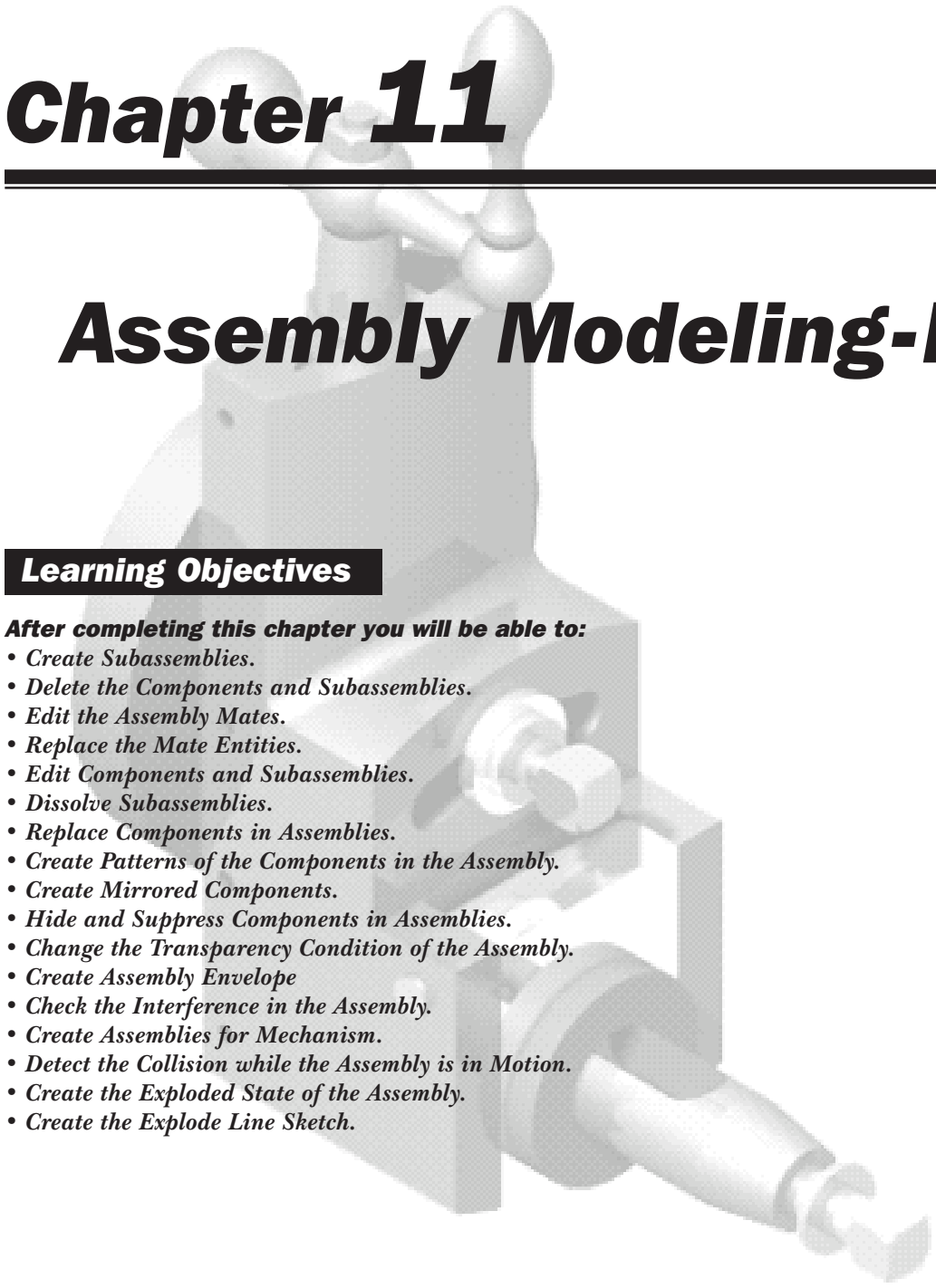
Chapter 11

Assembly Modeling-II



Learning Objectives

After completing this chapter you will be able to:

- *Create Subassemblies.*
- *Delete the Components and Subassemblies.*
- *Edit the Assembly Mates.*
- *Replace the Mate Entities.*
- *Edit Components and Subassemblies.*
- *Dissolve Subassemblies.*
- *Replace Components in Assemblies.*
- *Create Patterns of the Components in the Assembly.*
- *Create Mirrored Components.*
- *Hide and Suppress Components in Assemblies.*
- *Change the Transparency Condition of the Assembly.*
- *Create Assembly Envelope*
- *Check the Interference in the Assembly.*
- *Create Assemblies for Mechanism.*
- *Detect the Collision while the Assembly is in Motion.*
- *Create the Exploded State of the Assembly.*
- *Create the Explode Line Sketch.*

CREATING SUBASSEMBLIES

In the previous chapter, you learnt to place the components in the assembly document and apply the assembly mates to the components. In this chapter you will learn how to create the subassemblies and place the subassemblies in the main assembly.

For placing the subassemblies in the main assembly, there are two approaches that are followed in the assembly design environment. These two approaches are discussed next.

Bottom-up Subassembly Design

In the **Bottom-up Subassembly** design approach, the subassembly is created in the assembly environment and then it is saved as an assembly file. When you need to place the subassembly in the main assembly, open the main assembly document and choose **Insert > Component > From File** from the menu bar. The **Open** dialog box is displayed; choose **Assembly** (**.asm, *.sldasm*) from the **Files of Type** drop-down list. All the assemblies saved in the current location will be displayed in the display area. Select the subassembly from the display area and choose the **Open** button from the **Open** dialog box. Select a point in the drawing area to place the subassembly. Now, using the assembly mates, assemble the subassembly with the main assembly. Figure 11-1 shows a subassembly of piston and articulated rod. Figure 11-2 shows the main assembly of the piston and master rod. Figure 11-3 shows the main assembly after assembling the subassembly with the main assembly.

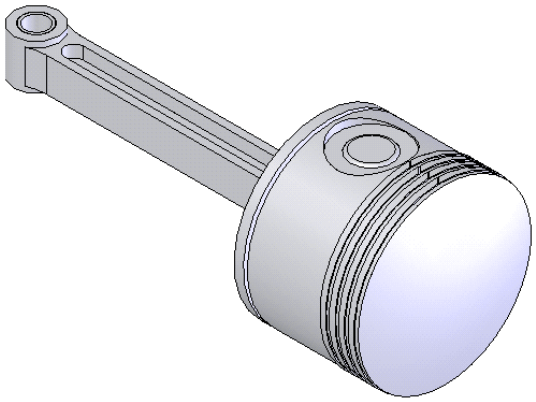


Figure 11-1 *A subassembly*

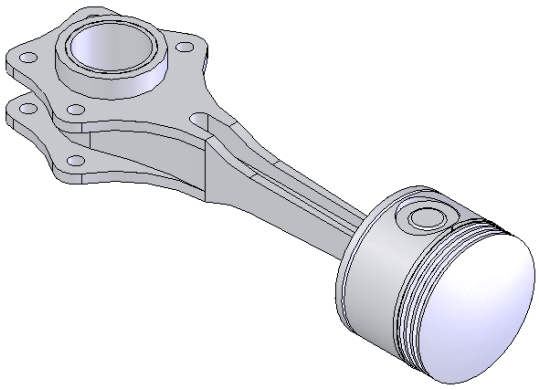


Figure 11-2 *A main assembly*

Top-down Subassembly Design

The **Top-down Subassembly** design approach is the most flexible subassembly design approach. In this, you create a new subassembly in the main assembly document. This approach is generally used in conceptual design or while managing a large assembly. To create a new subassembly in the assembly document, choose **Insert > Component > New Assembly** from the menu bar. The **Save As** dialog box is displayed; save the subassembly in the current location. You can drag and place the components in the new subassembly from the **FeatureManager Design Tree**. You will learn more about this approach while discussing the editing of assemblies and subassemblies.

Tip. *You can also place a subassembly in the main assembly using the drag and drop method that was discussed in the previous chapter.*

When you place a subassembly in the main assembly, an assembly icon will be displayed with the name of the subassembly in the ***FeatureManager Design Tree****. If you expand the subassembly in the* ***FeatureManager Design Tree****, all the parts assembled in the subassembly will be displayed.*

You can also create a subassembly of the components that are already placed in an assembly file. Press and hold down the CTRL key and select the components either from the drawing area or from the ***FeatureManager Design Tree****. Invoke the shortcut menu and choose* ***Form New Sub-assembly Here*** *from the menu bar or choose* ***Assembly from [Selected] Components*** *from the menu bar. The* ***Save As*** *dialog box is displayed; you need to enter the name of the subassembly and save it in the current location. You will observe that the selected components will be combined and an assembly icon will be displayed in the* ***FeatureManager Design Tree****.*

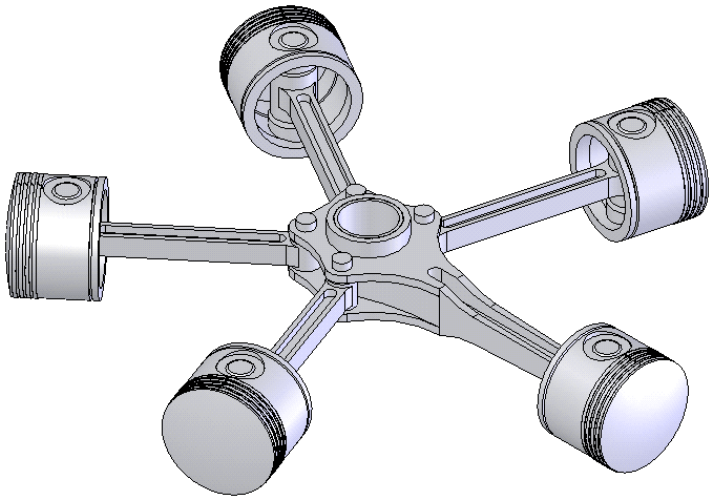


Figure 11-3 *Main assembly after assembling the subassemblies*

DELETING COMPONENTS AND SUBASSEMBLIES

After creating the assembly, at a certain stage of your design cycle you may need to delete any component or the subassembly. To delete a component of the assembly, select the component either from the drawing area or from the **FeatureManager Design Tree**. Invoke the shortcut menu and expand it; choose the **Delete** option from the shortcut menu. You can also delete the selected component by pressing the DELETE key from the keyboard. When you delete a component, the **Confirm Delete** dialog box is displayed. The name of the component and the items dependent on that component are displayed in this dialog box. Choose the **Yes** button from the **Confirm Delete** dialog box.

If you need to delete the subassembly, select the subassembly from the **FeatureManager Design Tree** and press the DELETE key. The **Confirm Delete** dialog box is displayed, choose the **YES** button from this dialog box. You should note that when you delete the subassembly, all the components of the subassembly are deleted.

EDITING ASSEMBLY MATES

Generally, after creating the assembly or during the process of assembling the components, you need to edit the assembly mates. The editing operations that can be performed on the assembly mates are modifying the type of assembly mate, modifying the angle and offset values, changing the component to which the mate was applied, and so on. In SolidWorks, for editing the mates, you first need to expand the **Mates** option available at the bottom of the **FeatureManager Design Tree**. Now, select the mate that you need to modify and invoke the shortcut menu. Choose the **Edit Definition** option from the shortcut menu. The **Mate PropertyManager** will be displayed. The name of the **Mate PropertyManager** will depend on the name and sequence of the mate applied. Figure 11-4 shows the **Mate PropertyManager** to edit the **Concentric** mate. You can edit the entities to mate, type of mate, value of offset, value of angle, and so on using this **PropertyManager**.

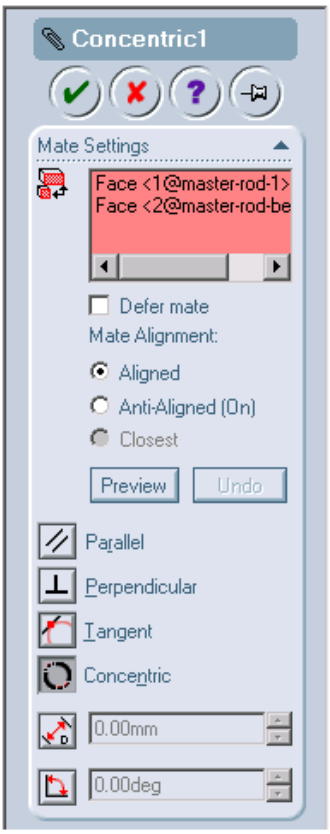


***Figure 11-4** The **Mate PropertyManager** to edit the **Concentric** mate*

Tip. *When you move the cursor on a mate in the* ***FeatureManager Design Tree****, the entities used in the mate are highlighted in red color in the drawing area. When you select the mate from the* ***FeatureManager Design Tree****, the entities used in the selected mate will be highlighted in green color in the drawing area.*

Replacing the Mated Entities

Toolbar: Assembly > Replace Mate Entities

As discussed, you can edit the mate entities using the **Mate PropertyManager**. The mate entities can also be modified using the **Mated Entities PropertyManager**. Select the **Mate** option from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Replace Mate Entities** option from the shortcut menu. The **Mated Entities PropertyManager** is displayed as shown in Figure 11-5. You can customize the **Assembly** toolbar and place the **Replace Mate Entities** button in the **Assembly** toolbar.

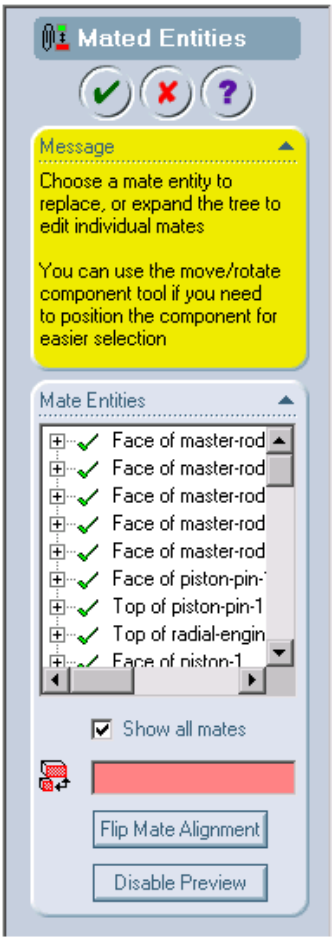


Figure 11-5 The ***Mated Entities PropertyManager***

When you invoke the **Mated Entities PropertyManager**, you are prompted to select the entity to be replaced. Select the entity from the **Mate Entities** display area. You can also expand the entity tree to edit the individual mate. After selecting, the selected face will be highlighted in green color in the drawing area. The name of the selected the entity will be displayed in the **Replacement Mate Entity** display area. You are prompted to select entity to be mated. Select the entity that will replace the previously selected entity. If the selected entity overdefines the mate, or the mating is not possible between the entities then the **SolidWorks** dialog box will be displayed. You will be informed about the possible cause of error.

The **Flip Mate Alignment** button is used to flip the direction of mate. The **Disable Preview** button is used to disable the preview of the assembly after replacing the mate entity. The **Show All** check box is used to display all the mated entities.

Tip. *Select the **Mate** option from the **FeatureManager Design Tree**, and invoke the shortcut menu. Choose the **Parent/Child Relationship** option from the shortcut menu. The **Parent Child Relationship** dialog box will be displayed. You can display the child and parent relationship of any component placed in the assembly using the **Parent Child Relationship** dialog box.*

EDITING THE COMPONENTS

Toolbar:	Assembly > Edit Part
Menu:	Edit > Part

After placing and mating the components in the assembly document, at some stage of design cycle you may need to edit the components. The editing of components includes editing the features, editing the sketches, and editing the sketch planes. For editing the components, you first need to select the component and invoke the part modeling environment in the assembly document. Select the component either from the drawing area or from the **FeatureManager Design Tree**. Now, choose the **Edit Part** button from the **Assembly** toolbar. The part modeling environment will be invoked and the complete assembly except the selected component will be displayed in transparent. The name of the component to be edited will be displayed in red in the **FeatureManager Design Tree**. Select the desired feature to edit, and invoke the shortcut menu. The **PropertyManager** relative to that feature will be displayed and you can easily edit the parameters of the feature. You can also add new features to the component. This type of edit is technically termed as **Editing in the Context of Assembly**. After editing the component, again choose the **Edit Part** button from the **Assembly** tool to return to the assembly environment.

Note

*If you need to edit components separately in their part documents, select the component and invoke the shortcut menu. Choose **Open**"part name" from the shortcut menu. The part document of the selected component will be opened. You can edit the component individually in the part document. After editing the component, save the part and close the part document and return to the assembly document. The **SolidWorks 2003** dialog box will be displayed as shown in Figure 11-6. This dialog box prompts you that the models contained within the assembly have changed. Would you like to rebuild the assembly now? Choose the **Yes** button from this dialog box.*

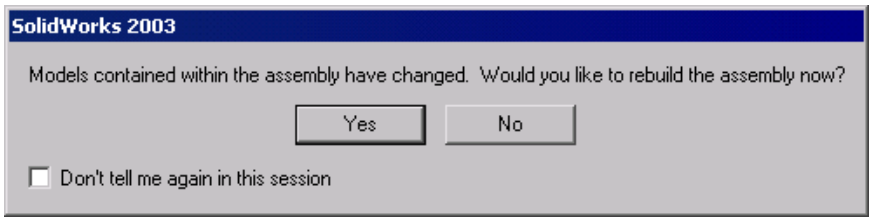


Figure 11-6 *The* ***SolidWorks 2003*** *dialog box*

Tip. *You can also modify the dimensions of a component assembled or placed in the assembly by double-clicking the feature of that component. All the dimensions of that feature will be displayed in the drawing area. Invoke the* ***Modify*** *dialog box by double-clicking the dimension to modify. Enter the new dimension in the* ***Modify*** *dialog box and choose the ENTER key from the key board. Since the dimension is modified, but the geometry of the feature has not changed, therefore, you need to rebuild the assembly. Choose the* ***Rebuild*** *button from the* ***Standard*** *toolbar to rebuild the entire assembly. You can also rebuild the assembly using CTRL+B key from the keyboard.*

While in the part editing mode in the assembly document, you can use the ***Move/Size Features*** *tool to edit the features dynamically using editing handles.*

EDITING THE SUBASSEMBLIES

Toolbar:	Assembly > Edit Part
Menu:	Edit > Part

To edit the subassemblies, select the subassembly from the **FeatureManager Design Tree** and choose the **Edit Part** button from the **Assembly** toolbar. The entire assembly except the selected subassembly will be displayed in transparent. You can add components in the subassembly, modify the mates, and replace the components while in the editing mode. After editing the subassembly, choose the **Edit Part** button to exit the editing mode.

Note

For editing a component of the subassembly, select the component from the drawing area and invoke the shortcut menu. Choose the ***Edit Part*** *option from the shortcut menu. The part editing mode will be invoked in the assembly document.*

To select the component of the subassembly from the ***FeatureManager Design Tree****, you need to expand the subassembly. All the components assembled in that subassembly will be displayed when you expand the subassembly.*

DISSOLVING THE SUBASSEMBLY

Dissolving the subassembly means the components of the subassembly become the components of the current assembly. When you dissolve a subassembly, the subassembly is removed from the main assembly and the components of the subassembly become the components of the main assembly. To dissolve a subassembly, select the subassembly from the **FeatureManager**

Design Tree and invoke the shortcut menu. Choose the **Dissolve Sub-assembly** option from the shortcut menu. The subassembly will be removed from the **FeatureManager Design Tree** and the components of the subassembly will be displayed as the components of the main assembly in the **FeatureManager Design Tree**.

REPLACING THE COMPONENTS

Toolbar: Assembly > Replace (*Customize to Add*)

Sometimes, in the assembly design cycle you may need to replace a component of the assembly with some other component. To replace a component, select the component to replace and invoke the shortcut menu. Choose the **Replace** option from the shortcut menu. The **Replace PropertyManager** will be displayed as shown in Figure 11-7. You can also invoke the **Replace PropertyManager** by choosing the **Replace** button from the **Assembly** toolbar after customizing it.

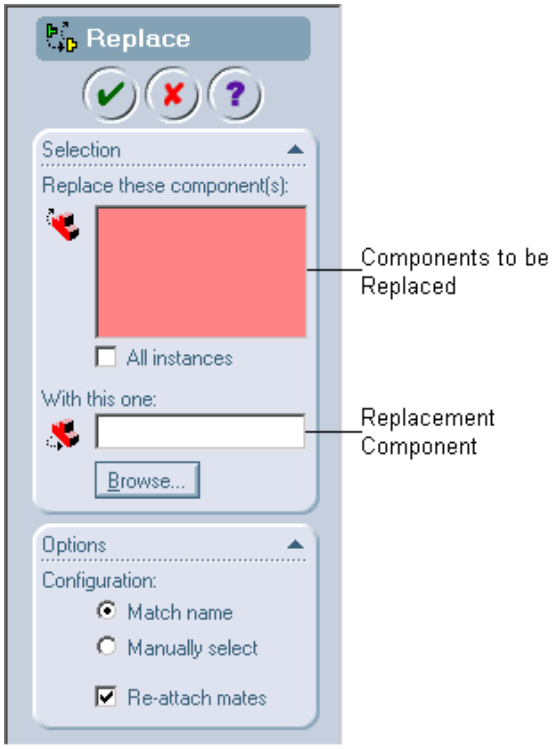


Figure 11-7** The **Replace PropertyManager

When you invoke the **Replace PropertyManager**, you are prompted to select components to be replaced. Select the component to be replaced. If the component is already selected before invoking this tool, then that component will be selected automatically. The name of the selected component is displayed in the **Components to be Replaced** display area. Next, you need to specify the replacement component. Choose the **Browse** button from the **Selection** rollout; the **Open** dialog box will be displayed. Select the replacement component and choose the

Open button from the **Open** dialog box. The name and location of the replacement component is displayed in the **Replacement Component** area.

The **Re-attach mates** check box is selected by default. This option is used to invoke the **Mate Entities PropertyManager** after you choose the **OK** button from the **Replace PropertyManager**. Using the **Mate Entities PropertyManager**, you can replace the mate entities.

If this check box is cleared, the **Rebuild Errors** dialog box is displayed after you exit the **Replace PropertyManager**. The **Rebuild Errors** dialog box informs you the name of the mates that contain errors. Therefore, you need to redefine the mates. For redefining the mates, expand the **Mates** option from the **FeatureManager Design Tree**. Select the mate that contains symbol on the left. Invoke the shortcut menu and choose the **Edit Definition** option. The **Mate PropertyManager** will be displayed and you can edit the mate entities.

The **All instances** check box available in the **Selection** rollout is used to replace all the instances of the selected component. The options available in the **Configuration** area are used to define the selection procedure of the configurations. You will learn more about configurations later.

Figure 11-8 shows the assembly in which the bolt is to be replaced by a pin. Figure 11-9 shows the faces of the pin to be used as mate entities after replacing the component. Figure 11-10 shows the isometric view of the assembly after bolts are replaced by pins.

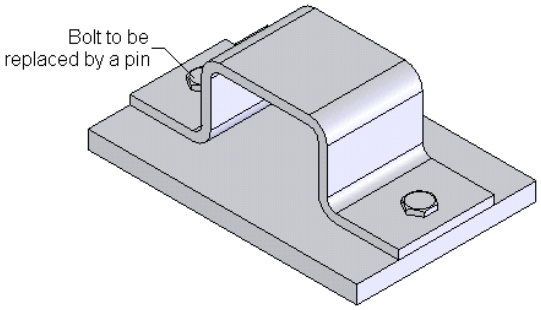


Figure 11-8 *Bolt to be replaced by a pin*

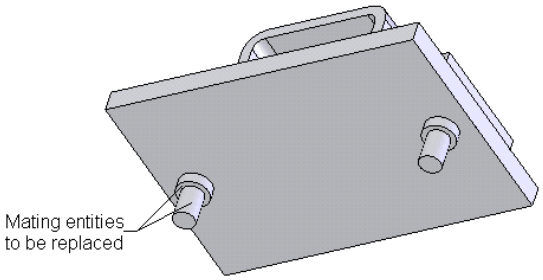


Figure 11-9 *Mating entities to be replaced*

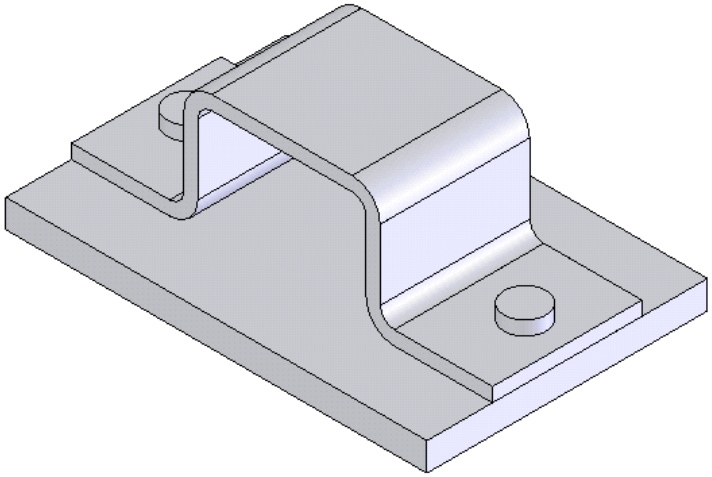


Figure 11-10 *Bolt replaced by pin in the assembly*

Tip. *If* ⚠ *symbol is displayed on the* ***Mates*** *option in the* ***FeatureManager Design Tree****, this means that this mate group has some errors. Expand the mate group; the mate that displays* ⚠ *sign will have some errors. Select that mate and invoke the shortcut menu. Choose* ***Whats Wrong?*** *from the shortcut menu to display the* ***Rebuild Errors*** *dialog box. The possible cause of error will be displayed in the* ***Rebuild Errors*** *dialog box.*

If you select the ***Mate Diagnostic*** *option from the shortcut menu, the* ***Diagnostics PropertyManager*** *will be displayed. The* ***Analyze*** *button is used to display the entities that are the cause of errors in the mate. Choose the* ***Analyze*** *button; the name of the mate and the entity will be displayed in the* ***Analyze Problem*** *rollout.*

Select the ***Mates*** *mategroup with error symbol from the* ***FeatureManager Design Tree*** *and invoke the shortcut menu. You can choose the* ***Whats Wrong?*** *or* ***Mate Diagnostics*** *option to analyze all the mates having errors.*

CREATING PATTERNS OF THE COMPONENTS IN AN ASSEMBLY

Menu: Insert > Component Pattern

While working in the assembly design environment of SolidWorks, you may need to assemble more than one instance of the component about a specified arrangement. Consider the case of a flange coupling where you have to assemble eight instances of the nut and bolt to fasten the coupling. Therefore, you need to make the instances of the nut and bolt and then assemble all the eight nuts and the eight bolts manually. However, this is a very tedious and time-consuming precess. Therefore, to reduce the time in the assembly design cycle,

SolidWorks has provided a tool to create the patterns of the components. There are two types of component patterns provided in SolidWorks. These are discussed next.

Derived Pattern

The **Derived Pattern** option is used to pattern the instances of the components using an existing pattern feature. For creating a derived pattern, choose **Insert > Component Pattern** from the menu bar. The **Pattern Type** dialog box is displayed as shown in Figure 11-11 and the **Use an existing feature pattern (Derived)** radio button is selected by default.

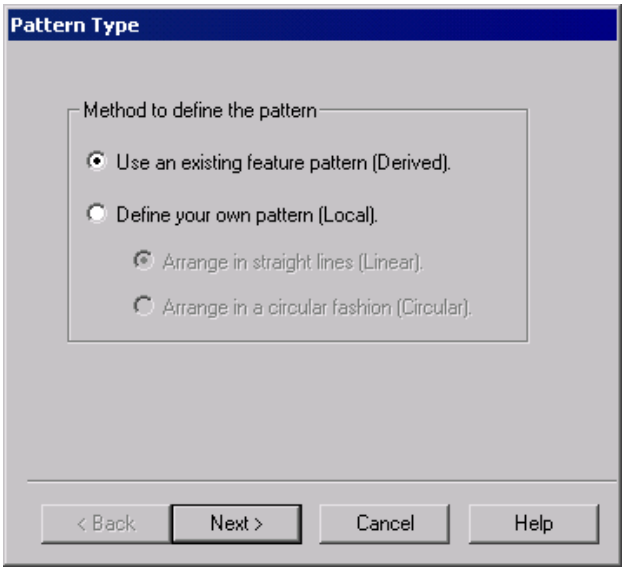


Figure 11-11 *The* ***Pattern Type*** *dialog box*

Choose the **Next** button from the **Pattern Type** dialog box. The **Derived Component Pattern** dialog box will be displayed as shown in Figure 11-12. The selection mode is active in the **Seed Component(s)** area. Select the component or components to pattern from the drawing area or from the **FeatureManager Design Tree**. Now, click once in the **Pattern Feature** area to invoke the selection mode and select any one instance of an existing pattern feature. Choose the **Finish** button from the **Derived Component Pattern** dialog box.

Figure 11-13 shows the components to be selected to pattern and an instance of an existing pattern feature to be selected. Figure 11-14 shows the assembly after creating the derived pattern feature.

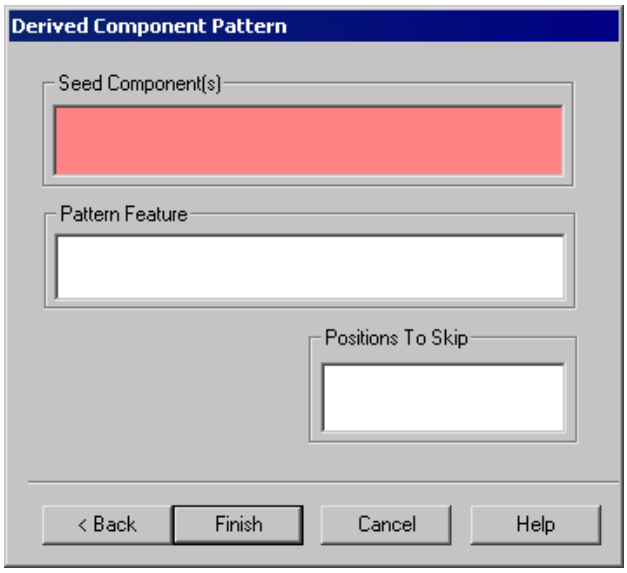


Figure 11-12 *The* ***Derived Component Pattern*** *dialog box*

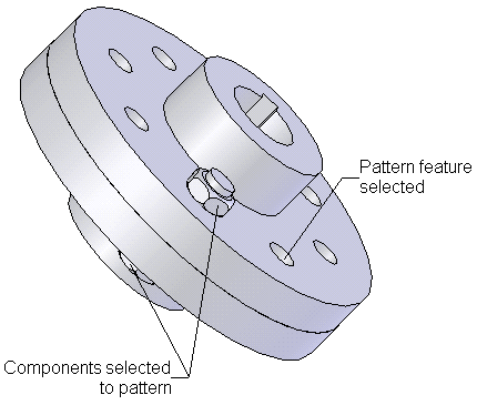


Figure 11-13 *Components and the instance of the pattern feature to be selected*

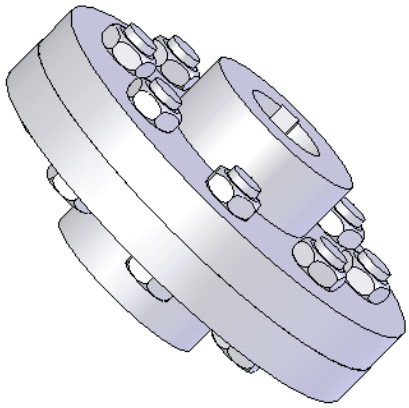


Figure 11-14 *Assembly after creating the derived component pattern*

Note

If you change the number of instances of the pattern feature using which the derived component pattern is created, the number of instances of the component pattern are automatically modified. This displays the associative and parametric nature of the derived component pattern.

After editing the number of entities in the pattern feature, you may have to choose the ***Rebuild*** *button.*

Remember that if the feature that was used to assemble the seed component is deleted while reducing the number of features in the pattern, you will have to modify the mates. This is because the original feature on which the mates were applied does not exist any more and so the mates give an error.

Tip. *If you need to skip some instances while creating the pattern of a component, then first create the component pattern as discussed above. After creating the pattern you will observe that the* ***DerivedCirPattern1*** *feature will be displayed in the* ***FeatureManager Design Tree****. If the derived pattern creates a rectangular pattern of the components, the name of the feature will be* ***DerivedLPattern1****. The number at the last of the feature display the sequence number of the derived pattern feature.*

When expand this feature, all the instances of the patterned component are displayed. Select the instance to be deleted from the ***FeatureManager Design Tree*** *and press the DELETE key. The* ***Confirmation*** *dialog box will be displayed; choose the* ***Yes*** *button from the dialog box.*

If you need to restore the deleted pattern instance, then select the derived pattern from the ***FeatureManager Design Tree*** *and invoke the shortcut menu. Choose the* ***Edit Definition*** *option from the shortcut menu. The* ***Derived Component Pattern*** *dialog box will be displayed. The number of the deleted instance will be displayed in the* ***Positions To Skip*** *area. Select the instance from the* ***Position To Skip*** *area and press the DELETE key to restore the deleted pattern instance. Choose the* ***Finish*** *button from the* ***Derived Component Pattern*** *dialog box.*

Local Pattern

You can also create the patterns of the components individually even if there is no pattern feature. This type of component pattern is known as local pattern. You can create two types of local pattern: first is the linear pattern and the other is the circular pattern. Both the types of local patterns are discussed next.

Linear Pattern

For creating the local linear pattern, invoke the **Pattern Type** dialog box. Select the **Define your own pattern (Local)** radio button from the **Pattern Type** dialog box. The **Arrange in straight lines (Linear)** radio button and the **Arrange in a circular fashion (Circular)** radio buttons are invoked. The **Arrange in straight lines (Linear)** radio button is selected by default. Choose the **Next** button from the **Pattern Type** dialog box. The **Local Component Pattern** dialog box for the linear pattern is displayed as shown in Figure 11-15.

The options available in this dialog box are used to create a linear local component pattern. The options of creating the linear pattern are the same as those discussed in Chapter 9.

Circular Pattern

For creating a local circular component pattern, invoke the **Pattern Type** dialog box. Select the **Define your own pattern (local)** radio button; next select the **Arrange in a circular fashion (Circular)** radio button. The **Local Component Pattern** dialog box for creating a circular component pattern will be displayed as shown in Figure 11-16.

The options available in this dialog box to create a circular pattern are the same as those discussed in Chapter 7 to create a circular pattern feature.

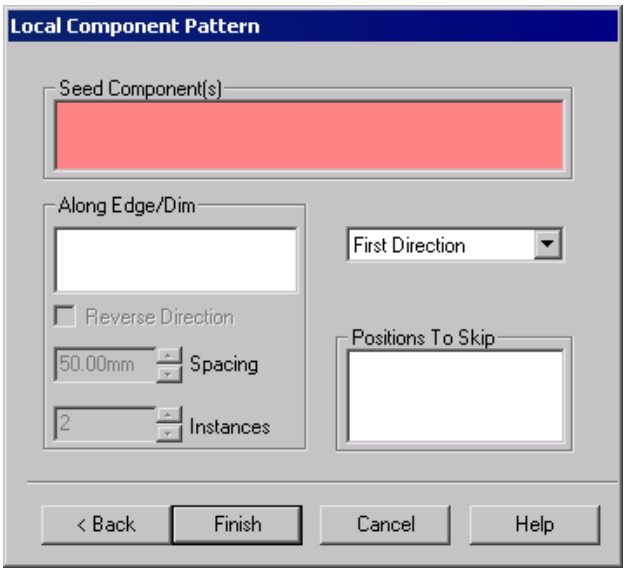


Figure 11-15 *The* ***Local Component Pattern*** *dialog box for linear pattern*

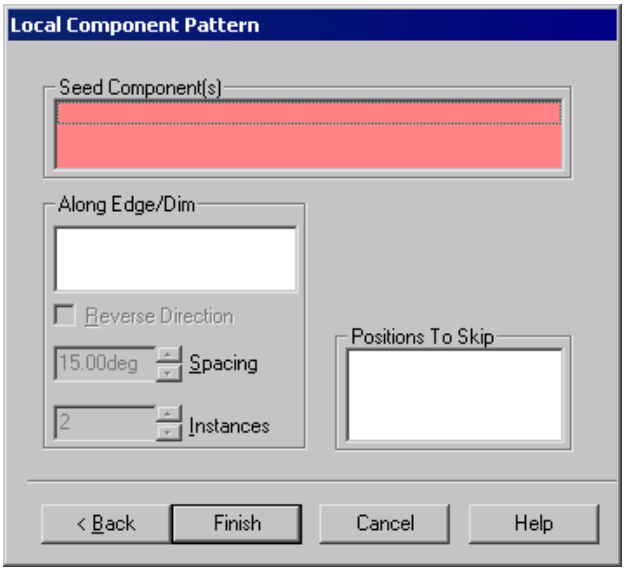


Figure 11-16 *The* ***Local Component Pattern*** *dialog box for circular pattern*

Tip. *If one instance of the component pattern is modified or edited, the other instances of the component will also be modified.*

You can also create the component pattern of a component pattern feature.

MIRRORING THE COMPONENTS

Menu: Insert > Mirror Components

In the assembly design environment of SolidWorks, you can also mirror a component to place the new instance of the component in the assembly document. For mirroring the component, choose **Insert > Mirror Components** from the menu bar. The **Mirror Components PropertyManager** is displayed as shown in Figure 11-17.

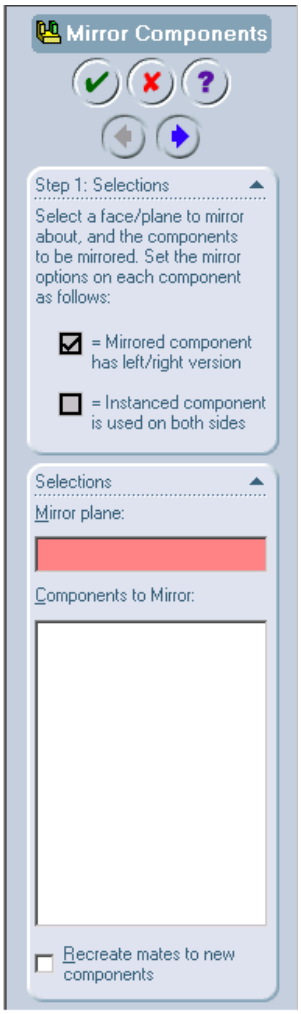


Figure 11-17** The **Mirror Components PropertyManager

Select the planar face or plane that act as mirror plane as shown in Figure 11-18. Now, select the component to mirror; you will observe that the name of the component with a check box is displayed in the **Components to Mirror** area. Leave this check box cleared and choose the **Next** button from the **Mirror Components PropertyManager**. The preview of the mirrored

component is displayed as shown in Figure 11-19. The **Orientation** rollout is displayed and using the options available in this rollout you can change the orientation of the mirrored component. Choose the **OK** button from the **Mirror Components PropertyManager**. The mirrored instance of the selected component is displayed in the drawing area as shown in Figure 11-20.

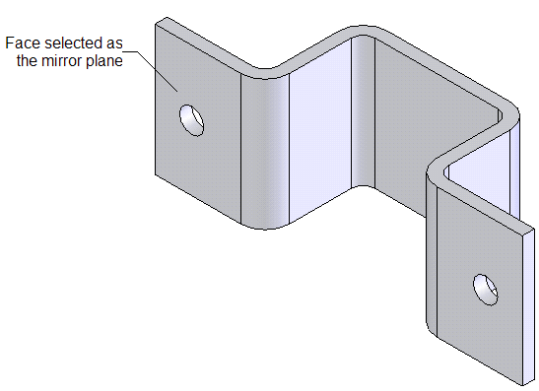


Figure 11-18 *Face to be selected as mirror plane*

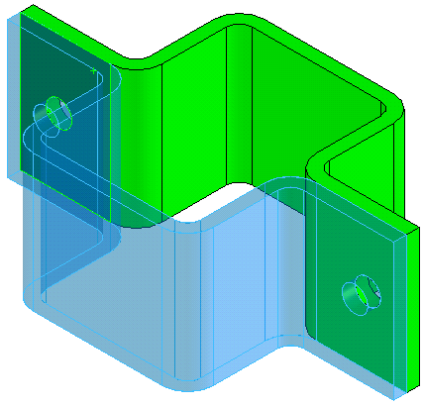


Figure 11-19 *Preview of the mirrored component*

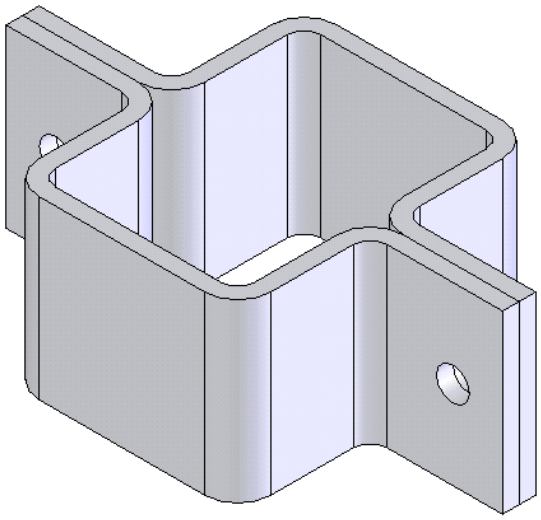


Figure 11-20 *Instance created by mirroring the component*

If you select the check box available with the name of the selected component in the **Components to Mirror** area, a new component that is mirrored along the selected mirror plane and saved in a specified location is created. Select the check box available along the name of the selected component and choose the **Next** button. The **Filenames** rollout is displayed; choose the **Browse** button available in the **Filenames** rollout. This button is used to specify the name and location where the file will be saved. Using the **Place files in one folder** check box, you can save all the mirrored instances in one folder. Using the **Add** drop-down list, you can add a suffix or a prefix to the file name.

SIMPLIFYING THE ASSEMBLIES USING VISIBILITY OPTIONS

When you are assembling the components, whether it be a large assembly or a small assembly, you may need to simplify the assembly using the visibility options. By simplifying, you can hide the components at any stage of the design cycle. You can also set the transparency of any component for simplifying the assembly. You can also suppress and unsuppress the components at any stage of the design cycle. The various methods of simplifying the assembly are discussed next.

Hiding the Components

Toolbar:	Assembly > Hide/Show Component
Menu:	Edit > Hide > This Configuration

If you need to hide the component placed in the assembly, choose the component from the drawing area or from the **FeatureManager Design Tree**. Using the CTRL key you can select more than one component to hide. Right-click to invoke the shortcut menu and choose the **Hide Components** option from the shortcut menu. The component will disappear from the drawing area. The icon of the component is displayed in transparent in the **FeatureManager Design Tree**. To unhide the hidden component, select the icon of the component from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Show Components** option from the shortcut menu. The hidden component will be redisplayed in the drawing area. You can also use the **Hide/Show Component** button from the **Assembly** toolbar to hide or show the components. But if this button is not available by default, then you have to customize the **Assembly** toolbar to add this button.

Tip. *You will observe that the **Lightweight** check box is provided in the **Open** dialog box. If you select this check box before opening the assembly file, the assembly will be opened with lightweight components.*

*A lightweight component is a component in which the feature information is available in the part document and only the graphical representation of the component is displayed in the assembly document. Therefore, the assembly environment becomes light. An icon of a lightweight component is displayed as a feather attached to the component icon in the **FeatureManager Design Tree**.*

*To get the feature information of the lightweight component, you need to resolve the component to normal state. Therefore, select the component from the drawing area or from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Set to Resolve** option from the shortcut menu. If you need to set a resolved component to a lightweight component, select the component and invoke the shortcut menu. Choose **Set to Lightweight** from the shortcut menu.*

Suppressing the Components

Toolbar: Assembly > Change Suppression State
Menu: Edit > Suppress > This Configuration

You can also suppress the components placed in the assembly to simplify the assembly representation. To suppress a component, select the component from the drawing area or from the **FeatureManager Design Tree**. Invoke the shortcut menu and choose **Suppress**. The component will not be displayed in the assembly document and the icon of the suppressed component will be displayed in grey in the **FeatureManager Design Tree**. To unsuppress the suppressed component, select the component to be resolved from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Set to Resolve** option from the shortcut menu.

You can also suppress the component using the **Change Suppression State** button from the **Assembly** toolbar. If this button is not available in the **Assembly** toolbar, you need to customize the toolbar. Using the **Change Suppression State** button, you can suppress the selected component as well as set the selected component to lightweight. When you select the component and choose the **Change Suppression State** button from the **Assembly** toolbar, a flyout is displayed. This flyout has three buttons. The first button is the **Suppress** button, the second button is the **Lightweight** button, and the third button is the **Resolve** button. The **Resolve** button is used to set the suppressed or lightweight components to resolve state.

Changing the Transparency Conditions

In SolidWorks you can change the transparency of the components or selected faces to simplify the assembly. First, you will learn how to change the transparency of the components. Select the component to change its transparency and invoke the shortcut menu. Choose the **Component Properties** option from the shortcut menu. The **Component Properties** dialog box will be displayed. Choose the **Color** button available in this dialog box. The **Assembly Instance Color** dialog box is displayed as shown in Figure 11-21.

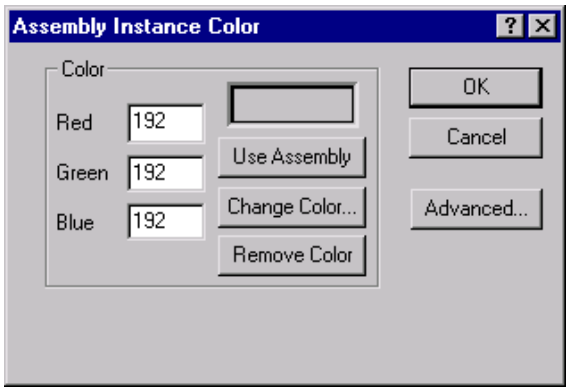


Figure 11-21 *The* ***Assembly Instance Color*** *dialog box*

Choose the **Advanced** button from this dialog box. The **Advanced Properties** dialog box is displayed as shown in Figure 11-22. You can set the transparency of the component from the **Transparency** slider. You can also set the various other advanced color settings such as **Ambient**, **Diffuse**, **Specularity**, **Shininess**, and **Emission** from this dialog box.

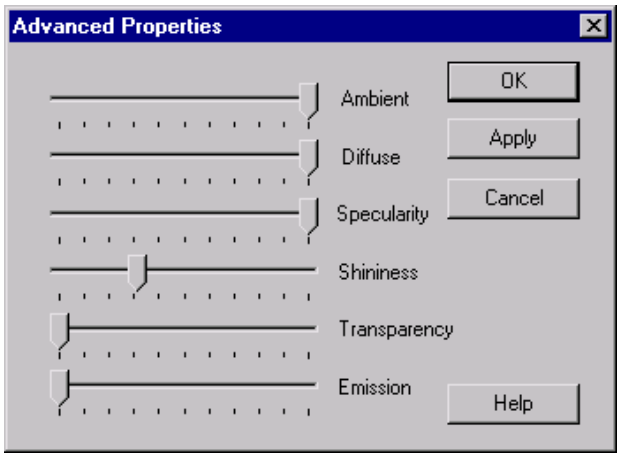


Figure 11-22 *The **Advanced Properties** dialog box*

You can also change the transparency of a selected face. To set the transparency of a face, select the face and invoke the shortcut menu and choose the **Face Properties** option from the shortcut menu. The **Entity Property** dialog box is displayed. Choose the **Advanced** button from this dialog box to invoke the **Advanced Properties** dialog box. Using this dialog box you can set the transparency of the selected face.

Tip. *The face that is made transparent cannot be selected by clicking. You need to select some other faces and then choose the **Select Other** option from the shortcut menu. Right-click until the desired face is highlighted. Once the desired face is highlighted, left-click to select it. Now, right-click anywhere in the drawing window to display the shortcut menu.*

CREATING THE ASSEMBLY ENVELOPE

Menu: Insert > Envelope > New/ From File

The assembly envelope is an option that is used to simplify the selection of the components in a complex assembly. An envelope is basically a box that is created around a specified component, see Figure 11-23. You can select all the components that lie in and around this box. To create an assembly envelope, choose **Insert** > **Envelope** > **New** from the menu bar. The **Save As** dialog box is displayed; save the envelope in the location where all the other components of that assembly are saved. The select cursor is replaced by the part placement cursor. Place the component on any one of the assembly default planes or on a planar face of the components of the assembly. The **Edit Part** button is chosen and the entire assembly is changed to transparent. The sketching environment is invoked with the placement plane or face of the assembly envelope as the sketching plane. You can exit the sketching plane and select another sketching plane for creating the sketch of the envelope. Create the sketch of the

envelope. Generally, the envelope is in the form of a cube or a cuboid, i.e., an extruded rectangle or extruded square. Create the sketch of the envelope and extrude the sketch. The sketch will be extruded on one side or on both the sides of the sketching plane, depending on the geometry of the assembly. Choose the **Edit Part** button from the **Assembly** toolbar to exit the part modeling environment. The assembly envelope will be displayed in the transparent blue color. The icon of the envelope is displayed with the name of the envelope in the **FeatureManager Design Tree**. Figure 11-23 shows an assembly with an assembly envelope.

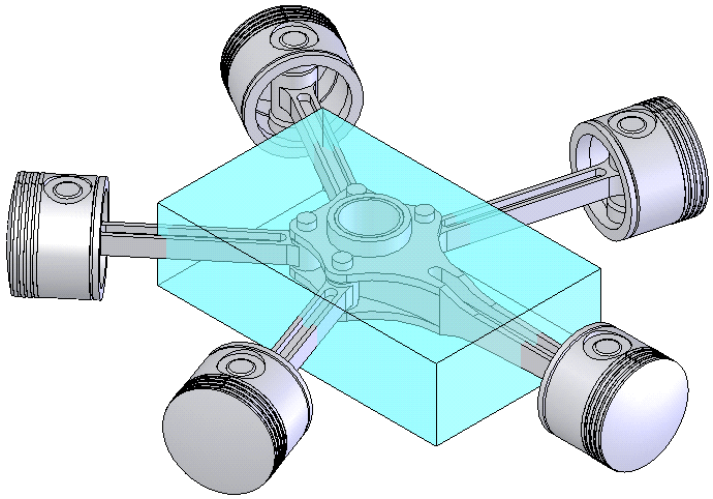


Figure 11-23 *An assembly with an assembly envelope*

Selecting the Components Using the Assembly Envelope

For selecting the components using the assembly envelope, choose the **ConfigurationManager** tab from the bottom of the **FeatureManager Design Tree** to invoke the **ConfigurationManager**. Select **Envelope1** from the **ConfigurationManager** and invoke the shortcut menu. Choose the **Select using envelope** option from the shortcut menu. The **Apply Envelope** dialog box is displayed as shown in Figure 11-24. The various options available in this dialog box are discussed next.

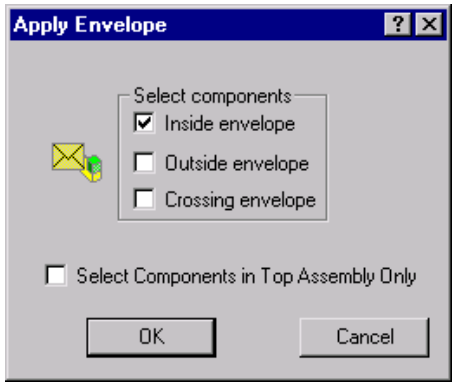


Figure 11-24 *The* ***Apply Envelope*** *dialog box*

Inside envelope

The **Inside envelope** check box is selected to select the components that are placed completely inside the assembly envelope.

Outside envelope

The **Outside envelope** check box is selected to select the components that are placed completely outside the assembly envelope.

Crossing envelope

The **Crossing envelope** check box is selected to select the components that are placed partially inside the envelope or that cross the boundary of the assembly envelope.

Select Components in Top Assembly Only

The **Select Components in Top Assembly Only** check box is selected to select the components of the top assembly only. In this case it considers the subassemblies as a single component. For example, consider a case in which a single part of subassembly is outside the assembly envelope. Now, if you select **Inside envelope** from the **Apply Envelope** dialog box, the entire subassembly will be selected.

Advanced Hide/Show the Components Using the Envelope

You can hide the components displayed in the assembly environment and also show the hidden components using the envelope. To hide/show the components using the envelope, select the envelope from the **ConfigurationManager** and invoke the shortcut menu. Choose the **Show/Hide using envelope** option from the shortcut menu. The **Apply Envelope** dialog box will be displayed as shown in Figure 11-25.

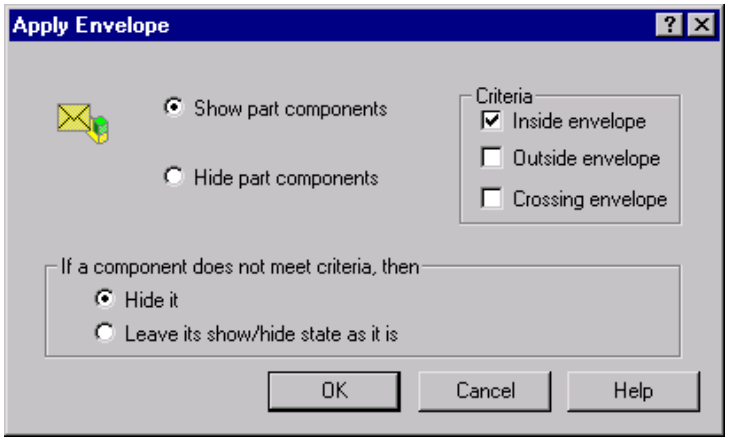


Figure 11-25 *The* ***Apply Envelope*** *dialog box*

By default, the **Show part components** radio button is selected in the **Apply Envelope** dialog box. The **Hide it** radio button is selected by default in the **If a component does not meet criteria, then** area. You can select the options from the **Criteria** area to define the components that should be affected. The options available in this area are discussed earlier. Therefore, with the current combination of selections the components that meet the criteria of the option

selected in the **Criteria** area will be shown as the other components will be defined in the hidden state. The **Hide part components** radio button is used to hide the components that meet the criteria of the option selected in the **Criteria** area. If the **Leave its show/hide state as it is** radio button available in the **If component does not meet criteria, then** area is selected, the current hide/show state of the components is maintained for the components that do not meet the option selected in the **Criteria area.**

CHECKING THE INTERFERENCES IN THE ASSEMBLY

After creating the assembly design, the first and the most essential step is to check the interference between the components of the assembly. If there is interference between the components, the components may not assemble properly after the they are out from the machine shop or tool room. Therefore, before sending the part file and assembly for the detailing and drafting purpose, it is essential to check the interference. To check the interference, choose **Tools > Interference Detection** from the menu bar. The **Interference Volumes** dialog box is displayed as shown in Figure 11-26.

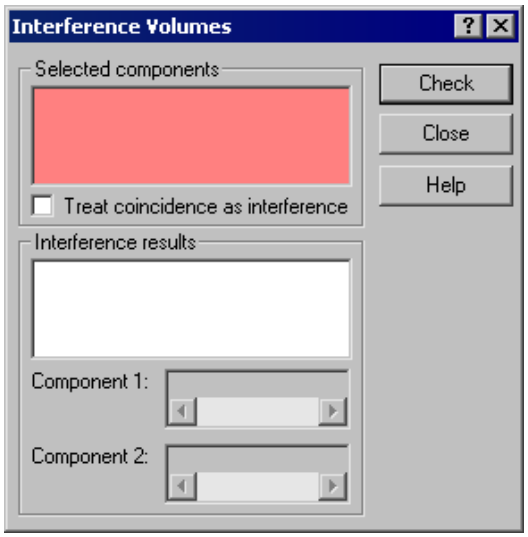


Figure 11-26 *The* ***Interference Volumes*** *dialog box*

The name of the current assembly is displayed in the **Selected components** display area. You can also check the interference between two or more than two components. To select the components, first clear the current selection set and select the components from the assembly. Choose the **Check** button from this dialog box to check the interference. If there is any interference between the components in the assembly, it will be displayed in the assembly and all the interferences are also displayed in the **Interference results** area. Select the interference from the **Interference** results area; the components between which the interference is detected will be displayed in the **Component 1** and **Component 2** display area. Also, as you select the interference, the components are highlighted in the assembly and the amount of interfering component is displayed in the assembly. The **Treat coincidence as interference** check box is used to display the coincident mates as interference. After analyzing the assembly you can edit or modify the part.

Tip. *You can also select a part created earlier as an envelope and place and assemble it in the assembly file for advanced selection or show/hide options. Choose* ***Insert > Envelope > From File*** *from the menu bar. The* ***Open*** *dialog box is displayed; browse and open the part to be used as envelope and assemble using the assembly mates.*

CREATING THE ASSEMBLIES FOR MECHANISM

As mentioned earlier, there are two types of assemblies. The first one is a fully defined assembly in which the relative movement of all the components is contained. The second type of assembly is that in which the components are not fully defined and some degree of freedom is kept unconstrained. As a result they can move in certain direction with respect to the surroundings of the assembly. This flexibility in turn helps you to create the mechanisms and then you can move the assembly to check the mechanism that you have designed. Consider the case of a Bench Vice in which you are assembling the vice jaw with the vice body. For this assembly to work, the linear movement of the Vice Jaw when placed on the vice body should be free. Therefore, while creating this assembly for mechanism, you should not apply the mates for constraining the linear motion of the Vice Jaw with respect to the vice body. Figure 11-27 shows the degree of freedom that has to be free to create an assembly for motion.

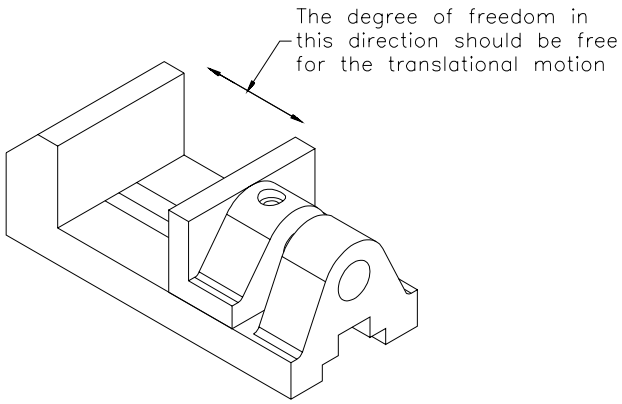


Figure 11-27 *Direction in which the degree of freedom should be free*

After creating the assembly for mechanism by defining minimum mates, invoke the **Move** tool. Select one of the face of the component that you need to move and drag the cursor to move the assembly. While moving the assembly for mechanism design, there are some options available for analyzing the assembly. These options of analyzing the assembly are discussed next.

Analyzing the Collisions Using the Collision Detection Tool

In SolidWorks, you can also analyze if there is any collision between the components of the assembly while the assembly is in motion. To analyze the collision when the assembly is in motion, invoke the **Move PropertyManager** and invoke the **Options** rollout as shown in

Figure 11-28. Select the **Collision Detection** radio button from the **Options** rollout. The **Check between** area is displayed; the options available in this area are used to specify the components between which the collision will be detected. The options available in the **Check between** area are discussed next.

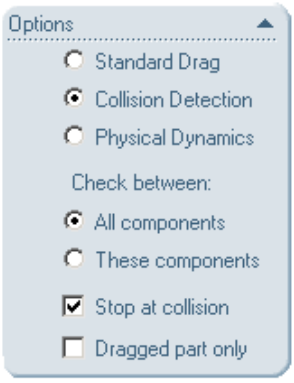


Figure 11-28 *The* ***Options*** *rollout of the* ***Move PropertyManager***

All components

The **All components** radio button is selected to check the collision between all the components when the assembly is in motion.

These components

The **These components** radio button is selected to check the collision only between the selected components when the assembly is in motion. When you select the **These components** radio button, the **Components for Collision Check** display area and the **Resume Drag** button are displayed in the **Options** rollout as shown in Figure 11-29. After selecting the components between which the collision has to be detected, the name of the components is displayed in the **Components for Collision Check** display area. After selecting the components, choose the **Resume Drag** button from the **Options** rollout and drag the cursor to move the assembly.

Stop at collision

The **Stop at collision** check box is selected to stop the motion of the assembly when one of the component collides with another component during the assembly motion.

Dragged part only

The **Dragged part only** check box is selected when you need to detect the collision only between the components that you selected to move. If you do not select this check box, the components that you selected to move and any other component that moves because of mates with the selected components are detected for collision.

After setting the options in the **Options** rollout, drag the assembly either using the **Move** tool or using the **Rotate** tool. You can also use the **Rotate** tool to move the assembly. To use the

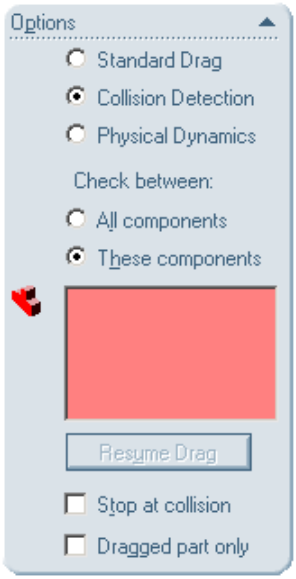


Figure 11-29 *The* ***Options*** *rollout of the* ***Move PropertyManager*** *with the* ***These components*** *radio button selected from the* ***Check between*** *area*

Rotate tool invoke the **Rotate** rollout and drag the assembly to move. If a component of the assembly collides with another component while the assembly is in motion, the faces of the components that collide with each other will be displayed in green color. If the **Stop at collision** check box is selected, the motion of the assembly will be stopped when one of the components collides with another component of the assembly.

Consider the assembly shown in Figure 11-30. In this assembly you need to move the slider in the given direction. Invoke the **Move PropertyManager** and select the **Collision Detection** radio button from the **Options** rollout. Drag the slider to move in the given direction. Figure 11-31 shows that the slider collides with the extrusion feature created in the vertical column of the base component. The faces of the component that collides is displayed in green color. If the **Stop at collision** check box is selected, you cannot move the component further after it collides with one of the components of the assembly.

Once the collision is detected in the assembly, you can edit and modify the components that collide during the assembly motion.

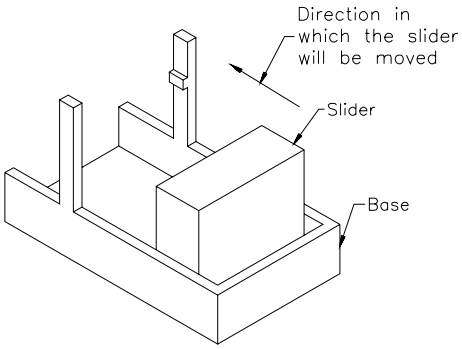


Figure 11-30 *Direction in which the slider will be moved inside the base*

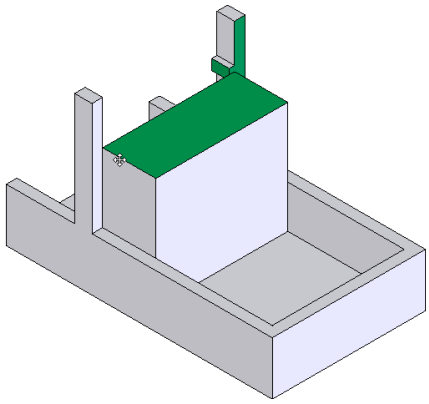


Figure 11-31 *The faces of the components highlighted in green after collision*

CREATING THE EXPLODED STATE OF THE ASSEMBLY

In SolidWorks, you can also create an exploded state of the assembly. The explode state of the assembly is created using the **Assembly Exploder** dialog box. To invoke this dialog box, first you need to invoke the **ConfigurationManager**. Select the **Default** option from the **ConfigurationManager** and invoke the shortcut menu. Choose the **New Explode View** option from the shortcut menu. The **Assembly Exploder** dialog box will be displayed as shown in Figure 11-32.

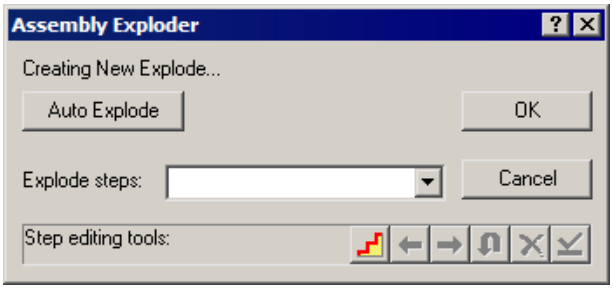


Figure 11-32 *The **Assembly Exploder** dialog box*

There are two options to create an exploded state of the assembly. The first option is to create the exploded state automatically. To create an automatic explode state, choose the **Auto Explode** button from the **Assembly Exploder** dialog box. The components of the assembly will be exploded and will be placed arbitrarily in the assembly document. Figure 11-33 shows an automatic exploded view of an assembly.

Creating the Systematic Explode State

To create a systematic explode state of the assembly, choose the **New** button from the **Assembly Exploder** dialog box. The **Assembly Exploder** dialog box will expand and some more options are displayed as shown in Figure 11-34.

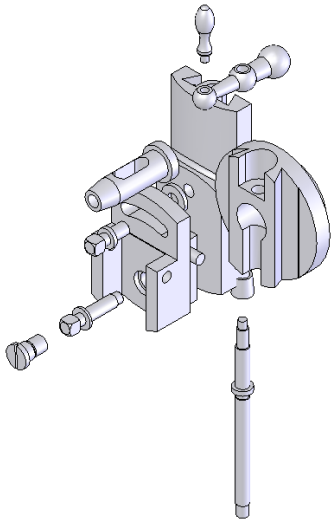


Figure 11-33 *An assembly exploded using the* ***Auto Explode*** *tool*

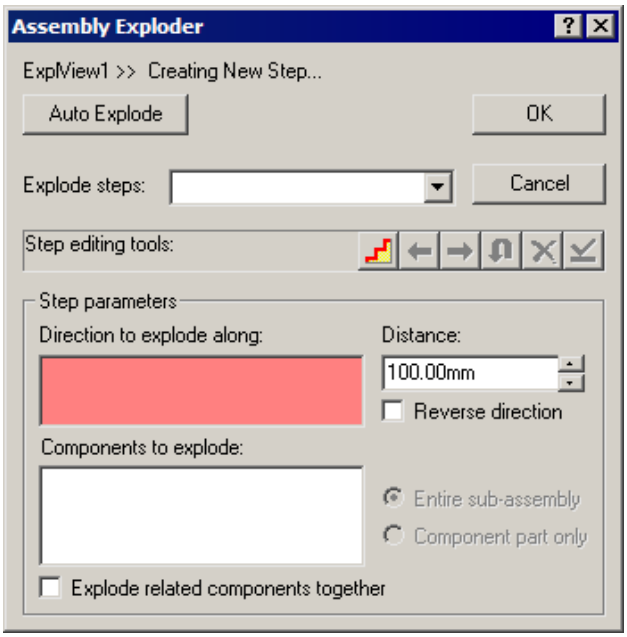


Figure 11-34 *The expanded* ***Assembly Exploder*** *dialog box*

As you expand this dialog box, you are prompted to create a new explode step. The selection is active in the **Direction to explode along** display area. Therefore, you need to select the direction in which you need to explode the component. Select an edge, face, plane, axis, or a sketch to define the direction along which the components will be exploded. After selecting the explode direction, the selection mode in the **Components to explode** area is activated. Select the components to be exploded in the current direction. Specify the explode distance using the **Explode** spinner. You can also reverse the direction of explode using the **Reverse**

direction check box. The **Explode related components together** check box is selected to explode all the components that related with the selected component using the assembly mates.

After exploding the components in a particular direction with same distance, you need to explode the other components. Therefore, you need to create another explode step. Choose the **New** button from the **Assembly Exploder** dialog box. Now, using the same procedure, explode the other set of components. You can create as many explode steps as you want while exploding the assembly.

After creating all the explode steps choose the **OK** button from the **Assembly Exploder** dialog box to complete the explode state creation. If you need to switch back to the unexplode state, expand the **Default** option from the **ConfigurationManager** and select **ExplView1**. Invoke the shortcut menu and choose **Collapse** from the shortcut menu. You can also edit each explode step by expanding **ExplView1** and select the explode step to be modified. Invoke the shortcut menu and choose the **Edit Definition** option from the shortcut menu. The **Assembly Exploder** dialog box is displayed and you can modify the parameters of the explode step. Figure 11-35 shows the systematic exploded view of an assembly.

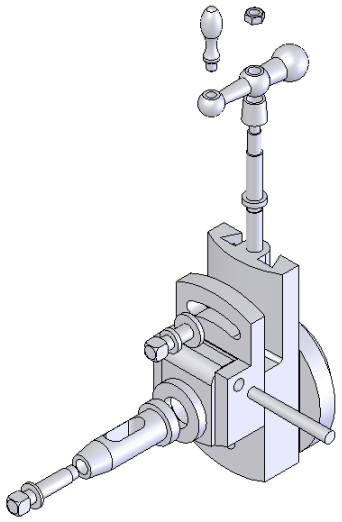


Figure 11-35 *The systematic exploded state of an assembly*

Creating the Explode Line Sketch

Toolbar:	Assembly > Explode Line Sketch
Menu:	Insert > Explode Line Sketch

Explode lines are the parametric axes that display the direction of explosion of the components in the exploded state. Figure 11-36 shows an exploded assembly with explode lines.

To create the explode line sketch, choose the **Explode Line Sketch** button from the **Drawing**

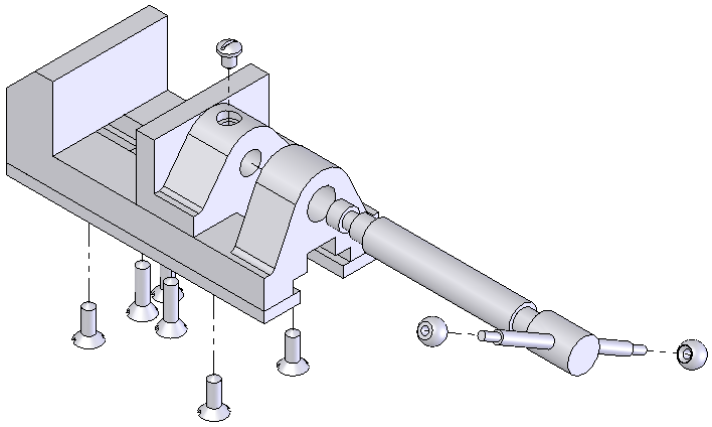


Figure 11-36 *Explode line sketch created on an exploded assembly*

toolbar or choose **Insert > Explode Line Sketch** from the menu bar. The **Route Line PropertyManager** is displayed as shown in Figure 11-37. You are prompted to select cylindrical face, planar face, vertex, point, arc, or line entities. One by one select the cylindrical faces of the two components to create an explode line between them. For example, to create an explode line between the oval fillister and the vice jaw, select the cylindrical face of the oval fillister that goes inside the vice jaw. Now, select the cylindrical hole of the vice jaw. The preview of the explode line appears. Choose **OK** to create the explode line. The names of the selected faces are displayed in the **Items To Connect** display area. As you select an entity to create an

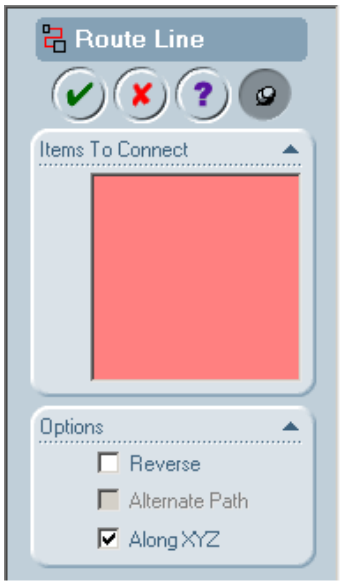


Figure 11-37 *The* ***RouteLine PropertyManager***

explode line, an arrow is also displayed with the line. You can use that arrow to reverse the direction of explode line creation. You can also choose the **Reverse** check box from the **Options** rollout of the **Route Line PropertyManager** to reverse the direction of the explode line creation. After creating the explode line in all the components, choose the **OK** button from the **Explode Line Sketch PropertyManager**.

Tip. *After creating all the explode lines, you need to exit the sketching environment using the* ***Confirmation Corner****.*

You can add jog lines using the ***Jog Line*** *tool.*

TUTORIALS

Tutorial 1

In this tutorial you will create the assembly shown in Figure 11-38. This assembly will be created in two parts, one will be the main assembly and the second will be the subassembly. You will also create the exploded state of the assembly and then create the explode line sketch. The explode state of the assembly is displayed in Figure 11-39. The views and dimensions of all the components of this assembly are displayed in Figures 11-40 through 11-43. **(Expected time: 3 hrs)**

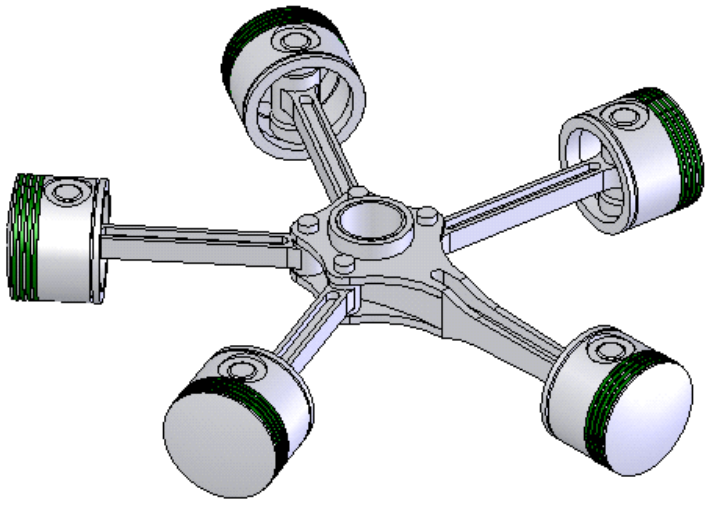


Figure 11-38 *The radial engine subassembly*

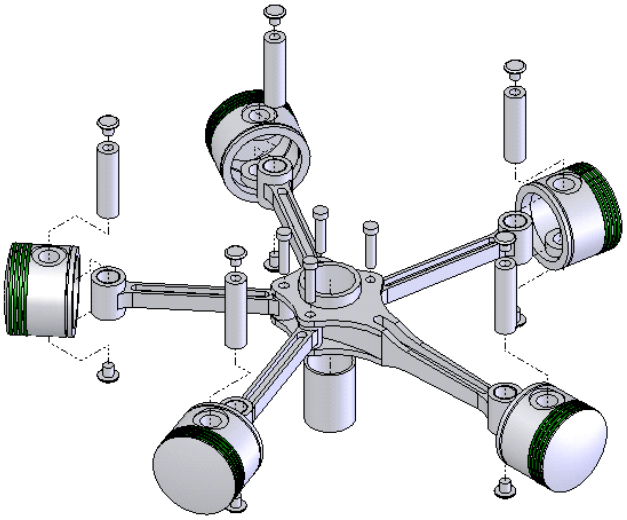


Figure 11-39 *Exploded view of the assembly*

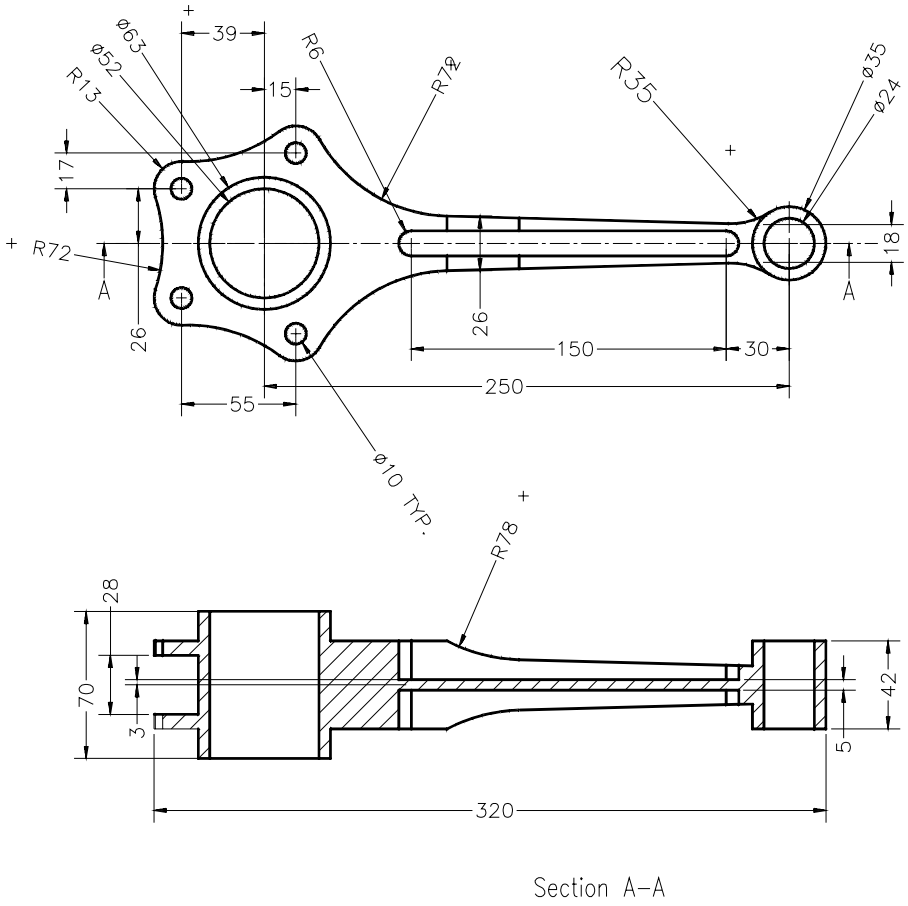


Figure 11-40 *Views and dimensions of Master rod*

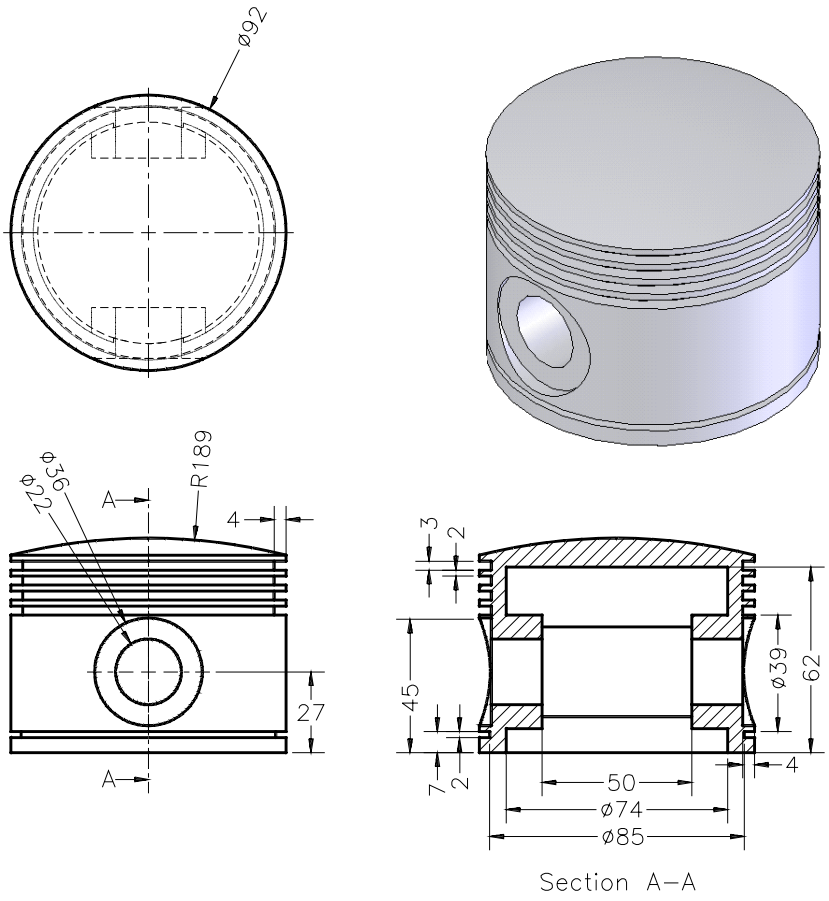


Figure 11-42 *Views and dimensions of the Piston*

The steps to be followed to complete this tutorial are discussed next:

Since this assembly is a large assembly, therefore, you need to divide this assembly in two parts. One will be the master assembly and the second will be the subassembly. First, you need to create the subassembly, which consists of Articulation Rod, Piston, Piston Rings, Piston Pin, Rod Bush Upper, Rod Bush Lower, and Piston Pin Plug. After creating this subassembly you will create the master assembly in which you will assemble the Master Rod with the Piston, Piston Rings, Piston Pin, Rod Bush Upper, and Piston Pin Plug. After assembling the components in the master assembly you will insert the subassembly in the master assembly and assemble it.

a. Create all the components of the assembly in the **Part** mode and save the components in a directory named **Radial Engine Subassembly**.

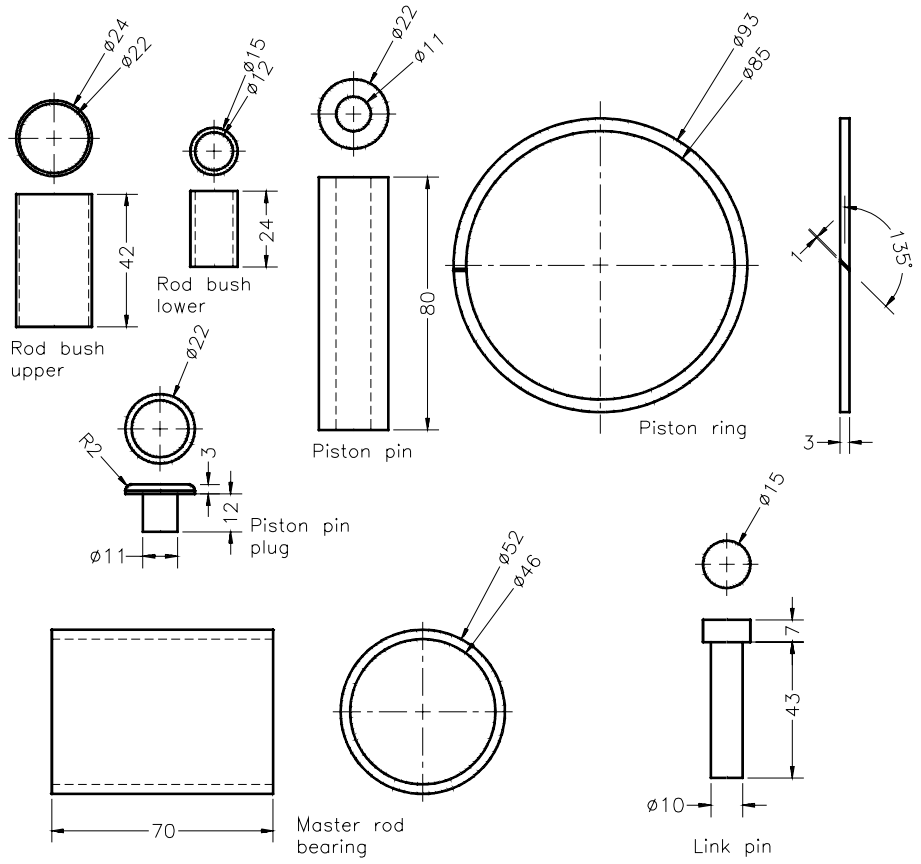


Figure 11-43 *Views and dimensions of other components*

b. Open a new assembly file and assemble the components to create a subassembly.
c. Open a new assembly file and assemble the components of the master assembly.
d. Insert the subassembly in the master assembly and assemble it with the master assembly.
e. Create the exploded view of the assembly and then create the explode lines.

Creating the Components

1. Create a directory with the name **Radial Engine Subassembly** in the */My Documents/SolidWorks/C11* directory. Create all the components in the individual part files and save them in this directory.

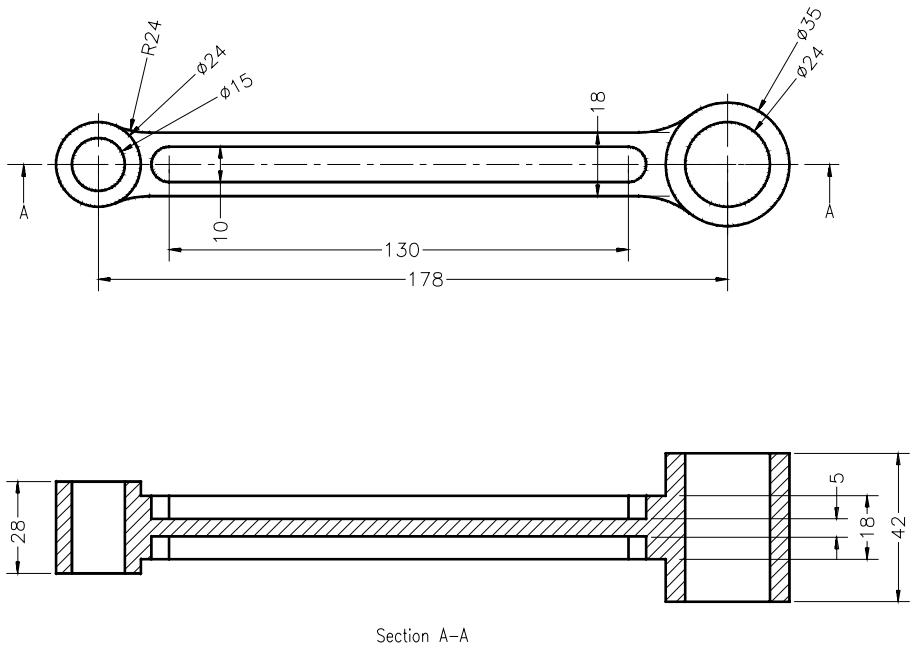


Figure 11-41 *Views and dimensions of the Articulated rod*

Note

When you create the Master Rod, remember that the holes on the left of the Master Rod should be created using the sketch-driven pattern. This is done because while assembling the Link Pin you will create the derived pattern of the Link Pin using the sketch driven pattern feature.

Creating the Subassembly

As discussed earlier, you will first create the subassembly. After that, you will assemble the subassembly with the master assembly.

1. Open a new assembly document and save it with the name **Piston Articulation Rod Subassembly** in the same directory in which the parts are created.

2. First place the Articulated Rod at the origin of the assembly and then place the other components such as Piston, Piston Pin, Piston Pin Plug, Rod Bush Upper, and Rod Bush Lower in the assembly document.

3. Apply the required mates to assemble these components. The assembly after assembling the Articulated Rod, Piston, Piston Pin, Piston Pin Plug, Rod Bush Upper, and Rod Bush Lower is shown in Figure 11-44.

   It is clear from the assembly that you need to assemble four instances of Piston Ring. You

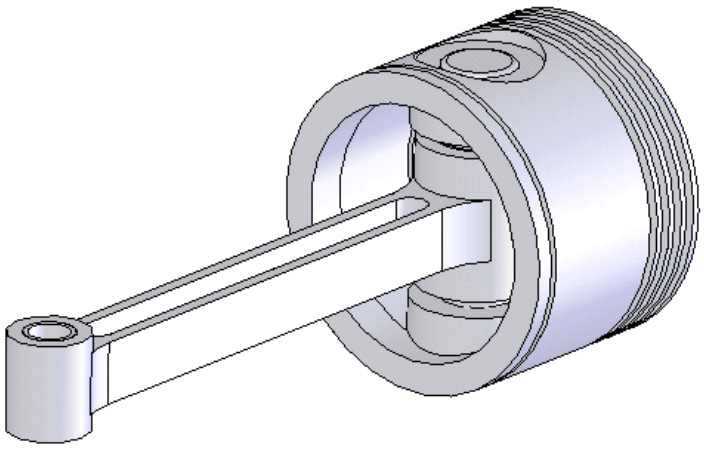


Figure 11-44 *Assembly of Articulated Rod, Piston, Piston Pin, Piston Pin Plug, Rod Bush Upper, and Rod Bush Lower*

will only assemble one instance of the Piston Ring at the uppermost groove of the ring. After assembling this instance, you will create the local pattern of the Piston Ring.

4. Insert the Piston Ring in the assembly document and assemble the Piston Ring at the uppermost groove of the Piston using the assembly mates, see Figure 11-45. In this figure, the color of Piston Ring is changed by selecting it and choosing the **Component Properties** option from the shortcut menu. From the **Component Properties** dialog box, choose the **Color** button to change the color.

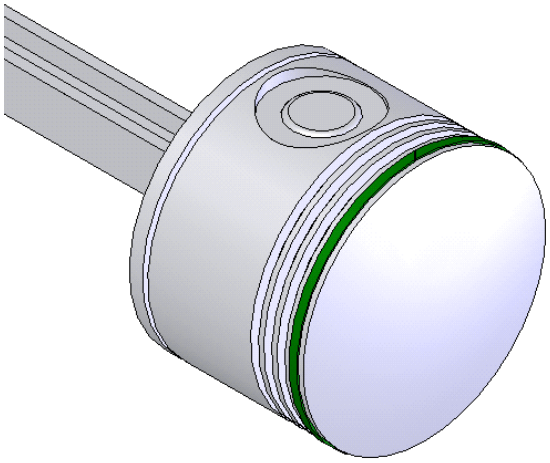


Figure 11-45 *First instance of the Piston Ring assembled with the Piston*

Next, you need to create the local pattern of the Piston Ring.

5. Choose **Insert > Component Pattern** from the menu bar. The **Pattern Type** dialog box is displayed.

6. Select the **Define your own pattern (Local)** radio button from this dialog box.

7. The **Arrange in straight lines (Linear)** radio button is selected by default. Choose the **Next** button from the **Pattern Type** dialog box. The **Local Component Pattern** dialog box will be displayed.

8. Select the Piston Ring from the assembly document; the name of the Piston Ring is displayed in the **Seed Component(s)** display area.

9. Now, click once in the **Along Edge/Dim** display area to activate the selection mode in this area.

10. Select any one of the horizontal edges of the Articulation Rod to define the direction of pattern creation.

11. Select the **Reverse Direction** check box to reverse the direction of pattern creation.

12. Set the value of the **Spacing** spinner to **5** and set the value of the **Instances** spinner to **4**.

13. Choose the **Finish** button from the **Local Component Pattern** dialog box.

The subassembly after patterning the Piston Ring is shown in Figure 11-46

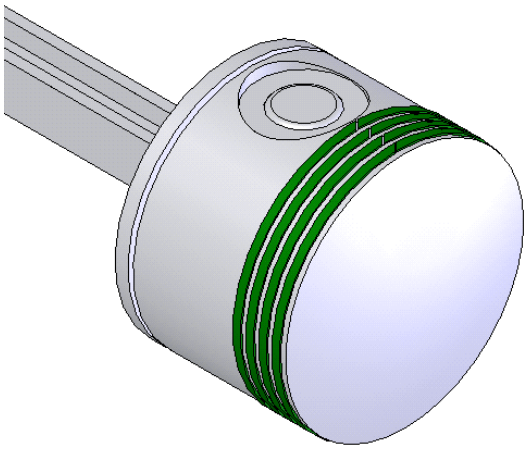


Figure 11-46 *Subassembly after patterning the Piston ring*

The subassembly is completed. Save the assembly document.

Creating the Master Assembly

Next, you will create the master assembly. After creating the master assembly you will place and assemble the subassembly with the master assembly.

1. Open a new assembly document and save it with the name **Radial Engine Subassembly** in the same directory in which the parts are created.

2. First, place the Master rod at the origin of the assembly and then place the Piston, Piston Pin, Piston Pin Plug, Piston Ring, Rod Bush Upper, and Master Rod Bearing in the assembly document.

3. Using the assembly mates and local pattern option assemble all the components of the master assembly.

The components after assembling in the master assembly are displayed in Figure 11-47.

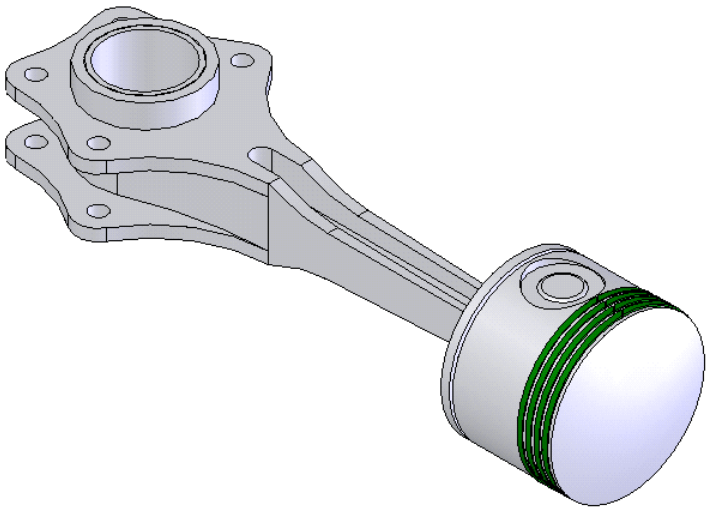


Figure 11-47 *Components assembled in the Master assembly.*

Assembling the Subassembly With the Master Assembly

Now, you will place the subassembly in the mater assembly and then assemble it with the subassembly.

1. Choose **Insert** > **Component** > **From File** from the menu bar. The **Open** dialog box is displayed.

2. Choose **Assembly** (**.asm, *.sldasm*) from the **Files of type** drop-down list.

3. Double-click on *Piston Articulated Rod Subassembly.SLDASM* and place the subassembly in the master assembly.

Figure 11-48 shows the subassembly placed in the master assembly.

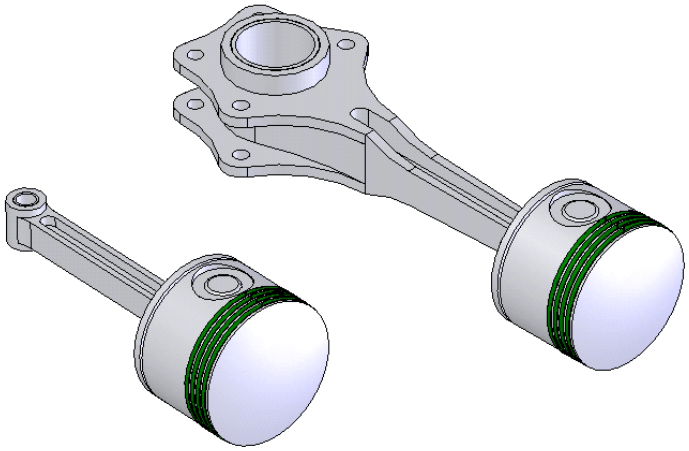


Figure 11-48 *Subassembly placed in the master assembly*

Tip. *You can create more than one instance of the subassembly by holding down the CTRL key; select and drag the subassembly from the* ***FeatureManager Design Tree****. Release the left mouse button to place the assembly in the assembly document.*

4. Using the assembly mates assemble the subassembly with the master assembly. Refer to Figure 11-49, which shows you the assembly structure that will help you in assembling the instances of the subassembly.

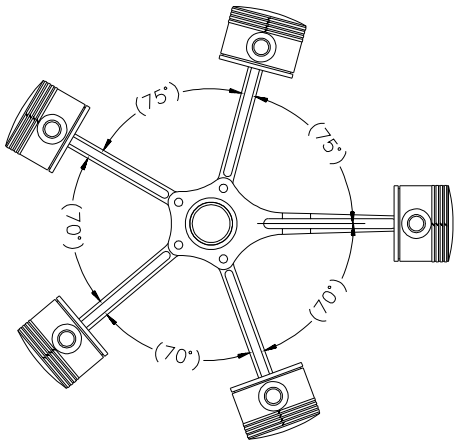


Figure 11-49 *Assembly structure*

Figure 11-50 shows all the instances of the subassembly assembled with the master assembly.

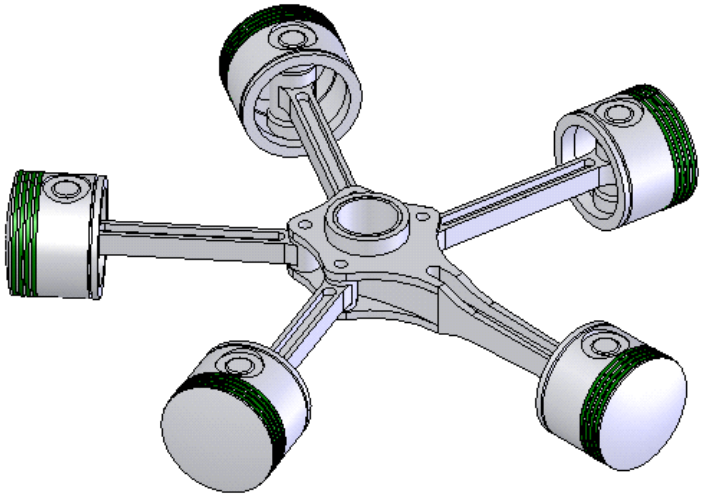


Figure 11-50 *Subassembly assembled to the Master assembly*

Assembling the Link Pin

After assembling the other components you need to assemble the Link Pin with the master assembly.

1. Place the Link Pin in the assembly document and using the assembly mates, assemble the Llink Pin with the master assembly. Figure 11-51 shows the first instance of Link Pin assembled with the master assembly.

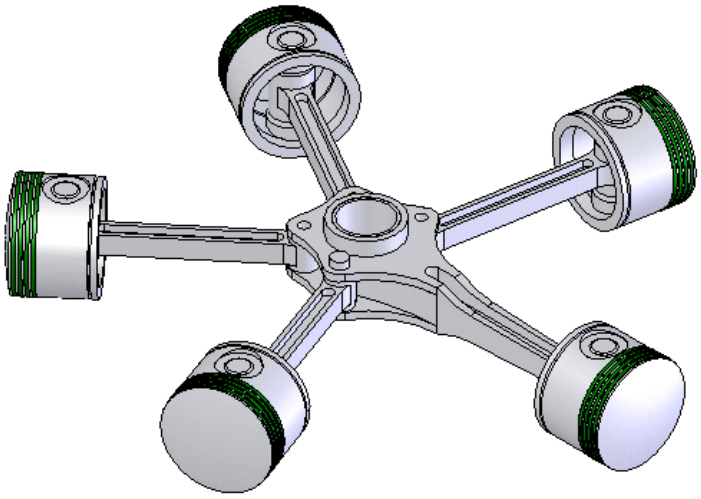


Figure 11-51 *First instance of the Link pin assembled with the master assembly*

As discussed earlier, the other instances of the Link Pin will be assembled as derived pattern using the sketch-driven pattern feature of the holes created on the left of the master rod.

2. Choose **Insert > Component Pattern** from the menu bar. The **Pattern Type** dialog box will be displayed.

   The **Use an existing feature pattern (Derived)** radio button is selected by default in the **Pattern Type** dialog box.

3. Choose the **Next** button from the **Pattern Type** dialog box. The **Derived Component Pattern** dialog box is displayed.

   The selection mode is active in the **Seed Component(s)** display area.

4. Select the Link Pin from the master assembly. The name of the Link Pin is displayed in the **Seed Component(s)** display area.

5. Click once in the **Pattern Feature** display area to activate the selection mode in this area.

6. Select any one of the hole instance created using the sketch-driven pattern. The name of the sketch pattern feature will be displayed in the **Pattern Feature** display area.

7. Choose the **Finish** button from the **Derived Component Pattern** dialog box.

   Figure 11-52 displays the final assembly.

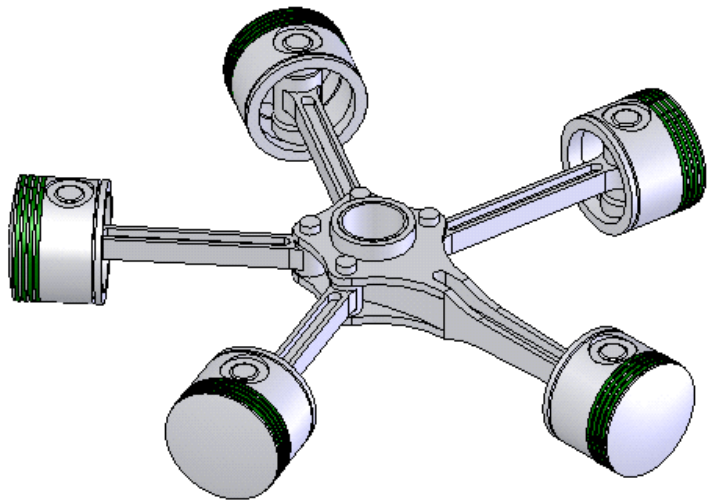


Figure 11-52 *The final assembly*

Exploding the Assembly

After creating the assembly you need to explode the assembly using the **Assembly Exploder** tool.

1. Invoke the **ConfigurationManager**. Select the **Default** option from the **ConfigurationManager** and invoke the shortcut menu. Choose the **New Exploded View** option from the shortcut menu.

   The **Assembly Exploder** dialog box is displayed.

2. Choose the **New** button from the **Assembly Exploder** dialog box.

3. Select any vertical edge from the assembly to define the direction in which the components will be exploded.

4. Select the **Component part only** radio button on the right of the **Components to explode** area.

5. Select all the Piston Pin Plugs from the assembly.

6. Set the value of the **Distance** spinner to **170** and choose the **Apply** button from the **Assembly Exploder** dialog box.

   All the instances of the Piston Pin Plug are exploded.

7. Choose the **New** button from the **Assembly Exploder** dialog box to create a new assembly exploder step.

8. Select any vertical edge from the assembly to define the direction of explode.

9. Select all the instances of the Piston Pin from the assembly.

10. Set the value of the **Distance** spinner to **140** and choose the **Apply** button to end the explode step creation.

    Similarly, explode the other components of the assembly. The assembly after exploding the components is shown in Figure 11-53.

Creating the Explode Line Sketch

After exploding the assembly, you need to create the explode line sketch of the exploded state of the assembly.

1. Choose the **Explode Line Sketch** button from the **Assembly** toolbar. The **Route Line PropertyManager** is displayed and you are prompted to select cylindrical face, planar face, vertex, point, arc, or line entities.

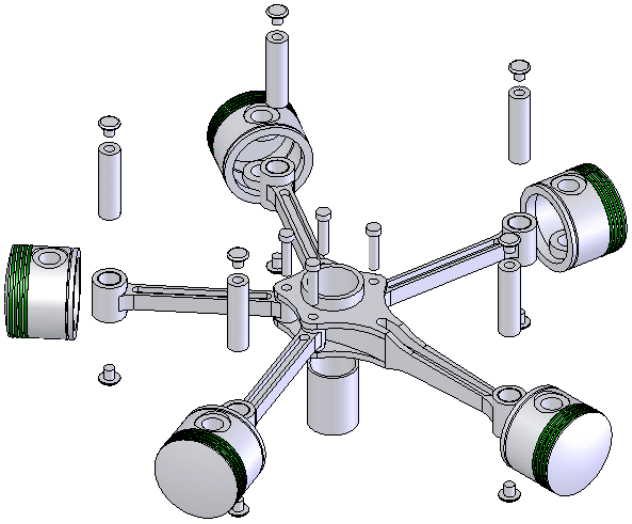


Figure 11-53 *Final assembly*

2. Choose the **Keep Visible** button from the **Route Line PropertyManager** if it is not chosen automatically to keep it visible on the screen.

3. Select the cylindrical face as shown in Figure 11-54 as the first selection. The name of the selected face will be displayed in the **Items To Connect** display area. The preview of the explode line sketch is displayed at the center of the selected face.

4. Refer to Figure 11-54 and select the cylinderical faces to create the explode line sketch.

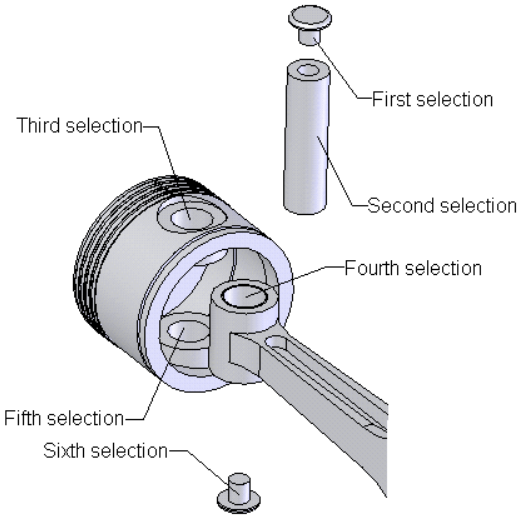


Figure 11-54 *Faces to be selected to create explode line sketch*

5. After selecting all the faces, choose the **OK** button. An exploded line will be created.

   Similarly, create the explode lines to connect the other parts of the assembly. Figure 11-55 shows the assembly after adding the explode line sketch.

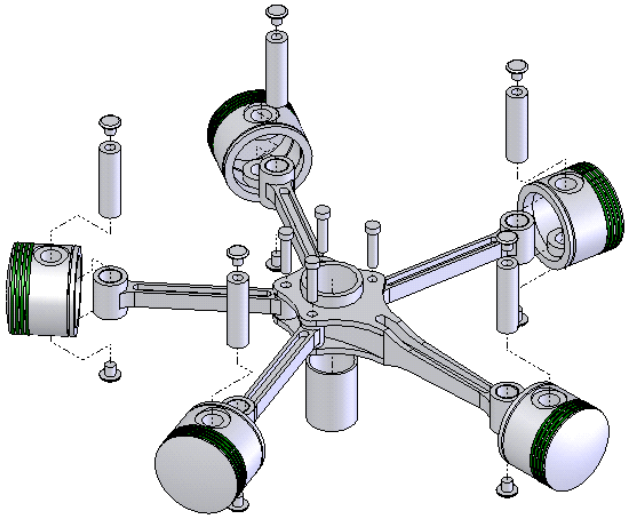


Figure 11-55 *Explode line sketch added to the exploded state of the assembly*

6. Invoke the **ConfigurationManager** and expand the **Default** option. Select **ExplView1** and invoke the shortcut menu. Choose the **Collapse** option from the shortcut menu to switch back to the collapse state of the assembly.

7. Save the assembly.

Tutorial 2

In this tutorial you will modify the assembly created in Tutorial 1 (Bench vice) of Chapter 10, *Assembly Modeling-I*. You will modify the design of the components of the assembly and then suppress some mates that enable the assembly to move along a particular degree of freedom. After that you will check the assembly for collision detection when the assembly is in motion and then you will modify the assembly and check the interference. **(Expected time: 1 hr)**

The steps that will be followed to complete this tutorial are discussed next:

a. Save the Bench Vice assembly directory in *c11* directory and then open the Bench Vice assembly.
b. Modify the design of the components within the context of assembly.
c. Suppress the mate to enable the vice jaw to move along the slide ways of the vice jaw.
d. Check the new assembly design for collision detection when the assembly is in motion. Modify the design if there is any collision between the components.
e. Check the interference in the modified assembly.

Opening the Bench Vice Assembly

The assembly created in the Tutorial 1 of Chapter 10 is the Bench vice assembly; You need to copy and save that assembly in the current directory of Chapter 11.

1. Copy the directory in which the Bench vice assembly is saved and then paste it in the *c11* directory.

2. Invoke the **Open** dialog box and browse the Bench Vice assembly document. Double-click on it to open the assembly document.

Modifying the Design of the Components of Bench Vice Assembly

Because of some alteration in the design of some of the components of the assembly, you need to modify the components of the assembly. You will modify and edit the components in the context of assembly.

Before you start modifying the components, it is recommended that you hide some of the components to simplify the assembly. Hiding the components will simplify the assembly and facilitate in selection while editing and modifying the components.

1. Hold down the CTRL key from the keyboard and select the Clamping Plate, Base Plate, Set Screw 1, Set Screw 2, Oval Fillister, Screw Bar, and Jaw Screw from the **FeatureManager Design Tree**.

2. Invoke the shortcut menu and choose the **Hide components** option from the shortcut menu.

   The selected components are hidden and are not displayed in the assembly document.

   With the design alteration, you need to create a through slot on the right face of the Vice Jaw. For modifying the design of the Vice Jaw, first you need to enable the part editing environment.

3. Select the Vice Jaw from the assembly and then choose the **Edit Part** button from the **Assembly** toolbar. The part modeling environment is invoked in the assembly document and the Vice Body is displayed in transparent as shown in Figure 11-56.

4. Select the right face of the Vice jaw and invoke the sketching environment.

5. Create the sketch of the slot as shown in Figure 11-57.

6. Choose **Insert > Cut > Extrude** from the menu bar. The **Cut-Extrude PropertyManager** is displayed.

7. Create the cut feature using the **Through All** option.

8. Choose the **Edit Part** button to exit the edit part environment.

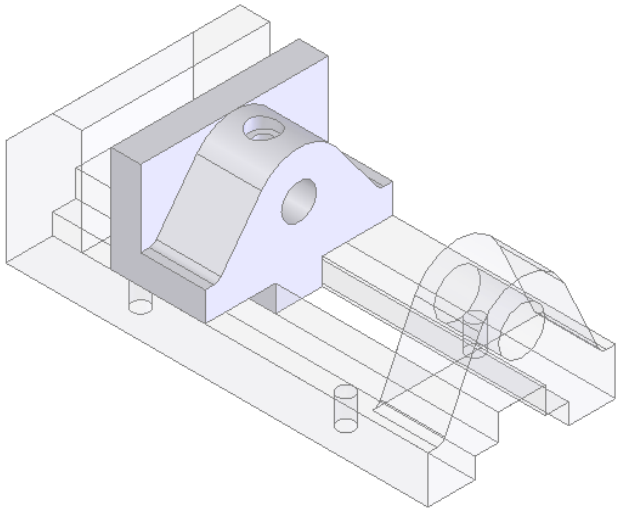


Figure 11-56 *Vice Jaw in the part edit mode in the assembly document*

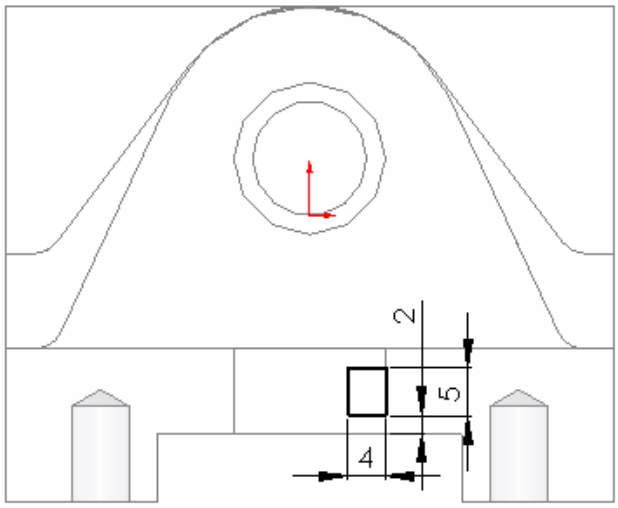


Figure 11-57 *Sketch of the slot*

Figure 11-58 shows the assembly after modifying the design of the Vice jaw.

9. Similarly modify the design of the Vice body. You need to create an extruded boss feature on the right face of the component.

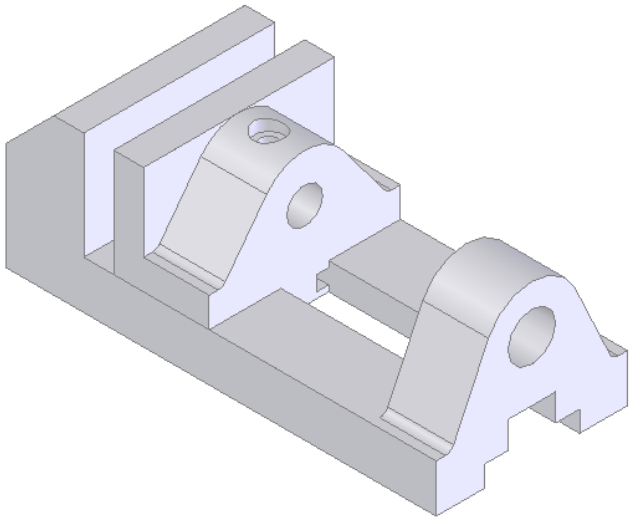


Figure 11-58 *Modified Vice Jaw*

The sketch of the feature is shown in Figure 11-59. You need to extrude the sketch to a distance of 60. Figure 11-60 shows the assembly after exiting the edit part environment.

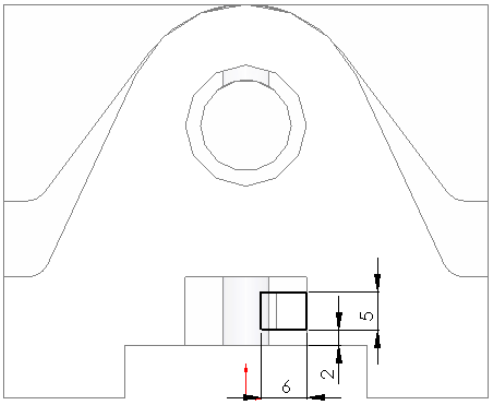


Figure 11-59 *Sketch of the extruded boss feature*

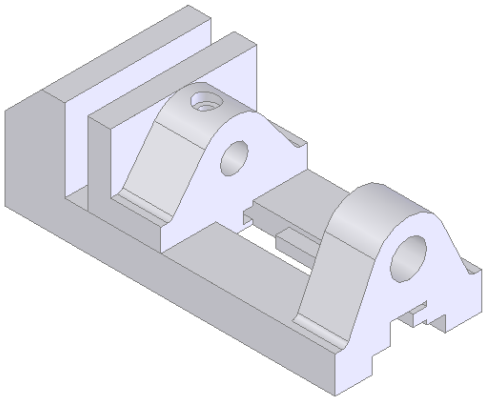


Figure 11-60 *Modified assembly*

10. Choose the **Save** button from the **Standard** toolbar. The **SolidWorks** information box is displayed and you are informed that some models referenced in the document are modified and they must be saved. Choose **Yes** to save the referenced models also.

Suppressing the Mate to Make the Movement of Vice jaw Free in a Specified Direction

To analyze the movement of the Bench vice assembly, you need to make the movement of the Vice jaw free in the X direction. By doing so the Vice Jaw will slide on the sideways of the Vice body.

1. Expand the **Mates** mategroup from the **FeatureManager Design Tree** and select **Distance1** mate. The planar faces of the Vice jaw and the Vice body are highlighted in green because this mate is applied to these faces.

2. Select this mate and invoke the shortcut menu. Choose the **Suppress** option from the shortcut menu.

   Now, the degree of freedom in the X direction is free.

3. Choose the **Move** button from the **Assembly** toolbar and select a horizontal edge of the Vice jaw. When you drag the cursor, you will observe that you can move the Vice jaw in the X direction.

4. Drag the Vice jaw back to its original position.

Analyzing the Collision Between the Components When the Assembly is in Motion

Next, you will analyze the collision between the components of the assembly when the assembly is in motion.

1. Choose the **Move Component** button from the **Assembly** toolbar. The **Move Component PropertyManager** is displayed. Select the **Collision Detection** radio button from the **Options** rollout.

2. Select Vice jaw and drag the cursor to move it in the direction as shown in Figure 11-61.

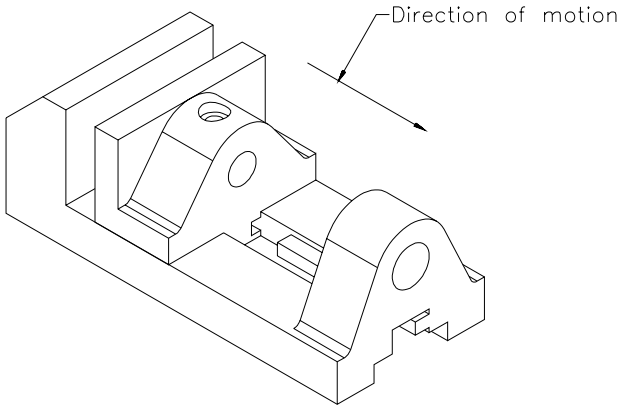


Figure 11-61 *Direction in which the Vice jaw will be moved*

3. When you move the Vice jaw in the specified direction, you will observe that the right face of the Vice jaw and the extrusion feature of the Vice body are highlighted in green as shown in Figure 11-62.

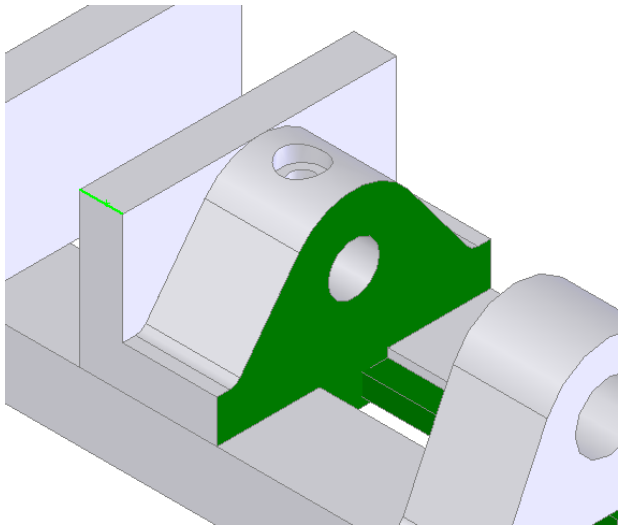


Figure 11-62 *Faces of the Vice jaw and the Vice body highlighted in green*

This indicates that the Vice jaw collides with the Vice body. Leave the assembly at this location.

4. Choose the **OK** button from the **Move Component PropertyManager**.

   Since the collision is detected in the assembly, you need to modify the design of one of the components. In this case, you will modify the dimensions of the extruded boss feature.

5. Double-click the newly created extruded boss feature of the Vice body. The dimensions of the newly created feature will be displayed.

6. Double-click the dimension that has a of value **6**; the **Modify** dialog box is displayed; set the value of the dimension to **4** and choose the ENTER key.

7. Press CTRL+B to rebuild the assembly.

8. Choose **Tools > Interference Detection** from the menu bar. The **Interference Volumes** dialog box is displayed. Choose the **Check** button from the **Interference Volumes** dialog box.

9. You will observe that **0 Interference** is displayed in the **Interference results** display area.

10. Choose the **Close** button from the **Interference Volumes** dialog box.

    Next, you need to need to unhide all the components.

11. Hold the CTRL key and select the hidden components from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Show components** option from the shortcut menu.

12. Save the assembly document and all the referenced part documents.

SELF-EVALUATION TEST

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

1. You can create subassemblies in the assembly environment of SolidWorks. (T/F)

2. You cannot create a subassembly of the components that are already placed in an assembly file. (T/F)

3. When you move the cursor on a mate in the **FeatureManager Design Tree**, the entities used in the mate are highlighted in red color in the drawing area. (T/F)

4. You cannot edit the assembly mates. (T/F)

5. While in the part editing mode in the assembly document, you can use the **Move/Size Features** tool to edit the features dynamically using editing handles. (T/F)

6. The component patterns created individually without the use of any existing pattern feature are known as _____ pattern.

7. The component patterns created using an existing pattern feature are known as _____ pattern.

8. In _____ component, the feature information is available in the part document and only the graphical representation of the component is displayed in the assembly document.

9. After selecting the component, choose the _____ option from the shortcut menu to change the transparency condition of the selected component.

10. To create the explode line sketch, choose the _____ button from the **Drawing** toolbar.

REVIEW QUESTIONS

Answer the following questions:

1. When you select the component and invoke the shortcut menu and choose the **Open** "part name" from the shortcut menu, which option is used to edit a component separately in its part document?

(a) **Modify** "part name" (b) **Edit** "part name"
(c) **Open** "part name" (d) None of these

2. Which option is used to define whether a component collides with another component of the assembly or not?

   (a) **Collision Detection** (b) **Interference Detection**
   (c) **Mass Properties** (d) None of these

3. Which check box is selected in the **Open** dialog box to open an assembly with light weight parts?

   (a) **Lightweight** (b) **Open Lightweight**
   (c) **Lightweight parts** (d) **Lightweight assembly**

4. Which button available in the **Assembly** toolbar is used to suppress a component?

   (a) **Change Suppression State** (b) **Suppress**
   (c) **Hide/Show Component** (d) **Move Component**

5. Which option is used to select the component placed partially inside and partially outside the assembly envelope?

   (a) **Inside Envelope** (b) **Outside Envelope**
   (c) **Crossing Envelope** (d) None of these

6. The _____ radio button is used to create a local linear pattern.

7. To unhide the hidden component, select the icon of the component from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the _____ option from the shortcut menu.

8. The _____ option is used to pattern the instances of the components using an existing pattern feature.

9. The explode state of the assembly is created using the _____ dialog box.

10. The _____ check box is selected to stop the motion of the assembly when one of the component collides with another component when the assembly is in motion.

EXERCISE

Exercise 1

Create the assembly as shown in Figure 11-63. Make sure that when you create this assembly, the back plate should be fixed and the entire assembly can move in the Y direction with respect to the back plate. Keep the rotational degree of freedom of the screw rod free so that it can also rotate on its axis. After creating the assembly you need to explode the assembly and create the explode line sketch. The exploded view of the assembly with explode line

sketch is shown in Figure 11-64. The dimensions of the model are given in Figures 11-65 thought 11-69. **(Expected time: 4 hrs)**

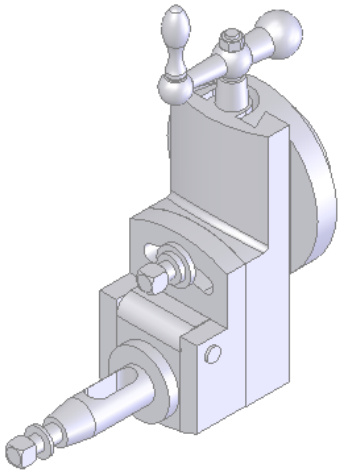


Figure 11-63 *Shaper tool holder assembly*

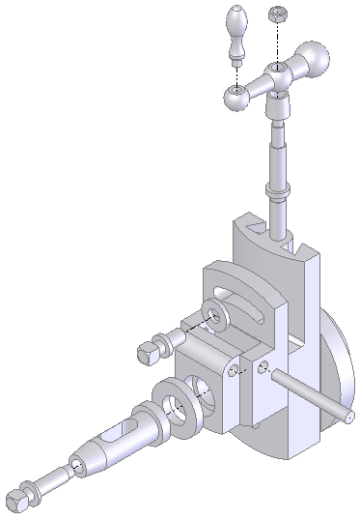


Figure 11-64 *Exploded view of the Shaper tool holder assembly with explode lines*

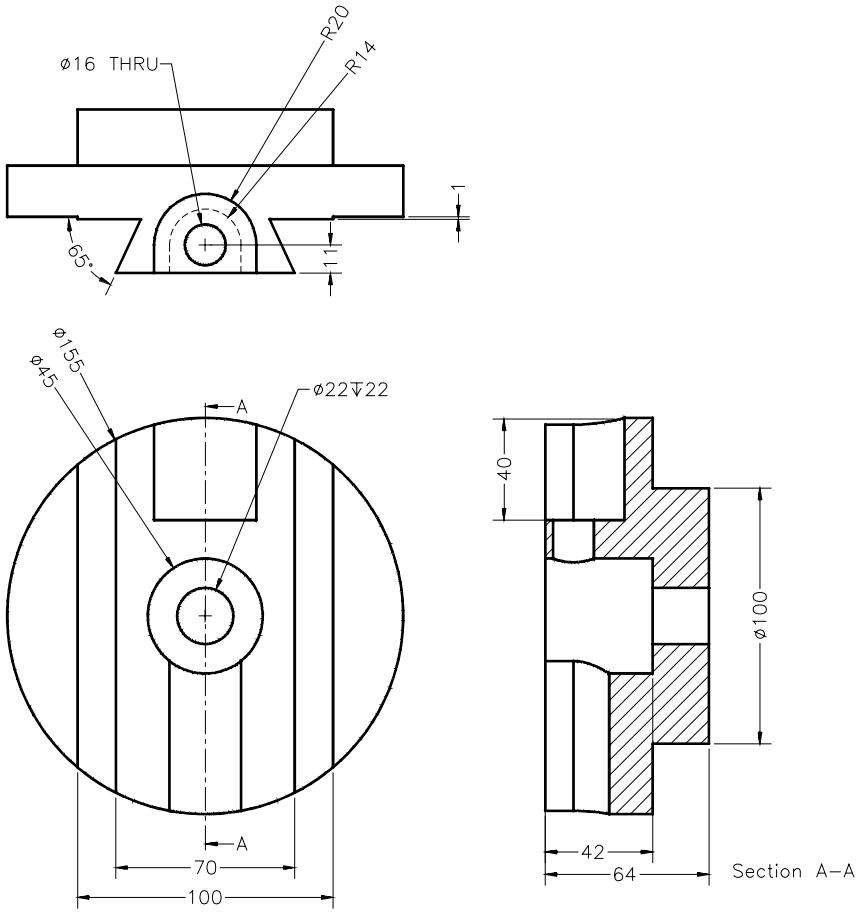


Figure 11-65 *Views and dimensions of Back Plate*

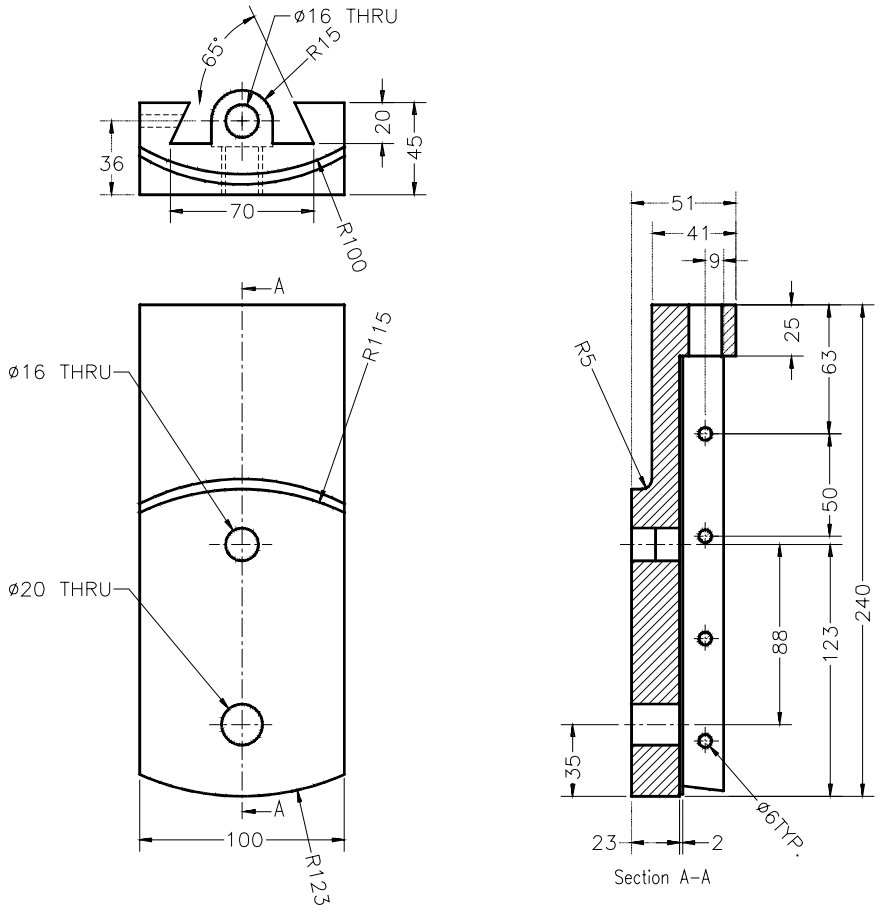


Figure 11-66 *Views and dimensions of Vertical Slide*

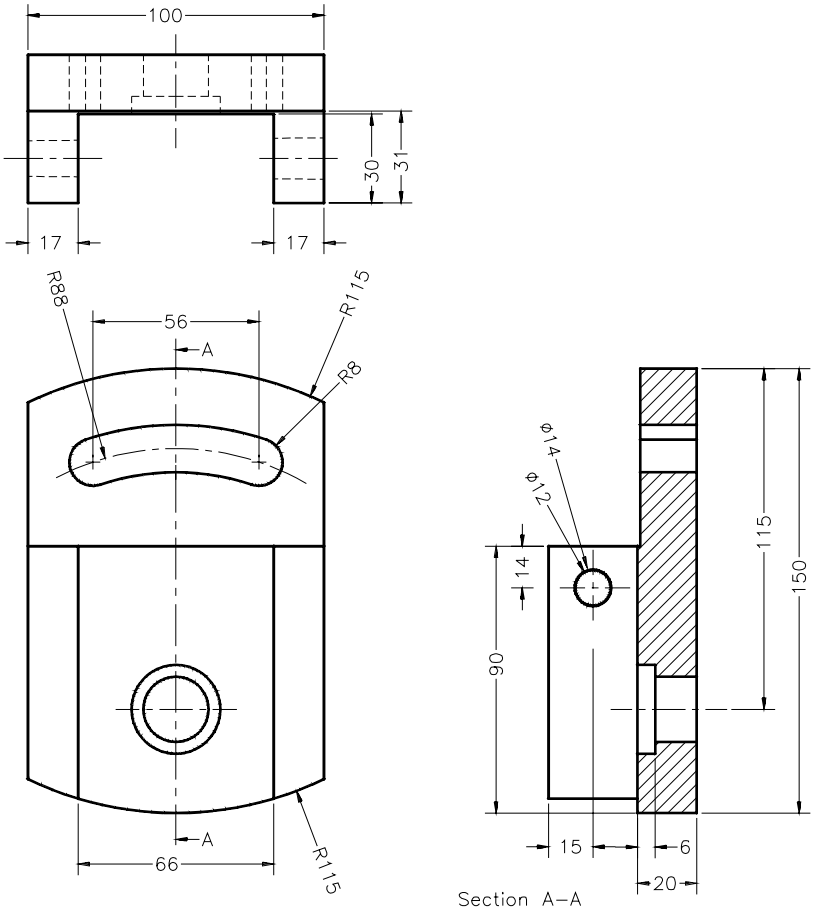


Figure 11-67 *Views and dimensions of Swivel Plate*

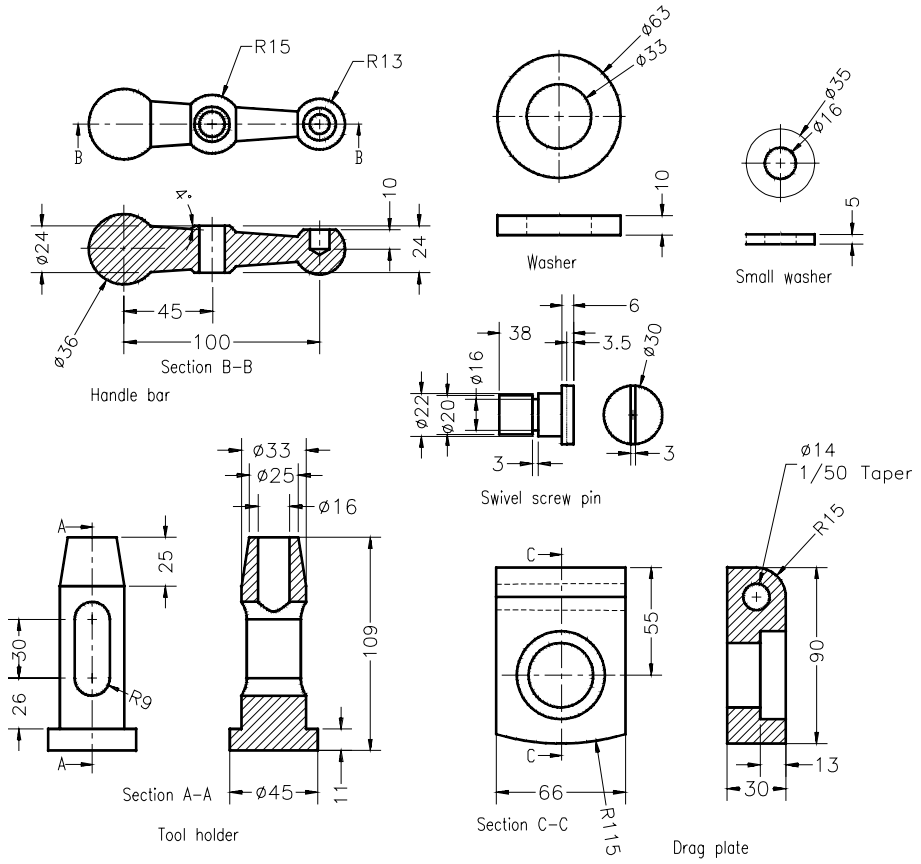


Figure 11-68 *Views and dimensions of the components*

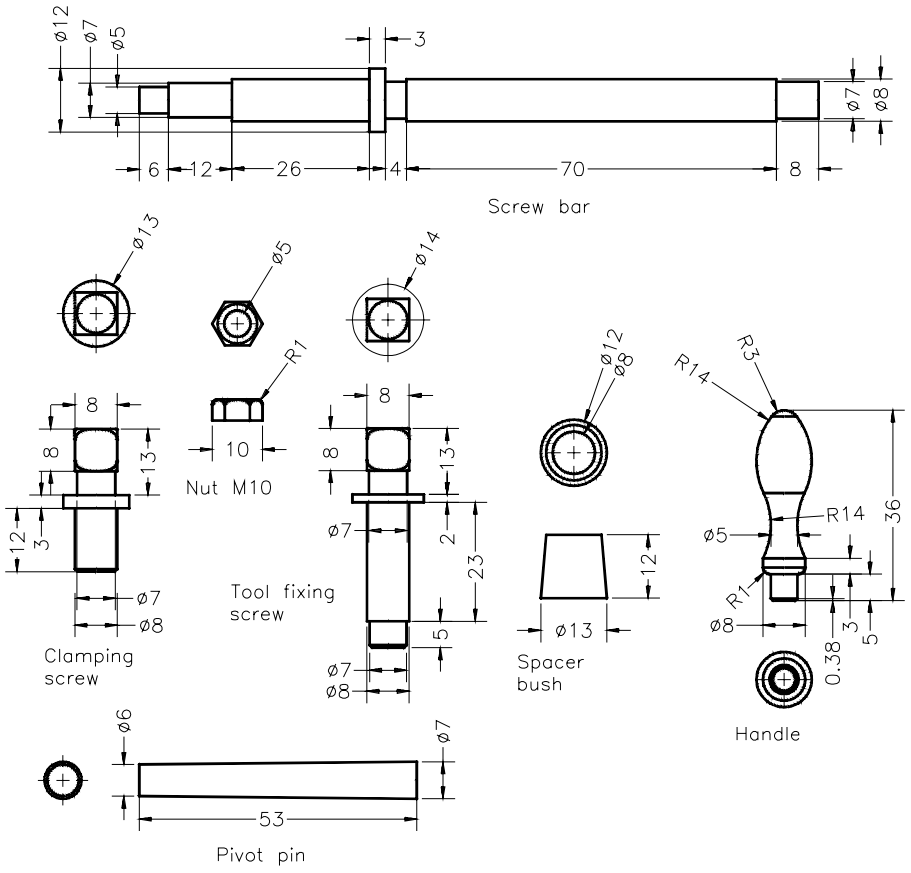


Figure 11-69 *Views and dimensions of the components*

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** T, **4.** F, **5.** F, **6. Local**, **7. Derived**, **8. Lightweight**, **9. Component Properties**, **10. Explode Line Sketch**