



Chapter 10

Assembly Modeling-I

Learning Objectives

After completing this chapter you will be able to:

- *Create bottom-up assemblies.*
- *Add mates to assemblies.*
- *Create top-down assemblies.*
- *Move individual components.*
- *Rotate individual components.*

Evaluation chapter. Logon to www.cadcam.com for more details

ASSEMBLY MODELING

An assembly design is a design consisting of two or more components assembled together at their respective work positions using the parametric relations. In SolidWorks, these relations are called mates. These mates allow you to constrain the degrees of freedom of the components at their respective work positions. To proceed to the **Assembly** mode of SolidWorks, invoke the **New SolidWorks Document** dialog box and choose the **Assembly** button as shown in Figure 10-1.

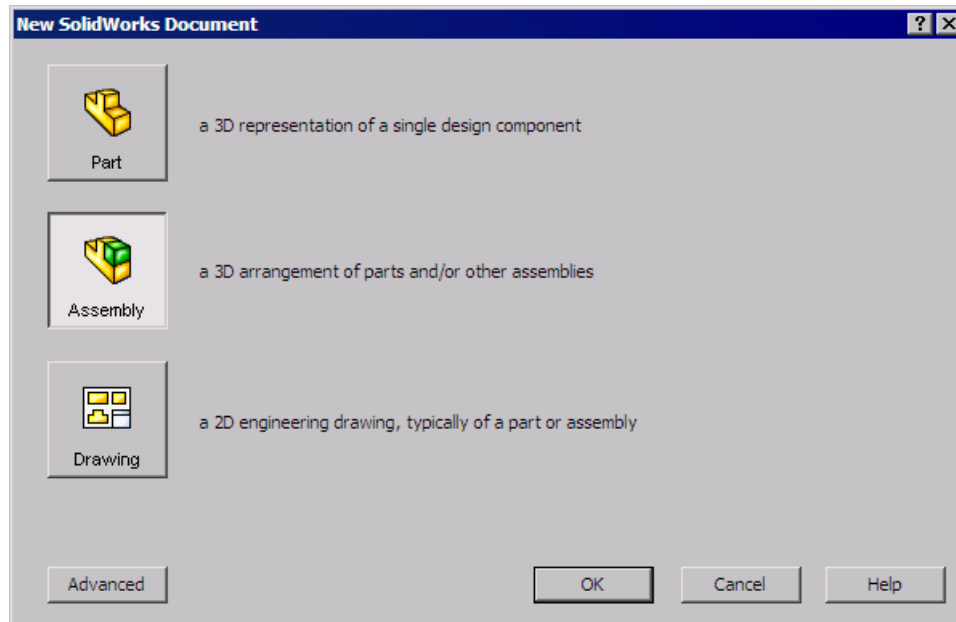


Figure 10-1 The New SolidWorks Document dialog box

Choose the **OK** button to create a new assembly document; new SolidWorks document will be started in the **Assembly** mode and the **Insert Component PropertyManager** will be invoked, as shown in Figure 10-2.

Types of Assembly Design Approach

In SolidWorks, the assemblies are created using two types of design approaches: bottom-up approach and the top-down approach. These design approaches are discussed next.

Bottom-up Assembly Design Approach

The bottom-up assembly design approach is the traditional and the most widely preferred approach of assembly design. In this assembly design approach, all components are created as separate part documents and are placed and referenced in the assembly as external components. In this type of approach, the components are created in the **Part** mode and saved as the *.sldprt* documents. After creating and saving all components of the assembly, you will start a new assembly document (*.sldasm*) and insert the components using the tools provided in the **Assembly** mode. After inserting you will assemble the components using the assembly mates.

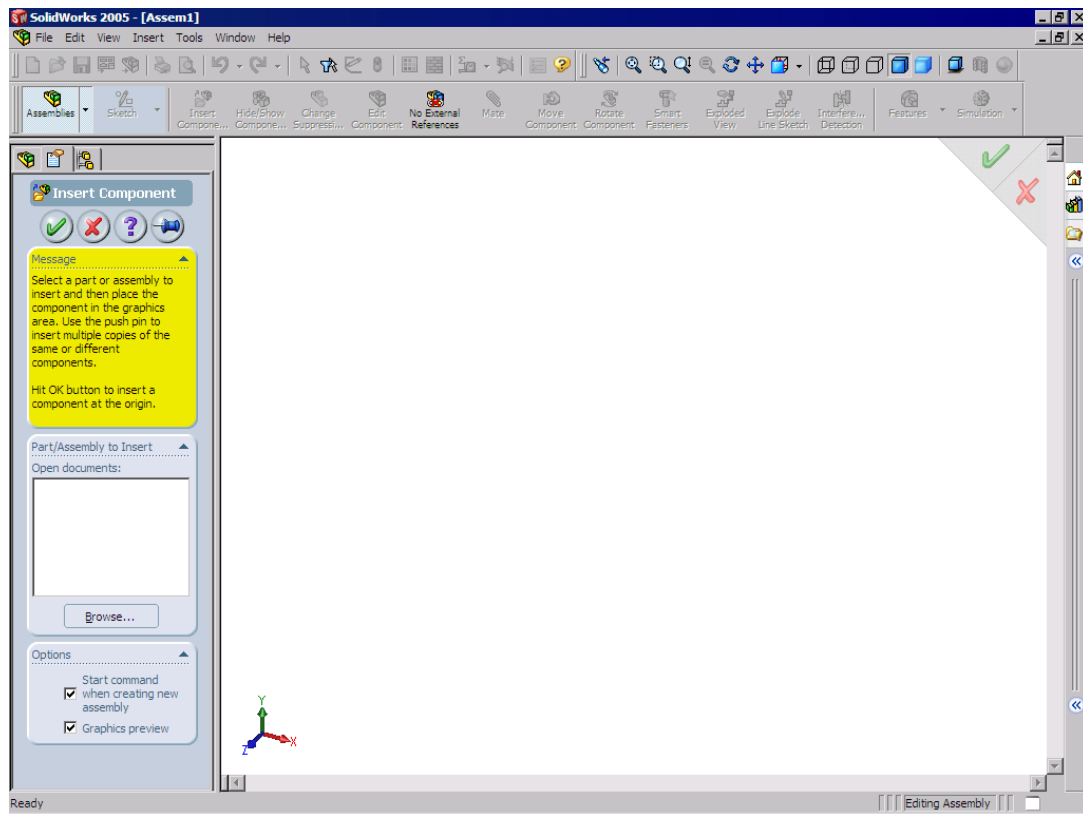


Figure 10-2 Screen display of SolidWorks in the **Assembly** mode

The main advantage of this assembly design approach is that because there is only a single part in the current file, the view of the part is not restricted. Hence, this approach allows you to pay more attention and focus more on the complex individual features. This approach is preferred while handling large assemblies or the assemblies with complex parts.

Top-down Assembly Design Approach

In the top-down assembly design approach, the components are created in the same assembly document, but saved as separate part files. Therefore, the top-down assembly design approach is entirely different from the bottom-up design approach. In this approach you will start your work in the assembly document and the geometry of one part will help you define the geometry of the other.



Note

You can also create an assembly with a combination of the bottom-up and the top-down assembly approaches.

CREATING BOTTOM-UP ASSEMBLIES

As mentioned earlier, the bottom-up assemblies are the assemblies in which the components are created as separate part documents in the **Part** mode. After creating the components, they are

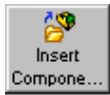
inserted in the assembly and then assembled using the assembly mates. For starting an assembly design with this approach, you first need to insert the components in the assembly. It is recommended that the first component should be placed at the origin of the assembly document. By doing this the default planes of the assembly and the part will coincide and the component will be in the same orientation as it was in the **Part** mode. When you place the first component in the assembly, that component will be fixed at its placement position. The techniques used to place the components in the assembly file are discussed next.

Placing Components in the Assembly Document

In SolidWorks, there are various options to place the components in the assembly. These options are discussed next.

Placing Components Using the Insert Component PropertyManager

CommandManager:	Assemblies > Insert Components
Menu:	Insert > Component > Existing Part/Assembly
Toolbar:	Assembly > Insert Components



When you start a new SolidWorks document in the **Assembly** mode, the **Insert Component PropertyManager** will be displayed, as shown in Figure 10-3. The **Message** rollout available in the **Insert Component PropertyManager** prompts you to select a part or an assembly and then place the component in the graphics area. When you choose the **Browse** button available in the **Part/Assembly to Insert** rollout, the **Open** dialog box will be displayed. Browse the location where the component is saved and then select the component and choose the **Open** button. The cursor will be replaced by the component cursor and graphic preview of the component will be also displayed. The name of the selected component will be displayed in the **Open documents** selection area of the **Part/Assembly to Insert** rollout. It is recommended that the origin of the first component should be aligned with the assembly origin. To place the component origin on the assembly origin choose the **OK** button from the **Insert Component PropertyManager**.

To place the second component, invoke the **Insert Component PropertyManager** again and choose the **Browse** button from the **Part/Assembly to Insert** rollout. Select the component from the **Open** dialog box; the cursor will be replaced by the component cursor and the preview of the component will also be displayed in the drawing area. Select a point anywhere in the drawing area to place the second component.

When you select a component to insert in the assembly, the **Thumbnail Preview** rollout will be displayed in the **Insert Component PropertyManager**. You can invoke this rollout to view the thumbnail preview of the selected component.

The **Start command when creating new assembly** check box available in the **Options** rollout is selected by default and is used to invoke the **Insert Component PropertyManager** automatically when you start a new SolidWorks assembly document. The **Graphics preview** check box available in the **Options** rollout is selected by default and is used to display the graphic preview of the component selected to insert.

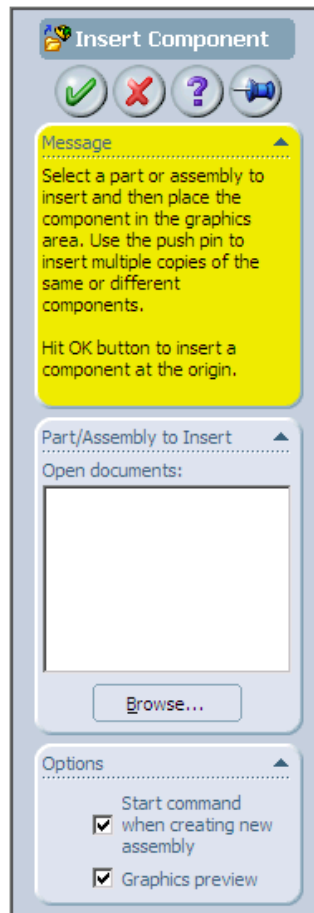


Figure 10-3 The *Insert Component PropertyManager*



Tip. If you need to place multiple components or multiple instances of the same component, choose the **Keep Visible** button from the **Insert Component PropertyManager** and select the placement points in the drawing area to place the multiple components. The **Keep Visible** button is available on the right of the **Help** button on top of the **Message** rollout.

If some part documents are opened in the SolidWorks window and you start an assembly document, those parts will be listed in the **Open documents** selection area. You can directly select the names of the parts from the **Open documents** selection area and place them in the drawing area.

Starting an Assembly From Within the Part Document

Menu:	File > Make Assembly from Part
Toolbar:	Standard > Make Assembly from Part/Assembly



Tip. When you insert a component in the assembly, it will be displayed in the **FeatureManager Design Tree**. The convention of naming the first component is **(f) Name of Component <1>**. In this convention **(f)** implies that the component is fixed. You cannot move a fixed component. You will learn more about fixed and floating components later in this chapter. After this the name of the component is displayed. After the name of the component, **<1>** symbol is displayed. This implies the serial number of the same component in the entire assembly.

The **(-)** symbol implies that the component is floating and is under defined. You need to apply the required mates to the component to fully define it. You will learn more about assembly mates later in this chapter.

The **(+)** symbol implies that the component is over defined.

If no symbol appears before the name of the component, the component will be fully defined.



You can also start an assembly document from within the part document. If the part document of the base component of the assembly is opened, choose the **Make Assembly from Part/Assembly** button from the **Standard** toolbar or choose **File > Make Assembly from Part** from the menu bar. You can also use CTRL+A on the keyboard as the shortcut key. If the **New SolidWorks Document** dialog box is invoked, choose the **OK** button from this dialog box. An assembly document will be started and the **Insert Component PropertyManager** will be invoked. Choose the **OK** button from this **PropertyManager** to place the component at the origin.



Note

If the last time you invoked the **New SolidWorks Document** dialog box in the novice mode, you just need to select the **OK** button from the **New SolidWorks Document** dialog box if it is displayed while creating an assembly from a part document. If you used the **New SolidWorks Document** in the advanced mode last time, you first need to select the assembly template and then choose the **OK** button. When you choose the **Make Assembly from Part/Assembly** button and if a **SolidWorks** warning box is displayed, choose **No** from the warning box. The **New SolidWorks Document** dialog box will be displayed in the advanced mode.

Placing Components Using the Opened Document Window

Another most widely used method of placing components in the assembly is the use of currently opened part or assembly documents. For example, the assembly that you are going to create consists of three components. Open the part document of the part that you want to insert and then start a new assembly document. Close the **Insert Component PropertyManager**. Now, choose **Window > Tile Horizontally** or **Tile Vertically** from the menu bar. All SolidWorks document windows will be tiled vertically or horizontally, depending on the option selected. You first need to place the first component in the assembly and this needs to be placed at the origin of the assembly. If the origin is not displayed by default, choose **View > Origins** from the menu bar. Now, move the cursor on the name of the first component in the **FeatureManager Design Tree** of the window of that component. Press and hold the left mouse button down on the name and then drag the cursor to the assembly origin in the assembly window. When the



Tip. Only the information about the mates is stored in the assembly file. The feature information of parts is stored in the individual part files. Therefore, the size of the assembly file is small.

It is recommended that all parts of an assembly should be saved in the same folder in which the assembly is saved. This is because if the location of the parts is changed, the component will not be displayed in the assembly and assembly file will show errors.

coincident symbol appears below the component cursor, release the left mouse button to place the component in the assembly. Similarly, place the other components in the assembly. Figure 10-4 displays placing the first component from an opened document window to the assembly document window. If another existing assembly document is opened, you can also drag and drop the part from that assembly document.

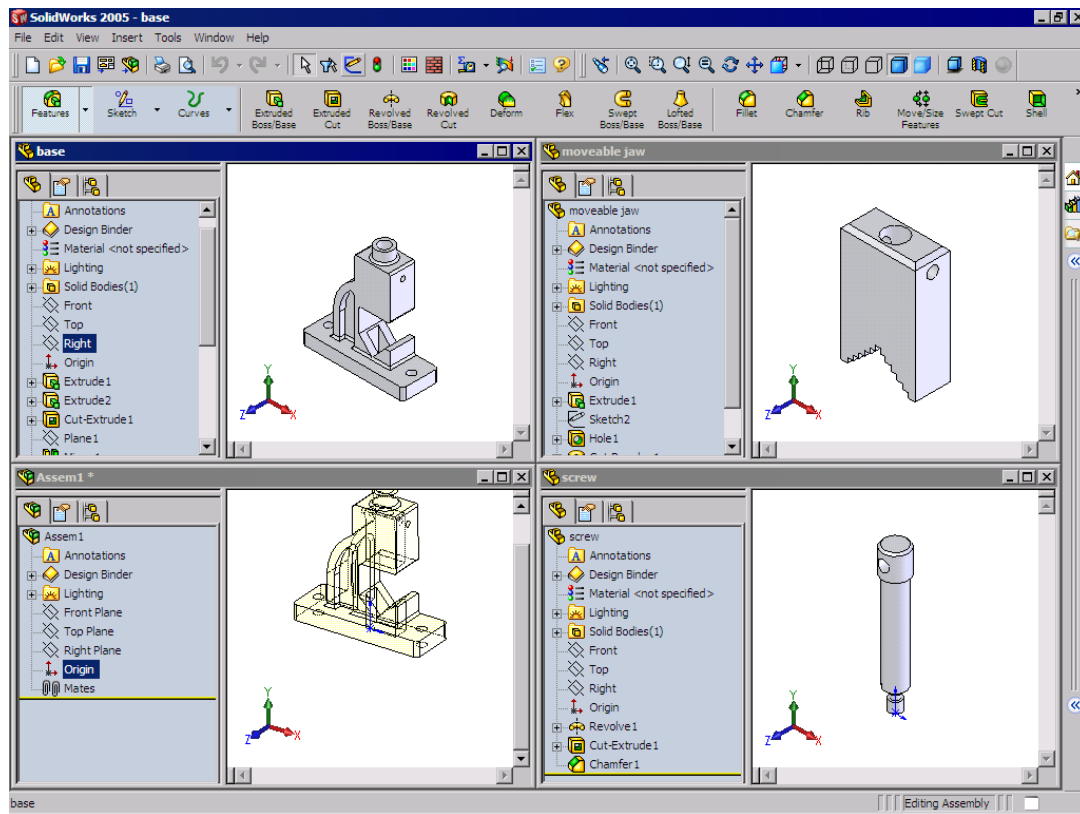


Figure 10-4 Placing a component in the assembly file from an existing window

Placing Components by Dragging From Windows Explorer

You can also place components in the assembly document by dragging them from Windows Explorer. Open Windows Explorer and browse the location where the part documents are saved. Tile the Windows Explorer window and the SolidWorks window such that you can view both

windows. Move the cursor on the icon of the part document in Windows Explorer. Press and hold the left mouse button down and drag the cursor to the assembly document window. Release the left mouse button at the origin of the assembly document to coincide the origin of the part with that of the assembly document. Figure 10-5 shows the part being dropped in the assembly window.

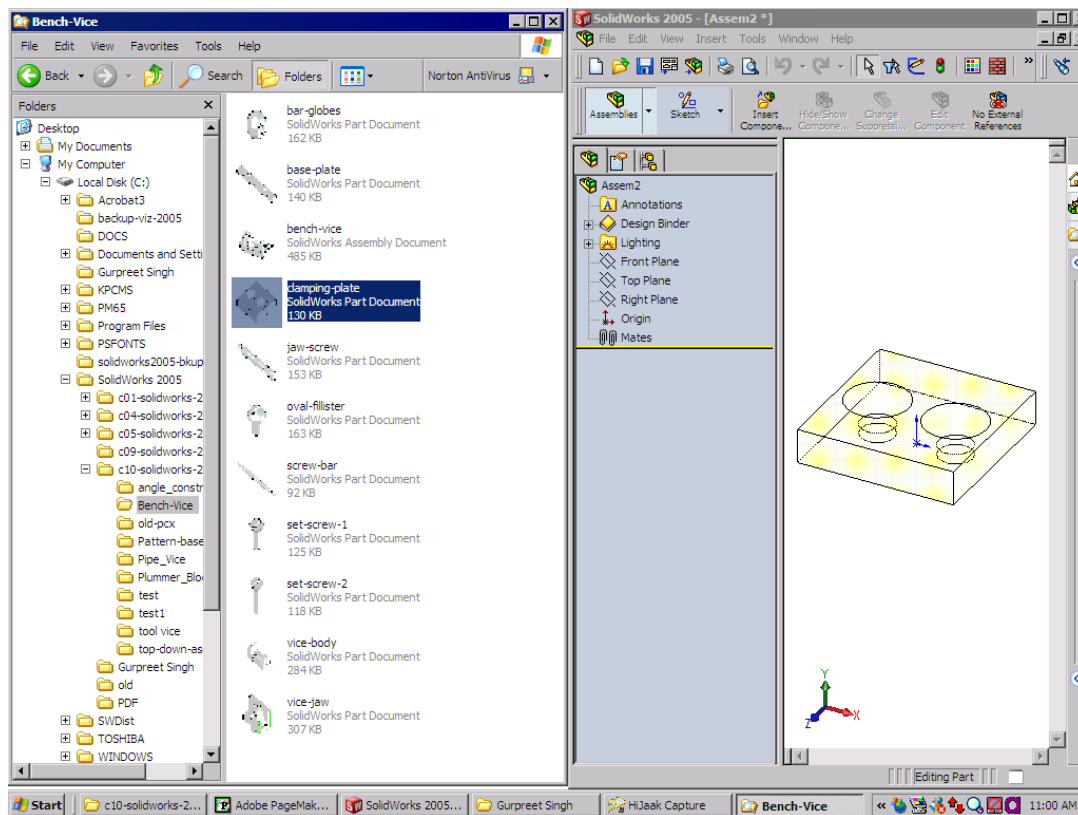


Figure 10-5 Dropping the part from Windows Explorer to the assembly window

Placing Components From Internet Explorer

You can also place the components from Internet Explorer. You need Internet Explorer 4.0 or later for that. Browse the location of the SolidWorks part file link on the Web. Drag the hyperlink and drop it in the drawing area of the assembly document. The **Save As** dialog box will be displayed; save the part document at the desired location.

Placing Additional Instances of Existing Component in the Assembly

Sometimes you need more than one instance of the component to be placed in the assembly document. To do so, press and hold the CTRL key down on the keyboard. Now, select the component in the assembly file and drag the cursor to the location where you want to place the instance of the selected component. Release the left mouse button to drop the new instance of the component. Similarly, you can place as many copies of the existing component as you want by following the above-mentioned procedure.

Placing the Component Using the Feature Palette Window

You can also place a component in the assembly file from the **Feature Palette** window. If you have saved the component in the **Feature Palette** window, you can place that component by dragging and dropping from this window. The **Feature Palette** window is invoked by choosing **Tools > Feature Palette** from the menu bar.

Assembling the Components

After placing the components in the assembly document, you need to assemble them. By assembling the components, you will constrain the degrees of freedom of the components. As mentioned earlier, the components are assembled using mates. Mates help you precisely place and position the component with respect to the other components and the surroundings in the assembly. You can also define the linear and rotatory movement of the component with respect to the other components. In addition, you can create a dynamic mechanism and check the stability of the mechanism by precisely defining the combination of mates. There are two methods of adding mates to the assembly. The first method is using the **Mate PropertyManager** and the second and the most widely used method of adding mates to the assembly is using the **Smart Mates**. Both methods are discussed next.

Assembling Components Using the Mate PropertyManager

CommandManager:	Assemblies > Mate
Menu:	Insert > Mate
Toolbar:	Assembly > Mate



In SolidWorks, mates can be applied using the **Mate PropertyManager**. Choose the **Mate** button available in the **Assemblies CommandManager** or choose **Insert > Mate** from the menu bar. The **Mate PropertyManager** will be invoked, as shown in Figure 10-6.

Select a planar face, curved face, axis, or point on the first component and then select the entity from the second component. The selected entities will be highlighted in green. The names of the selected entities will be displayed in the **Entities to Mate** selection area of the **Mate Selections** rollout. The **Mate** pop-up toolbar will be invoked, as shown in Figure 10-7. The most suitable mates to be applied to the current selection set are displayed in the **Mate** pop-up menu toolbar and in the **Standard Mates** rollout of the **Mate PropertyManager**. The most appropriate mate is selected by default. The preview of the assembly using most appropriate mate is displayed in the drawing area. You can also select the mates from the given list of the mates that can be applied to the current selection set. As you select some other mate from the **Mate** pop-up toolbar the preview of the assembly will be displayed using the newly selected mate. Now, choose the **Add/Finish Mate** button from the **Mate** pop-up toolbar; the **Mate PropertyManager** will be still displayed, and you can add other mates to the assembly. After adding all mates choose the **OK** button from the **Mate PropertyManager**. Various types of mates that can be applied are discussed next.

Coincident



The **Coincident** mate is applied to make two planar faces coplanar. But you can also apply the **Coincident** mate to other entities. The details of the geometries on which

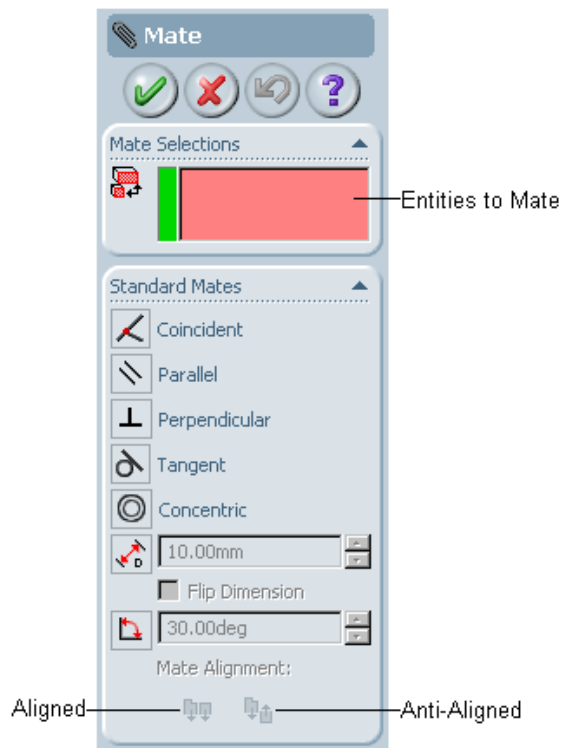


Figure 10-6 The Mate PropertyManager

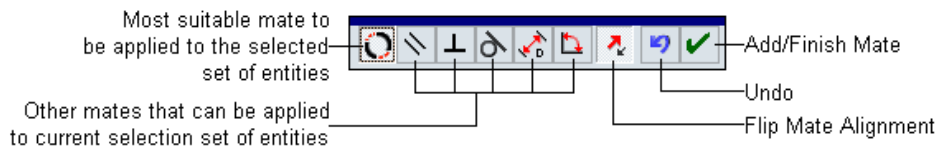


Figure 10-7 The Mate pop-up toolbar

the **Coincident** mate can be applied are shown in Figure 10-8.

When you choose the **Coincident** button from the **Mate** pop-up toolbar, the preview of the model will be displayed according to the current selection of the mate and the model will be assembled in aligned or anti-aligned direction, depending on the current orientation of the model. Depending on the preview of the assembled component the **Aligned** or **Anti-Aligned** button is selected in the **Standard Mates** rollout. You can choose the alignment of the assembled component by choosing the **Flip Mate Alignment** button from the **Mate** pop-up toolbar or switch between **Aligned** and **Anti-Aligned** buttons in the **Mate PropertyManager**. Figure 10-9 displays the faces to be selected to apply the coincident mate. Figure 10-10 shows the resulting mate applied with the default option, that is, **Anti-Aligned** selected. Figure 10-11 shows the coincident mate applied with the **Aligned** button selected.

ASSEMBLY MATE COMBINATIONS (USING COINCIDENT MATE)		Second Component									
		Cone	Cylinder	Line	Point	Sphere	Circular/ Arc Edge	Extrusion	Surface	Plane	Cam
First Component	Cylinder	X	X	✓	✓	X	✓	X	X	X	X
	Sphere	X	X	X	✓	X	X	X	X	X	X
	Cone	✓	X	X	✓	X	✓	X	X	X	X
	Circular/ Arc Edge	X	✓	X	X	X	✓	X	X	✓	X
	Line	X	✓	✓	✓	X	X	X	X	✓	X
	Point	X	✓	✓	✓	✓	X	✓	✓	✓	✓
	Extrusion	X	X	X	✓	X	X	X	X	X	X
	Surface	X	X	X	✓	X	X	X	X	X	X
	Plane	X	X	✓	✓	X	✓	X	X	✓	X

Figure 10-8 Table displaying the combinations for applying the **Coincident** mate

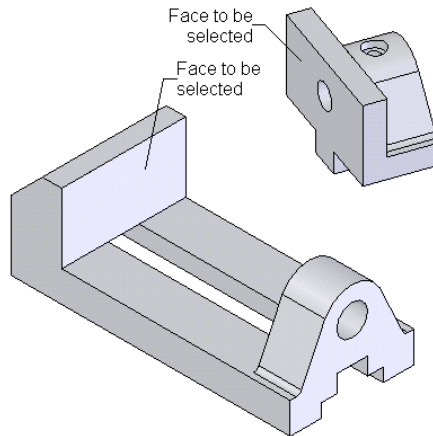


Figure 10-9 Faces to be selected

Concentric



The **Concentric** mate is generally used to align the central axis of one component with that of the other. You generally need to select the circular faces to apply the concentric mate. You can also apply the **Concentric** mate between a point and a circular face or edge. The other combinations of applying the **Concentric** mate are displayed in the table given in Figure 10-12. For applying a **Concentric** mate, invoke the **Mate PropertyManager**. Select two entities from two different components; the names of the selected entities will be displayed in the **Entities to Mate** selection area. The **Concentric** button will be chosen in the **Mate** pop-up toolbar. If this button is not chosen by default, you need to manually choose it. The preview of the models being assembled after applying this mate will be displayed in the graphics area. You can choose the **Aligned** and **Anti-Aligned** buttons on the basis of the design requirement. Choose the **Add/Finish Mate** button from the **Mate** pop-up toolbar.

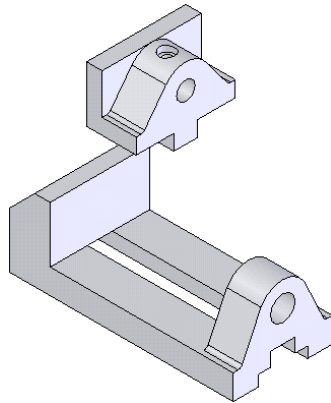


Figure 10-10 *Coincident mate applied with the **Anti-Aligned** button selected*

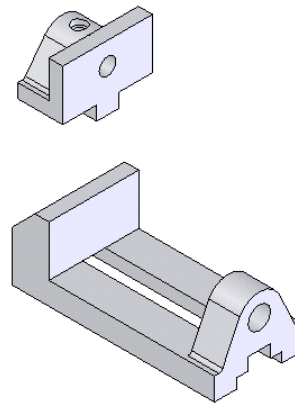


Figure 10-11 *Coincident mate applied with the **Aligned** button selected*

ASSEMBLY MATE COMBINATIONS (USING CONCENTRIC MATE)		Second Component									
		Cone	Cylinder	Line	Point	Sphere	Circular/ Arc Edge	Extrusion	Surface	Plane	Cam
First Component	Cylinder	✓	✓	✓	✓	✓	✓	✗	✗	✗	✗
	Sphere	✗	✓	✓	✓	✓	✗	✗	✗	✗	✗
	Cone	✓	✓	✓	✓	✗	✓	✗	✗	✗	✗
	Circular/ Arc Edge	✗	✓	✓	✗	✗	✓	✗	✗	✗	✗
	Line	✓	✓	✗	✗	✓	✓	✗	✗	✗	✗
	Point	✓	✓	✗	✗	✓	✗	✗	✗	✗	✗
	Extrusion	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Surface	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Plane	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗

Figure 10-12 *Table displaying the combinations for applying the **Concentric** mate*

Figure 10-13 shows the faces to be selected to apply the concentric mate. Figure 10-14 shows the **Concentric** mate applied with the **Aligned** button chosen and Figure 10-15 shows the **Concentric** mate applied with the **Anti-Aligned** button chosen.

Distance



The **Distance** button is chosen to apply the **Distance** mate between two components. To apply this mate, invoke the **Mate PropertyManager** and select the entities from both components. Choose the **Distance** button from the **Mate** pop-up toolbar; the **Distance** spinner will be displayed in the **Mate** pop-up toolbar. Set the value of the distance in the **Distance** spinner. The preview of the assembly will be updated automatically after you set the value of the distance. Using the **Flip Dimension** button available on the left of the **Distance** spinner, you can specify a negative distance value. If needed, you can choose the

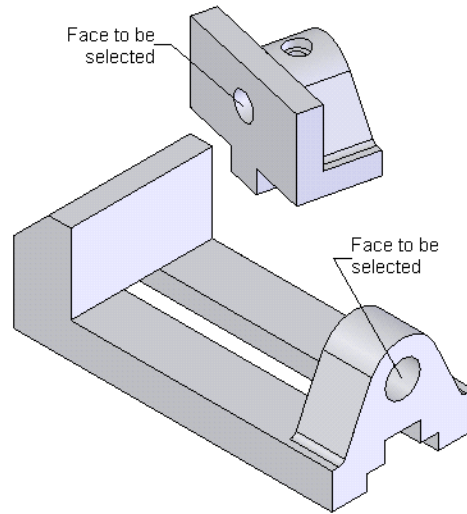
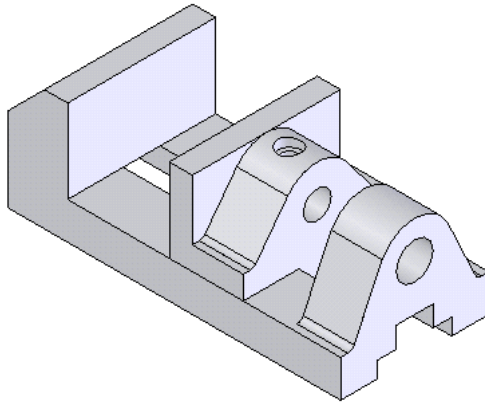
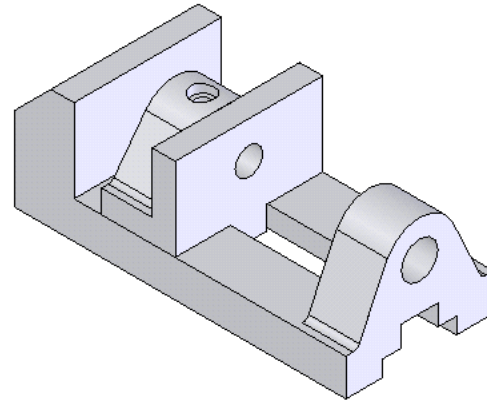


Figure 10-13 Faces to be selected



*Figure 10-14 Concentric mate applied with the **Aligned** button selected*



*Figure 10-15 Concentric mate applied with the **Anti-Aligned** button selected*

Aligned button or the **Anti-Aligned** button. Figure 10-16 shows the combinations of components to apply the **Distance** mate. Figure 10-17 shows the faces to be selected and Figure 10-18 shows the **Distance** mate applied between two components.

Angle



The **Angle** button is used to apply the **Angle** mate between two components. This mate is used to specify the angular position between the selected plane, planar face, or edges of the two components. To apply this mate, invoke the **Mate PropertyManager** and select the entities from the two components. Choose the **Angle** button from the **Mate** pop-up toolbar. The preview of the models will be modified with the default value of the angle. Also, the **Angle** spinner will be invoked and you can set the value of the angle in this spinner. You can also change the angle direction using the **Flip Dimension** button provided on the right of the angle spinner. You can choose the **Aligned** button and

ASSEMBLY MATE COMBINATIONS (USING DISTANCE MATE)		Second Component									
		Cone	Cylinder	Line	Point	Sphere	Plane	Extrusion	Surface	Circular/ Arc Edge	Cam
First Component	Cylinder	X	✓	✓	✓	X	✓	X	X	X	X
	Sphere	X	X	✓	✓	✓	X	X	X	X	X
	Cone	✓	X	X	X	X	X	X	X	X	X
	Plane	X	✓	✓	✓	✓	✓	X	X	X	X
	Line	X	✓	✓	✓	✓	✓	X	X	X	X
	Point	X	✓	✓	✓	✓	✓	X	X	X	X
	Extrusion	X	X	X	X	X	X	X	X	X	X
	Surface	X	X	X	X	X	X	X	X	X	X
	Circular/ Arc Edge	X	X	X	X	X	X	X	X	X	X

Figure 10-16 Table displaying the combinations for applying the **Distance** mate

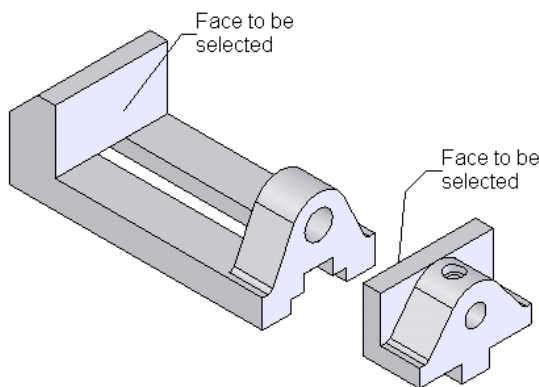


Figure 10-17 Selecting the faces to apply the **Distance** mate

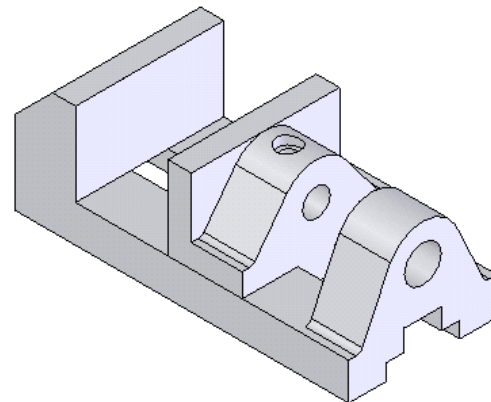


Figure 10-18 **Distance** mate applied to the selected faces

the **Anti-Aligned** button on the basis of the design requirement. After adding the mate, choose the **Add/Finish Mate** button from the **Mate** pop-up toolbar. Figure 10-19 shows the table of combinations for applying the **Angle** mate.

Figure 10-20 shows the faces to be selected to apply the **Angular** mate and Figure 10-21 shows the assembly after applying the **Angular** mate with the angle value of 90-degrees.

Parallel



The **Parallel** button available in the **Mate** pop-up toolbar is used to apply the **Parallel** mate between two components. To apply the **Parallel** mate, invoke the **Mate PropertyManager** and select the two entities from the two components. Choose the **Parallel** button from the **Mate** pop-up menu to apply the mate. You can also choose the **Align** button and the **Anti-Aligned** button. After applying the mate, choose the **Add/Finish**

ASSEMBLY MATE COMBINATIONS (USING ANGLE MATE)		Second Component									
		Cylinder	Extrusion	Line	Plane	Sphere	Circular/ Arc Edge	Cone	Surface	Point	Cam
First Component	Cylinder	✓	✓	✓	×	×	×	×	×	×	×
	Extrusion	✓	✓	✓	×	×	×	×	×	×	×
	Line	✓	✓	✓	×	×	×	×	×	×	×
	Plane	×	×	×	✓	×	×	×	×	×	×
	Sphere	×	×	×	×	×	×	×	×	×	×
	Circular/ Arc Edge	×	×	×	×	×	×	×	×	×	×
	Cone	✓	✓	✓	×	×	×	✓	×	×	×
	Point	×	×	×	×	×	×	×	×	×	×
	Surface	×	×	×	×	×	×	×	×	×	×

Figure 10-19 Table displaying the combinations for applying the Angle mate

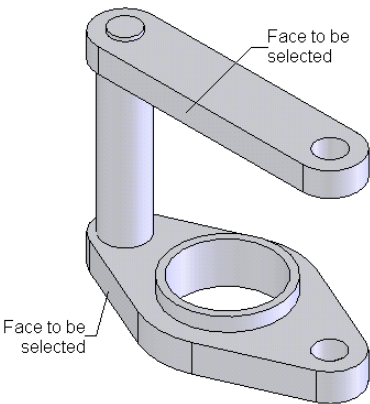


Figure 10-20 Faces selected to apply the Angle mate

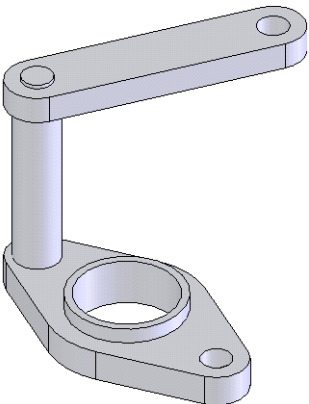


Figure 10-21 Assembly after applying the Angle mate

Mate button from the **Mate** pop-up toolbar. Figure 10-22 shows the combinations to components on which you can apply the **Parallel** mate. Figure 10-23 shows the entities to be selected to apply the **Parallel** mate and Figure 10-24 shows the assembly after applying the **Parallel** mate.

Perpendicular



The **Perpendicular** button available in the **Mate Settings** rollout is used to apply the **Perpendicular** mate between two components. Invoke the **Mate PropertyManager** and select two entities from the two components. Choose the **Perpendicular** button from the **Mate** pop-up toolbar. You can also toggle between the **Aligned** button and the **Anti-Aligned** button. Choose the **Add/Finish Mate** button from the **Mate**

ASSEMBLY MATE COMBINATIONS (USING PARALLEL MATE)		Second Component									
		Cylinder	Extrusion	Line	Plane	Sphere	Circular/ Arc Edge	Cone	Surface	Point	Cam
First Component	Cylinder	✓	✓	✓	✗	✗	✗	✗	✗	✗	✗
	Extrusion	✓	✓	✓	✗	✗	✗	✗	✗	✗	✗
	Line	✓	✓	✓	✓	✗	✗	✗	✗	✗	✗
	Plane	✗	✗	✓	✓	✗	✗	✗	✗	✗	✗
	Circular/ Arc Edge	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Sphere	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Cone	✓	✓	✓	✗	✗	✗	✓	✗	✗	✗
	Surface	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗

Figure 10-22 Table displaying the combinations for applying the **Parallel** mate

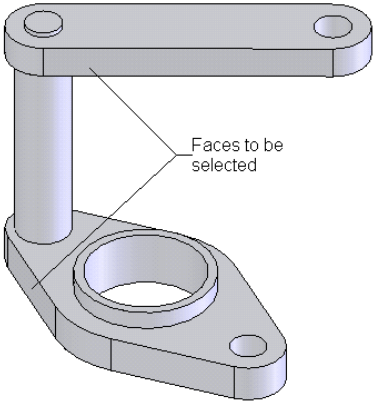


Figure 10-23 Faces selected to apply the **Parallel** mate

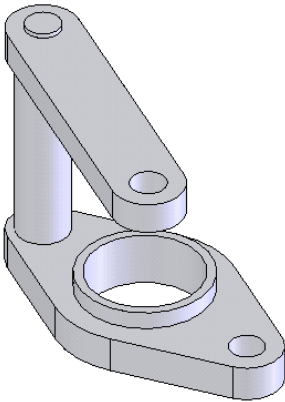


Figure 10-24 Assembly after applying the **Parallel** mate

pop-up toolbar. Figure 10-25 shows the table displaying the combinations for applying the **Perpendicular** mate. Figure 10-26 shows the entities to be selected and Figure 10-27 shows the **Perpendicular** mate applied to the assembly.

ASSEMBLY MATE COMBINATIONS (USING PERPENDICULAR MATE)		Second Component									
		Cylinder	Extrusion	Line	Plane	Sphere	Circular/ Arc Edge	Cone	Surface	Point	Cam
First Component	Cylinder	✓	✓	✓	✗	✗	✗	✗	✗	✗	✗
	Extrusion	✓	✓	✓	✗	✗	✗	✗	✗	✗	✗
	Line	✓	✓	✓	✓	✗	✗	✗	✗	✗	✗
	Plane	✗	✗	✓	✓	✗	✗	✗	✗	✗	✗
	Circular/ Arc Edge	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Sphere	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗
	Cone	✓	✓	✓	✗	✗	✗	✓	✗	✗	✗
	Surface	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗

Figure 10-25 Table displaying the combinations for applying the **Perpendicular** mate

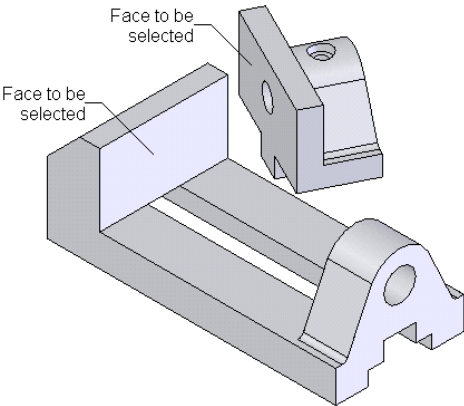


Figure 10-26 Faces selected to apply the **Perpendicular** mate

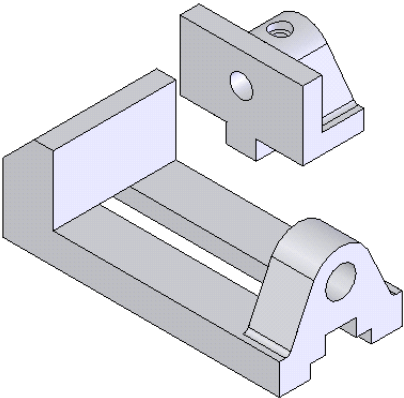


Figure 10-27 Assembly after applying the **Perpendicular** mate

Tangent



The **Tangent** button available in the **Mate** pop-up toolbar is used to apply the **Tangent** mate between two components. Figure 10-28 shows the table displaying the combinations for applying the tangent mate. Figure 10-29 shows the entities to be selected to apply the **Tangent** mate and Figure 10-30 shows the **Tangent** mate applied to the assembly.



Note
The **Advanced Mates** are discussed in Chapter 11.

ASSEMBLY MATE COMBINATIONS (USING TANGENT MATE)		Second Component								
		Cone	Cylinder	Line	Point	Sphere	Plane	Surface	Cam	Extrusion
First Component	Cylinder	✓	✓	✓	✗	✓	✓	✓	✓	✓
	Sphere	✓	✓	✓	✗	✓	✓	✗	✗	✗
	Cone	✗	✗	✗	✗	✓	✓	✗	✗	✗
	Plane	✗	✓	✗	✗	✓	✗	✓	✓	✓
	Line	✗	✓	✓	✓	✓	✓	✓	✓	✓
	Extrusion	✗	✓	✗	✗	✗	✓	✗	✗	✗
	Surface	✗	✓	✗	✗	✗	✓	✗	✗	✗
	Cam	✗	✗	✗	✗	✗	✗	✗	✗	✗

Figure 10-28 Table displaying the combinations for applying the **Tangent** mate

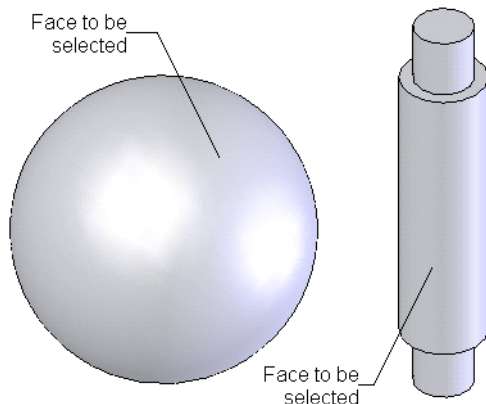


Figure 10-29 Entities to be selected to apply the **Tangent** mate

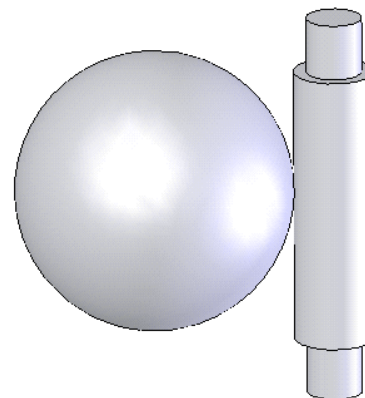


Figure 10-30 Assembly after applying the **Tangent** mate

The options available in the **Mate PropertyManager** are discussed next.

Mate

The **Mate** rollout is used to display the mates that are applied between the selected entities.

Options

The **Add to new folder** check box available in this rollout is used to add the currently applied mate in a folder. This folder is placed in the **Mates** mategroup in the **FeatureManager Design Tree**. You can also drag and drop the other mates from the **Mates** mategroup in the newly created folder. The **Show popup dialog** check box is selected by default and is used to display

the **Mate** pop-up toolbar. The **Show preview** check box is selected by default and is used to display the dynamic preview of the assembly as you apply the mates to the components. The **Use for positioning only** check box is used only to define the position of the component by applying the mate. This mate is displayed in the **Mate** rollout, but when you exit the **Mate PropertyManager**, this mate will not be displayed in the **Mates** mategroup in the **FeatureManager Design Tree**.

Assembling Components Using the Smart Mates

Smart Mates is the most attractive feature of the assembly design environment of SolidWorks. The **Smart Mates** technology speeds up the design process in the assembly environment of SolidWorks. To add smart mates to the components, choose the **Move Component** button from the **Assembly CommandManager**; the **Move Component PropertyManager** will be displayed. Now, choose the **SmartMates** button available in the **Move** rollout; the **Move Component PropertyManager** will be replaced by the **SmartMates PropertyManager**, as shown in Figure 10-31. Also, the select cursor will be replaced by the move cursor.

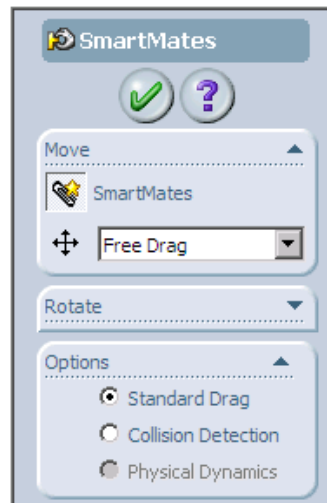


Figure 10-31 The *SmartMates PropertyManager*

Next, double-click on the entity of the first component to add a mate; the component will appear transparent and the cursor will be replaced by the smart mates cursor. Press and hold the left mouse button down on the selected entity and drag the cursor to the entity with which you want to mate the previously selected entity. The symbol of the constraint that can be applied between the two entities will be displayed below the smart mates cursor. You can use the TAB key to toggle between the aligned and anti-aligned options while applying **Smart Mates**. When the symbol of the mate is displayed below the cursor, release the left mouse button; the **Mate** pop-up toolbar will be displayed. Choose the **Add/Finish Mate** button from this toolbar. Figure 10-32 shows the face to be selected to apply a smart mate and Figure 10-33 shows the component being dragged.

Figure 10-34 shows that the concentric symbol appears below the smart mates cursor when the cursor is placed near a circular face of the other component.

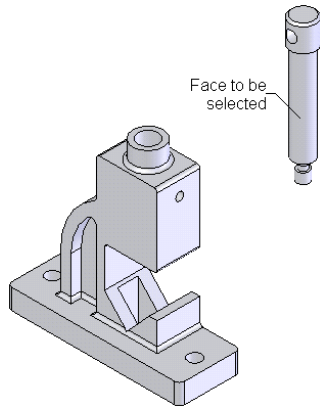


Figure 10-32 Face to be selected to apply a smart mate

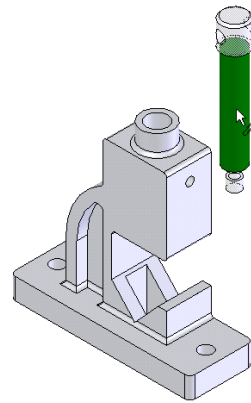


Figure 10-33 Component being dragged to apply a smart mate

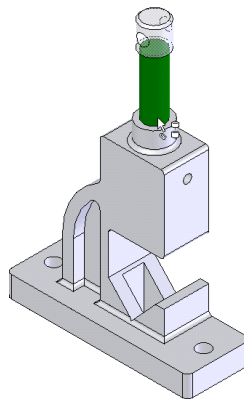


Figure 10-34 Concentric symbol appears below the smart mates cursor

Figure 10-35 shows a planar face selected to apply a smart mate. You can use the TAB key to toggle between the aligned and anti-aligned mates. Figure 10-36 shows that the coincident symbol appears below the cursor and Figure 10-37 shows the assembly after applying the **Coincident** mate using the smart mate.



Note

To apply smart mates without invoking the **SmartMates** tool, press and hold the **ALT** key down and select the entity of the first component. Drag the component to the entity of the other component to which you need to apply the mate. As you release the left mouse button, the **Mate** pop-up toolbar will be displayed. Choose the **Add/Finish Mate** button from this toolbar.

When you drag a component for applying a smart mate, the selected entity of the first component will snap all corresponding entities of the second component. You can press the **ALT** key on the keyboard to exit the snap. To enter the snap mode again press the **ALT** key on the key board.

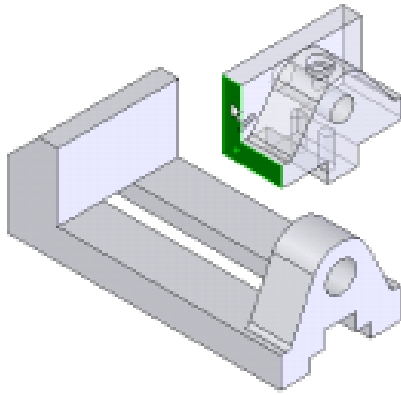


Figure 10-35 Planar face to be selected to apply a mate using smart mates

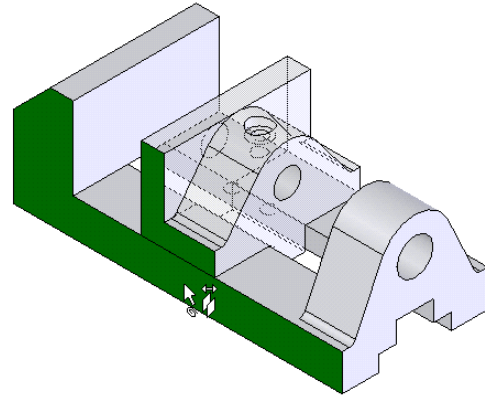


Figure 10-36 Coincident symbol appears after dragging the component near another planar face

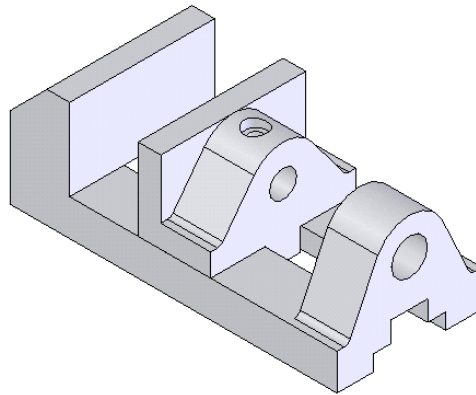


Figure 10-37 Coincident mate applied to the assembly using smart mates



Tip. You can also add smart mates without dragging the component. To add a smart mate without dragging, invoke the **Smart Mates** tool. Double-click on the entity from the first component. The component will be displayed in transparent. Now, select the entity from the second component. The preview with the most appropriate mate will be displayed and the **Mate** pop-up toolbar is invoked.

Geometry-based Mates

In the assembly design environment of SolidWorks, you can also add geometry-based mates. Geometry-based mates are also a type of smart mate and are applied while you are placing a component in the assembly environment. Consider a case in which the first component is already placed in the assembly environment. Now, open the part document of the second component. Choose **Window > Tile Horizontal** or **Tile Vertical** from the menu bar.

Suppose you need to insert the revolved feature of the second component in the circular slot of the first component and at the same time you need to align the larger bottom face of the second

component with the upper face of the first component. To do so, press and hold the left mouse button down on the edge of the second component, as shown in Figure 10-38. Drag the cursor to the assembly window near the upper edge of the circular slot of the first component; the second component mated with the first component will be displayed in temporary graphics, as shown in Figure 10-39. The coincident symbol will also be displayed below the smart mates cursor.

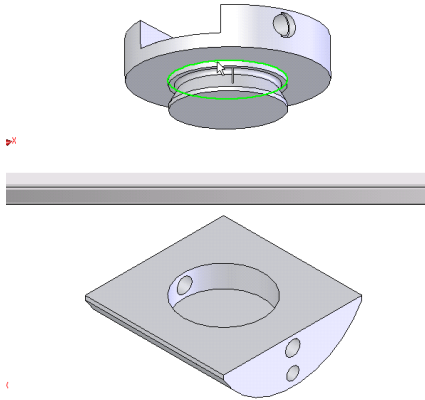


Figure 10-38 Edge of the second component to be selected.

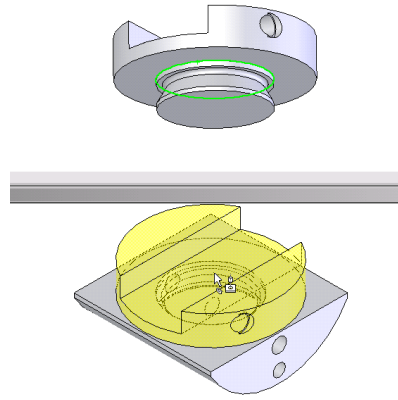


Figure 10-39 Component being dragged into the assembly window for applying geometry-based mates

You can also toggle the direction of placement of the component. Figure 10-40 shows that the direction of placement is flipped using the TAB key. Again press the TAB key to return to the default direction. Release the left mouse button to place the component. If you expand the **Mates** option from the **FeatureManager Design Tree**, you will notice that two mates are applied to the assembly; one is the coincident mate and the other is the concentric mate. Figure 10-41 shows the assembly after adding the geometry-based smart mates.

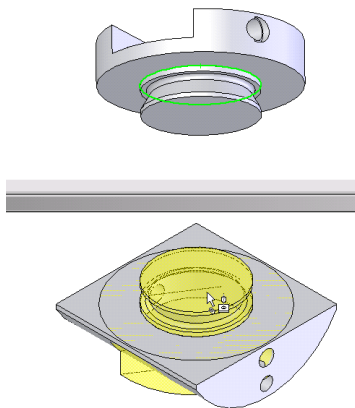


Figure 10-40 The placement direction of component is flipped using the TAB key

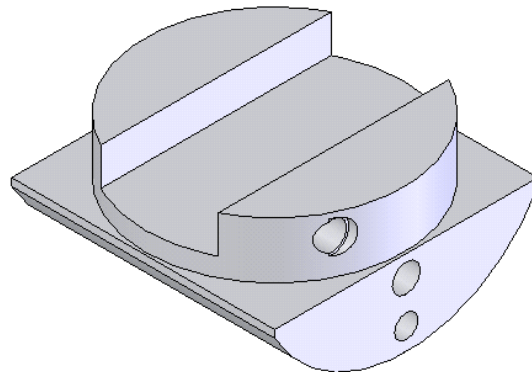


Figure 10-41 Assembly after adding a geometry-based mate



Tip. You can add the geometry-based mates between two linear edges, two planar faces, two vertices, two conical faces, two axes, between an axis and a conical face, and between two circular edges.

Feature-based Mates

In the assembly design environment of SolidWorks, you can also add feature-based mates. For adding feature-based mates, one of the features of the first component must be a circular base or boss feature and the second component must have a hole or a circular cut feature. The feature can be an extruded or a revolved feature. Also, in the assembly document, one of the parts must be placed earlier. Open the part document of the component to be placed using feature-based mates and tile both document windows. In the **FeatureManager Design Tree** of the part document, select the extruded or revolved feature and drag it to the assembly window. Place the cursor at a location where you need to place the component. You can also change the alignment or direction of placement using the TAB key. Release the left mouse button to drop the component. Figure 10-42 shows the component being dragged by selecting the revolved feature from the **FeatureManager Design Tree** of the part document. Figure 10-43 shows the resulting component assembled using feature-based mates.

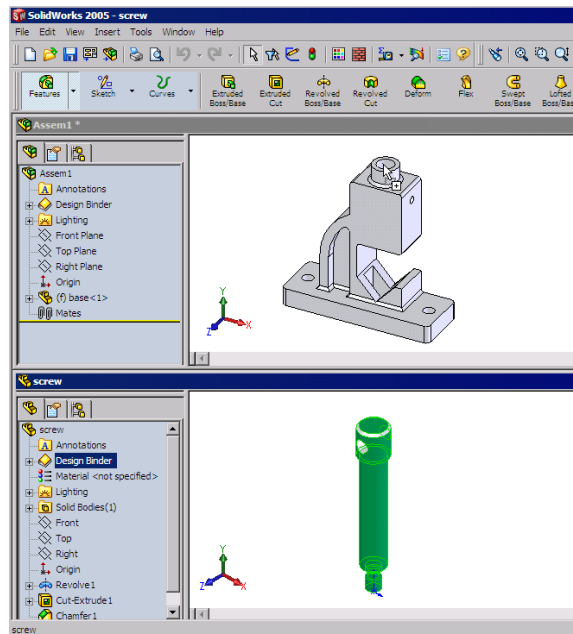


Figure 10-42 Component being dragged by selecting the revolved feature

Pattern-based Mates

Pattern-based mates are used to assemble the components that have a circular pattern created on the circular feature. The best example for these type of components is the flange or a shaft coupling. Remember that to create pattern-based mates, all components that will be assembled must have a circular pattern on the mating faces. To create pattern-based mates, select the outer edge of the

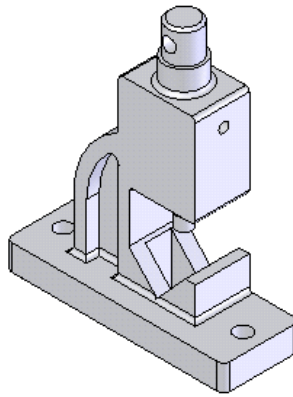


Figure 10-43 Component assembled using feature-based mates



Tip. The feature-based mates are applied only to the components having cylindrical or conical features. You cannot add feature-based mates using features other than cylindrical or conical geometry. If you are adding feature-based mates to a component having a conical face, the second component also must have a conical face. You cannot add a feature-based mate if the geometry of the feature of one component is cylindrical and that of the other is conical.

If you are adding feature-based mates using the features having conical geometry, there must be a planar face adjacent to the conical face of both features.

second component and drag it to the circular edge of the first component that is already placed in the assembly document. The preview of the component assembled with the first component will be displayed. Using the TAB key, you can switch the part with respect to the pattern instances. Release the left mouse button to drop the part. Figure 10-44 shows the component being dragged to the assembly document window. Figure 10-45 shows the preview of the component being assembled and Figure 10-46 shows the component assembled using pattern-based mates.

Assembling Components Using Mate Reference

CommandManager:	Reference Geometry > Mate Reference	(Customize to Add)
Menu:	Insert > Geometry > Mate Reference	
Toolbar:	Reference Geometry > Mate Reference	(Customize to Add)



In SolidWorks, you can define the mate reference to the part in the **Part** mode or in the **Assembly** mode. The mate references allow you to define the mating references such as planar surfaces, axis, edges, and so on before assembling the component.

It is recommended that the mate reference should be added to the component in the **Part** mode. The mate references are created using the **Mate Reference PropertyManager**, which is invoked by choosing the **Mate Reference** button from the **Reference Geometry Command** after customizing it. The **Mate Reference PropertyManager** is shown in Figure 10-47.

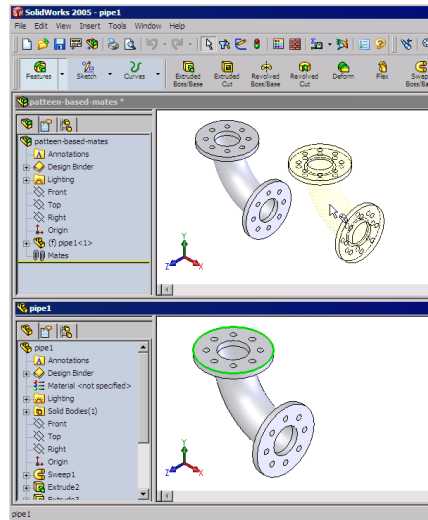


Figure 10-44 Component being dragged into the assembly document

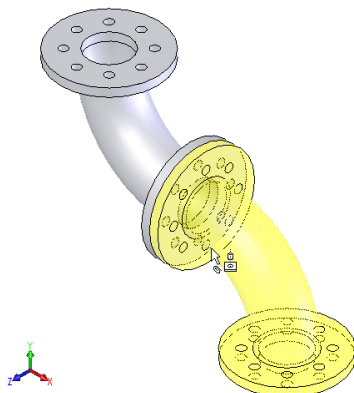


Figure 10-45 Preview of the component being assembled using pattern-based mates

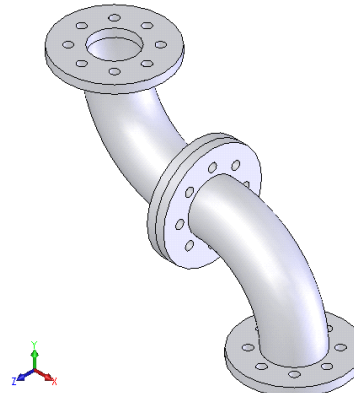


Figure 10-46 Component assembled using pattern-based mates

The **Mate Reference Name** edit box is used to define the name of the mate reference. The **Primary Reference Entity** rollout is used to define the primary mate reference. The **Mate Reference Type** drop-down list is used to define the type of mate. The **Mate Reference Alignment** drop-down list is used to define the type of alignment.

The **Secondary Reference Entity** rollout is used to define the secondary mate reference. The **Tertiary Reference Entity** rollout is used to define the tertiary mate reference.

To assemble a component using the mate reference, you need to define the mate references for two components. Also, the names of the mate references should be the same for both components. After defining the mate references for both components, place the first component coincident with the origin in the assembly document. Now, drag the second component in the assembly

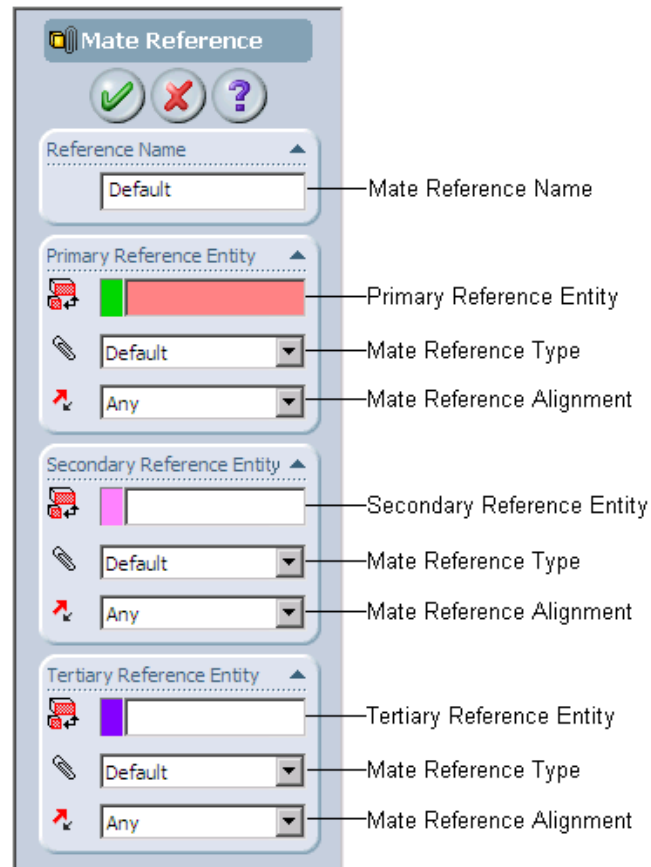


Figure 10-47 The Mate Reference PropertyManager

document and you will notice that the second component is aligned to the references that were defined as mate references. Therefore, you do not need to apply mates in the assembly environment.

CREATING TOP-DOWN ASSEMBLIES

As mentioned earlier, top-down assemblies are the assemblies in which all components are created in the same assembly file. However, to create the components, you require an environment in which you can draw the sketches or the sketched features and the environment where you can convert the sketches into features. In other words, you need a sketching environment and a part modeling environment in the assembly file. In SolidWorks, you can invoke the sketching environment and the part modeling environment in the assembly document itself. The basic procedure for creating the components in the assembly or in other words the procedure for creating the top-down assembly is discussed next.

Creating the Component in the Top-Down Assembly

CommandManager:	Assemblies > New Part	(Customize to Add)
Menu:	Insert > Component > New Part	
Toolbar:	Assembly > New Part	(Customize to Add)



Before creating the first component in the top-down assembly, you first need to save the assembly document. After creating a new assembly document, choose the **Save** button from the **Standard** toolbar to save the assembly file. It is recommended that you create a new folder and save the assembly file and the other referenced file in the same folder.

Now, choose the **New Part** button from the **Assemblies CommandManager** after customizing it. The **Save As** dialog box will be displayed where you need to specify the name and location to save the new component. Enter the name of the new component in the **File name** edit box and choose the **Save** button. The select cursor will be replaced by the new component cursor. Now, you need to place the new component in the assembly document. Using the left mouse button place the component on the **Front** assembly plane, which is displayed in the **FeatureManager Design Tree**. The plane on which you place the component is selected as the sketching plane and the sketching environment is invoked automatically. You will notice that the **Edit Part** button available in the **Assemblies CommandManager** is selected. This means that the part modeling environment is invoked in the assembly document. You can draw the sketch of the base feature in the current sketching environment or you can also exit the sketching environment and select any other sketching plane to create the sketch. After creating the sketch of the base feature, exit the sketching environment. Now, as the **Features CommandManager** is not available by default, you need to invoke it and use the tools available in the **Features CommandManager** to convert the sketch into a model. Similarly, create the remaining features in the model.

After creating all features, choose the **Edit Part** button from the **Assembly** toolbar to exit the part modeling environment. The newly created component will have an **Inplace** mate with the default assembly plane on which it was placed earlier. Therefore, the newly created component is fixed. Using the procedure mentioned above, create other components.

Whenever you create a component in a top-down assembly, the component is fixed using the **Inplace** mate. You can delete this mate by selecting the **Inplace** mate by expanding the **Mates** option available in the **FeatureManager Design Tree**. After selecting the mate, press the DELETE key. Now, this component is floating and you can move this component. You can also assemble this component according to your requirement. You will learn more about fixed and floating components later in this chapter.

**Note**

*As discussed earlier, when you place the first component in the assembly in bottom-up assembly design approach, that component is fixed by default. Therefore, you cannot apply any mates to a fixed component. If you want to add some mates to the fixed component, you first need to float the component. To do so, select the component from the drawing area or from the **FeatureManager Design Tree**. Right-click to invoke the shortcut menu and choose the **Float** option.*

*The components placed after the first component are floating components by default. If you need to fix a floating component, select the component and invoke the shortcut menu. Choose the **Fix** option from the shortcut menu.*

MOVING THE INDIVIDUAL COMPONENTS

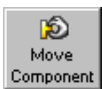
In SolidWorks you can move the individual unconstrained components in the assembly document without affecting the position and location of the other components. There are three methods of moving an individual component. Two methods of moving the individual component are discussed next. The third method is discussed later in this chapter.

Moving the Individual Components by Dragging

One of the major enhancements in the **Assembly** mode of SolidWorks 2005 is that you can move the component placed in the assembly without invoking any tool. To move an individual component, simply select the component and drag the cursor to move the component. Release the left mouse button to place the component at the desired location.

Moving Individual Components Using the Move Component Tool

CommandManager:	Assemblies > Move Component
Toolbar:	Assembly > Move Component



You can also move an individual component using the **Move Component** tool. Choose the **Move Component** button from the **Assemblies CommandManager** to invoke the **Move Component PropertyManager** is invoked. You will notice that the **Free Drag** option is selected in the **Move** drop-down list in the **Move** rollout. You will be prompted to select a component and drag to move it. The select cursor will be replaced by the move cursor; select the component and drag the cursor to move the component. Release the left mouse button to move the component to the desired location. The other options available in the **Move** drop-down list to move the component are discussed next.

Along Assembly XYZ

Using the **Along Assembly XYZ** option from the **Move** drop-down list, you can move the component dynamically along the X, Y, and Z axes of the assembly document. Select the **Along Assembly XYZ** option from the **Move** drop-down list in the **Move** rollout. An assembly coordinate system in magenta will be displayed in the drawing area and you will be prompted to select a component and drag parallel to an assembly axis to move along that axis. Select the component and drag the cursor to move the component along any one of the assembly axes.

Along Entity

The **Along Entity** option is used to move the component along the direction of the selected entity. When you invoke this option, the **Selected item** selection area will be displayed. You will be prompted to select an entity to drag along, then select a component and drag to move it. Select an entity to define the direction in which you need to move the component. The name of the selected entity is displayed in the **Selected item** selection area. Now, select the component and drag the cursor to move the component along the direction of the selected entity.

By Delta XYZ

The **By Delta XYZ** option available in the **Move** drop-down list is used to move the selected component to a given distance in a specified direction. When you select this option, the **Delta X**, **Delta Y**, and **Delta Z** spinners will be invoked and you will be prompted to select a component and enter the distance to move in the PropertyManager. An assembly coordinate system will also be displayed in magenta. Select the component to move and specify the distance in the respective direction spinners in the X, Y, or Z direction. Choose the **Apply** button to move the component.

To XYZ Position

The **To XYZ Position** option is used to specify the coordinates of the origin of the part where the component will be placed after moving. When you select this option, the **X Coordinate**, **Y Coordinate**, and **Z Coordinate** spinners will be invoked and you will be prompted to select a component and enter the XYZ coordinates for the part's origin. An assembly coordinate system will also be displayed in magenta. Select the component and enter the respective coordinates of the part origin in the spinners and choose the **Apply** button to move the component.

ROTATING INDIVIDUAL COMPONENTS

In SolidWorks you can rotate an individual unconstrained component in the assembly document without affecting the position and location of the other components. The **Rotate Component** tool is used to rotate the component. There are three methods of rotating an individual component. Two methods of rotating an individual component are discussed next. The third method is discussed later in this chapter.

Rotating the Individual Components by Dragging

In SolidWorks 2005, you can rotate the component placed in the assembly without invoking any tool. To rotate an individual component, simply select the component using the right mouse button and drag the cursor to rotate the component. Release the right mouse button after attaining the desired orientation of the individual component.

Rotating Individual Components Using the Rotate Component Tool

CommandManager:	Assemblies > Rotate Component
Toolbar:	Assembly > Rotate Component



You can also rotate an individual component using the **Rotate Component** tool. To rotate an individual component using this tool choose the **Rotate Component** button from the **Assemblies CommandManager**; the **Rotate Component PropertyManager** will be invoked. You will notice that the **Free Drag** option is selected in the **Rotate** drop-down list in the **Rotate** rollout. Therefore, you will be prompted to select a component and drag it to rotate. The select cursor will be replaced by the rotate cursor; select the component and drag the cursor to rotate the component. The other options available in the **Rotate** drop-down list to rotate the component are discussed next.

About Entity

The **About Entity** option available in the **Rotate** drop-down list is used to rotate the component with respect to a selected entity. The selected entity is defined as the rotational axis. When you invoke this option, the **Selected item** selection area will be displayed and you will be prompted to select an axis entity to rotate about. Select an entity to define the rotational axis. The name of the selected entity will be displayed in the **Selected item** selection area. Now, select the component and drag the cursor to rotate the component about the selected axis.

By Delta XYZ

The **By Delta XYZ** option available in the **Rotate** drop-down list is used to rotate the selected component by a given incremental angle along the specified axis. When you select this option, the **Delta X**, **Delta Y**, and **Delta Z** spinners will be invoked and you will be prompted to select a component and enter the desired rotation in the PropertyManager. Select the component to rotate and specify the rotation angle in the respective spinners in the direction in which you need to rotate the component. Choose the **Apply** button to rotate the component.



Tip: You can toggle between the **Move PropertyManager**, **Rotate PropertyManager**, and **SmartMates PropertyManager**. If you need to invoke the **Move PropertyManager**, invoke the **Rotate** rollout to switch to the **Rotate PropertyManager**. If you select the **SmartMates** button, the system will switch you to the **SmartMates PropertyManager**. From the **Rotate** or **SmartMates PropertyManager**, if you invoke the **Move** rollout, the system will switch you to the **Move PropertyManager**.

MOVING AND ROTATING AN INDIVIDUAL COMPONENT USING THE TRIAD

To move an individual component using the triad, you first need to select the component and then right-click to invoke the shortcut menu. Choose the **Move with Triad** option from the shortcut menu; the component will appear transparent and the triad will be displayed on it. Using this triad you can move and rotate a component. The different components of the triad are displayed in Figure 10-48.

Press and hold the left mouse button down on the X arm; the select cursor will be replaced by the move/rotate cursor. Use the left mouse button to drag the selected component in the X direction. If you use the right mouse button and drag the cursor, the selected component will rotate about the X axis. Similarly, you can select the Y or the Z arm and drag the cursor to move/rotate the component in the Y or Z direction.

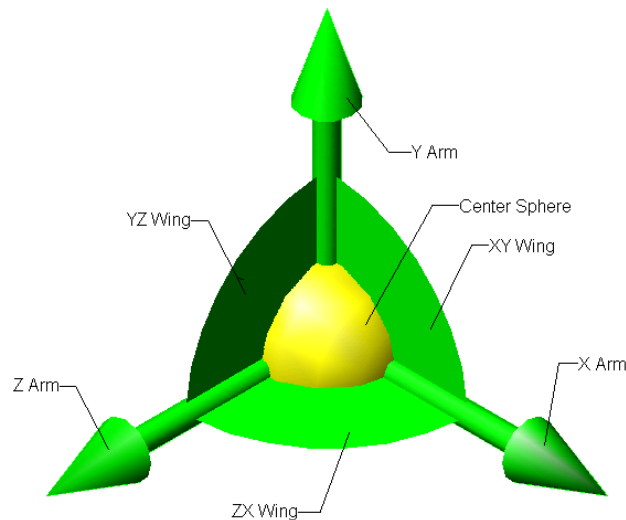


Figure 10-48 Triad

You can also move a component in the XY plane. To do so, select the XY wing of the triad and drag the cursor to move the component. To move the component in the YZ plane, select the YZ wing of the triad and drag the cursor to move the component. Similarly, by selecting the ZX wing and dragging the cursor, you can move the component in the ZX plane.

If you select the center sphere and invoke the shortcut menu, various options will be available, which are discussed next.

Show Translate XYZ Box

The **Show Translate XYZ Box** option available in the shortcut menu is used to display the **Translate XYZ** box. Using this box you can specify the value of the X, Y, and Z coordinates of the destination where you need to place the selected component. Specify the X, Y, and Z coordinates in the respective edit boxes and then choose **OK** from the box. When you move the component by dragging, the values of the X, Y, and Z spinners will change automatically.

Show Translate Delta XYZ Box

The **Show Translate Delta XYZ Box** option available in the shortcut menu is used to display the **Translate Delta XYZ** box. Using this box you can specify the incremental value by which you need to move the selected component in the X, Y, or Z direction. Set the incremental value in the X, Y, or Z edit box and then choose **OK** from the box.

Show Rotate Delta XYZ Box

The **Show Rotate Delta XYZ Box** option available in the shortcut menu is used to display the **Rotate Delta XYZ** box. Using this box, you can specify the incremental value by which the selected component will rotate in the X, Y, or Z direction. Set the incremental value in the X, Y, or Z edit box and then choose **OK** from the box.

Align to Component

The **Align to Component** option available in the shortcut menu is used to align the triad with respect to the current component origin.

Evaluation chapter. Logon to www.cadcam.com for more details

TUTORIALS

Tutorial 1

In this tutorial, you will create all components of the Bench Vice and then assemble them. The Bench Vice assembly is shown in Figure 10-49. The dimensions of various components of the assembly are given in Figures 10-50 through 10-53. **(Expected time: 2 hrs 45 min)**

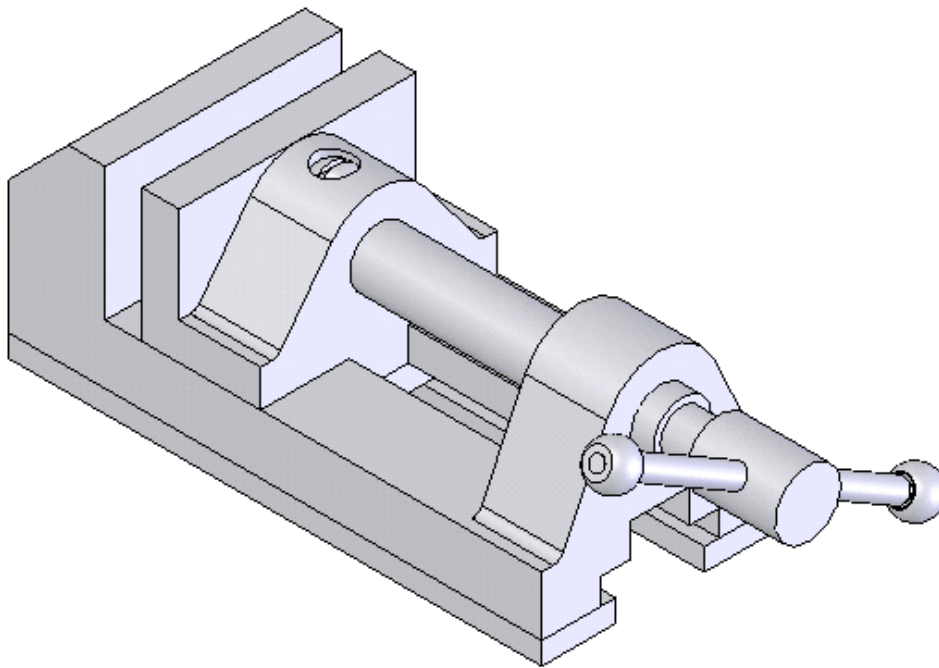


Figure 10-49 Bench Vice assembly

The steps to be followed to complete the assembly of this tutorial are listed below.

- Create all components in individual part documents and save them. The part documents will be saved in the *My Documents\SolidWorks\c10\Bench Vice* folder.
- Open Vice Body and Vice Jaw part documents and define the mate references in both part documents, refer to Figures 10-54 and 10-55
- Create a new assembly document and open all part documents. Place the first component, which is Vice Body, by dragging and dropping from the part document window. Now, drag and drop the Vice Jaw in the assembly document. It will automatically assemble with the

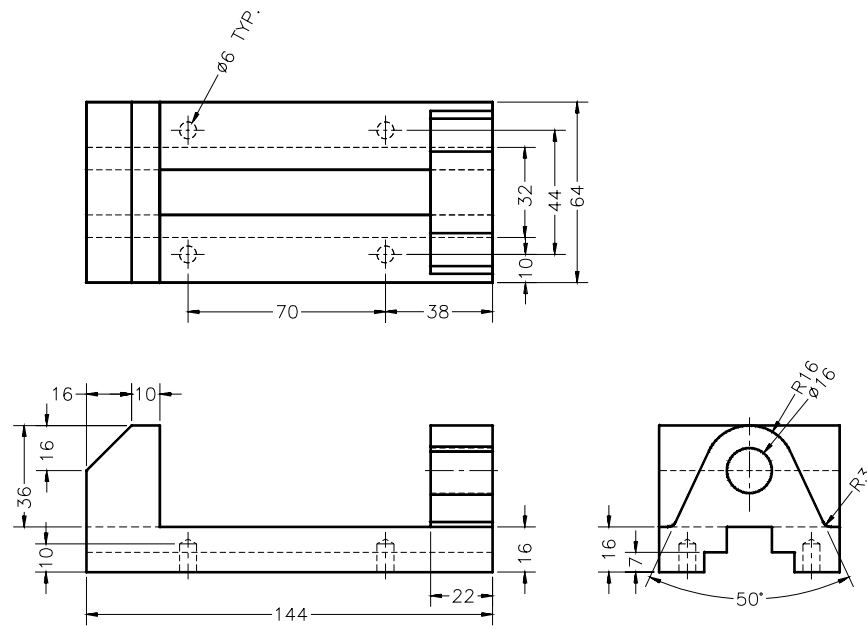


Figure 10-50 Views and dimensions of the Vice Body

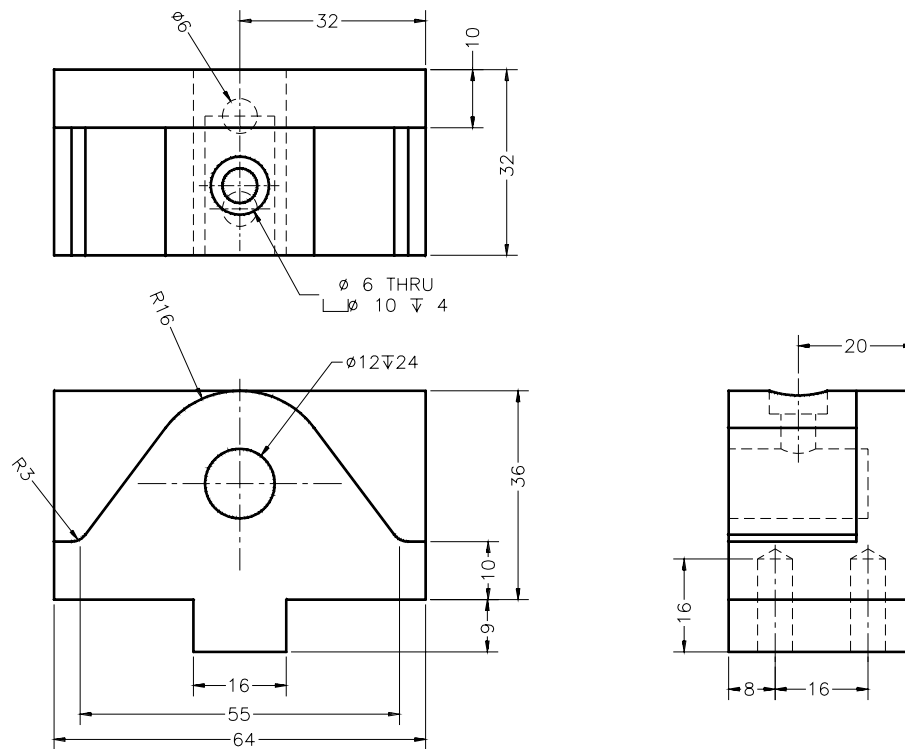


Figure 10-51 Views and dimensions of Vice Jaw

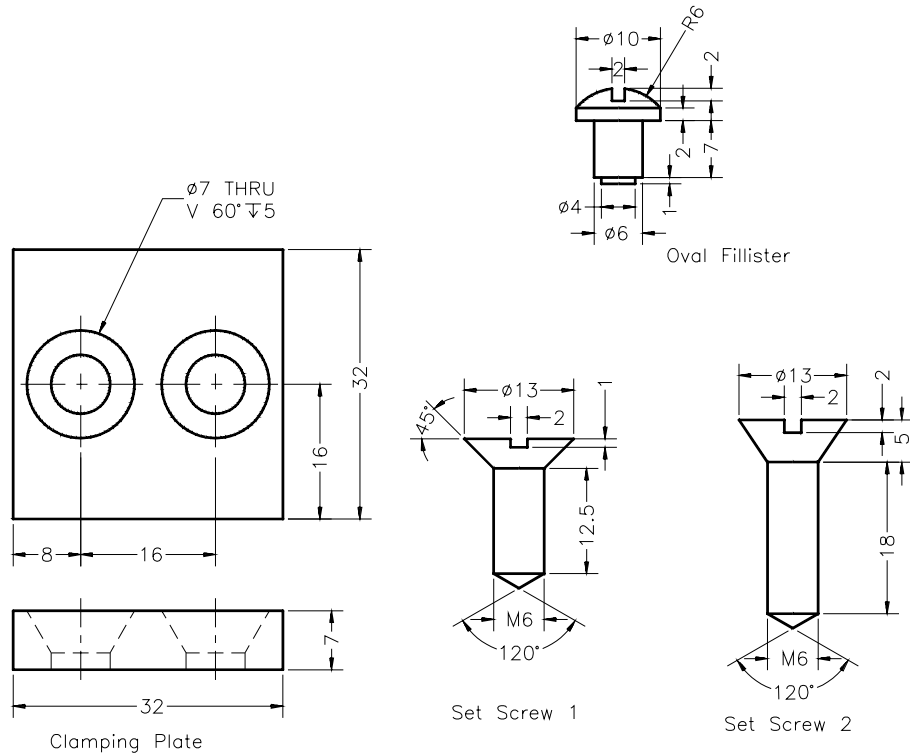


Figure 10-52 Views and dimensions of Clamping Plate, Oval Fillister, Set Screw 1, and Set Screw 2

- Vice Jaw because the mate references are already defined in both part documents, refer to Figures 10-56 and 10-57.
- Drag and drop the jaw screw in the assembly document and apply the required mates, refer to Figures 10-59 through 10-64.
 - Next, analyze the assembly for degrees of freedom of the components.
 - After analyzing the assembly, apply the required mates to constrain all degrees of freedom, refer to Figures 10-65 and 10-66.
 - Next, assemble the Clamping Plate, refer to Figures 10-67 through 10-70.
 - Next, assemble the Oval Fillister using the feature-based mates, refer to Figure 10-71.
 - Similarly, assemble the other components.

Creating the Components

- Create all the components of the Bench Vice assembly as separate part documents. Specify the names of the documents as shown in Figures 10-50 through 10-53. The files should be saved in the folder `\My Documents\SolidWorks\c10\Bench Vice`. Make sure that the *Bench Vice* is your current folder.

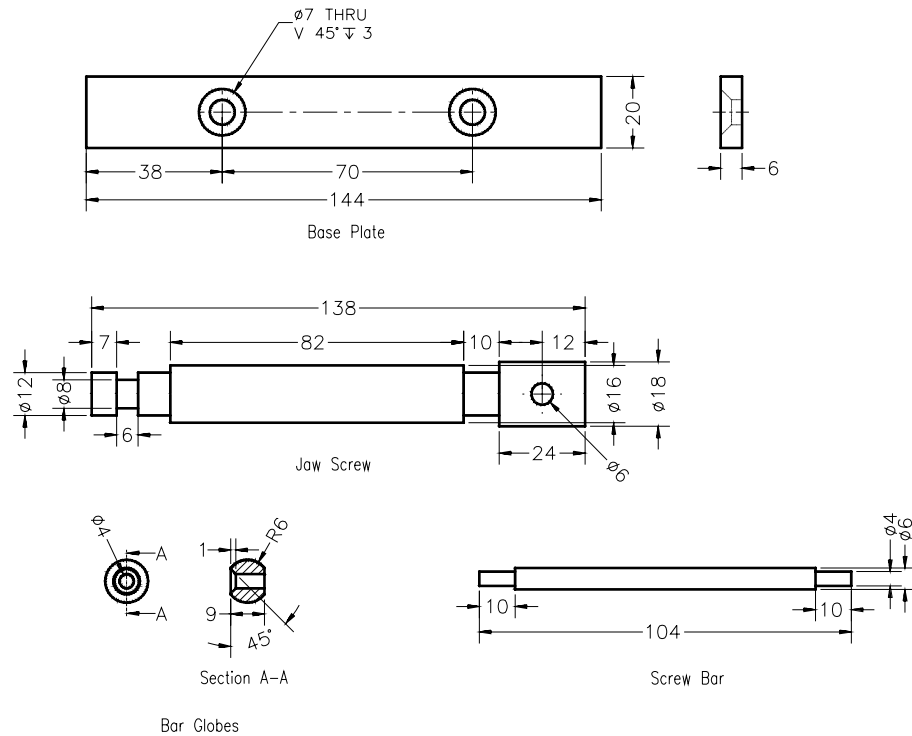


Figure 10-53 Views and dimensions of Base Plate, Jaw Screw, Screw Bar, and Bar Globes

Creating the Mate References

In this tutorial, you need to assemble the first two components of the assembly using the mate references. For assembling the components using the mate references, first you need to create the mate reference. Therefore, you need to open the part documents in which you will add the mate references.

1. Invoke the **Open** dialog box from the **Standard** toolbar.
2. Double-click on the Vice Body; the vice-body part document is opened in the SolidWorks window.
3. Choose **Insert > Reference Geometry > Mate Reference** from the menu bar; the **Mate Reference PropertyManager** is invoked. The selection mode in the **Primary Reference Entity** selection area is active.
4. Select the planar face of the model shown in Figure 10-54 as the primary reference. The selected planar face is highlighted in green.

5. Select the **Coincident** option from the **Mate Reference Type** drop-down list in the **Primary Reference Entity** rollout. The selection mode in the **Secondary Reference Entity** selection area is active.
6. Select the planar face of the model shown in Figure 10-54 of the model as the secondary reference. The selected face is highlighted in magenta.
7. Select the **Coincident** option from the **Mate Reference Type** drop-down list in the **Secondary Reference Entity** rollout.

The selection mode in the **Tertiary Reference Entity** selection area is active.

8. Select the planar face of the model shown in Figure 10-54 as the tertiary reference. The selected face is highlighted in blue.

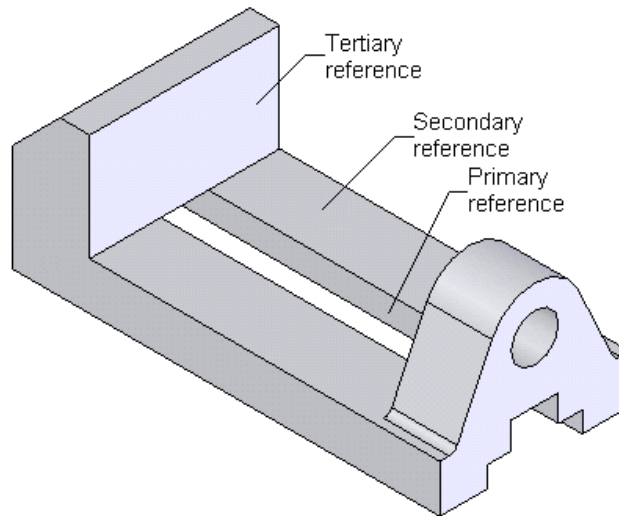


Figure 10-54 Faces to be selected as mate references

9. Select the **Parallel** option from the **Mate Reference Type** drop-down list in the **Tertiary Reference Entity** rollout.
10. Enter **Vice Mate Reference** as the name of mate reference in the **Mate Reference Name** edit box available in the **Reference Name** rollout.
11. Choose the **OK** button from the **Mate Reference PropertyManager**.
12. Similarly, create the mate reference in the Vice Jaw part document. The faces to be selected as reference are displayed in Figure 10-55. The name of the mate reference in the **Reference Name** rollout should be the same in both part documents.
13. Close all part documents, except *Vice Body.sldpart* and *Vice Jaw.sldprt*, if they are opened.

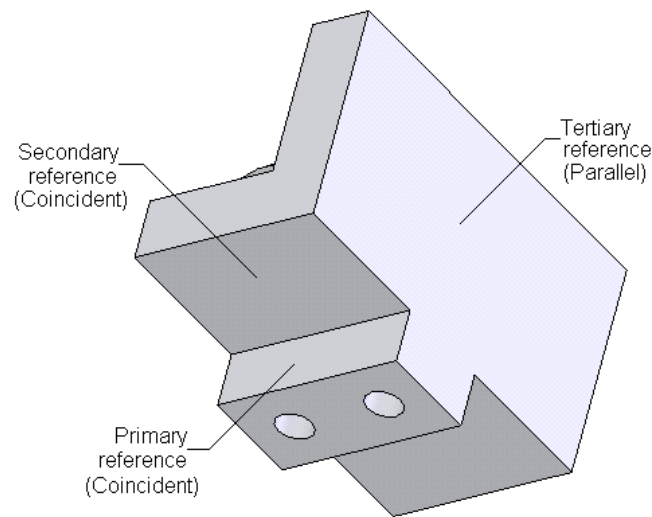


Figure 10-55 Faces to be selected as mate references

Assembling the First Two Components of the Assembly

After creating the mate references in the part documents, you need to assemble the components. To do so, you need to start a new SolidWorks assembly document.

1. Start a new SolidWorks assembly document; the **Insert Component PropertyManager** is invoked automatically and the names of components that were opened are displayed in the **Open documents** selection area.
2. Select Vice Body from the **Open documents** selection area; the preview of the Vice Body is displayed along with the component cursor.
3. Choose the **Keep Visible** button from the **Insert Component PropertyManager** because after placing this component, you also need to place the second component that is displayed in the **Open documents** selection area.

It is recommended that the first component of the assembly should be placed at the assembly origin.

4. Choose the **OK** button from the **Insert Component PropertyManager** to place the first component coincident to the origin.

As the **Keep Visible** button is selected, the preview of the Vice Body is again displayed with the component cursor in the drawing area.

5. Change the current view to isometric.

Next, you need to place the second component in the assembly. As discussed earlier, the second component of the assembly, which is the Vice Jaw, is assembled with the Vice Body using the mate references.

6. Select the Vice Jaw from the **Open documents** selection area; the preview of the Vice Jaw is available in the drawing area.

When you move the cursor close to the Vice Body in the assembly document, the preview of the Vice Jaw assembled after applying the mates with the Vice Body is displayed in the assembly document.

7. Place the component at the required location. The mates specified in the mate references are applied between the Vice Jaw and the Vice Body. Choose the **Keep Visible** button and exit the **Mate PropertyManager**.

Figure 10-56 shows the second component being placed in the assembly document and Figure 10-57 shows the Vice Jaw assembled with the Vice Body.

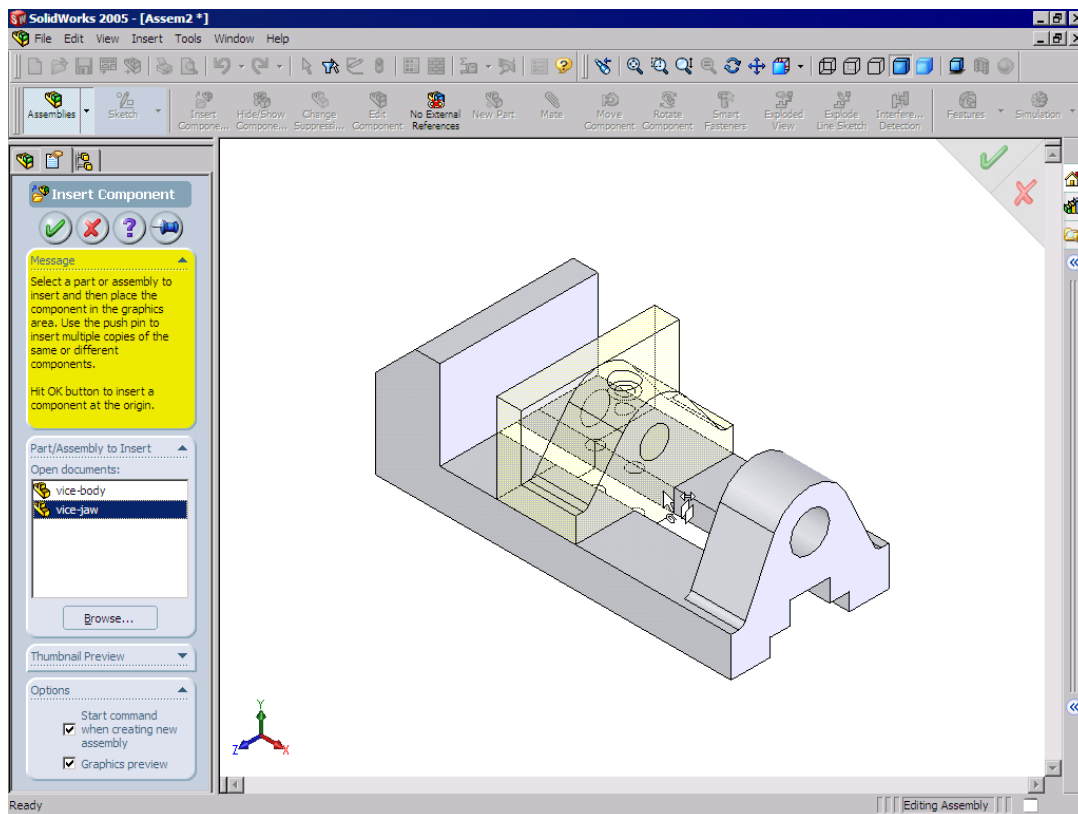


Figure 10-56 Second component being dragged

Before proceeding further, it is recommended that you close all part documents.

8. Close the part document windows of all parts that are placed in the assembly document.

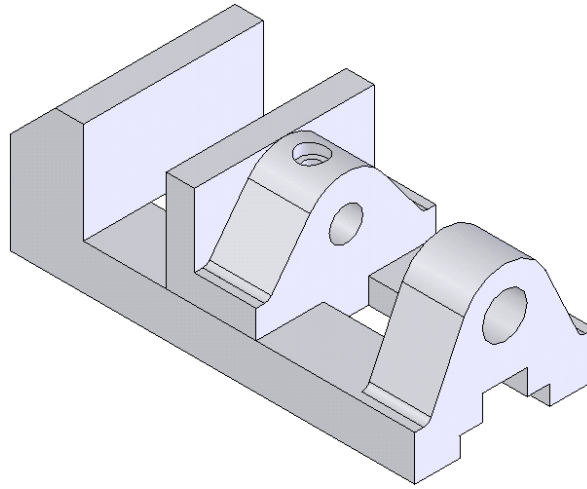


Figure 10-57 Vice Jaw assembled with the Vice Body



Tip. The mates that are defined in the mate reference are applied to the components when you place the components in the assembly. You can view the mates applied to both components by expanding the **Mates** option from the **FeatureManager Design Tree** of the assembly document.

Assembling the Jaw screw

Now, you need to place the Jaw Screw in the assembly document.

1. Choose the **Insert Component** button from the **Assemblies CommandManager**.
2. Choose the **Browse** button from the **Part/Assembly to Insert** rollout to display the **Open**.
3. Double-click on the Jaw Screw to open the part document of Jaw Screw. The preview of the Jaw Screw is displayed in the drawing area.
4. Click anywhere in the drawing area to place the Jaw Screw.

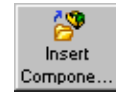


Figure 10-58 shows the Jaw Screw placed arbitrarily in the assembly document. Next, you need to add the assembly mates to assemble the components placed in the assembly.

5. Press and hold the ALT key down on the keyboard and then select the face from the location shown in Figure 10-59.
6. Drag the Jaw Screw to the hole in the Vice Body, as shown in Figure 10-60; the Jaw Screw appears transparent and the select cursor is replaced by the smart mates cursor. Also, the symbol of concentric mate is displayed below the cursor.
7. Release the left mouse button at this location; the **Mates** pop-up toolbar is displayed and the **Concentric** button is chosen by default, which suggests that the **Concentric** mate is the most

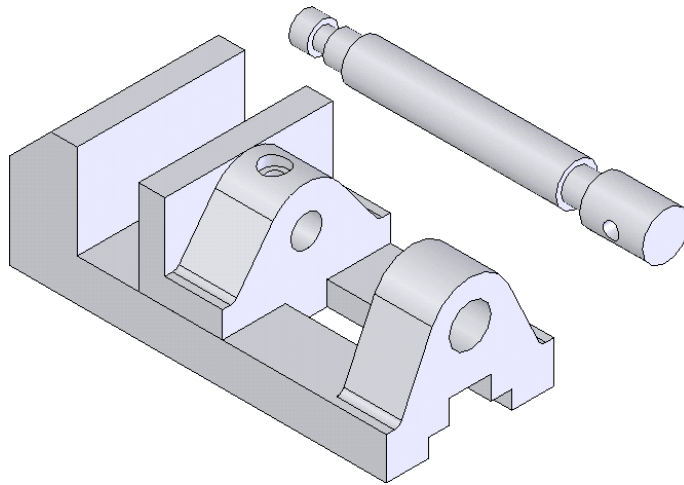


Figure 10-58 Jaw Screw placed in the assembly document

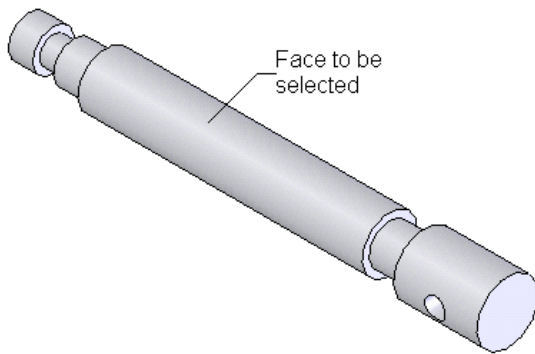


Figure 10-59 Face to be selected

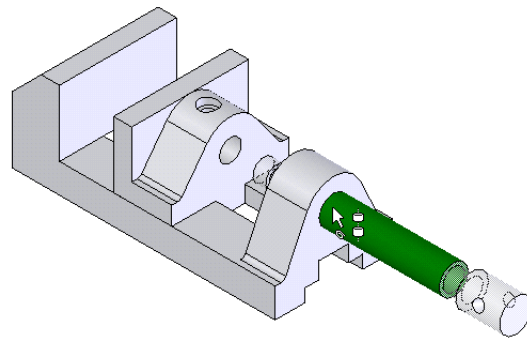


Figure 10-60 Jaw Screw being dragged

appropriate mate to be applied. Choose the **Add/Finish Mate** button from the **Mates** pop-up toolbar. Figure 10-61 shows the Jaw Screw assembled to the Vice Body.

Next, you need to apply the coincident mate between the planar faces of the Jaw Screw and the Vice Jaw.

8. Choose the **Mate** button from the **Assemblies CommandManager** to invoke the **Mate PropertyManager**.
9. Rotate the assembly view and select the face of the Vice Jaw, as shown in Figure 10-62. Next, select the face of the Jaw Screw, as shown in Figure 10-63.



As soon as you select the faces, the **Mates** pop-up toolbar is invoked and the preview of the assembly with coincident mate is displayed in the drawing area.

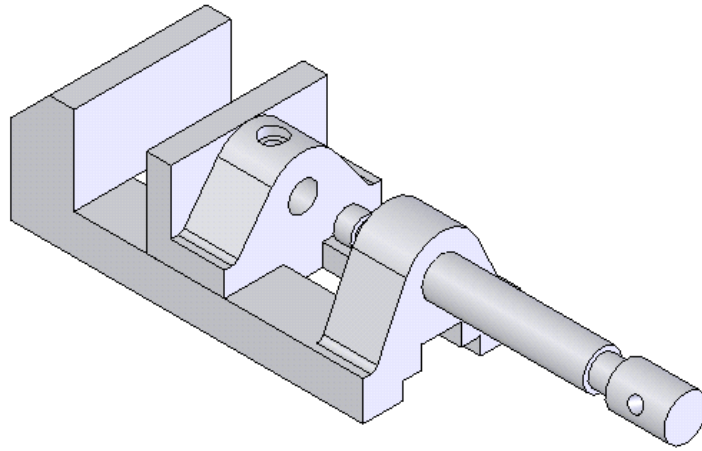


Figure 10-61 Concentric mate applied between the Jaw Screw and the Vice Body

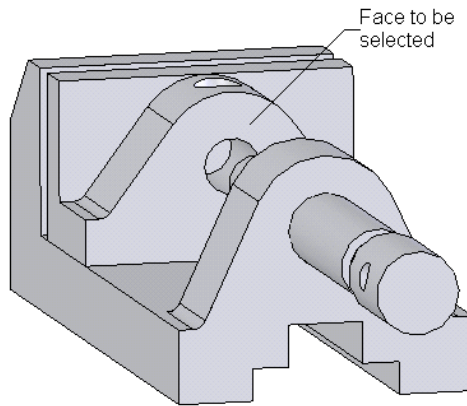


Figure 10-62 Face to be selected

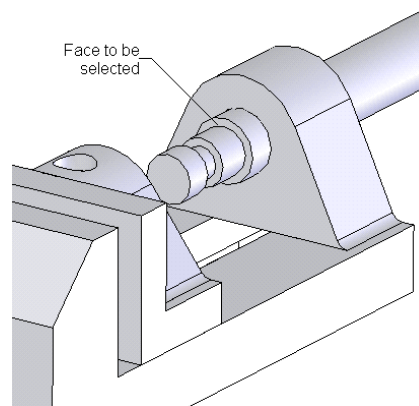


Figure 10-63 Face to be selected

10. Choose the **Add/Finish Mate** button from the **Mate** pop-up toolbar. The assembly after adding the **Coincident** mate is displayed in Figure 10-64.

In real world, there are two types of assemblies. The first is the fully defined assembly in which all degrees of freedom of all components are restricted. The other type of assembly is that in which some degrees of freedom of the components are left free so that they can be moved or rotated. These type of assemblies are used for mechanism that you will learn in the next chapter.

After adding the **Coincident** mate, you need to move the assembly to analyze the degree of freedom of the components of the assembly. After analyzing the components of an assembly you need to add the mates to constrain that degree of freedom.

11. Select the circular face of the Jaw Screw using the left mouse button and drag the cursor.

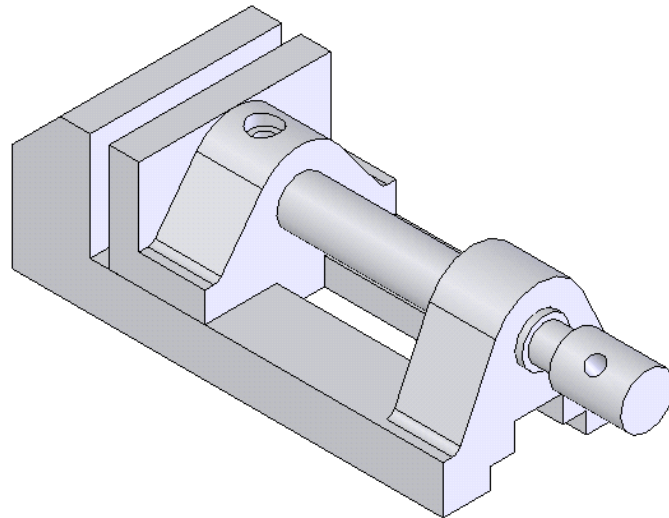


Figure 10-64 Assembly after applying the **Coincident** mate to the Jaw Screw

You will notice that the Jaw Screw is rotating about its axis and is moving along the X axis. Also, it is forcing the Vice Jaw to move along the X axis. Originally, this degree of freedom of Vice Jaw and Jaw Screw needs to be left free so that the assembly can function in the mechanism. But in this chapter, you need to restrict this degree of freedom so as to create a fully defined assembly.

12. Select the two faces, one of the Vice Body and the other of the Jaw Screw, as shown in Figures 10-65 and 10-66.

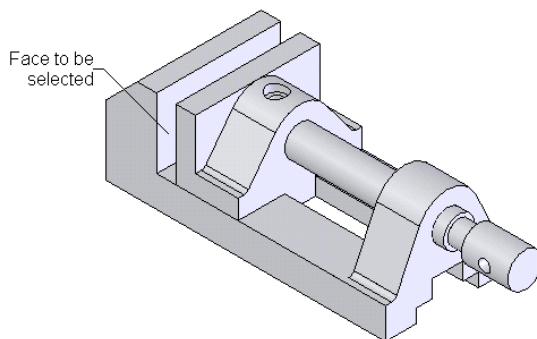


Figure 10-65 Face to be selected

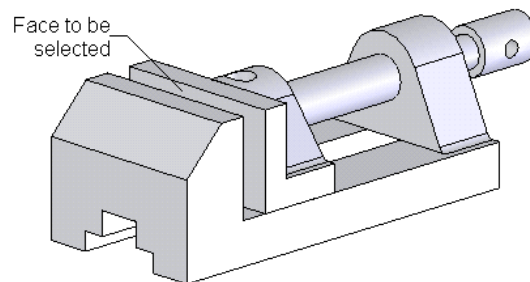


Figure 10-66 Face to be selected

13. Choose the **Distance** button from the **Standard Mates** rollout and set the **Distance** spinner value to **10**. As you choose the **Distance** button, the **PropertyManager** changes to the **Distance 1 PropertyManager**.

14. Choose the **OK** button from the **Distance 1 PropertyManager** to again display the **Mate PropertyManager**.
15. Expand the **FeatureManager Design Tree** displayed in the drawing area and select the **Top Plane**. Now, expand the **Jaw Screw** from the **FeatureManager Design Tree** and select the **Top Plane**. The **Mates** pop-up toolbar is displayed.
16. Choose the **Angle** button from the **Mate** pop-up toolbar.
17. Set the value of the **Angle** spinner to **45** and choose the **Add/Finish Mate** button from the **Mates** pop-up toolbar.
18. Choose the **OK** button from the **Mate PropertyManager** to exit it.

Assembling the Clamping Plate

Next, you need to assemble the Clamping Plate with the assembly.

1. Invoke the **Insert Component PropertyManager** and choose the **Browse** button to invoke the **Open** dialog box. Open and place the Clamping Plate in the drawing area.
2. Rotate the assembly such that the bottom face of the assembly is displayed as shown in Figure 10-67.
3. Select the Clamping Plate using the right mouse button and drag the cursor to rotate the Clamping Plate as shown in Figure 10-68.

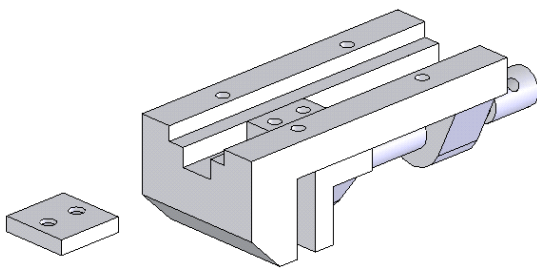


Figure 10-67 Rotated assembly

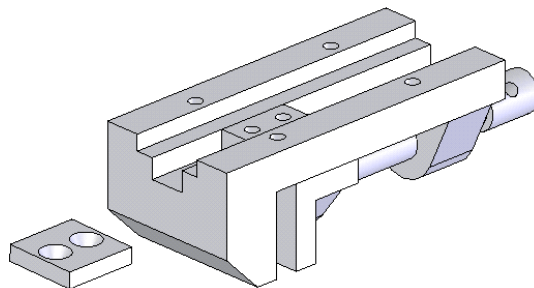


Figure 10-68 Clamping Plate after rotating

4. Apply the **Concentric** mate between the two cylindrical faces of the clamping plate and the two holes of the Vice Jaw, refer to Figure 10-69. You may have to move the Clamping Plate after applying the first mate.
5. Move the Clamping Plate by dragging. Now, apply the **Coincident** mate between the faces of the Clamping Plate and the Vice Jaw, as shown in Figure 10-70.

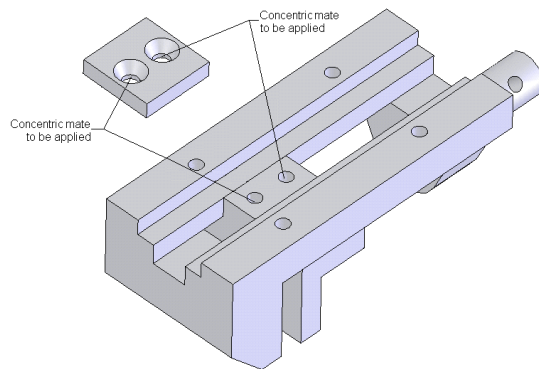


Figure 10-69 Faces to be selected to apply mate

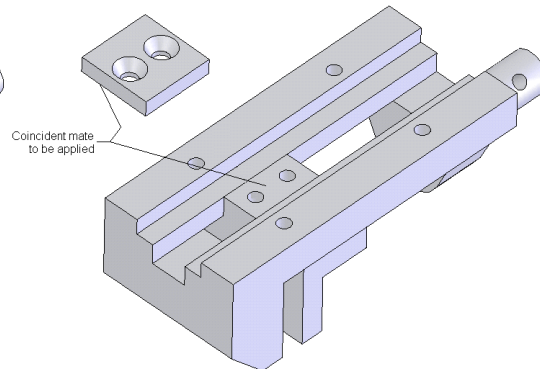


Figure 10-70 Faces to be selected to apply mate

6. Similarly, assemble the Screw Bar, Support Plates, and Bar Globes. The assembly after assembling all these components is shown in Figure 10-71.

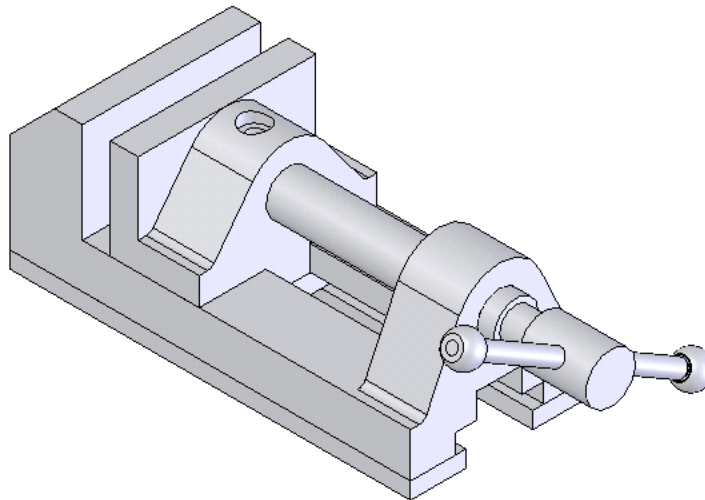


Figure 10-71 Assembly after assembling the Vice Body, Vice Jaw, Jaw Screw, Screw Bar, Clamping Plate, Base Plate, and Bar Globes

Assembling the Remaining Components

Next, you need to assemble the Oval Fillister, Set Screw 1, and Set Screw 2. These fasteners are assembled using feature-based mates.

1. Open the part documents of Oval Fillisters, Set Screw 1, and Set Screw 2.
2. Choose **Window > Tile Horizontally** from the menu bar to rearrange the windows.

3. Select the **Revolve1** feature from the **FeatureManager Design Tree** of the Oval Fillister part document. Drag the cursor to place the component in the assembly, as shown in Figure 10-72.

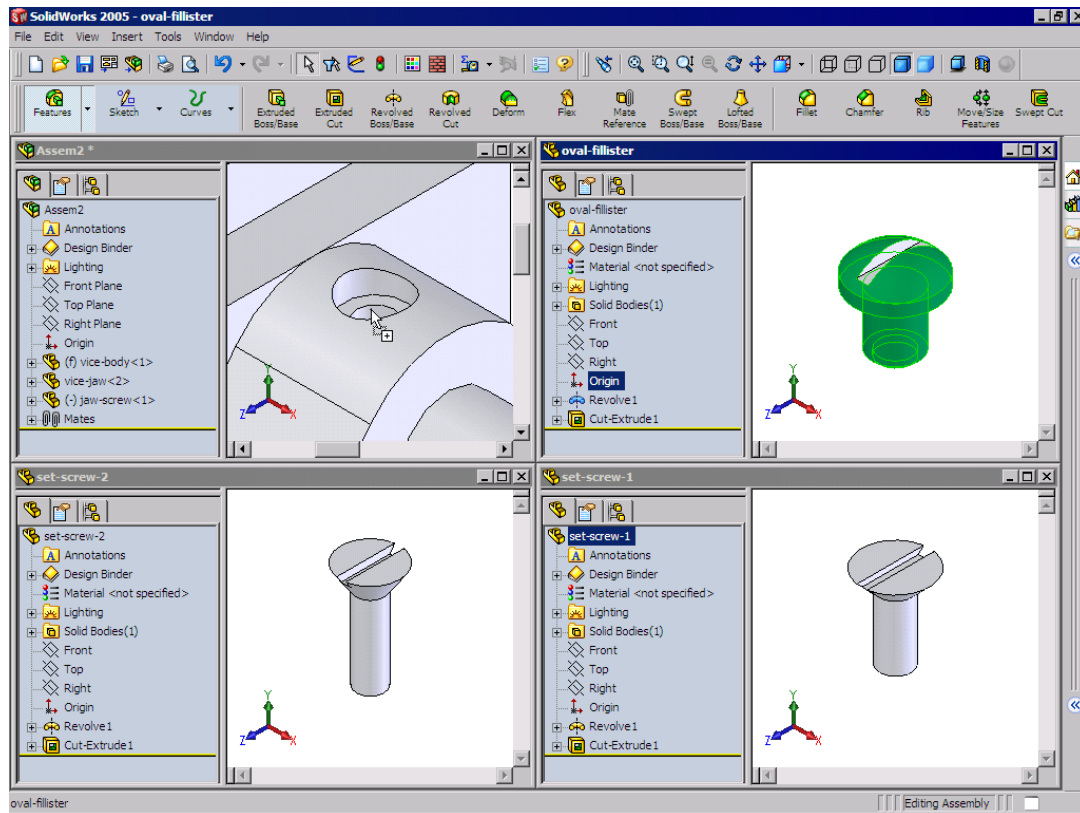


Figure 10-72 Dragging the Oval Fillisters in the assembly using feature-based mates

4. Drop the component at this location.

Similarly, assemble Set Screw 1 and Set screw 2 using feature-based mates. Use the TAB key on the keyboard to reverse the direction. The rotated view of the assembly after assembling Set Screw 1 and Set Screw 2 is shown in Figure 10-73.

5. Choose the **Save** button to save the assembly with the name *Bench Vice* in the folder |My Documents|SolidWorks|Bench Vice.

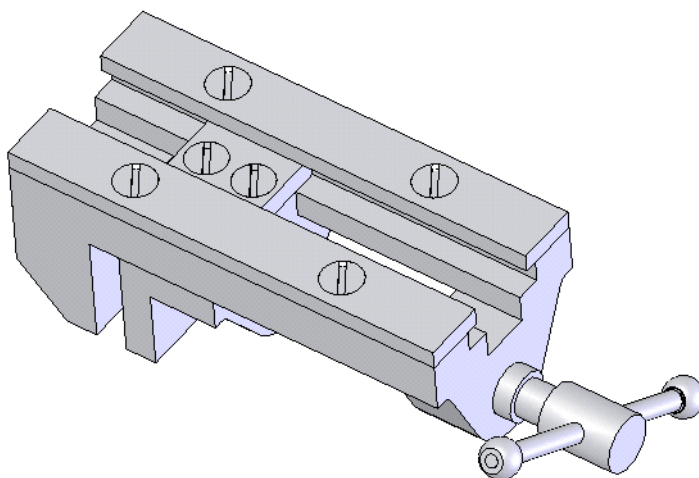


Figure 10-73 Viewing the final assembly from the bottom

Tutorial 2

In this tutorial, you will create all components of the Pipe Vice and then assemble them. The Pipe Vice assembly is shown in Figure 10-74. The dimensions of various components of this assembly are given in Figures 10-75 and 10-76. **(Expected time: 2 hrs 45 min)**

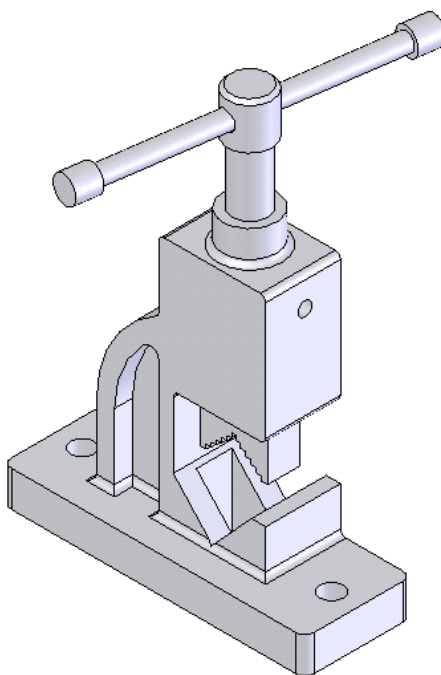


Figure 10-74 Pipe Vice assembly

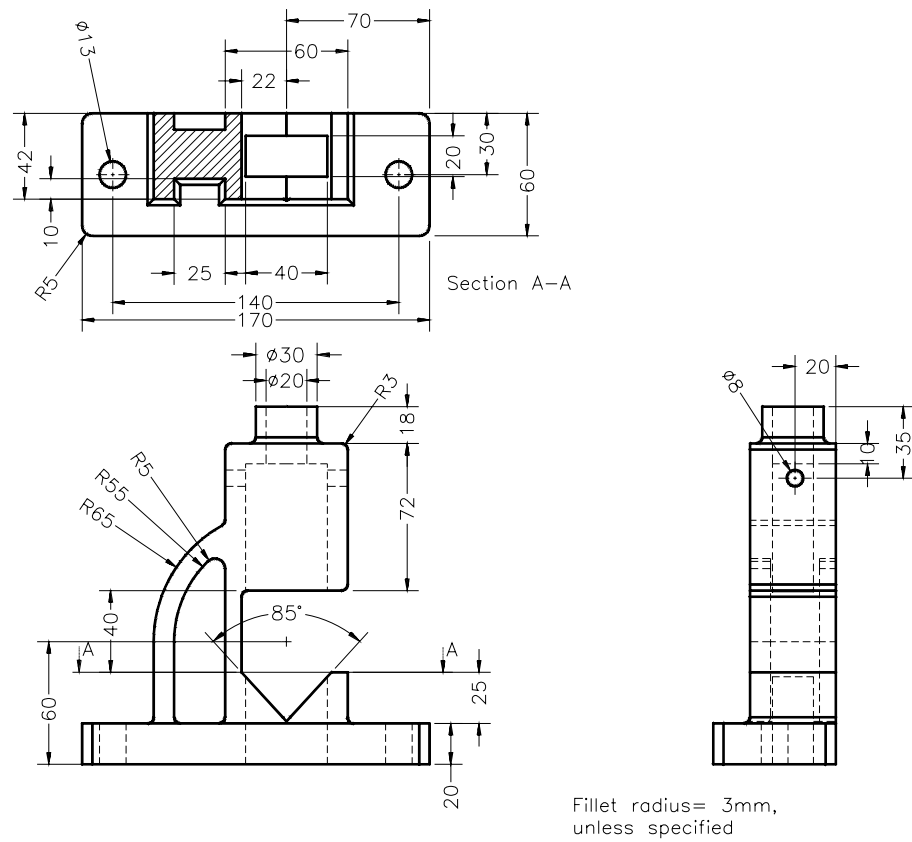


Figure 10-75 Views and dimensions of the base

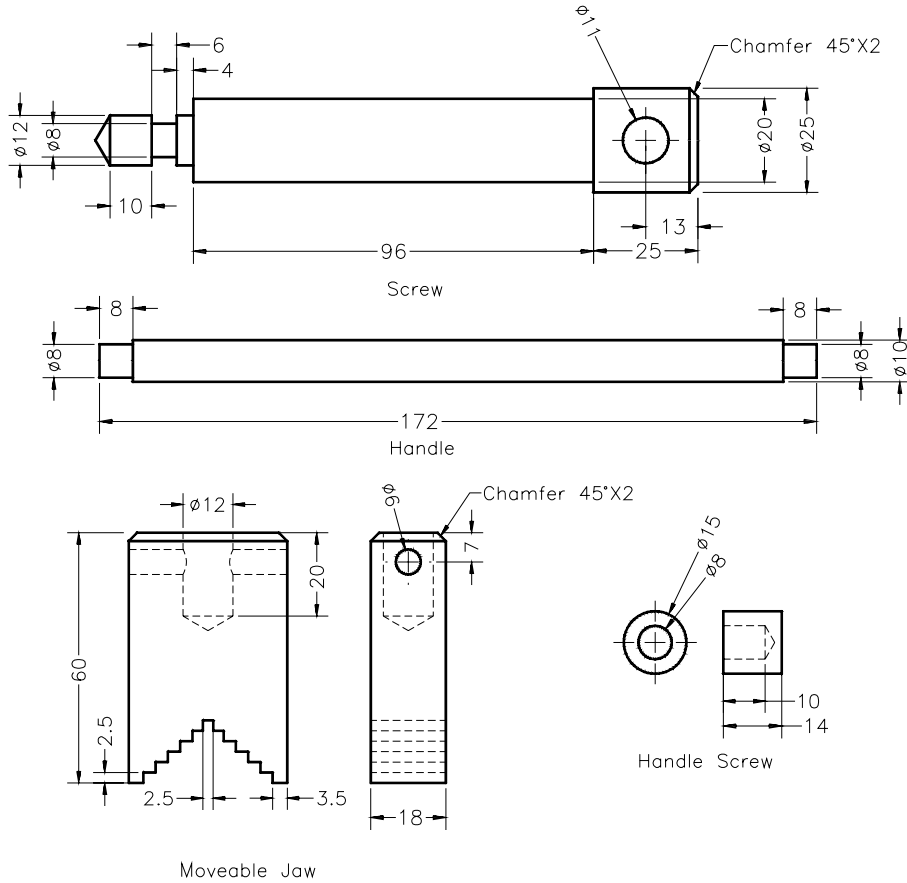


Figure 10-76 Views and dimensions of the Screw, Handle, Moveable Jaw, and Handle Screw

You need to create all components of the Pipe Vice assembly as separate part documents. After creating the parts, you will assemble them in the assembly document. Therefore, in this tutorial, you need to use the bottom-up approach for creating the assembly.

The steps to be followed to complete the assembly of this tutorial are listed below.

- Create all components in the individual part documents and save them. The part documents will be saved in `\My Documents\SolidWorks\c10\Pipe Vice`.
- Place the base at the origin of the assembly.
- Place the Moveable Jaw and the Screw in the assembly. Apply the mates between Moveable Jaw and the Screw, refer to Figures 10-77 through 10-79.
- Assemble the Screw with the Base.
- Place the other components in the assembly and apply the required mates to the assembly, refer to Figure 10-80.

Creating the Components

1. Create all components of the Pipe Vice assembly as separate part documents. Specify the names of the files, as shown in Figures 10-75 and 10-76. The documents should be saved in the folder */My Documents/SolidWorks/c10/Pipe Vice*.

Inserting the First Component in the Assembly

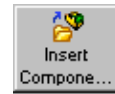
After creating all components of the Pipe vice Assembly, you need to start a new SolidWorks assembly document.

1. Start a new SolidWorks assembly document; the **Insert Component PropertyManager** is invoked by default.
2. Choose the **Browse** button from the **Part/Assembly to Insert** rollout to display the **Open** dialog box. Double-click on Base.
3. Choose the **OK** button from the **Insert Component PropertyManager** to place the Base origin coincident to the origin of the assembly document.
4. Change the view orientation to isometric.

Inserting and Assembling the Moveable Jaw and the Screw

After placing the first component in the assembly document, you need to place the Moveable Jaw and the Screw in the assembly document. After placing these components you need to apply the required mates.

1. Choose the **Insert Components** button from the **Assemblies CommandManager**. Then, choose the **Keep Visible** button from the **PropertyManager** and invoke the **Open** dialog box by choosing the **Browse** button from the **Part/Assembly to Insert** rollout.
2. Double-click on the Moveable Jaw. Place the component anywhere in the assembly document such that it does not interfere with existing component.
3. Similarly, place the Screw in the assembly document and choose the **OK** button from the **Insert Component PropertyManager**. Figure 10-77 shows the Moveable Jaw, Screw, and Base placed in the assembly document.



First, you need to assemble the Screw with the Moveable Jaw. Therefore, for assembling the Screw with the Moveable Jaw, you need to fix the Moveable Jaw.

4. Select the Moveable Jaw from the drawing area or from the **FeatureManager Design Tree**. Right-click to invoke the shortcut menu.
5. Choose the **Fix** option from the shortcut menu. The Moveable Jaw is fixed and you cannot move or rotate the Moveable Jaw.

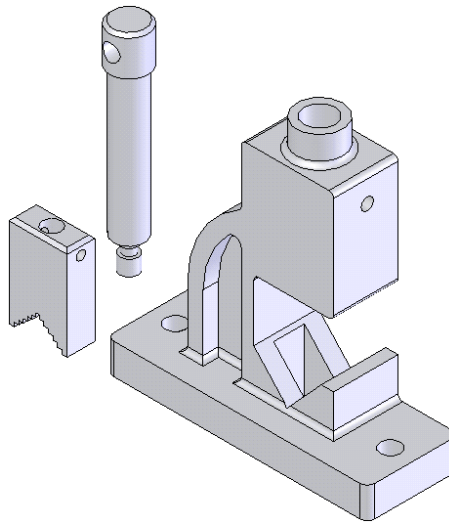


Figure 10-77 The Moveable Jaw, Screw, and Base placed in the assembly document

6. Invoke the **Move PropertyManager** and choose the **SmartMates** button from the **Move** rollout. Double-click on the lowermost cylindrical face of the Screw; the Screw appears transparent.
7. Drag the cursor to the hole located on the top of the Moveable Jaw. Release the left mouse button as soon as the concentric symbol is displayed below the cursor. Choose the **Add/Finish Mate** button from the **Mates** pop-up toolbar.
8. Select the Screw and move it up so that it is not inside the Moveable Jaw.
9. Right-click in the drawing area and choose **Clear Selections** to clear the current selection.
10. Rotate the assembly and double-click on the lower flat face of the Screw; the Screw appears transparent.
11. Again, rotate the model and select the top planar face of the Moveable Jaw. The **Coincident** relation is applied between the two selected faces. Choose the **Add/Finish Mate** button from the **Mates** pop-up toolbar.
12. Choose the **OK** button from the **SmartMates PropertyManager**. Figure 10-78 shows the Screw after applying the mates.

Next, you need to assemble the Screw and the Moveable Jaw with the Base.

13. Select the Moveable Jaw and invoke the shortcut menu. Choose the **Float** option from the shortcut menu.

Now, the Moveable Jaw and the Screw assembled to it can be moved.

14. Using smart mates, add a **Concentric** mate between the cylindrical face of the Screw and the hole created on the top face of the Base.
15. Invoke the **Mate PropertyManager** and select the front planar face of the Moveable Jaw and the front planar face of the Base.
16. Choose the **Parallel** button from the **Mates** pop-up toolbar and choose the **Add/Finish Mate** button to add a parallel mate between the selected faces.
17. Next, select the faces as shown in Figure 10-79.

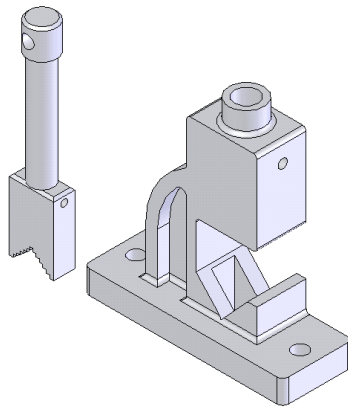


Figure 10-78 The Screw assembled with the Moveable Jaw

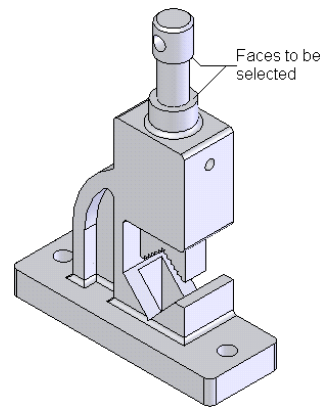


Figure 10-79 Faces to be selected

18. Choose the **Distance** button from the **Mates** pop-up toolbar and set the value of the **Distance** spinner to **35**.
19. Choose the **Add/Finish Mate** button from the **Mates** pop-up toolbar and then choose the **OK** button from the **Mate PropertyManager**.
20. Similarly, assemble the other components of Pipe Vice. Figure 10-80 shows the final Pipe Vice assembly.
21. Choose the **Save** button to save the assembly document in the `\My Documents\SolidWorks\Pipe Vice` folder.

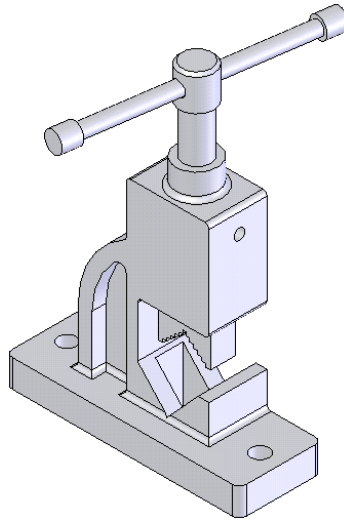


Figure 10-80 Final Pipe Vice assembly

SELF-EVALUATION TEST

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

1. The bottom-up assembly design approach is the traditional and the most widely preferred approach of assembly design. (T/F)
2. In the top-down assembly design approach, all components are created in the same assembly document. (T/F)
3. The **Coincident** mate is generally applied to make two planar faces coplanar. (T/F)
4. The most suitable mates that can be applied to the current selection set are displayed in the **Mate Selections** rollout of the **Mate PropertyManager**. (T/F)
5. Feature-based mates are applied only to the components having cylindrical features. (T/F)
6. Pattern-based mates are used to assemble the components that have a circular pattern created on the circular feature. (T/F)
7. Choose the _____ button from the **Assemblies CommandManager** to invoke the **Rotate Component PropertyManager**.
8. The _____ option available in the **Rotate** drop-down list is used to rotate the selected component by a given incremental angle about the specified axis.

9. The _____ mate is generally used to align the central axis of one component with that of the other.
10. The _____ option available in the **Move** drop-down list is used to move the component along the direction of the selected entity.

REVIEW QUESTIONS

Answer the following questions.

1. The names of the selected entities are displayed in the _____ selection area of the **Mate PropertyManager**.
2. Choose _____ from the menu bar to place a component in the assembly document.
3. Using the _____ option from the **Move** drop-down list, you can move the component dynamically along the X, Y, and Z axes of assembly document.
4. The _____ button available in the **Standard Mates** rollout is used to make the two selected entities normal to each other.
5. The _____ option is used to specify the coordinates of the origin of the part where the component will be placed after moving.
6. If you are adding feature-based mates using the features having conical geometry, then there must be a _____ face adjacent to the conical face of both features.
7. The most widely used method of adding the mates to the components in the assembly in SolidWorks is
 - (a) **Smart Mates**
 - (b) **Mate PropertyManager**
 - (c) By dragging from part document
 - (d) None of these
8. Which button is used to make the **Mate PropertyManager** available after applying a mate to the selected entities?
 - (a) **Help**
 - (b) **OK**
 - (c) **Keep Visible**
 - (d) **Cancel**
9. Which option available in the **Rotate** drop-down list is used to rotate the component with respect to the selected entity?
 - (a) **Along Entity**
 - (b) **Selected Edge**
 - (c) **Reference Entity**
 - (d) None of these

10. Which option is used to specify the coordinates of the origin of the part where the component will be placed after moving?

(a) **To XYZ Position**

(b) **Reference Position**

(b) **Along Entity**

(c) None of these

EXERCISE

Exercise 1

Create the Plummer Block assembly as shown in Figure 10-81. The dimensions of the components of this assembly are shown in Figures 10-82 through 10-84. **(Expected time: 1 hr)**

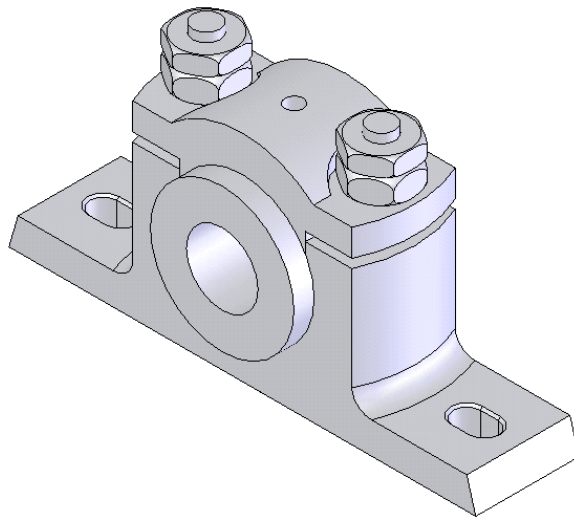


Figure 10-81 Plummer Block assembly

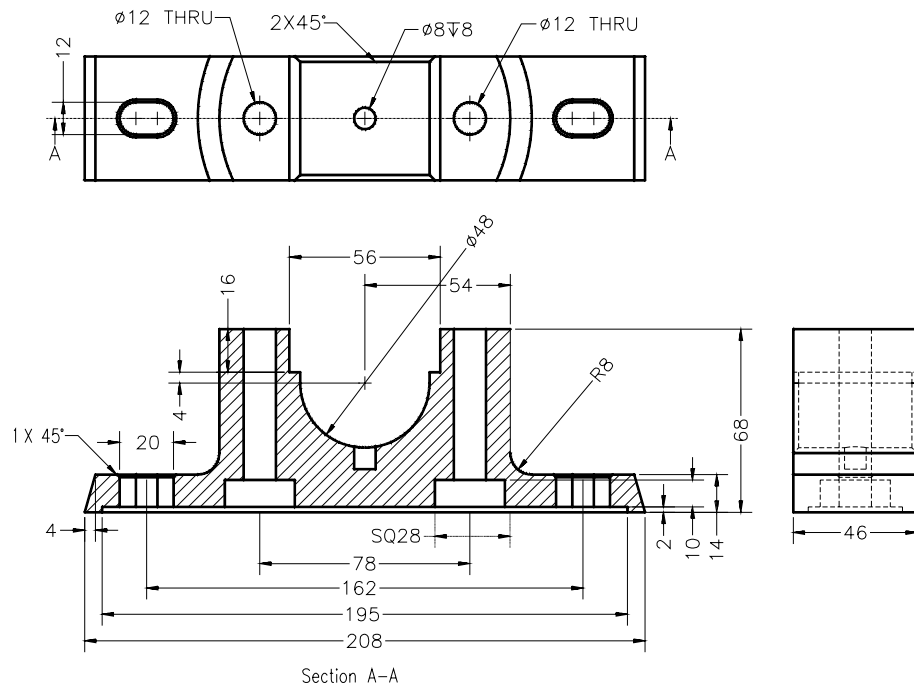


Figure 10-82 Views and dimensions of Casting

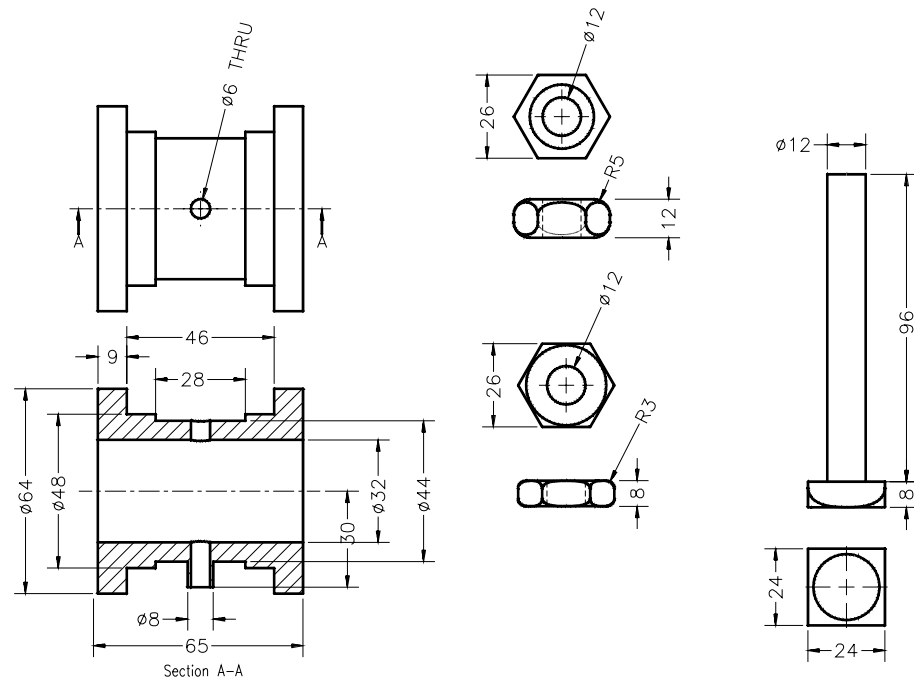


Figure 10-83 Views and dimensions of Brasses, Nut, Lock Nut, and Bolt

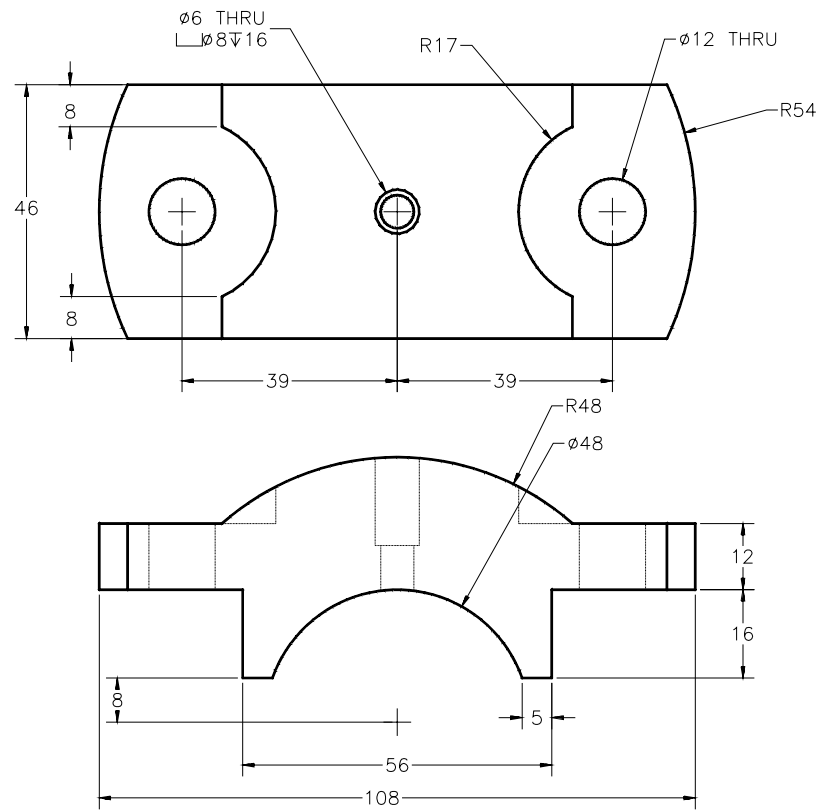


Figure 10-84 Views and dimensions of Cap

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. T, 6. T, 7. Rotate Component, 8. By Delta XYZ, 9. Coincident, 10. Along Entity