



Chapter 12

Assembly Modeling-II (Traditional & Synchronous)

Learning Objectives

After completing this chapter, you will be able to:

- *Create subassemblies.*
- *Edit assembly relationships.*
- *Edit assembly components.*
- *Disperse an assembly.*
- *Replace components in an assembly.*
- *Set the visibility options of an assembly.*
- *Check the interference in an assembly.*
- *Create the exploded state of an assembly.*

CREATING SUBASSEMBLIES

In the previous chapter, you learned to place components in the assembly file and apply the assembly relationships to the components. In this chapter, you will learn to create subassemblies and place them in the main assembly.

Sometimes when an assembly has many parts, it becomes easy to design it by segregating it into subassemblies. To create a subassembly, you need to start a new assembly file, assemble components in it, and save it with a name. This subassembly will then be inserted into the new assembly file and assembled with the other parts. To create subassemblies, you need to follow the same procedure as that for creating assemblies. Figure 12-1 shows the subassembly of the articulated rod and piston and Figure 12-2 shows the subassembly of the master rod and the piston. Figure 12-3 shows the main assembly that is created using the two subassemblies.

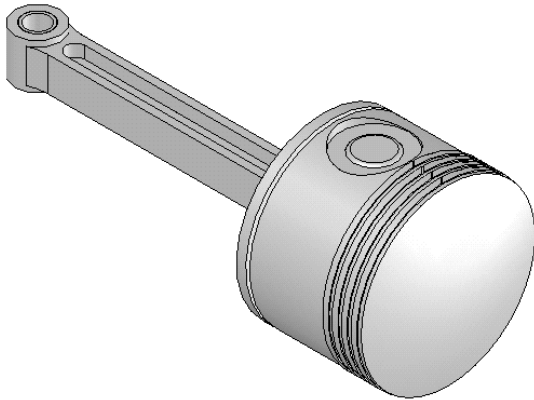


Figure 12-1 A subassembly of the articulated rod and piston

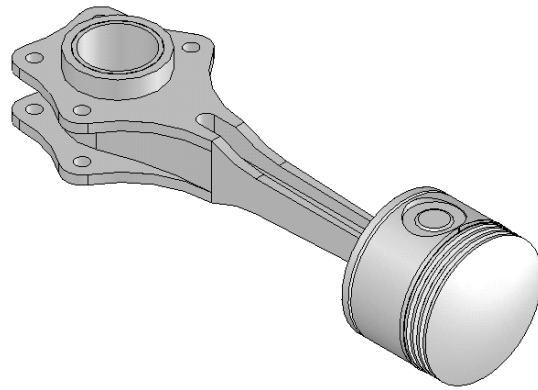


Figure 12-2 A subassembly of the master rod and piston

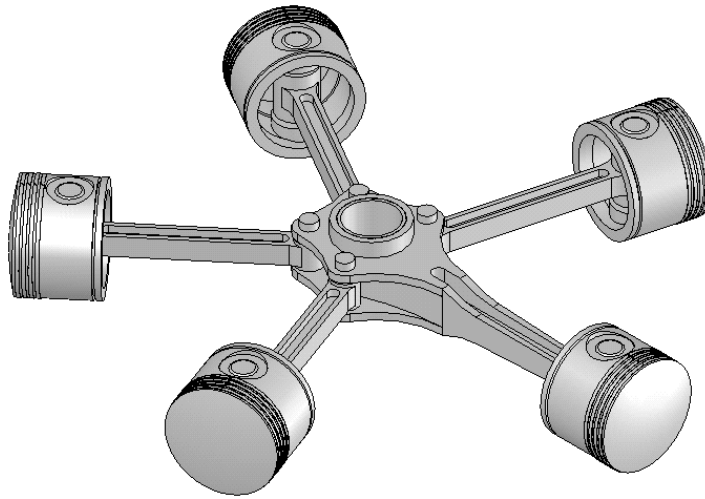


Figure 12-3 Main assembly created after assembling two subassemblies

To create a subassembly, you can follow the top-down approach or the bottom-up approach of the assembly design.



Note

Right-click in the top pane of the **Parts Library** area and choose the **Use Configurations** option from the shortcut menu. If this option is not chosen, the subassembly you place in the assembly document will not be activated.



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 docking window. If you expand the subassembly in the docking window, all parts assembled in it will be displayed.

EDITING ASSEMBLY RELATIONSHIPS

Generally, after creating the assembly or during the process of assembling the components, you need to edit the assembly relationships. The editing operations that can be performed on the assembly relationships are listed next.

- Modifying the angle and distance offset values
- Adding relationships to a partially constrained component
- Modifying the assembly relationships



Note

When you select a part from the top pane of the **PathFinder**, the relationships that are applied to the selected part and the parts before it will be displayed above the dashed line. The remaining relationships that are associated with the selected part will be displayed below the dashed line.

Modifying the Values

The following steps explain the procedure for modifying the angle or distance offset values:

1. Select a part from the docking window; the selected part will be highlighted in the assembly window and the relationships will be displayed in the bottom pane of the docking window.
2. You can recognize the relationship name by viewing its symbol. Select the mate relationship from the bottom pane; the faces to which the mate relationship was applied will get highlighted.
3. Enter a new offset value in the **Offset Value** edit box of the **Placement** rollout in the **Mate** command bar; the distance offset value of the assembly relationship will be modified and the modifications will be updated automatically in the assembly. You can also enter a negative value for the offset distance. The negative value enables you to position the part in the opposite direction.

Similarly, you can also modify the angular value of an assembly relationship.

Applying Additional Relationships

Sometimes, you may need to keep a part partially positioned with other parts in an assembly. This is because you can move the partially positioned parts along the X, Y, or Z axis. As discussed in the previous chapter, the symbol for the partially positioned part is different from that of the fully positioned part. Figure 12-4 shows a table describing the meaning of the symbols displayed in the top pane of the **Pathfinder** tab in the docking window.
















	Active part
	Inactive part
	Hidden part
	Unloaded part
	Part that is not fully positioned
	Part that has conflicting relationships
	Linked part
	Simplified part
	Displayed assembly
	Adjustable assembly
	Pattern group
	Pattern item
	Reference planes
	Reference plane
	Sketch

Figure 12-4 Symbols displayed in the docking window

The following steps explain the procedure of applying additional relationships to a part:

1. Select the partially positioned part from the docking window; the selected part will get highlighted in the assembly window and the existing relationships will be displayed in the bottom pane of the docking window.
2. If you are in the **Assembly** environment, choose the **Edit Definition** button from the **Select** command bar; the **Assemble** command bar will be displayed. But, in case, you are working in the **Synchronous Assembly** environment, then on selecting a component, its name will be displayed, which is called editing handle. If you click on the editing handle, the **Assemble** command bar will be displayed.
3. Choose the assembly relationship from the **Relationship Types** flyout to apply new relationships to the part.



Note

While editing parts using the above steps, the **Relationship List** drop-down list is displayed in the **Options** rollout of the **Assemble** command bar. In this drop-down list, you can view the existing relationships applied to the selected part.

Modifying Assembly Relationships

Sometimes, you may need to modify the existing assembly relationships of parts in an assembly. For example, you may need to modify the **Mate** relationship into the **Planar Align** relationship. To do so, select the component in the **Pathfinder** tab of the docking window. If you are working in the **Synchronous Assembly** environment, click on the editing handle. But if you are in the **Assembly** environment, you need to choose the **Edit Definition** button from the **Select** command bar. Now, select the relationship that you want to replace from the **Relationship List** drop-down list. Use the **Relationship Types** flyout to choose the new relationship. Next, you can select the elements to which the new relationship needs to be applied.

You can also delete the existing relationship by selecting it from the bottom pane of the **PathFinder**. After selecting the relationship, right-click to invoke the shortcut menu and choose the **Delete Relationship** option. You can also press the DELETE key to delete the selected relationship. After deleting the relationship, you can apply a new relationship to the part.

The bottom pane of the **Pathfinder** is used to view and modify the relationships between the selected part and the other parts in the assembly. Figure 12-5 shows the list of symbols that are displayed in the bottom pane. Their meanings are also mentioned in the same figure.

	Ground relationship
	Mate relationship
	Planar align relationship
	Axial align relationship
	Connect relationship
	Angle relationship
	Tangent relationship
	Suppressed relationship
	Failed relationship

Figure 12-5 Symbols of relationships

Reversing the Orientation of a Part

You can reverse the orientation of a part that has been positioned using the **Axial Align**, **Planar Align**, **Mate**, **Parallel**, **Tangent**, or **Cam** relationship. To flip a part, select it and then select the relationship. Right-click to invoke the shortcut menu and choose the **Flip** option.



Tip. When you modify a fixed offset type to a floating offset, you may also need to modify the other relationships to make the part fully positioned.



Note

You can also use the **Flip** button available in the **Assemble** command bar to reverse the orientation of the part. When you flip a part assembled using the **Mate** relationship, this relationship will be converted into a **Planar Align** relationship.

*The rotatory movement of the part will be locked on applying the **Axial Align** relationship. You can unlock it by editing the relationship from the bottom pane of the docking window. When you select the **Axial Align** relationship, the **Unlock Rotation** and **Lock Rotation** buttons will be displayed in the command bar. Choose the **Unlock Rotation** button.*

EDITING ASSEMBLY COMPONENTS

After inserting and positioning components in an assembly file, you may need to edit components at some later stage of your assembly design cycle. The editing of components includes editing features, editing sketch profile, and modifying reference planes or faces. The following steps explain the procedure of editing components:

1. Select the component from the top pane of the **PathFinder** tab of the docking window. Right-click on the component and choose the **Edit** option from the shortcut menu; the **Part** or **Synchronous Part** environment will be invoked in the assembly file.
2. Select the feature to be edited from the docking window. You can edit or delete an existing feature. Also, you can add a new feature to the assembly component. This type of editing is known as **Editing in the Context of Assembly**.
3. After the part is edited, choose **Close > Close and Return** from the **Ribbon** to close and return to the **Assembly** environment.



Tip: To edit a component separately from the **Assembly** environment, select the component and invoke the shortcut menu. Next, choose the **Open in Solid Edge Part** option from the shortcut menu; the component will open in the corresponding **Part** environment. Edit the component and then save the changes.

MODIFYING SYNCHRONOUS ASSEMBLY COMPONENTS

You can modify the components that are assembled in the **Synchronous Assembly** environment using the steering wheel. However, to do so, such components must be created in the **Synchronous Part** environment. To modify a component, select the faces to be modified of that component; a steering wheel will be displayed. By default, you may not be able to select the faces. To select the required faces, select the **Faces Priority** option from the flyout displayed on choosing the black arrow besides the **Select** tool in the **Ribbon**. Alternatively, move the cursor toward the required component; the **QuickPick** tool will be displayed. Next, right-click; the **QuickPick** list box with the list of nearest possible selections will be displayed. Select the face to be modified from this list box. You can also move or rotate the selected faces of Synchronous components using the steering wheel. The working of steering wheel is the same as discussed in Chapter 10.

DISPERSING SUBASSEMBLIES

Ribbon: Modify > Disperse



Dispersing the subassembly means that the components of the subassembly become the components of the next higher level assembly or subassembly. To disperse a subassembly, select it from the **Pathfinder** tab of the docking window and choose the

Disperse button from the **Modify** group of the **Ribbon**; the **Disperse Assembly** information box will be displayed, as shown in Figure 12-6. In this information box, choose **Yes** to accept the transfer of components of the subassembly to the next higher level assembly or subassembly.

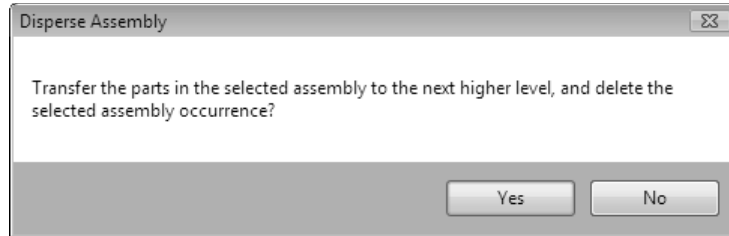


Figure 12-6 The Disperse Assembly information box

REPLACING COMPONENTS

Ribbon: Modify > Replace Part



Sometimes in the assembly design, you may need to replace a component of a assembly with some other components. To replace a component, choose the **Replace Part** button from the **Ribbon**; you will be prompted to select the component to be replaced. Select the component; the **Replacement Part** dialog box will be displayed. Note that if the component is already selected before invoking the **Replace Part** button, there will be no prompt and the **Replacement Part** dialog box will be displayed directly. Select the component to be replaced and choose the **Open** button from the **Replacement Part** dialog box; the assembly component will be replaced by a new part. While replacing the components, Solid Edge compares the geometry of two components, and if their geometry matches properly, the new component will be fully positioned with the other component in the assembly. To place the new component, Solid Edge uses the same relationships that were used to place the original component. However, if there is a change in the geometry of the new component, the assembly relationships may fail and the **Replace Part** information box will be displayed, as shown in Figure 12-7.

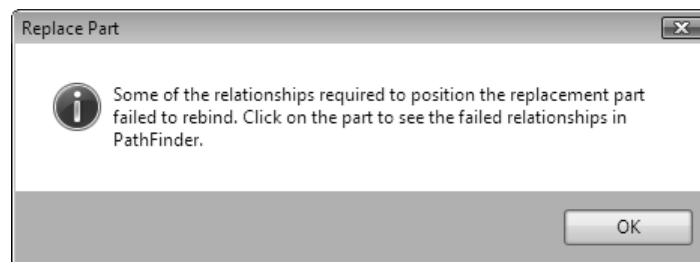


Figure 12-7 The Replace Part information box

If the relationships fail, their symbols will be displayed in red color at the bottom pane of the **Pathfinder**. You need to delete these relationships and apply new ones to fully position the replaced component.

SIMPLIFYING ASSEMBLIES USING VISIBILITY OPTIONS

When you are assembling components, whether it is a large assembly or a small one, you may need to simplify it 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. The methods of simplifying the assembly are discussed next.

Hiding and Displaying Components

To hide a component placed in an assembly, select the component from the assembly or from the **PathFinder** and clear the check box in front of the required component; the display of the component will be turned off. Also, the symbol next to the component in the **PathFinder** will change, indicating that the component is hidden. Alternatively, select the component and right-click; a shortcut menu will be displayed. Choose **Hide** from the shortcut menu to hide the component.

To show the hidden component, select the hidden component from the **PathFinder** and then select the check box in front of it; the hidden component will be displayed again in the assembly. Alternatively, select the component and right-click; a shortcut menu will be displayed. Choose **Show** from the shortcut menu to display the component.



Tip. You will notice that when you open an assembly by using the **Open** button from the **Quick Access toolbar**, the **Inactivate all** radio button is selected by default in the **Open File** dialog box. As a result, the assembly will open with all its parts inactivated. This helps in opening the assembly faster. The symbol of an inactivated component is also different from that of the activated component.

To activate a component, right-click on the inactivated component and choose the **Activate** option from the shortcut menu.

Changing Transparency Conditions

In Solid Edge, you can change the transparency of components to simplify the assembly. Select the component to change its transparency. Next, select the required color from the **Face Style** drop-down list in the **Style** group of the **View** tab in the **Ribbon**; the color of the selected component will be changed.

INTERFERENCE DETECTION IN ASSEMBLIES

After creating an assembly design, the most essential step is to check the interference between components of an assembly. If there is an interference between components, the components may not assemble properly after they are manufactured. Therefore, before sending components for manufacturing, it is essential to check the assembly for interference. To check interference, choose **Inspect > Evaluate > Check Interference** from the **Ribbon**; the **Check Interference** command bar will be displayed. The tabs in this command bar are discussed next.

Interference Options

The **Options** button is used to set the output options for checking the interference. On choosing this button, the **Interference Options** dialog box will be displayed, as shown in Figure 12-8. The options available in this dialog box are discussed next.

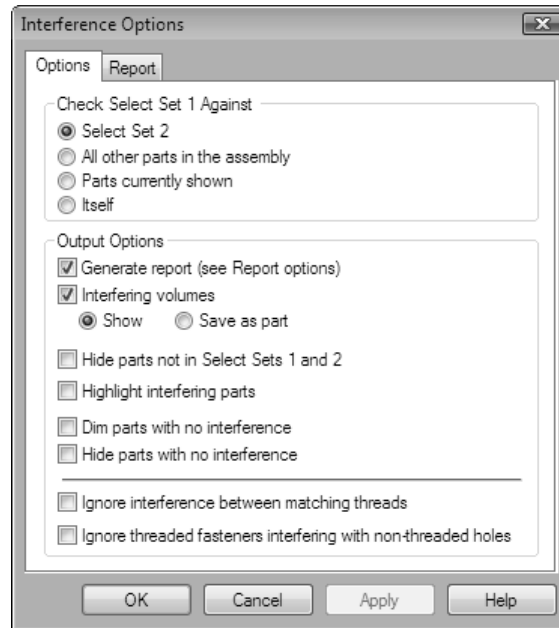


Figure 12-8 The Interference Options dialog box

Options Tab

This tab is chosen by default. The options available in this tab are discussed next.

Check Select Set 1 Against Area

The options in this area enable you to determine the method of selecting components for checking the interference. These options are discussed next.

Select Set 2. This option is selected by default. When this option is selected, you need to select the second set of components individually from the **Pathfinder** or from the assembly window after selecting the first set of components.

All other parts in the assembly. This option considers the remaining parts of the assembly as the second selection set.

Parts currently shown. This option checks the interference between the components you selected as the first set and the remaining components that are displayed in the assembly.

Itself. This option checks the interference among the first set of components.

Output Options Area

The options in this area enable you to determine the format in which you need the output from the interference check. These options are discussed next.

Generate report. This option enables you to save the interference check results into a text file; the configuration of this file can be set in the **Report** tab of the same dialog box.

Interfering volumes. This option enables you to determine the state of the interfering volumes. The **Show** radio button allows you to display the interfering volume in the assembly. Whereas, the **Save as part** option allows you to save the interfering volume as a part file. This volume is saved as a separate part and is automatically grounded.

Hide parts not in Select Sets 1 and 2. This option enables you to hide the parts that are not included in sets 1 and 2.

Highlight interfering parts. As the name suggests, this option highlights the parts that have an interference in the assembly.

Dim parts with no interference. If this option is selected, the parts that do not have interference with any part are displayed as dim.

Hide parts with no interference. This option enables you to hide the parts that are not in any of the selection sets and do not take part in the interference check.

Ignore interferences between matching threads. On selecting this check box, the interference is not detected, if the thread pitch does not match between a bolt and a threaded hole with the same nominal diameters.

Ignore threaded fasteners interfering with non-threaded holes. This option ignores the interference between a threaded cylinder and a non threaded hole.

Report Tab

When you choose the **Report** tab, the **Interference Options** dialog box will be modified, as shown in Figure 12-9. The options in this tab are discussed next.

File name

The **File name** edit box is used to specify the name of the report file. You can choose the **Browse** button to locate the folder in which you want to save the file.

Part names

When you select this option, the names of the parts that are causing the interference will be listed in the report file.

Part centers of gravity

When you select this option, the centers of gravity of the interfering parts will be listed in the report file.

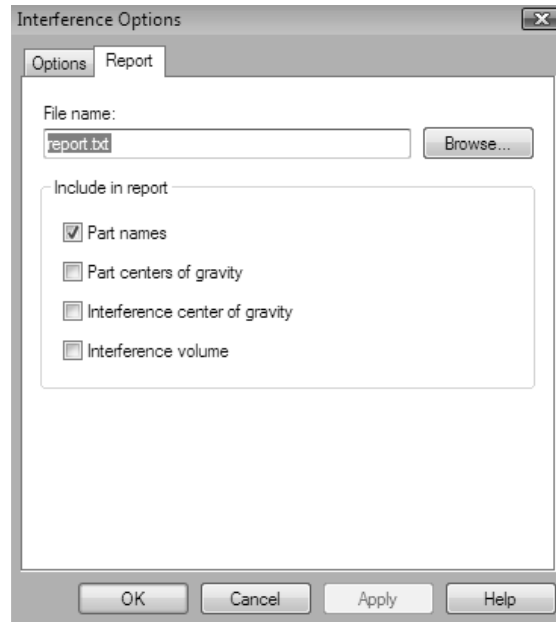


Figure 12-9 The **Report** tab in the **Interference Options** dialog box

Interference center of gravity

When you select this option, the center of gravity of the volume of the interference will be listed in the report file.

Interference volume

When you select this option, the volume of the interference will be listed in the report file.

Checking for the Interference

After setting the options for the interference check, you need to select the first set of components. You can select the components from the assembly window or from the **Pathfinder**. After selecting the first set of components, choose the **Accept** button on the **Check Interference** command bar or right-click to accept the selection. Now, you need to select the parts for set 2. After selecting the parts, right-click to process the interference check. If there is an interference, then depending on the output options you have set, the interference volume will be displayed. You can use the reference of the interference volume to edit the components for eliminating the interference.

CREATING THE EXPLODED STATE OF ASSEMBLIES

Ribbon: Tools > Environs > ERA



The exploded state of an assembly is created in the **Explode - Render - Animate** environment. To invoke this environment, choose **Tools > Environs > ERA** from the **Ribbon** in the **Assembly** environment. As soon as you enter **Explode - Render - Animate**

environment, the explode, render, and animate tools will be displayed in the **Ribbon**. The **Explode** group of the **Ribbon** provides tools to create the exploded state of an assembly. There are two methods of creating the exploded state of an assembly. These are discussed next.

Automatic Explode

Ribbon: Explode > Auto Explode



The first option is to create the exploded view automatically. Whenever you explode an assembly using this method, flowlines are automatically created. To create an automatic exploded state, choose the **Auto Explode** button from the **Explode** group of the **Ribbon**; the **Auto Explode** command bar will be displayed. The options in this command bar are discussed next.

Select Step



The **Select Step** button is chosen by default in the **Auto Explode** command bar. This button enables you to select the assembly that you want to explode.

Select Drop-down List

There are two options in the **Select** drop-down list. These are discussed next.

Top-level assembly

By default, the **Top-level assembly** option is selected in the **Select** drop-down list. As a result, the subassemblies in the assembly are considered as a single component while exploding. Choose the **Accept** button from the command bar to accept the **Top-level assembly** option. Once you accept it, the options in the command bar will change. Next, choose the **Automatic Explode Options** button from the command bar; the **Automatic Explode Options** dialog box will be displayed, as shown in Figure 12-10. Clear the **Bind all subassemblies** check box and select the **By subassembly level** radio button in the **Explode Technique** area, if it is not already selected. Choose the **OK** button from the **Automatic Explode Options** dialog box. Inactivate the **Automatic Spread Distance** button by choosing it from the **Auto Explode** command bar and set the distance value in the **Distance** edit box. Choose the **Explode** button from the command bar. Figure 12-11 shows the exploded state created using the **Auto Explode** tool.

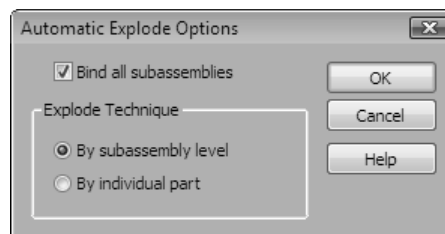


Figure 12-10 The Automatic Explode Options dialog box

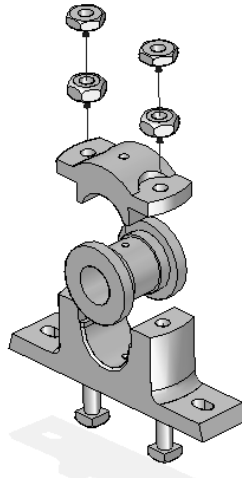


Figure 12-11 Exploded view of the Plummer Block assembly

Subassembly

This option allows you to explode the components of the selected subassembly only, as shown in Figure 12-12. The explosion procedure is similar to that discussed earlier in the top-level assemblies.

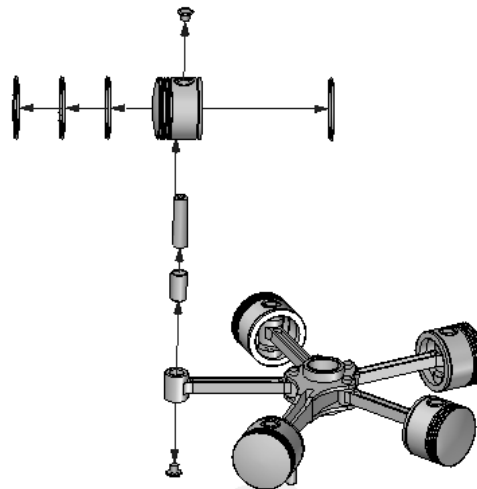


Figure 12-12 The selected subassembly exploded



Tip. The automatic explode method explodes the component of an assembly based on the relationships applied to it.

Unexploding Assemblies

Ribbon: Explode > Unexplode



To unexplode an assembly, choose the **Unexplode** button from the **Explode** group of the **Ribbon**; the **Solid Edge** information box will be displayed informing you that the

current explosion will be deleted, as shown in Figure 12-13. Choose the **Yes** button from this information box.

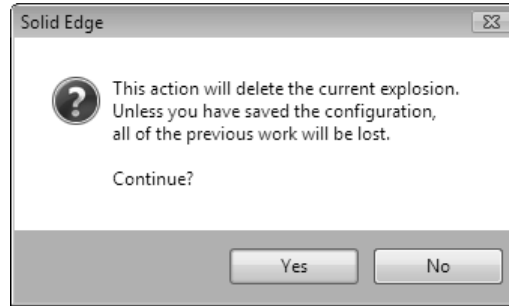


Figure 12-13 The Solid Edge information box

Exploding Assemblies Manually

Ribbon: Explode > Explode



The automatic explode method does not give the desired results every time. Therefore, the manual method is used to achieve the required exploded state. To explode the assembly manually, choose the **Explode** button from the **Explode** group of the **Ribbon**; the **Explode** command bar will be displayed and you will be prompted to select the parts you need to explode. Note that while selecting the parts, do not select the part that is stationary. Also, select only those components that will be exploded in the same direction. Remember that the order of the selection of the components should be such that the component closest to the stationary part in the exploded state is selected first. For example, in Figure 12-14, component A should be selected first and then the component B. The stationary part will remain stationary and the parts that were not selected will also remain at their position. After selecting the parts to be exploded, choose the **Accept** button. Now, select the part that will remain stationary and then select a face or reference plane on the stationary part to specify the direction of explosion. The explosion will take place normal to the selected plane. After you select the explode direction, the **Explode Options** dialog box will be displayed, as shown in Figure 12-15. The options in the **Explode Options** dialog box are discussed next.

Explode Technique Area

The options in this area are discussed next.

Move components as a unit

This option allows you to move the selected components as a single unit from their original position through the distance specified in the **Distance** edit box, refer to Figure 12-16.

Spread components evenly

This option will move the selected components individually from their original position through the distance specified in the **Distance** edit box, refer to Figure 12-17. The individual components are listed in the **Explode order** list box.

After specifying the required option, choose the **OK** button from the **Explode Options** dialog box to accept and exit it. Next, choose the **Explode** button from the **Explode** command bar; the component will explode in the graphics area depending upon the option specified, as shown in Figure 12-17. Choose **Finish** and then **Cancel** to exit exploding.

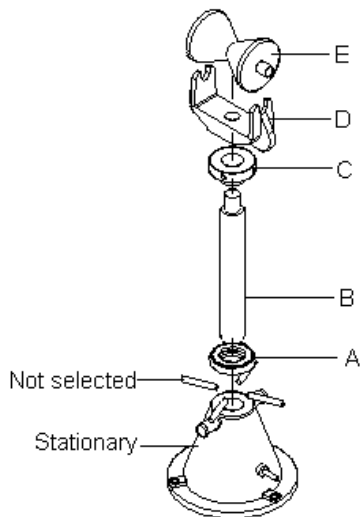


Figure 12-14 Exploded state of an assembly



Figure 12-15 The **Explode Options** dialog box

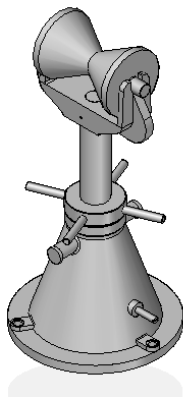


Figure 12-16 Exploding components as a single unit

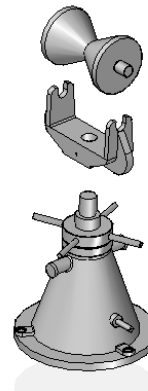


Figure 12-17 Exploding individual parts of components

Changing the Distance between Components



Ribbon: Explode > Drag Component



After exploding an assembly, you may need to modify the distance between the components or the orientation of components. To do so, choose the **Drag Component** button from the **Explode** group of the **Ribbon**; the **Drag Component** command bar will be displayed and you will be prompted to select a component. Move the cursor on the component to be moved; the component will get highlighted. Next, drag the cursor to move the highlighted component to the required position. In the **Drag Component** command bar, the **Move Dependent Parts** button is chosen by default. As a result, you can select and move a component along with its dependents. To move a single component, choose the **Move Selected Part** button from the command bar. Next, select the component and drag it to move. You can change the orientation of the components by choosing the **Rotate** button from the **Drag Component** command bar. Similarly, on choosing **Move Planar**, you can move the selected component in the same plane.

Repositioning the Parts

Ribbon: Explode > Reposition



When you are not able to move a part beyond its parent part in the exploded state of assembly, you can invoke the **Reposition** tool from the **Explode - Render - Animate** environment and reposition the selected component with respect to a reference part. To reposition a part, choose **Explode > Reposition** from the **Ribbon**; the **Reposition** command bar will be displayed and you will be prompted to select the part to reposition. Select the part, and then select another part. Next, select the direction in which you want to place the part. You will notice that the repositioned part changes its position in the exploded state of the assembly. Now, if required, you can move the repositioned part to the desired location.



Note

The distance and position of only the exploded parts can be modified.

Removing the Parts

Ribbon: Explode > Remove



The **Remove** button will be enabled in the **Explode** group of the **Ribbon** only after you select an exploded part. When you choose this button, the selected part will be hidden and moved to its original position in the assembly. To redisplay the hidden component, select the check box in front of the component in the docking window. Alternatively, right-click on the hidden component in the docking window and choose the **Show** option from the shortcut menu; the hidden part will be redisplayed.

Generating Flowlines

Ribbon: Explode > Flow Lines
View > Show > Flow Lines



When you create the exploded state of an assembly, flowlines are automatically generated. These flowlines indicate the direction of explosion of the component. The arrow at the end of the flowline is called flowline terminator. You can set the display of the flowline and the flowline terminator using the **Flow Lines** tool. Figures 12-18 and 12-19 show the exploded state without and with flowlines, respectively.

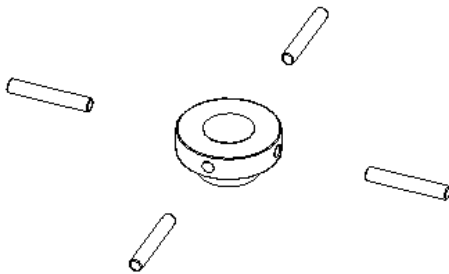


Figure 12-18 Exploded state without flowlines

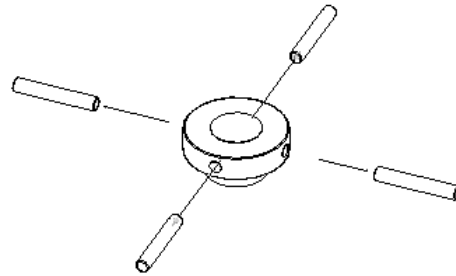


Figure 12-19 Exploded state with flowlines

TUTORIALS

Tutorial 1

Traditional Assembly

In this tutorial, you will create the exploded view of the Stock Bracket assembly created in Tutorial 1 of Chapter 11. The exploded state of the assembly is shown in Figure 12-20. After creating the exploded state, save the configuration with the name *Stock Bracket.cfg* at the location given below:

|Solid Edge|c12|Stock Bracket

(Expected time: 30 min)

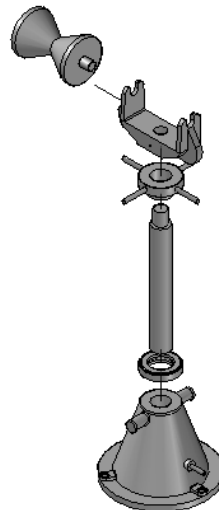


Figure 12-20 Exploded view of the Stock Bracket assembly

The following steps are required to complete this tutorial:


- a. Copy all part and assembly files of the Stock Bracket assembly from the *c11* folder to the *c12* folder. The files will be saved at *\Solid Edge\c12\Stock Bracket*.
- b. Open the assembly file of the Stock Bracket in the **Assembly** environment.
- c. Create the exploded state of the assembly, refer to Figure 12-21.
- d. Save the exploded state of the assembly in a configuration file.

Copying Files to the Current Folder

1. Copy all part and assembly files of the Stock Bracket assembly from the *c11* folder to *\Solid Edge\c12\Stock Bracket* folder. Remember that the extension of the part files is *.par* and that of assembly files is *.asm*.

Creating the Exploded State of the Assembly

To create the exploded state of the assembly, you need to invoke the **Explode - Render - Animate** environment in the **Assembly** environment.

1. Open the *stock bracket.asm* file in the **Assembly** environment.
2. Choose **Tools > Environs > ERA** from the **Ribbon**; the **Explode - Render - Animate** environment is invoked.
3. Choose the **Explode** button from the **Explode** group of the **Ribbon**; you are prompted to click on the parts to be exploded. Remember that the part that has to be kept closest to the stationary part must be selected first. In this assembly, the stationary part is the Stock Support Base. 
4. Select the components in the following order, Thrust Bearing, Support Adjusting Screw, Adjusting Screw Nut, Support Roller Bracket, and Stock Support Roller.
5. Choose the **Accept** button from the **Explode** command bar; you are prompted to click on the stationary part.
6. Select the Stock Support Base as the stationary part; you are prompted to select a face of the stationary part. This face is needed to determine the direction of explosion.
7. Select the top face of the Stock Support Base; a red arrow pointing in upward direction is displayed. If the arrow does not point upward, move the cursor upward to point it upward.
8. Click to specify the direction when the arrow points in the upward direction.
9. As you specify the direction, the **Explode Options** dialog box is displayed. In this dialog box, select the **Spread components evenly** radio button and choose the **OK** button to close it. Next, set the value **30** in the **Distance** edit box in the **Explode** command bar.
10. Choose the **Explode** button to explode the assembly.

11. Choose the **Fit** button from the status bar; the exploded view of the assembly is displayed, as shown in Figure 12-21.
12. To exit the **Explode** tool, choose the **Finish** button and then the **Cancel** button from the command bar.

Note that the exploded view obtained is not the desired view. This is because the Stock Support Roller is not in the correct direction. Therefore, you need to further apply some operations on it to get the desired view.

Collapsing the Component

Ideally, the Stock Support Roller needs to be exploded on the right or left of the Stock Roller Bracket. Therefore, you need to first unexplode the Stock Support Roller and then explode it again in the horizontal direction.

1. Select the Stock Support Roller and choose the **Collapse** button from the **Explode** group of the **Ribbon**; the selected component moves to its original position with respect to its parent component, as shown in Figure 12-22. Alternatively, select the component and right-click to invoke a shortcut menu. Choose **Collapse** from the shortcut menu.



Figure 12-21 Exploded view of the assembly

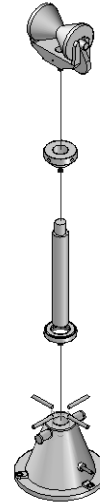


Figure 12-22 Exploded view after collapsing the Stock Support Roller

Exploding the Stock Support Roller

Now, you need to explode the Stock Support Roller in the horizontal direction.

1. Choose **Explode > Explode** from the **Ribbon**; you are prompted to click on the parts to be exploded.
2. Select the Stock Support Roller from the assembly window and choose the **Accept** button; you are prompted to select the stationary part.

3. Select the Support Roller Bracket as the stationary component; you are prompted to select a face on the stationary component to specify the direction of explosion.
4. Select the face of the stationary part, as shown in Figure 12-23, to display a red arrow on it.
5. When the arrow points in the direction shown in Figure 12-23, click to specify the direction.
6. Enter **25** in the **Distance** edit box and then choose the **Explode** button from the **Explode** command bar; the selected component is exploded, as shown in Figure 12-24.

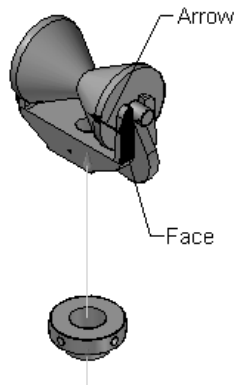


Figure 12-23 Arrow showing the direction of explosion

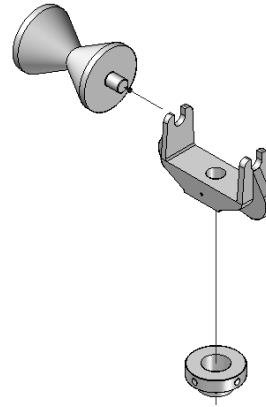



Figure 12-24 The selected part exploded in the specified direction

Reducing the Gap between Components

The distance between the components in the exploded view is large. You need to reduce the gap between the components.

1. Choose **Explode > Drag Component** from the **Ribbon**; the **Drag Component** command bar is displayed. 
2. Move the cursor toward the part that is nearest to the stationary component to highlight it and then drag it toward the stationary part to reduce the gap between them.
3. After dragging the first component, select the second component and drag it closer to the stationary part, as shown in Figure 12-25.
4. Similarly, drag other components to reduce the gap. The exploded state of the assembly, after reducing the gap, is shown in Figure 12-26.

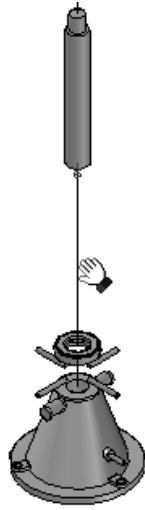


Figure 12-25 Dragging the components

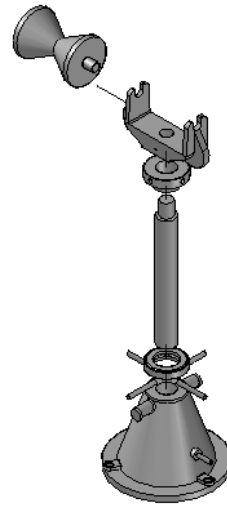



Figure 12-26 Assembly after reducing the gap



Tip: You can also select the instances and specify a new distance value for the exploded parts in the command bar that is displayed on selecting the exploded instances.

Moving the Components

The four instances of the Adjusting Nut Handle are not at the desired location. Therefore, you need to move them manually one-by-one.

1. Choose **Explode > Drag Component** from the **Ribbon**; the **Drag Component** command bar is displayed and you are prompted to select the component to move. 
2. Select any one of the instances of the Adjusting Nut Handle and choose the **Accept** button from the **Drag Component** command bar; a triad is displayed on the selected instance, as shown in Figure 12-27.
3. Select the Z axis of the triad by clicking on it. Drag the component and place it, as shown in Figure 12-28.



Tip: To move the Adjusting Nut Handle more precisely to the desired location in the assembly, zoom in and view the assembly location of this part from the front view. While viewing the part from the front view, move it to the desired location. You can also change the display of the assembly by choosing **View Styles > Visible and Hidden Edges** from the status bar.

4. Similarly, drag and place other instances of the Adjusting Nut Handle. To select the next instance, choose the **Select Part** button from the **Drag Component** command bar.

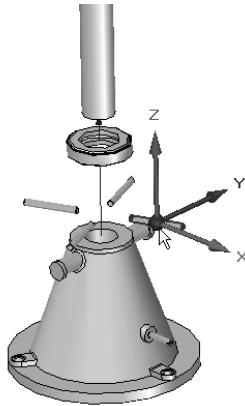


Figure 12-27 Moving the component along the z axis

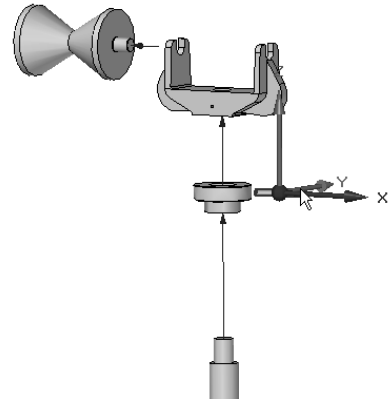


Figure 12-28 The selected part placed at the required position

The assembly after moving all instances of the Adjusting Nut Handle is shown in Figure 12-29.

After moving the instances, you will notice that the instances still touch the Adjusting Screw Nut. Therefore, you need to move all instances of the Adjusting Nut Handle away from the Adjusting Screw Nut by using the **Drag Component** tool.

5. Choose the **Drag Component** button from the **Explode** group of the **Ribbon**; the **Drag Component** command bar is displayed.
6. Select any one instance of the Screw Nut Handle and choose the **Accept** button from the command bar. Drag the instance to the required position. Alternatively, enter the distance value in the **Distance** edit box; the part moves to a new location.
7. Similarly, drag all instances of the Adjusting Nut Handle one-by-one and place them, as shown in Figure 12-30.

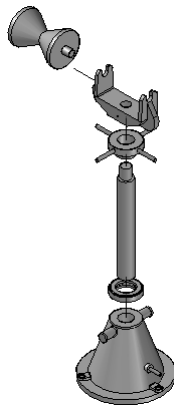


Figure 12-29 Assembly after moving all instances of the Adjusting Nut Handle

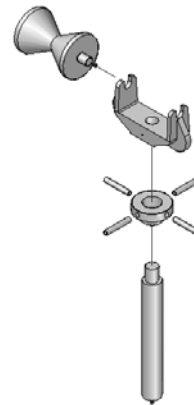


Figure 12-30 The Adjusting Nut Handle after dragging all instances

8. Exit the **Drag Component** tool by invoking the **Select** tool; all parts are exploded, as required.

Exploding the Last Component

The last component is the Adjusting Screw Guide and you need to explode it manually.

1. Choose **Explode > Explode** from the **Ribbon**.
2. Select the Adjusting Screw Guide from the assembly and choose the **Accept** button.
3. Select the Stock Support Base as the stationary part.
4. Select the face of the stationary part, as shown in Figure 12-31, to specify the direction of explosion.
5. Click when the red arrow points away from the base.
6. Choose the **Explode** button from the command bar; the selected part will explode in the specified direction, as shown in Figure 12-32.
7. Choose **Finish** and then **Cancel** to exit the **Explode** tool.

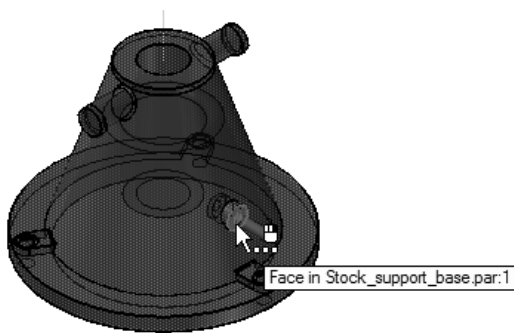


Figure 12-31 Arrow showing the direction of explosion

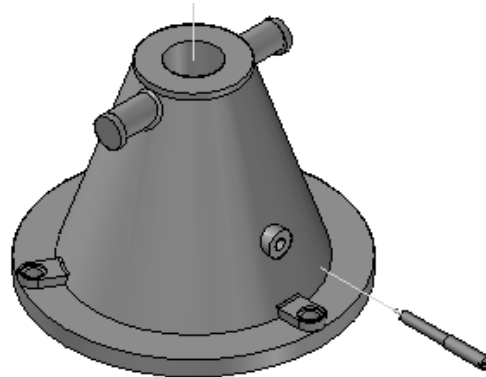


Figure 12-32 The selected part exploded in the specified direction

Saving the Configuration

You may need to use this exploded state of the assembly in the later chapters. Therefore, you need to save the exploded state of this assembly in a configuration file. To create the configuration file, follow the steps given next.

1. Choose the **Configurations** button in the title bar of the **Configurations** group of the **Ribbon**; the **Display Configurations** dialog box is displayed.
2. Enter **Stock bracket** as the name of the configuration in the **Configuration name** text box of the **Display Configurations** dialog box.

3. Choose the **Save** button and then the **Close** button.
4. To return to the assembly window, choose **Close > Close ERA** from the **Ribbon**.

Tutorial 2

Traditional Assembly

In this tutorial, you will create the Radial Engine assembly shown in Figure 12-33. This assembly will be created in two parts, the subassembly and the main assembly. The exploded state of the assembly is shown in Figure 12-34. The views and dimensions of all components of this assembly are shown in Figures 12-35 through 12-38. After creating the assembly, save it with the name *Radial Engine.asm* at the location given below:

\\Solid Edge\\c12\\Radial Engine

(Expected time: 3 hrs)

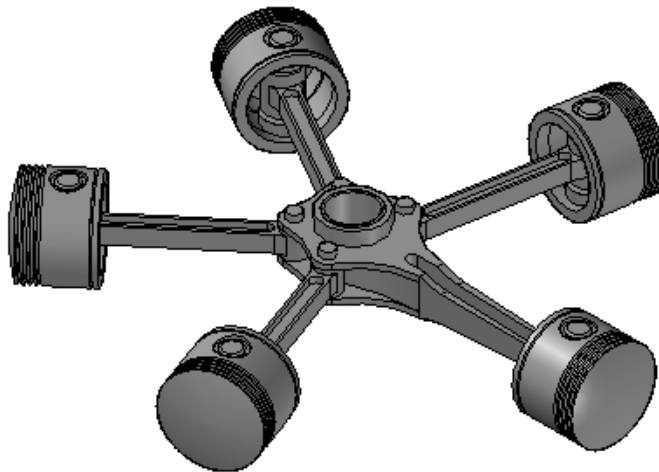


Figure 12-33 The Radial Engine assembly

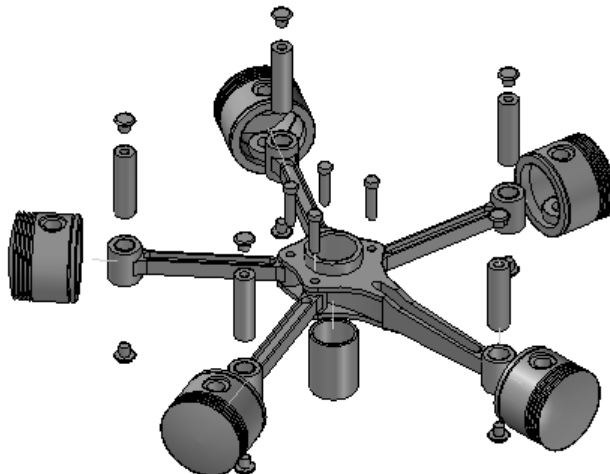


Figure 12-34 Exploded view of the assembly

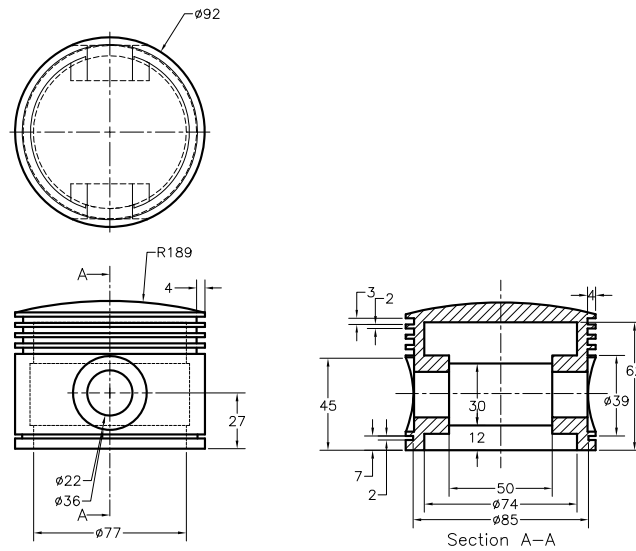


Figure 12-35 Views and dimensions of the Piston

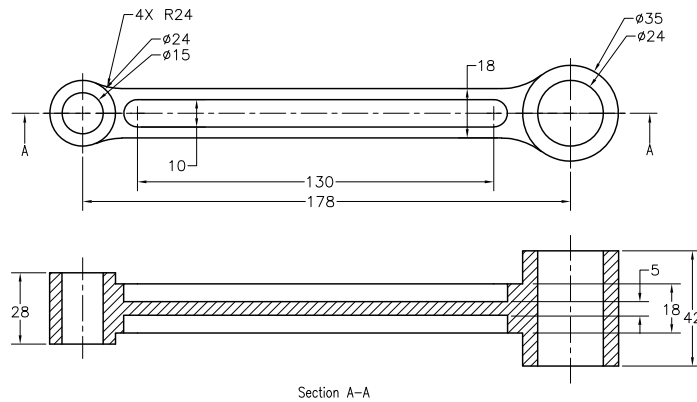


Figure 12-36 Views and dimensions of the Articulated Rod

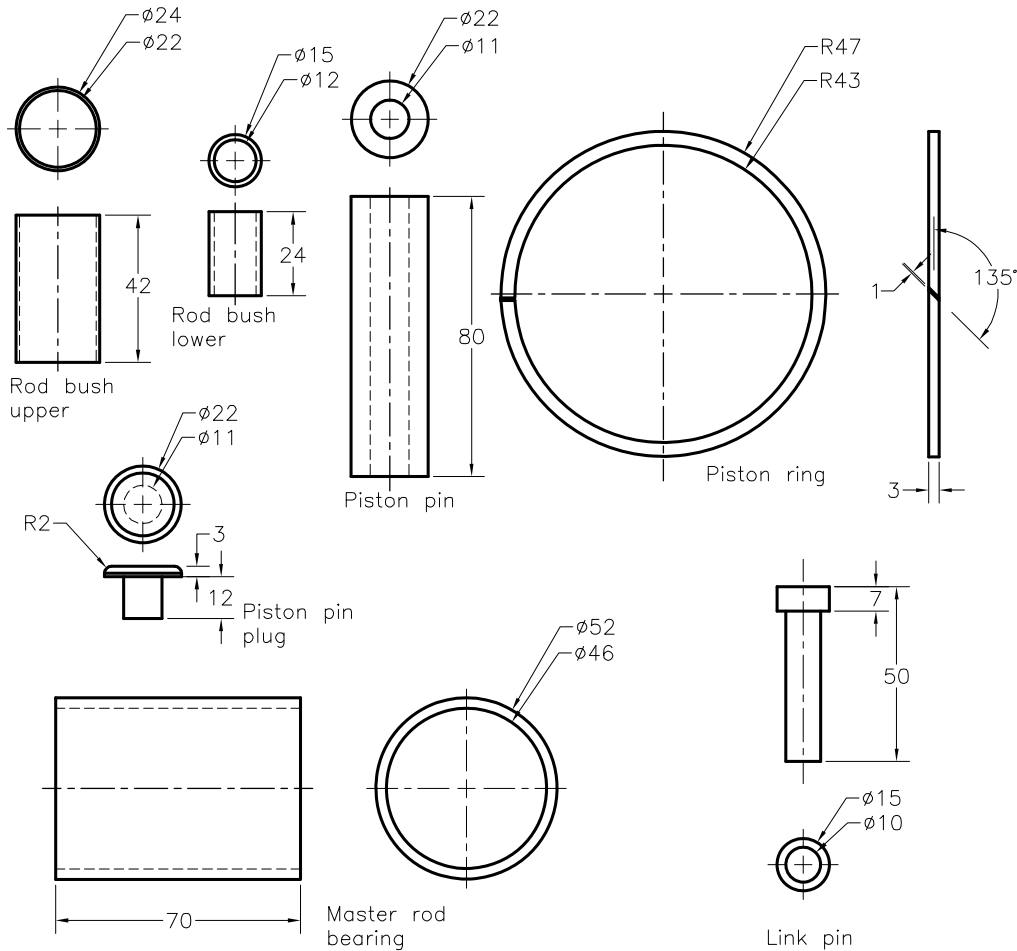


Figure 12-37 Views and dimensions of other components

This assembly has a large number of components, so you need to divide it into two assemblies such that one is the subassembly and the other is the main assembly. First you need to create the subassembly, which consists of the Articulated Rod, Piston, Piston Rings, Piston Pin, Rod Bush Upper, Rod Bush Lower, and Piston Pin Plug. After creating this subassembly, you will create the main 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 main assembly, you will assemble the subassembly with the main assembly.

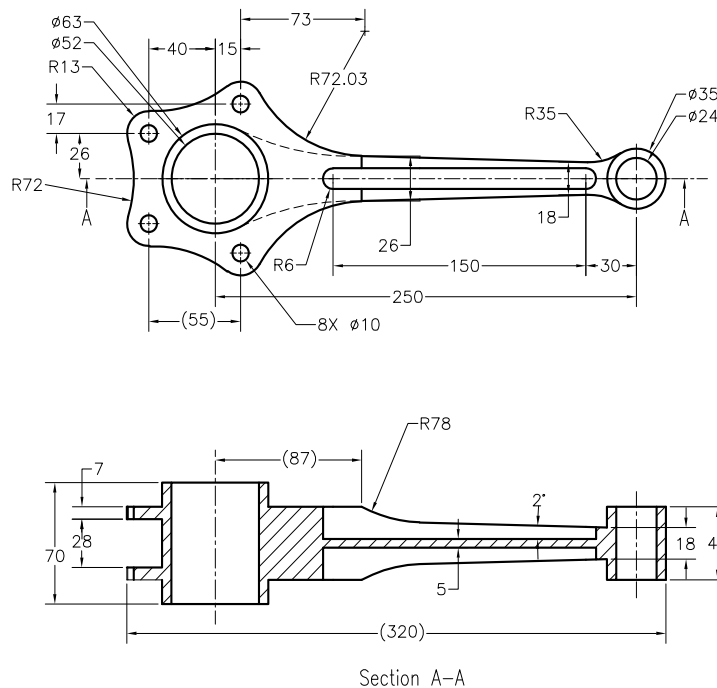


Figure 12-38 Views and dimensions of the Master Rod

The following steps are required to complete this tutorial:

- Create all components of the assembly in the **Part** environment and save them in the *Radial Engine* folder.
- Start a new assembly file and assemble the components to create the subassembly.
- Start a new assembly file and assemble the components to create the main assembly.
- Assemble the subassembly to the main assembly.

Creating Components

- Create a folder with the name *Radial Engine* at `|Solid Edge|c12`. Create all components in the individual part files and save them in this folder.

Creating the Subassembly

As discussed earlier, you first need to create the subassembly and then assemble it with the main assembly.

- Start a new file in the **Assembly** environment. Name this file *Piston Articulated Rod subassem* and save in the same folder in which the parts were saved.
- Choose the **Parts Library** button from the bottom of the docking window. Now, drag and drop the Articulated Rod in the assembly window.

3. Assemble the Rod Bush Upper with the Articulated Rod using the assembly relationships. Figure 12-39 shows the sequence in which you need to assemble the components. The exploded view and the flowlines are shown for your reference only. The assembly after assembling the Articulated Rod, Rod Bush Upper, Rod Bush Lower, Piston, Piston Pin, and Piston Pin Plugs is shown in Figure 12-40.
4. Insert the Piston Ring in the subassembly and assemble it with the Piston, as shown in Figure 12-40. Similarly, assemble the other instances of the Piston Ring with the Piston.

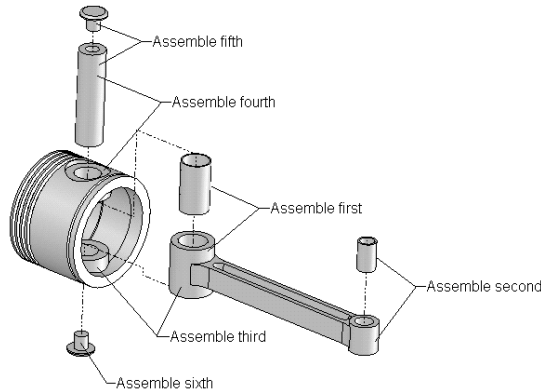


Figure 12-39 Assembly sequence of the Articulated Rod, Piston, Piston Pin Plug, Rod Bush Upper, and Rod Bush Lower

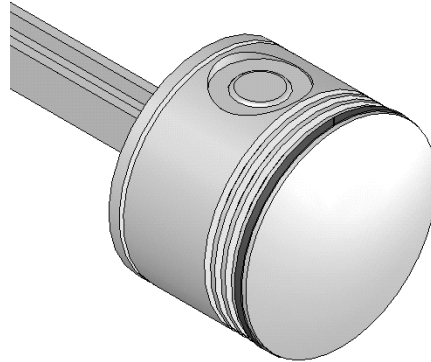


Figure 12-40 First instance of the Piston Ring assembled with the Piston

Now, you need to change the color of the Piston Ring.

5. Choose **View > Style > Color Manager** from the **Ribbon**; the **Color Manager** dialog box is displayed.
6. Select the **Use individual part styles** radio button from the dialog box and choose **OK**.
7. Now, select the Piston Ring from the docking window or from the subassembly.
8. In the **Face Style** drop-down list of the **Style** group in the **Ribbon**, select the color that you want to apply to the selected component. On doing so, the **Multiple Part Occurrences** message box is displayed prompting you to apply the selected style to the selected part only or all of its occurrences.
9. Choose the **All Occurrences** button to apply the selected style to all occurrences of the selected part. The subassembly after assembling the Piston Rings and changing their color is shown in Figure 12-41. Next, save and close the subassembly file.

Creating the Main Assembly

Next, you need to create the main assembly and then assemble the subassembly with it.

1. Start a new file in the Assembly environment and save it with the name **Radial Engine Assembly** in the same folder in which the parts were saved.

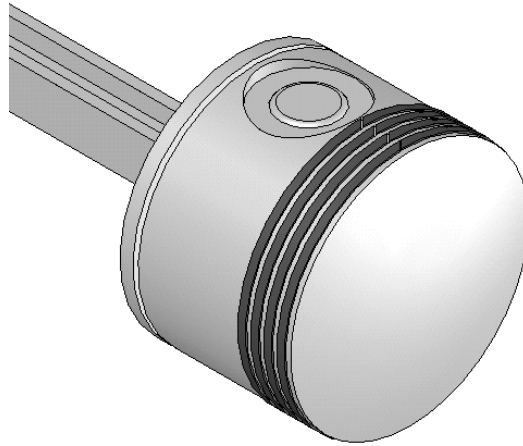


Figure 12-41 Subassembly after assembling the Piston Rings and changing their color

2. Place the Master Rod in the assembly so that it automatically gets assembled with the assembly reference planes.
3. After placing the first component, use the assembly relationships to place the Piston, and then the Piston Pin, Piston Pin Plugs, Piston Rings, Rod Bush Upper, and Master Rod Bearing one-by-one.

The components after being assembled with the main assembly will be displayed as shown in Figure 12-42.

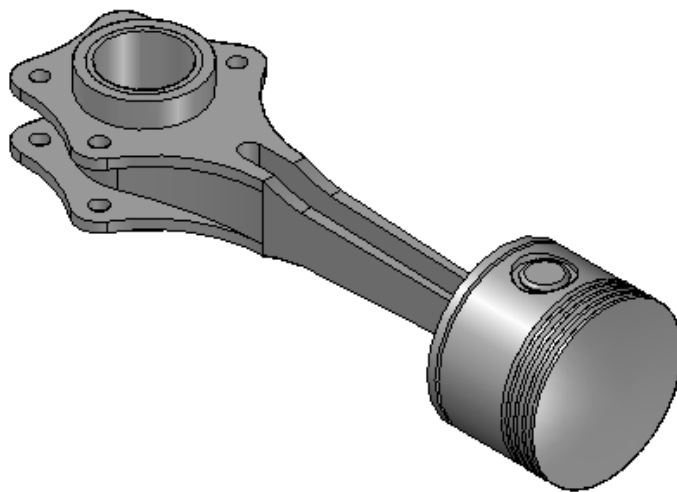


Figure 12-42 Components assembled with the Main assembly

Assembling the Subassembly with the Main Assembly

Next, you need to assemble the subassembly with the main assembly using the assembly relationships.

1. If the parts list is not open, then choose the **Parts Library** button from the docking window.
2. Right-click in the docking window and choose the **Use Configurations** option. Otherwise, after you place the subassembly in the assembly window, some components in the subassembly will not be activated.
3. Drag and drop the *Piston Articulation Rod subassem* into the assembly window; the **Use Configuration** dialog box is displayed. In this dialog box, the **Activate all parts** radio button is selected by default.
4. Choose the **OK** button to exit the dialog box.
5. Assemble the subassembly with the main assembly using the **Mate**, **Axial Align**, and **Angle** assembly relationships. The first instance of the subassembly after assembling with the main assembly is shown in Figure 12-43. Note that while applying the **Angle** relationship, you need to choose the **4** button from the **Angle Format** flyout of the **Placement** rollout in the **Angle** command bar.

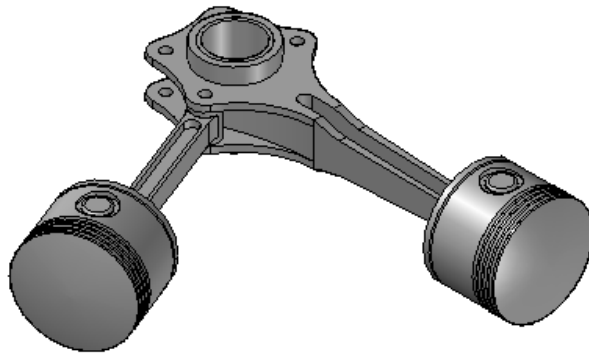


Figure 12-43 The first instance of the subassembly assembled with the main assembly

Refer to Figure 12-44, which shows the assembly structure that will help you in assembling the instances of the subassembly.

Assembling the Link Pin

After assembling the subassembly with the main assembly, you need to assemble the Link Pin with the main assembly.

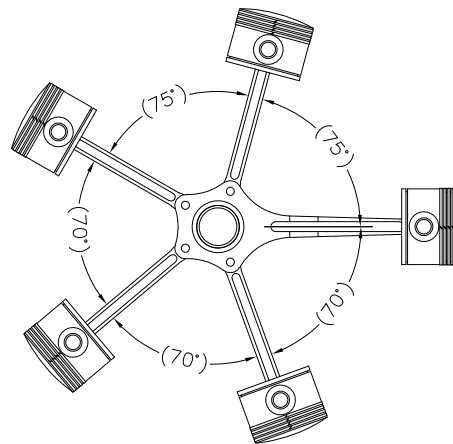


Figure 12-44 Assembly structure

1. Place the Link Pin in the current assembly and assemble it with the main assembly by using the assembly relationships. Figure 12-45 shows the first instance of the Link Pin assembled with the main assembly.
2. Before placing other instances of the Link Pin, invoke the **Options** dialog box and then select the **Automatically Capture Fit while placing parts** check box from it.

Figure 12-46 shows all the instances of the Link Pin assembled with the main assembly.

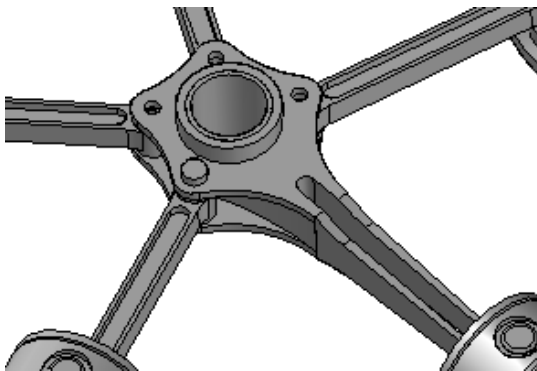


Figure 12-45 First instance of the Link Pin assembled with the main assembly

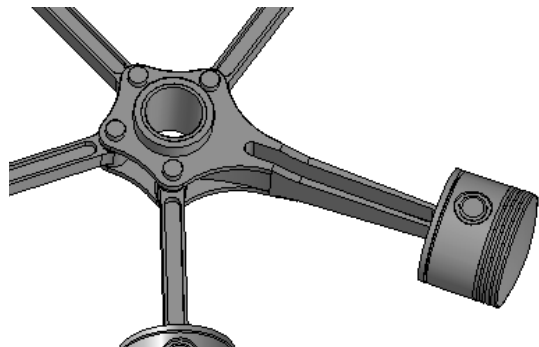


Figure 12-46 All instances of the Link Pin assembled with the main assembly

3. Save the assembly.

Tutorial 3

Synchronous Assembly

In this tutorial, you will edit the Synchronous parts of the Plummer Block assembly created in Exercise 1 of Chapter 11, as shown in Figure 12-47. Modify the width of the components from **46** to **56** mm. Also, you need to remove the taper of the base element. The resultant model is shown in Figure 12-48. Save the resultant model with the name *Plummer Block* at the location given below:

`\\Solid Edge\\c12\\Plummer Block`

(Expected time: 30 min)

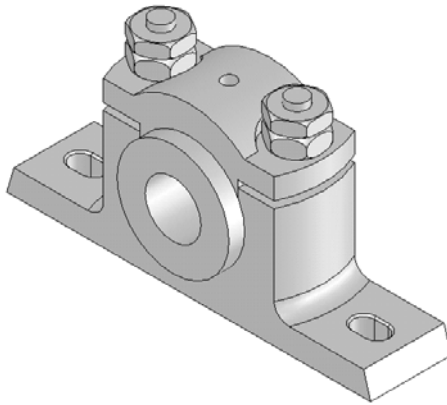


Figure 12-47 The original Plummer Block assembly

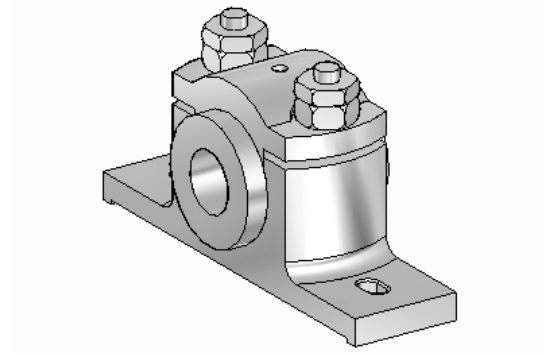


Figure 12-48 The Plummer Block assembly after editing

The following steps are required to complete this tutorial:

- Copy all part and assembly files of the Plummer Block assembly from *c11* folder to the *c12* folder. The files will be saved at `\\Solid Edge\\c12\\Plummer Block`.
- Open the file of the Plummer Block assembly.
- Modify the width of the base component, refer to Figure 12-49.
- Modify the width of the Brasses, refer to Figures 12-50 and 12-51.
- Remove the taper of the base component, refer to Figure 12-52.

Copying Files to the Current Folder

- Copy all part and assembly files of the Plummer Block assembly from the *c11* folder to the *c12\\Plummer Block* folder. Remember that the extension of the part files is *.par* and that of assembly files is *.asm*.
- Choose the **Open** button from the **Quick Access toolbar**; the **Open File** dialog box is displayed. Browse to the *plummer block.asm* file. Select the **Activate all** radio button to activate the opened components. Next, choose the **Open** button; the file is opened in the **Synchronous Assembly** environment, as it was created and assembled in the Synchronous environment.

Modifying the Width of the Casting and Cap

To modify the width of the component, you need to select the face of the base component first and then modify it. Next, you need to make the other components coincident with the base component.

1. To modify the width of the base component, you need to move its face to a new position. To select the face of the component, choose the black arrow on the **Select** button of the **Ribbon**; a flyout is displayed. Choose **Faces Priority** option from the flyout displayed.
2. Move the cursor on the front face of the base component; the face gets highlighted. Select the front face of the base component.
3. On selecting the front face, the steering wheel, **QuickBar**, **Live Rules**, and dimensions that were added to the feature are displayed.
4. Unlock the locked dimensions. Note that the locked dimensions will be red in color.
5. Click on the primary axis of the steering wheel; an edit box is displayed on the component. Enter **5** in the edit box to specify the distance to move the front face, as shown in Figure 12-49. On doing so, you will notice that the face moves toward both sides. But this will happen only when the base component is extruded symmetrically.

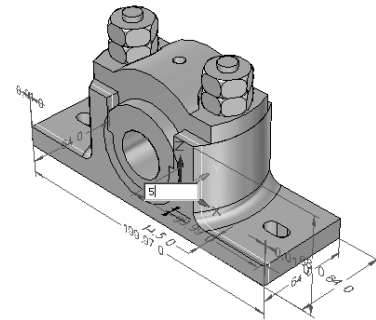


Figure 12-49 Moving the front face of the base component

The base component gets modified.

Similarly, modify the Cap component.

Modifying the Width of the Brasses

Next, you need to modify the **Brasses**. You cannot modify them in a single modification step because they do not extrude symmetrically. Therefore, you need to modify each face one by one. In order to do so, first you need to suppress the cap, bolts, and lock nuts.

1. In the **Pathfinder** tab of the docking window, clear the check boxes beside the cap, bolts, nuts, and lock nuts; the respective components get hidden.
2. Invoke the **Select** tool and select the inner face of the outer cylinder; the steering wheel and the **QuickBar** will be displayed.
3. Click on the primary axis of the steering wheel and move the face toward the planar face of the base component. Next, select the end keypoint of the base component; the inner face of the outer cylinder will be in plane with the base component, as shown in Figure 12-50.
4. Invoke the **Select** tool and select the outer face of the outer cylinder; the steering wheel is displayed again.

5. Click on the primary axis of the steering wheel; an edit box is displayed. Enter **7** in the edit box; both the outer cylinders get modified based on the newly specified width.
6. Now, select the inner face of the second outer cylinder; the steering wheel is displayed, as shown in Figure 12-51. Click on the primary axis and move the face outward; an edit box is displayed. Next, enter **3** in it; both the second outer cylinders get modified.

Now, you need to display all hidden components.

7. In the **PathFinder** tab of the docking window, select the check boxes beside the hidden components.

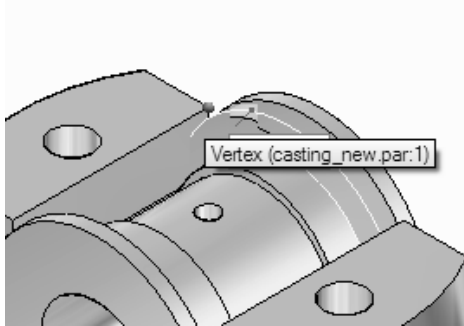


Figure 12-50 *Modifying the outer cylinders of the **Brasses***

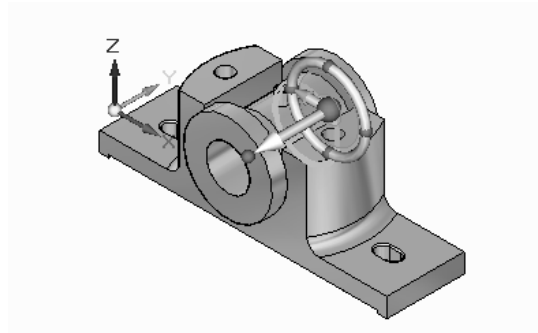


Figure 12-51 *Modifying the next outer cylinders of the **Brasses***

Removing the Taper of the Base Component

1. Select the tapered face of the base component, as shown in Figure 12-52; the steering wheel, the **QuickBar**, and **Live Rules** are displayed for the selected face. Remember that to select the face, you need to choose **Home > Selection Filters > Smart Priority > Faces** from the **Ribbon**.
2. Move the steering wheel to the edge of the tapered face, as shown in Figure 12-52.
3. Click on the Torus of the steering wheel; an edit box along with the preview of rotation is displayed on the screen.
4. Enter **-16** in the edit box to make it perpendicular to the base.

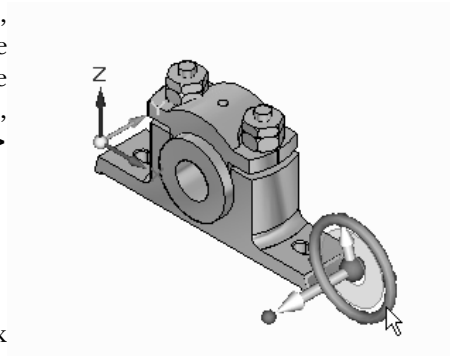


Figure 12-52 *Rotating the face of the base component*

Similarly, rotate the other tapered face also. The resultant model will be as shown in Figure 12-48.

**Note**

If you do not get the desired result on entering **-16** in step 4, enter **16**.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. You can delete an existing assembly relationship by selecting it from the bottom pane of the **Pathfinder** tab in the docking window and then pressing the DELETE key. (T/F)
2. The bottom pane of the **Pathfinder** tab in the docking window is used to view and modify the relationships between the selected part and the other parts in the assembly. (T/F)
3. To create subassemblies, you need to follow a different procedure than the one required for creating assemblies. (T/F)
4. By simplifying assemblies, you can hide components at any stage of the design cycle. (T/F)
5. Before sending the part file and the assembly for detailing and drafting, it is essential to check for interference in the assembly. (T/F)
6. You cannot set output options before starting the interference check. (T/F)
7. The _____ is the file extension of the files created in the **Assembly** and **Synchronous Assembly** environments of Solid Edge.
8. The _____ environment in the **Assembly** and **Synchronous Assembly** environments is used to create the exploded states of the assembly.
9. You can turn on the display of a hidden component by selecting the hidden component and choosing the _____ option from the shortcut menu that is displayed on right-clicking.
10. To explode an assembly manually, choose the _____ button from the **Explode** group of the **Ribbon** in the **Explode-Render-Animate** environment.

Review Questions

Answer the following questions:

1. Which of the following options is used to open a component separately in a part file?
 - (a) **Modify**
 - (b) **Open**
 - (c) **Open in Solid Edge**
 - (d) None of these

2. Which of the following environments within the **Assembly** or **Synchronous Assembly** environment is used to create the exploded state of an assembly?
- (a) **Virtual Studio** (b) **Explode-Render-Animate**
(c) **Motion** (d) None of these
3. Which of the following radio buttons should be selected in the **Open File** dialog box so that all components of the assembly to be opened are activated?
- (a) **Activate all** (b) **Inactivate all parts**
(c) **Use all simplified parts** (d) None of these
4. Which of the following options should be chosen from the shortcut menu to hide a component?
- (a) **Show** (b) **Hide**
(c) **Activate** (d) None of these
5. You cannot modify the dimensions of a component in the **Assembly** environment. (T/F)
6. In the exploded state, when you choose the **Remove** button from the **Explode** group of the **Ribbon** in the **Explode-Render-Animate** environment, the selected part gets hidden and it moves to its original position in the assembly. (T/F)
7. The exploded state of an assembly is created by changing the environment to the **Explode-Render-Animate** environment in the **Assembly** or **Synchronous Assembly** environment. (T/F)
8. You can set the transparency of any component to simplify the assembly. (T/F)
9. The automatic explode method does not give the desired results every time. Therefore, the manual method is used to achieve the desired exploded state. (T/F)
10. When you place a subassembly in the main assembly, an assembly icon is displayed with the name of the subassembly in the docking window. (T/F)

Exercise

Exercise 1

Create the Shaper Tool Head assembly, as shown in Figure 12-53. After creating the assembly, create its exploded state, as shown in Figure 12-54. The dimensions of the model are given in Figures 12-55 through 12-59. After creating the assembly, save it with the name *Shaper.asm* at the location given below:

|Solid Edge|c12|Shaper Tool Head

(Expected time: 4 hrs)

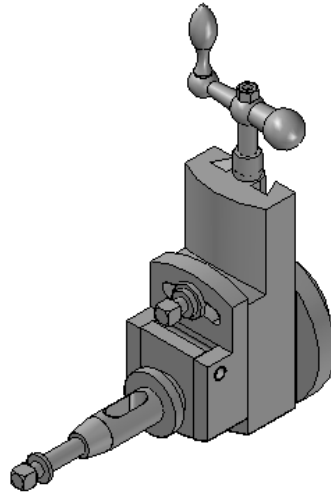


Figure 12-53 Shaper Tool Head assembly

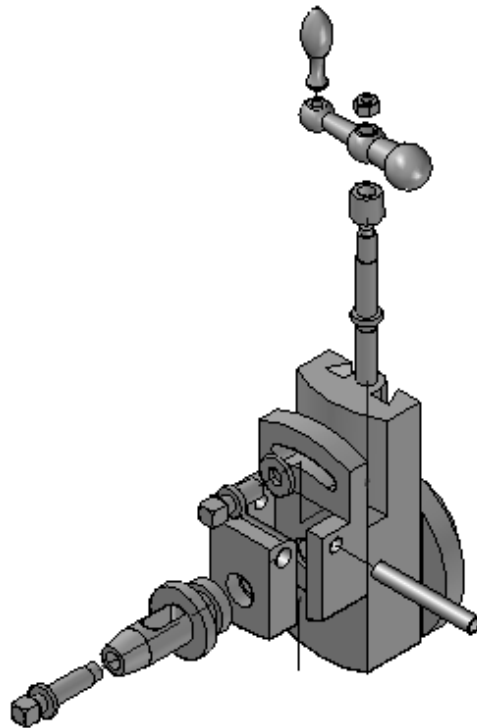


Figure 12-54 Exploded state of the assembly

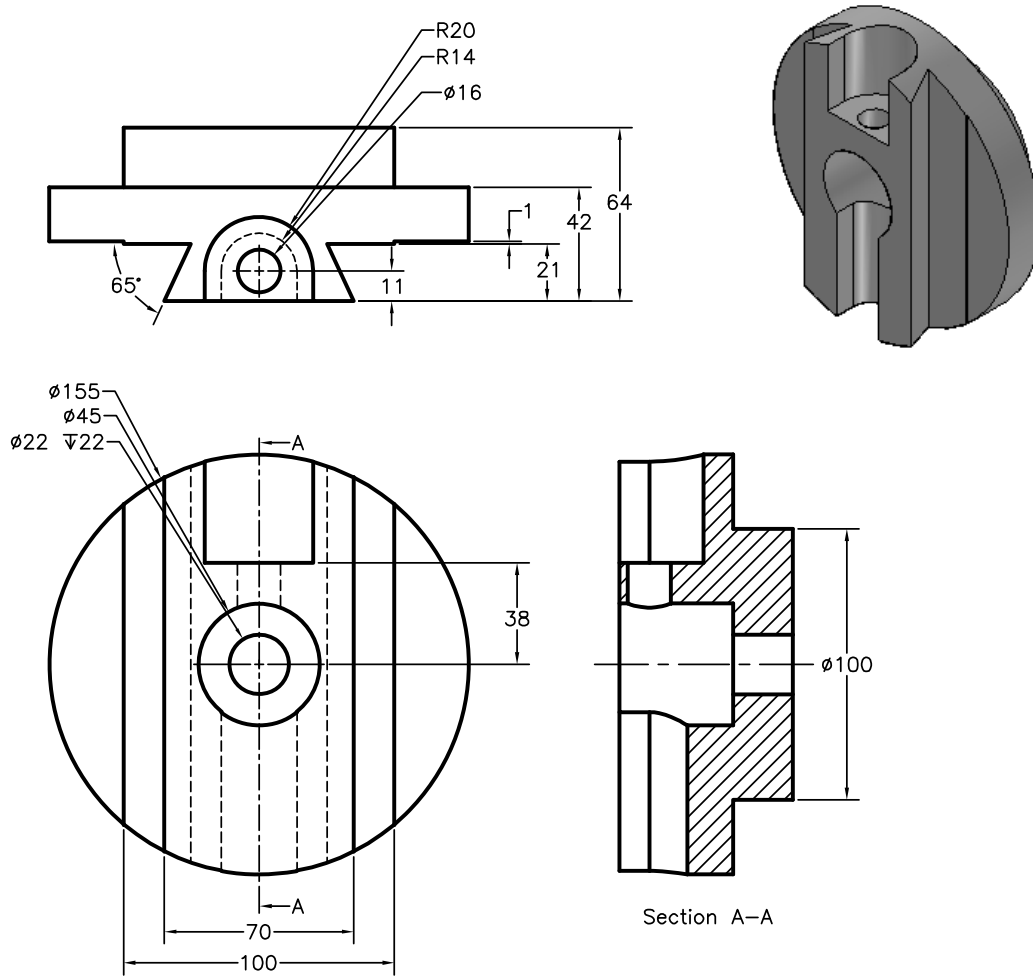


Figure 12-55 Views and dimensions of the Back Plate

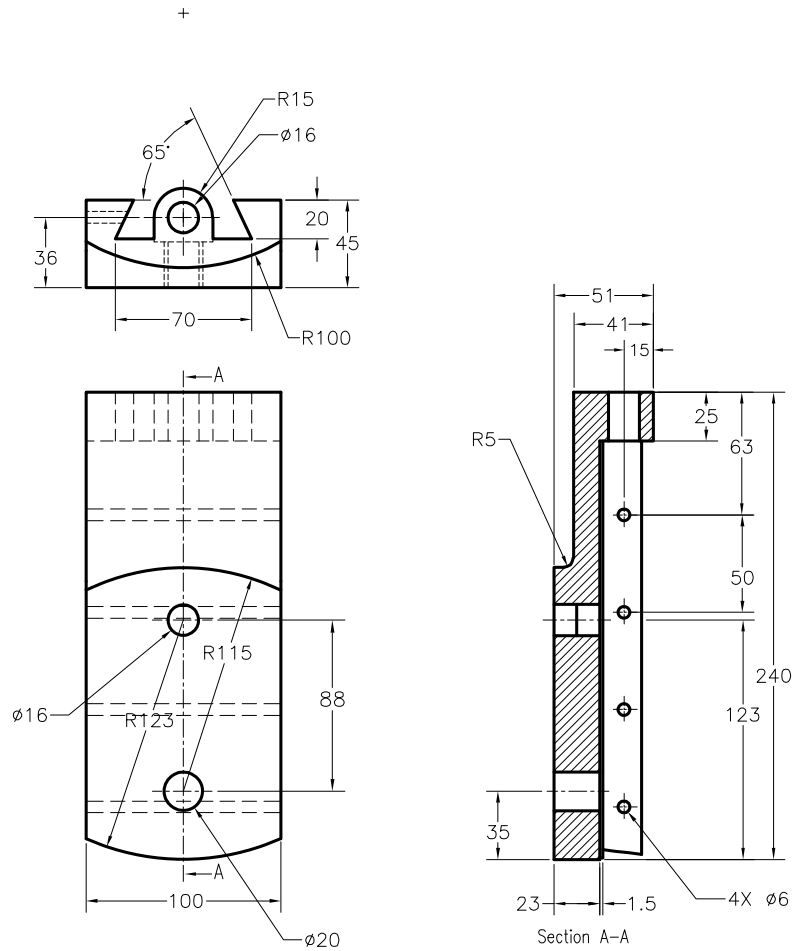


Figure 12-56 Views and dimensions of the Vertical Slide

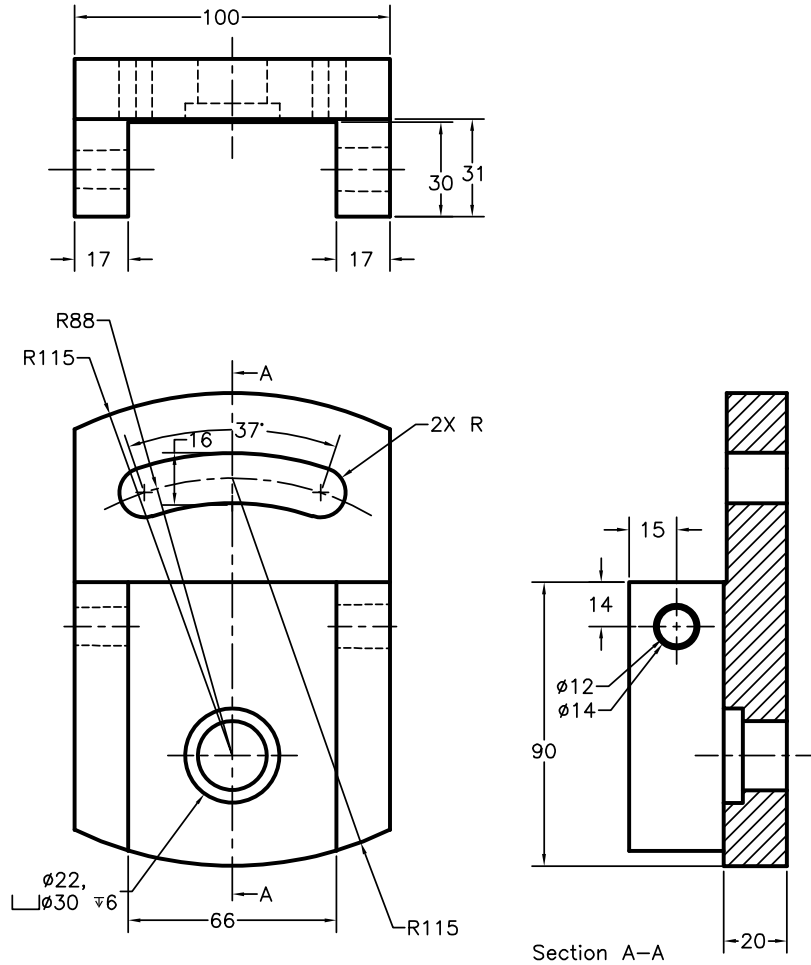


Figure 12-57 Views and dimensions of the Swivel Plate

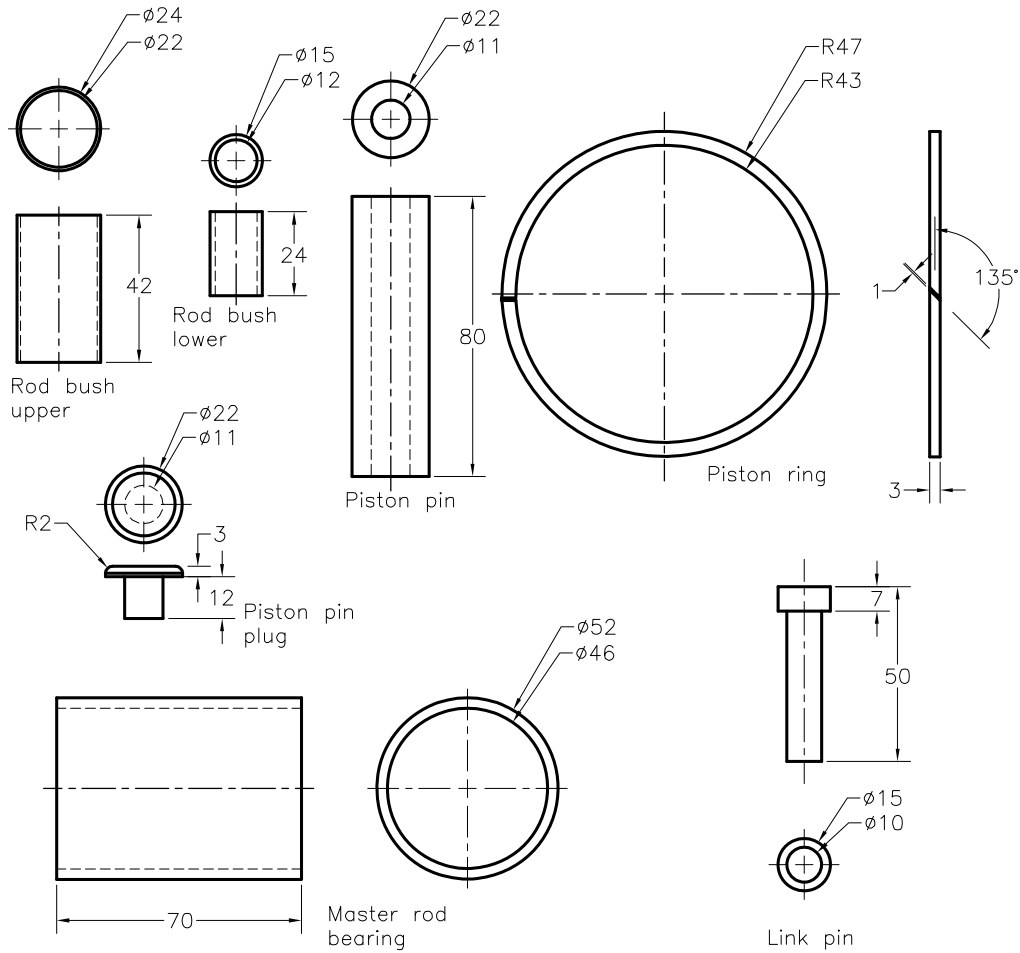


Figure 12-58 Views and dimensions of various components

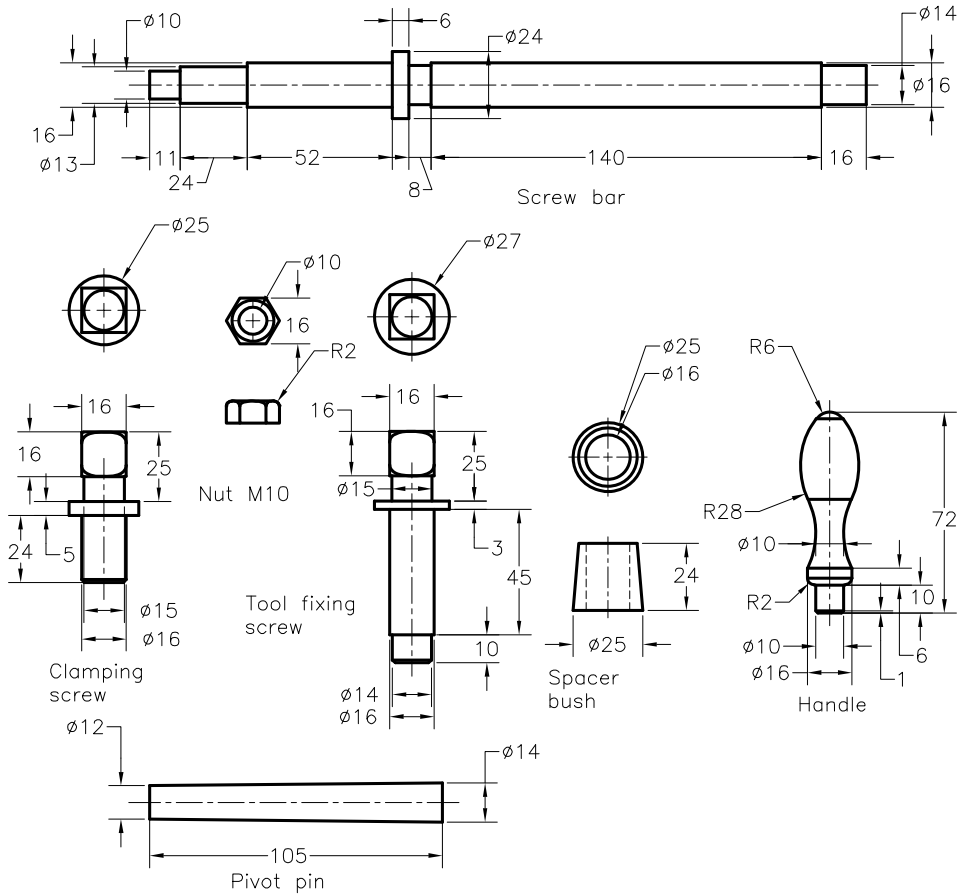


Figure 12-59 Views and dimensions of various components

Answers to Self-Evaluation Test

1. T, 2. T, 3. F, 4. T, 5. T, 6. F, 7. .asm, 8. Explode - Render - Animate, 9. Show, 10. Explode