options in this dialog box, to define tolerance, are discussed next.

Tolerance

The options in the **Tolerance** area are used to specify the geometrical condition for the tolerance and the value of tolerance. To do so, choose **Tolerance Feature modifier** button in the **Tolerance** area; a flyout menu is displayed. Choose the type of geometrical condition to apply from the flyout. Set the value of tolerance in the **Tolerance Value** edit box on the right of the **Tolerance Feature modifier** button.

Reference

The options in the **Reference** area of the **Geometrical Tolerance** dialog box are used to define the reference for applying the tolerance. After specifying the geometric condition and the value of tolerance, define the primary reference for applying the tolerance in the **Primary Datum Text** edit box, which is invoked in the **Reference** area. The **Secondary Datum Text** edit box is invoked, after you specify the primary reference. You can specify the second reference in the **Secondary Datum Text** edit box. After specifying the secondary reference, the **Third Datum Reference** edit box is invoked. This is used to define the third reference.

The **Upper Text** edit box is provided to specify the text above the geometrical tolerance. The **Lower Text** edit box is provided to specify the text below the geometrical tolerance.

The **Next line** button in the **Geometrical Tolerance** dialog box is used to move to the next line to define the parameters of specifying the second geometrical tolerance. When you choose this button, another set of **Tolerance** and **Reference** areas are displayed. You can set the tolerances in these areas. The **Previous line** button, above the **Next line** button, is used to return back to the previous line.

The **Insert Symbol** flyout can be used to insert symbols in the geometric tolerance box. The **Reset** button is used to set the parameters of the **Geometrical Tolerance** to default. After setting all parameters, choose the **OK** button from the **Geometrical Tolerance** dialog box.

Figure 13-18 shows a drawing after adding datum features and tolerances.

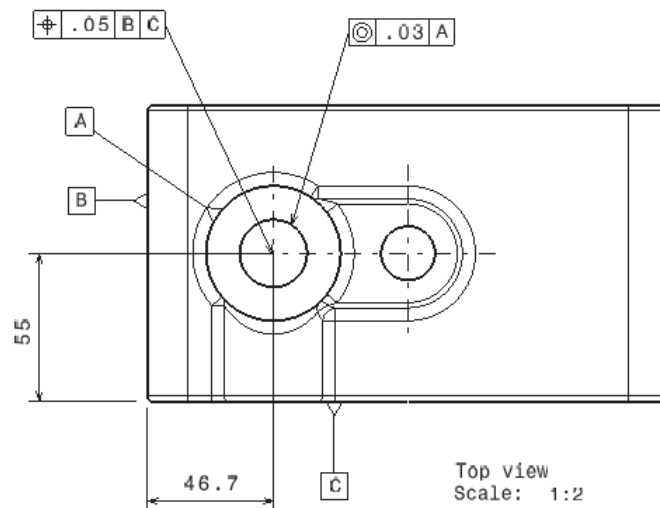


Figure 13-18 The drawing view, after adding datum feature and tolerance

Adding Surface Finish Symbols

Menu: Insert > Annotations > Symbols > Roughness Symbol
Toolbar: Annotations > Symbols > Roughness Symbol



The **Roughness Symbol** tool is used to add a roughness symbol to the drawing views. Choose the **Roughness Symbol** button from the **Symbols** toolbar, after invoking it from the **Annotations** toolbar. The **Symbols** toolbar is shown in Figure 13-19.

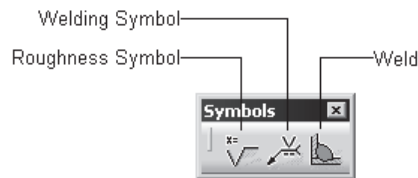


Figure 13-19 The Symbols toolbar

You are prompted to click on the roughness anchor. Click on the surface from where you need to place the roughness symbol. Its preview is displayed attached to the selected point and the **Roughness Symbol** dialog box is displayed, as shown in Figure 13-20.

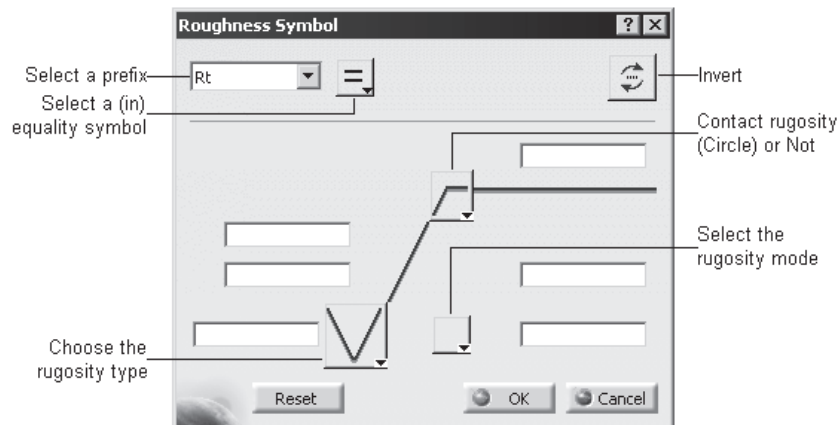


Figure 13-20 The Roughness Symbol dialog box

Select the prefix of the roughness symbol from the **Select a prefix** drop-down list. You can also specify the equality symbol using the **Select a (in) equality symbol** flyout on the right of the **Select a prefix** drop-down list. You can choose the type of rugosity using the **Choose the rugosity type** flyout. You can also choose the mode of rugosity using the **Select the rugosity mode** flyout. Set the value of the surface roughness using the edit boxes provided in the **Roughness Symbol** dialog box. You can also invert the surface roughness symbol using the **Invert** button in the **Roughness Symbol** dialog. After setting the parameters, choose the **OK** button.

Adding Welding Symbols

Menu: Insert > Annotations > Symbols > Welding Symbol
Toolbar: Annotations > Symbols > Welding Symbol



To add the welding symbol, choose the **Welding Symbol** button from the **Symbols** toolbar; you are prompted to select a first element or indicate the leader anchor point. Select the first element on which you need to add the welding symbol from the drawing sheet. Next, you are prompted to select the second element; select the second element from the drawing sheet. The preview of the welding symbol is attached to the cursor. Move the cursor and click on an appropriate location on the drawing sheet to place it; the **Welding creation** dialog box is displayed, as shown in Figure 13-21.

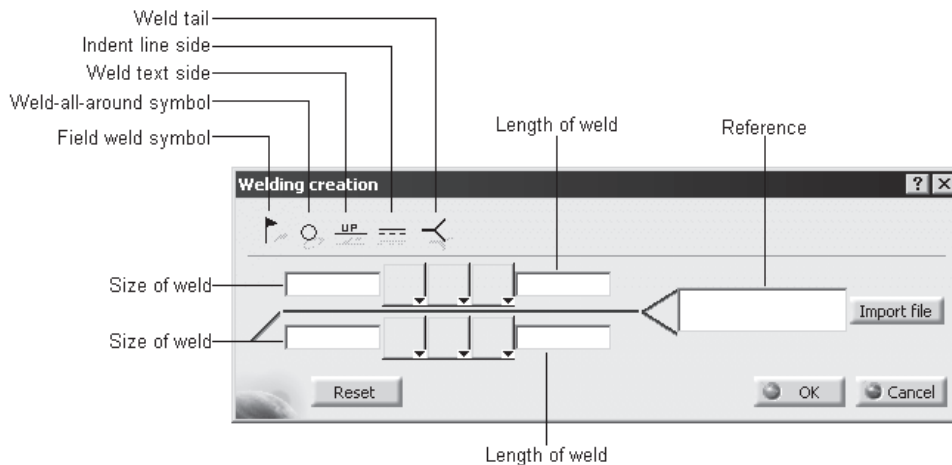


Figure 13-21 The Welding creation dialog box

The buttons on the upper row of the **Welding creation** dialog box are used to define the conditions of welding such as field weld symbol, weld allaround, side of the weld text, side of indent line, and the welding symbol tail.

Set the size of the weld in the **Size of weld** edit box. You can also define the parameters for the other side of the weld, if you weld the component on both sides, by setting the value of the size in the lower **Size of weld** edit box. All parameters regarding the size, length, and other properties on one side can also be defined for the other side.

The first button provided on the right of the **Size of weld** edit box is used to define the type of the weld such as the V-groove weld, single V-groove weld, fillet weld, and so on. To define the type of weld, choose the first button on the right of the **Size of weld** edit box; the flyout is displayed. Select the type of weld from this menu.

The second button provided on the right of the **Size of weld** edit box is used to define the profile of the weld such as straight, convex, concave, and so on. To define the profile of weld,

choose the second button on the right of the **Size of weld** edit box; the flyout is displayed. Select the type of weld profile from this menu.

The third button, provided on the right of the **Size of weld** edit box, is used to define the type of finish of the weld such as chiseled, hammered, machined, and so on. To define the profile of the weld, choose the third button on the right of the **Size of weld** edit box; the flyout is displayed. Select the type of welding finish from this menu.

The **Length of weld** edit box is used to specify the length of the weld. You can also define the reference of weld in the **Reference** edit box. The **Import file** button the left of the **Reference** edit box is used to browse the text file to insert the reference.

After setting all parameters, choose the **OK** button from the **Welding creation** dialog box. Figure 13-22 shows a welding symbol attached to a drawing view.

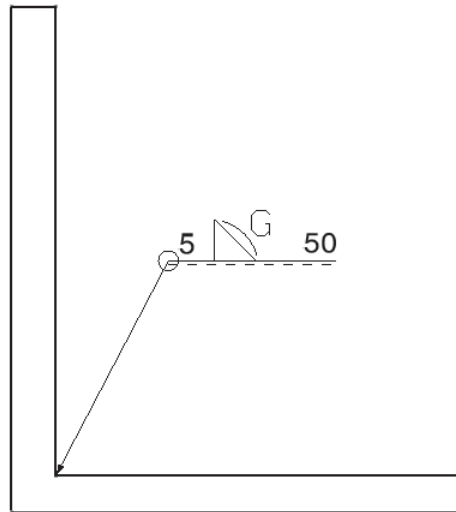


Figure 13-22 The drawing view, after adding the welding symbol

Applying Weld

Menu: Insert > Annotations > Symbols > Weld
Toolbar: Annotations > Symbols > Weld



To apply a weld, choose the **Weld** button from the **Symbols** toolbar; you are prompted to select the first edge. Select the first edge on which you need to apply the weld; you are prompted to select the second edge. Select the second edge from the drawing sheet; the **Welding Editor** dialog box is displayed, as shown in Figure 13-23.

The preview of the weld is also displayed on the drawing sheet. Depending on the entities selected, the system will choose the most appropriate type of weld. You can also select the weld type to be applied from the flyout that is displayed, when you choose the **Change Type**



Figure 13-23 The Welding Editor dialog box

button on the right of the **Thickness** spinner. Set the value of thickness using the **Thickness** spinner. If required, set the value of the angle using the **Angle** spinner.

After setting all parameters, choose the **OK** button from the **Welding Editor** dialog box. Figure 13-24 shows a drawing view after applying the weld.

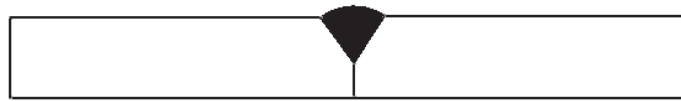


Figure 13-24 The drawing view, after adding the welding symbol

EDITING ANNOTATIONS

You can edit the annotations added to the drawing views by double-clicking on them to display their respective dialog boxes. You can edit the parameters of the annotations using these dialog boxes.

GENERATING THE BILL OF MATERIAL (BOM)

After generating the drawing views of an assembly, it is very necessary to generate the Bill of Material (BOM). The BOM is a table that provides you information related to the number of components in an assembly, their name, their quantity, and so on. You can add the BOM to the drawing sheet to display the part list of the components used in the assembly. The BOM placed on the drawing sheet is parametric in nature. Therefore, if you add or delete a part from the assembly, the change will be reflected in the BOM on the drawing sheet. Before generating the BOM, you need to apply numbers to the components of the assembly. This will help in providing the serial number to the component, which will in turn help to number the components, while generating balloons.

To number the components, switch to the assembly file from which the drawing views are generated. Choose the **Generate Numbering** button from the **Product Structure Tools** toolbar. Now, select **Product1** from the specification tree; the **Generate Numbering** dialog box is displayed, as shown in Figure 13-25.

The **Integer** radio button is selected by default in the **Mode** area of the **Generate Numbering** dialog box. To generate the numbering in alphabets, select the **Letters** radio button.

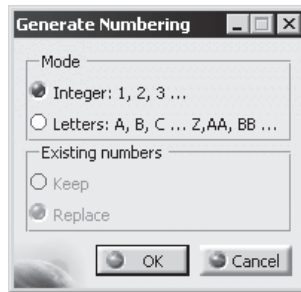


Figure 13-25 The *Generate Numbering* dialog box

The options in the **Existing numbers** area are used to specify whether you need to keep or remove the existing numbers, if you have generated them once. Choose the **OK** button from the **Generate Numbering** dialog box.

After generating the numbers, you need to edit the format of the BOM in the assembly environment. This is because by default, the number column is not available in the BOM format. To edit the BOM format, choose **Analyze > Bill of Material** from the menu bar; the **Bill of Material : Product1** dialog box is displayed. Choose the **Define formats** buttons from the **Bill of Material : Product1** dialog box; the **Bill of Material : Define formats** dialog box is displayed. Select **Number** from the **Hidden properties** selection area and choose the **Show Properties** button. The selected property is now displayed in the **Displayed properties** selection area. You will notice that the **Number** property is placed at the bottom in the **Display properties** selection area. You need to place it on the top in the **Display properties** selection area. To reposition the property, select **Number** from the **Display properties** selection area and choose the **Change order** button. Now, select the property on the top in the **Display properties** selection area. You will notice that **Number** properties is now placed on the top. You also need to hide some of the unwanted properties displayed in the **Display properties** selection area, such as **Nomenclature** and **Revision**. To hide a property, select the property to be hidden, from the **Display properties** selection area and choose the **Hide properties** button.

After setting the properties, choose the **OK** button from the **Bill of Material : Define formats** dialog box and then choose the **OK** button from the **Bill of Material : Product1** dialog box. Save the assembly file and switch back to the drawing window.

To generate a BOM, choose **Insert > Generation > Bill of Material** from the menu bar. You are prompted to click at a location to insert the Bill of Material. Click on the drawing sheet. The BOM and the recapitulation list are placed on the selected point. Figure 13-26 shows the drawing sheet after generating the BOM.

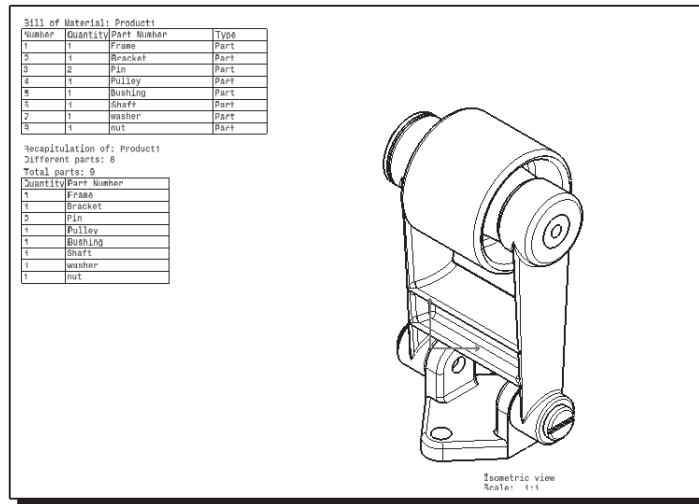


Figure 13-26 The drawing sheet, after generating the BOM

GENERATING BALLOONS

Menu: Insert > Generation > Balloon generation
Toolbar: Generation > Dimension Generation > Generate Balloons



The **Generate Balloons** tool is used to generate balloons that are attached to the drawing view of an assembly. The naming of the balloons depends on the sequence of the parts in the BOM. To generate balloons, you need to make sure that the numbering is done to the assembly. Numbering the components of the assembly has been discussed earlier, while generating the BOM. Now, choose the **Generate Balloons** button from the **Dimension Generation** toolbar. Balloons are automatically attached to the components of the assembly in the active drawing view. By default, they are placed arbitrarily on the drawing sheet. You need to move them manually and place them at an appropriate location. Figure 13-27 shows a drawing sheet after generating balloons.

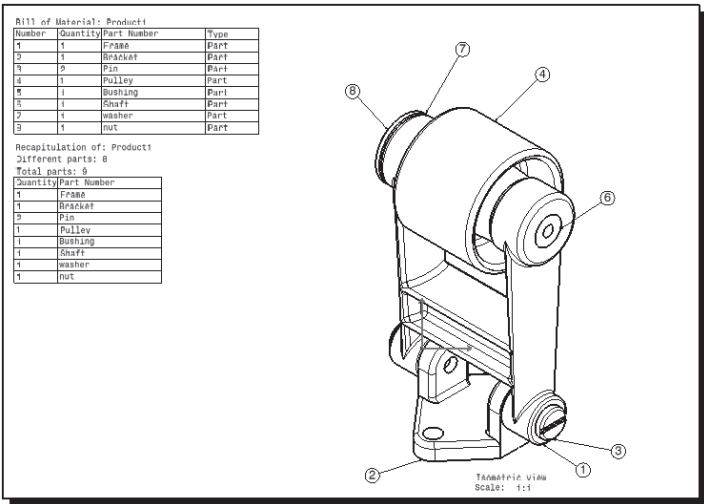


Figure 13-27 The drawing sheet, after generating balloons

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 13-28. After creating it, you need to generate the front, top, and the isometric views in the **Drafting** workbench. You also need to generate the dimensions in the drawing views. The views and dimensions are shown in Figure 13-29. **(Expected time: 45 min)**

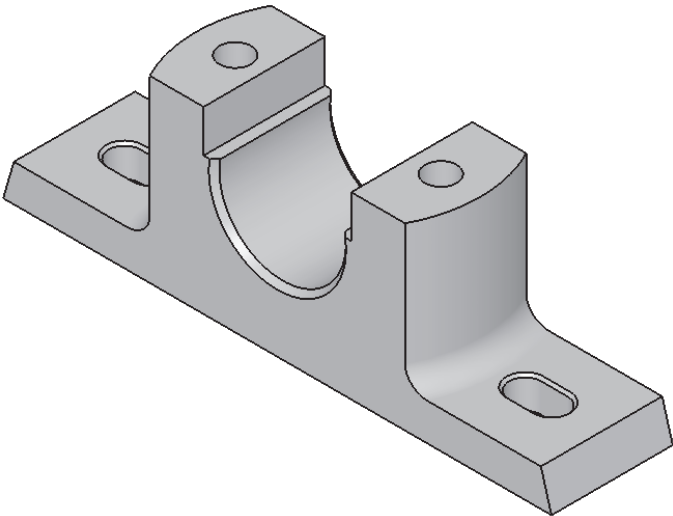


Figure 13-28 Model for Tutorial 1

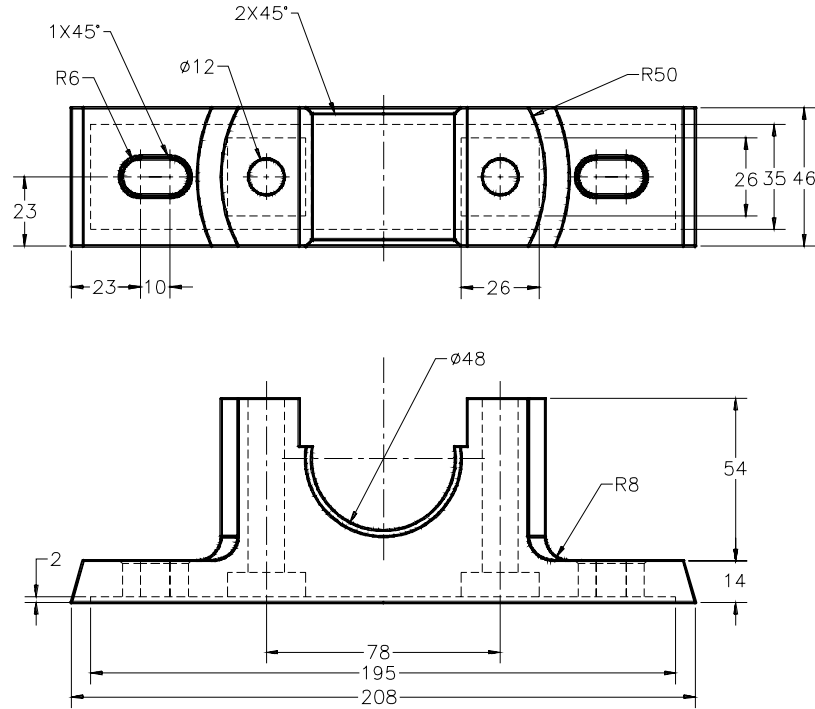


Figure 13-29 Views and dimensions for Tutorial 1

The following steps are required to complete this tutorial:

- Create the model in the **Part** workbench and save the file in the *c13* folder.
- Start a new file in the **Drafting** workbench with a standard A2 sheet.
- Set the projection standard to the third angle.
- Create a standard title block and frame in the background editing mode, refer to Figure 13-30.
- Generate the front view, refer to Figure 13-31.
- Generate the top view, refer to Figure 13-32.
- Generate the isometric view, refer to Figure 13-33.
- Generate the dimensions, refer to Figure 13-35.
- Arrange the dimensions and delete the unwanted ones, refer to Figure 13-36.

Creating the Model

First you need to create the model in the **Part** workbench.

- Start a new file in the **Part** workbench.
- Create the model, refer to Figure 13-29 for dimensions.

3. Create a folder with the name *c13* in the *CATIA* folder and save the model in it with the name *c13tut1*.

Start the New File in the Drafting Workbench

After creating and saving the model, you need to starting a new file in the **Drafting** workbench.

1. Choose **File > New** from the menu bar. Select **Drawing** from the **New** dialog box and choose the **OK** button; the **New Drawing** dialog box is displayed.
2. Select the **A2 ISO** option from the **Format** drop-down list. Choose the **OK** button from the **New Drawing** dialog box. Set the projection standard to the third angle.

A new file in the **Drafting** workbench is started.

3. Set the projection mode to the third angle using the **Properties** dialog box.

Creating the Title Block and Frame

After starting a new file in the **Drafting** workbench, you need to create the title block and frame by invoking the background editing mode.

1. Choose **Edit > Background** from the menu bar to invoke the background editing mode. The color of the sheet changes to grey.
2. Choose the **Frame Creation** button from the **Drawing** toolbar; the **Insert Frame and Title Block** dialog box is displayed.



The **Creation** option is selected by default in the **Action** list box. You can select this option to create the feature and the title block.

3. Choose the **OK** button from the **Insert Frame and Title Block** dialog box. The drawing sheet, after creating the title block and frame, is shown in Figure 13-30.

After creating the title block, you need to exit the background editing mode.

4. Choose **Edit > Working Views** from the menu bar. You are returned back to the working views mode.

Generating the Front View

After creating the title block and frame, you need to generate the front view.

1. Choose the **Front View** button from the **Projections** toolbar. You are prompted to select a reference plane on a 3D geometry.
2. Choose **Window > c13tut1.CATPart** from the menu bar. The part document is displayed.
3. Select the YZ plane from the specifications tree. The drawing file is invoked.

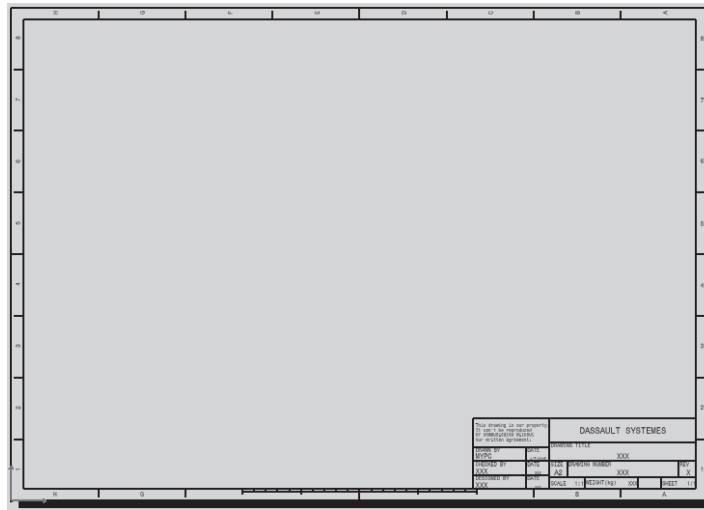


Figure 13-30 The drawing sheet, after creating the title block and frame

The preview of the front view is displayed and the knob is also provided to set the orientation of the front view.

4. Move the cursor on the frame of the preview of the front view; the cursor is replaced by a hand cursor. Press and hold down the left mouse button and drag the cursor close to the lower left corner of the drawing sheet, refer to Figure 13-31.
5. Click anywhere on the drawing sheet to generate the front view. The drawing sheet, after generating the front view, is shown in Figure 13-31.

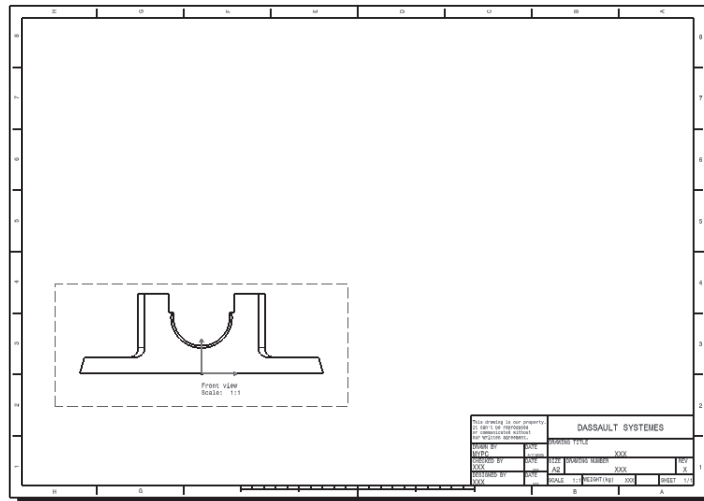


Figure 13-31 The drawing sheet, after generating the front view

Generating the Top View

After generating the front view, you need to generate the top view by using the **Projection View** tool.

1. Choose the **Projection View** button from the **Projections** toolbar.
2. Move the cursor vertically upward from the front view; the preview of the projected view is displayed attached to the cursor.
3. Click on the drawing sheet to place the top view, refer to Figure 13-32.
4. Move the view notes by dragging them, refer to Figure 13-32.

The drawing sheet, after generating the top view, is shown in Figure 13-32.

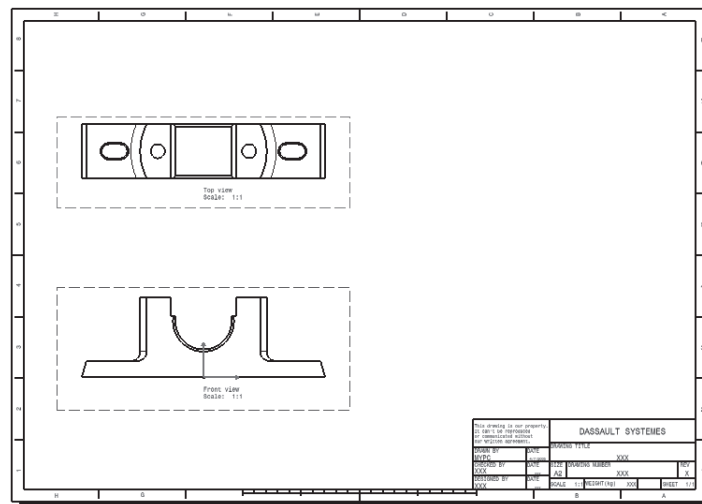


Figure 13-32 The drawing sheet, after generating the top view

Generating the Isometric View

Next, you need to generate the isometric view.

1. Choose the **Isometric View** button from the **Projections** toolbar; you are prompted to select a reference plane on a 3D geometry.
2. Choose **Window > c13tut1.CATPart** from the menu bar; the part window is displayed.
3. Select the front face of the model from the geometry area; the drawing window is displayed. The preview of the isometric view is displayed on the drawing sheet, along with the knob, to orient the isometric view.

You may need to move the isometric view, if it is placed outside the drawing sheet.

4. Drag the view to an appropriate location by holding its frame, refer to Figure 13-33.
5. Click anywhere on the drawing sheet to generate the isometric view. The drawing sheet, after generating the isometric view, is shown in Figure 13-33.

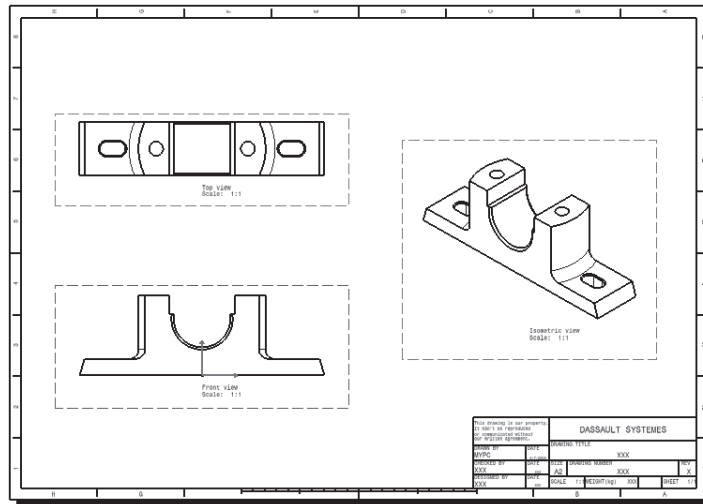


Figure 13-33 The drawing, sheet after generating the isometric view

Generating Dimensions

After generating the drawing view, you need to generate the dimensions. Before generating them, you need to turn on the display of the hidden lines of the top and the front view in order to define some of the dimensions from the hidden lines. You also need to turn on the display of the center lines. After this, you need to turn off the display of the view frames.

1. Hold the CTRL key down and select **Front view** and **Top view** from the specification tree. Right-click to invoke the contextual menu.
2. Choose the **Properties** option; the **Properties** dialog box is displayed.
3. Select the **Hidden Lines** and the **Center line** check boxes from the **Dress-up** area and choose the **OK** button from the **Properties** dialog box.

Next, you need to turn the display of the view frames off.

4. Hold the CTRL key down and select **Front view**, **Top view**, and **Isometric view** from the specification tree.
5. Invoke the contextual menu and choose the **Properties** option from it.
6. Clear the **Display View Frame** check box from the **Visualization and Behavior** area and choose the **OK** button from the **Properties** dialog box.

The drawing sheet, after modifying the display, is shown in Figure 13-34.

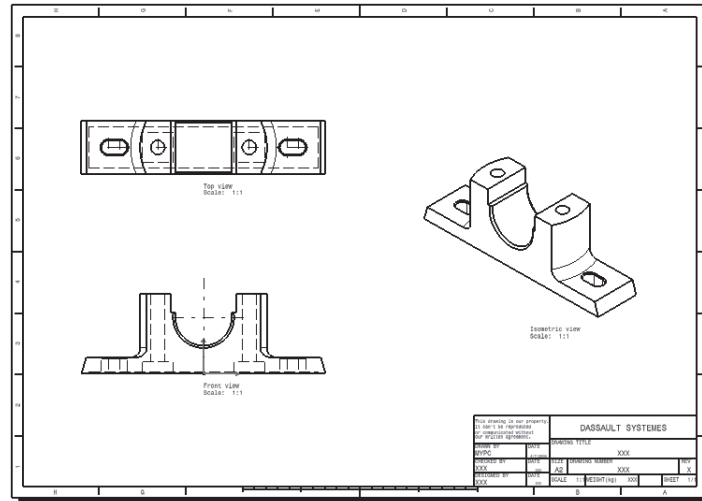


Figure 13-34 Drawing sheet after modifying the display

Now, you need to generate the dimensions.

7. Choose the **Generate Dimensions** button from the **Dimension Generation** toolbar; the **Dimension Generation Filters** dialog box is displayed.
8. Select the **associated with unrepresented elements** check box from the **Options** area of the **Dimension Generation Filters** dialog box and choose the **OK** button. Choose the **OK** button from the **Generated Dimension Analysis** dialog box, if it is displayed.

The progress bar is displayed and all the dimensions are generated in the front and top views, as shown in Figure 13-35.

After generating the dimensions, you may need to delete the repeated ones, if they are generated. Also, the dimensions are placed staggered on the drawing sheet, and so you need to arrange them.

9. Delete the repeated dimensions by selecting the dimensions and pressing the DELETE key.
10. Select the dimensions one by one and drag them to the desired location, refer to Figure 13-36.

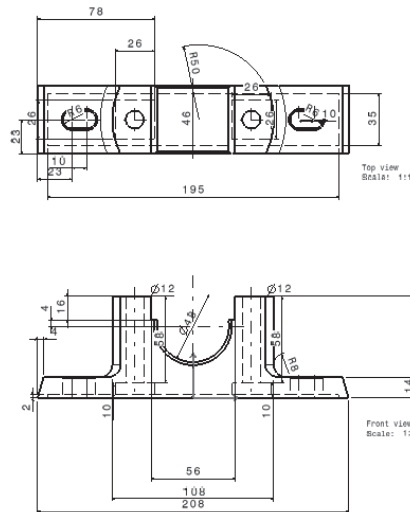


Figure 13-35 The drawing sheet, after generating dimensions

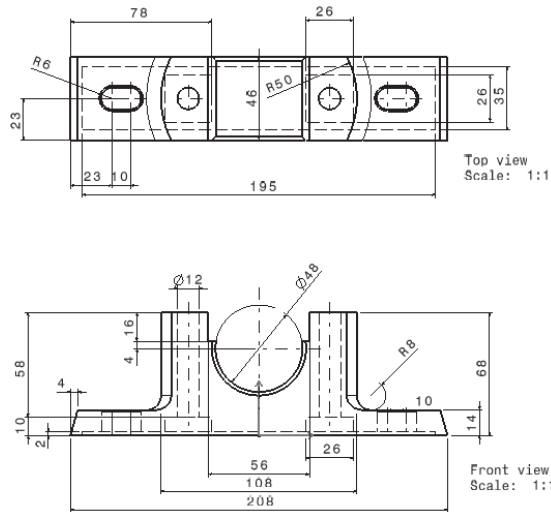


Figure 13-36 Drawing sheet after arranging dimensions

The final drawing sheet is shown in Figure 13-37.

Saving and Closing the Files

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.
2. Enter the name of the file as *c13tut1.CATDrawing* in the **File name** edit box and choose the **Save** button. The file will be saved in the *|My Documents|CATIA|c13* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

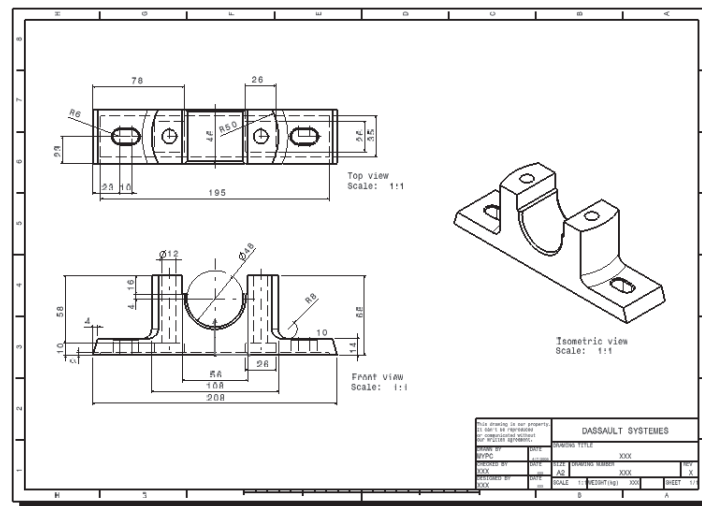


Figure 13-37 Final drawing sheet

Tutorial 2

In this tutorial, you will generate the front, top, right, and isometric views of the V-Block assembly created in the Exercise 1 of Chapter 12. You also need to generate the BOM and balloons. **(Expected time: 30 min)**

The following steps are required to complete this tutorials.

- Copy the V-Block folder to the c13 folder.
- Open the assembly file.
- Start a new file in the **Drafting** workbench.
- Generate the front view, refer to Figure 13-38.
- Generate the top view, refer to Figure 13-38.
- Generate the right view, refer to Figure 13-38.
- Generate the isometric view, refer to Figure 13-38.
- Generate the BOM, refer to Figure 13-39.
- Generate the balloons, refer to Figure 13-39.

Copying and Opening the Assembly Document

- Copy the *V-Block* folder from the `\My Document\CATIA\c12` folder to the *c13* folder.
- Start CATIA V5 and open the assembly document of Exercise 1 of Chapter 12 that you copied in the folder of the current chapter.

Start the New File in the Drafting Workbench

- Choose **File > New** from the menu bar. Select **Drawing** from the **New** dialog box and choose the **OK** button from the **New** dialog box. The **New Drawing** dialog box is displayed.

2. Select the **A2 ISO** option from the **Format** drop-down list. Choose the **OK** button from the **New Drawing** dialog box.
3. Set the projection standard to the third angle.

Creating the Title Block and Frame

After starting a new file in the **Drafting** workbench, you need to create the title block and frame by invoking the background editing mode.

1. Choose **Edit > Background** from the menu bar to invoke the background editing mode.
2. Choose the **Frame Creation** button from the **Drawing** toolbar; the **Insert Frame and Title Block** dialog box is displayed.
3. Accept the default option and choose the **OK** button from the dialog box.



After creating the title block, you need to exit the background editing mode.

4. Choose **Edit > Working Views** from the menu bar.

Generating the Drawing Views

Next, you need to generate the front, top, right, and isometric drawing views of the V-Block assembly.

1. Generate the front, top, right, and the isometric views of the V-Block assembly. The drawing sheet, after generating the views, is shown in Figure 13-38.

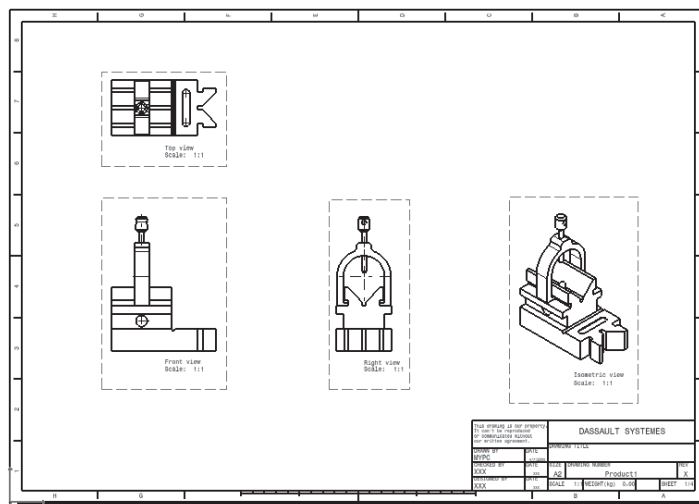



Figure 13-38 The drawing sheet, after generating the front, top, right, and isometric views

Generating the BOM and Balloons


Before generating the BOM and balloons, you need to number the components of the assembly and then modify the BOM format.

1. Choose **Window > c12exr2.CATProduct** from the menu bar.


The assembly window is invoked. Numbering the components of the assembly will help you generate the balloons and number the components in the BOM.

2. Choose the **Generate Numbering** button from the **Product Structure Tools** toolbar. You are prompted to select a component to manage its representation. 
3. Select **Product1** from the specification tree; the **Generate Numbering** dialog box is displayed.
4. Choose the **OK** button from the **Generate Numbering** dialog box.

After numbering the components of the assembly, you need to redefine the BOM format.

5. Choose **Analyze > Bill of Material** from the menu bar; the **Bill of Material : Product1** dialog box is displayed.
6. Choose the **Define formats** button from the **Bill of Material : Product1** dialog box; the **Bill of Material : Define formats** dialog box is invoked.
7. Select **Number** from the **Hidden properties** selection area and choose the **Show Properties** button on the left of the **Hidden properties** selection area. The **Number** property will be displayed in the **Displayed properties** selection area. 

You will notice that the **Number** property is displayed in the **Displayed properties** selection area at the end. You need to change the sequence of the properties by moving the **Number** property to the top.

8. Select the **Number** property from the **Displayed properties** selection area and choose the **Change order** button provided on the right of the **Displayed properties** selection area. 
9. Select the **Quantity** property from the **Displayed properties** selection area. You will notice that the **Number** property is now placed on the top.

Next, you need to remove some of the properties from the **Displayed properties** selection area.

10. Select **Nomenclature** from the **Displayed properties** selection area and choose the **Hide properties** button provided on the right. 

The **Nomenclature** property is moved to the **Hidden properties** selection area.

11. Similarly, remove the **Revision** property from the **Display properties** selection area.
12. Choose the **OK** button from the **Bill of Material : Define formats** dialog box and then choose the **OK** button from the **Bill of Material : Product1** dialog box.
13. Choose **File > Save** from the menu bar. Invoke the drawing window.

Next, you need to generate balloons on the isometric view.

14. Double-click on the isometric view in the specification tree.
15. Choose the **Generate Balloons** button from the **Dimensions Generation** toolbar.
The balloons are generated and displayed in the isometric view.
16. Move the balloons to an appropriate location, if they are placed arbitrarily.



Next, you need to generate the BOM.

17. Choose **Insert > Generation > Bill of Material** from the menu bar; you are prompted to click at a location to insert the Bill of Material.
18. Click near the upper right corner of the drawing sheet to define the upper left corner of the BOM, refer to Figure 13-39.
19. Move the BOM, if it is placed outside the drawing sheet. Turn off the display of the view frames of all the views.

The final drawing sheet, after generating the BOM and balloons, is shown in Figure 13-39.

20. Save the file.

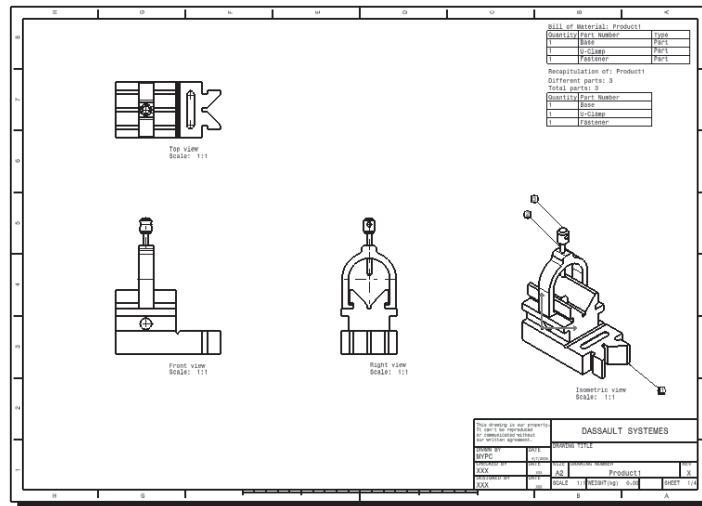


Figure 13-39 The final drawing sheet

SELF-EVALUATION TEST

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

1. You cannot insert additional sheets to the current drafting file. (T/F)
2. The **Type of Titleblock** drop-down list in the **Frame and Titleblock** area of the **Insert Frame and Title Block** dialog box is used to set the style of the frame and titleblock. (T/F)
3. The options in the **Reference** area of the **Geometrical Tolerance** dialog box are used to define the reference for applying the tolerance. (T/F)
4. You cannot apply the datum feature symbol to the drawing views. (T/F)
5. The naming of the balloons depends on the sequence of the parts in the BOM. (T/F)
6. The _____ tool is used to add the welding symbol to the drawing views.
7. The options in the _____ area of the **Dimension Generation Filters** dialog box are used to define the type of constraint that you need to generate.
8. Use the _____ edit box from the **Welding creation** dialog box to set the size of the weld.
9. The _____ option in the **Action** list box of the **Insert Frame and Title Block** dialog box is used to delete the existing frame and title block.

10. The options in the _____ area of the **Geometrical Tolerance** dialog box are used to specify the geometrical condition for the tolerance and the value of tolerance.

REVIEW QUESTIONS

Answer the following questions.

1. To switch the dimensions from one view to another, use the **Transfer** button from the **Step By Step Generation** dialog box. (T/F)
2. The **Upper Text** edit box provided in the **Geometrical Tolerance** dialog box is used to specify the text above the geometrical tolerance. (T/F)
3. To add the surface finish symbols, choose the **Roughness Symbol** button from the **Symbols** toolbar. (T/F)
4. You cannot insert a logo in the title block. (T/F)
5. You cannot generate a BOM in the **Drafting** workbench of CATIA V5. (T/F)
6. The _____ option in the **Geometrical Tolerance** dialog box is used to set all the parameters of the geometrical tolerance to default.
7. The _____ edit box provided in the **Welding creation** dialog box is used to specify the length of the weld.
8. To exit the background editing mode, after inserting the frame and title block, choose the _____ from the menu bar.
9. The _____ option in the **Action** list box of the **Insert Frame and Title Block** dialog box is used to resize the frame and title block.
10. The _____ tool is used to generate the dimensions step by step.

EXERCISES

Exercise 1

Generate the front, top, right, and isometric views of the Blower assembly created in Tutorial 1 of Chapter 11. The drawing view will be generated with the scale factor of 1:5. After generating the drawing views, you need to generate the BOM and balloons, as shown in Figure 13-40. (Expected time: 30 min)

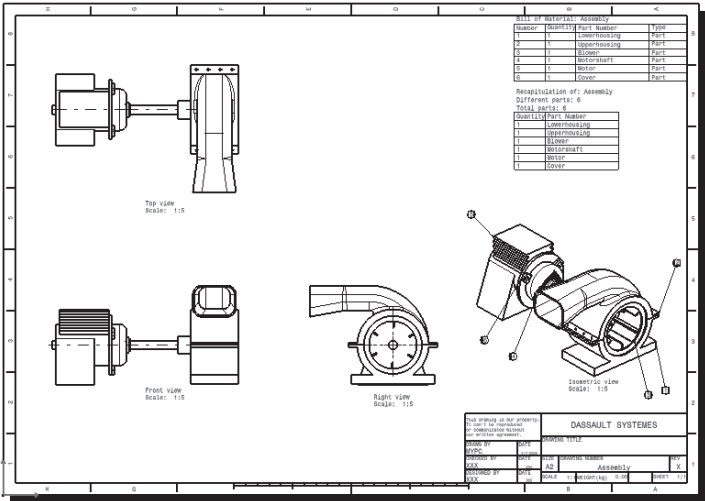


Figure 13-40 Drawing views for Exercise 1

Answers to Self-Evaluation Test

1. F, 2. T, 3. T, 4. F, 5. T, 6. Welding Symbol, 7. Type of constraint, 8. Size of weld, 9. Deletion, 10. Tolerance