

A 3D CAD model of a mechanical assembly, likely a pump or valve component, rendered in a light gray color. The assembly consists of a main body with a large central opening, a flange with four bolt holes, and a long, angled arm extending from the bottom. The model is shown in a perspective view, highlighting its complex geometry and assembly structure.

Chapter 8

Assembly Modeling-I

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the concept of the bottom-up and top-down assemblies.*
- *Create components of the top-down assemblies in the assembly file.*
- *Insert components of the bottom-up assemblies in the assembly file.*
- *Understand various assembly constraints and use them to assemble components.*
- *Move and rotate individual components in the assembly file.*

ASSEMBLY MODELING

An assembly is defined as a design consisting of two or more components tied together at their respective working positions. In Autodesk Inventor, the components of the assembly will be bound using the parametric assembly constraints. Because the assembly constraints are parametric in nature, you can modify or delete them whenever you want. In Autodesk Inventor, the assemblies are created in the **Assembly** module. To proceed to the assembly module, invoke the **Open** dialog box and select the **Standard (mm).iam** file, as shown in Figure 8-1.

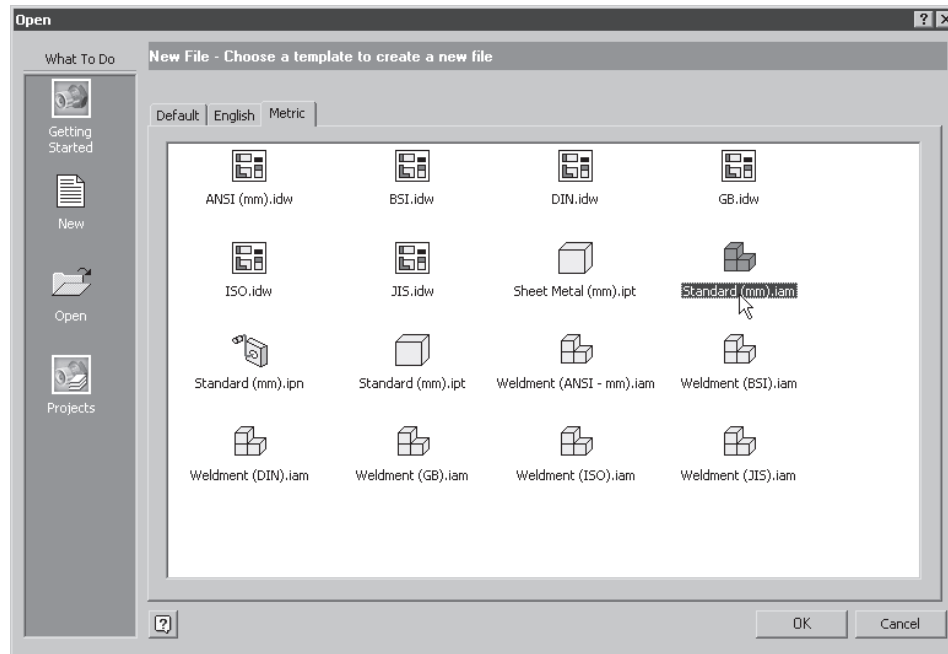


Figure 8-1 Opening an assembly file from the **Metric** tab of the **Open** dialog box

When you select the assembly file, the assembly environment will be activated. The screen display of Autodesk Inventor in the **Assembly** module is shown in Figure 8-2. This figure also displays the **Assembly Panel** panel bar, browser, and the **Assembly Panel** toolbar.



Note

*When you enter the **Assembly** module, you will notice that very few tools in the **Assembly Panel** panel bar and toolbar are available. All these tools will be available once you insert or create a component.*

TYPES OF ASSEMBLIES

In Autodesk Inventor, you can create two types of assemblies: top-down assemblies and bottom-up assemblies. Both these assemblies are discussed next.

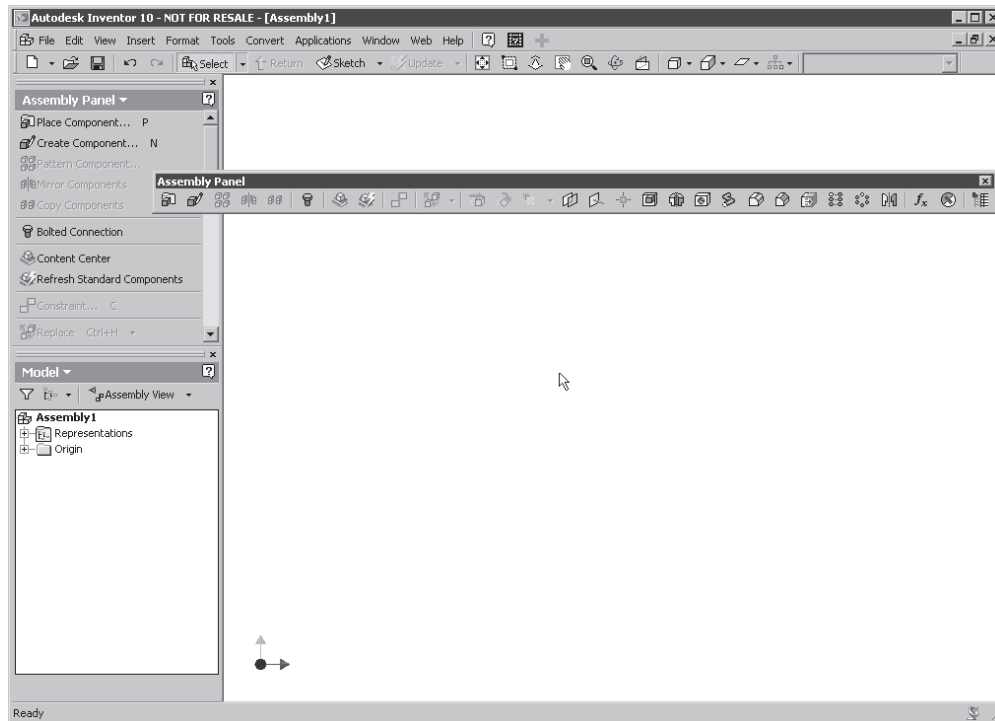


Figure 8-2 Screen display in the **Assembly** module

Top-down Assemblies

A top-down assembly is an assembly whose components are created within the assembly file. In this type of assembly, the components are created in the assembly file and then assembled using the assembly constraints. The process of creating the components in the **Assembly** module of Autodesk Inventor is designed in such a way that the components you create in the **Assembly** module are also saved as individual parts or assembly files. This eliminates the risk of losing the individual components, in case there is an error in the assembly file. Also, the assembly file has the information related to only the assembly, which keeps the size of the assembly file to the minimum.

Bottom-up Assemblies

A bottom-up assembly is an assembly whose all components are created as separate part files and are referenced in the assembly file as external components. In this type of assembly, the components are created in the **Part** module as part files (.ipt). Once all components of the assembly are created, you will open an assembly file (.iam) and then insert all the component files using the tool in the **Assembly** module. After inserting the components, they are assembled using the assembly constraints. Because the assembly file has information related only to the assembling of components, this file size is not large and so it requires less hard disk space. However, remember that if any of the components referenced in the assembly is moved from its original location, it will not show up when you open the assembly next time. This is because Autodesk Inventor will look for the component only in the folder, in which it was originally stored. Because the component is not found at its original location, the **Resolve**

Link dialog box will be displayed, see Figure 8-3. Using this dialog box, you will have to specify the new location of the component.

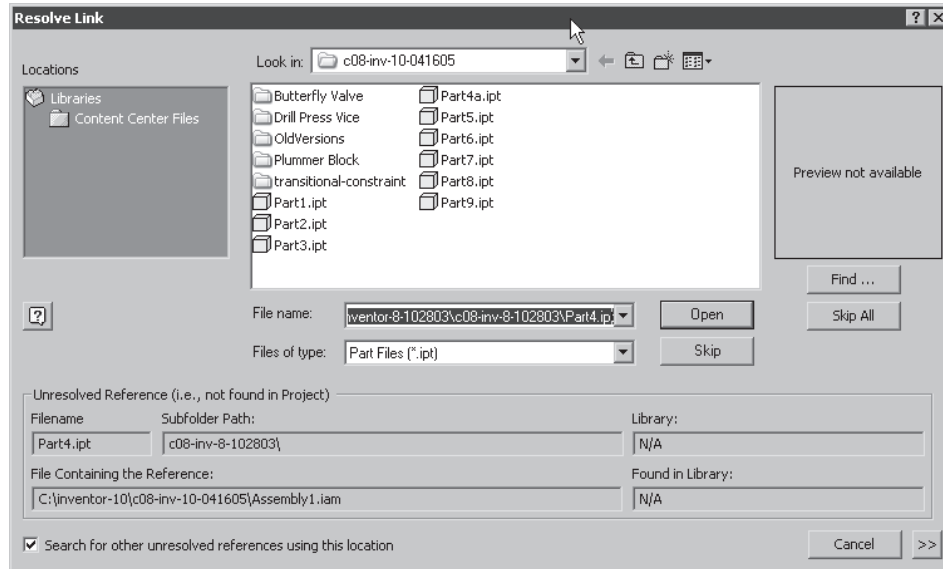


Figure 8-3 The Resolve Link dialog box



Note

An assembly, in which you use a combination of bottom-up and top-down approach, is called the **middle-out assembly**.

CREATING TOP-DOWN ASSEMBLIES

As mentioned earlier, top-down assemblies are those, in which all components are created within the assembly file. To create the components, you require the environment where you can draw the sketches of the sketched features and the environment where you can convert the sketches into features. In other words, to create the components in the assembly file, you require the sketching environment and the part modeling environment. Autodesk Inventor provides you the liberty of invoking both these environments in the **Assembly** module also by using the **Create Component** tool. The use of this tool is discussed next.

Creating Components in the Assembly Module

Toolbar: Assembly Panel > Create Component
Panel bar: Assembly Panel > Create Component



In Autodesk Inventor, you can create components in the **Assembly** module also. One of the advantages of creating the components in the **Assembly** module is that these components can also be saved as a separate part file (*.ipt*) or an assembly file (*.iam*). Therefore, in case you again require any component created in the **Assembly** module, you can use the individual part or assembly file. The components in the **Assembly** module are created using the **Create Component** tool. When you invoke this tool, the **Create In-Place**



Component dialog box will be displayed, see Figure 8-4. The options under this dialog box are discussed next.

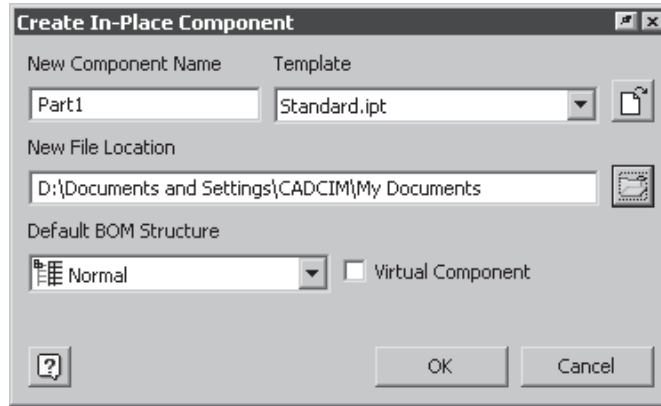


Figure 8-4 The *Create In-Place Component* dialog box

New Component Name

The **New Component Name** text box is used to specify the name of the new component that will be created.

Template

The **Template** drop-down list is used to select the template for the new file. There are four default templates in this drop-down list: **Sheet Metal.ipt**, **Standard.iam**, **Standard.ipt**, and **Weldment.iam**. You can also select the template by choosing the **Browse Templates** button. When you choose this button, the **Open Template** dialog box will be displayed, see Figure 8-5. Using this dialog box, you can select the required template for the new file.



Tip. The assembly template is used to create smaller subassemblies that consist of a few components. These smaller assemblies can be later assembled in a separate assembly file to form the main assembly.

New File Location

The **New File Location** edit box is used to specify the location for saving the new file. You can either specify the location in this edit box or choose the **Browse to New File Location** button provided on the right of the **File Type** drop-down list to specify the location. When you choose this button, the **New Component** dialog box will be displayed, see Figure 8-6. Using this dialog box, you can select the folder, in which you want to save the new file.

Default BOM Structure

This drop-down list is used to specify the type of Bill of Material (BOM) structure for the new component. You will learn more about BOM structure in the next chapter.

Virtual Component

This check box is used to create a virtual component only for the purpose of adding a row in the BOM.

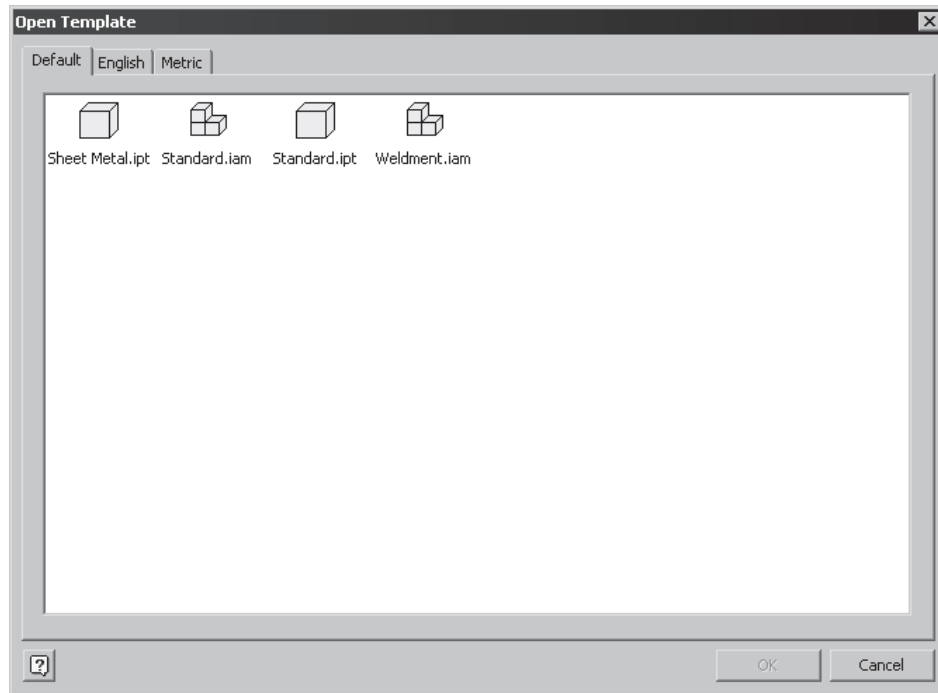


Figure 8-5 The *Open Template* dialog box

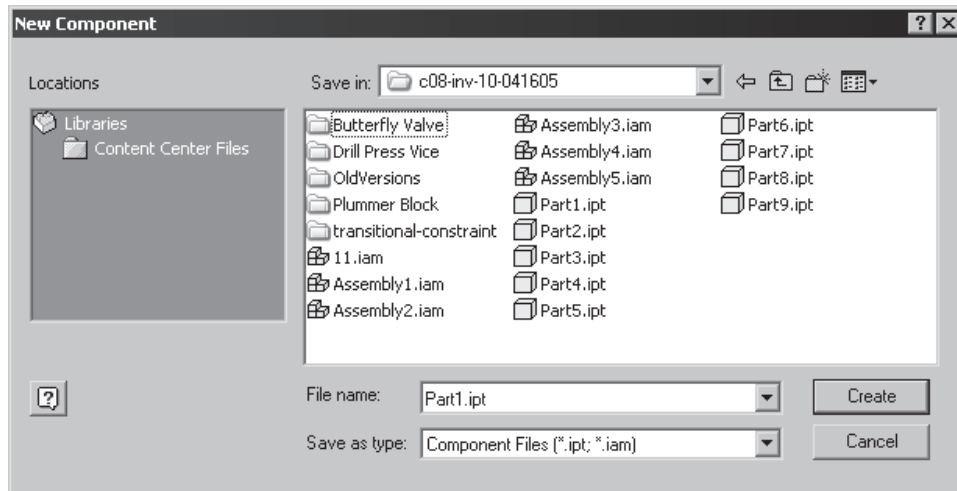


Figure 8-6 The *New Component* dialog box

Constrain sketch plane to selected face or plane

The **Constrain sketch plane to selected face or plane** check box is used to constrain the plane on which the base feature of the model will be created to the selected face of an existing model or work plane. If the component that you create is the first component, it will be grounded and this check box will not be available in the **Create In-Place Component** dialog

box. A grounded component is the first component of the assembly using which the remaining components will be assembled. All the degrees of freedom of the grounded component will be eliminated so that it is not able to move from its original location. Even if you are placing external components, by default, the first component will be grounded.



Tip. In the browser, you can easily distinguish between a grounded and an ungrounded component. A grounded component will have a push pin icon on the left of its name in the browser. You can make a grounded component ungrounded by right-clicking on its name in the browser. When you right-click on a grounded component in the browser, you will notice a check mark in front of the **Grounded** option in the shortcut menu. To unground the component, choose this option again; the push pin icon will be replaced by the original part icon, suggesting that the component is ungrounded.

After setting all the options in the **Create In-Place Component** dialog box, choose **OK**; the sketching environment will be activated and you can draw the sketch for the base feature of the model. After creating the sketch, choose the **Return** button. The part modeling environment will be activated with all the part modeling tools, similar to the **Part** module. Once you have created a part using the sketching environment and the part modeling environment, you can switch back to the **Assembly** module by again choosing **Return** from the **Inventor Standard** toolbar. The **Part Features** panel bar is replaced by the **Assembly Panel** panel bar and all the tools in the **Assembly Panel** panel bar will be available.

Similarly, you can create as many components as you want in the assembly. Once all the components are created, you can start assembling them using the assembly constraints.

CREATING BOTTOM-UP ASSEMBLIES

As mentioned earlier, bottom-up assemblies are those in which all the components are created as separate part files. All the individual part files are then inserted in an assembly file and are assembled using the assembly constraint. The first component inserted in the assembly will be grounded and its origin will coincide with that of the assembly file. Also, the three default planes of the part file will be placed in the same orientation as that of the default planes of the assembly file. The individual components are inserted in the assembly file using the **Place Component** tool. This tool is discussed next.

Placing Components in the Assembly File

Toolbar: Assembly Panel > Place Component
Panel bar: Assembly Panel > Place Component



The **Place Component** tool is used to insert any inventor file in the current assembly file. When you invoke this tool, the **Open** dialog box will be displayed, see Figure 8-7.

This dialog box is similar to the simple **Open** dialog box for opening the files. You can also preview the component before inserting it using this dialog box. The type of file you want to insert can be selected from the **Files of type** drop-down list.

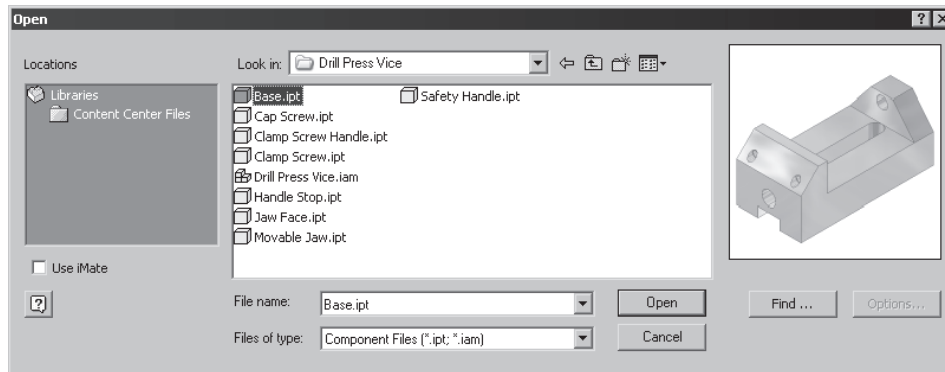


Figure 8-7 The **Open** dialog box

Select the file to be placed from this dialog box and then choose the **Open** button. If it is the first component in the assembly file, one instance of the selected component will be placed automatically in the current file and you will be prompted to place another instance of the selected component. You can place as many copies of the selected component as you want by specifying the point on the screen. Once you have placed the required number of instances of the component, right-click and choose **Done** from the shortcut menu. As mentioned earlier, the first component will be a grounded component. Therefore, the origin of the first component that will be placed automatically will coincide with the origin of the assembly file.

Similarly, you can place the other components using the **Place Component** tool. However, remember that if one or more components are already placed in the current assembly file, no instance of the selected component will be placed automatically. You will have to manually specify the location of the first instance of the component also.

ASSEMBLING COMPONENTS

Toolbar: Assembly Panel > Place Constraint
Panel bar: Assembly Panel > Constraint



In Autodesk Inventor, the components are assembled using four types of assembly constraints, two types of motion constraints, and a transitional constraint. All these constraints are applied using the various tabs of **Place Constraint** dialog box. This dialog box is displayed when you invoke the **Constraint** tool. All the seven types of constraints that can be applied using the **Place Constraint** dialog box are discussed next.

Mate Constraint



The **Mate** constraint is applied using the first button provided under the **Type** area of the **Assembly** tab, see Figure 8-8. This constraint is used to make the selected planar face, axis, or point of a component coincident with that of another component. Depending on the solution selected from the **Solution** area, the components will be assembled with the normal of the faces pointing in the same direction or in the opposite direction. The options in the **Assembly** tab of the **Place Constraint** dialog box when you choose the **Mate** button are discussed next.

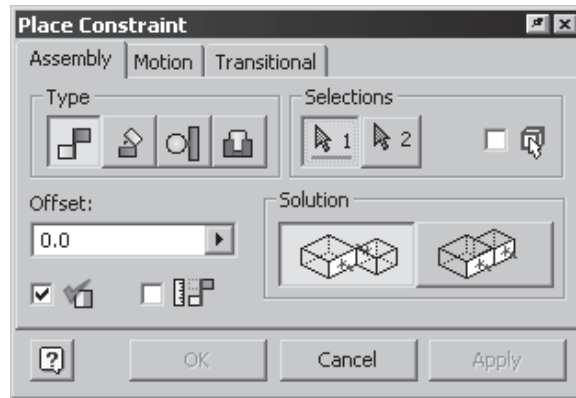


Figure 8-8 The **Mate** constraint options in the **Assembly** tab of the **Place Constraint** dialog box

Selections Area

The options under this area are used to select the faces, axes, edges, or points of the selected model for applying the **Mate** constraint. These options are discussed next.



Tip. You can press and hold the **F4** key to rotate the view of the model for the selection purpose. You can also rotate the view of an individual component, which is discussed later in this chapter.

1 (First Selection)



This button is automatically chosen when you invoke the **Mate** constraint and is used to select a face, axis, edge, or point on the first component to apply the **Mate** constraint. Move the cursor close to the component you want to select. If the cursor is close to a face, it will be highlighted and an arrow will be displayed along with a cross. This arrow will point in the direction of the normal of the selected face. The components are assembled in the direction of the normal of the faces. Similarly, if you move the cursor close to an edge, axis, or a point, it will be highlighted.

2 (Second Selection)



This button is automatically chosen after you select the first component and is used to select a face, axis, edge, or point on the second component to apply the **Mate** constraint. Figure 8-9 shows the **Mate** constraint being applied on the faces of two components. As evident from Figure 8-9, an arrow is displayed on the selected face of both the components. These arrows point in the direction of the normal of the selected face. The selected components will be assembled in the direction of these faces.

Pick part first

The **Pick part first** check box is provided on the right side in the **Selections** area. This check box is used for the assembly that has a large number of components and it is hard to select the axis, edge, face, or point of one of the components due to the complicity. If this check box is selected, you will first have to select the component and then select the element in that component to apply the constraint.

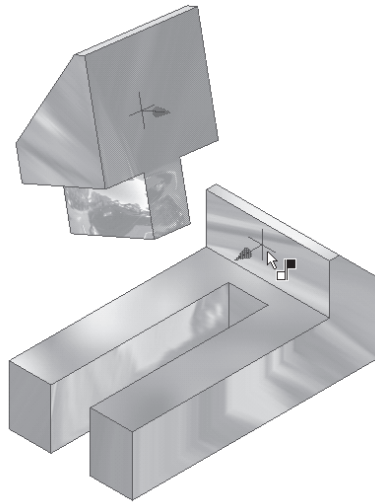


Figure 8-9 Applying the **Mate** constraint on faces



Note

You can preview the assembling of the components on the screen after you have selected both the components to apply the constraint. However, remember that until you choose the **Apply** button in the **Place Constraint** dialog box, the constraint will not be actually applied.

Offset

The **Offset** edit box is used to specify the offset distance between the mating components. If the offset distance is zero, the mating entities will be in contact with each other. However, if there is an offset distance between the mating components, they will be placed at a distance from each other. Figure 8-10 shows the components assembled with an offset distance of 0 mm and Figure 8-11 shows the components assembled with an offset distance of 10 mm between the faces.

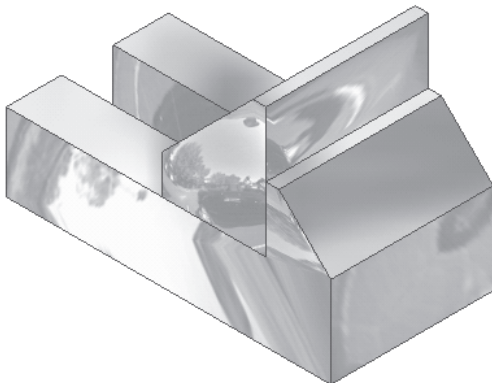


Figure 8-10 Components assembled with an offset of 0 mm

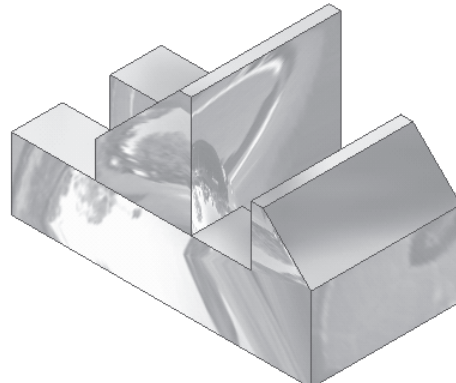


Figure 8-11 Components assembled with an offset of 10 mm



Tip. Generally, the components are not assembled with a single constraint. Depending on the components, you require two to three constraints. It can be these many number of the same constraint or a combination of constraints.

Show Preview

The **Show Preview** check box is selected to display the preview of the assembling components. When you select two components to apply the constraint, a preview of the assembly will be displayed even if you have not chosen the **Apply** button. This is because the **Show Preview** check box is selected. If this check box is cleared, the preview of the assembly will not be displayed.

Predict Offset and Orientation

The **Predict Offset and Orientation** check box is selected to allow Autodesk Inventor to predict the offset and the orientation of the selected components. The predicted offset value is automatically specified in the **Offset** edit box.

Solution Area

The buttons provided under the **Solution** area are used to specify whether the components being assembled should be placed in a mating position or in a flushing position. A mating position is the one in which the normal of the faces are facing in the opposite direction, see Figure 8-12. A flushing position is one, in which the normal of the faces are facing in the same direction, see Figure 8-13.

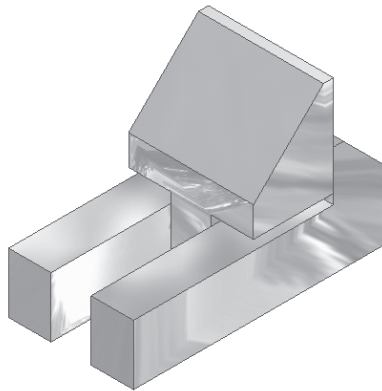


Figure 8-12 Mating position

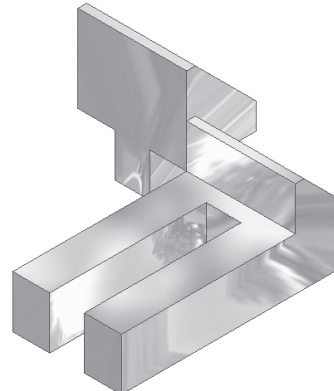


Figure 8-13 Flushing position

Angle Constraint



The **Angle** constraint is applied by using the second button in the **Type** area of the **Assembly** tab. This constraint is used to specify the angular position of the selected planar faces or edges of two components. The options that will be displayed when you choose the **Angle** button are displayed in Figure 8-14. Some of the options under this constraint are the same as those under the **Mate** constraint. The remaining options are discussed next.

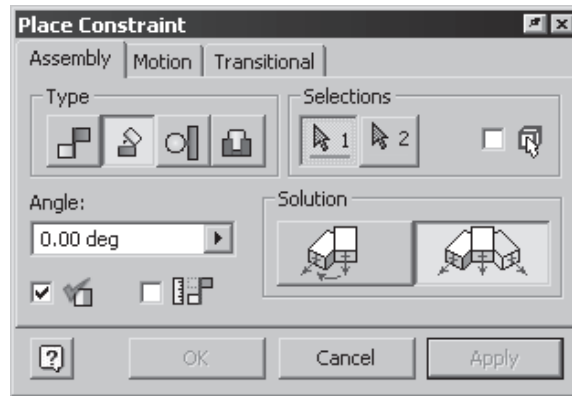


Figure 8-14 The **Angle** constraint options in the **Assembly** tab of the **Place Constraint** dialog box

Angle

This edit box is used to specify the angle between the selected planar faces or edges of two components. The components will be separated by an angle value specified in this edit box. You can specify a positive or a negative value in this edit box.

Solution Area

This area provides two buttons. The first button is the **Directed Angle** button and is used to apply the **Angle** constraint based on the right hand thumb rule. The second button is the **Undirected Angle** button and is used to apply the constraint based on the default orientation of the components.

Figure 8-15 shows the components selected to apply the **Angle** constraint and Figure 8-16 shows the components after applying the **Angle** constraint of 90-degree.

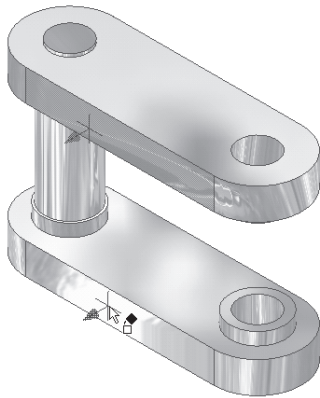


Figure 8-15 Selecting the faces to apply the angle constraint

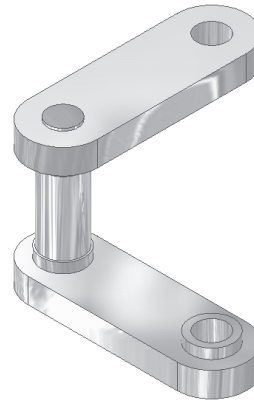


Figure 8-16 Components after applying the **Angle** constraint of 90-degree



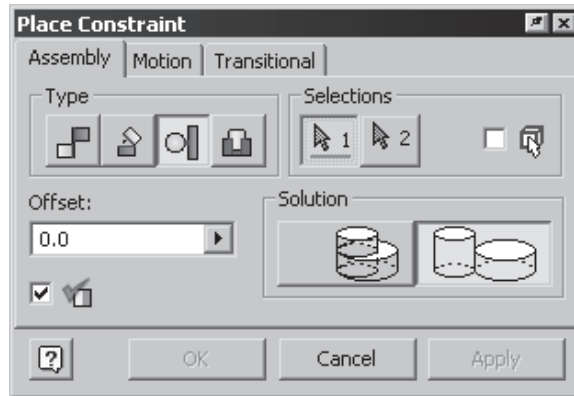
Note

The remaining options under the **Angle** constraint are the same as those discussed under the **Mate** constraint.

Tangent Constraint



The **Tangent** constraint is applied by choosing the third button in the **Type** area. This constraint forces the selected circular face of the component to become tangent to the circular face of the other component. The options that are displayed when you choose the **Tangent** button are displayed in Figure 8-17. Some of the options are similar to those discussed earlier. The remaining options are discussed next.

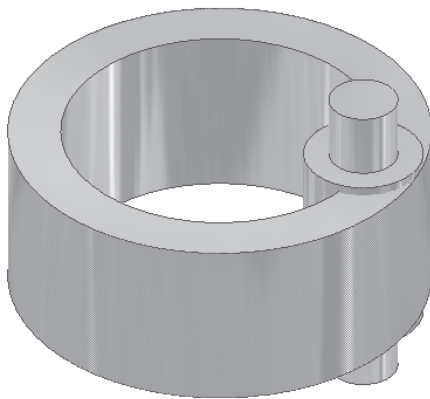


*Figure 8-17 The **Tangent** constraint options in the **Assembly** tab of the **Place Constraint** dialog box*

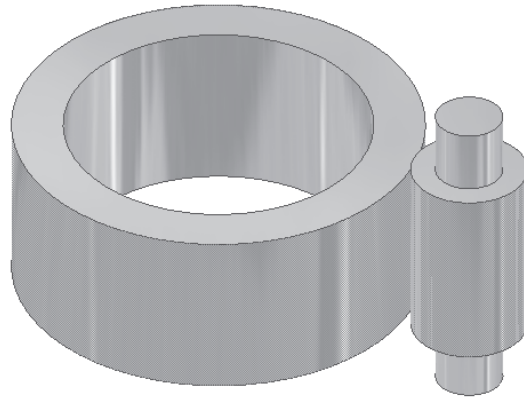
Solution Area

The **Solution** area provides the **Inside** and **Outside** buttons for the **Tangent** constraint. Choosing the **Inside** button applies the **Tangent** constraint between the outside of the first selected face and inside of the second selected face. In doing so, the first component is moved inside the second component. Choosing the **Outside** button applies this constraint between the outside of both the selected faces.

Figure 8-18 shows the **Tangent** constraint with **Inside** solution and Figure 8-19 shows the **Tangent** constraint with **Outside** solution.



*Figure 8-18 The **Inside** solution*



*Figure 8-19 The **Outside** solution*

Insert Constraint



The **Insert** constraint is applied using the fourth button in the **Type** area of the **Assembly** tab of the **Place Constraint** dialog box. This constraint is used to force two different cylindrical or conical components or features of components to share the same location and orientation of the central axis. This constraint also makes the selected face of the first component coplanar with the selected face of the other component. The options that will be displayed in the **Assembly** tab are shown in Figure 8-20. These options are discussed next.

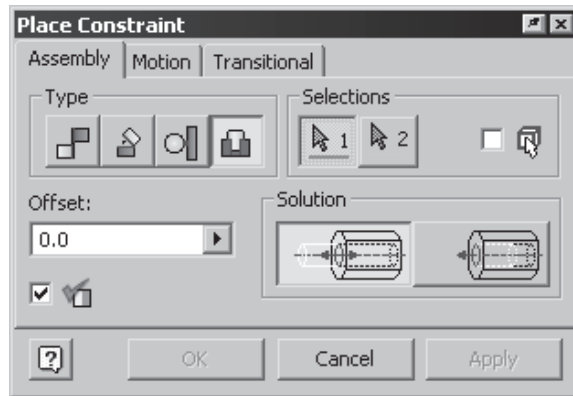


Figure 8-20 The **Insert** constraint options in the **Assembly** tab of the **Place Constraint** dialog box

Solution Area

The options provided under the **Solution** area are used to specify whether the normal of the mating faces will point in the same direction or in the opposite direction. If you choose the **Opposed** button, the normal of the mating faces will point in the opposite direction. If you choose the **Aligned** button, the mating faces will point in the same direction. However, the central axes of both the components will share the same orientation. Figure 8-21 shows the components being selected to apply the **Insert** constraint and Figure 8-22 shows the components after applying the **Insert** constraint.

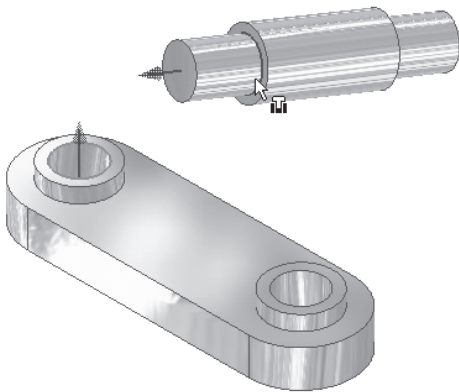


Figure 8-21 Selecting the components

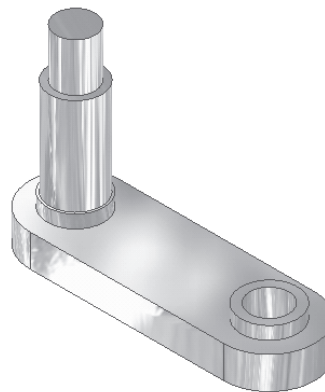


Figure 8-22 Components after applying the **Insert** constraint

**Note**

The remaining options under the **Tangent** and **Insert** constraints are the same as those discussed under the **Mate** constraint.

Rotation Constraint



The **Rotation** constraint is applied by choosing the first button in the **Type** area of the **Motion** tab. This constraint is used to rotate one of the components in relation with the other component. The components rotate about the specified central axis. The options that are displayed when you choose the **Rotation** button from the **Type** area of the **Motion** tab of the **Place Constraint** dialog box are shown in Figure 8-23. These options are discussed next.

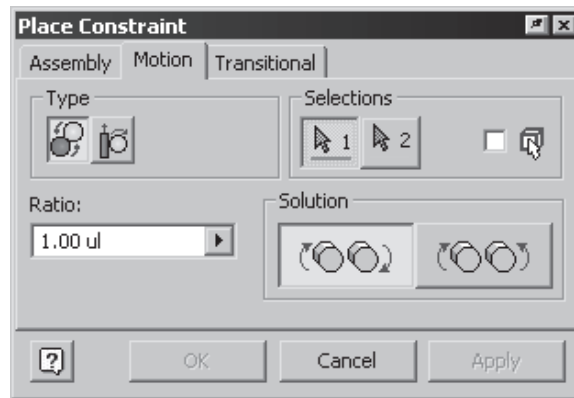


Figure 8-23 The **Rotation** constraint options in the **Motion** tab of the **Place Constraint** dialog box

Ratio

The **Ratio** edit box is used to specify the ratio by which the second component will rotate with respect to one complete rotation of the first component. For example, if you enter a value of 2 in this edit box, the second component will rotate two times if the first component is rotated once. Similarly, if you enter a value of 10, the second component will rotate ten times if the first component is rotated once.

Solution

The options provided under the **Solution** area are used to specify the direction of rotation of the components. Choose the **Forward** button to rotate the components in the forward direction or choose the **Reverse** button to rotate the components in the reverse direction.

**Note**

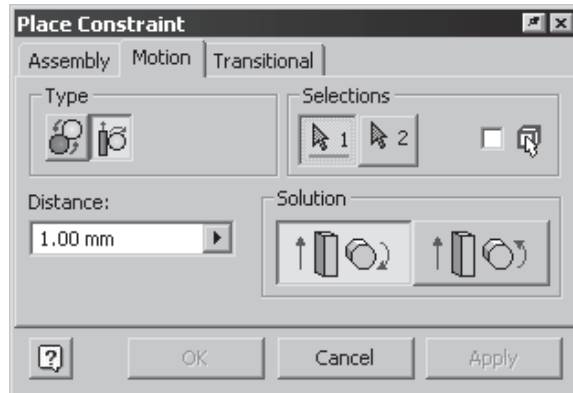
The constraints in the **Motion** tab of the **Place Constraint** dialog box work only with the degree of freedom that is not restricted. These constraints do not interfere with the other assembly constraints.

The remaining options under the **Rotation** constraint are the same as those discussed under the **Mate** constraint.

Rotation-Translation Constraint



The **Rotation-Translation** constraint is applied by choosing the second button in the **Type** area of the **Motion** tab. This constraint is used to rotate the first component in relation with the translation of the second component. The options in the **Motion** tab of the **Place Constraint** dialog box are shown in Figure 8-24. These options are discussed next.



*Figure 8-24 The **Rotation-Translation** constraint options in the **Motion** tab of the **Place Constraint** dialog box*

Distance

The **Distance** edit box is used to specify the distance by which the second component will move in relation with one complete rotation of the first component. For example, if you enter a value of 2 mm in this edit box, the second component will move a distance of 2 mm for one complete rotation of the first component.

Solution

The buttons in the **Solution** area are used to specify whether the second component will move in the forward direction or the reverse direction for every forward rotation of the first component. Choose the **Forward** button to move the component in the forward direction and the **Reverse** button to move the component in the reverse direction.

Transitional Constraint

The **Transitional Constraint** is applied by choosing its button from the **Type** area of the **Transitional** tab, see Figure 8-25. A transitional constraint ensures that the selected face of the cylindrical component maintains contact with the other selected face when you slide the cylindrical component about the degree of freedom that is not eliminated. The options under this tab are similar to those discussed in the previous constraints.

USING ALT+ DRAG TO APPLY ASSEMBLY CONSTRAINTS

Autodesk Inventor allows you to apply the assembly constraints without actually invoking the **Place Constraint** dialog box. This is done by pressing the ALT key and then dragging the

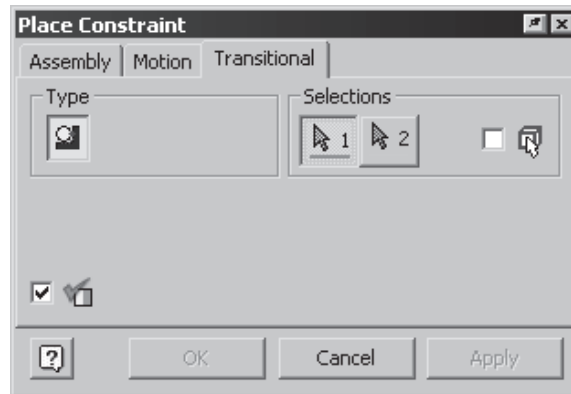


Figure 8-25 The *Transitional* constraint options in the *Transitional* tab of the *Place Constraint* dialog box

component. The following steps explain the procedure to apply assembly constraints using the ALT+Drag method.

1. Press the ATL key and start dragging the component that you want to select as the first component; the symbol of the **Mate** constraint is displayed below the cursor. This is because when you use the ALT+Drag method, by default the **Mate** constraint is applied.
2. Release the ALT key but make sure you do not release the left mouse button. This is because if you release the left mouse button, the constraints will no more be applied. Press the SPACEBAR to change the mate position to the flush position. Drag the selected component to the component that you want to select as the second component and then release the left mouse button. The assembly constraint is applied.

If you do not want to apply the **Mate** constraint, you can use the keyboard keys, after releasing the ALT key, to apply the other relations. The following table provides the details of the various keys that you can use to apply the assembly constraints:

Key	Description	Function of SPACEBAR
M or 1	Applies the Mate constraint	Changes the mate position to the flush position
A or 2	Applies the Angle constraint	Reverses the direction of applying the Angle constraint
T or 3	Applies the Tangent constraint	Reverses to inside or outside face for applying the Tangent constraint
I or 4	Applies the Insert constraint	Reverses the direction of the insertion
R or 5	Applies the Rotation constraint	Reverses the direction of rotation
S or 6	Applies the Rotation-Translation constraint	Reverses the direction of translation
X or 8	Applies the Transitional constraint	No use of SPACEBAR

MOVING INDIVIDUAL COMPONENTS

Toolbar: Assembly Panel > Move Component
Panel bar: Assembly Panel > Move Component



Autodesk Inventor allows you to move the individual unconstrained components in the assembly file without disturbing the position and location of the other components in the assembly file. This is done using the **Move Component** tool. When you invoke this tool, you will be prompted to drag the component to a new location. As you move the cursor close to any component, it will be highlighted. Select the component and then drag it to the desired location. The component will be relocated and the other components in the assembly file will not be disturbed.

ROTATING INDIVIDUAL COMPONENTS IN 3D SPACE

Toolbar: Assembly Panel > Rotate Component
Panel bar: Assembly Panel > Rotate Component



You can also rotate individual unconstrained components in the current assembly file without changing the orientation of the other components. This is done using the **Rotate Component** tool. When you invoke this tool, you will be prompted to drag the component to a new location. Select the component that you want to reorient. As soon as you select it, the rim along with the handles will be displayed around the model. Also, the cursor will be changed to rotation mode cursor.

You can use the same tool to rotate other individual components also. After you have finished rotating a component, click on the other component using the rotation mode cursor. You will notice that the rim is now displayed around the component that you selected. Similarly, you can select any individual component to rotate in 3D space.

TUTORIALS

Tutorial 1

In this tutorial, you will create the components of the Butterfly Valve and then assemble them. The Body and the Shaft will be created in the assembly file and the remaining components will be created as individual parts in separate part files. As a result, you will use the combination of top-down and bottom-up assemblies. The dimensions for the components are given in Figures 8-26 through 8-32. Assume the missing dimensions for the components and the parameters for the threads.
(Expected time: 3 Hrs 30 min)

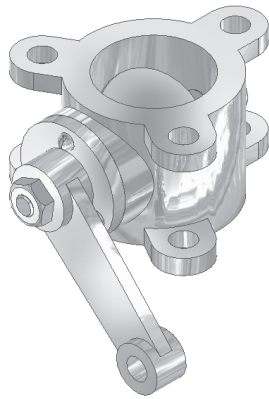


Figure 8-26 Butterfly Valve assembly

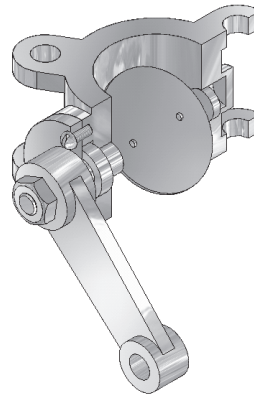


Figure 8-27 Inside view of the Butterfly Valve

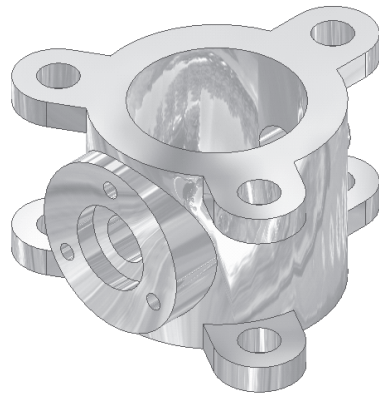


Figure 8-28a Solid model of the Body

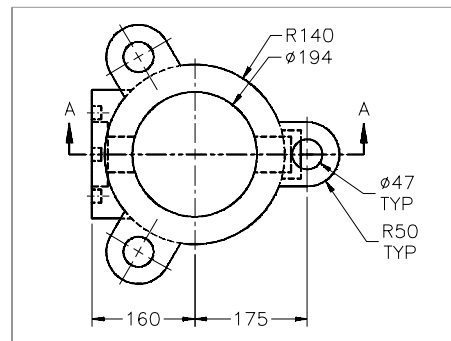


Figure 8-28b Top view of the Body

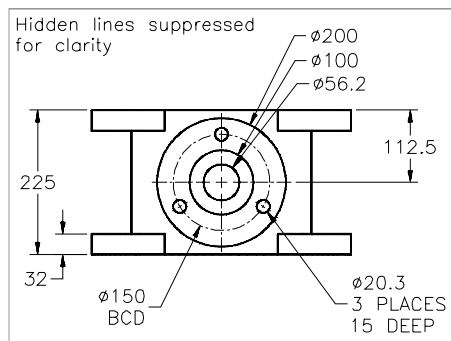


Figure 8-28c Left-side view of the Body

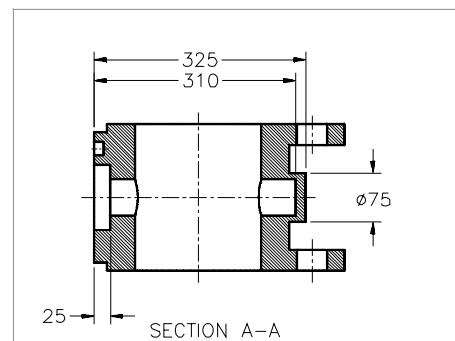


Figure 8-28d Sectioned front view of the Body

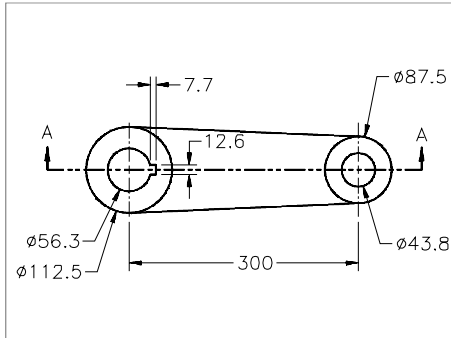


Figure 8-29a Top view of the Arm

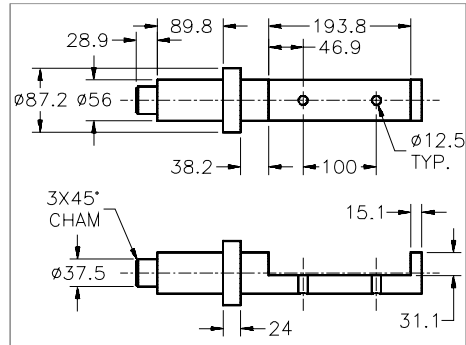


Figure 8-30 Dimensions of the Shaft

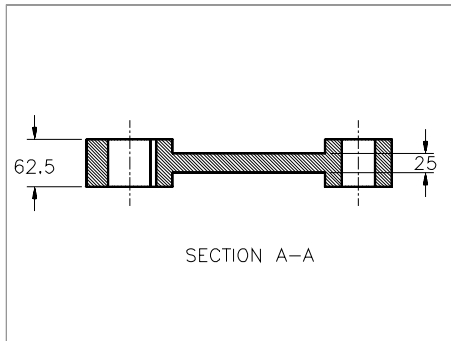


Figure 8-29b Sectioned front view of the Arm

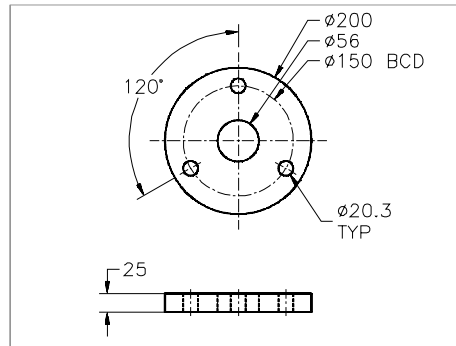


Figure 8-31 Dimensions of the Retainer

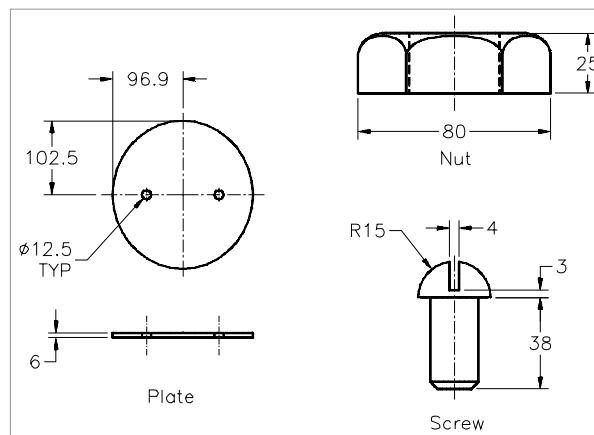


Figure 8-32 Dimensions of the Plate, Nut, and Screw

The following steps are required to create the assembly:

- Create the Body and the Shaft in the assembly file and then assemble these two components using the **Place Constraint** dialog box. Save and close the assembly file.
- Start new metric standard part files and, one by one, create the other individual components.
- Open the assembly file and insert the individual components in the assembly file using the **Place Component** tool.
- Assemble the components using the **Place Constraint** dialog box to complete the Butterfly Valve assembly.

Creating the Body

The Body and the Shaft will be created in the assembly file and therefore, you will use the top-down approach of assembly modeling. To create these two components, you first need to start a new metric assembly file.

- Choose the **New** button from the **Inventor Standard** toolbar to invoke the **Open** dialog box. Double-click on **Standard (mm).iam** in the **Metric** tab (Figure 8-33) to start a metric assembly file. If you have started a new Autodesk Inventor session, this dialog box is automatically displayed.

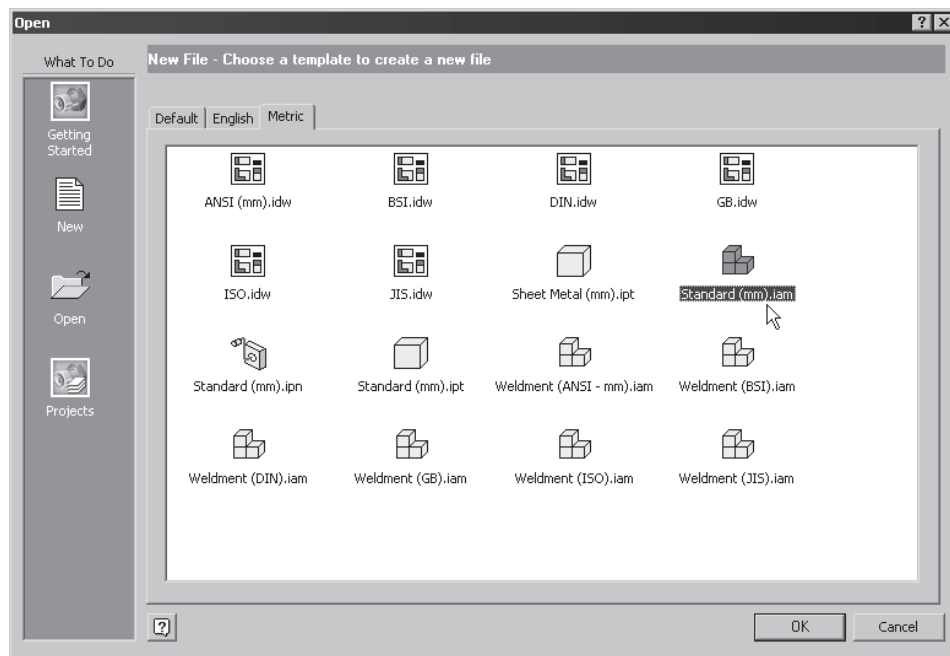


Figure 8-33 Starting a new metric assembly file using the **Open** dialog box

The assembly modeling environment is activated with the **Assembly Panel** panel bar on the left of the drawing window, as shown in Figure 8-34.

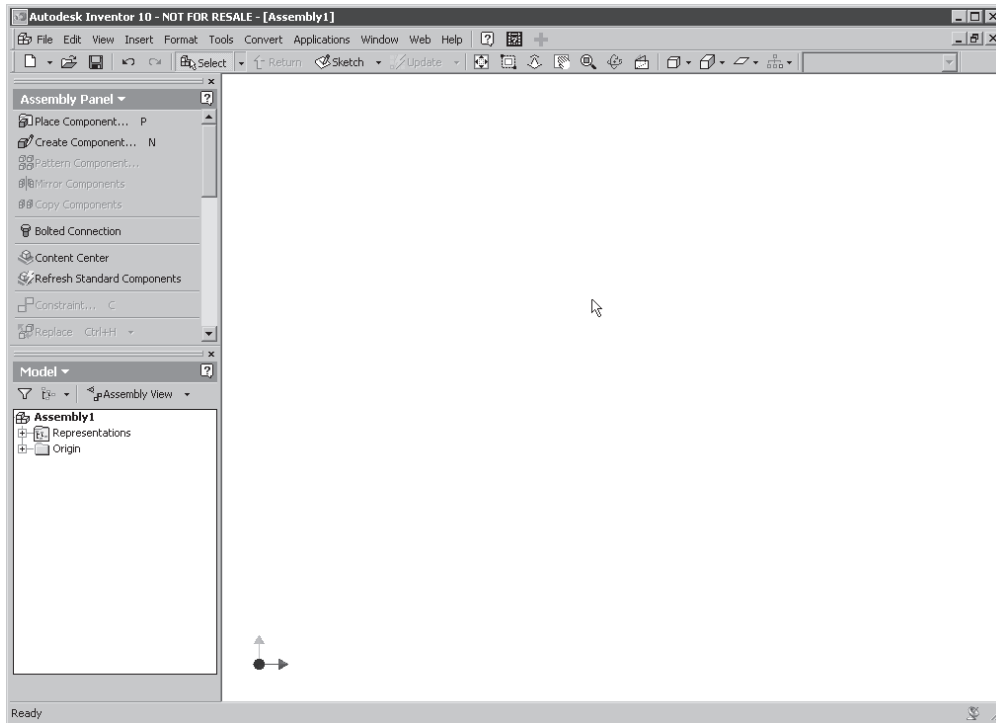



Figure 8-34 Screen display in the *Assembly* module

You will notice that very few tools are available in the **Assembly Panel** panel bar. This is because no component is present in the assembly file. Once a component is placed or created, all the other tools will be available for use.

2. Choose the **Create Component** button from the **Assembly Panel** toolbar or choose the **Create Component** button from the **Assembly Panel** panel bar to invoke the **Create In-Place Component** dialog box. 
3. Enter the name of the new part file as **Body.ipt** in the **New Component Name** edit box.
4. Select **Standard.ipt** from the **Template** drop-down list, if not already selected. Choose the **OK** button.
5. Specify the location of the new part file in the **New File Location** edit box as `\\PersonalProject\\c08\\Butterfly Valve`.

The **Autodesk Inventor 10** information box is displayed informing you that the folder you have mentioned does not exist. It will also prompt you if you want to create this folder. Because you did not create any folder for storing the component files before starting the assembly, you need to create it. Therefore, if you choose **OK** in the **Autodesk Inventor 10** information box, it will create the folder for you.

6. Choose **OK** in the **Autodesk Inventor 10** information box.



Tip. It is recommended that you create separate folders for storing the individual component files of assemblies. The reason for this is that a lot of assemblies have components with similar names. For example, the name *Body* is common to a number of assemblies. Therefore, if you create the *Body* and store it in the folder of a particular assembly, there will be no confusion in placing the components. Also, when you open the assembly next time, there will be no confusion in referring to the component.

As soon as you choose **OK** from the **Autodesk Inventor 10** information box, it is closed and you are prompted to select the sketching plane for the base feature. You can now create the *Body* of the *Butterfly Valve* assembly.

**Note**

Remember that if you save the file when the part modeling environment is active, the part file will be saved not the assembly file. This means that while creating the *Body*, if you choose the **Save** button from the **Inventor Standard** toolbar, the *Body.ipt* file will be saved not the current assembly file. To save the current assembly file, you need to exit the part modeling environment and then choose the **Save** button in the assembly modeling environment. You can also save the assembly file before you start creating the components.

7. Create the *Body* of the *Butterfly Valve* using the given dimensions. The screen display of the assembly file after creating the *Body* is shown in Figure 8-35.

You will notice that the part modeling environment is still active in the assembly file. To proceed further, you need to save the part file and exit the part modeling environment by choosing the **Return** button from the **Inventor Standard** toolbar.

8. Choose the **Save** button to save the part file and then choose the **Return** button from the **Inventor Standard** toolbar to exit the part modeling environment.

When you choose the **Return** button, you will notice that the assembly modeling environment is activated and the **Part Features** panel bar of the part modeling environment is replaced by the **Assembly Panel** panel bar of the assembly modeling environment.

As mentioned earlier, until you exit the part modeling environment, only the part file will be saved when you choose the **Save** button. The assembly file will be saved only after you exit the part modeling environment.

9. Choose the **Save** button from the **Inventor Standard** toolbar and save the assembly with the name *Butterfly Valve.iam* in the *Butterfly Valve* folder.

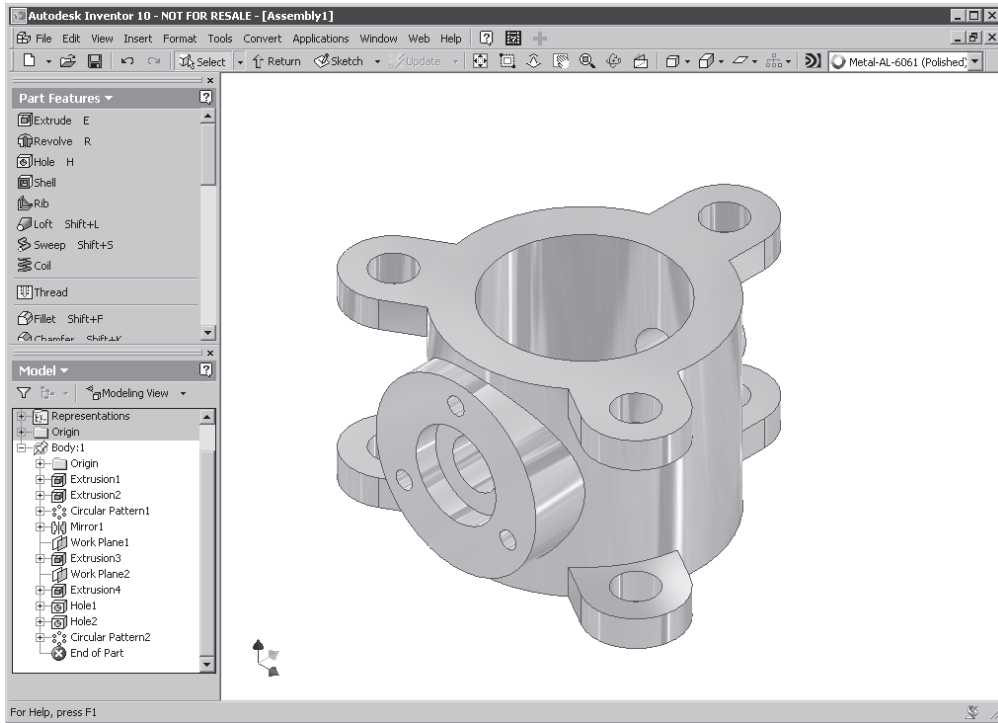



Figure 8-35 Assembly file in the part modeling environment after creating the Body

Creating the Shaft

The second component that has to be created in the assembly file is the Shaft. Therefore, you need to again activate the sketching environment and the part modeling environment to create the Shaft. However, because the Body is already present in the assembly file, there are chances that it might restrict with the view of the part that you create next. Considering this, the part modeling environment in the assembly file is designed in such a way that when you start creating the components in the assembly file, all the existing components become transparent so that they do not restrict the view of the new part that you create.

1. Choose the **Create Component** button from the **Assembly Panel** toolbar or choose **Create Component** from the **Assembly Panel** panel bar to invoke the **Create In-Place Component** dialog box. 
2. Enter the name of the new part file as **Shaft.ipt** in the **New Component Name** edit box.
3. Specify the location of the new part file in the **New File Location** edit box as `\\PersonalProject\\c08\\Butterfly Valve`.
4. Clear the **Constrain sketch plane to selected face or plane** check box. Choose **OK**.

5. You are prompted to select the plane on which you want to sketch the base feature. Select the XY plane from the browser.

As soon as you select the XY plane, the sketching environment is activated and the Body becomes transparent. You can now proceed with creating the Shaft.

6. After creating the Shaft, save it before exiting the part modeling environment so that the *Shaft.ipt* file is saved. Exit the part modeling environment by choosing the **Return** button from the **Inventor Standard** toolbar. Save the assembly file by choosing the **Save** button from the **Inventor Standard** toolbar.

When you exit the part modeling environment, you will notice that the Body is no more transparent. Also, both the components in the assembly file interfere with each other. Therefore, before proceeding with assembling these components, you need to move one of the components such that it does not interfere with the other. You can move the individual component using the **Move Component** tool.

7. Choose **Move Component** from the **Assembly Panel** panel bar. You are prompted to drag the component to a new location. Select the Body and drag it to a new location where it does not interfere with the Shaft. Choose the **Zoom All** button to increase the display area. The screen display of the assembly file with both the components is shown in Figure 8-36.

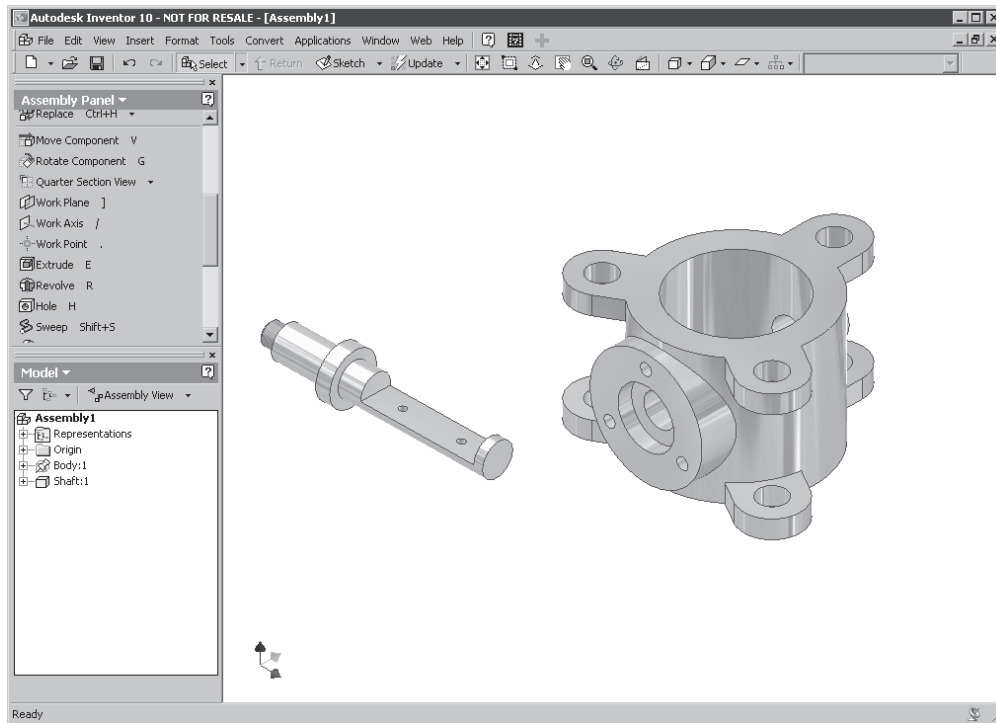


Figure 8-36 The assembly file after creating the Body and the Shaft

**Note**

If the orientation of the Shaft and the Body is different from the one shown in Figure 8-36, you can reorient them using the **Rotate Component** tool.

Assembling the Components

The Shaft is a circular part that has to be inserted in the counterbore hole of the Body. Therefore, you can use the **Insert** constraint to assemble these components. As mentioned earlier, the **Insert** constraint forces the selected components or features to share the same location and orientation of the central axis and at the same time makes the selected faces coplanar. Therefore, the Shaft will be assembled with the Body using the **Insert** constraint. Now, the flat part of the Shaft has to be at an angle to the top face of the Body. To assemble it at an angle with the top face, you will have to use the **Angle** constraint. Therefore, the second constraint that will be used is the **Angle** constraint.

1. Choose the **Place Constraint** button from the **Assembly Panel** toolbar or choose **Constraint** from the **Assembly Panel** panel bar to invoke the **Place Constraint** dialog box.



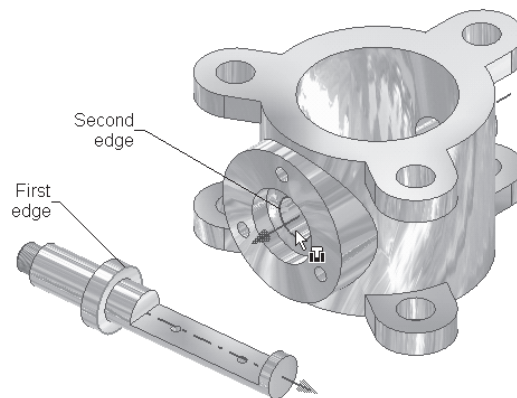
By default, the **Mate** constraint is selected. For assembling the Shaft with the Body, you require the **Insert** constraint. Therefore, you will have to choose the **Insert** button from the **Type** area of the **Assembly** tab of the **Place Constraint** dialog box.

2. Choose the **Insert** button from the **Type** area. You will notice that the **Insert** constraint symbol is attached to the cursor. This symbol is displayed along with the cursor when you move the cursor in the drawing window.



3. Select the edge of the Shaft, as shown in Figure 8-37.

You will notice that the selected edge is highlighted and an arrow is displayed along the direction of the central axis of the Shaft. This arrow will also point in the direction in which the Shaft will be assembled. Also, the **2** button in the **Selections** area of the **Place Constraint** dialog box is automatically chosen.



4. Select the inner edge of the counterbore hole, as shown in Figure 8-37. As soon you select the second edge for applying the constraint, the Shaft will assemble with the Body. This is because by default, the **Show Preview** check box in the **Place Constraint** dialog box is selected.

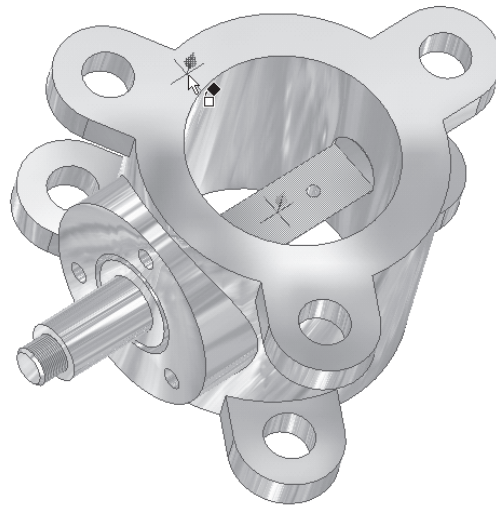
Figure 8-37 Selecting the faces to apply the **Insert** constraint

5. Choose the **Apply** button to assemble the Shaft with the Body.
6. Choose the **Angle** button to apply the **Angle** constraint.



As the assembly is in the isometric view, the flat face of the Shaft is not visible. Therefore, you will have to reorient the assembly using the **Rotate** tool. This tool can be used when you are inside another tool and so when you invoke this tool, you will temporarily switch to the assembly rotating mode. The assembling of components will resume when you exit the **Rotate** tool.

7. Choose the **Rotate** button from the **Inventor Standard** toolbar and rotate the assembly such that the flat face of the Shaft is visible, refer to Figure 8-38. Right-click to display the shortcut menu and choose **Done** to exit this tool.
8. The symbol of the **Angle** constraint will be reattached to the cursor, suggesting that the assembling of components is resumed. Select the flat face of the Shaft as the first face to apply the **Angle** constraint, see Figure 8-38.
9. Select the top face of the Body as the second face to apply the **Angle** constraint, see Figure 8-38.



*Figure 8-38 Selecting faces to apply the **Angle** constraint*

10. Enter the value of angle as **45** in the **Angle** edit box. The flat face rotates toward the right side. If the flat face rotates toward the left, enter **-45** in the **Angle** edit box and choose the **Directed Angle** button from the **Solution** area to make sure the face rotates toward the right. Choose **Apply** to apply the constraint. Exit the dialog box by choosing the **Cancel** button. Change the view back to the previous view by pressing the F5 key.

Creating other Components

1. Save the current assembly file and then close it by choosing **File > Close** from the menu bar.
2. Create the other components as individual part files and save them with their names in the *Butterfly Valve* folder.

3. Exit the part file and then again open the *Butterfly Valve.iam* file.

Assembling the Retainer

The next component that has to be assembled is the Retainer. The Retainer is also a circular part and so can be assembled using the **Insert** constraint. The three holes of the Retainer have to match those on the front planar face of the Body. Also, the central hole of the Retainer has to match with the central hole of the front planar face of the Body. Therefore you need to apply the **Insert** constraint twice, the first time to align one of the smaller holes on the Retainer with one of the smaller holes on the left flat face of the Body. The second time it will be used for the central hole. But first, you need to place the Retainer in the assembly using the **Place Component** tool.

1. Choose **Place Component** from the **Assembly Panel** panel bar to invoke the **Open** dialog box.



By default, the current folder in the **Open** dialog box will be the **Butterfly Valve** folder. All the components that you created are displayed in this folder.

2. Double-click on the Retainer. You can also select the Retainer and then choose the **Open** button.

The **Open** dialog box is closed and the Retainer is attached to the cursor. Also, you are prompted to place the component.

3. Place the Retainer at a location where it does not interfere with the existing components.

After you have placed an instance of the Retainer, you are again prompted to place the component. Because you need to place only one instance of the Retainer, you can exit the component placement option.

4. Right-click in the drawing window to display the shortcut menu and choose **Done** to exit the component placement option.

5. Choose **Constraint** from the **Assembly Panel** panel bar; the **Place Constraint** dialog box is displayed. If the **Place Constraint** dialog box is restricting the viewing of the components in the drawing window, you can move it by picking it from the blue strip on top and dragging.



6. Choose the **Insert** button from the **Type** area. Select the circular edge of one of the smaller holes on the top face of the Retainer as the first edge, see Figure 8-39.



7. Select the circular edge of one of the smaller holes on the front planar face of the circular feature on the Body to apply the constraint, see Figure 8-39. Choose **Apply**.

As soon as you select the second face, the Retainer moves from its location and is assembled with the Body such that both the selected holes are concentric and the top face of the Retainer is coplanar with the front planar face of the circular feature on the Body. However,

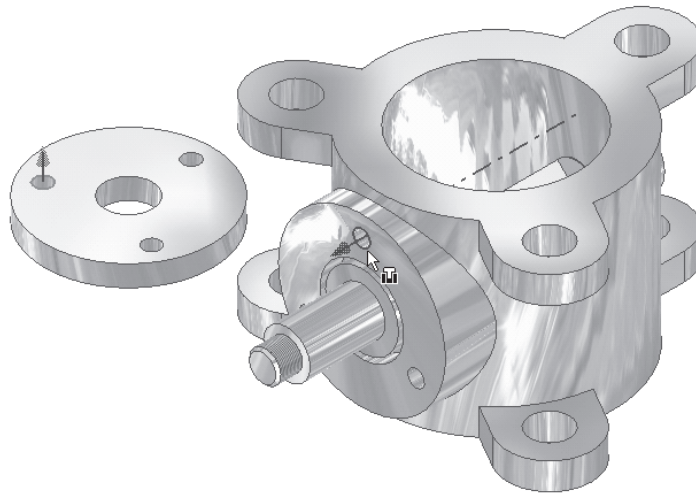


Figure 8-39 Selecting the faces to apply the **Insert** constraint

you will notice that the central hole of the Retainer is not concentric with the central hole of the left circular feature of the Body and the Shaft. Therefore, you need to apply the **Insert** constraint once again to align them.

8. Select the inner edge of the Retainer that is coplanar with the Body as the first edge to apply the constraint, see Figure 8-40. You may have to rotate the model to select this edge.
9. Select the outer circular edge on the flat face of the front circular feature on the Body as the second face to apply the constraint, see Figure 8-40. Choose **Apply** to assemble the components and then choose **Cancel** and exit the dialog box.

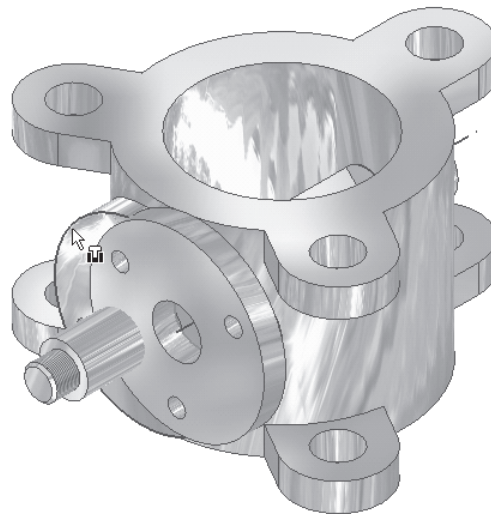





Figure 8-40 Selecting the faces to apply the **Insert** constraint again

Assembling the Arm

The next component that has to be assembled is the Arm. Because the Arm is assembled at a certain angle with respect to the Body, you need to use two constraints to assemble it. The first constraint will be the **Insert** constraint, which will insert the already assembled Retainer in the bigger hole of the Arm. The second constraint will be the **Angle** constraint, which will be used to apply an angle between the XZ plane of the Arm and the top face of the Body. You will place the Arm using the **Place Component** tool.

1. Choose the **Place Component** button from the **Assembly Panel** panel bar to invoke the **Open** dialog box. 
2. Double-click on the Arm.
3. Place the Arm at a location where it does not interfere with the existing components.
4. Right-click in the drawing window to display the shortcut menu and choose **Done** to exit the component placement option.
5. Choose the **Constraint** button from the **Assembly Panel** panel bar; the **Place Constraint** dialog box is displayed. 
6. Choose the **Insert** button and then select the top circular edge of the hole with the keyway in the Arm as the first face, see Figure 8-41. 
7. Select the outer circular edge on the front planar face of the Retainer as the second face to apply the **Insert** constraint, see Figure 8-41. Choose the **Apply** button.

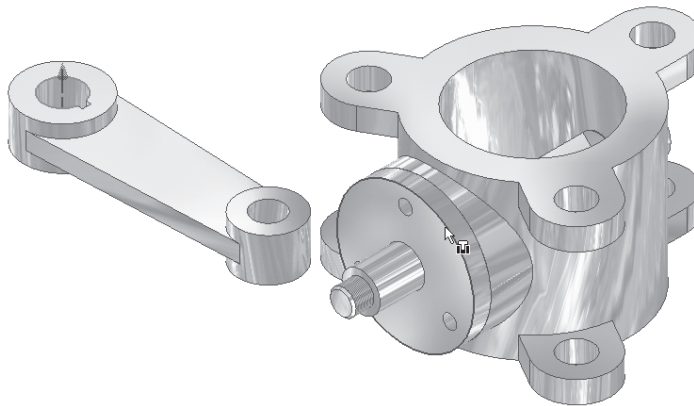


Figure 8-41 Selecting the faces to apply the **Insert** constraint

The Arm will be assembled with the Retainer and the Shaft will be inserted in the bigger hole of the Arm. The second constraint will be used to reorient the Arm such that it is

assembled at an angle to the top face of the Body. This angle is the same as the angle between the top face of the Body and the flat face of the Shaft.



Note

*In this case, it is presumed that the cylindrical features of the Arm are created on the XY plane. Also, the bigger and smaller cylindrical features are created from left to right respectively along the X axis direction when placed on the XY plane. Therefore, the XZ plane will pass through the center of the two cylindrical features. This XZ plane will be used to apply the **Angle** constraint.*

8. Choose the **Angle** button from the **Type** area. Select the top face of the Body as the first face to apply the constraint.



The second face that has to be selected to apply the constraint is the XZ plane of the Arm. The XZ plane of the Arm will not be displayed in the browser. You need to click on the + sign located on the left of the Arm in the browser to display the **Origin** folder and then from this folder, select the XZ plane.

9. Click on the + sign located on the left of the Arm in the browser to display the **Origin** folder. Click again on the + sign located on the left of the **Origin** folder to display all the work planes of the Arm. Select the XZ plane.
10. Enter the angle as **-135** in the **Angle** edit box. Choose **Apply** to apply the constraint and then **Cancel** to close the dialog box. Click on the - sign on the left of the Arm to close the folders. The assembly, after assembling the Arm, is shown in Figure 8-42.

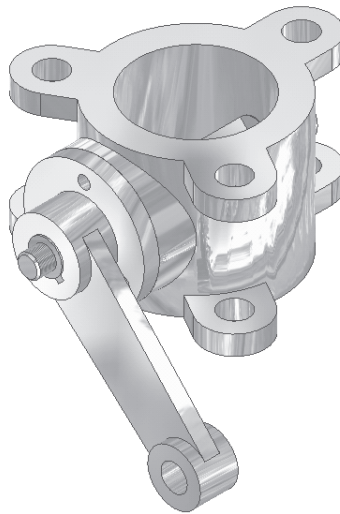


Figure 8-42 Assembly, after assembling the Arm

Assembling the Plate

The next component that has to be assembled is the Plate. You need to place it in the assembly file and then assemble it on the flat feature of the Shaft. You need to apply the **Insert** constraint twice to assemble the Plate with the Shaft. The first constraint will align

one of the holes on the Plate with one of the holes on the Shaft. The second constraint will align the second hole on the Plate with that on the Shaft.

Because the Shaft is assembled inside the Body, the Body will cause a restriction in viewing the components being assembled. To avoid this, Autodesk Inventor allows you to turn off the display of the components that you do not require for assembling the other components. Therefore, before proceeding with the assembling of the Plate, you can turn off the display of the Body. This is done using the browser.

1. Right-click on the Body in the browser to display the shortcut menu. You will notice that in the shortcut menu, there is a check mark in front of the **Visibility** option, see Figure 8-43. This suggests that the display of this component is turned on. Choose the **Visibility** option again to turn off the display of the Body.

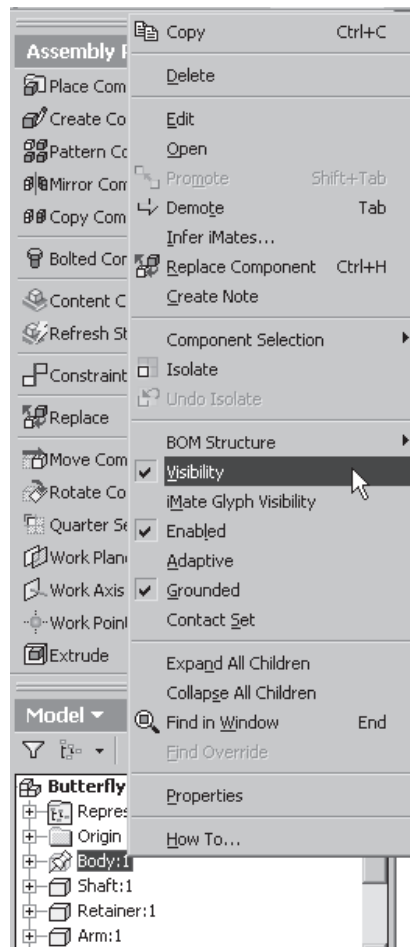





Figure 8-43 Turning off the display of the Body

The component whose visibility is turned off will be displayed in gray color in the browser.

2. Choose **Place Component** from the **Assembly Panel** panel bar to invoke the **Open** dialog box. 
3. Double-click on the Plate.
4. Place the Plate at a location where it does not interfere with the existing components.
5. Right-click in the drawing window to display the shortcut menu and choose **Done** to exit the component placement option.
6. Choose **Constraint** from the **Assembly Panel** panel bar. The **Place Constraint** dialog box is displayed. 
7. Choose the **Insert** button and then select the circular edge of one of the holes on the top face of the Plate as the first face to apply the constraint, see Figure 8-44. 
8. Select the circular edge of the right hole on the flat face of the Shaft as the second face to apply the constraint, see Figure 8-44. Choose the **Apply** button to apply the constraint.

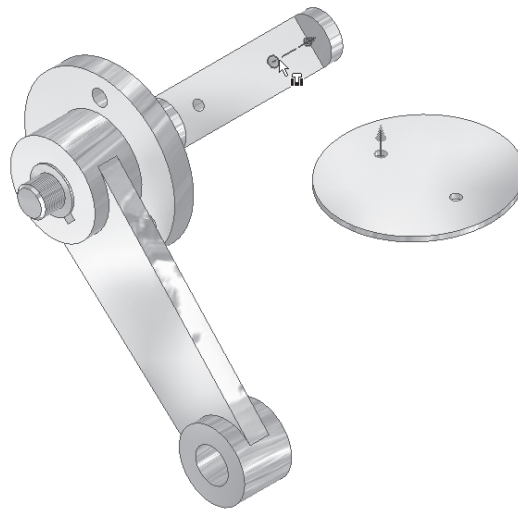


Figure 8-44 *Selecting the faces to apply the constraint*

As soon as you select the second face to apply the constraint, the Plate will move from its location and will be assembled with the Shaft. Now, the second constraint has to be applied on the other hole of the Plate. But the hole has to be selected on the face that is made coplanar with the flat face of the Shaft. Therefore, you need to reorient the model such that the back face of the Plate is visible and you can select the hole on that face to apply the constraint.

9. Reorient the assembly using the **Rotate** tool such that the back face of the Plate is visible.

10. Select the circular edge of the hole on the back face of the Plate as the first face to apply the constraint.

Because you rotated the model such that the back face of the Plate is visible, the flat face of the Shaft is not visible in the current view. Therefore, you need to switch back to the previous view. Sometimes, when you are inside the **Place Constraint** tool, you cannot use the F5 key to invoke the previous view. You need to invoke the **Rotate** tool again and then right-click to display the shortcut menu and choose **Previous View** from this menu.

11. Press the F5 key or choose the **Rotate** button from the **Inventor Standard** toolbar and right-click in the drawing window to display the shortcut menu. Choose the **Previous View** option to switch back to the previous view. Again right-click and choose **Done** to exit the **Rotate** tool.
12. Select the other hole on the flat face of the Shaft to apply the constraint. Choose the **Apply** button and then choose the **Cancel** button to exit the dialog box. The assembly, after assembling the Plate, is shown in Figure 8-45.

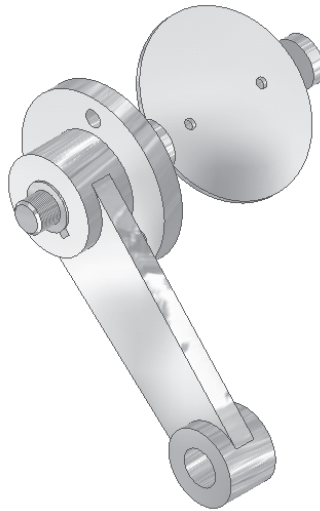





Figure 8-45 Assembly after assembling the Plate

Assembling the Screws




There are three instances of the Screw that need to be assembled such that they are inserted in the three holes on the Retainer. But before assembling the Screws, you need to turn off the display of the Arm so that it does not interfere in the display while assembling the other components.

1. Right-click on the Arm in the browser and choose **Visibility** to turn off the display of the Arm in the assembly.

2. Choose **Place Component** from the **Assembly Panel** panel bar to invoke the **Open** dialog box. 
3. Double-click on the Screw.
4. Place three instances of the Screw at a location where they do not interfere with the existing components.
5. Choose the **Place Constraint** button from the **Assembly Panel** toolbar or choose **Constraint** from the **Assembly Panel** panel bar. The **Place Constraint** dialog box is displayed. 
6. Choose the **Insert** button and then select the circular edge on the flat face of the head of the Screw as the first face to apply the constraint. 
7. Select the circular edge of one of the smaller holes on the front face of the Retainer as the second face to apply the constraint. Choose **Apply** to apply the constraint.
8. Similarly, assemble the other two Screws with the other two smaller holes on the Retainer.

Assembling the Nut

The Nut has to be assembled with the Shaft. Since the threaded portion of the Shaft has to be inserted inside the hole of the Nut, you will use the **Insert** constraint to assemble these components.

1. Choose **Place Component** from the **Assembly Panel** panel bar to invoke the **Open** dialog box. 
2. Double-click on the Nut.
3. Place the Nut at a location where it does not interfere with the existing components. Rotate the Nut using the **Rotate Component** tool such that the flat face of the Nut is visible in the current view.
4. Choose the **Place Constraint** button from the **Assembly Panel** toolbar or choose **Constraint** from the **Assembly Panel** panel bar. The **Place Constraint** dialog box is displayed. 
5. Choose the **Insert** button and then select the hole on the flat face of the Nut as the first face to apply the constraint. 
6. Select the end face (not on the side of the chamfered edge) of the threaded feature of the Shaft. Choose **Apply** and then choose **Cancel** to exit the dialog box.

Turning on the Display of the Body and the Arm

1. Right-click on the Body in the browser and choose **Visibility** from the shortcut menu to turn on the display of the Body in the assembly.
2. Similarly, right-click on the Arm in the browser and choose **Visibility**. The display of the Arm will be turned on in the assembly. Choose the **Zoom All** button from the **Inventor Standard** toolbar. The final Butterfly Valve assembly is shown in Figure 8-46.

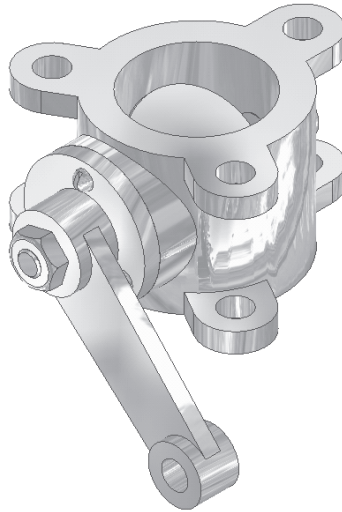


Figure 8-46 Final Butterfly Valve assembly

3. Save the assembly and close the file.

Tutorial 2

In this tutorial, you will create the components of the Plummer Block assembly and then assemble them in the assembly file. All components should be created as separate part files. After creating the components, place them in the assembly file and then assemble them. The dimensions of the components are shown in Figures 8-47 through 8-52. Assume the missing dimensions and the parameters for the threads. **(Expected time: 3 Hrs)**



Note

The orientation of the Casting you draw should match the orientation of the Casting shown in the assembly in Figure 8-47. This is because when you place the first component in the assembly file, it is placed on the same plane on which it was originally created in the part file. Since the Casting will be the first component you place in the assembly file, the base of it should be created on the XY plane. Therefore, when you place it in the assembly file, it is placed on the XY plane. The orientation of the other components also depends on the first component that you place in the assembly file.

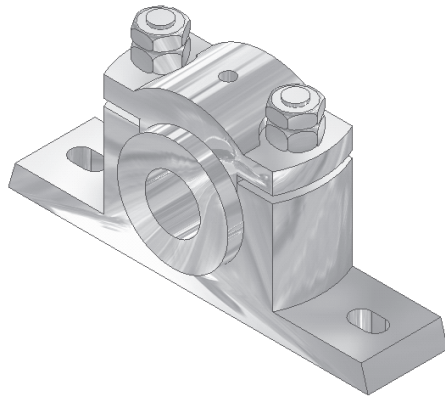


Figure 8-47 Plummer Block assembly

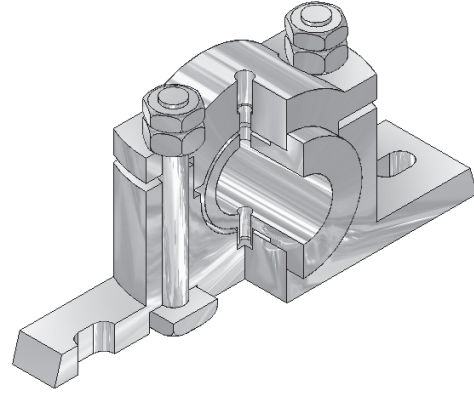


Figure 8-48 Sectioned isometric view of the Plummer Block assembly

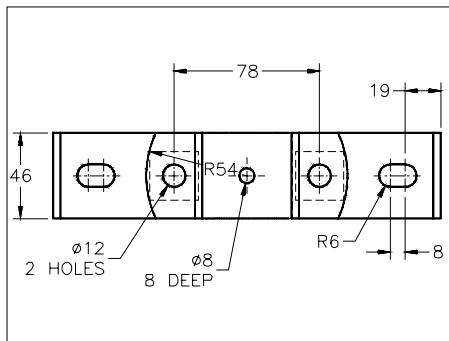


Figure 8-49a Top view of the Casting

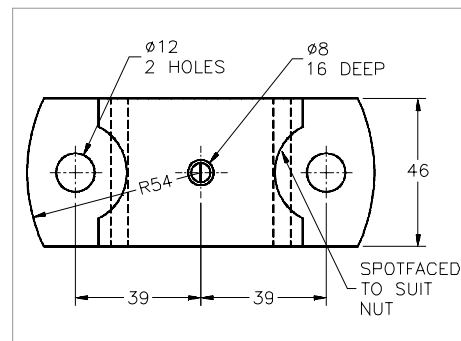


Figure 8-50a Top view of the Cap

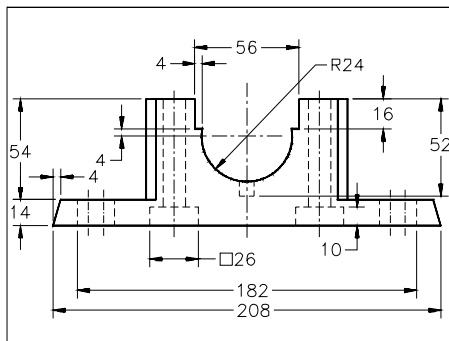


Figure 8-49b Front view of the Casting

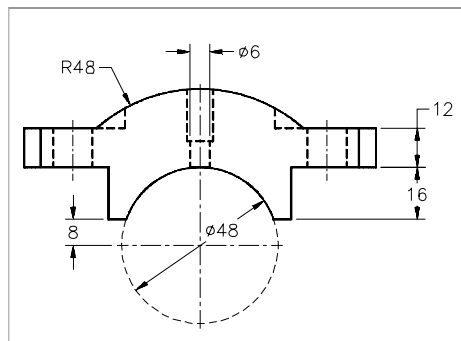


Figure 8-50b Front view of the Cap

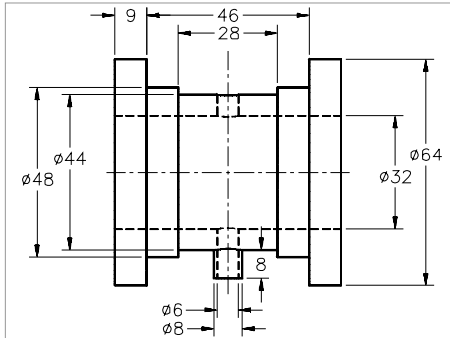


Figure 8-51 Dimensions of the Brasses

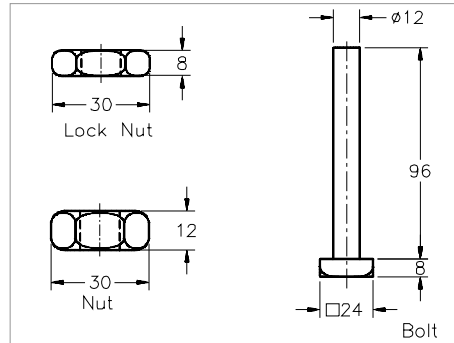


Figure 8-52 Dimensions of the Lock Nut, Nut, and Bolt

The following steps are required to create the assembly:

- Create all the components of the assembly as separate part files and save them in the `\PersonalProject\c08\Plummer Block` folder.
- Start a new metric assembly file and using the **Place Component** tool, place the Casting and the Cap.
- Assemble the two components using the assembly constraints.
- Next, turn off the display of the Cap and then place the Brasses in the assembly. Assemble the Brasses with the Casting and then turn off the display of the Brasses too.
- Place two instances of the Bolt in the assembly file and then assemble them with the Casting.
- Turn on the display of the Cap and place two instances of the Nut and the Lock Nut. Assemble both the instances of the Nut with the Cap and then the Lock Nut with the Nut.
- Finally, turn on the display of the Brasses to complete the Plummer Block assembly.


Creating the Components




- Create all the components of the Plummer Block assembly as separate part files. Specify the names of the files as shown in Figures 8-49 through 8-52. The files should be saved in the `\PersonalProject\c08\Plummer Block` folder.


Assembling the Casting and the Cap

The Casting and the Cap will be assembled using two assembly constraints. The first constraint is the **Insert** constraint that will align one of the holes on the top face of the Cap with its corresponding hole on one of the horizontal face of the Casting. You will also apply an offset of 4 mm between the mating faces in this constraint. The other constraint is the **Mate** constraint that will align the front face of the Cap with the front face of the Casting.

- Start a new assembly file. Save it with the name *Plummer Block.iam* in the `\PersonalProject\c08\Plummer Block` folder. This is the folder, in which all the individual part files are saved.

1. Choose **Place Component** from the **Assembly Panel** panel bar to invoke the **Open** dialog box. 
2. Double-click on the Casting.

Notice that one instance of the Casting is automatically placed in the assembly file. This instance is grounded and has a push pin icon in front of it in the browser. You are again prompted to place the component.
3. Right-click and choose **Done** from the shortcut menu. Similarly, place one instance of the Cap in the current assembly file. The location of the Cap should be such that it does not interfere with the Casting.
4. Choose **Rotate Component** from the **Assembly Panel** panel bar and rotate the Cap such that its bottom face is visible in the current view. 
5. Choose **Constraint** from the **Assembly Panel** panel bar; the **Place Constraint** dialog box is displayed. 
6. Choose the **Insert** button and then select the circular edge of the right hole on the bottom face of the Cap as the first face to apply the constraint. 
7. Select the circular edge of the right hole on the top face of the Casting as the second face to apply the constraint. Enter the value of the offset as **4** in the **Offset** edit box. Choose the **Apply** button.

The Cap is assembled with the Casting. However, the alignment of the front face of the Cap does not match the alignment of the front face of the Casting. Therefore, you need to apply the **Mate** constraint.
8. Choose the **Mate** button from the **Type** area and then select the front face of the Cap as the first face to apply the constraint. 
9. Select the front face of the Casting as the second face to apply the constraint and then choose the **Flush** button from the **Solution** area. Choose **Apply** and then choose **Cancel** to exit the dialog box. The assembly, after assembling the Cap and the Casting, is shown in Figure 8-53.

Assembling the Brasses

The Brasses is a circular part and so can be assembled using the **Insert** constraint. Notice that in the Brasses, there is a circular join feature (snug) created on the cylindrical face. The snug has to match with that of the hole on the circular face of the Casting. To proceed with assembling the Brasses, you first need to turn off the display of the Cap because it is not required in assembling the Brasses.

1. Right-click on the Cap in the browser and choose **Visibility** from the shortcut menu to turn off its display.

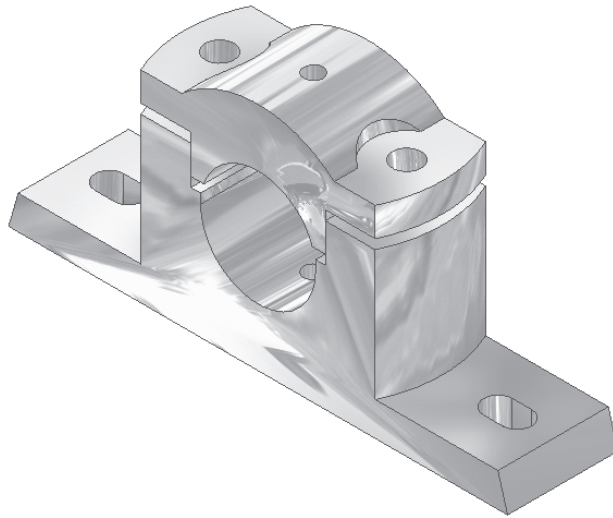


Figure 8-53 Assembly after assembling the Cap with the Casting

2. Place the Brasses using the **Place Component** tool. Invoke the **Place Constraint** dialog box using the **Place Constraint** tool. Choose the **Insert** button from the **Type** area and select the semicircular edge of the Casting as the first edge to apply the constraint, see Figure 8-54.
3. Select the circular edge of the Brasses shown in Figure 8-54 as the second edge to apply the constraint. Make sure the arrow points in the backward direction. Choose the **Apply** button.

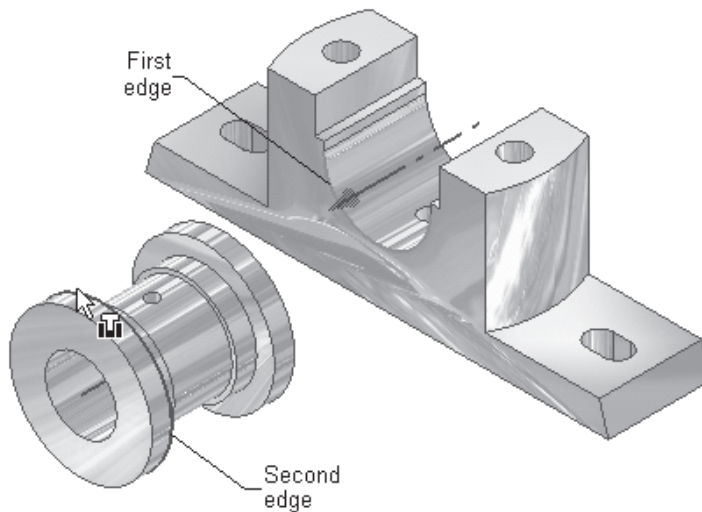


Figure 8-54 Selecting the faces to apply the **Insert** constraint

The next constraint is used to align the central axis of the snug created on the cylindrical face of the Brasses with that of the hole on the cylindrical face of the Casting. However, because the hole in the casting will not be visible even if you change the orientation of the assembly, you need to change the display mode from shaded to wireframe. In the wireframe display, the inside holes and features will also be displayed.

4. Change the display mode to wireframe by choosing the down arrow on the right of the **Shaded Display** button in the **Inventor Standard** toolbar and choosing **Wireframe Display**. Choose the **Mate** button from the **Type** area and move the cursor close to the snug on the cylindrical face of the Brasses. When you move the cursor close to the snug, the central axis of the snug will be displayed. If not, you can use the **Cycle Through** tool to select the central axis of the circular join feature as the first element to apply the constraint.
5. Select the central axis of the hole in the cylindrical face of the Casting as the second selection set. Choose the **Apply** button and then choose the **Cancel** button to apply the constraint and exit the dialog box.
6. Turn on the visibility of the Cap using the browser and change the display type to **Shaded Display**. The assembly is shown in Figure 8-55.

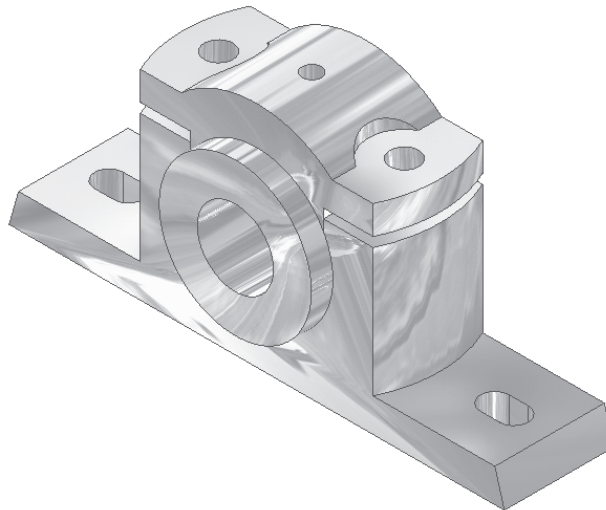


Figure 8-55 Assembly after assembling the Brasses

Assembling the Bolts

There are two instances of the Bolts that have to be assembled in the current assembly. But because the Brasses is not required for assembling the Bolts or the Nuts, you can turn off its display. After turning off the display of the Brasses, you will assemble the Bolts.

1. Turn off the display of the Brasses using the browser. Place two instances of the Bolt using the **Place Component** tool.

2. Invoke the **Place Constraint** dialog box and then choose the **Insert** button from the **Type** area. Select the circular edge on the top face of the base square feature of the Bolt as the first face to apply the constraint, see Figure 8-56.
3. Change the display type to wireframe and then select the circular edge on the top face of the square cut on the bottom face of the Casting, see Figure 8-56. Choose **Apply**.

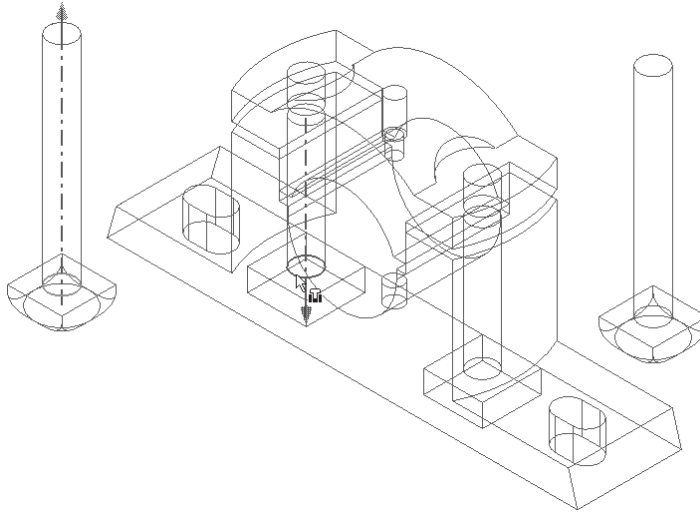


Figure 8-56 Selecting the faces to apply the constraint

4. Similarly, assemble the other Bolt and then change the display type to **Shaded Display**.

Assembling the Nuts and the Lock Nuts

1. Place two instances each of the Nut and the Lock Nut using the **Place Component** tool.
2. Invoke the **Place Constraint** dialog box and choose the **Insert** button. Select the circular edge of the hole on the top face of one of the Nut as the first face to apply the constraint.
3. Select the circular edge of the left hole on the top face of the Cap as the second face to apply the constraint. The Nut will be assembled with the Cap. Choose the **Apply** button to apply the constraint.
4. Select the circular edge of the hole on the top face of one of the Lock Nut as the first face to apply the constraint.
5. Select the circular edge of the hole on the top face of the Nut that is assembled with the Cap as the second face to apply the constraint. Choose the **Apply** button to apply the constraint.
6. Similarly, assemble the other Nut and the Lock Nut. Turn on the display of all the invisible components using the browser. The final Plummer Block assembly is shown in Figure 8-57.

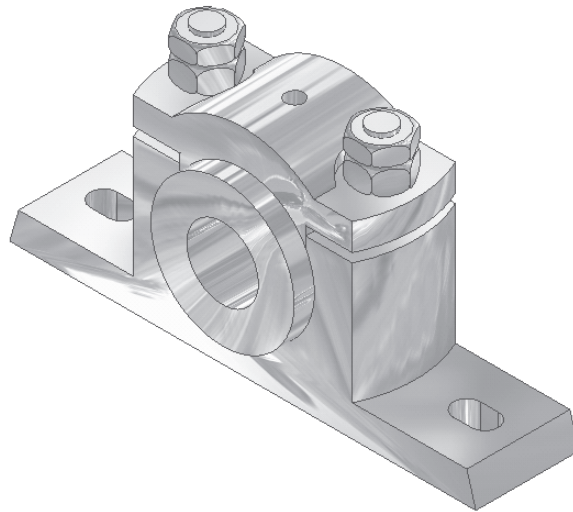


Figure 8-57 Final Plummer Block assembly

7. Save the assembly by choosing the **Save** button from the **Inventor Standard** toolbar.

Self-Evaluation Test

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

1. In Autodesk Inventor, you can use the bottom-up approach as well as the top-down approach for creating the assemblies. (T/F)
2. An assembly in which you use a combination of top-down and bottom-up approach is called a middle-out assembly. (T/F)
3. You can rotate individual components in the assembly file. (T/F)
4. You cannot invoke the sketching environment in the assembly file. (T/F)
5. The _____ tool is used to place the components in the assembly file.
6. The _____ icon is displayed in front of the grounded component in the browser.
7. When you invoke the **Place Constraint** tool, the _____ dialog box is displayed.

8. The _____ constraint is used to make the selected planar face, axis, or point of a component coincident with that of another component.
9. By default, the first component placed in the assembly file is _____.
10. The individual components in the assembly file can be moved using the _____ tool.

Review Questions

Answer the following questions.

1. You can change the display type of the components even when you are using a tool to perform a function. (T/F)
2. The components that are not grounded by default can also be grounded when required. (T/F)
3. The display of the components that are not required for assembling the other components can be turned off using the browser. (T/F)
4. If the component files are moved from their original location, they will not show up the next time you open the assembly file. (T/F)
5. The top-down assemblies are those in which all the components are created as individual part files and are placed in the assembly file. (T/F)
6. How many types of assembly constraints are available in Autodesk Inventor?
 - (a) 4
 - (b) 5
 - (c) 7
 - (d) 8
7. How many types of motion constraints are available in Autodesk Inventor?
 - (a) 2
 - (b) 3
 - (c) 4
 - (d) 5
8. Which tool is used to rotate the individual components in the assembly file.
 - (a) **Rotate Component**
 - (b) **Move Component**
 - (c) **Rotate**
 - (d) You cannot rotate the individual components
9. Which constraint is used to rotate one of the components in relation with the other component?
 - (a) **Rotation**
 - (b) **Rotation-Translation**
 - (c) **Mate**
 - (d) **Tangent**

Exercise

Exercise 1

Create the components of the Drill Press Vice assembly and then assemble them, as shown in Figures 8-58 and 8-59. The dimensions for the components are shown in Figures 8-60 through 8-63b. Create a folder with the name `\PersonalProject\c08\Drill Press Vice` and save all the component files and the assembly file in this folder. Assume the missing dimensions. You will use the bottom-up approach for creating this assembly.

(Expected time: 3 Hrs 15 min)

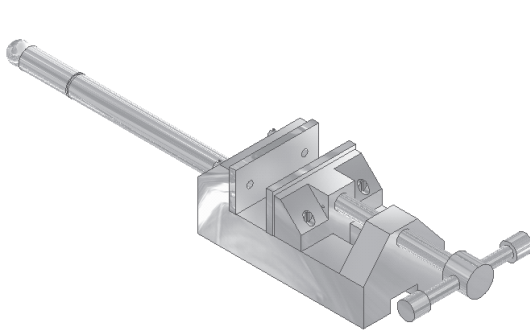


Figure 8-58 Drill Press Vice assembly

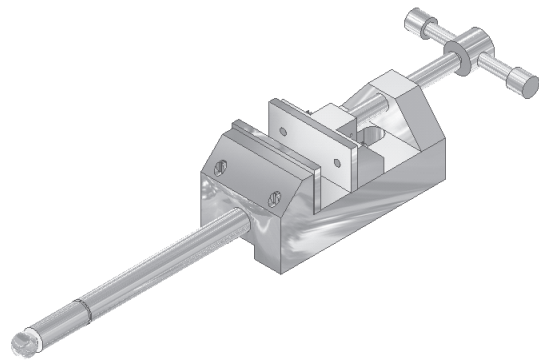


Figure 8-59 Drill Press Vice assembly

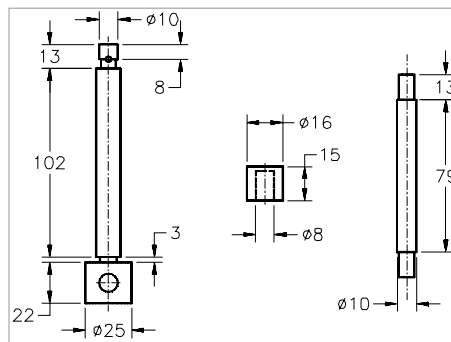


Figure 8-60 Dimensions of the Clamp Screw, Handle Stop, and Clamp Screw Handle

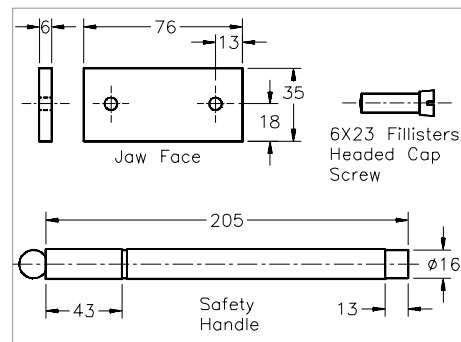


Figure 8-61 Dimensions of the Jaw Face, Cap Screw, and Safety Handle

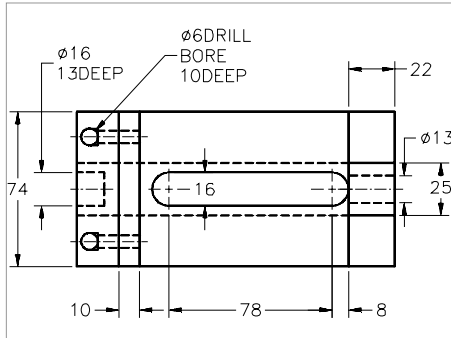


Figure 8-62a Top view of the Base

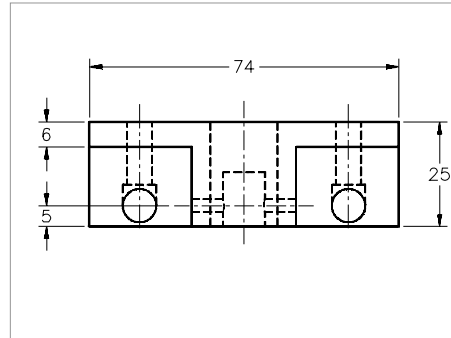


Figure 8-63a Top view of the Movable Jaw

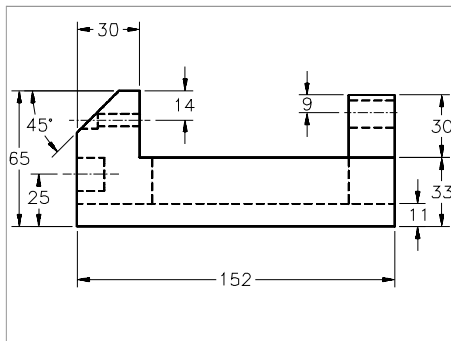


Figure 8-62b Front view of the Base

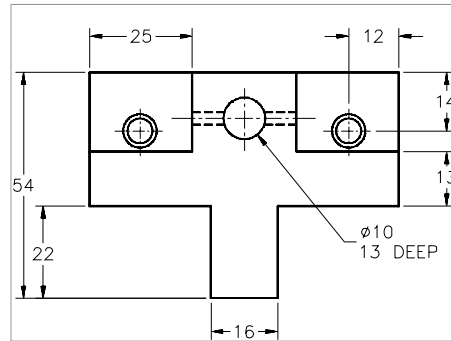


Figure 8-63b Front view of the Movable Jaw

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. F, 5. Place Component, 6. push pin, 7. Open, 8. Mate, 9. grounded, 10. Move Component