



# Chapter 8

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## Assembly Modeling-I

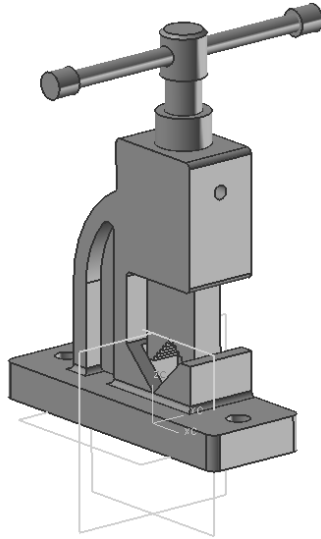
### Learning Objectives

**After completing this chapter, you will be able to:**

- *Start the Assembly application in NX.*
- *Understand the types of assembly design approaches in NX.*
- *Distinguish the meaning and usage of the assembly relations.*
- *Create assemblies using the bottom-up assembly design approach.*
- *Modify assembly relations.*
- *Modify a component in the Assembly application.*
- *Manipulate the components in the Assembly application.*
- *Create a pattern of components in an assembly.*

## THE ASSEMBLY APPLICATION

The **Assembly** application is used to create the inter relationship between the component parts, which are assembled together by applying a parametric link, both in dimensional and positional aspects. The assembly constraints are parametric in nature and so any type of modification with the assembly constraints at any stage of the assembling procedure is possible. Figure 8-1 shows the Pipe Vice assembly created in the **Assembly** application of NX.



*Figure 8-1 The Pipe Vice assembly created in NX*

The assembly files in NX have the file name extension as *.prt*. The **Assembly** application in NX is interactive and bidirectionally associative. The component can be modified in the assembly application itself by setting the part to be modified as the workpart. After assembling the components, you can also check the interference between them. This increases the efficiency of the assembly and also eliminates the errors while actually manufacturing the components. You can also create the exploded state of the assembly, which helps the technician in the assembly line to understand which part is to be inserted in a component and also their sequence of assembling.

## Starting a New File in the Assembly Application of NX

<b>Menu:</b>	Application > Assemblies
<b>Toolbar:</b>	Application > Assemblies



As mentioned earlier, the **Assembly** application is also invoked in the *.prt* file. Therefore, you need to start a new *.prt* file. Before invoking the **Assembly** application, it is recommended that you invoke the **Modeling** application and create the three default datum planes. These planes are generally used to assemble the first component of the assembly. Next, choose the **Assemblies** button from the **Application** toolbar or choose **Application > Assemblies** from the menu bar to invoke the **Assembly** application. The NX window, with the **Assembly** application, is shown in Figure 8-2.

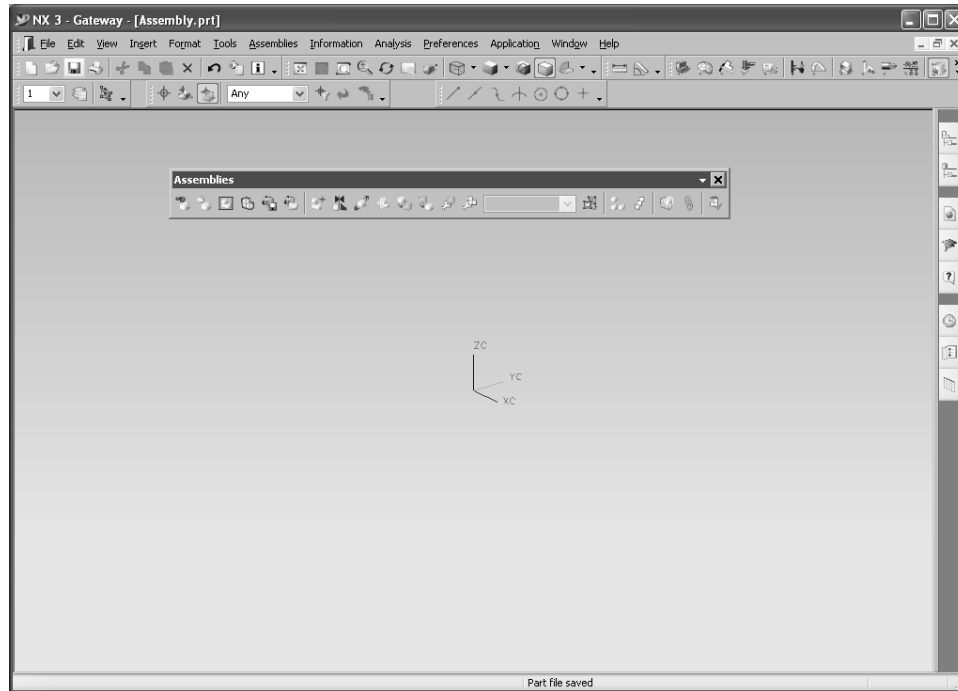


Figure 8-2 The NX 3 window in the *Assembly* application

## Types of Assembly Design Approaches

In NX, the assemblies can be created using two types of design approaches: the bottom-up assembly design approach and the top-down assembly design approach. In this chapter, you will learn about the bottom-up assembly design approach. The top-down approach is discussed in the next chapter.

### Bottom-up Assembly Design Approach

The bottom-up assembly design approach is the traditional and the most widely preferred approach of the assembly design. In this approach, all components are created as separate part documents and are placed in the assembly as external components. You need to create the components separately in the **Modeling** application and save them as *.prt* files. After creating all components, start a new file in the **Assembly** application and place the components using the tools in the **Assemblies** toolbar. Next, assemble them using the assembly constraints.

The main advantage of this assembly design approach is that you can pay more attention to the complexity of the component design. You can also capture the design intent easily. This approach is preferred while handling large assemblies or the assemblies with complex components.

## CREATING BOTTOM-UP ASSEMBLIES

As mentioned earlier, in bottom-up assemblies, the components are created as separate parts in the **Modeling** application and are placed in a new file that is started in the **Assembly** application. In the assembly file, they are assembled using the assembly constraints. Before you start assembling the components, it is recommended that you invoke the **Modeling** application and create the three default datum planes. The procedure of creating these planes is the same that you used in earlier chapters, while creating the components.

**Note**

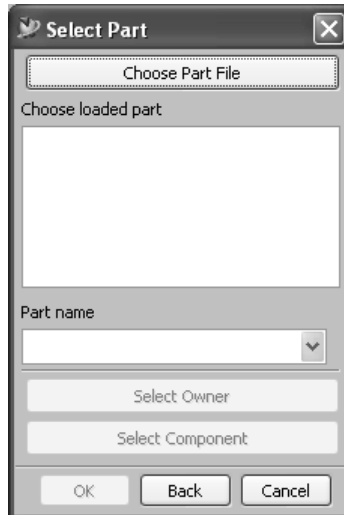
*The **Assemblies** toolbar should be invoked before inserting the base component. The **Assemblies** toolbar can be invoked by right-clicking on the toolbar area and selecting the **Assemblies** option from the shortcut menu.*

## Placing Components in the Assembly

<b>Menu:</b>	Assemblies > Component > Add Existing Component
<b>Toolbar:</b>	Assemblies > Add Existing Component



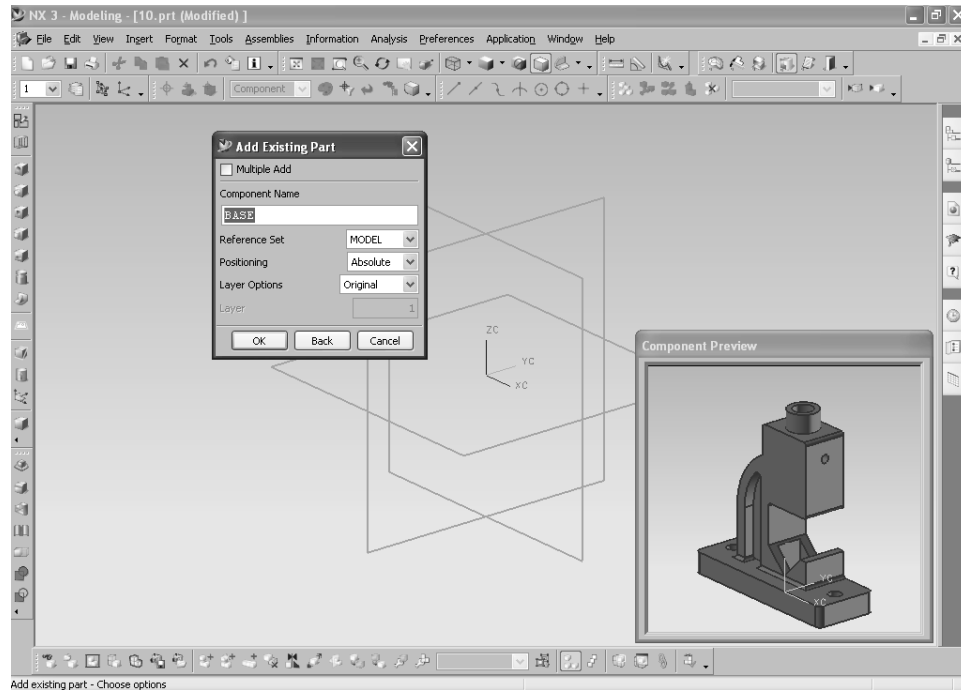
In NX, the components are placed in the **Assembly** application using the **Add Existing Component** tool. To place the base component, choose the **Add Existing Component** button from the **Assemblies** toolbar; the **Select Part** dialog box will be displayed, as shown in Figure 8-3.



*Figure 8-3 The **Select Part** dialog box*

Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box will be displayed. Browse and select the component to place and choose the **OK** button from **Part Name** dialog box; the **Add Existing Component** dialog box will be displayed, along with the **Components Preview** window. In this window, you can preview the selected component. The component that is displayed in the **Component Preview** window is known as the displayed

part. The **Assembly** application window with the **Add Existing Component** and the **Component Preview** window is shown in Figure 8-4.



**Figure 8-4** Assembly graphics window with the **Add Existing Part** dialog box and the **Component Preview** window

The name of the component is displayed in the **Component Name** edit box of the **Add Existing Part** dialog box. The **Multiple Add** check box in this dialog box is selected to add more than one instance of the same component. The remaining options in the **Add Existing Part** dialog box are discussed next.

### Reference Set

The options in the **Reference Set** drop-down list are used to specify the occurrence state of component references in the assembly. If you select the **Model** option from the **Reference Set** drop-down list, only the model will be placed in the assembly file. If you select the **Entire Part** option, the model will be placed in the assembly file along with the datum planes and sketches used for creating the model. If you select the **Empty** option, the component will be placed in the assembly file as an empty part without the model or its reference sets.



#### Note

To assemble the base component, you need to select the **Entire Part** option from the **Reference Set** drop-down list.

### Positioning

The options in the **Positioning** drop-down list are used to define the type of positioning required for

the component in the assembly file. To place assemble the components, the **Mate** option is selected from this drop-down list. This option allows you to apply assembly constraints between the component and the default datum planes in the assembly file. If the **Absolute** option is selected from the **Positioning** drop-down list, you cannot use the assembly constraints to make the component fully constrained. You can only position the component in the assembly file. If you select the **Reposition** option from the **Positioning** drop-down list, the component can be oriented in all the three directions by using the options in the **Reposition Component** dialogue box. The tools in the **Reposition Component** dialog box are discussed in the later part of this chapter.

**Note**

*If the component to be assembled is already placed in the assembly, then for subsequent placements, you can directly retrieve it from the **Select Part** dialog box.*

## Changing the Reference Set of the Component

<b>Menu:</b>	Assemblies > Component > Replace Reference Set
<b>Toolbar:</b>	Assemblies > Replace Reference Set



As mentioned earlier, when you place a component using the **Enter Part** option from the **Reference Set** drop-down list of the **Add Existing Part** dialog box, datum planes and sketches used to create that component are also displayed in the assembly file. After assembling the component, you do not need them as they cause confusion in the assembly file and also restrict the display of the other components.

To do this, right-click on the component in the assembly window and choose **Replace Reference Set > MODEL**; the datum planes and sketches of the component are no more displayed. You can also select the model from the assembly window and then select the **MODEL** option from the **Replace Reference Set** drop-down list in the **Assemblies** toolbar.

If you choose the **Replace Reference Set** button from the **Assemblies** toolbar and select the components, the **Replace Reference Set** dialog box will be displayed. This dialog box can also be used to select the required reference set.

## Applying Assembly Constraints to Components

In NX, the assembly constraints are applied using the **Mating Conditions** dialog box. This dialog box will be displayed when you select the reference set and the positioning options from the **Add Existing Part** dialog box and choose **OK**. Figure 8-5 shows the **Mating Conditions** dialog box.

The assembly constraints are used to constrain the six degrees of freedom of the component in an assembly. By constraining these degrees of freedom, you precisely place and position the components with respect to the other components in the assembly. There are eight types of assembly constraints in the **Mating Type** area of the **Mating Conditions** dialog box, as shown in Figure 8-6. All these assembly constraints are discussed next.

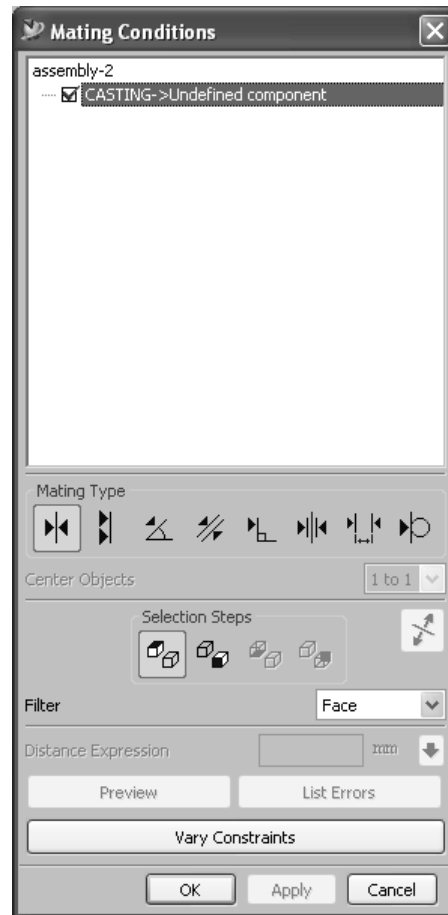


Figure 8-5 The Mating Conditions dialog box

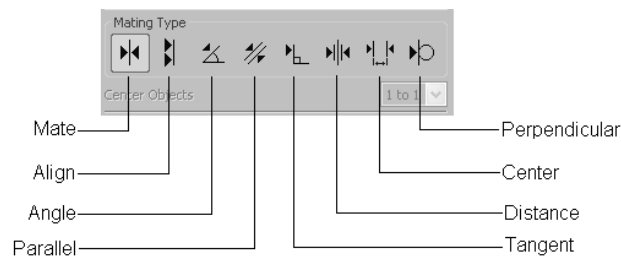


Figure 8-6 The Mating Type area

## Mate

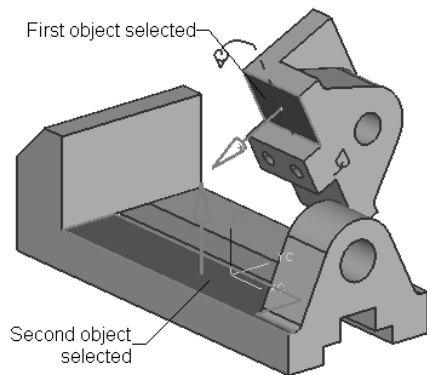


The **Mate** constraint allows you to make two selected reference faces or reference planes coplanar. The normal of the faces will point in the opposite direction. This constraint can also be used to make edges or axis collinear or points coincident. To apply this constraint, choose the **Mate** button from the **Mating Type** area of the **Mating Conditions**

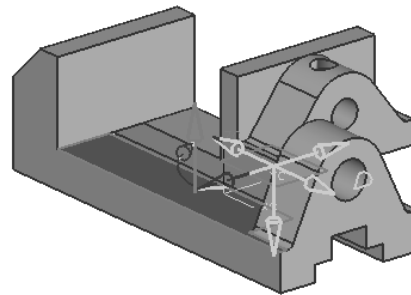
dialog box; you will be prompted to select the reference object from the component to be mated. You can select a planar face, curved face, edge, datum axis, or a point as the reference object from the component to be mated. You can select the type of reference object to be selected from the **Filter** drop-down list of the **Mating Conditions** dialog box. The reference object that you select from the component to be mated will be highlighted in red and you will be prompted to select the reference object from the component to mate to.

Select the assembly datum plane or any other reference object from the component displayed in the assembly window. After you select the reference objects from both the components, the **Degree of Freedom** arrows of purple and orange will be displayed from both the components, as shown in Figure 8-7. The orange arrows represent the constrained degrees of freedom as a result of applying the **Mate** constraint. The purple arrows represent the free degrees of freedom for the component that is to be constrained. If you selected edges to apply this constraint, the **Alternate Solution** button will be available in the **Mating Conditions** dialog box. By choosing the **Alternate Solution** button, you can flip the possible solutions for the specified **Mate** constraint.

You can choose the **Preview** button from the **Mating Conditions** dialog box to preview the resulting position of the components. Choose the **Apply** button from the **Mating Conditions** dialog box to apply the **Mate** constraint, as shown in Figure 8-8.



**Figure 8-7** Faces selected as reference objects for applying the **Mate** constraint



**Figure 8-8** Components after applying the **Mate** constraint

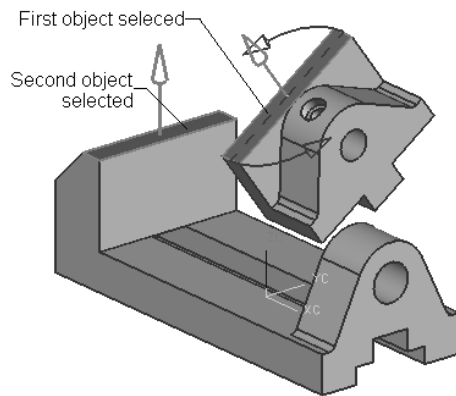
## Align



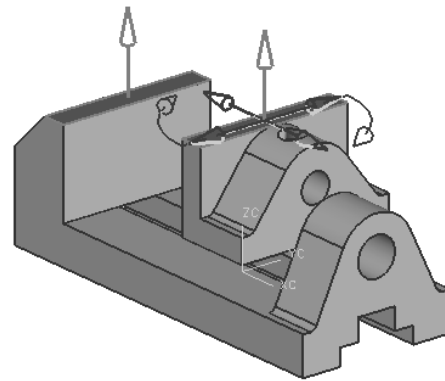
The **Align** constraint allows you to align the selected reference objects in the same direction. You can use this constraint to make the two selected reference objects coplanar. This is the reverse constraint of **Mate**. Choose the **Align** button from the **Mating Type** area of the **Mating Conditions** dialog box; you will be prompted to select the reference object that needs to be aligned from the component to be mated. You can select a planar face, curved face, edge, datum axis, or a point as the reference object from the component. You can select the type of reference object to be selected from the **Filter** drop-down list of the **Mating Conditions** dialog box. The reference object that you select from the component to be mated will be highlighted in red and you will be prompted to select the reference object from the component to mate to.



Select the reference object from the component to mate to displayed in the assembly window. After you select the reference objects, the degree of freedom arrows will be displayed, as shown in Figure 8-9. If you select edges as the reference objects, the **Alternate Solution** button will be available in the **Mating Conditions** dialog box. Choose the **Alternate Solution** button to flip the possible solutions for the specified **Align** constraint. Choose the **Preview** button from the **Mating Conditions** dialog box to preview the resulting position of the components. Choose the **Apply** button from the **Mating Conditions** dialog box to apply the **Align** constraint, as shown in Figure 8-10.



**Figure 8-9** Faces selected as reference objects for applying the **Align** constraint



**Figure 8-10** Components after applying the **Align** constraint

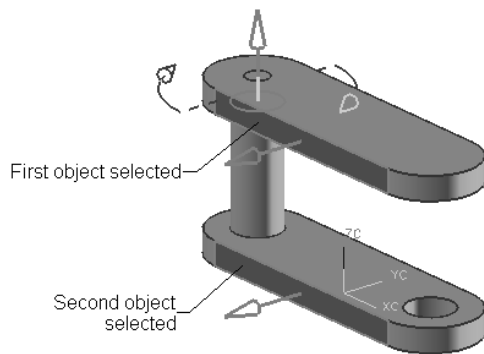
## Angle



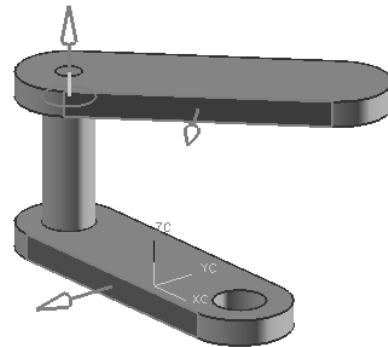
The **Angle** constraint is used to specify an angle between the two selected reference objects of the components. Choose the **Angle** constraint from the **Mating Type** area of the **Mating Conditions** dialog box; the **Angle Options** drop-down list will be available in the **Mating Conditions** dialog box. By default, the **Planar** option will be selected from this drop-down list and you will be prompted to select the reference object from the component to be mated. You can select a planar face, curved face, edge, datum axis or point as the reference object from the component to be mated. The selected reference object will be highlighted in red and you will be prompted to select the reference object from the component to mate to.

Select the reference object from the component in the assembly window; the degree of freedom arrows will be displayed, as shown in Figure 8-11. To apply the **Angle** constraint as the first constraint between the selected components, select the reference objects from the component to be mated; you will be prompted to select the rotation axis. You can select a planar face, curved face, edge, or datum axis for defining the rotation axis. At this stage, the **Angle Expression** edit box will be available in the **Mating Conditions** dialog box. Enter the angle to be subtended in the same edit box and choose the **Preview** button from the **Mating Conditions** dialog box to preview the resulting position of the components. The preview is shown in Figure 8-12.

If you choose the **3D** option from the **Angle Options** drop-down list, you can specify only positive angles between the selected reference objects. Even if you enter a negative angle value in the **Angle Expression** edit box, it will subtend the angle in the positive direction. If you



**Figure 8-11** Faces selected as reference objects for applying the **Angle** constraint



**Figure 8-12** Components after applying the **Angle** constraint

choose the **3D** option from the **Angle Options** drop-down list and apply the **Angle** constraint between the components for the first time, it will not prompt you to select the rotation axis. Instead, it will take the reference object selected from the component to mate to as the rotation axis and try to rotate the component about it.

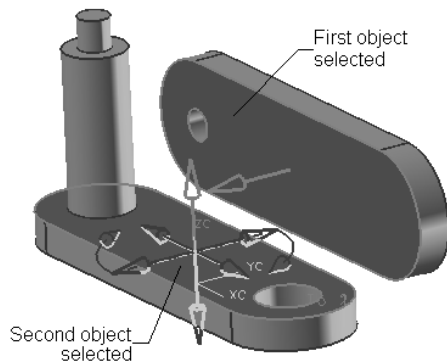
If you choose the **Orient** option from the **Angle Options** drop-down list, you will be allowed to mix the type of reference objects, while selecting them. For example, you can select a face as the reference object in the component to be mated and an edge as the reference object in the component to mate to and subtend the required angle between them. But the **Angle** constraint, after you select the **Orient** option from the **Angle Options** drop-down list, cannot be applied as the first constraint between the selected components.

## Parallel

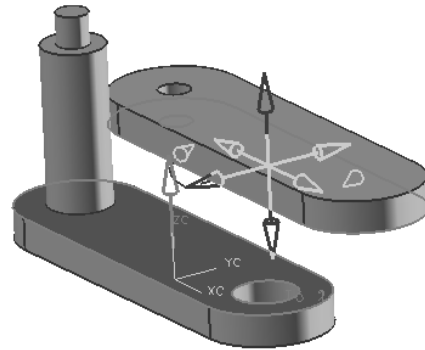


The **Parallel** constraint is used to place the selected reference objects parallel to each other. To apply the **Parallel** constraint, choose the **Parallel** button from the **Mating Type** area of the **Mating Conditions** dialog box. Select the reference objects from the two components. The degree of freedom arrows will be displayed, as shown in Figure 8-13. Choose the **Preview** button from the **Mating Conditions** dialog box to preview the resulting position of the components after you apply the **Parallel** constraint. Choose the **Alternate Solution** button to switch to the other possible solutions for the current selection set. Next, choose the **Apply** button and then the **OK** button from the **Mating Conditions** dialog box to apply the **Parallel** constraint, as shown in Figure 8-14.

You can apply the **Parallel** constraint between two cylindrical faces. In this case, the axes of the selected cylindrical faces are made parallel to each other. If you apply the **Parallel** constraint between a cylindrical face and a planar face, then the axis of the cylindrical face will become parallel to the normal of the selected planar face. You can also make an axis of the cylindrical face parallel to the selected edge, and vice versa.



**Figure 8-13** Faces selected as reference objects for applying the **Parallel** constraint

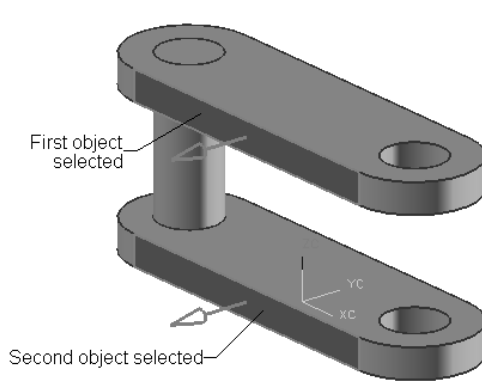


**Figure 8-14** Components after applying the **Parallel** constraint

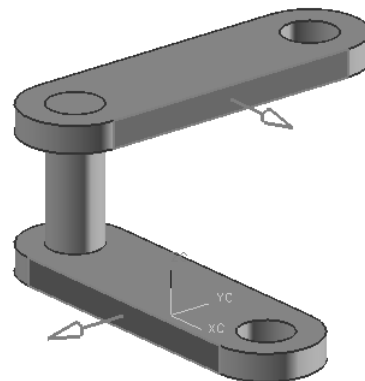
### Perpendicular



The **Perpendicular** constraint is used to place the selected reference objects normal to each other. To apply the **Perpendicular** constraint, choose the **Perpendicular** button from the **Mating Type** area of the **Mating Conditions** dialog box. Select the reference objects from the **From** and **To** components. The degree of freedom arrows will be displayed, as shown in Figure 8-15. Choose the **Preview** button from the **Mating Conditions** dialog box to preview the resultant position of the components. Next, choose the **Apply** button and then the **OK** button from the **Mating Conditions** dialog box for applying the **Perpendicular** constraint, as shown in Figure 8-16.



**Figure 8-15** Faces selected as reference objects for applying the **Perpendicular** constraint



**Figure 8-16** Components after applying the **Perpendicular** constraint

You can apply the **Perpendicular** constraint between two cylindrical faces. In this case, the axes of the selected cylindrical faces will be made normal to each other. If you apply the **Perpendicular** constraint between a cylindrical face and a planar face, the axis of the cylindrical face will become normal to the selected planar face. If you apply the **Perpendicular** constraint between two curved edges, then the selected curved edges will be placed normal to each other. If

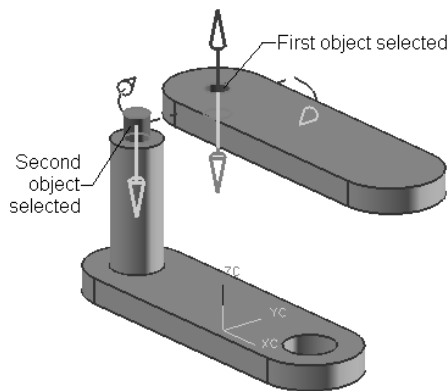
you apply the **Perpendicular** constraint between a linear edge and the curved edge, then the curved edge will be pivoted about the linear edge. Also, it can be rotated about a selected linear edge.

### Center

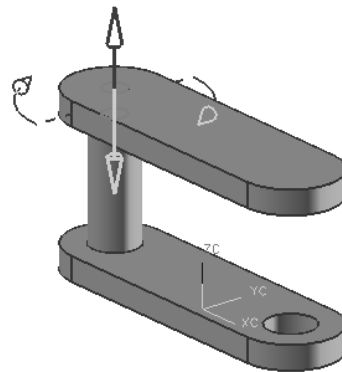


The **Center** constraint is used to coincide the central axis of a cylindrical face or a curved edge with a straight edge or the axis of a curved edge or a curved face. The limitation is that one of the two selected reference objects should be an entity that can represent its own central axis. Choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box; the **Center Objects** drop-down list will be available in the **Mating Conditions** dialog box and by default, the **1 to 1** option will be selected from this drop-down list. The general syntax of the **Center Objects** drop-down list options is the number of reference objects that can be selected from the component to be mated to the number of reference objects that can be selected from the component to mate to. For example, if you select the **2 to 1** option from the **Center Objects** drop-down list, you can select two reference objects from the component to be mated and one reference object from the component to mate to. By using these options, you can constrain more than one degree of freedom at a time.

On selecting this constraint, you will be prompted to select the reference object from the component to be mated. You can select a planar face, curved face, edge, datum axis, or point as the reference object from the component. On selecting the reference object, you will be prompted to select the reference object from the component to mate to. Select the reference object from the component displayed in the assembly window. The selected reference object will be highlighted in red, along with the degree of freedom arrows, as shown in Figure 8-17. Choose the **Apply** button from the **Mating Conditions** dialog box to apply the **Center** constraint, as shown in Figure 8-18.



**Figure 8-17** Faces selected as reference objects for applying the **Center** constraint



**Figure 8-18** Components after applying the **Center** constraint

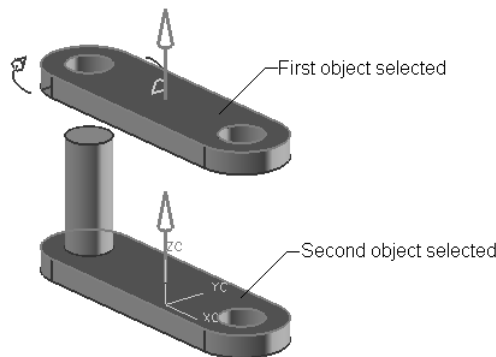
### Distance



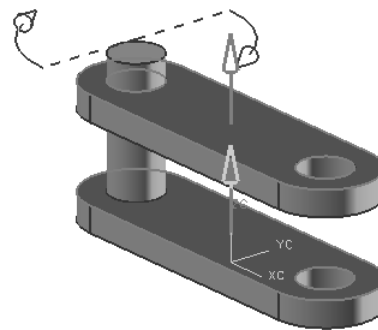
The **Distance** constraint is used to maintain the required offset between two selected reference objects. To apply this constraint, choose the **Distance** button from the **Mating Type** area of the **Mating Conditions** dialog box; you will be prompted to select the reference object from the component to be mated. Select the first reference object, as shown in

Figure 8-19; you will be prompted to select the reference object from the component to mate to. After you select the second reference object, the **Distance Expression** edit box will be available in the **Mating Conditions** dialog box.

Enter the offset value in the **Distance Expression** edit box and choose the **Preview** button to preview the resulting position of the components. If the resulting position of the components is not satisfied, choose the **Alternate Solution** button in the **Mating Conditions** dialog box to flip the possible solutions for the selection set. The resulting position of the components, after you apply the **Distance** constraint, is shown in Figure 8-20.



**Figure 8-19** Faces selected as reference objects for applying the **Distance** constraint



**Figure 8-20** Components after applying the **Distance** constraint

You can also apply the **Distance** constraint between the two cylindrical faces or circular edges. Select the reference objects from both the components and enter the offset value in the **Distance Expression** edit box. The axes of the two circular entities will be made parallel and they will be placed at a distance equal to the value of the offset distance.

## Tangent

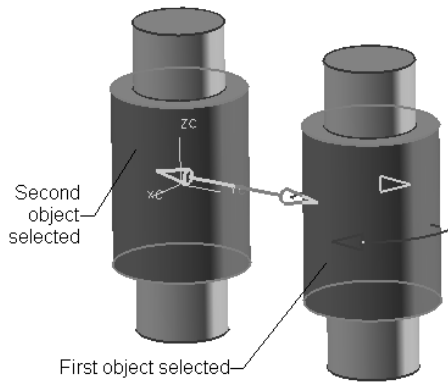


The **Tangent** constraint is used to create a tangential relation between a cylindrical surface and a planar surface or between two cylindrical surfaces. Choose the **Tangent** button from the **Mating Type** area of the **Mating Conditions** dialog box; you will be prompted to select the reference object from the component to be mated. You can select the cylindrical surface or a circular edge or a linear edge. The limitation in selection is that at least one of the reference objects should be a cylindrical or a circular entity. On selecting the first object, you will be prompted to select the reference object from the component to mate to. After you select the reference objects, as shown in Figure 8-21, the **Alternate Solution** button will be available in the **Mating Conditions** dialog box. Choose the **Alternate Solution** button to define the inward or outward tangency between the selected reference objects and then choose the **Apply** button for applying the **Tangent** constraint, as shown in Figure 8-22.

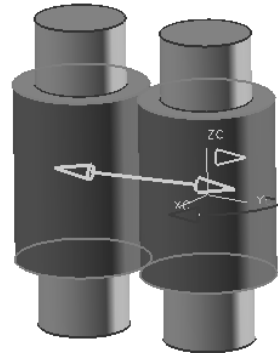


### Note

You cannot select a spherical face as a reference object for applying the **Tangent** constraint.



**Figure 8-21** Faces selected as reference objects for applying the **Tangent** constraint



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

## Using the Options in the Mating Conditions Dialog Box

The use of the other buttons in the **Mating Conditions** dialog box are discussed next.

### List Errors

The **List Errors** button is used to detect the inappropriate assembly constraints applied between the components in an assembly. When conflicting assembly constraints have been applied, you will be informed that conflicting constraints have been found. At this stage, the **List Errors** button will be available. Choose the **List Errors** button to view the errors in the assembly constraints in the **Information** dialog box.

### Vary Constraints

The **Vary Constraint** button is used to reorient the component in the assembly. The **Vary Constraints** button cannot be used to override the **Mating** conditions.

## Points to Remember while Assembling Components

The following points should be kept in mind to work efficiently in the **Assembly** application of NX.

1. The first component should be assembled in the assembly file using the datum planes.
2. It is important to make the components fully constrained using the assembly constraints before placing the next component. The component that is partially constrained will be disturbed from its location while using it to assemble the other components.
3. If you have to place the same component at various positions in the same assembly, the second time onward, you can directly select the component from the **Select Part** dialog box. There is no need of going to the **Part Name** dialog box again.

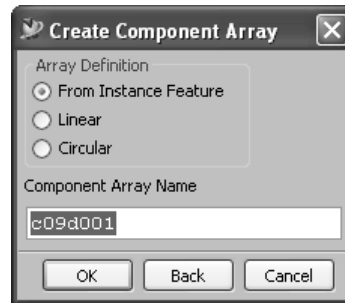
4. While you are in the process of making the component fully constrained, you can locate the temporary placement of the target component in the assembly using the **Preview** button. At least, three assembly constraints will be needed to make the component fully constrained.

## Creating a Component Array in an Assembly

**Menu:** Assemblies > Create Array  
**Toolbar:** Assemblies > Create Component Array



The **Create Component Array** tool is used to pattern a component in an assembly. It is similar to the method of patterning a feature in a component. For example, if in an assembly, there are ten bolts to be assembled, you don't have to assemble them ten times. Instead, assemble one of the bolts in any one of the holes and pattern it using the **Create Component Array** tool. To do this, choose the **Create Component Array** button from the **Assemblies** toolbar; the **Create Component Array** dialog box will be displayed, as shown in Figure 8-23.



*Figure 8-23 The Create Component Array dialog box*

The procedure for creating the component array in an assembly is discussed next.

1. Enter the name of the component array that is to be defined in the **Component Array Name** edit box; the name will identify the array in the assembly.
2. To create a circular array, select the **Circular** radio button and choose the **OK** button. The **Create Circular Array** dialog box will be displayed, as shown in Figure 8-24, and you will be prompted to define an object as an array axis. You can select a cylindrical face, datum axis, or an edge of the component to define the array axis. Enter the number of instances in the **Total Number** edit box and the angle of rotation in the **Angle** edit box in the **Create Circular Array** dialog box.
3. In case of a linear pattern, select the **Linear** radio button and choose the **OK** button. The **Create Linear Array** dialog box will be displayed, as shown in Figure 8-25, and you will be prompted to select the object to define the X direction. After you define the X direction, you will be prompted to select the object to define the Y direction. You can select a datum plane, edge, datum axis or a face normal to the object for defining the X and Y directions.

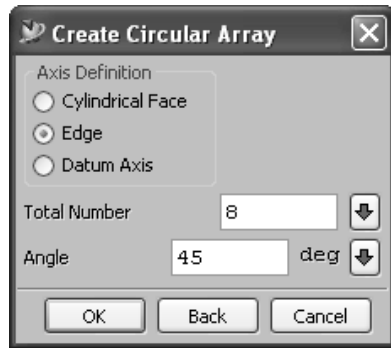


Figure 8-24 The **Create Circular Array** dialog box

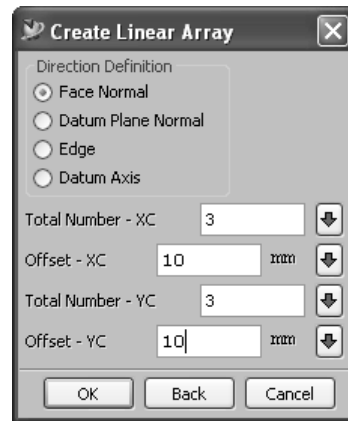


Figure 8-25 The **Create Linear Array** dialog box

Enter the number of instances required along the X direction in the **Total Number - XC** edit box and enter the offset distance to be maintained between the components along the same direction in the **Offset - XC** column. Similarly, enter the number of instances required along the Y direction in the **Total Number - YC** edit box and enter the offset distance to be maintained between the components along the same direction in the **Offset - YC** edit box of the **Create Linear Array** dialog box.

4. The **From Instance Feature** radio button of the **Create Component Array** dialog box is used to pattern a component with reference to the array feature created in the component in the **Modeling** application. For example, if you have created a circular pattern of the holes in a component at the **Modeling** application, you can use it to assemble a bolt on each of the patterned holes. By using the pattern feature created in the **Modeling** application as a reference, you can create the pattern of a bolt in the **Assembly** application. After assembling one of the instances of the Bolt using the assembly constraints shown in Figure 8-26, invoke the **Create Component Array** dialog box. Next, select the **From Instance Feature** radio button and choose the **OK** button to automatically pattern the selected component in the assembly with reference to the created in the **Modeling** application, as shown in Figure 8-27.

## Replacing a Component in an Assembly

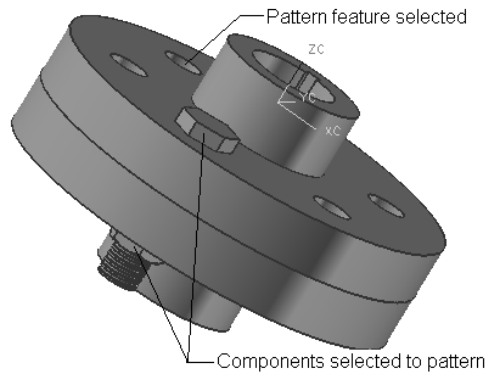
**Menu:** Assemblies > Component > Substitute Component  
**Toolbar:** Assemblies > Substitute Component



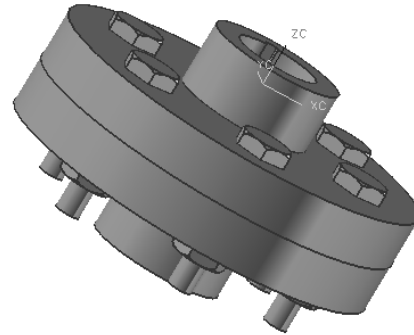
The **Substitute Component** tool is used to replace a component that is already placed in the assembly. Invoke the **Substitute Component** tool from the **Assemblies** toolbar; you will be prompted to select the component for an operation. Select the component and choose the **OK** button from the **Substitute Component Icon Options**; the **Substitute Component** message window will be displayed, as shown in Figure 8-28.

If you choose the **Remove and Add** button from the **Substitute Component** message window, the **Select Part** dialog box will be displayed and you will be prompted to select the replacement



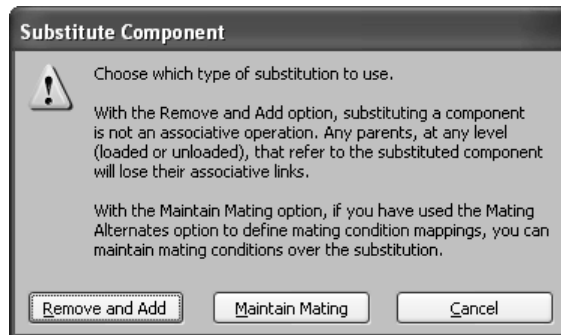


**Figure 8-26** Components and the instance of the pattern feature to be selected

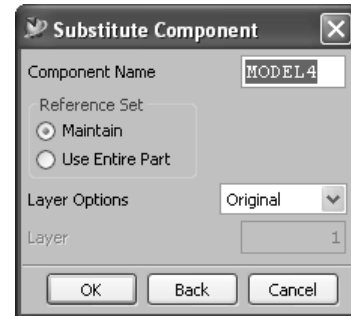


**Figure 8-27** Assembly after creating the derived component pattern

part. You can select the replacement part from the list box of the **Select Part** dialog box if the part name is listed. Alternatively, you can browse for the part by choosing the **Choose Part File** button from the dialog box. After you select the replacement part, the **Substitute Component** dialog box will be displayed, as shown in Figure 8-29. Select the **Maintain** radio button from the **Reference Set** area to maintain the same reference set type with the replacement component or select the **Use Entire Part** radio button, if you want to have all the reference entities, along with the replacement component. If you adopt this method of substitution, the previously defined assembly constraints for the replaced component will be deleted and you will need to apply the assembly constraints again for making the replacement component a fully constrained one.



**Figure 8-28** The *Substitute Component* message window



**Figure 8-29** The *Substitute Component* dialog box

If you select the **Maintain** radio button from the **Reference Set** area, the older component will be replaced with a new component and the assembly constraints applied to the older component will be retained in the new component.

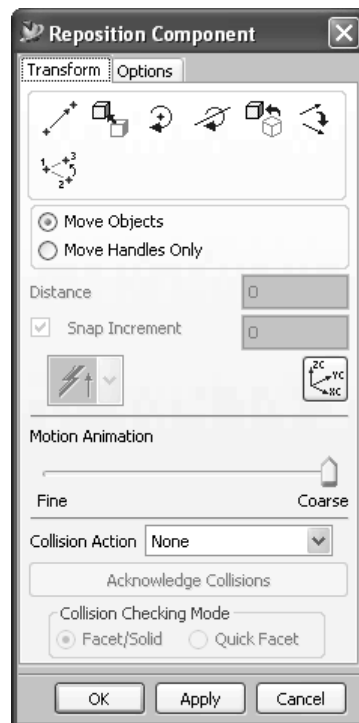
## Repositioning a Component in an Assembly

**Menu:** Assemblies > Components > Reposition Component  
**Toolbar:** Assemblies > Reposition Component



The **Reposition Component** tool is used to move and reorient the assembled component in the 3D space, with respect to the available degrees of freedom. Remember that the component cannot be reoriented along the constrained degrees of freedom using the **Reposition Component** tool.

Invoke the **Reposition Component** tool from the **Assemblies** toolbar; you will be prompted to select the component that is to be repositioned. Select the component and choose the **OK** button from the **Reposition Component Icon Options**; the **Reposition Component** dialog box will be displayed, as shown in Figure 8-30.



**Figure 8-30** The **Reposition Component** dialog box

Also, the handles of the selected component will be activated. The possible degrees of freedom for the component will be identified by noting the available translation or rotational cubes in the handle. The grayed out handle cube indicates the constrained degrees of freedom of the component. You can reorient the component dynamically by selecting the required handle cubes and dragging them by pressing the left mouse button. If you select the **Move Objects** radio button option from the **Reposition Component** dialog box; the selected component will

be repositioned. If you select the **Move Handles Only** radio button option from the dialog box, only the handles of the selected component will be repositioned. By default, when the **Reposition Component** dialog box is displayed, the **Move Objects** radio button is selected. In addition to the dynamic method of repositioning, you can also reposition the component using the methods in the **Reposition Component** dialog box. These methods are discussed next.

### Point to Point

If you choose the **Point to Point** button from the **Reposition Component** dialog box, the **Point Constructor** dialog box will be displayed, and you will be prompted to select the first point from the component to be repositioned. Specify the first point from the selected component. The **Point Constructor** dialog box will be displayed and you will be prompted to specify the second point or destination point for repositioning the component. After you specify the points, the component will be moved to the new position.

### Translate

The **Translate** button in the **Reposition Component** dialog box will be used to place the selected component in the 3D space by specifying the X, Y, and Z coordinates. By choosing the **Translate** button from the **Reposition Component** dialog box, the **Transformations** dialog box will be displayed, as shown in Figure 8-31. The **Transformations** dialog box has separate edit boxes for entering the X, Y, and Z coordinates to position the selected component in the 3D space with respect to the WCS of the assembly file.

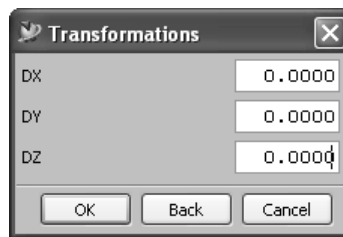


Figure 8-31 The Transformations dialog box

### Rotate About a Point

By choosing the **Rotate About a Point** button from the **Reposition Component** dialog box, you can rotate a component about a specified point. On choosing this button, the **Point Constructor** dialog box will be displayed and you will be prompted to specify a rotation point. After specifying the rotation point, the handles of the selected component will be moved to the specified point. By selecting the required handle cube of rotation, you can rotate the component about the specified point.

### Rotate About a Line

By choosing the **Rotate About a Line** button from the **Reposition Component** dialog box, you can rotate the selected component about a line by considering it as the axis of rotation. After you select this button, the **Point Constructor** dialog box will be displayed and you will be prompted to specify the start point of the line. After you specify the start point, the **Vector Constructor**

dialog box will be displayed and you will be prompted to specify the direction for creating the line from the specified start point. After you specify the vector direction for constructing the line, the handles of the selected component will be moved to the specified start point. By selecting the required rotational handle cube, you can rotate the selected component about the specified line.

### Reposition

The **Reposition** button from the **Reposition Component** dialog box will be used to move the component from one point to another by constructing new coordinate systems in the 3D space. Choose the **Reposition** button from the **Reposition Component** dialog box, the **CSYS Constructor** dialog box will be displayed. Construct the new coordinate system and choose the **OK** button from the dialog box; the selected component will be moved to a new position.

### Rotate Between Axes

The **Rotate Between Axes** button allows you to rotate the selected component between two selected vectors. On choosing this button, the **Point Constructor** dialog box will be displayed and you will be prompted to specify the reference point. After you specify the reference point, the **Vector Constructor** dialog box will be displayed twice for specifying both the axes about which the component will be rotated. After you specify the two axes, the coordinate system will be formed at the reference point specified. Now, enter the rotational angle in the **Angle** edit box of the **Reposition Component** dialog box and the selected component will be rotated accordingly.

### Rotate Between Points

By specifying three points on the same plane, you can rotate the selected component using the **Rotate Between Points** button from the **Reposition Component** dialog box. Select the **Rotate Between Points** button from the **Reposition Component** dialog box; you will be prompted to specify the rotation point. Specify the rotation point; you will be prompted to specify the first point. Specify the first point for representing the X axis; you will be prompted to specify the second point. Specify the second point for representing the Y axis. Enter the angle of rotation in the **Angle** edit box and choose the **OK** button for rotating the component.

### Snap Handles to WCS

By choosing the **Snap Handles to WCS** button from the **Reposition Component** dialog box, you can make the handles of the selected component coincide with the WCS of the assembly file.

### Motion Animation

The **Motion Animation** sliding bar of the **Reposition Component** dialog box will be used to define the quality of motion which the component undergoes when it is being repositioned. The motion of the component can be controlled from **Fine** to **Coarse** using the sliding bar. The **Fine** option visualizes the displacement of the component in a better way.

### Collision Action

The **Collision Action** drop-down list is used to restrict the collision of one component with another component. If the **Highlight Collision** option is selected from the **Collision Action** drop-down list, the collision between the components will be highlighted in red during the

process of repositioning. If the **Stop on Collision** option is selected, the movement of the component will be stopped as it will collide with the other component. No further movement is possible after collision in the latter case.

## Mirroring a Component in an Assembly

**Menu:** Assemblies > Component > Mirror Assembly  
**Toolbar:** Assemblies > Mirror Assembly



The **Mirror Assembly** tool is used for mirroring the assembly or any of the individual components in the assembly about a reference plane. Note that you cannot select a planar face of a component for this purpose. When you choose the **Mirror Assembly** button from the **Assemblies** toolbar, the **Mirror Assembly Wizard** dialog box will be displayed with a **Welcome Page**, as shown in Figure 8-32.



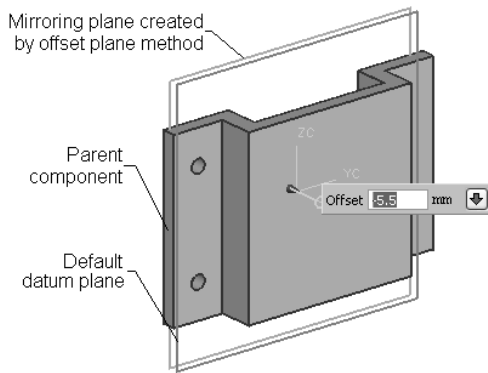
*Figure 8-32 The Welcome page of the Mirror Assembly Wizard dialog box*

To select components or assembly to mirror, choose the **Next** button; the **Select Components** page will be displayed and you will be prompted to select the components to be mirrored. Select the components that are to be mirrored and choose the **Next** button to proceed to the **Select Plane** page.

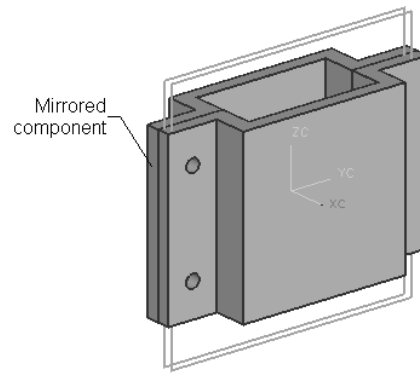
Select any of the reference planes as the mirroring plane. You can also create a mirroring plane, as shown in Figure 8-33, by choosing the **Create Datum Tool** in the **Select Plane** page. After selecting the plane, choose the **Next** button to proceed to the **Mirroring Setup** page. There are two types of mirroring operations available. By default, the **Assign Reposition Operation** type will be chosen. If the mirroring operation is performed by using this type, the mirrored component will be the same part file. That means an associative link will be maintained between the parent and the mirrored component. The name of the mirrored component will be the same as that of the parent component.

If the mirroring operation is performed by using the **Assign Mirror Geometry Operation** option, the mirrored component will be a separate copy of the parent component and also an associative link will be maintained between the parent and the mirrored components. Note that the **Assign Mirror Geometry Operation** button will be available only after you select the components from the list box in this area.

After choosing the mirror type, choose the **Next** button; the **Mirror Review** page will be displayed. On this page, you can use the **Cycle Mirror Solutions** button to cycle through the possible solutions. You can also directly select the required solution from the **Cycle Mirror Solutions** drop-down list. After you select the solution, choose the **Finish** button to view the mirrored component in the assembly, as shown in Figure 8-34.



**Figure 8-33** The parent part after creating the mirroring plane



**Figure 8-34** The assembly with the mirrored component

If you selected the **Assign Mirror Geometric Operations** option from the **Mirroring Setup** page, the **Naming Policy** page will be displayed. You can create a new name for the mirrored component by adding a prefix or suffix to the parent component's name. To add a suffix, select the **Add this as suffix to the original name** radio button from the **Naming Rule** area and enter the suffix name in the edit box provided in the same area. To add a prefix, select the **Add this as prefix to the original name** radio button from the **Naming Rule** area and enter the suffix name in the edit box provided in the same area.

To save the mirrored component in a different folder, select the **Add new parts to the specified directory** radio button from the **Directory Rule** area of the **Naming Policy** page; the **Browse** button will be available. When you choose this button, the **Choose New Part Save Directory** dialog box will be displayed. You can browse to a new directory for saving the mirrored component.

Next, choose the **Finish** button to exit the **Mirror Assembly Wizard** dialog box. To completely change the name of the mirrored component, choose the **Next** button from the **Naming Policy** page; the **Name New Part Files** page will be displayed. Double-click on the part name to be changed; the **Rename New Part File** dialog box will be displayed where you can specify a new name for the file. After changing the name, choose the **OK** button, the **Name New Part Files** page will be displayed with the new name, as shown in Figure 8-35.



Figure 8-35 The Name New Part Files page of the Mirror Assembly Wizard dialog box



#### Note

The mirrored components do not have assembly constraints. You need to manually add these constraints to the mirrored components.

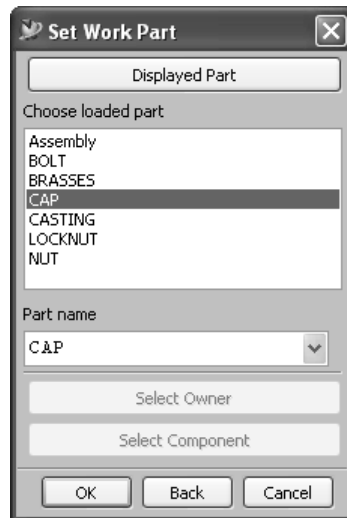
## Modifying a Component in the Assembly File

**Menu:** Assemblies > Context Control > Set Work Part  
**Toolbar:** Assemblies > Make Work part



The **Make Work part** tool is used to modify or add sketches and features to the component that are placed in the assembly file by bringing the component into the current **Work Part** category. The procedure for modifying the component in the assembly file is as discussed next.

Invoke the **Make Work part** tool from the **Assemblies** toolbar; the **Set Work Part** dialog box will be displayed, as shown in Figure 8-36, and you will be prompted to select the component to be brought in the category of the work part. Select the component in the assembly and choose the **OK** button. The component color will be changed to distinguish it from the assembly. After converting the component to the **Work Part**, you can modify or perform any type of feature and sketch-based operations. The changes made in the component in the assembly are also updated in the part file of the component. To exit to the assembly mode, choose the **Assembly Navigator** button from the resource bar; the cascade menu will be displayed. Select the assembly name from the assembly node tree and right-click on it. Choose the **Make Work Part** option from the shortcut menu. After you select the assembly as a work part, the whole assembly will be displayed in the same color.



**Figure 8-36** The *Set Work Part* dialog box



**Note**

You can select only one component at a time in the **Work Part** category. After you make the changes in the component, you can directly save the changes in the component file by saving the assembly file.

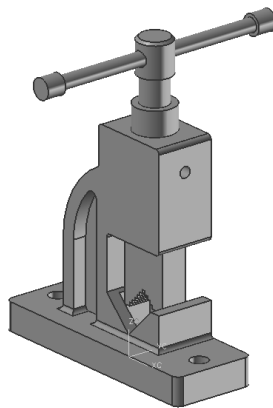
## TUTORIALS

### Tutorial 1

In this tutorial, you will create all components of the Pipe Vice assembly shown in Figure 8-37 and then assemble them. The dimensions of the components are given in Figures 8-38 and 8-39. After completing the tutorial, save the file with the name given below.

*NX 3\c08\Pipe Vice\Pipe Vice.prt*

**(Expected Time : 2.5 hrs)**



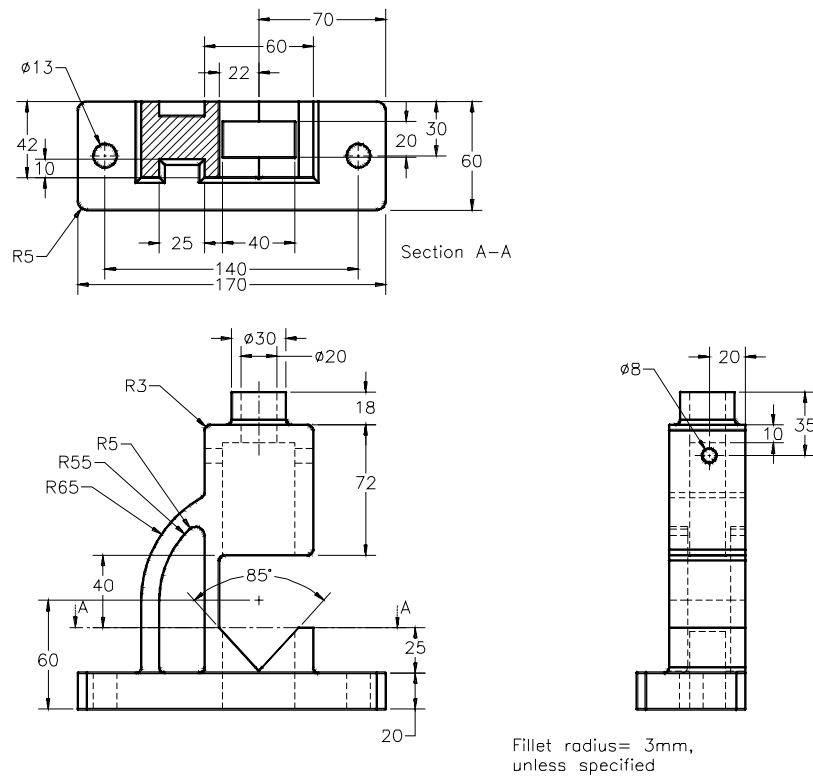
**Figure 8-37** The *Pipe Vice* assembly





The following steps are required to complete this tutorial:

- Create all components as individual part files and save them. The part files will be saved in the `|NX 3|c08|Pipe Vice` folder.
- Start the new file in the **Modeling** application and create the three default planes. Invoke the **Assembly** application.
- Insert the Base in the assembly window and make it fully constrained using the assembly constraints, refer to Figure 8-40.
- Insert the Screw in the assembly window and apply the required assembly constraints to make it a fully constrained part, refer to Figures 8-41 through 8-47.
- Insert the Movable Jaw in the assembly window and apply the required assembly constraints, refer to Figures 8-48 through 8-54.
- Insert the Handle into the assembly window and apply the required assembly constraints, refer to Figures 8-55 through 8-61.
- Insert the Handle Stop into the assembly window and apply the required assembly constraints,



*Figure 8-39 Views and dimensions of the Base*

refer to Figures 8-62 to 8-69. Similarly, assemble the other instances of the Handle Stop to the assembly.

### Creating the Assembly Components


1. Create all the components of the Pipe Vice assembly as separate part files. Specify the names of the files shown in Figures 8-38 and 8-39. The files should be saved in the `|NX 3|c08|Pipe Vice` folder.

### Starting the NX session in the Modeling Application




1. Start a new part file in the **Modeling** application and name it `Pipe Vice.prt`. Next, create the three default datum planes using the **Datum Plane** tool.
2. Invoke the **Assembly** application by choosing **Application > Assemblies** from the menu bar, if it is not already invoked.

### Assembling the Base with the Three Default Datum Planes

The Base will be the first component to be placed in the assembly file. Rest of the components will be assembled with the Base.

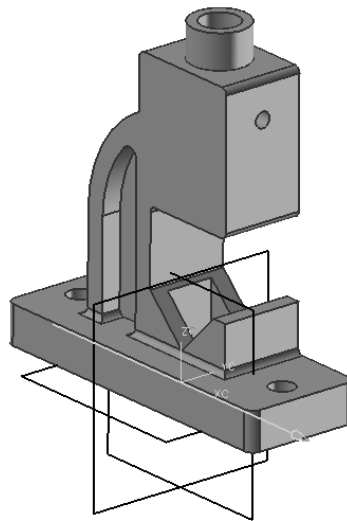
1. Choose the **Add Existing Component** button from the **Assemblies** toolbar to invoke the **Select Part** dialog box. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Select the Base from the **Part Name** dialog box and choose the **OK** button.

The Base is displayed in the **Component Preview** window at the lower right corner of the main window. The **Add Existing Part** dialog box is displayed on the upper left corner of the main window.

4. Select the **Entire Part** option from the **Reference Set** drop-down list and select the **Mate** option from the **Positioning** drop-down list. Choose the **OK** button from the **Add Existing Part** dialog box; the **Mating Conditions** dialog box is displayed.
5. Select the **Datum Plane** option from the **Filter** drop-down list of the **Mating Conditions** dialog box before applying the assembly constraints.
6. Choose the **Align** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
7. Select the right reference plane of the base from the **Component Preview** window and the right reference plane from the main window. Do not choose the **Apply** button. Else, the Base will be displayed in the assembly window.
8. Choose the **Align** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
9. Select the top reference plane of the base from the **Component Preview** window and the top reference plane from the main window.
10. Choose the **Align** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
11. Select the front reference plane of the Base from the **Component Preview** window and the front reference plane from the main window. At this stage, you will be informed in the status bar that the mating condition is fully constrained.
12. Select **Any** from the **Filter** drop-down list. Choose the **Apply** button and then the **Cancel** button from the **Mating Conditions** dialog box.

Next, you need to change the reference set of the Base so that the datum planes and sketches used to create it are no more displayed.

13. Right-click on the Base and choose **Replace Reference Set > MODEL** from shortcut menu. The assembly, after assembling the Base, is shown in Figure 8-40.



**Figure 8-40** The assembly after assembling the Base




**Note**

*If you do not select **Any** from the **Filter** drop-down list, you may not be allowed to select any object other than the datum planes.*

14. Turn off the display of the assembly datum planes.

### Assembling the Second Component


The second component that you need to assemble is the Screw. This component will be assembled with the Base.

1. Choose the **Add Existing Component** button from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button; the **Part Name** dialog box is displayed. Select the Screw from it and choose the **OK** button.

The Screw is displayed in the **Component Preview** window. You will notice that the **Mate** option is selected by default in the **Positioning** drop-down list.

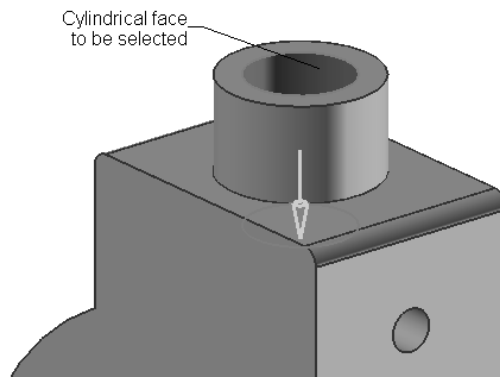
3. Select the **Entire Part** option from the **Reference Set** drop-down list and choose the **OK**

button from the **Add Existing Part** dialog box; the **Mating Conditions** dialog box is displayed.


4. Select the **Face** option from the **Filter** drop-down list.
5. Choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
6. Select the cylindrical face of the Screw, as shown in Figure 8-41. Next, select the cylindrical face from the Base, as shown in Figure 8-42.

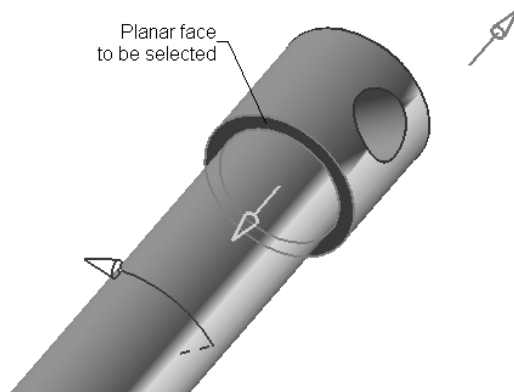


**Figure 8-41** Cylindrical face of the Screw to be selected

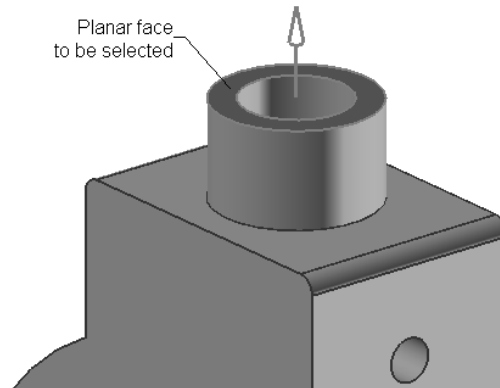


**Figure 8-42** Cylindrical face of the Base to be selected


7. Choose the **Distance** button from the **Mating Type** area. 
8. Select the planar face of the Screw, as shown in Figure 8-43. Next, select the planar face of the Base component, as shown in Figure 8-44.

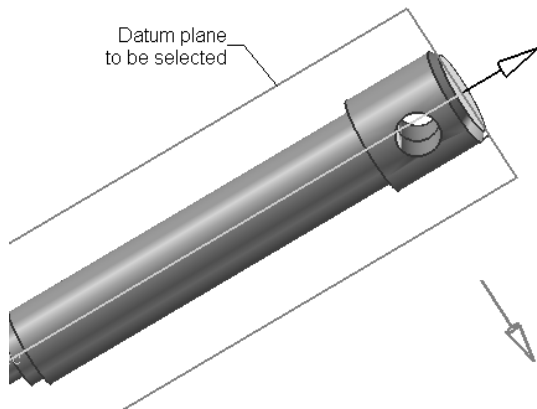


**Figure 8-43** Planar face of the Screw to be selected

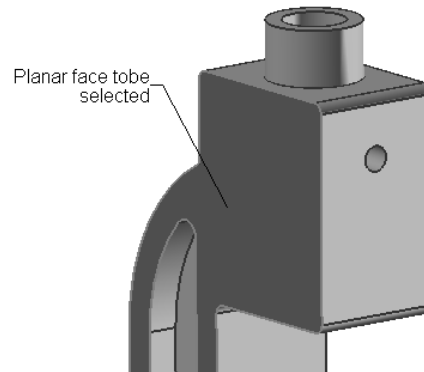


**Figure 8-44** Planar face of the Base to be selected

9. Type the offset value as **35** in the **Distance Expression** edit box.
10. Select the **Any** option from the **Filter** drop-down list.
11. Choose the **Parallel** button from the **Mating Type** area. 
12. Select the front reference plane from the Screw, as shown in Figure 8-45. Select the front face of the Base, as shown in Figure 8-46; you are informed that the mating condition is fully constrained.

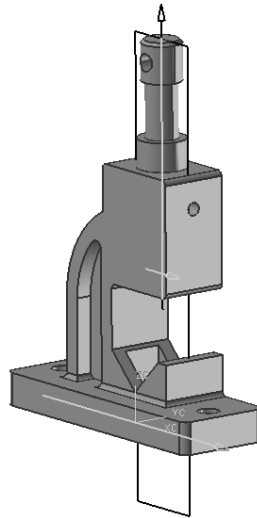


**Figure 8-45** Datum plane of the Screw to be selected



**Figure 8-46** Planar face of the Base to be selected

13. Choose the **Apply** button from the **Mating Conditions** dialog box. The assembly, after assembling the Screw, is shown in Figure 8-47.

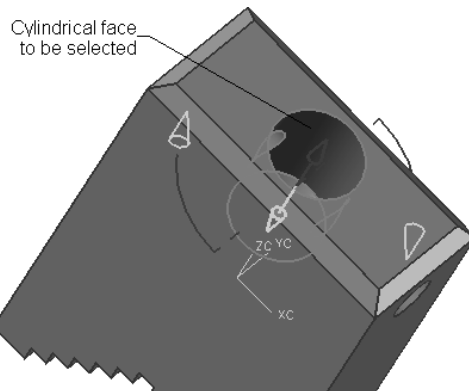


**Figure 8-47** The assembly after assembling the Screw

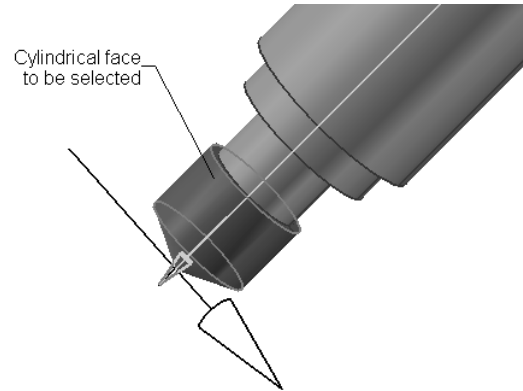
### Assembling the Third Component

Next, you need to assemble the Moveable Jaw with the Screw. It is recommended that before assembling the **Moveable Jaw**, you should hide the Base. This will provide a better display of the Screw.

1. Choose the **Assembly Navigator** button from the resource bar; the **Assembly Navigator** cascade menu is displayed. Click on the check box available on the left of the Base; the display of the Base is turned off.
2. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Select the Moveable Jaw from the **Part Name** dialog box and choose the **OK** button; the Moveable Jaw is displayed in the **Component Preview** window.
4. Select the **MODEL** option from the **Reference Set** drop-down list, if it is not selected. Choose the **OK** button.
5. Choose the **Center** button from the **Mating Type** area and then select the cylindrical face from the Moveable Jaw as the first object, as shown in Figure 8-48. Next, select the bottom cylindrical face from the Screw as the second object, as shown in Figure 8-49.

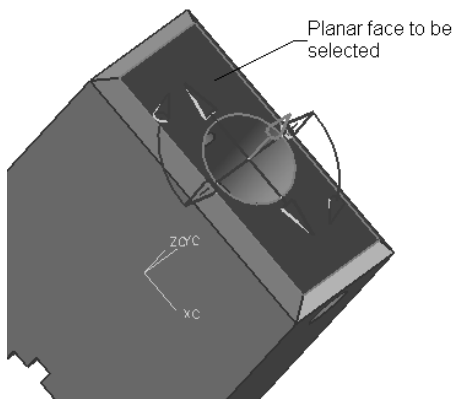


**Figure 8-48** Cylindrical face of the Moveable Jaw to be selected

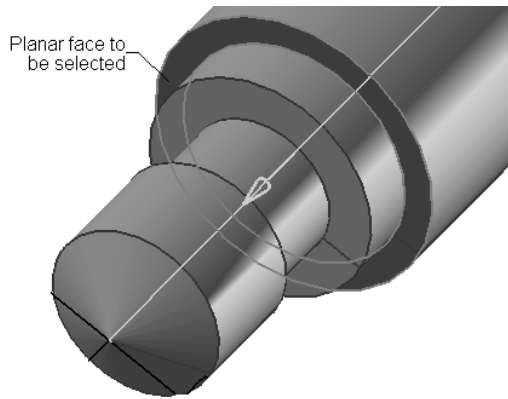


**Figure 8-49** Cylindrical face of the Screw to be selected

6. Choose the **Mate** button from the **Mating Type** area.
7. Select the top face of the Moveable Jaw as the first object, as shown in Figure 8-50. Select the planar face from the Screw as the second object, as shown in Figure 8-51.
8. Choose the **Parallel** button from the **Mating Type** area.

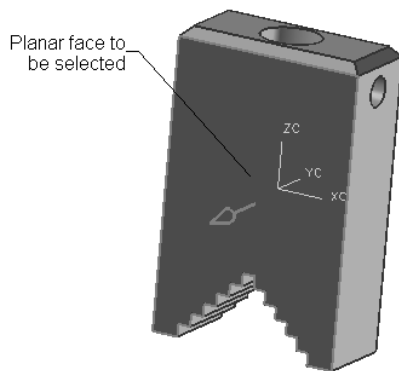


**Figure 8-50** Planar face of the Moveable Jaw to be selected

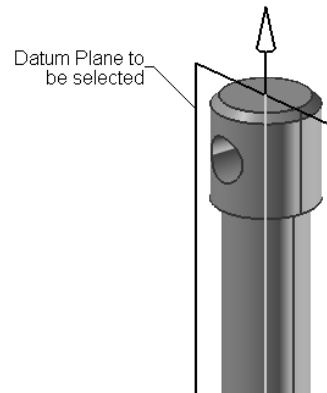


**Figure 8-51** Planar face of the Screw to be selected

9. Select the front face of the Moveable Jaw as the first object, as shown in Figure 8-52. Next, select the front datum plane from the Screw as the second object, as shown in Figure 8-53. For selecting the datum plane, you may need to select the **Any** option from the **Filter** drop-down list.



**Figure 8-52** Planar face of the Moveable Jaw to be selected



**Figure 8-53** Datum plane of the Screw to be selected

After you apply the three assembly constraints between the Moveable Jaw and the Screw, you will be informed that the mating condition is fully constrained. This implies that the Movable Jaw is fully constrained to the Screw.

10. Choose the **Apply** button from the **Mating Conditions** dialog box and then choose the **Cancel** button to exit it.

Now, you need to turn on the display of the Base.

11. Invoke the **Assembly Navigator** cascade menu. Click on the check box on the left of the Base; the display of the Base is turned on.



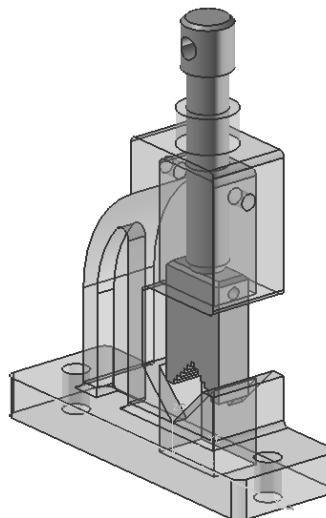
It is recommended that you change the solid texture of the Base to a transparent texture. This is done to display the components that are assembled inside the Base such as the Moveable Jaw and the Screw.

12. Choose **Edit > Object Display** from the menu bar; you are prompted to select the objects to edit. Select the Base and choose the **OK** button from the **Object Display Options**; the **Edit Object Display** dialog box is displayed.

By moving the **Translucency** sliding bar, you can vary the transparency property of the Base from 0 to 100.

13. Move the **Translucency** sliding bar to a value of 75 and choose the **OK** button. If the **Translucency Performance** window is displayed, choose the **OK** button. This window informs you that enabling the translucency will decrease the performance of the graphics.

The assembly, after modifying the display of the Base, is shown in Figure 8-54.




**Figure 8-54** The assembly after making the Base translucent

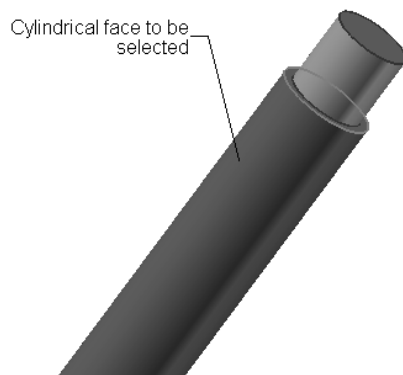
### Assembling the Handle

Next, you need to assemble the **Handle** with the **Screw**.

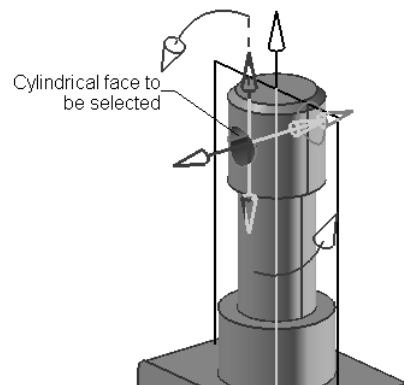
1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed.
2. Choose the **Choose Part File** button. Select the Handle from **Part Name** dialog box and choose the **OK** button.




3. Select the **Entire Part** option from the **Reference Set** drop-down list and choose the **OK** button from the **Add Existing Part** dialog box. The **Mating Conditions** dialog box is displayed.
4. Select the **Face** option from the **Filter** drop-down list of the **Mating Conditions** dialog box.
5. Choose the **Center** button from the **Mating Type** area. 
6. Select the cylindrical face as the first object from the Handle, as shown in Figure 8-55. Next, select the cylindrical face from the Screw as the second object, as shown in Figure 8-56.

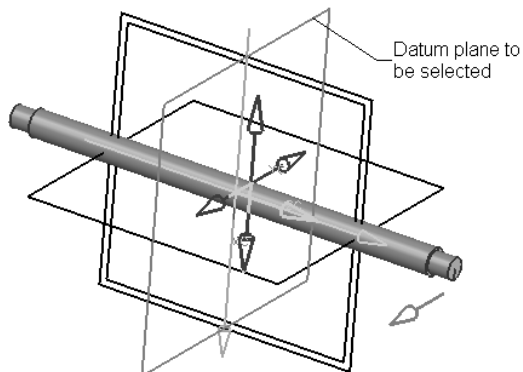


**Figure 8-55** Cylindrical face from the Handle to be selected

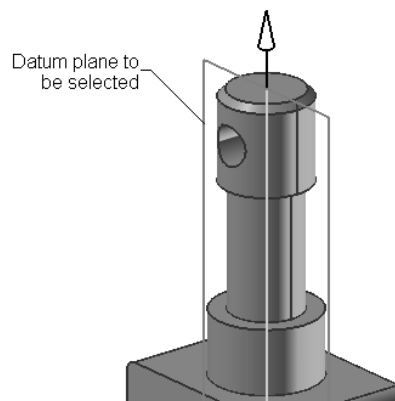


**Figure 8-56** Cylindrical face from the Screw to be selected

7. Choose the **Mate** button from the **Mating Type** area and select the **Datum Plane** option from the **Filter** drop-down list. 
8. Select the right datum plane of the Handle as the first object, as shown in Figure 8-57 and select the right datum plane of the Screw as the second object, as shown in Figure 8-58.



**Figure 8-57** Datum plane of the Handle to be selected

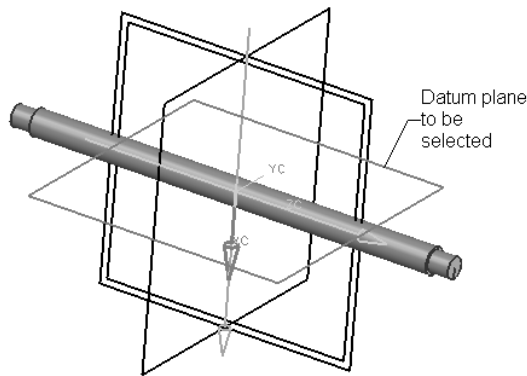


**Figure 8-58** Datum plane of the Screw to be selected

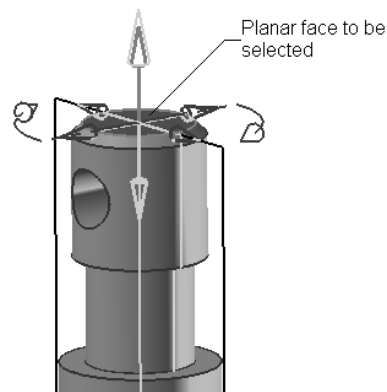
9. Choose the **Parallel** button from the **Mating Type** area.



10. Select the top datum plane of the Handle as the first object, as shown in Figure 8-59. Now, select the **Face** option from the **Filter** drop-down list and select the top face of the Screw as the second object, as shown in Figure 8-60.



**Figure 8-59** Datum plane of the Handle to be selected



**Figure 8-60** Planar face of the Screw to be selected

11. Choose the **Apply** button from the **Mating Conditions** dialog box and then choose the **Cancel** button to exit the dialog box.
12. Choose the **Replace Reference Sets** button from the **Assemblies** toolbar; you are prompted to select the components. Select the Handle and the Screw from the assembly and choose the **OK** button from the **Replace Reference Sets Icon Options**; the **Replace Reference Sets** dialog box is displayed.
13. Select the **MODEL** option from the list box. Choose the **Apply** button and then the **OK** button. The display of the reference sets from the Handle and the Screw are turned off. The resulting assembly, after assembling the Handle, is shown in Figure 8-61.

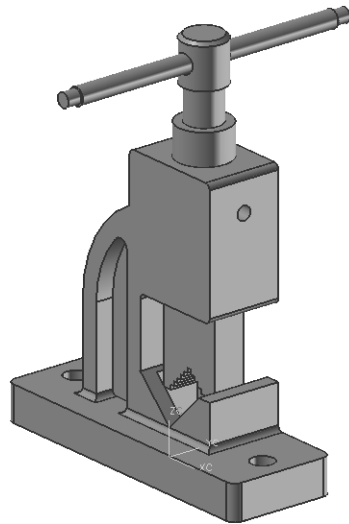


### Assembling the Handle Stop


Assemble the Handle Stop using the following steps:

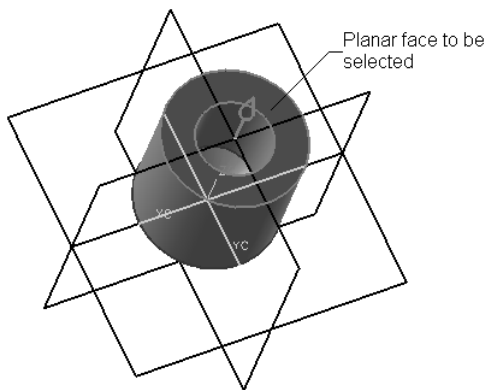
1. Change the reference set of the Handle back to **Entire Part**.
2. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed.
3. Choose the **Choose Part File** button from the **Select Part** dialog box and double-click on the Handle Stop from the **Part Name** dialog box. Choose the **OK** button from the **Add Existing Part** dialog box.



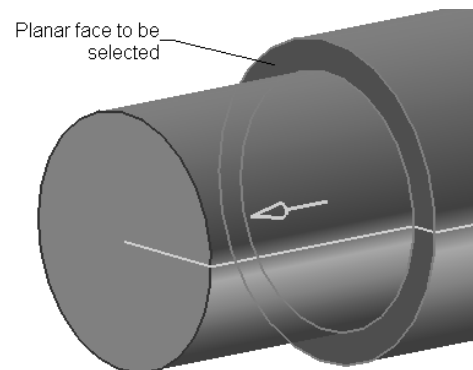


*Figure 8-61 The assembly after assembling the Handle*

4. Choose the **Mate** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
5. Select the planar face from the Handle Stop, as shown in Figure 8-62, and then select the planar face from the Handle, as shown in Figure 8-63.



**Figure 8-62** Planar face to be selected from the Handle Stop

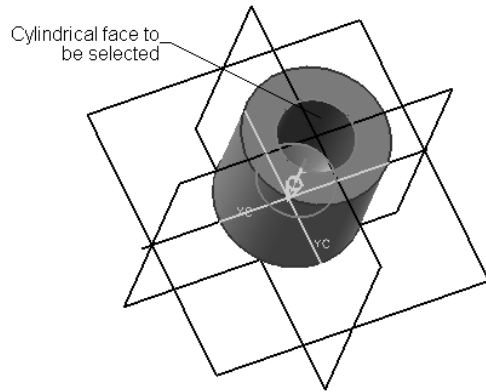


**Figure 8-63** Planar face to be selected from the Handle

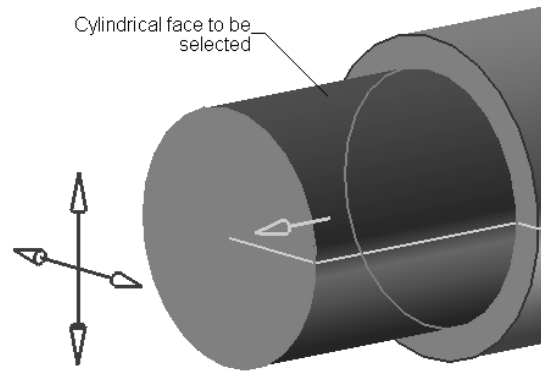
6. Choose the **Center** button from the **Mating Type** area.



7. Select the cylindrical face from the Handle Stop, as shown in Figure 8-64 and select the cylindrical face of the Handle, as shown in Figure 8-65.

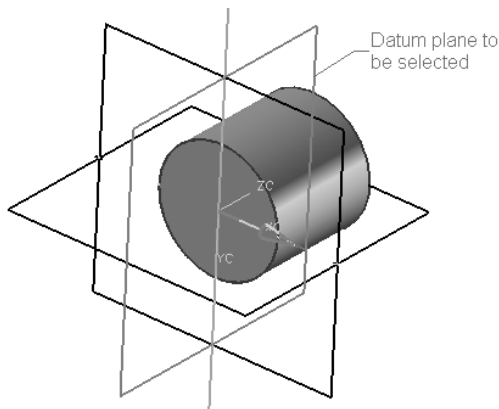


**Figure 8-64** Cylindrical face to be selected from the Handle Stop

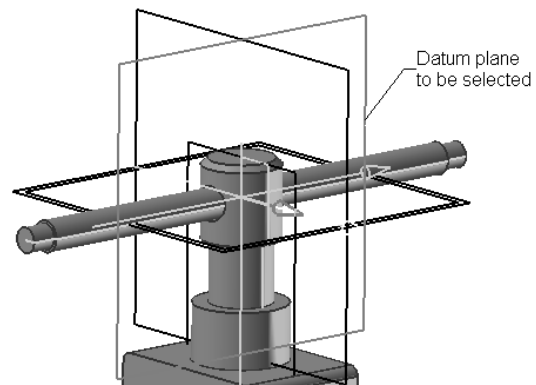


**Figure 8-65** Cylindrical face to be selected from the Handle

8. Choose the **Parallel** button from the **Mating Type** area.
9. Now, choose the **Datum Plane** option from the **Filter** drop-down list.
10. Select the right datum plane from the Handle Stop, as shown in Figure 8-66, and then select the right datum plane of the Handle, as shown in Figure 8-67.

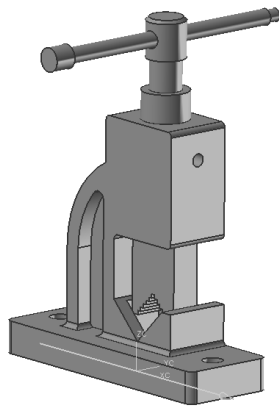


**Figure 8-66** Datum plane to be selected from the Handle Stop

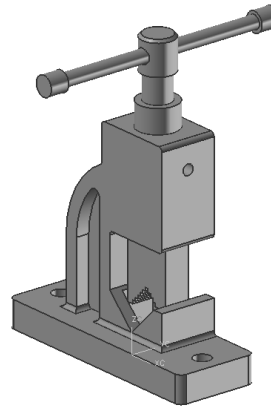


**Figure 8-67** Datum plane to be selected from the Handle

11. Choose the **Apply** button and then the **Cancel** button from the **Mating Conditions** dialog box to exit it.
12. Change the reference set of the Handle Stop to **MODEL** by right-clicking on it. The assembly, after assembling one of the Handle Stop, is shown in Figure 8-68.
13. Similarly, assemble the other instance of the Handle Stop at the other end of the Handle.
14. Change the reference set of all components to **MODEL**. The final Pipe Vice assembly is shown in Figure 8-69.



**Figure 8-68** The assembly after assembling the right side Handle Stop



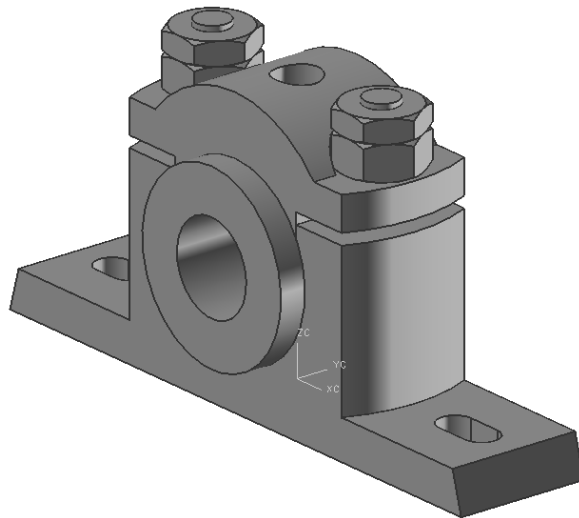
**Figure 8-69** Completed Pipe Vice assembly after assembling the left side Handle Stop

## Tutorial 2

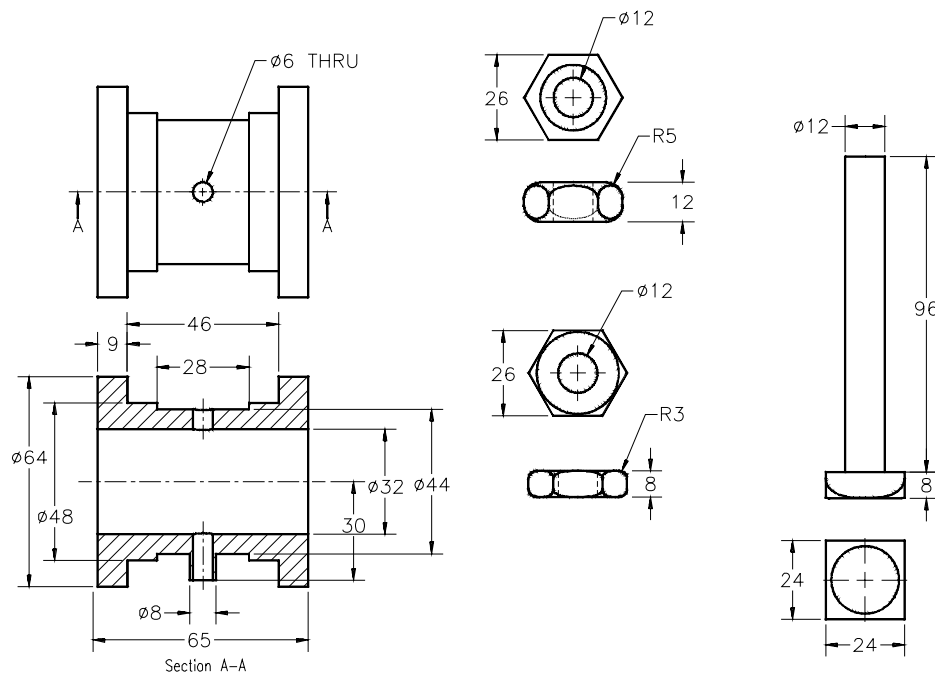
In this tutorial, you will create the components of the Plummer Block assembly shown in Figure 8-70 and then assemble them. The dimensions of the components are given in Figures 8-71 through 8-73. After completing the tutorial, save the file with the name given below.

*NX 3\c08\Plummer Block\Plummer Block.prt*

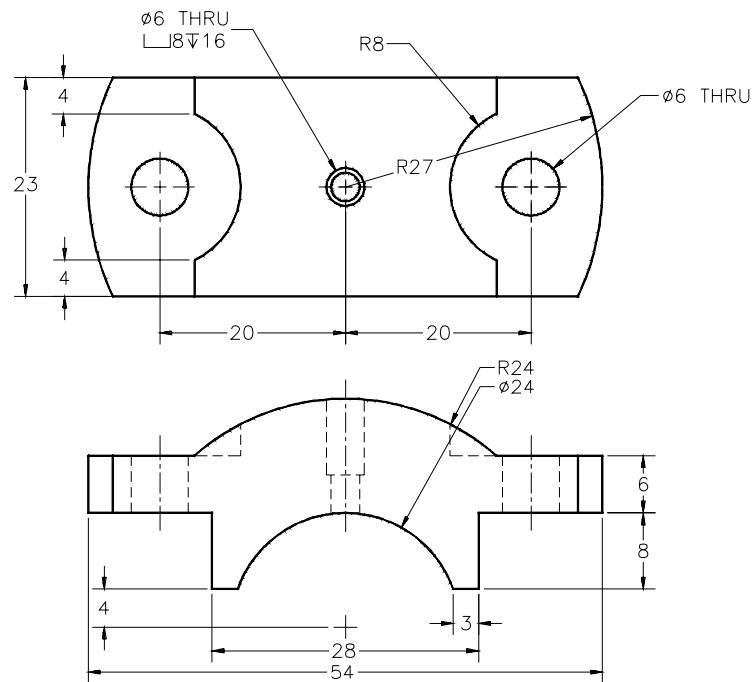
**(Expected time : 2.5 hrs)**



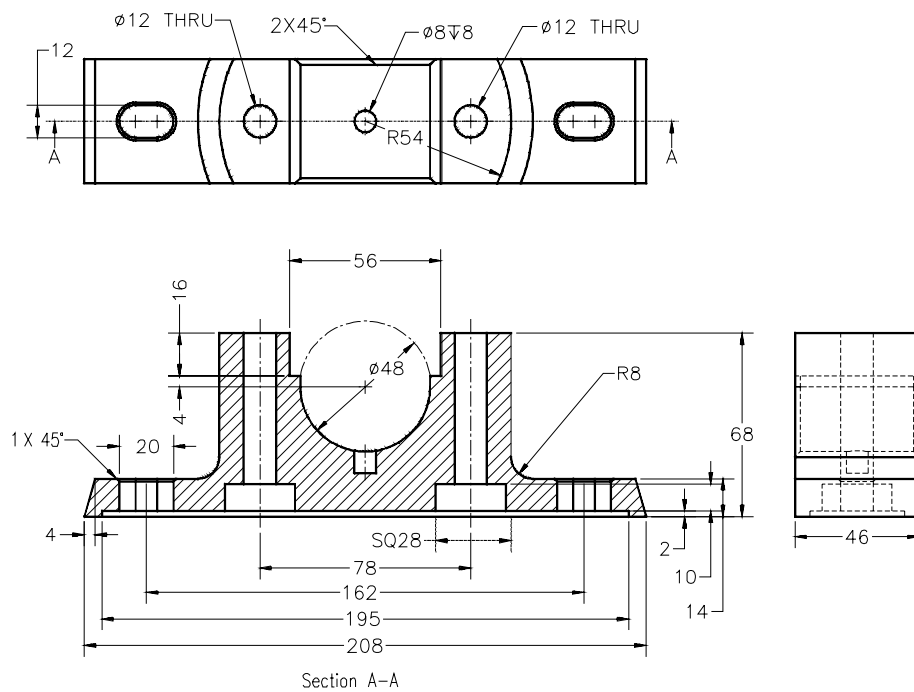
**Figure 8-70** The Plummer Block assembly



**Figure 8-71** Views and dimensions of Brasses, Nut, Lock Nut, and Bolt



**Figure 8-72** Views and dimensions of the Casting



**Figure 8-73** Views and dimensions of the Casting



The following steps are required to complete this tutorial:

- a. Create all the components in individual part files and save them in the |NX 3|c08|Plummer Block folder.
- b. Start a new file in the **Modeling** application. Create the three default planes and then invoke the **Assembly** application.
- c. Insert the Casting in the assembly window and assemble it, refer to Figure 8-74.
- d. Insert the Brasses in the assembly window and apply the required assembly constraints to make it a fully constrained part, refer to Figures 8-75 through 8-81.
- e. Insert the Cap in the assembly window and apply the required assembly constraints, refer to Figures 8-82 through 8-88.
- f. Assemble the Bolts using apply the required assembly constraints, refer to Figures 8-89 to 8-95.
- g. Assemble the Nuts using the required assembly constraints, refer to Figures 8-96 to 8-100.
- h. Assemble the Lock Nuts using the required assembly constraints, refer to Figures 8-101 to 8-106.


### Creating the Assembly Components

1. Create all the components of the Plummer Block assembly as separate part files. Specify the names of the files as mentioned in the drawing views. The files should be saved in the |NX 3|c08|Plummer Block folder.

### Starting the NX session in the Modeling Application




1. Start NX in the **Modeling** application and create the three default datum planes.
2. Choose the **Application > Assemblies** from the menu bar to invoke the **Assembly** application, it is not already chosen.

### Assembling the Casting with the Three Default Datum Planes

1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Double-click on the Casting from the **Part Name** dialog box.

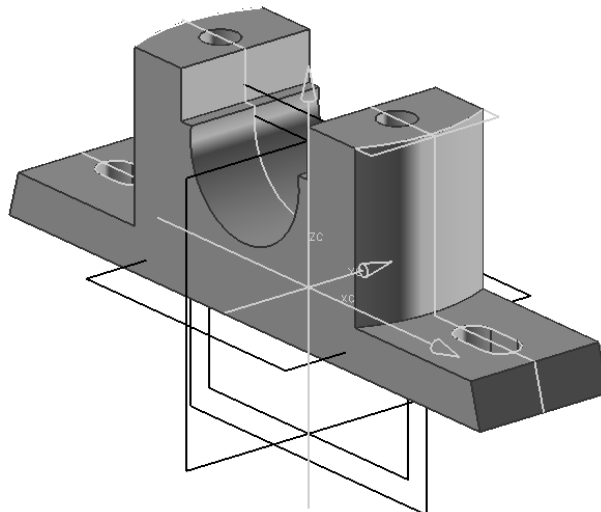
The Casting is displayed in the **Component Preview** window at the lower right corner of the main window; the **Add Existing Component** dialog box is displayed in the upper left corner of the main window.

4. Select the **Entire Part** option from the **Reference Set** drop-down list and the **Mate** option from the **Positioning** drop-down list. Choose the **OK** button from the **Add Existing Part** dialog box; the **Mating Conditions** dialog box is displayed.

5. Select the **Datum Plane** option from the **Filter** drop-down list of the **Mating Conditions** dialog box.
6. Choose the **Align** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
7. Select the right reference datum plane of the Casting from the **Component Preview** window and the right reference datum plane from the assembly window.
8. Choose the **Align** button from the **Mating Type** area. 
9. Select the top reference datum plane of the Casting from the **Component Preview** window and the top reference datum plane from the assembly window.
10. Choose the **Align** button from the **Mating Type** area. 
11. Select the front reference datum plane of the Casting from the **Component Preview** window and the front reference datum plane from the assembly window.

After applying the three assembly constraints, you will be informed that the mating condition is fully constrained.

12. Select **Any** from the **Filter** drop-down list and then choose the **Apply** button from the **Mating Conditions** dialog box. Choose the **Cancel** button to exit the dialog box.
13. Right-click on the Casting and choose **Replace Reference Set > MODEL** from the shortcut menu. The assembly, after assembling the Casting, is shown in Figure 8-74.





**Figure 8-74** Casting assembled with the three default planes

14. Turn off the display of the datum planes.

### Assembling the Second Component

After inserting the Casting, you need to assemble the Brasses as the second component.

1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Double-click on the Brasses from the **Part Name** dialog box; the Brasses is displayed in the **Component Preview** window.
4. Select the **Model** option from the **Reference Set** drop-down list and the **Mate** option from the **Positioning** drop-down list. Choose the **OK** button from the **Add Existing Part** dialog box; the **Mating Conditions** dialog box is displayed.
5. Select the **Face** option from the **Filter** drop-down list.
6. Choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box 
7. Select the cylindrical face from Brasses, as shown in Figure 8-75, and then select the cylindrical face from the Casting, as shown in Figure 8-76.

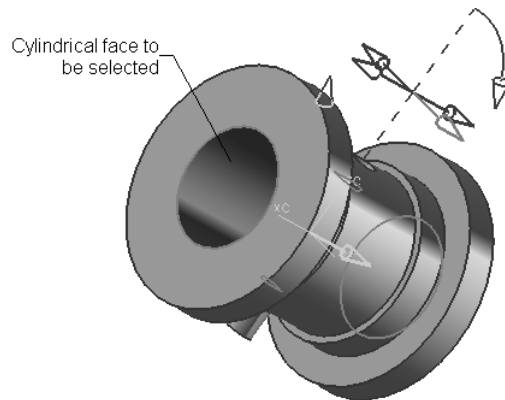


Figure 8-75 Cylindrical face from the Brasses to be selected

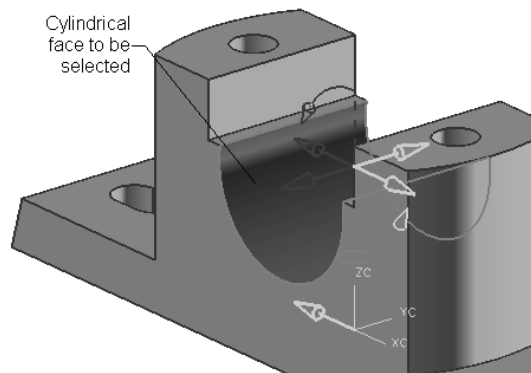

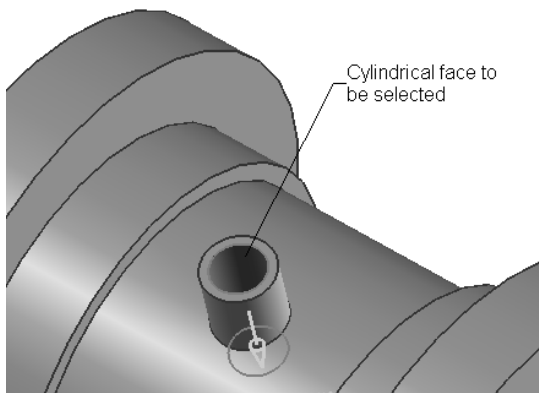
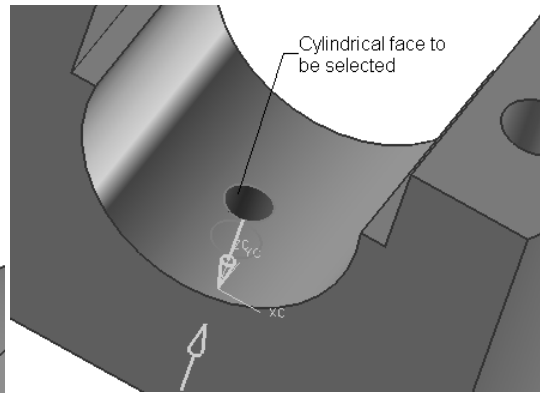


Figure 8-76 Cylindrical face from the Casting to be selected

8. Choose the **Center** button from the **Mating Type** area. 
9. Select the cylindrical face from the Brasses, as shown in Figure 8-77, and then select the cylindrical face from the Casting, as shown in Figure 8-78.



**Figure 8-77** Cylindrical face from the Brasses to be selected

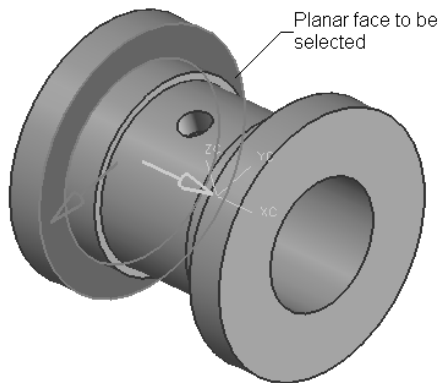


**Figure 8-78** Cylindrical face from the Casting to be selected

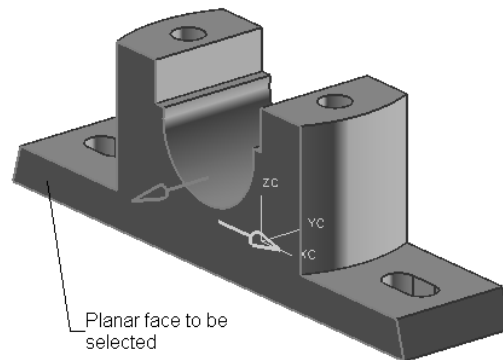
10. Choose the **Mate** button from the **Mating Type** area.



11. Select the planar face of the Brasses, as shown in Figure 8-79, and then select the front face from the Casting, as shown in Figure 8-80.



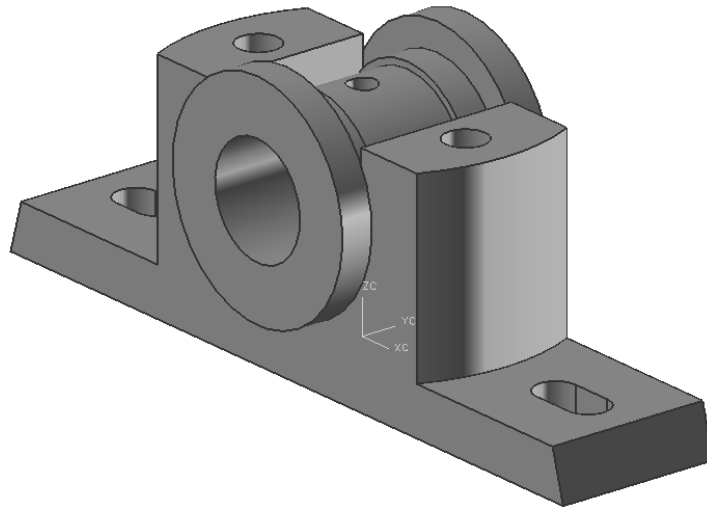
**Figure 8-79** Planar face from the Brasses to be selected



**Figure 8-80** Planar face from the Casting to be selected

After you apply the three constraints, you are informed that the mating condition is fully constrained.




12. Choose the **Apply** button from the **Mating Conditions** dialog box and then the **Cancel** button to exit.
13. Change the reference set of the Brasses to **MODEL** by right-clicking on it. The assembly, after assembling the Brasses is shown in Figure 8-81.

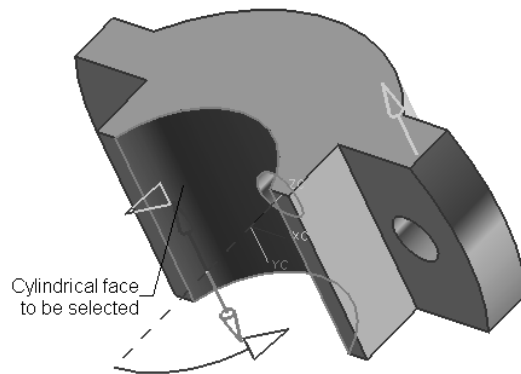


*Figure 8-81 Resulting assembly after assembling the Brasses*

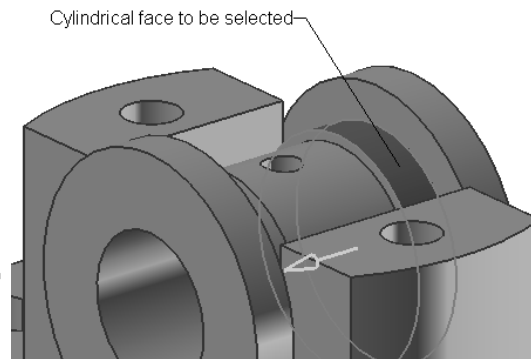
### Assembling the Third Component

Next, you need to assemble the Cap.

1. Choose the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Double-click on the Cap from the **Part Name** dialog box; the Cap is displayed in the **Component Preview** window.
4. Select the **Model** option from the **Reference Set** drop-down list and the **Mate** option from the **Positioning** drop-down list and choose **OK** from this dialog box.
5. Choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
6. Select the cylindrical face from the Cap, as shown in Figure 8-82. Next, select the cylindrical face from the Brasses, as shown in Figure 8-83.
7. Choose the **Align** button from the **Mating Type** area. 

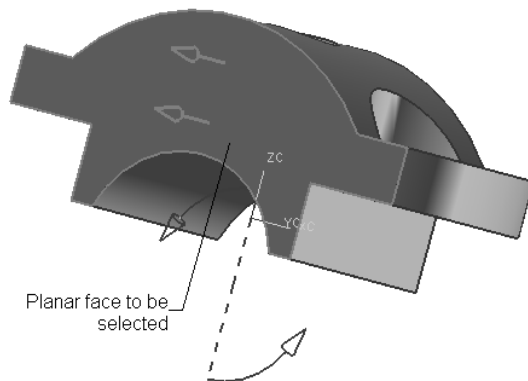


**Figure 8-82** Cylindrical face from the Cap to be selected

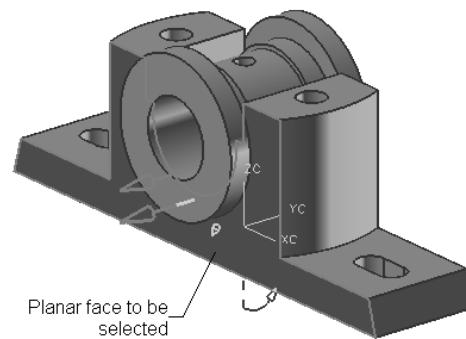


**Figure 8-83** Cylindrical face from the Brasses to be selected

8. Select the front face from the Cap, as shown in Figure 8-84, and then select the front face from the Casting, as shown in Figure 8-85.



**Figure 8-84** Planar face from the Cap to be selected



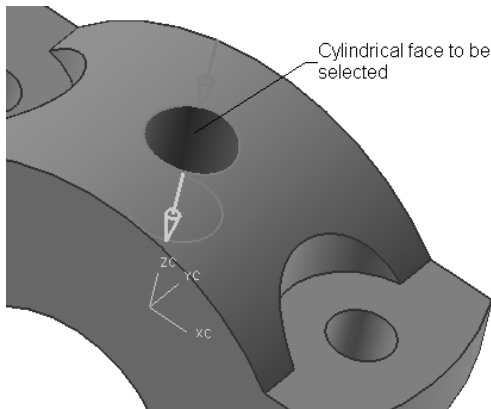
**Figure 8-85** Planar face from the Casting to be selected

9. Choose the **Center** button from the **Mating Type** area.

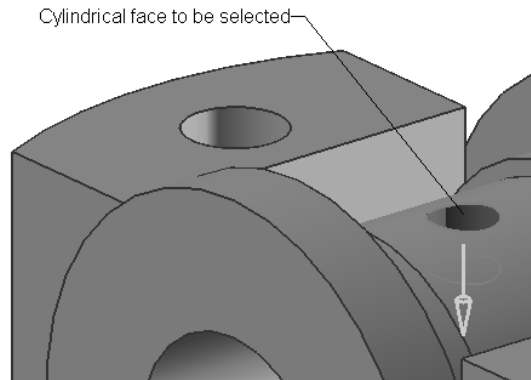


10. Select the cylindrical face from the Cap, as shown in Figure 8-86. Next, select the cylindrical face from the Brasses, as shown in Figure 8-87.

After you apply the three assembly constraints, you will be informed that the mating condition is fully constrained.

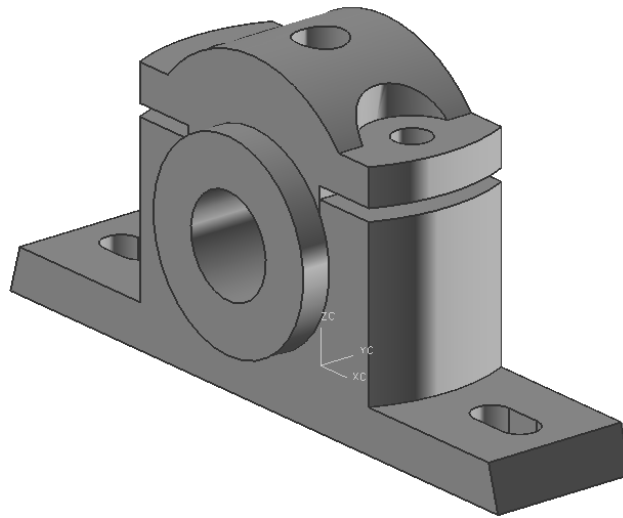


**Figure 8-86** Cylindrical face from the Cap to be selected



**Figure 8-87** Cylindrical face from the Brasses to be selected


11. Choose the **Apply** button and then the **Cancel** button from the **Mating Conditions** dialog box to exit. The resulting assembly, after assembling the Cap, is shown in Figure 8-88.




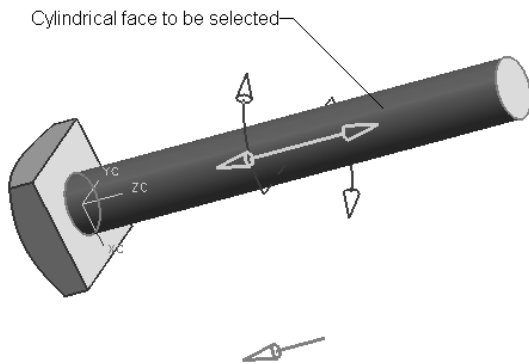
**Figure 8-88** The assembly after assembling the Cap

### Assembling the Bolts

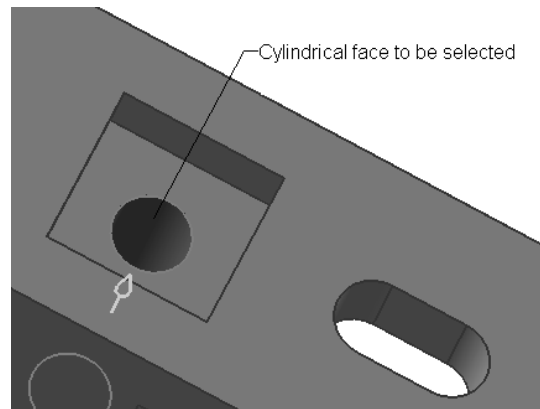
Now, you need to assemble the Bolt as the fourth component.

1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.



3. Double-click on the Bolt from the **Part Name** dialog box; the Bolt is displayed in the **Component Preview** window.
4. Select the **Model** option from the **Reference Set** drop-down list and select the **Mate** option from the **Positioning** drop-down list. Next, choose the **OK** button from the **Add Existing Part** dialog box.
5. Choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
6. Select the cylindrical face from the Bolt, as shown in Figure 8-89. Next, select the cylindrical face from the Casting, as shown in Figure 8-90.



**Figure 8-89** Cylindrical face from the Bolt to be selected



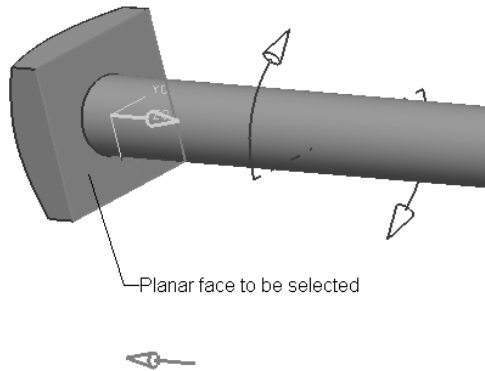
**Figure 8-90** Cylindrical face from the Casting to be selected

7. Choose the **Mate** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
8. Select the planar face of the bolt, as shown in Figure 8-91. Select the planar face from the Casting, as shown in Figure 8-92.
9. Choose the **Parallel** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
10. Select the side planar face from the bolt, as shown in Figure 8-93. Select the planar face from the Casting, as shown in Figure 8-94.

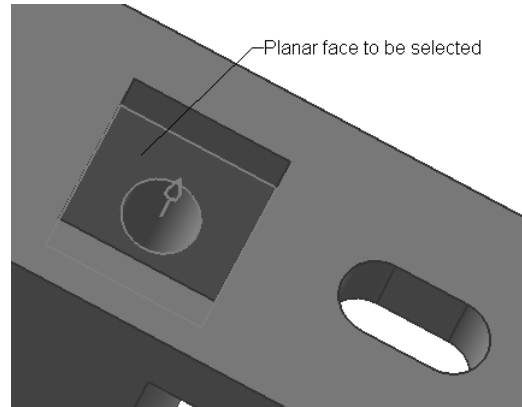
After you apply the three assembly constraints, you will be informed that the mating condition is fully constrained.

11. Choose the **Apply** button and then choose the **Cancel** button from the **Mating Conditions** dialog box to exit.

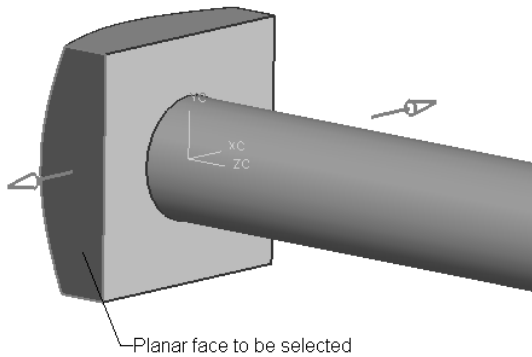




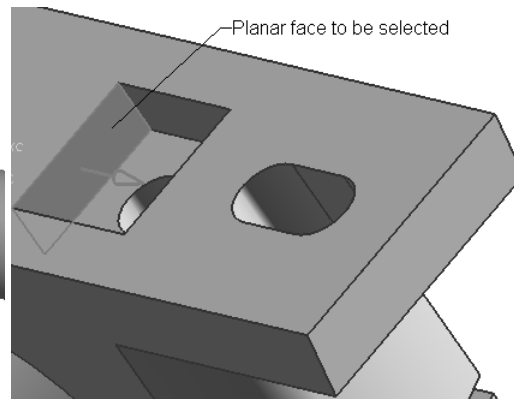
**Figure 8-91** Planar face from the Bolt to be selected



**Figure 8-92** Planar face from the Casting to be selected



**Figure 8-93** Planar face from the Bolt to be selected




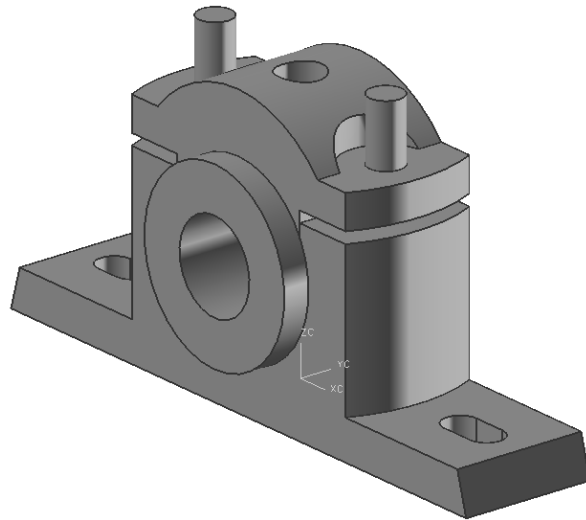
**Figure 8-94** Planar face from the Casting to be selected

12. Similarly, assemble the second instance of the Bolt on the other side. The same three assembly constraints should be applied. The assembly, after assembling the bolts, is shown in Figure 8-95.


### Assembling the Nuts

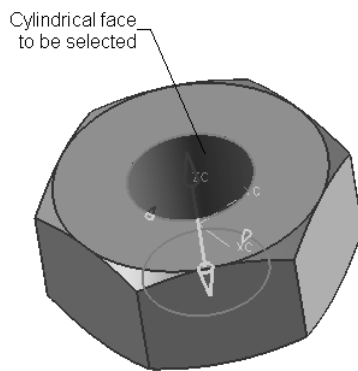
Next, you need to assemble the Nut as the fifth component.

1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed. 
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.

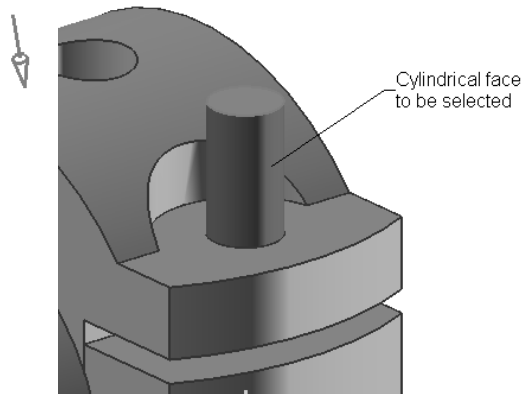


**Figure 8-95** The assembly after assembling the Bolts

3. Double-click on the Nut in the **Part Name** dialog box; the Nut is displayed in the **Component Preview** window.
4. Accept the default options from the **Add Existing Part** dialog box. Next, choose the **Center** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
5. Select the cylindrical face from the Nut, as shown in Figure 8-96. Next, select the cylindrical face from the Bolt, as shown in Figure 8-97.



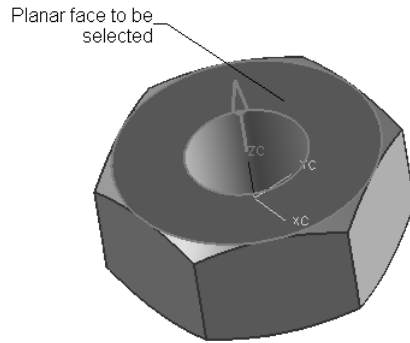
**Figure 8-96** Cylindrical face from the Nut to be selected



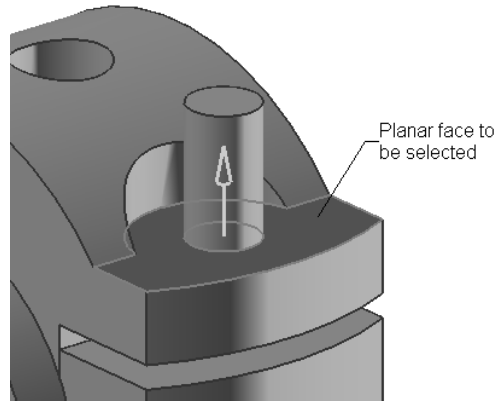
**Figure 8-97** Cylindrical face from Bolt to be selected

6. Choose the **Mate** button from the **Mating Type** area of the **Mating Conditions** dialog box. 

7. Rotate the view of the Nut and select its bottom planar face, as shown in Figure 8-98. Next, select the planar face from the Cap, as shown in Figure 8-99.



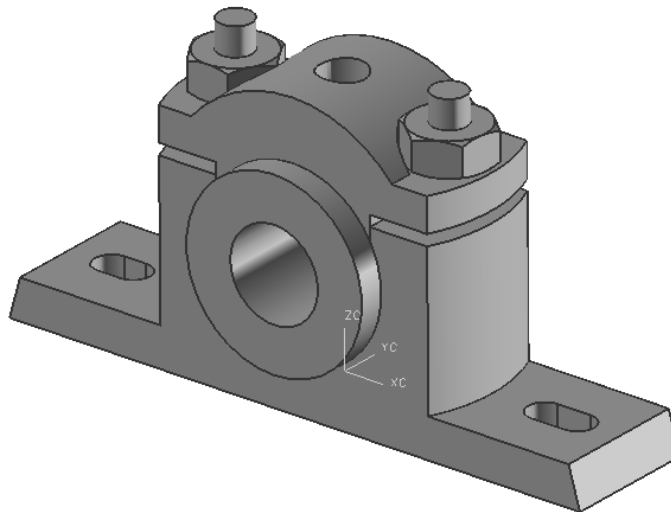
**Figure 8-98** Planar face from the Nut to be selected



**Figure 8-99** Planar face from the Cap to be selected

After applying the two assembly constraints, you will be informed that one degree of freedom is remaining. This implies that the rotational degree of freedom of the assembled Nut is unconstrained.

8. Choose the **Apply** button and then the **Cancel** button from the **Mating Conditions** dialog box to exit.
9. Similarly, assemble the second instance of the Nut on the other side. The assembly, after assembling the Nuts is shown in Figure 8-100.




**Figure 8-100** The assembly after assembling the Nuts

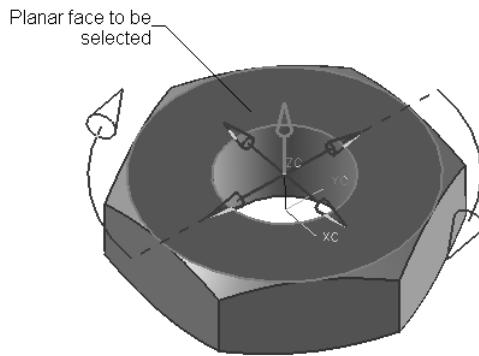


**Tip.** During the process of assembling the Nut in the assembly, you should not constraint its rotational degree of freedom. The Nut should have the rotational degree of freedom, so that it can pass through the bolt threads to be tightened to lock the Cap and Brasses with the Casting.

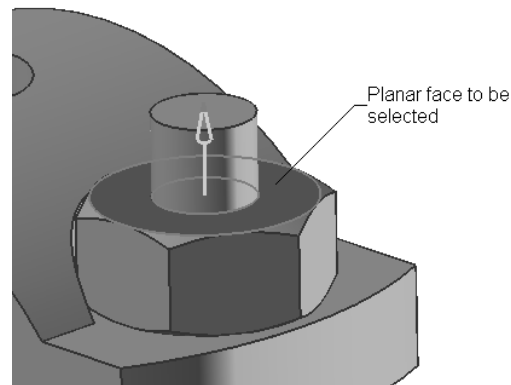
## Assembling the Lock Nut

Next, you need to assemble the Lock Nut as the sixth component.


1. Invoke the **Add Existing Component** tool from the **Assemblies** toolbar; the **Select Part** dialog box is displayed.
2. Choose the **Choose Part File** button from the **Select Part** dialog box; the **Part Name** dialog box is displayed.
3. Double-click on the Lock Nut in the **Part Name** dialog box; the Lock Nut is displayed in the **Component Preview** window.
4. Accept the default options from the **Add Existing Part** dialog box and choose **OK**.  
Next, choose the **Mate** button from the **Mating Type** area of the **Mating Conditions** dialog box. 
5. Select the planar face from the Lock Nut, as shown in Figure 8-101. Next, select the planar face from the Nut, as shown in Figure 8-102.

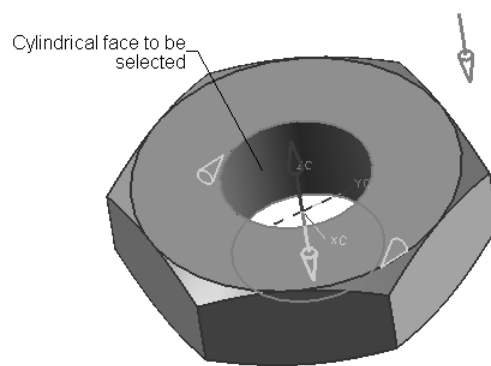


**Figure 8-101** Planar face from the Lock Nut to be selected

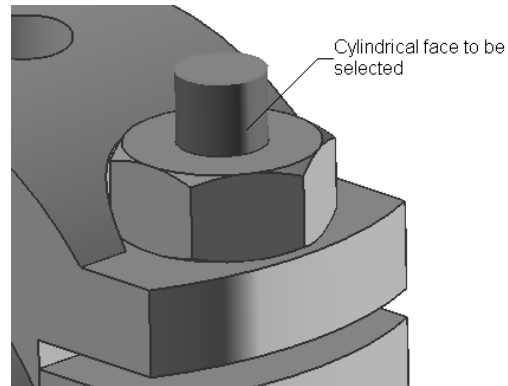


**Figure 8-102** Planar face from the Nut to be selected

6. Choose the **Center** button from the **Mating Type** area. 
7. Select the cylindrical face from the Lock Nut, as shown in Figure 8-103. Next, select the cylindrical face from the Bolt, as shown in Figure 8-104.



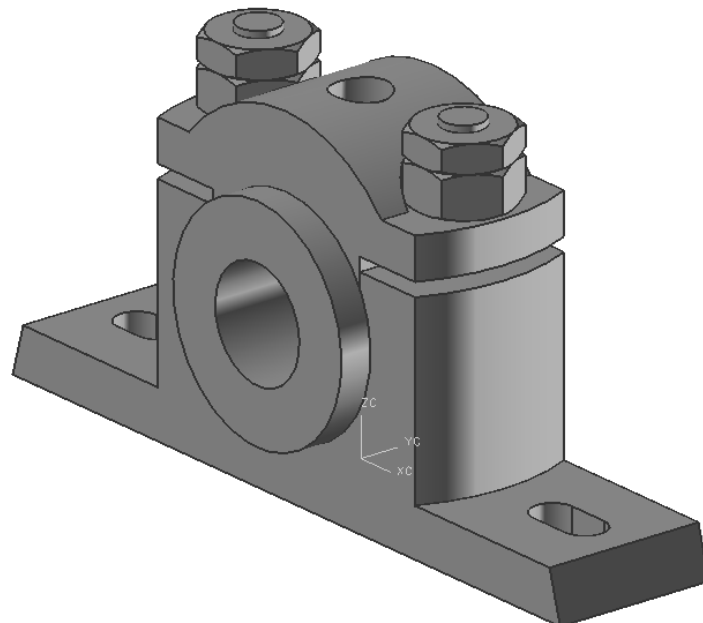
**Figure 8-103** Cylindrical face from the Lock Nut to be selected



**Figure 8-104** Cylindrical face from the Bolt to be selected

After you apply the two assembly constraints, you will be informed that one degree of freedom is remaining. This implies that the rotational degree of freedom is left unconstrained.

8. Choose the **Apply** button and then the **Cancel** button to exit the dialog box.
9. Similarly, assemble the second instance of the Lock Nut on the other side. The completed Plummer Block assembly is shown in Figure 8-105.



**Figure 8-105** The final Plummer Block assembly

### Self-Evaluation Test

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

1. In the top-down assembly design approach, all components are created within the same assembly file. (T/F)
2. Using the **Parallel** constraint, you can maintain an offset value between the two faces. (T/F)
3. You cannot create a component in the **Assembly** application. (T/F)
4. The original component of the instance assembled is known as the **Component Part**. (T/F)
5. The **Replace Reference Set** tool is used to turn on or off the reference set of the component assembled. (T/F)
6. For assembling the base component, it is mandatory to select the **Absolute** option from the **Positioning** edit box in the **Add Existing Part** dialog box. (T/F)
7. The \_\_\_\_\_ is the file name extension for the assembly files.
8. The \_\_\_\_\_ assembly constraint is used to maintain an offset value between the reference faces.
9. The **Create Component Array** tool is used to create \_\_\_\_\_ and \_\_\_\_\_ arrays.
10. The component in the **Component Preview** window is known as \_\_\_\_\_.

### Review Questions

Answer the following questions:

1. Which button do you have to choose to invoke the **Mirror Assembly Wizard**?
  - a. **Mirror Assembly**
  - b. **Create Component Array**
  - c. **Replace Reference Set**
  - d. None of the above

2. Which button is used to make two faces coplanar?
  - a. **Mate**
  - b. **Parallel**
  - c. **Angle**
  - d. None of the above
3. Which button is used to align the center axis of the two cylindrical components?
  - a. **Align**
  - b. **Perpendicular**
  - c. **Center**
  - d. None of the above
4. Which button is used to open the **Mating Conditions** dialog box directly?
  - a. **Mate Component**
  - b. **Replace Reference Set**
  - c. **Add Existing Part**
  - d. None of the above
5. Assembly constraints once created cannot be edited. (T/F)
6. After placing the component in the assembly, a further modification of the component is not possible in the assembly application. (T/F)
7. The **Make Workpart** tool is used to create a component in the assembly application. (T/F)
8. To align the axes of two cylindrical components, you can select their cylindrical faces. (T/F)
9. Reference sets are readily available in the assembly file when a new assembly file is opened. (T/F)
10. The **Reposition Component** tool is used to reorient the component in 3D space irrespective of the assembly constraints given. (T/F)

## Exercise

### Exercise 1

Create the components of the Butterfly Valve Assembly and then assemble them, as shown in Figures 8-106. The dimensions of the components are shown in Figures 8-107 through 8-111. Create the folder with the name \c08\Butterfly Valve Assembly and save all the component files and the assembly file in it. Assume the missing dimensions.

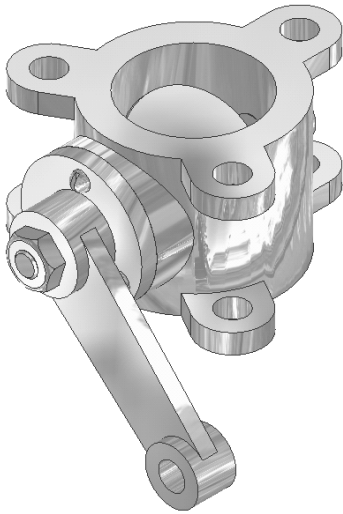


Figure 8-106 Final Butterfly Valve assembly

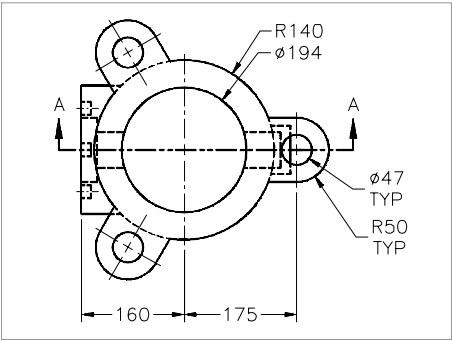


Figure 8-107a Top View of the Body

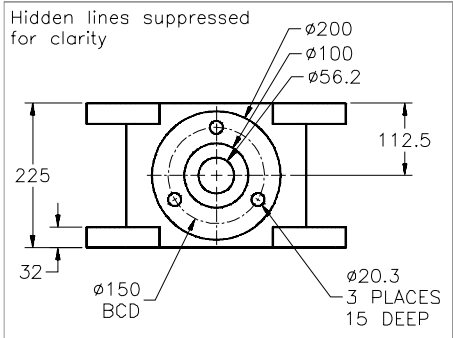


Figure 8-107b Left side view of the Body

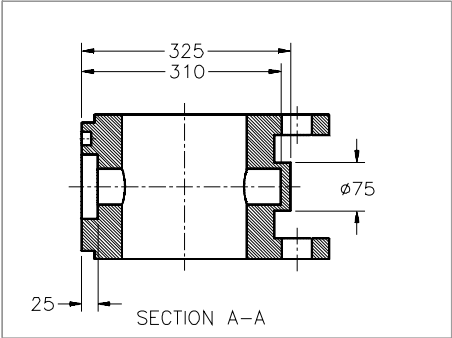


Figure 8-107c Sectional front view of the Body



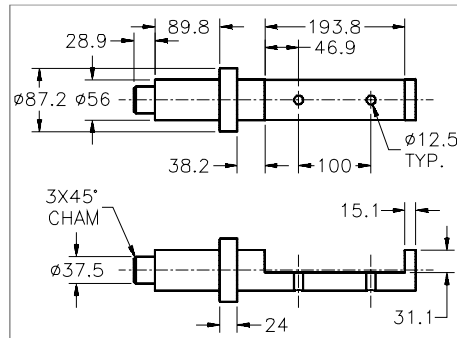


Figure 8-108 Dimensions of the Shaft

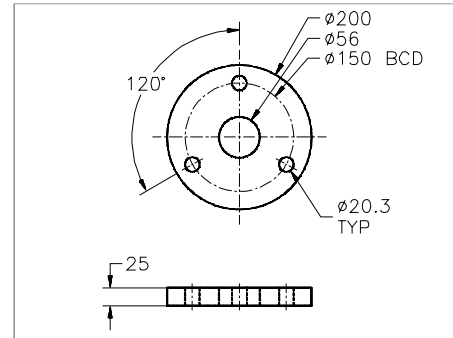


Figure 8-109 Dimensions of the Retainer

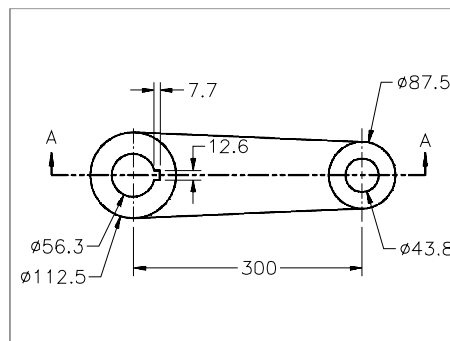


Figure 8-110a Top view of the Arm

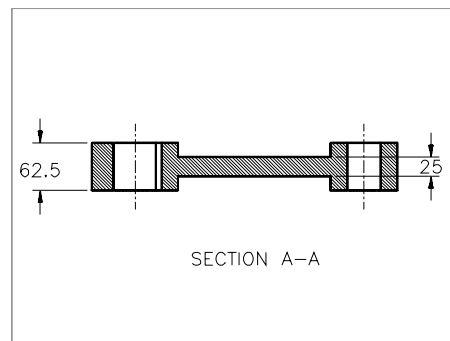
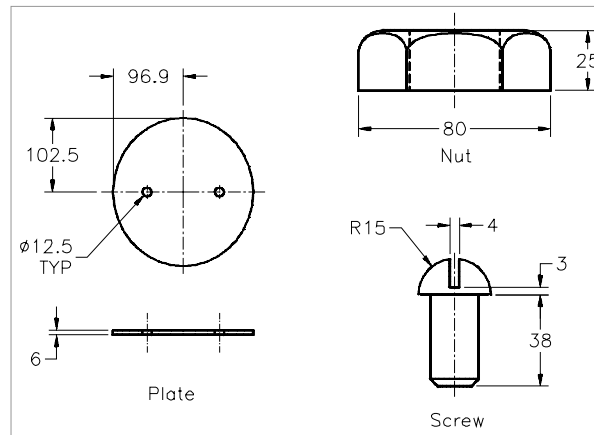


Figure 8-110b Sectional front view of the Arm



*Figure 8-111 Dimensions of the Plate, Nut, and Screw*

**Answers to Self-Evaluation Test**

1. T, 2. F, 3. F, 4. T, 5. T, 6. F, 7. \*.prt, 8. Distance, 9. Linear and Circular, 10. Displayed part