Chapter 13
Working with Assemblies - II

In this chapter, you will learn the following:

- Creating Assembly by using the Top-down Approach
- Editing Assembly Components
- Editing Mates
- Patterning Assembly Components
- Creating a Pattern Driven Component Pattern
- Creating a Chain Component Pattern
- Mirroring Components of an Assembly
- Creating Assembly Features
- Suppressing or Unsuppressing Components
- Inserting the Parts having Multiple Configurations
- Creating and Dissolving Sub-Assemblies
- Creating an Exploded View
- Collapsing an Exploded View
- Animating an Exploded View
- Editing an Exploded View
- Adding Explode Lines in an Exploded View
- Creating Bill of Material (BOM) of an Assembly

In the previous chapter, you have learned about creating assemblies by using the Bottom-up Assembly Approach. You have also learned about different types of mates and how to move or rotate individual components of an assembly. In this chapter, you will learn about creating assemblies by using the Top-down Assembly Approach, editing assembly components, patterning and mirroring assembly components, creating assemblies features, exploding assemblies, and so on.

Creating Assembly by using the Top-down Approach

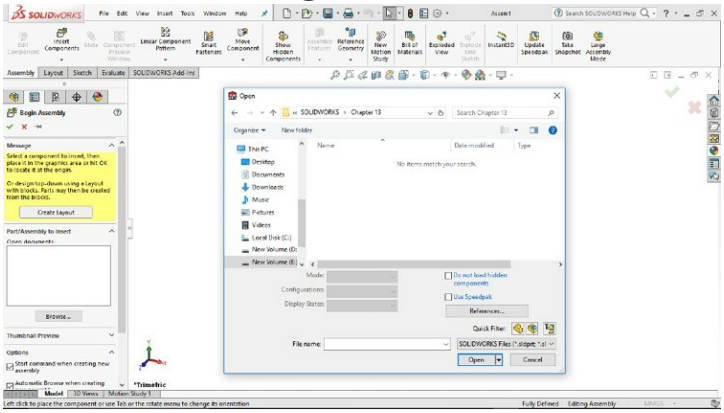
In the Top-down Assembly Approach, all components of an assembly are created within the Assembly environment itself. Creating components in the Assembly environment helps in taking reference from the existing components of the assembly. By using this approach, you can create concept-based design, where new components of an assembly can be created by taking reference from the existing components and maintain the

relationships between them. The procedure for creating assembly by using the Top-down Assembly Approach is as follows:

Procedure for Creating Assembly by using the Top-down Approach

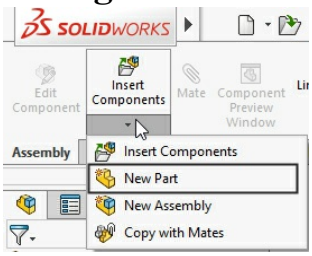
1. Invoke the Assembly environment by using the **New** tool of the **Standard** toolbar, see Figure 13.1. Note that the **Open** dialog box appears along with the **Begin Assembly PropertyManager** in the initial screen of the Assembly environment, by default.

Figure 13.1



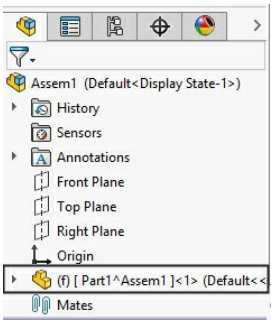
2. Close the **Open** dialog box by clicking on the **Close** button and then close the **Begin Assembly PropertyManager** by clicking on the red cross mark available at its top. The reason behind closing the **Open** dialog box and the PropertyManager is to create components in the Assembly environment itself instead of importing them.
3. Click on the arrow at the bottom of the **Insert Components** tool in the **Assembly CommandManager**. A flyout appears, see Figure 13.2.

Figure 13.2



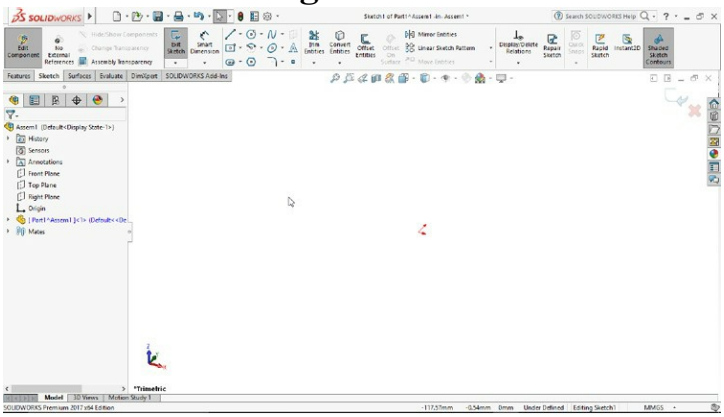
4. Click on the **New Part** tool in the flyout. A new empty part is inserted in the Assembly environment and its default name appears in the FeatureManager Design Tree, see Figure 13.3. Also, a green color tick mark appears attached to the cursor in the graphics area.

Figure 13.3



5. Click on a reference plane in the FeatureManager Design Tree. The Sketching environment is invoked within the Assembly environment such that the selected reference plane becomes the sketching plane for creating the base feature of the part, see Figure 13.4.

Figure 13.4



Note:

*You can click anywhere in the graphics area to define the position of the part instead of selecting a reference plane. On clicking anywhere in the graphics area, the position of the part gets defined such that the origin of the part becomes coincident to the origin of the assembly. But, the Sketching environment for creating the base feature is not invoked. In this case, to invoke the Sketching environment, click on the name of the part in the FeatureManager Design Tree and then click on the **Edit Component** tool in the **Assembly CommandManager**. Next, click on the **Sketch** tool in the **Sketch CommandManager** and then select a plane or a planar face as the sketching plane for creating the base feature.*

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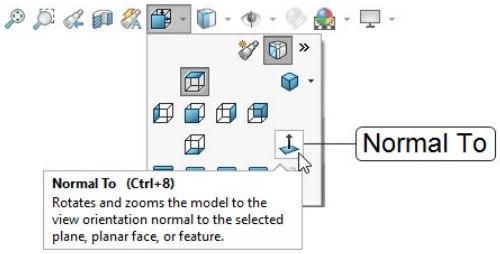
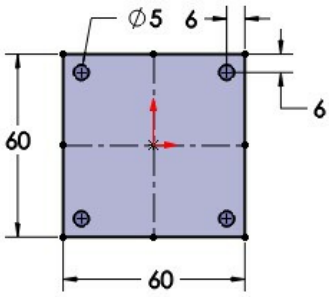
6. Press CTRL + 8 to change the current orientation of the model as normal to the viewing direction. Alternatively, click on the **Normal To** tool in the **View Orientation** flyout, see Figure 13.5.

Figure 13.5



7. Draw the sketch of the base feature by using the sketching tools of the **Sketch CommandManager**, refer to Figure 13.6.

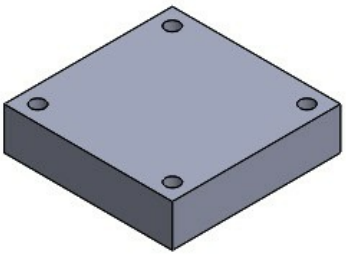
Figure 13.6



After creating the sketch, you need to covert it into a solid feature by using the solid modeling tools.

8. Click on the **Features** tab in the CommandManager. The tools of the **Features CommandManager** are displayed. Now, by using the tools such as **Extruded Boss/Base** or **Revolved Boss/Base**, you can convert the sketch into a solid feature, refer to Figure 13.7. In this figure, the sketch is extruded to the depth of 15 mm by using the **Extruded Boss/Base** tool.

Figure 13.7



You need to create the second feature of the component.

9. Invoke the Sketching environment again by selecting a planar face or a plane as the sketching plane for creating the second feature of the component.
10. Create the sketch of the second feature, refer to Figure 13.8 and then convert it into a feature by using the solid modeling tools of the **Features CommandManager**, refer to Figure 13.9. In Figure 13.9, the sketch is extruded to the depth of 75 mm by using the **Extruded Boss/Base** tool. Similarly, you can create the remaining features of the part one after another.

Figure 13.8

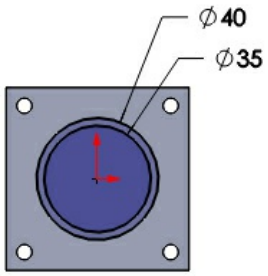
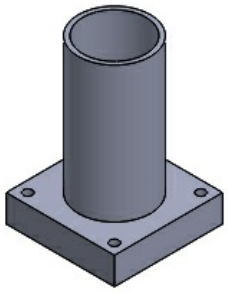


Figure 13.9



11. Once all the features of the component have been created, click on the **Edit Component** tool in the **Features CommandManager**. The first component is created and the Assembly environment is invoked.

Note:

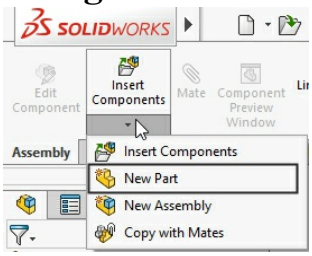
By default, the components created in the Assembly environment are fixed components and their degrees of freedom are restricted. This is because the Inplace mate is applied automatically between the plane of the component and the plane of the assembly. You can convert a fixed component into a floating component by deleting the Inplace mate. To delete the Inplace mate, expand the Mates node in the FeatureManager Design Tree and then select the InPlace mate to be deleted. Next, press the DELETE key.

If you have defined the placement of a component in the Assembly environment by clicking in the graphics area instead of selecting a plane then the component becomes fixed in the Assembly environment without applying the Inplace mate. In such case, to convert a fixed component into a floating component, select the component from the FeatureManager Design Tree and then right-click to display a shortcut menu. Next, click on the ***Float*** *option in the shortcut menu.*

After creating the first component, you can create the second component of the assembly.

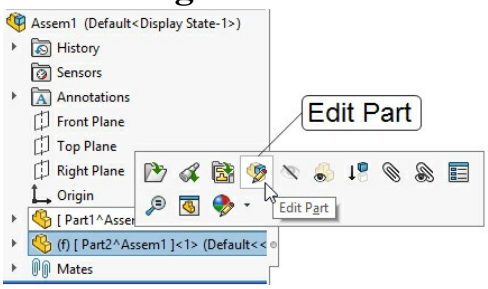
12. Click on the arrow at the bottom of the **Insert Components** tool to invoke a flyout, see Figure 13.10. Next, click on the **New Part** tool in this flyout. A new empty part is inserted in the Assembly environment and its default name appears in the FeatureManager Design Tree.

Figure 13.10



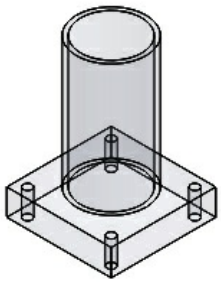
13. Click anywhere in the graphics area. The position of the part gets defined such that the origin of the part becomes coincident to the origin of the assembly.
14. Click on the name of the newly inserted component in the FeatureManager Design Tree. A Pop-up toolbar appears, see Figure 13.11.

Figure 13.11



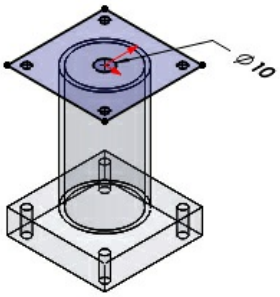
15. Click on the **Edit Part** tool in the Pop-up toolbar, see Figure 13.11. The Part modeling environment is invoked within the Assembly environment for creating the second component. Also, the first component becomes transparent so that you can easily create the second component and take the reference of the first component while creating the second component, refer to Figure 13.12.

Figure 13.12



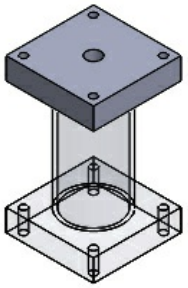
16. Invoke the Sketching environment by selecting a plane or a planar face of the first component as the sketching plane for creating the base feature of the second component.
17. Create the sketch of the base feature of the second component by taking the reference of the first component, refer to Figure 13.13. In this figure, the rectangle and the four circles of the sketch have been created by projecting the edges of the first component onto the sketching plane.

Figure 13.13



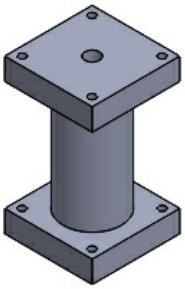
18. Convert the sketch into a feature by using the tools of the **Features CommandManager**, refer to Figure 13.14. In this figure, the sketch is extruded to the depth of 15 mm by using the **Extruded Boss/Base** tool. Similarly, you can create the remaining features of the second component one by one.

Figure 13.14



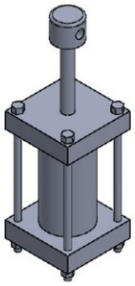
19. Once all the features of the second component have been created, click on the **Edit Component** tool in the **Features CommandManager**. The second component is created and the Assembly environment is invoked. Also, all components of the assembly appear in the shaded display style, refer to Figure 13.15.

Figure 13.15



20. Similarly, create the remaining components of the assembly one after another. Figure 13.16 shows an assembly whose all components are created one after another in the Assembly environment itself by using the Top-down Assembly Approach.

Figure 13.16



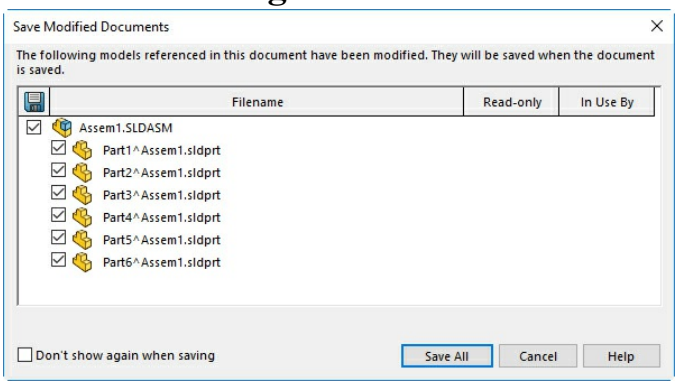
Note:

*As discussed, the components created by using the Top-down Assembly Approach become fixed in the Assembly environment. You can make these components floating components by deleting their respective Inplace mates or by selecting the **Float** option from the shortcut menu, which appears on right-clicking on the component. A floated component can translate and rotate in all directions, which means its all degrees of freedom are free. You can restrict the required degrees of freedom of a floated component and assemble it with the other components of the assembly by applying the required mates. The method of applying mates between the components, which are created by using the Top-down Assembly Approach is the same as discussed in the earlier chapter.*

After creating all the components of an assembly, you can save the assembly file and its components externally or internally in the assembly file.

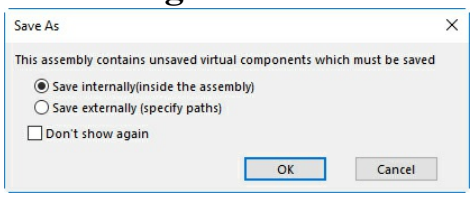
21. Click on the **Save** button in the **Standard** toolbar. The **Save Modified Documents** dialog box appears, see Figure 13.17.

Figure 13.17



22. Click on the **Save All** button in the dialog box. The **Save As** dialog box appears. Next, browse to the location where you want to save the assembly.
23. Enter the name of the assembly in the **File name** field of the dialog box and then click on the **Save** button. The another **Save As** dialog box appears, see Figure 13.18.

Figure 13.18



By default, the **Save internally (inside the assembly)** radio button is selected in the **Save As** dialog box, see Figure 13.18. As a result, all the components of the assembly will be saved internally in the assembly file. On selecting the **Save externally (specify paths)** radio button, all the components of the assembly will be saved externally as individual components in the same folder, where the assembly file is saved.

24. Select the **Save externally (specify paths)** radio button and then click on the **OK** button. All the components of the assembly and the assembly file are saved in the specified folder, individually.

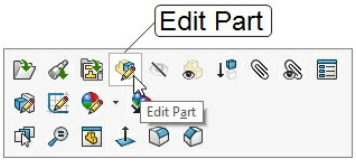
Editing Assembly Components

In the process of creating an assembly, you may need to edit it components several times depending upon the changes in the design, revisions, or validate the design. SOLIDWORKS allows you to edit each component of an assembly within the Assembly environment as well as in the Part modeling environment. Different methods of editing assembly components are as follows:

Editing Assembly Components within the Assembly Environment

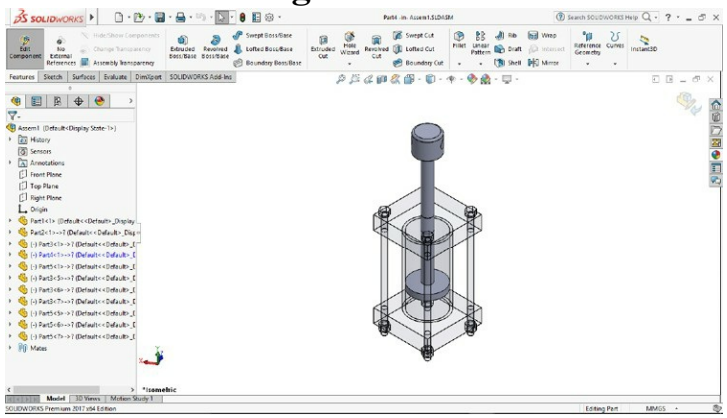
To edit a component of an assembly within the Assembly environment, select the component to be edited either from the graphics area or from the FeatureManager Design Tree. A Pop-up toolbar appears, see Figure 13.19.

Figure 13.19



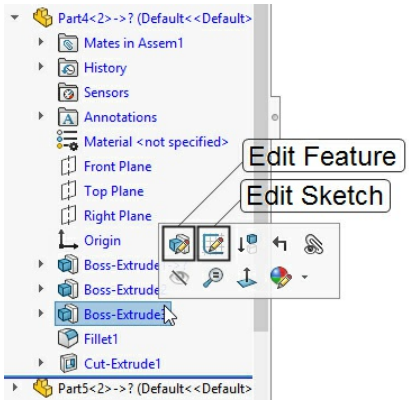
In this Pop-up toolbar, click on the **Edit Part** tool. The editing mode to edit the selected component is invoked within the Assembly environment itself, see Figure 13.20. Also, the other components of the assembly becomes transparent and the name of the component selected for editing appears blue in the FeatureManager Design Tree, see Figure 13.20.

Figure 13.20



Expand the node of the component being edited in the FeatureManager Design Tree to list all its features, see Figure 13.21. Next, select the feature to be edited from the expanded node in the FeatureManager Design Tree. A Pop-up toolbar appears, see Figure 13.21. Next, click on the **Edit Feature** tool in the Pop-up toolbar for editing the feature parameters such as extrusion depth and end condition. If you want to edit the sketch of the feature then click on the **Edit Sketch** tool in the Pop-up toolbar, see Figure 13.21. Depending upon the tool selected (**Edit Feature** or **Edit Sketch**), the respective environment gets invoked for editing the selected feature or sketch of the component. In addition to editing the existing feature of a component, you can also create new features in the component by using the tools of the **Features CommandManager**. Once all the required editing operations have been performed on the component, click on the **Edit Component** tool in the **Features CommandManager** to exit the editing mode and switch back to the Assembly environment. Alternatively to exit the editing mode, click on the Confirmation corner , which is available at the upper right corner of the graphics area.

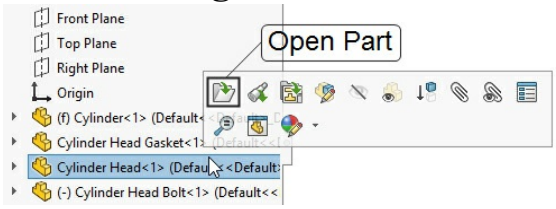
Figure 13.21



Editing Assembly Components in the Part Modeling Environment

In addition to editing the components of an assembly in the Assembly environment, you can also open a component of an assembly in the Part modeling environment and then perform the editing operations. To edit a component of an assembly in the Part modeling environment, click on the component to be edited either in the FeatureManager Design Tree or in the graphics area. A Pop-up toolbar appears, see Figure 13.22. Next, click on the **Open Part** tool in the Pop-up toolbar, see Figure 13.22. The selected component is opened the Part modeling environment. Now, you can edit the component by editing its features and sketch. To edit a feature, click on the feature to be edited in the FeatureManager Design Tree and then click on the **Edit Feature** tool in the Pop-up toolbar appeared. To edit the sketch of a feature, click on the **Edit Sketch** tool in the Pop-up toolbar. Depending upon the tool selected (**Edit Feature** or **Edit Sketch**), the respective environment gets invoked, which allows you to edit the selected feature. You can also create new features in the component by using the tools of the **Features CommandManager**.

Figure 13.22



Once you have edited the component by using the tools of the Part modeling environment, click on the **Save** tool in the **Standard** toolbar to save the modified component. Next, click on **Window > *name of the assembly*** in the SOLIDWORKS menus to switch to the Assembly environment. The **SOLIDWORKS** message window appears. Click on the **Yes** button in this window. The process of updating the assembly starts and once the assembly has been updated, the updated assembly with the modified component appears in the Assembly environment. Note that the modifications made in the component are also reflected in the Assembly environment.

Note:

SOLIDWORKS has bi-directional associative properties, which means the modifications made into a component in any environment are also

Editing Mates

In SOLIDWORKS, you can edit existing mates of an assembly, which are applied between the components. To edit existing mates, expand the Mates node available at the bottom of the FeatureManager Design Tree, see Figure 13.23. The **Mates** node consists of a list of all mates applied between components of an assembly. Next, click on the mate to be edited in the expanded **Mates** node of the FeatureManager Design Tree. A Pop-up toolbar appears, see Figure 13.24. Also, the entities between which the selected mate is applied get highlighted in the graphics area. Next, click on the **Edit Feature** tool in the Pop-up toolbar, see Figure 13.24. The PropertyManager appears depending upon the type of mate selected. By using the options of the PropertyManager, you can select new entities for the mate, change the type of mate, type of mate alignment, and so on. Once the editing has been done, click on the green tick mark in the PropertyManager to accept the change and to exit the PropertyManager.

Figure 13.23

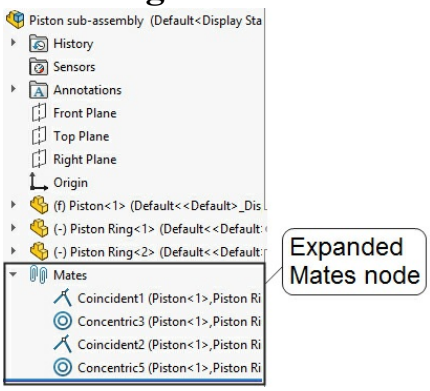
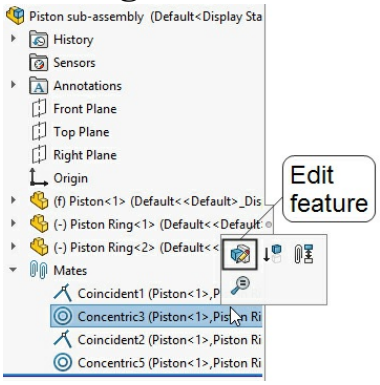


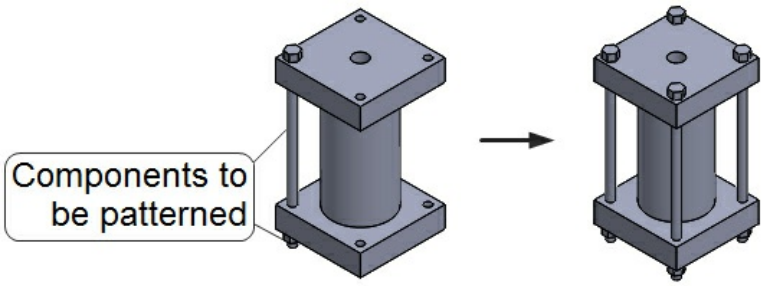
Figure 13.24



Patterning Assembly Components

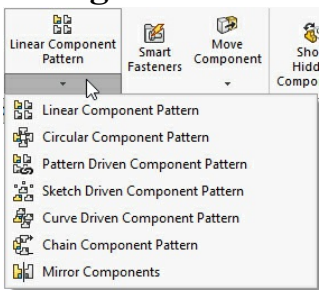
Similar to patterning a feature of a component in the Part modeling environment, you can also pattern a component or components of an assembly in the Assembly environment, see Figure 13.25. In this figure, two components of the assembly are patterned to create their other instances.

Figure 13.25



In the Assembly environment, you can create different types of patterns such as linear component pattern, circular component pattern, sketch driven component pattern, curve driven component pattern, pattern driven component pattern, and chain component pattern by using the respective tools available in the **Pattern** flyout of the **Assembly CommandManager**, see Figure 13.26. The method of creating linear component pattern, circular component pattern, sketch driven component pattern, and curve driven component pattern by using the respective tool is the same as discussed earlier while creating patterns in the Part modeling environment with the only difference that in the Part modeling environment, you pattern features to create their multiple instances. However, in the Assembly environment, you pattern components to create their multiple instances. The method of creating remaining types of patterns are as follows:

Figure 13.26



Creating a Pattern Driven Component Pattern

A pattern driven component pattern is created by patterning a component in an assembly with respect to a pattern feature of another component. In this type of pattern, the component to be patterned in the assembly drives by a pattern feature of another component. Consider a case of an assembly shown in Figure 13.27, which has three components: Component 1, Component 2, and Component 3. The Component 3 is the component to be patterned with respect to the circular pattern feature of the Component 2. Figure 13.28 shows the resultant assembly, in which a pattern driven component pattern is created by patterning the Component 3 with respect to the circular pattern feature of the Component 2. Note that on modifying the number of instances of the pattern feature, the number of instances of the pattern driven component pattern are also modified, automatically. This is because, the instances of the pattern driven component pattern are driven by the instances of the pattern feature.

Figure 13.27

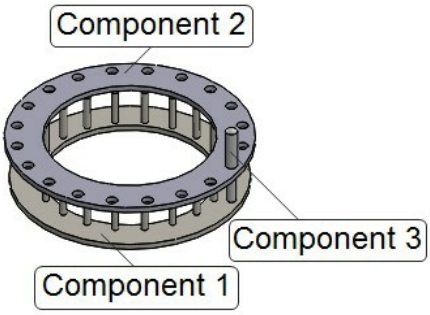
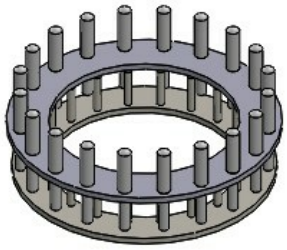
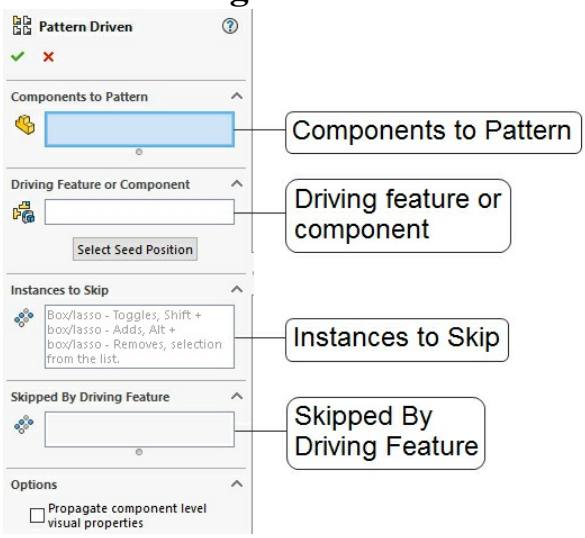


Figure 13.28



To create a pattern driven component pattern, invoke the **Pattern** flyout and then click on the **Pattern Driven Component Pattern** tool, refer to Figure 13.26. The **Pattern Driven PropertyManager** appears, see Figure 13.29. The options of the PropertyManager are as follows:

Figure 13.29



Components to Pattern

The **Components to Pattern** field is used to select components to be patterned. By default, this field is activated. As a result, you can select components to be patterned either from the graphics area or from the FeatureManager Design Tree.

Driving feature or component

The **Driving feature or component** field is used to select an instance of a pattern feature of a component as the driving feature, see Figure 13.30. To select a pattern

instance (driving feature), click on this field in the PropertyManager and then select a pattern instance of a pattern. After selecting the component to be patterned and a pattern instance (driving feature), the preview of the pattern driven component pattern appears in the graphics area, see Figure 13.31.

Figure 13.30

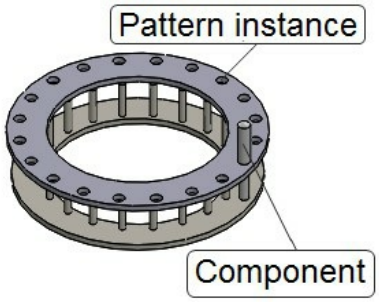
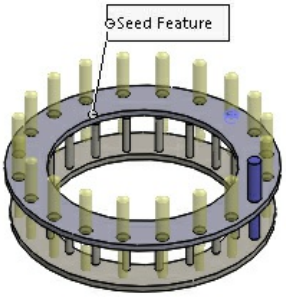


Figure 13.31



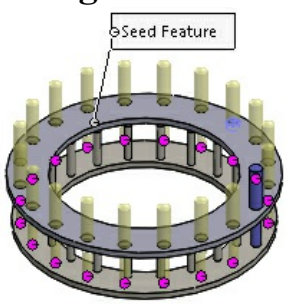
Note:

By default, the position of the seed/parent instance of the pattern is taken as the position of the component being patterned. You can change the default position of the seed pattern instance by using the ***Select Seed Position*** *button of the* ***Driving Feature or Component*** *rollout. On clicking the* ***Select Seed Position*** *button, a blue dot appears in the preview of the each pattern instance in the graphics area. You can click on the blue dot of the pattern instance to select it as the seed feature of the pattern.*

Instances to Skip

The **Instances to Skip** field is used to skip pattern instances of the pattern. To skip pattern instances, click on the **Instances to Skip** field in the rollout. A pink dot appears on each pattern instance in the graphics area, see Figure 13.32. Move the cursor over the instance to be skipped and then click on it.

Figure 13.32



Skipped By Driving Feature

The **Skipped By Driving Feature** field is used to display the list of pattern instances, which are skipped in the pattern feature (driving) of the component.

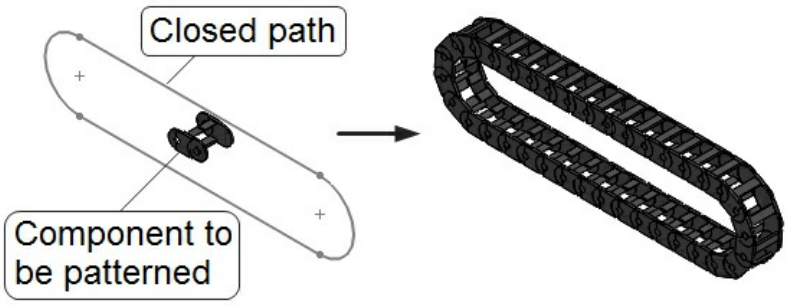
Procedure for Creating the Pattern Driven Component Pattern

1. Invoke the **Pattern** flyout in the **Assembly CommandManager**.
2. Click on the **Pattern Driven Component Pattern** tool in the **Pattern** flyout.
3. Click on the component to be patterned in the graphics area.
4. Click on the **Driving feature or component** field and then click on an instance of a pattern feature of the component as the driving feature.
5. If needed, you can skip the pattern instances by using the **Instances to Skip** rollout.
6. Click on the green tick mark in the PropertyManager. The pattern driven component pattern is created.

Creating a Chain Component Pattern

A chain component pattern is created by patterning a component along an open or a closed path to simulate the chain drive or cable carrier mechanism, dynamically in an assembly, see Figure 13.33. In SOLIDWORKS, you can create a chain component pattern by using the **Chain Component Pattern** tool.

Figure 13.33



In the chain component pattern, the component drives along an open or closed path such that you can simulate its motion dynamically by dragging the pattern instances. To create a chain component pattern, invoke the **Pattern** flyout, see Figure 13.34 and then click on the **Chain Component Pattern** tool. The **Chain Pattern PropertyManager** appears, see Figure 13.35.

Figure 13.34

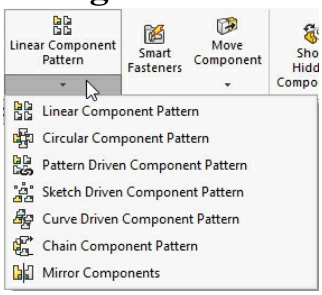
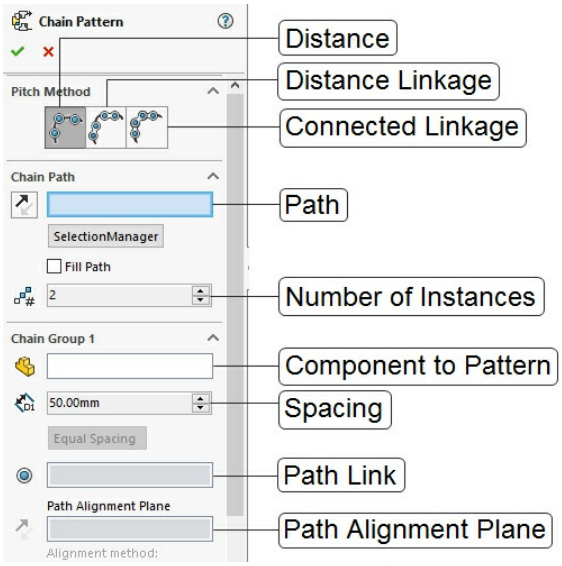


Figure 13.35

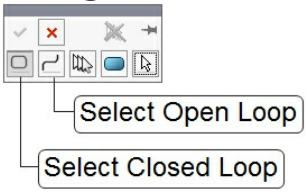


The **Chain Pattern PropertyManager** is used to create three types of chain patterns: Distance, Distance Linkage, and Connected Linkage. The Distance chain pattern is used to pattern a component with a single link among the pattern instances along a chain path. The Distance Linkage chain pattern is used to pattern a component with two non-connected links among the pattern instances along the path. The Connected Linkage chain pattern is used to pattern a component with connected links among the pattern instances along the path. The procedure of creating different types of chain patterns are as follows:

Procedure for Creating the Distance Chain Pattern

1. Click on the **Chain Component Pattern** tool in the **Pattern** flyout. The **Chain Pattern PropertyManager** appears.
2. Make sure the **Distance** button is selected in the **Pitch Method** rollout of the PropertyManager.
3. Click on the **SelectionManager** button in the **Chain Path** rollout. The **Selection** toolbar appears, see Figure 13.36.

Figure 13.36



4. Click on the **Select Closed Loop** or **Select Open Loop** button in the **Selection** toolbar, depending upon the type of path (closed or open) to be selected.
5. Click on the path to be selected in the graphics area, see Figure 13.37. Next, click on the green tick mark in the **Selection** toolbar. The path is selected.
6. Either select the **Fill Path** check box in the **Chain Path** rollout to fill in the path with pattern instances or specify the number of instances to be created in the **Number of Instances** field.
7. Click on the **Component to Pattern** field in the **Chain Group 1** rollout to activate it.

8. Click on the component to be patterned in the graphics area, see Figure 13.37.
9. Click on the **Path Link** field in the **Chain Group 1** rollout to activate it.
10. Click on a cylindrical face, a circular edge, a linear edge, or a reference axis as the link among the pattern instances in the graphics area, see Figure 13.37.
11. Select a plane or a planar face as the alignment plane for aligning the pattern instances along the path, see Figure 13.37. The preview of the chain pattern appears, see Figure 13.38.
12. Specify the spacing between the pattern instances in the Spacing field.

Figure 13.37

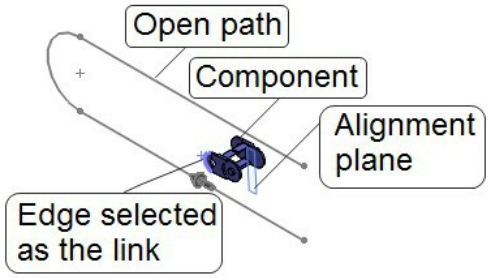
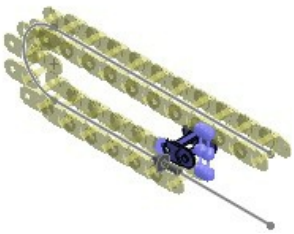
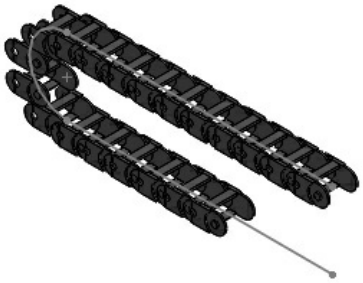


Figure 13.38



13. Click on the green tick mark in the PropertyManager. The chain component pattern is created, see Figure 13.39.

Figure 13.39



Procedure for Creating the Distance Linkage Chain Pattern

1. Click on the **Chain Component Pattern** tool in the **Pattern** flyout. The **Chain Pattern PropertyManager** appears.
2. Click on the **Distance Linkage** button in the **Pitch Method** rollout of the PropertyManager.
3. Click on the **SelectionManager** button in the **Chain Path** rollout. The **Selection**

toolbar appears.

4. Click on the **Select Closed Loop** or **Select Open Loop** button in the **Selection** toolbar, depending upon the type of path (closed or open) to be selected.
5. Click on the path to be selected in the graphics area, see Figure 13.40. Next, click on the green tick mark in the **Selection** toolbar. The path is selected.
6. Either specify the number of pattern instances to be created along the path in the **Number of Instances** field or select the **Fill Path** check box to fill in the path with pattern instances.
7. Click on the **Component to Pattern** field in the **Chain Group 1** rollout to activate it.
8. Click on the component to be patterned in the graphics area.
9. Click on the **Path Link** 1 field in the **Chain Group 1** rollout to activate it.
10. Click on a cylindrical face, a circular edge, a linear edge, or a reference axis as the link 1 among the pattern instances in the graphics area, see Figure 13.40.
11. Click on a cylindrical face, a circular edge, a linear edge, or a reference axis as the link 2 among the pattern instances, see Figure 13.40.
12. Click on a plane or a planar face as the alignment plane, see Figure 13.40. The preview of pattern appears, see Figure 13.41.

Figure 13.40

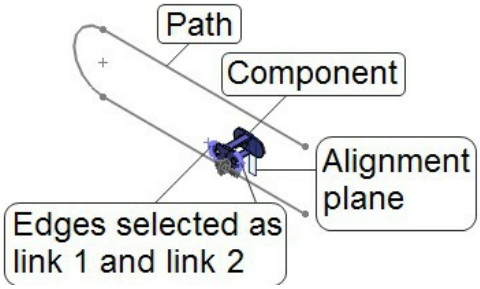
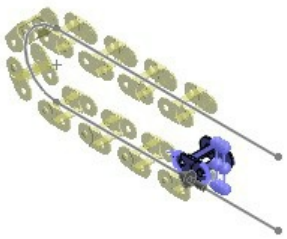


Figure 13.41



13. Specify the spacing between the pattern instances in the **Spacing** field.
14. Click on the green tick mark in the PropertyManager. The distance linkage chain pattern is created, see Figure 13.42.

Figure 13.42

Note:

*In the **Options** rollout of the **Chain Pattern PropertyManager**, the **Dynamic** radio button is selected by default, see Figure 13.43. As a result, you can drag any pattern instance to move the chain. On selecting the **Static** radio button, you can move the chain only by dragging the parent or seed component of the pattern. The **Static** radio button helps in improving the overall performance of the system for large assemblies.*

Figure 13.43

Procedure for Creating the Connected Linkage Chain Pattern

1. Click on the **Chain Component Pattern** tool in the **Pattern** flyout to invoke the **Chain Pattern PropertyManager**.
2. Click on the **Connected Linkage** button in the **Pitch Method** rollout of the PropertyManager.
3. Select an open or a closed path from the graphics area by using the **SelectionManager** button of the **Chain Path** rollout, see Figure 13.44.
4. Either specify the number of pattern instances to be created along the path in the **Number of Instances** field or select the **Fill Path** check box to fill in the path with pattern instances.
5. Click on the **Component to Pattern** field in the **Chain Group 1** rollout to activate it.
6. Click on the component to be patterned in the graphics area.
7. Click on the **Path Link 1** field in the **Chain Group 1** rollout to activate it.
8. Click on a cylindrical face, a circular edge, a linear edge, or a reference axis as the link 1 between the pattern instances in the graphics area, see Figure 13.44.
9. Click on a cylindrical face, a circular edge, a linear edge, or a reference axis as the link 2 between the pattern instances, see Figure 13.44.
10. Click on a plane or a planar face as the alignment plane, see Figure 13.44. The preview of the pattern appears, see Figure 13.45. If the pattern preview does not appear in the graphics area, you need to reverse the direction of the pattern by clicking on the arrow appeared along the path in the graphics area.

Figure 13.44

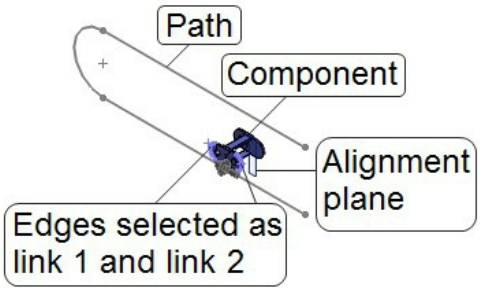
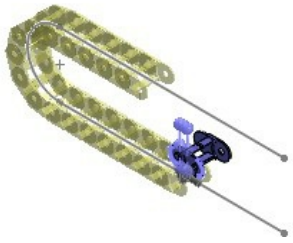


Figure 13.45



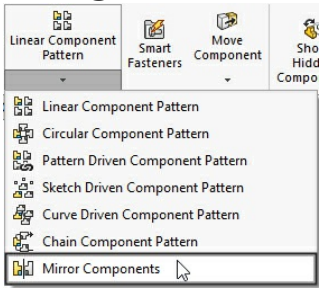
11. Click on the green tick mark of the PropertyManager. The connected linkage chain pattern is created, see Figure 13.46.

Figure 13.46

Mirroring Components of an Assembly

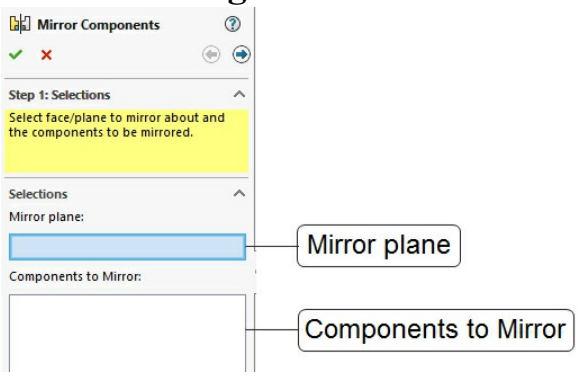
Similar to mirroring features in the Part modeling environment, you can also mirror components in the Assembly environment by using the **Mirror Components** tool of the **Pattern** flyout, see Figure 13.47.

Figure 13.47



To mirror components of an assembly, invoke the **Pattern** flyout and then click on the **Mirror Components** tool, see Figure 13.47. The **Mirror Components PropertyManager** appears, see Figure 13.48. The options of the PropertyManager are as follows:

Figure 13.48



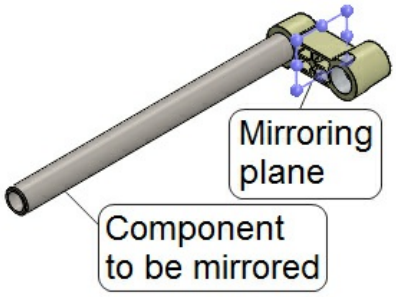
Mirror plane

The **Mirror plane** field is used to select a mirroring plane for mirroring the selected components of an assembly. By default, this field is activated. As a result, you can select a plane or a planar face as the mirroring plane, see Figure 13.49.

Components to Mirror

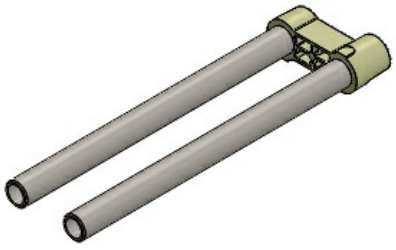
The **Components to Mirror** field is used to select components to be mirrored about the mirroring plane. This field gets activated as soon as you select the mirroring plane. Select a component or components to be mirrored either from the graphics area or from the FeatureManager Design Tree, see Figure 13.49.

Figure 13.49



After selecting the mirroring plane and a component to be mirrored, click on the green tick mark in the PropertyManager. The selected component is mirrored about the mirroring plane, see Figure 13.50.

Figure 13.50



Note:

*You can also change the orientation of the mirrored component. To change the orientation of the mirrored component, click on the **Next** button in the **Mirror Components PropertyManager**. Next, click on the double arrow in the **Reorient components** area of the PropertyManager to view the second or next possible orientation of the mirrored component. You can cycle through four possible orientations of the mirrored component by clicking on this double arrow in the **Reorient components** area.*

*If you have selected a flexible sub-assembly as the component to be mirrored then you can synchronize the movement of the components of the mirrored sub-assembly with respect to the movement of the parent sub-assembly (flexible) by selecting the **Synchronize movement of flexible subassembly components** check box of the PropertyManager. After selecting this check box, if you move the components of the flexible sub-assembly (parent), the respective components of the mirrored sub-assembly are also moved, respectively, and vice versa. Note that this check box is enabled only if the selected sub-assembly to be mirrored is a flexible sub-assembly and the **Create opposite hand version** button is activated in the PropertyManager. The **Create opposite hand version** button is used to create the mirror image (opposite version) of the selected component.*

Procedure for Mirroring Components of an Assembly

1. Invoke the **Pattern** flyout in the **Assembly CommandManager**.
2. Click on the **Mirror Components** tool in the **Pattern** flyout.
3. Select a plane or a planar face as the mirroring plane.
4. Select a component or components to be mirrored about the mirroring plane.
5. Click on the green tick mark in the PropertyManager. The selected components are mirrored.

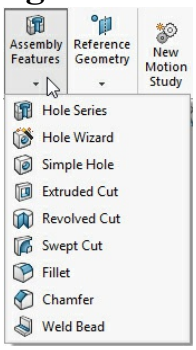
Creating Assembly Features

In a manufacturing unit or a shop floor, after assembling all components of an assembly, several cut operations may take place in components in order to give final touch-up and align components perfectly with respect to each other. For this, SOLIDWORKS has tools to create cut features in the Assembly environment. Cut features created in the Assembly environment are known as assembly features and do not affect the original geometry of the components. For example, if you create an assembly feature (cut feature) on a component of an assembly in the Assembly environment; the assembly feature created will exist only in the assembly and if you open the same component in the Part modeling environment, you will not find the existence of the assembly feature. It means the assembly features exist in the assembly only and will not affect the original geometry of the component.

In SOLIDWORKS, you can create assembly features such as holes, extruded cut, revolved cut, swept cut, and fillets. The tools to create assembly features are provided in the **Assembly Features** flyout, see Figure 13.51. To invoke the **Assembly Features** flyout, click on the arrow at the bottom of the **Assembly Features** tool in the **Assembly CommandManager**. The procedure of creating assembly features is the same as creating features in the Part modeling environment. For example, to create an extruded cut feature, click on the **Extruded Cut** tool in the **Assembly Features** flyout. The **Extrude PropertyManager** appears. Select a plane or a planar face as the sketching plane. The Sketching environment is invoked. Create the sketch of the extruded cut feature and then exit the Sketching environment. As soon as you exit from the Sketching environment, the preview of the cut feature appears in the graphics area. Specify the

required parameters for the extrusion in the PropertyManager and then click on the green tick mark. The extruded cut feature is created in the Assembly environment.

Figure 13.51



Note:

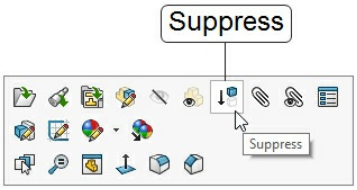
*You can also invoke the tools for creating the assembly features by clicking on **Insert > Assembly Feature** in the SOLIDWORKS menus. In addition to creating the assembly features (cut features), you can mirror the assembly features about a mirroring plane by using the **Mirror** tool of the **Assembly Features** flyout. You can also create a linear pattern, circular pattern, table driven pattern, sketch driven pattern, and so on of the assembly features by using the respective tools available in the **Assembly Features** flyout. The tools to create mirror and pattern features become available in the **Assembly Features** flyout only after creating an assembly feature in the Assembly environment.*

Suppressing or Unsuppressing Components

In SOLIDWORKS, you can suppress or unsuppress components of an assembly. A suppressed component is removed from the assembly and does not appear in the graphics area. Also, the name of the suppressed component appears in gray color in the FeatureManager Design Tree. Note that a suppressed component is not deleted from the assembly, it is only removed or disappeared such that it is not loaded into the RAM (random access memory) while rebuilding the assembly. This helps you speed up the overall performance of the system when you are working with large assemblies.

To suppress a component of an assembly, select the component to be suppressed either from the graphics area or from the FeatureManager Design Tree. A Pop-up toolbar appears, see Figure 13.52. In this Pop-up toolbar, click on the **Suppress** tool. The selected component is suppressed.

Figure 13.52

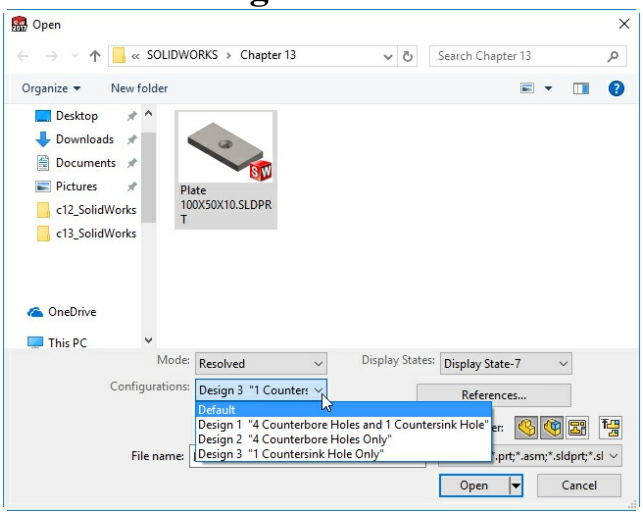


To unsuppress a suppressed component, select the suppressed component from the FeatureManager Design Tree. A Pop-up toolbar appears. Next, click on the **Unsuppress** tool in the Pop-up toolbar. The component is now unsuppressed and appears in the assembly.

Inserting the Parts having Multiple Configurations

In SOLIDWORKS, you can choose a configuration of a component to be inserted in the Assembly environment. To choose a configuration of a component to be inserted in the Assembly environment, click on the **Insert Components** tool in the **Assembly CommandManager**. The **Open** dialog box appears along with the **Insert Component PropertyManager**, automatically. Note that if the **Open** dialog box does not appear then click on the **Browse** button in the Part/Assembly to Insert rollout of the PropertyManager to open the **Open** dialog box. In the **Open** dialog box, browse to the location where the component to be inserted has been saved. Next, select the component that has multiple configurations and then invoke the **Configurations** drop-down list in the **Open** dialog box, see Figure 13.53. Now, select the required configuration of the component from this drop-down list that is to be inserted in the Assembly environment. Next, click on the **Open** button in the dialog box. The selected configuration of the component is attached to the cursor. Now, click in the graphics area to specify the position of the component.

Figure 13.53



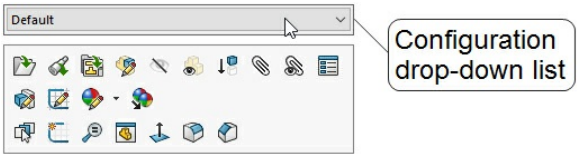
Note:

In SOLIDWORKS, you can create multiple configurations of a component in the Part modeling environment. For example, if a bolt of same geometry has to be used in an assembly several times with the difference in its diameter, then you can create a single bolt with multiple configurations having different diameters. The different methods of creating multiple configurations are discussed in Chapter 11.

You can also choose a configuration of a component to be inserted in the Assembly environment by using the **Configuration** drop-down list of the **Insert Component PropertyManager** or the **Begin Assembly PropertyManager**. Additionally, you can change the configuration of a component even after inserting it in the Assembly environment. To change the configuration of an already inserted component, click on the component whose configuration has to be changed. A Pop-up toolbar appears with the **Configuration** drop-down list, see Figure 13.54. Next, invoke the **Configuration** drop-

down list by clicking on its down arrow and then select the required configuration of the component. Next, click on the green tick mark appeared in front of the drop-down list to confirm the selection of the configuration. The configuration of the component changes to the selected configuration in the Assembly environment.

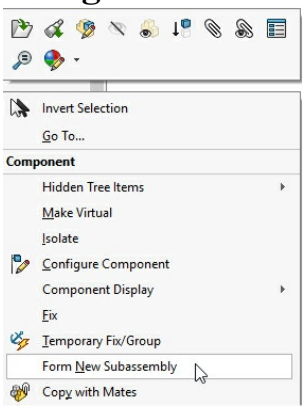
Figure 13.54



Creating and Dissolving Sub-Assemblies

SOLIDWORKS allows you to create sub-assemblies from the components of an assembly within the Assembly environment. To create a sub-assembly, select components to be included in the sub-assembly from the FeatureManager Design Tree by pressing the CTRL key and then right-click. A shortcut menu appears. In this shortcut menu, click on the **Form New Subassembly** option, see Figure 13.55. As soon as you select this option, a sub-assembly is created with a default name and the selected components become the part of the sub-assembly. In case, the **Assembly Structure Editing** window appears after selecting the **Form New Subassembly** option then click on the **Move** button. You can also include patterned and mirrored components in the sub-assembly. To rename the sub-assembly, select the sub-assembly in the FeatureManager Design Tree and then right-click. Next, click on the **Rename Assembly** option in the shortcut menu appeared. The name of the sub-assembly appears in an edit field. Now, you can enter a new name for the sub-assembly.

Figure 13.55



You can also dissolve the created sub-assembly in the Assembly environment. To dissolve the sub-assembly, select the sub-assembly to be dissolved from the FeatureManager Design Tree and then right-click to display a shortcut menu. Next, click on the **Dissolve Subassembly** option in the shortcut menu. The selected sub-assembly is dissolved and its components become the individual components of the main assembly.

Creating an Exploded View

Creating an exploded view of an assembly is important from the presentation point of view. Also, an exploded view of an assembly helps you easily identify the position of each component of an assembly. Moreover, an exploded view helps in making technical documentation as well as it helps technical and non-technical clients easily understand about various components of the assembly. Figure 13.56 shows an assembly and Figure 13.57 shows the exploded view of the assembly.

Figure 13.56

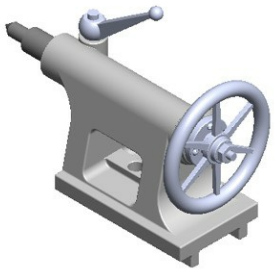
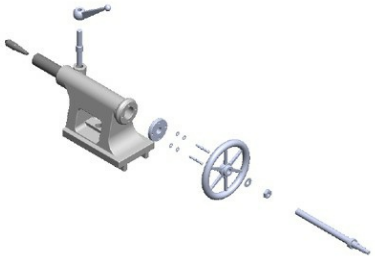


Figure 13.57



To create an exploded view of an assembly, click on the **Exploded View** tool in the **Assembly CommandManager**, see Figure 13.58. The **Explode PropertyManager** appears, see Figure 13.59. The options of the PropertyManager are as follows:

Figure 13.58

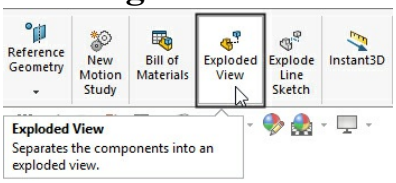
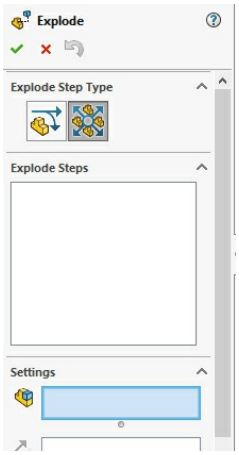


Figure 13.59



Explode Step Type

The **Explode Step Type** rollout of the PropertyManager is used to choose the type of exploded step to be created. By activating the **Regular step** button in the **Explode Step Type** rollout, you can explode the components of an assembly by translating and rotating them along and about an axis, see Figure 13.60. By activating the **Radial step** button, you can explode the components of an assembly by aligning them radially or cylindrically about an axis, see Figure 13.61.

Figure 13.60

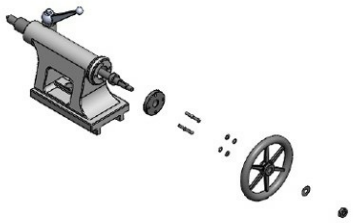
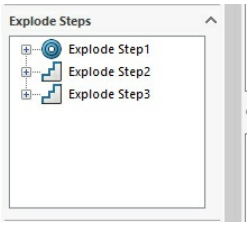


Figure 13.61

Explode Steps

The **Explode Steps** rollout displays the list of exploded steps created for an exploded assembly, see Figure 13.62. Note that to create an exploded assembly, you may need to create multiple exploded steps (regular and radial steps). In each exploded step, one or more than one component can be exploded.

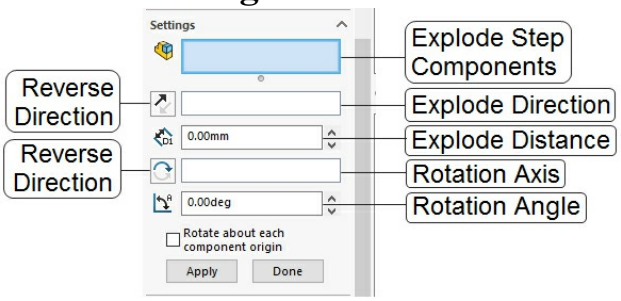
Figure 13.62



Settings

The options in the **Settings** rollout are used to create exploded steps. Note that the availability of the options in this rollout depends on the button (**Regular step** or **Radial step**) selected in the **Explode Step Type** rollout. Figure 13.63 shows the **Settings** rollout when the **Regular step** button is selected. The options of the **Settings** rollout are as follows:

Figure 13.63



Explode Step Components

The **Explode Step Components** field is used to select components of the assembly to be exploded in the exploded step. You can select one or more than one component to be exploded in an exploded step from the graphics area. As soon as you select the components to be exploded, three rotation handles and three translation handles appear in the graphics area, see Figure 13.64. Note that these handles (three rotation and three translations) appear in the case of creating regular exploded step by using the **Regular step** button. In the case of creating the radial exploded step by using the **Radial step** button, one rotation handle and one translation handle appears in the graphics area as soon as you select the components to be exploded, see Figure 13.65.

Figure 13.64

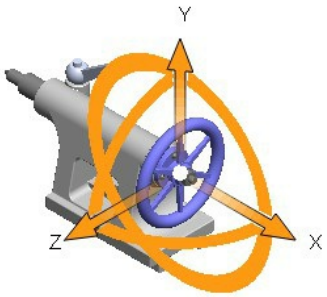


Figure 13.65

You can drag the required rotation handle or translation handles for creating the exploded step. A translation handle is used to translate the selected components along its respective direction, whereas a rotation handle is used to rotate the components about its respective axis.

Tip:

For translation movement, move the cursor over the required translation handle and then drag it by pressing and holding the left mouse button. The selected component starts translating along the direction of translation handle. Once the desired location has been achieved, release the left mouse button. An explode step is created and appears in the ***Explode Steps*** *rollout.*

For rotational movement, move the cursor over the required rotation handle and then drag it by pressing and holding the left mouse button. The selected component starts rotating about the axis of the rotational handle. Next, release the left mouse button. An exploded step is created.

Explode Direction

The **Explode Direction** field displays, translation direction along which selected components can translate. Note that the display of exploded direction depends upon the translation handle selected.

Explode Distance

The **Explode Distance** field is used to specify translation distance value for translating the selected components along with the translation direction. After specifying translation distance value in this field, click on the **Apply** button and then on the **Done** button in the **Settings** rollout of the PropertyManager. Note that if you translate the components by dragging the translation handle, then the translation distance value updates automatically in this field.

Rotation Axis

The **Rotation Axis** field displays rotation axis about which the selected components are to be rotated. The display of the rotational axis depends upon the selection of rotation handle.

Rotation Angle

The **Rotation Angle** field is used to specify a rotation angle value for rotating the selected components. After entering a rotation angle value in this field, click on the **Apply** button and then on the **Done** button. Note that if you rotate the components by dragging the rotation handle, then the rotational angle value updates automatically in this

field.

Rotate about each component origin

On selecting the **Rotate about each component origin** check box, the selected components rotate about the origin of the component. This check box is available only when the **Regular Step** button is activated in the **Explode Step Type** rollout of the PropertyManager.

Diverge from axis

On selecting the **Diverge from axis** check box, the selected components explode away from an axis. This check box is available only when the **Radial Step** button is activated in the **Explode Step Type** rollout of the PropertyManager.

Apply

The **Apply** button is used to apply the exploded parameters and display the preview of exploded step.

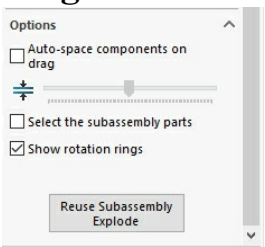
Done

The **Done** button is used to accept the preview of exploded step and create an exploded step.

Options

The options in the **Options** rollout control the exploded step, see Figure 13.66. Note that the availability of the options in this rollout depends upon the button (**Regular step** or **Radial step**) selected in the **Explode Step Type** rollout of the PropertyManager. Figure 13.66 shows the **Options** rollout when the **Regular step** button is selected. The options of the **Options** rollout are as follows:

Figure 13.66



Auto-space components on drag

The **Auto-space components on drag** check box is used to translate or rotate a set of selected components with equal spacing, automatically on exploding the components along or about the selected direction. Note that the equal spacing value for exploding a set of selected components with equal spacing can be increased or decreased by using the **Adjust the spacing between chain components** slider.

Adjust the spacing between chain components

The **Adjust the spacing between chain components** slider is used to adjust the spacing

between a set of selected components for exploding them, equally.

Show rotation rings

By default, the **Show rotation rings** check box is selected. As a result, the rotation handles/rings appear in the graphics area by selecting the components to be exploded. If you uncheck this check box, the display of rotation handles/rings is disabled.

Select the subassembly's parts

By selecting the **Select the subassembly's parts** check box, you can select individual components of a sub-assembly for explosion.

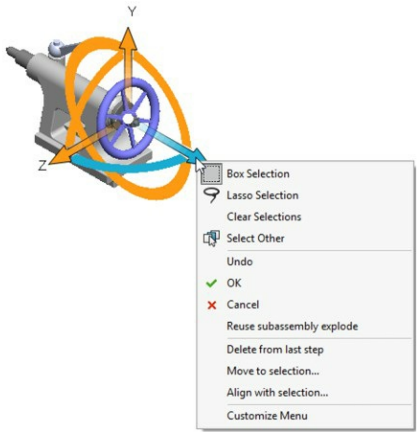
Reuse Subassembly Explode

The **Reuse Subassembly Explode** button is used to explode components by using the previously defined exploded step for a sub-assembly.

Note:

*By default, the selected components can only explode along the X, Y, and Z axes by using the handles appeared in the graphics area. To translate the component other than X, Y, and Z axes, move the cursor over a translation handle appeared in the graphics area and then right-click. A shortcut menu appears, see Figure 13.67. In this shortcut menu, click on the **Align with selection** or **Align to selection** option. Next, select a linear edge of a component. The selected handle is aligned along the direction of the selected linear edge. Now, you can drag the aligned handle to translate the components along the direction of the aligned handle.*

Figure 13.67



Procedure for Creating the Regular Exploded View

1. Click on the **Exploded View** tool. The **Explode PropertyManager** appears.
2. Make sure that the **Regular step** button is selected in the **Explode Step Type** rollout.
3. Select a set of components to be exploded as the first regular exploded step.
4. Drag a translation handle or a rotation handle appeared in the graphics area by pressing and holding the left mouse button. Alternatively, enter the translation distance or rotational angle value in the respective fields of the Settings rollout and then click on the **Apply** button and the **Done** button to create the first exploded step.
5. Similarly, create the other exploded steps for the assembly.
6. Click on the green tick mark button in the PropertyManager. The exploded view of

the assembly is created.

Procedure for Creating the Radial Exploded View

1. Click on the **Exploded View** tool. The **Explode PropertyManager** appears.
2. Make sure that the **Radial step** button is selected in the **Explode Step Type** rollout.
3. Select a set of components to be exploded as the first exploded step.
4. Drag the translation handle or the rotation handle to a required distance by pressing and holding the left mouse button. Alternatively, enter the translation distance or rotation angle value in the respective fields of the Settings rollout and then click on the **Apply** button and then the **Done** button to create the first exploded step.
5. Similarly, create the other exploded steps for the assembly.
6. Click on the green tick mark in the PropertyManager. The exploded view of the assembly is created.

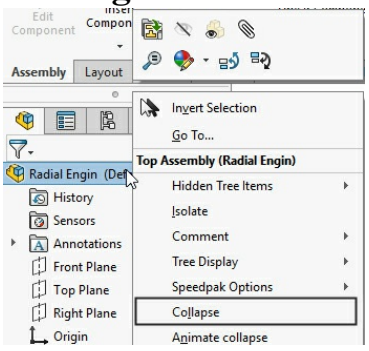
Note:

You can create exploded view of an assembly with the combination of the regular and radial exploded steps.

Collapsing an Exploded View

After creating an exploded view of an assembly, you can restore the components of the assembly back to their original positions by collapsing the exploded view. To collapse the exploded view of an assembly, select the name of the assembly from the FeatureManager Design Tree and then right-click. A shortcut menu appears, see Figure 13.68. In this shortcut menu, click on the **Collapse** option.

Figure 13.68



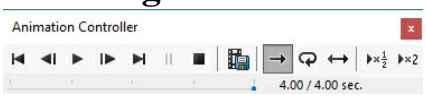
After the collapsed view of an assembly has been displayed, you can display the exploded view again. To display the exploded view of an assembly again, select the name of the assembly from the FeatureManager Design Tree and then right-click to display a shortcut menu. In this shortcut menu, click on the **Explode** option. The exploded view of the assembly is displayed.

Animating an Exploded View

After creating an exploded view of an assembly, you can animate the components of the

assembly to display its collapsed and exploded views. To animate the collapsed or exploded view of an assembly, select the name of the assembly from the FeatureManager Design Tree and then right-click to display a shortcut menu. Next, click on the **Animate collapse** or **Animate explode** option in the shortcut menu. The components start animating and the **Animation Controller** toolbar appears, see Figure 13.69. By using the **Animation Controller** toolbar, you can control the animation of the components. You can also record the animation and save it as a *.avi*, *.bmp*, and *.tga* file.

Figure 13.69



Note:

*The availability of the **Animate collapse** or **Animate explode** option in the shortcut menu depends upon the current state of the assembly. If the assembly appears in its exploded view in the graphics area, then the **Animate collapse** option is available in the shortcut menu. If the assembly appears in its collapsed view, then the **Animate explode** option is available in the shortcut menu.*

Editing an Exploded View

You can edit the existing exploded view of an assembly to make necessary changes in its existing exploded steps and to create new exploded steps. To edit the exploded view of an assembly, invoke the **ConfigurationManager** by clicking on the **ConfigurationManager** tab, see Figure 13.70. The **ConfigurationManager** displays the list of all configurations of the assembly. Figure 13.70 shows the default configuration of the assembly (Default [*name of the assembly*]). Expand the configuration of the assembly by clicking on the arrow that appears in its front, see Figure 13.71.

Figure 13.70

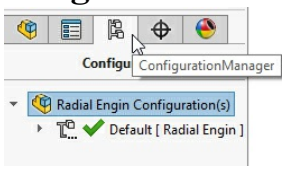


Figure 13.71

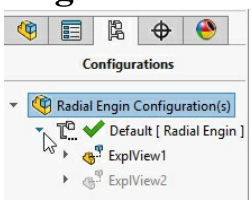
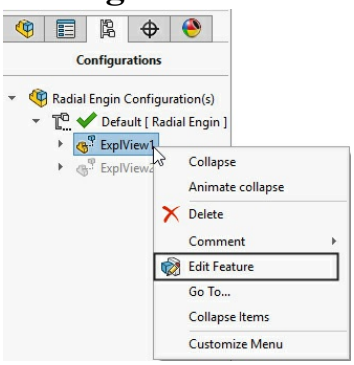


Figure 13.71 shows the two exploded views: ExplView1 and ExplView2 created for the assembly and out of which the ExplView1 is activated, by default. You can activate the required exploded view by double-clicking on its name. To edit an exploded view, select the exploded view to be edited and then right-click to display a shortcut menu, see

Figure 13.72. Next, click on the **Edit Feature** option in the shortcut menu. The **Explode PropertyManager** appears. Now, by using the options of this PropertyManager, you can edit the exploded view, as required.

Figure 13.72



Adding Explode Lines in an Exploded View

After creating an exploded view of an assembly, you can add exploded lines in it. Figure 13.73 shows an assembly and Figure 13.74 shows an exploded view of the assembly with exploding lines.

Figure 13.73

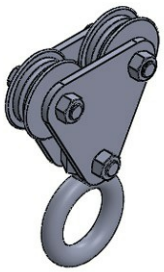
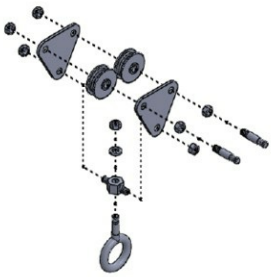


Figure 13.74



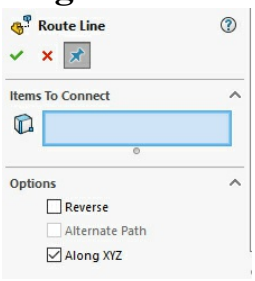
Exploded lines are used to show relationships between components in an exploded view. You can create exploded lines in an exploded view by using the **Explode Line Sketch** tool of the **Assembly CommandManager**. The procedure to create exploded lines in an exploded view of an assembly is as follows:

Procedure for Creating Exploded Lines

1. Click on the **Explode Line Sketch** tool in the **Assembly CommandManager**. The

Route Line PropertyManager appears, see Figure 13.75.

Figure 13.75



2. Select faces, circular edges, straight edges, or planar faces of the components having same assembly line one after another to connect them with a single route line.
3. Select the **Reverse** check box to reverse the direction of route line, if needed. Also, you can select the **Alternate Path** check box to see the alternate route between the selected components.
4. Click on the green tick mark in the PropertyManager. A route line is created among the selected components, which represents the assembly line of the components.
5. Similarly, create route lines for remaining sets of components having the same assembly line.
6. Once you have created all the exploded lines, click on the green tick mark in the PropertyManager.

Creating Bill of Material (BOM) of an Assembly

A Bill of Material (BOM) is one of the important features of any drawing. It contains information related to the number of components, material, quantity, and so on. In addition to creating Bill of Material (BOM) in a drawing, SOLIDWORKS also allows you to create BOM in the Assembly environment. You will learn about creating Bill of Material (BOM) in a drawing in chapter 14. To create BOM in the Assembly environment, click on the **Bill of Materials** tool in the **Assembly CommandManager**. The **Bill of Materials PropertyManager** appears, see Figure 13.76. Accept the default parameters specified in the PropertyManager for creating the BOM by clicking on the green tick mark in the PropertyManager. The Bill of Material (BOM) is attached to the cursor and as you move the cursor in the graphics area, the BOM moves accordingly. Now, you need to specify the location for the BOM in the graphics area. Click in the graphics area. The BOM is placed on the specified location, see Figure 13.77.

Figure 13.76