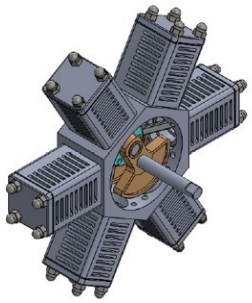
Chapter 12
Working with Assemblies - I

In this chapter, you will learn the following:

- Bottom-up Assembly Approach
- Top-down Assembly Approach
- Creating Assembly by using Bottom-up Approach
- Working with Degrees of Freedom
- Applying Relations or Mates
- Working with Standard Mates
- Working with Advanced Mates
- Working with Mechanical Mates
- Moving and Rotating Individual Components
- Working with SmartMates

In the earlier chapters, you have learned about the basic and advance techniques of creating real world mechanical components. In this chapter, you will learn about different techniques of creating mechanical assemblies. An assembly is made up of two or more than two components joined together by applying relations/mates. You will learn about applying relations/mates later in this chapter. Figure 12.1 shows an assembly, in which multiple components are assembled with respect to each other by applying required mates.

Figure 12.1



In SOLIDWORKS, you can create assemblies in the Assembly environment by using two approaches: Bottom-up Assembly Approach and Top-down Assembly Approach. Moreover, you can use the combination of both the approaches for creating an assembly. Both the approaches are as follows:

Bottom-up Assembly Approach

The Bottom-up Assembly Approach is most widely used approach for assembling components. In this approach, first all the components of an assembly have created one by one in the Part modeling environment and then saved in a common location. Later, all the components are inserted one by one in the Assembly environment and then assembled with respect to each other by applying the required mates.

Tip:

SOLIDWORKS has the bidirectional association between all its environments. As a result, if any, change or modification is made into a component in the Part modeling environment, the same change automatically replicates or reflects in the Assembly and Drawing environments as well, and vice-versa.

Top-down Assembly Approach

The Top-down Assembly Approach is mainly used for creating concept-based design, in which new components of an assembly are created by taking reference from the existing components of the assembly in the Assembly environment and maintain the relationships between them.

In the Top-down Assembly Approach, all the components of an assembly are created in the Assembly environment. Creating all components of an assembly in the Assembly environment helps you create components by taking reference from existing components of the assembly.

Creating Assembly by using Bottom-up Approach

After creating all components of an assembly in the Part modeling environment and saving them in a common location, you need to invoke the Assembly environment of SOLIDWORKS for assembling them. To invoke the Assembly environment, click on the **New** tool in the **Standard** toolbar. The **New SOLIDWORKS Document** dialog box appears, see Figure 12.2. In this dialog box, click on the **Assembly** button and then click on the **OK** button. The Assembly environment is invoked with the **Begin Assembly PropertyManager** on its left. Also, the **Open** dialog box appears on the screen, see Figure 12.3. Note that along with the **Begin Assembly PropertyManager**, the **Open** dialog box appears every time on invoking the Assembly environment. This is because, in the **Options** rollout of the **Begin Assembly PropertyManager**, the **Automatic Browse when creating new assembly** check box is selected, by default. It is used to invoke the **Open** dialog box automatically, if no components are opened in the current session of SOLIDWORKS. By using the **Open** dialog box, you can insert a component in the Assembly environment. The methods for inserting components in the Assembly environment are discussed next.

Figure 12.2

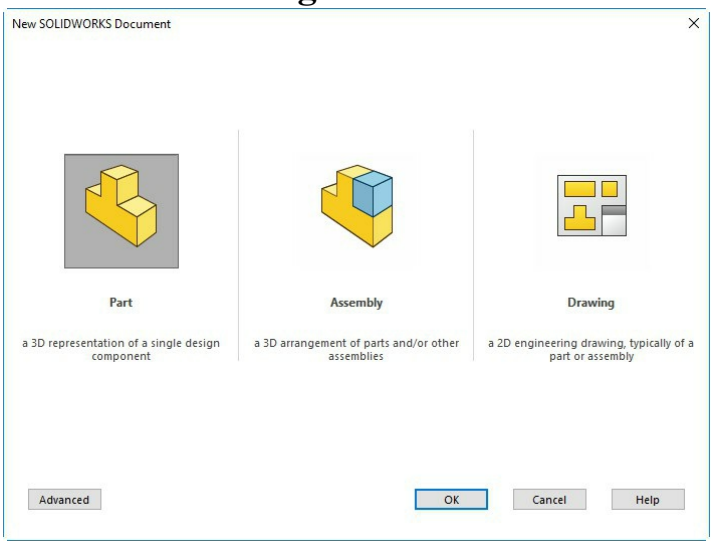
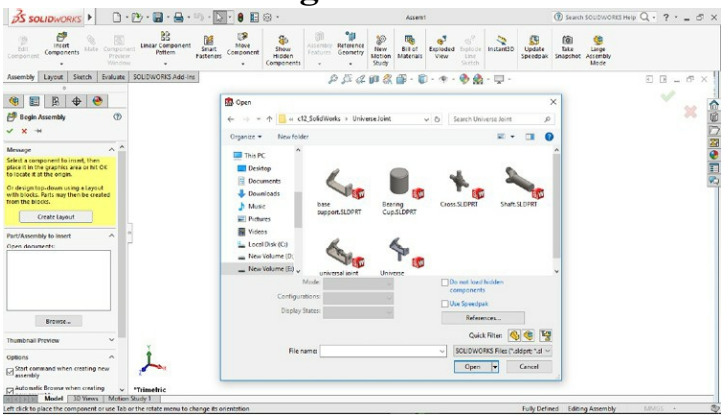


Figure 12.3



Note:

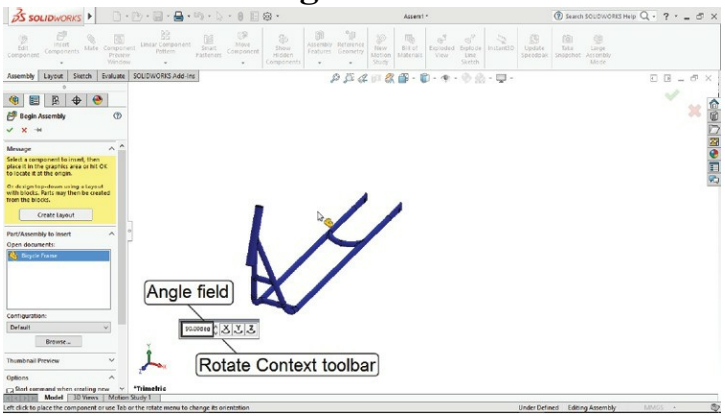
*The **Begin Assembly PropertyManager** appears every time on invoking the Assembly environment for inserting components in the Assembly environment. This is because, in the **Options** rollout of the **Begin Assembly PropertyManager**, the **Start command when creating new assembly** check box is selected, by default, refer to Figure 12.3. If you uncheck this check box, next time when you invoke the Assembly environment, the **Begin Assembly PropertyManager** as well as the **Open** dialog box do not appear. In such a case, you can insert components in the Assembly environment by using the **Insert Components** tool of the **Assembly CommandManager**. You will learn about inserting components by using the **Insert Components** tool later in this chapter.*

Inserting Components in the Assembly Environment Updated

As discussed, on invoking the Assembly environment, if no components are opened in the current session of SOLIDWORKS, the **Open** dialog box appears, automatically along with the **Begin Assembly PropertyManager**. If the **Open** dialog box does not appear then click on the **Browse** button in the **Begin Assembly PropertyManager** to invoke the **Open** dialog box. In the **Open** dialog box, browse to the location where all components of the assembly are saved and then select a component to be inserted. Next, click on the **Open** button in the dialog box. The selected component gets attached to the cursor and the **Rotate Context** toolbar appears in the graphics area, see Figure 12.4. If needed, you can change the orientation of the attached component by using the tools of

the **Rotate Context** toolbar. By default, 90 degrees is entered in the **Angle** field of the **Rotate Context** toolbar. As a result, when you click on the **X**, **Y**, or **Z** tool in this toolbar; the component rotates 90 degrees about the X, Y, or Z axis, respectively. You can enter any required value for the angle of rotation in the **Angle** field and rotate the component about X, Y, or Z axis by using this toolbar.

Figure 12.4

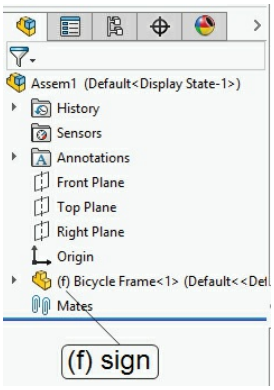


Note:

*By default, the **Rotate Context** toolbar appears ever time when you insert a component in the Assembly environment because the **Show Rotate context toolbar** check box is selected in the **Options** rollout of the **Begin Assembly PropertyManager**.*

Once the orientation of the component has been set by using the **Rotate Context** toolbar, click anywhere in the graphics area. The component moves toward the origin of the assembly and becomes a fixed component with respect to the origin of the assembly. Notice that the name of the inserted component gets listed in the FeatureManager Design Tree with '(f)' sign in front of its name, see Figure 12.5. The '(f)' sign indicates that all degrees of freedom of the component are fixed and the component cannot move or rotate in any direction. In SOLIDWORKS, the first component you insert in the Assembly environment becomes a fixed component automatically and does not allow any translational or rotational movement. You can also change the fixed component to a floating component, whose all degrees of freedom are free. Means the floating component is free to move or rotate in the graphics area. To change a fixed component to a floating component, right-click on the name of the fixed component in the FeatureManager Design Tree and then click on the **Float** option in the shortcut menu appeared. In addition, notice that as soon as the first component is inserted in the Assembly environment, the **Begin Assembly PropertyManager** gets closed. Therefore, to insert the remaining components of the assembly in the Assembly environment, you need to use the **Insert Components** tool of the Assembly CommandManager. The method of inserting components by using the **Insert Components** tool is as follows:

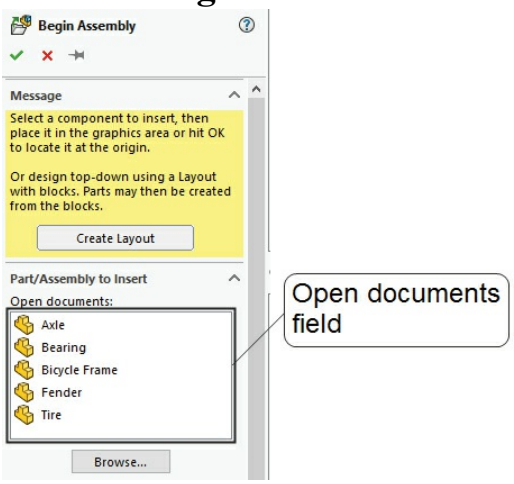
Figure 12.5



Note:

If any component is opened in the current session of SOLIDWORKS then on invoking the Assembly environment; the ***Open*** *dialog box will not be opened, automatically and the name of the opened component is listed in the* ***Open documents*** *field of the* ***Begin Assembly PropertyManager****, see Figure 12.6. In this figure, four components are listed in the* ***Open documents*** *field. You can select a component to be inserted in the Assembly environment from the* ***Open documents*** *field or click on the* ***Browse*** *button to open the* ***Open*** *dialog box.*

Figure 12.6



Tip:

If you pin the ***Begin Assembly PropertyManager*** *by clicking on the* ***Keep Visible*** *icon available at the upper right corner of the PropertyManager then the display of the PropertyManager will not be closed after inserting the component and you can continue inserting the components by using the* ***Begin Assembly PropertyManager****.*

Inserting Components by using the Insert Components Tool

To insert a component in the Assembly environment by using the **Insert Components** tool, click on the **Insert Components** tool in the **Assembly CommandManager**, see Figure 12.7. The **Insert Component PropertyManager** and the **Open** dialog box appear. Figure 12.8 shows the **Insert Component PropertyManager**.

Figure 12.7

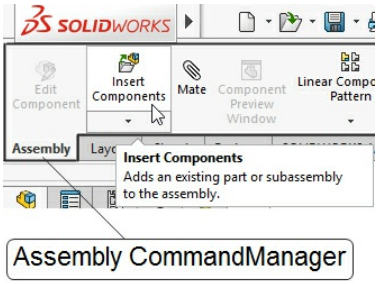
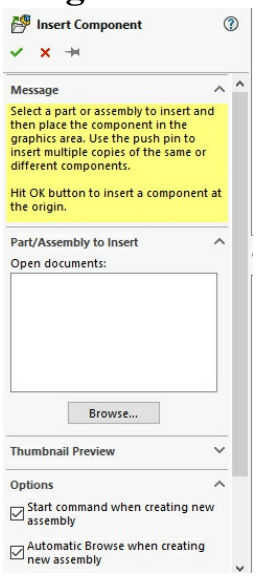


Figure 12.8



Note that in the **Insert Component PropertyManager**, the **Automatic Browse when creating new assembly** check box is selected, by default. As a result, the **Open** dialog box appears, automatically if no components are opened in the current session of SOLIDWORKS. If the **Open** dialog box does not appear, then click on the **Browse** button in the **Insert Component PropertyManager** to invoke the **Open** dialog box. In the **Open** dialog box, browse to the location where all components of the assembly are saved and then select a component. Next, click on the **Open** button in the dialog box. The selected component is attached to the cursor. Also, the **Rotate Context** toolbar appears in the graphics area, see Figure 12.9.

Figure 12.9

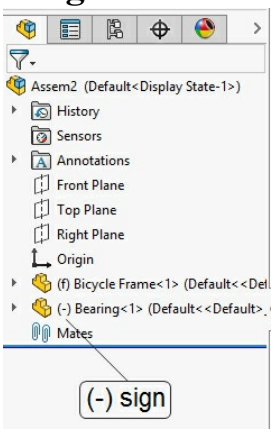
Note:

*The **Start command when creating new assembly** check box of the **Insert Component PropertyManager** is used to turn on or off the display of the **Begin Assembly PropertyManager** on invoking the Assembly environment.*

Change the current orientation of the component by using the **Rotate Context** toolbar, as needed. Once the orientation of the component has been set, as required then click the left mouse button anywhere in the graphics area to define the placement point for the attached component. The attached component is placed on the location defined and the

PropertyManager is closed. Also, the name of the inserted component is added in the FeatureManager Design Tree with '(-)' sign in front of its name, see Figure 12.10. The '(-)' sign indicates that all degrees of freedom of the component are not defined. Means the component is free to move or rotate in the graphics area. You need to assemble the free component with the existing components of the assembly by applying the required relations or mates. You will learn about applying relations or mates later in this chapter. Similarly, you can insert multiple components in the Assembly environment one by one by using the **Insert Component PropertyManager**.

Figure 12.10



Tip:

While defining the placement point for a component in the graphics area, make sure that the component does not intersect with any existing component of the assembly.

Note:

*If you pin the **Insert Component PropertyManager** by clicking on the **Keep Visible** icon available at its upper right corner then the display of the PropertyManager will not be closed after inserting the component and notice that the second instance of the inserted component is attached to the cursor. Click in the graphics area to insert the second instance of the same component. Similarly, you can insert multiple instances of a component in the Assembly environment by clicking the left mouse button.*

After inserting the second component, it is recommended that you first assemble the second component with the first component by applying required mates before inserting the third or next component in the Assembly environment. However, before you learn applying mates between assembly components, first it is important to understand the concept of degrees of freedom, which is as follows:

Working with Degrees of Freedom

A free component within the Assembly environment has six degrees of freedom: three translational and three rotational. It means a free component in the Assembly environment can move along the X, Y, and Z axes and rotate about the X, Y, and Z axes. As discussed earlier, the first component inserted in the Assembly environment becomes the fixed component automatically and does not allow any translational or rotational

movement. It means all its degrees of freedom are fixed. However, the second or further components inserted in the Assembly environment of SOLIDWORKS are free for all movements, which means degrees of freedom of components are not restricted. In such cases, a designer needs to fix the degrees of freedom of components by applying mates. It is not about fixing the degrees of freedom of components, you need to maintain actual relationships between them as exactly it is in the real world assembly. You need to allow movable components of an assembly to move freely in the respective movable directions. For example, the working of a shaft in an assembly is to rotate about its axis therefore, you need to retain its rotation degree of freedom free to rotate.

Note:

*To check the degrees of freedom of a component, you can move or rotate the component along or about its degrees of freedom by using the **Move Component** tool or **Rotate Component** tool, respectively. You can also move or rotate components along or about their free degrees of freedom by dragging them in the graphics area. You will learn about moving or rotating individual component of an assembly later in this chapter.*

Applying Relations or Mates Updated

In SOLIDWORKS, you can assemble components together by using three types of mates: Standard, Advanced, and Mechanical. All these types of mates can be applied by using the **Mate PropertyManager** that appears after clicking on the **Mate** tool in the **Assembly CommandManager**.

To apply a mate between components, click on the **Mate** tool in the **Assembly CommandManager**. The **Mate PropertyManager** appears, see Figure 12.11. Note that the **Entities to Mate** field in the **Mate Selections** rollout of the PropertyManager is activated by default. As a result, you can select entities of two different components to mate. Select the required entities such as faces, edges, planes, or a combination of these to apply a mate. Note that the first selected component for applying a mate becomes transparent in the graphics area, see Figure 12.12. It helps you easily select the second component, especially when the second component is not visible or behind the first component in the graphics area. This is because the **Make first selection transparent** check box is selected in the **Options** rollout of the **Mate PropertyManager**. Notice that as soon as you select entities for applying a mate, the **Mate** Pop-up toolbar appears in the graphics area, see Figure 12.12. The availability of tools in the **Mate** Pop-up toolbar depends on the entities selected for applying the mate. Also, in **Mate** Pop-up toolbar, the best suitable mate that can be applied between the selected entities is activated, by default and the preview of components appears after applying the best suitable mate, see Figure 12.12. In this figure, two cylindrical faces of the components are selected for applying a mate. As a result, the Concentric mate is activated, by default and the preview of selected entities appears concentric to each other. To accept the default selected mate, click on the green tick mark in the **Mate** Pop-up toolbar. You can also apply a mate other than the mate selected by default in the **Mate** Pop-up toolbar. To apply a mate other than the default selected mate, click on the required mate

tool to be applied between the entities in the **Mate** Pop-up toolbar. In addition to applying a mate by using the **Mate** Pop-up toolbar, you can also apply a mate between the entities by using the **Mate PropertyManager**. Different types of mates are as follows:

Figure 12.11

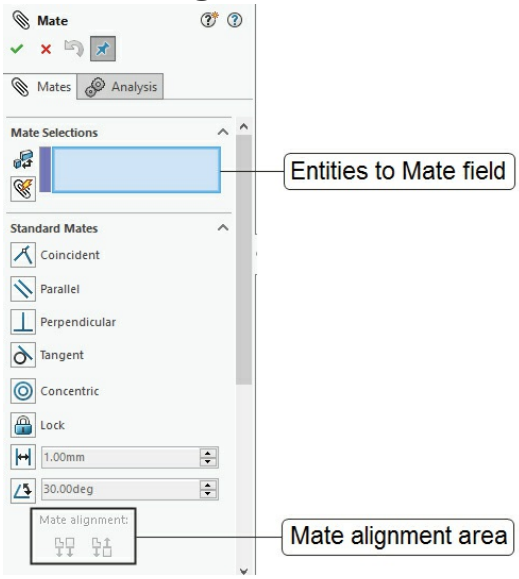
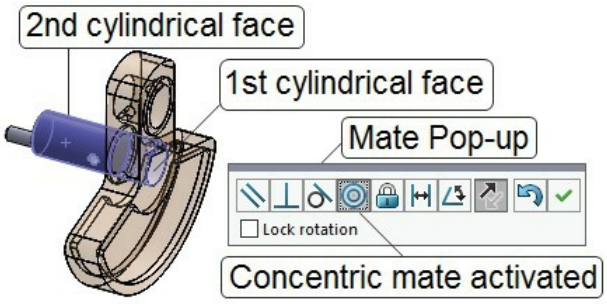


Figure 12.12



Note:

*The **Flip Mate Alignment** tool of the **Mate** Pop-up toolbar is used to flip the mate alignment between the selected entities. You can also flip the mate alignment between the entities by using the **Aligned** and **Anti-Aligned** buttons of the **Mate alignment** area in the **Mate PropertyManager**, refer to Figure 12.11. Note that these buttons are enabled only after selecting the entities. You will learn more about mate alignment between the entities later in this chapter.*

Working with Standard Mates Updated

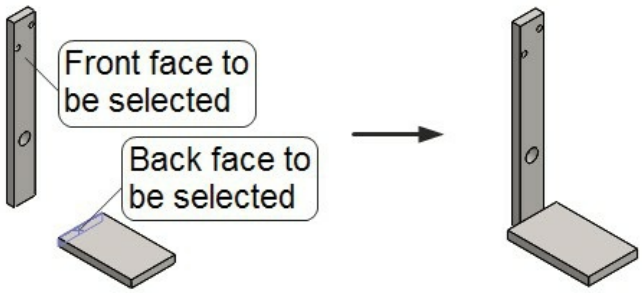
Standard mates are used for positioning components of an assembly by restricting or reducing the degrees of freedom of the components. You can apply standard mates such as coincident, parallel, perpendicular, concentric, and tangent between entities of the components to position them in the assembly. The standard mates are as follows:

Coincident

Coincident mate is used to make the selected entities of two different components

coincident to each other, see Figure 12.13. You can select faces, edges, planes, vertices, or a combination of these as entities for applying the coincident mate. On applying the coincident mate, the selected entities get aligned or share the same plane. Figure 12.13 shows two planar faces for applying coincident mate and the resultant model after applying the coincident mate.

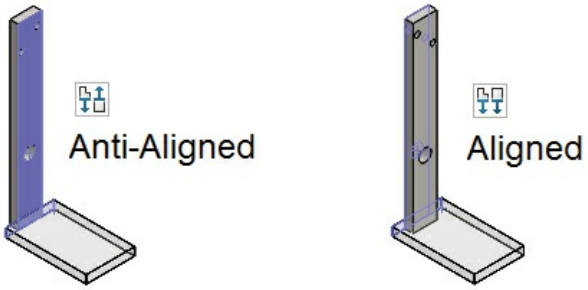
Figure 12.13



Note:

*If you click on the **Flip Mate Alignment** tool in the **Mate** Pop-up toolbar, the alignment of the selected entities changes from Anti-Aligned to Aligned or vice versa, see Figure 12.14. You can also flip the alignment by using the **Aligned** and **Anti-Aligned** buttons of the **Mate alignment** area in the **Mate PropertyManager**.*

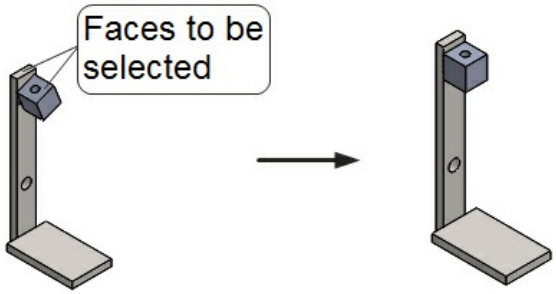
Figure 12.14



Parallel

The parallel mate is used to make the selected entities parallel to each other, see Figure 12.15. You can select planar faces, edges, planes, or a combination of these as entities for applying parallel mate. Figure 12.15 shows two planar faces and the resultant model after applying the parallel mate.

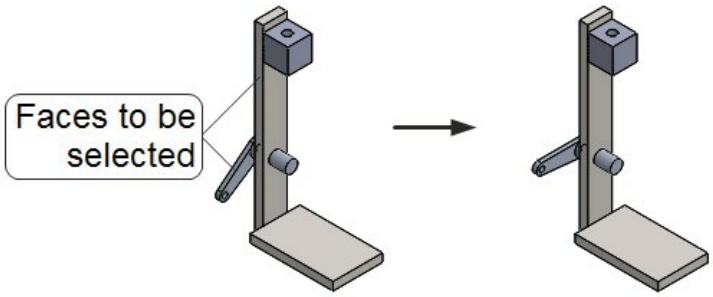
Figure 12.15



Perpendicular

The perpendicular mate is used to make the selected entities perpendicular to each other, see Figure 12.16. You can select planar faces, edges, planes, or a combination of these as entities for applying perpendicular mate.

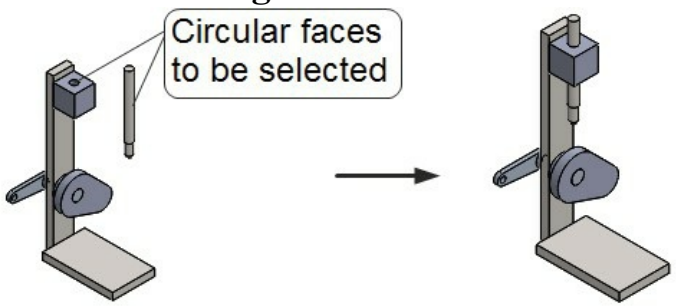
Figure 12.16



Concentric

The concentric mate is used to make the selected circular entities concentric to each other, see Figure 12.17. You can apply concentric mate between two circular or semi-circular faces and edges. On applying concentric mate, both the selected circular entities share a common axis.

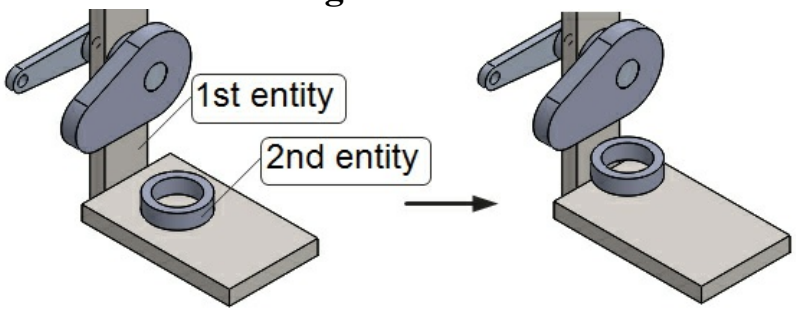
Figure 12.17



Tangent

The tangent mate is used to make the selected entities tangent to each other. You can select a planar face, a curved face, an edge, or a plane as the first entity and a cylindrical, conical, or spherical face as the second entity for applying the tangent mate. Figure 12.18 shows two faces for applying tangent mate and the resultant model after applying the tangent mate.

Figure 12.18



Distance Updated

The distance mate is used to keep distance between two entities. You can select planar faces, cylindrical faces, and planes as the entities to apply the distance mate. To apply the distance mate, after selecting the entities, click on the **Distance** button in the **Standard Mates** rollout of the PropertyManager or in the **Mate** Pop-up toolbar. Next, enter the required distance value in the **Distance** field. Figure 12.19 shows planar faces and the resultant model after applying the distance mate. In SOLIDWORKS 2017, while applying the distance mate between two cylindrical faces: **Center to Center**, **Minimum Distance**, **Maximum Distance**, and **Custom Distance** buttons become available in the **Standard Mates** rollout of the PropertyManager, see Figure 12.20. By using these buttons, you specify the type of distance measurement between the selected cylindrical faces, see Figure 12.21. In this figure, the distance mate is applied by measuring the center to center distance between the entities.

Figure 12.19

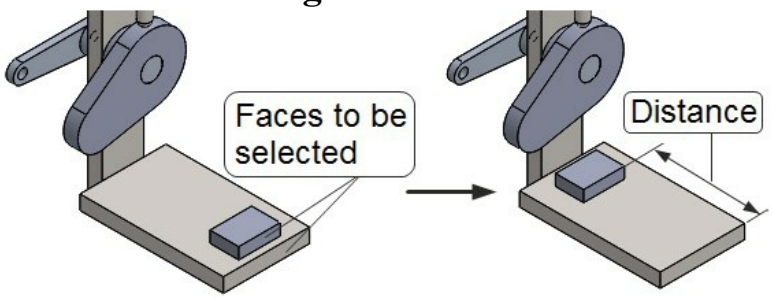
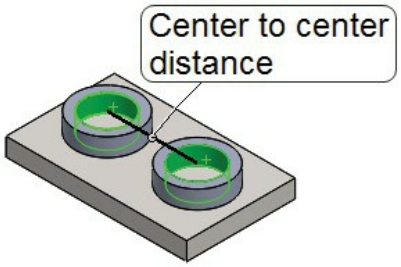


Figure 12.20

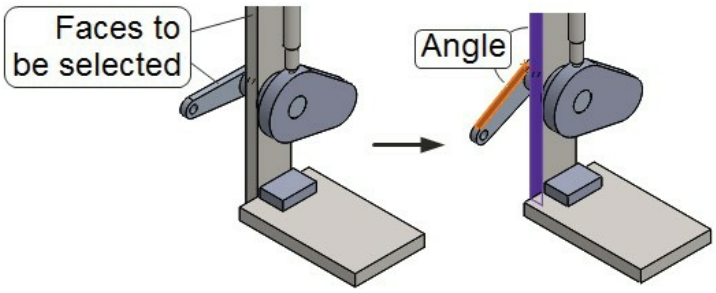
Figure 12.21



Angle

The angle mate is used to make angle between two selected entities. Figure 12.22 shows faces to be selected and the resultant model after applying the angle mate.

Figure 12.22



Note:

In case of the coincident, parallel, distance, and angle mates, you can flip the mate alignment between the selected entities from aligned to anti-aligned or vice versa.

Lock

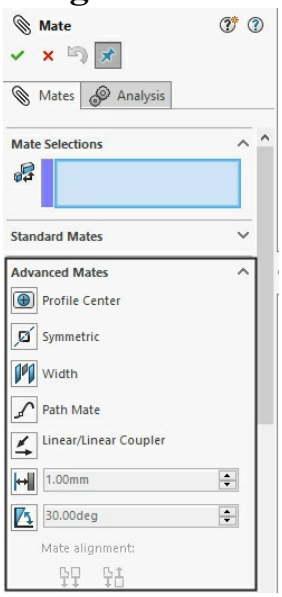
The lock mate is used to lock the selected entities at a desired position in the graphics area. On applying the lock mate, all degrees of freedom of the selected entities get fixed.

Working with Advanced Mates

Advanced mates are special types of mates and work one step forward above the standard mates for restricting or reducing degrees of freedom of components. You can apply advanced mates such as profile center, symmetric, width, path, and linear/linear coupler between components by using the **Mate** tool.

To apply advanced mates, click on the **Mate** tool in the **Assembly CommandManager**. The **Mate PropertyManager** appears. In this PropertyManager, expand the **Advanced Mates** rollout, see Figure 12.23. By using the buttons such as **Profile Center**, **Symmetric**, **Width**, and **Path Mate** in the **Advanced Mates** rollout, you can apply the respective advanced mates between components. The different advanced mates are as follows:

Figure 12.23



Profile Center Mate

The profile center mate is used to center-align two rectangular profiles, two circular profiles, or a rectangular and a circular profiles of two different components with each other, see Figures 12.24 through 12.26.

Figure 12.24

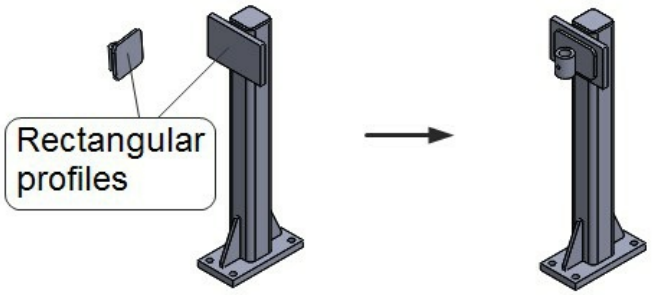


Figure 12.25

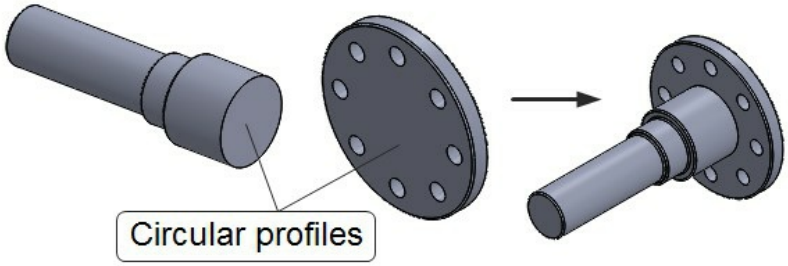
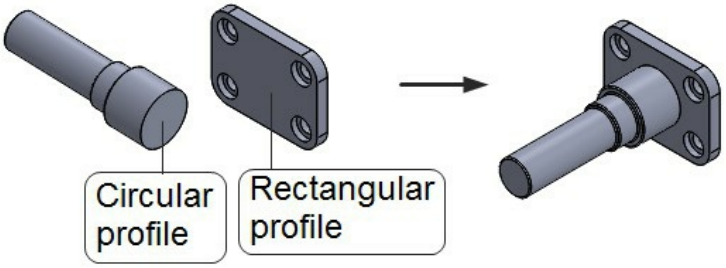
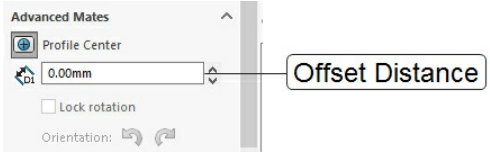


Figure 12.26



To apply the profile center mate, click on the **Profile Center** button in the **Advanced Mates** rollout of the **Mate PropertyManager**. The **Offset Distance** field is enabled in the **Advanced Mates** rollout, see Figure 12.27.

Figure 12.27



Select entities for applying the profile center mate in the graphics area. You can select two rectangular profiles, two circular profiles (circular faces or circular edges), or a rectangular profile and a circular profile as the entities for applying the profile center mate, refer to Figures 12.24 through 12.26. As soon as you select entities, the preview appears such that the selected entities get centrally aligned to each other. You can also specify offset distance between the selected entities/profiles by using the **Offset Distance** field of the **Advanced Mates** rollout. By default, **0** (zero) is entered in this field. You can enter an offset distance between the selected profiles, as required. The

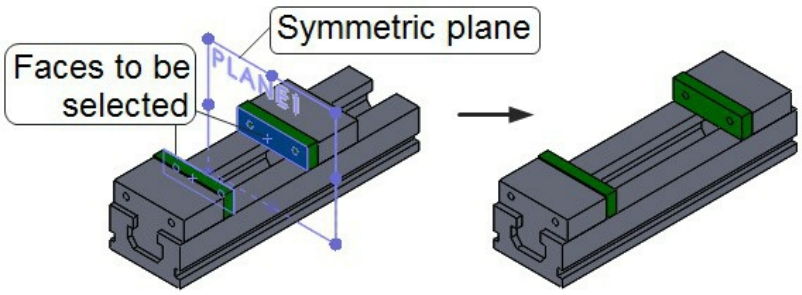
Flip dimension check box of the **Advanced Mates** rollout is used to flip the direction of offset dimension between the entities. The **Lock rotation** check box is used to restrict the rotation movement of the circular profile. Note that the **Lock rotation** check box is enabled if any of the selected entity is a circular profile.

Symmetric Mate

The symmetry mate is used to make two entities of different components symmetric about a plane or a planar face. You can select two vertices, sketch points, edges, axes, sketch lines, planes, planar faces, curved faces of same radii, and so on as the entities to be symmetric.

To apply the symmetric mate, click on the **Symmetric** button in the **Advanced Mates** rollout. The **Entities to Mate** and **Symmetry Plane** fields are enabled in the **Mate Selections** rollout of the PropertyManager. By default, the **Symmetry Plane** field is activated. As a result, you can select a plane or a planar face as the symmetric plane. Select a plane or a planar face as the symmetric plane in the graphics area. The **Entities to Mate** field gets activated, automatically. Select two entities of different components to be symmetric about the symmetric plane in the graphics area. Next, click on the green tick mark in the PropertyManager. The symmetric mate is applied between the entities with respect to the symmetric plane. Figure 12.28 shows two planar faces as the entities to be symmetric about a symmetric plane and the resultant model after applying the symmetric mate.

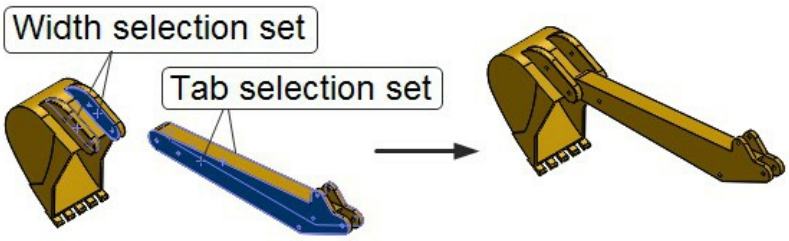
Figure 12.28



Width Mate

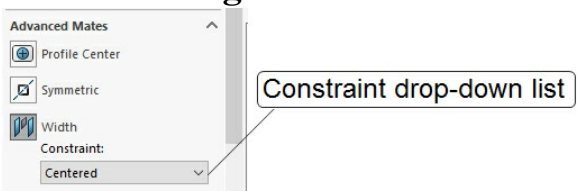
The width mate is used to make two planar faces, a cylindrical face, or an axis of a component centered between two planar faces of another component, see Figure 12.29. The width mate needs two pairs of selections: one pair of selection is known as width selection and the other pair of selection is known as tab selection, see Figure 12.29. After applying the width mate, the tab selection set is aligned centrally between the width selection set, by default. You can select two planar faces (parallel or non parallel), a cylindrical face, or an axis as the tab selection set. For a width selection set, you can select two parallel or non parallel faces. Figure 12.29 shows planar faces selected as the width selection set and the tab selection set, and the resultant model after applying the width mate.

Figure 12.29



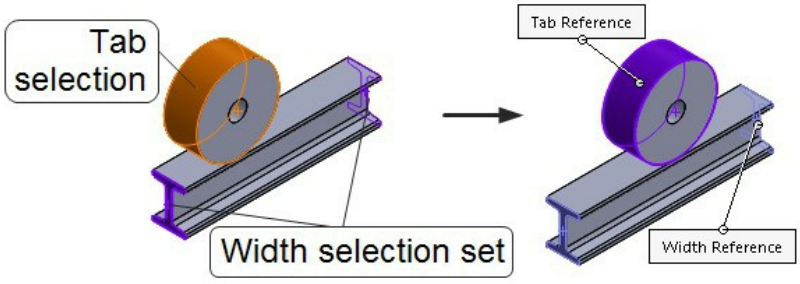
To apply the width mate, click on the **Width** button in the **Advanced Mates** rollout. The **Width selections** and **Tab selections** fields are enabled in the **Mate Selections** rollout of the PropertyManager. Also, the **Constraint** drop-down list becomes available in the **Advanced Mates** rollout, see Figure 12.30.

Figure 12.30



By default, the **Width selections** field is activated in the **Mate Selections** rollout. As a result, you can select width selection set in the graphics area. Select two entities (parallel or non parallel faces) as the width selection set. As soon as you select the width selection set, the **Tab selections** field gets activated. Select two planar faces or a cylindrical face as the tab selection set. The preview of the width mate appears. Figure 12.31 shows the two planar faces as the width selection, a cylindrical face as the tab selection, and the preview of the resultant width mate.

Figure 12.31



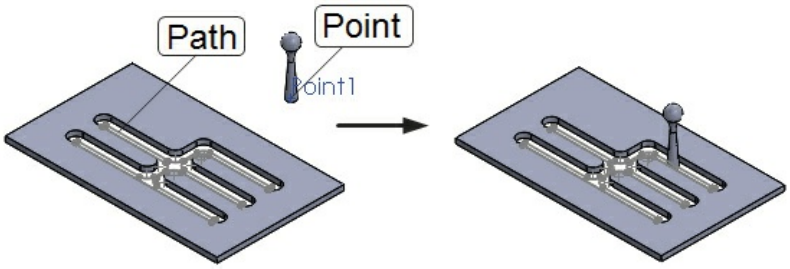
Note that the **Centered** option is selected in the **Constraint** drop-down list of the **Advanced Mates** rollout, refer to Figure 12.30. As a result, the tab selection set is centered between the width selection set. You can also select the **Free**, **Dimension**, or **Percent** option from the **Constraint** drop-down list, as required. On selecting the **Free** option, the tab selection set can move freely within the limit of the width selection set. On selecting the **Dimension** option, you can control the position of the tab selection set by specifying a distance value in the **Distance from the End** field of the rollout. On selecting the **Percent** option, you can control the position of the tab selection set by specifying a percentage value in the **Percentage of Distance from the End** field of the

rollout. After selecting the required option in the **Constraint** drop-down list, click on the green tick mark in the PropertyManager. The width mate is applied between the selected entities.

Path Mate Updated

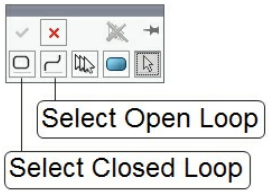
The path mate is used to constraint a point or a vertex of a component with a path such that the component can move along the defined path, see Figure 12.32.

Figure 12.32



To apply the path mate, click on the **Path Mate** button in the **Advanced Mates** rollout. The **Component Vertex** and **Path Selection** fields are enabled in the **Mate Selections** rollout of the PropertyManager. By default, the **Component Vertex** field is activated. As a result, you can select a point or a vertex of a component for applying the path mate, refer to Figure 12.32. As soon as you select a point or a vertex, the **Path Selection** field gets activated. Select a sketch (open or close) or an edge as the path from the graphics area by clicking the left mouse button. You can use the **SelectionManager** button of the **Mate Selections** rollout to select a sketch having multiple entities. On clicking this button, the **Selection** Pop-up toolbar appears, see Figure 12.33. By using the **Select Closed Loop** and the **Select Open Loop** tools of this Pop-up toolbar, you can select a closed loop and an open loop sketch as the path, respectively. After selecting the path by using the **Selection** Pop-up toolbar, click on the green tick mark in the Pop-up toolbar.

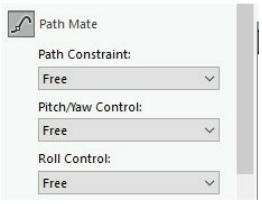
Figure 12.33



Note that the **Free** option is selected in the **Path Constraint** drop-down list in the **Advanced Mates** rollout, see Figure 12.34. As a result, the component moves freely along the selected path. You can also select the **Distance Along Path** or **Percent Along Path** option from the **Path Constraint** drop-down list, as required. On selecting the **Distance Along Path** option, you can control the position of the component along the path by specifying a distance value in the **Distance from the End** field of the rollout. On selecting the **Percent Along Path** option, you can control the position of the component

along the path by specifying a percentage value in the **Percentage of Distance from the End** field of the rollout. Also, in the **Pitch/Yaw Control** and **Roll Control** drop-down lists in the rollout, the **Free** option is selected, by default. As a result, the pitch, yaw, and roll of the component are not constrained and the component is allowed to move freely along the path. On selecting the **Follow Path** option in the **Pitch/Yaw Control** drop-down list, you make an axis (X, Y, or Z) of the component tangent to the path by selecting the respective radio button which appeared on selecting the **Follow Path** option. On selecting the **Up Vector** option in the **Roll Control** drop-down list, you can align an axis (X, Y, or Z) of the component to a vector. You can select a linear edge or a planar face as the vector by using the field which appeared on selecting the **Up Vector** option. After selecting the required option in the **Path Constraint**, **Pitch/Yaw Control**, and **Roll Control** drop-down lists, click on the green tick mark in the PropertyManager. The path mate is applied.

Figure 12.34



Tip:

To review the movement of the component along the path after applying the path mate, select the moveable component and then drag it such that it travels along the defined path.

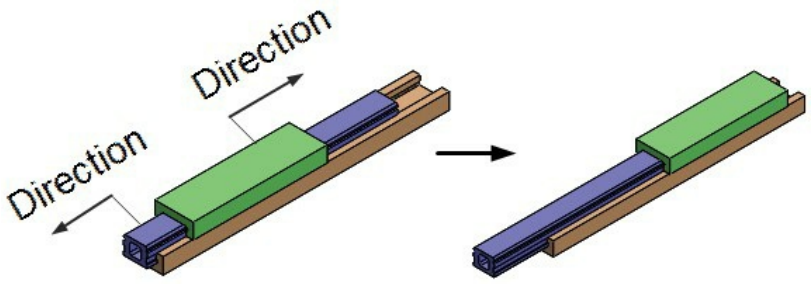
Note:

To achieve the correct path motion, you need to define the degrees of freedom of the component to be moved such that the component can only move along the path.

Linear/Linear Coupler Mate

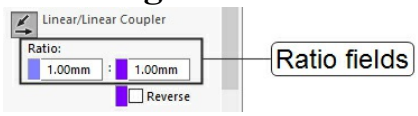
The linear/linear coupler mate is used to translate the motion of components with respect to each other. After applying the linear/linear coupler mate between two components, when you move a component, the other component also moves accordingly, see Figure 12.35.

Figure 12.35



You can specify the translation ratio between the components by using the **Ratio** fields of the PropertyManager, see Figure 12.36. If the translation ratio is 1: 2 in mm unit, on translating one component to a distance of 1 mm in a direction; the second component is translated automatically to a distance of 2 mm. You can reverse the translation direction of components by using the **Reverse** check box.

Figure 12.36



To apply the linear/linear coupler mate, click on the **Linear/Linear Coupler** button in the **Advanced Mates** rollout. Next, select linear edges of two components one by one as the direction to move the components with respect to each other. Next, specify the translation ratio in the **Ratio** fields of the PropertyManager and then click on the green tick mark in the PropertyManager. The linear/linear coupler mate is applied.

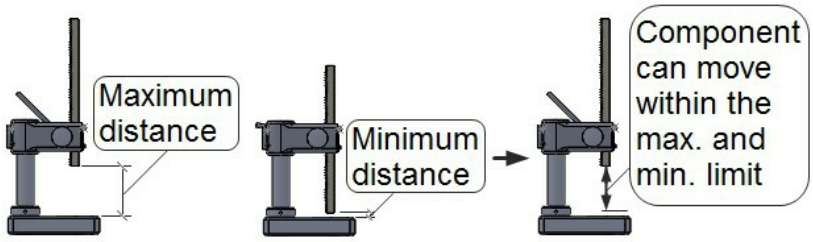
Tip:

To review the translation motion between two components after applying the linear/linear coupler mate, select a component and then drag it.

Distance Mate

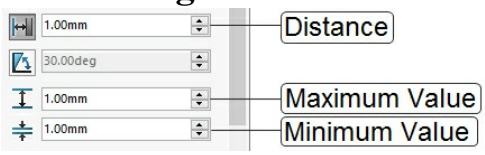
The advanced distance mate of the **Advance Mates** rollout is used to specify the minimum and maximum distance limit between two components. After applying this mate, the components can move or translate within the specified distance limit, see Figure 12.37.

Figure 12.37



To apply the advanced distance mate, click on the **Distance** button in the **Advanced Mates** rollout. The **Distance**, **Maximum Value**, and **Minimum Value** fields get enabled in the PropertyManager, see Figure 12.38. Specify the maximum distance value in the **Maximum Value** field. Next, specify the minimum distance value in the **Minimum Value** field. After specifying the maximum and minimum distance values, select two entities (faces, planes, edges, points, vertices, or a combination of these) of two different components and then click on the green tick mark in the PropertyManager. The advanced distance mate is applied between the selected entities of the components such that the moveable component can move within the specified maximum and minimum distance limit, refer to Figure 12.37.

Figure 12.38



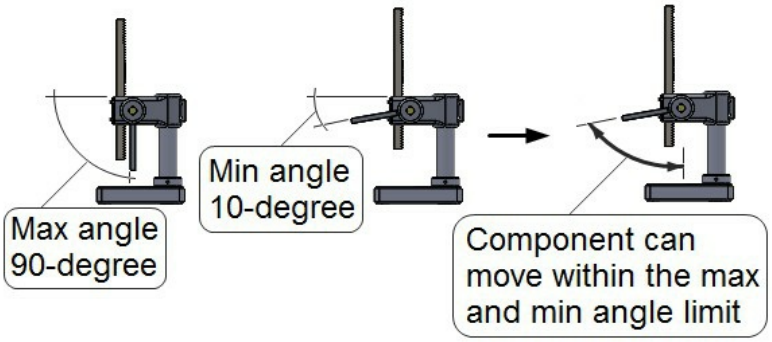
Tip:

To review the distance motion between components, select the moveable component and then drag it.

Angle mate

The advanced angle mate of the **Advance Mates** rollout is used to specify the minimum and maximum angle limit between two components. After applying this mate, the components can rotate within the specified angle limit, see Figure 12.39.

Figure 12.39



To apply the advanced angle mate, click on the **Angle** button in the **Advanced Mates** rollout. The **Angle**, **Maximum Value**, and **Minimum Value** fields are enabled in the PropertyManager, see Figure 12.40. Specify the maximum angle value in the **Maximum Value** field. Next, specify the minimum angle value in the **Minimum Value** field of the rollout. After specifying the maximum and minimum angle values, select two faces of two different components and then click on the green tick mark in the PropertyManager. The advanced angle mate is applied between the selected faces of the components such that the moveable component can rotate within the specified maximum and minimum

angle limit, see Figure 12.39.

Figure 12.40

Tip:

To review the angle motion between components, select the moveable component and then drag it.

Working with Mechanical Mates

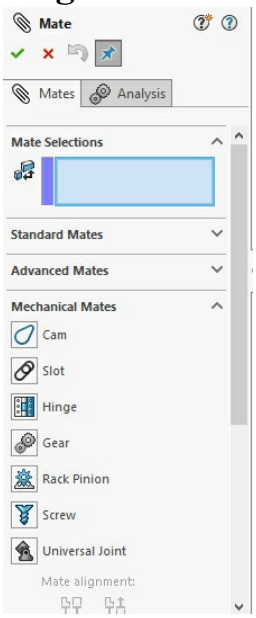
Mechanical mates are used to create mechanism between components of an assembly. You can create cam and follower, gear, hinge, rack and pinion, screw, and universal joint mechanisms between components of an assembly by using mechanical mates. Different types of mechanical mates are available in the **Mechanical Mates** rollout of the **Mate PropertyManager**.

Note:

To create a mechanism between components of an assembly by applying mechanical mates, it is necessary to have required degrees of freedom of components such that the components can only be moved or rotated in the desired directions. For example, to create a gear mechanism between two gear components, you need to first constraint the gear components such that they can only rotate about their axes. You can constraint degrees of freedom of a component by using the standard and advanced mates.

To apply a mechanical mate, expand the **Mechanical Mates** rollout of the **Mate PropertyManager**, see Figure 12.41. Different types of mechanical mates are as follows:

Figure 12.41



Cam Mate

The cam mate is used to create cam and follower mechanism between two components of an assembly.

To apply the cam mate, click on the **Cam** button in the **Mechanical Mates** rollout. The **Cam Path** and **Cam Follower** fields appear in the **Mate Selections** rollout of the PropertyManager. By default, the **Cam Path** field is activated. As a result, you can select a face of the cam component. Select a face of the cam component, which is tangent to the series of other faces of the cam component and forms a closed loop, see Figure 12.42. As soon as you have selected a face of the cam component, all faces that are tangent to the selected face get selected automatically and the cam component becomes transparent in the graphics area, see Figure 12.42. Also, the **Cam Follower** field gets activated in the **Mate Selections** rollout. As a result, you can select a face of the follower component. Select a cylindrical, a semi-cylindrical, or a planar face of the follower component, see Figure 12.43. The selected face of the follower component is placed over the cam component, see Figure 12.44. Next, click on the green tick mark in the PropertyManager. The cam mate is applied between the cam and follower components. Now, on rotating the cam component, the follower component moves up and down with respect to the cam profile such that it follows the cam and follower mechanism.

Figure 12.42

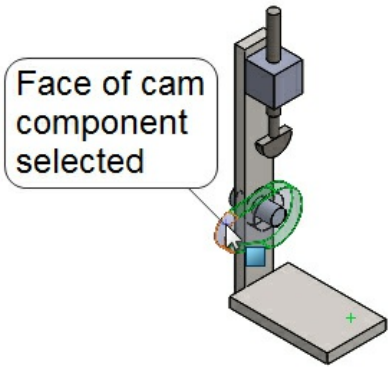


Figure 12.43

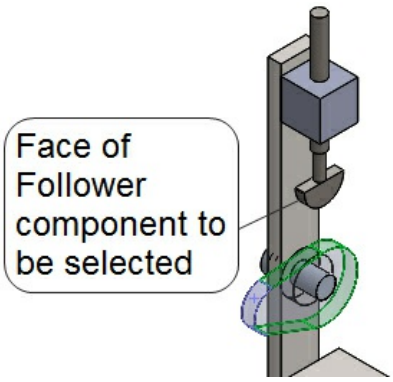
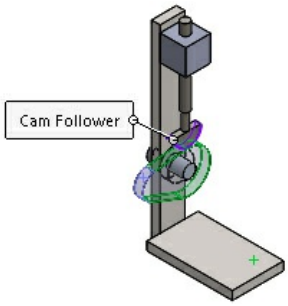


Figure 12.44



Tip:

To review the cam and follower mechanisms between the components, select the cam component and then drag it such that it rotates around its axis of rotation.

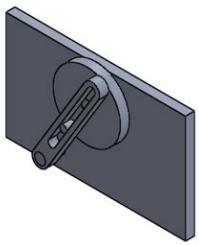
Note:

To select faces of components for applying mates, you may need to move or rotate the assembly or its individual components. To rotate an assembly, drag the cursor by pressing and holding the middle mouse button. To pan the assembly, drag the cursor by pressing and holding the CTRL key pulse middle mouse button. Alternatively, you can use the ***Rotate*** *and* ***Pan*** *tools to rotate and pan the assembly, respectively. You will learn about moving or rotating individual components of an assembly later in this chapter.*

Slot Mate

The slot mate is used to drive a bolt, a pin, or a cylindrical feature of a component along the slot of another component in an assembly, see Figure 12.45. You can also apply the slot mate mechanism between two slots of the components.

Figure 12.45



To apply the slot mate, click on the **Slot** button in the **Mechanical Mates** rollout. Next, select a face of the slot of a component, see Figure 12.46. All the remaining tangent faces of the slot are selected, automatically and the component becomes transparent in the graphics area, see Figure 12.46. Next, select a cylindrical face of another component (a bolt, a pin, or a cylindrical feature), see Figure 12.47. The cylindrical face is placed between the slot, see Figure 12.47. Next, click on the green tick mark in the PropertyManager. The slot mate is applied between the components and now you can review the slot mechanism by dragging the components.

Figure 12.46

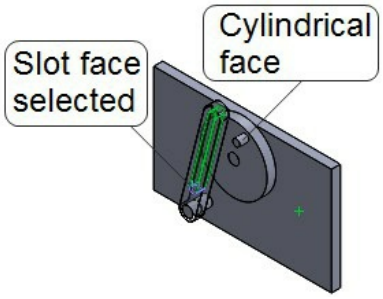
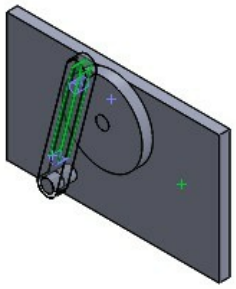


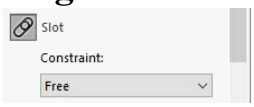
Figure 12.47



Note:

*When you click on the **Slot** button, the **Constraint** drop-down list appears in the rollout, see Figure 12.48. By default, the **Free** option is selected in this drop-down list. As a result, the component having the bolt, pin, or cylindrical feature can move freely within the slot of the other component. You can constraint the movement of components by using the **Center in Slot**, **Distance Along Slot**, or **Percentage Along Slot** option of the **Constraint** drop-down list.*

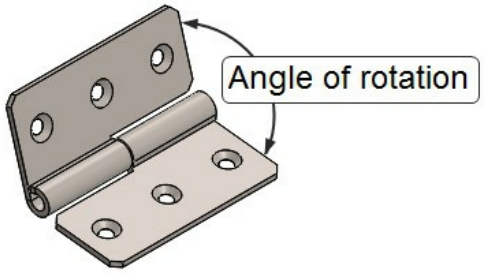
Figure 12.48



Hinge Mate

The hinge mate is used to create hinge mechanism between two components by fixing all degrees of freedom of the components except the rotational degree of freedom, see Figure 12.49. You can also limit the rotational degree of freedom by specifying the minimum and maximum angle of rotation.

Figure 12.49



To apply the hinge mate, click on the **Hinge** button, the **Concentric Selections** and **Coincident Selections** fields are enabled in the **Mate Selections** rollout of the

PropertyManager. Select two circular faces or circular edges of two different components to be concentric with each other, see Figure 12.50. Next, select two planar faces of the components to be coincident with each other, see Figure 12.51. The concentric and coincident mates are applied between the selected faces of the components, see Figure 12.52. Next, click on the green tick mark in the PropertyManager. The hinge mate is applied and now you can review the hinge mechanism by dragging the components.

Figure 12.50

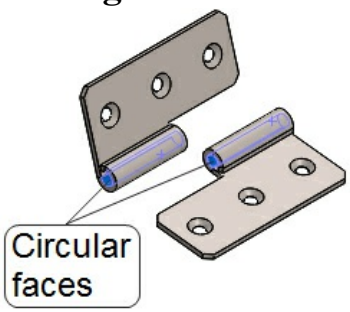


Figure 12.51

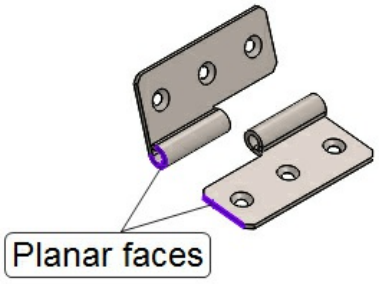
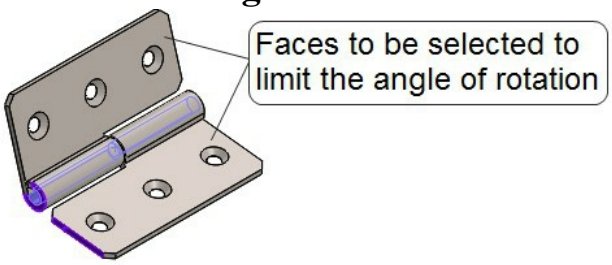


Figure 12.52



Note:

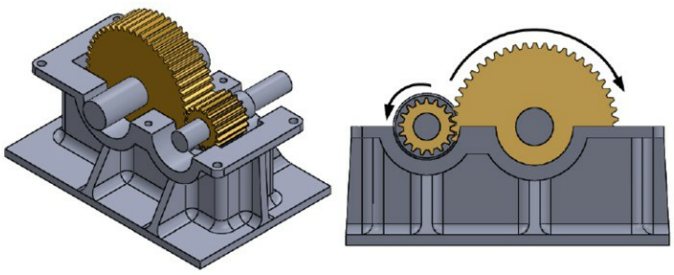
By default, the angle of rotation for the hinge mechanism is 360 degrees. To limit the angle of rotation for the hinge mechanism between the components, click on the ***Specify angle limits*** *check box in the* ***Mate Selections*** *rollout of the PropertyManager. The* ***Angle Selections***, ***Angle***, ***Maximum Value***, *and* ***Minimum Value*** *fields are enabled in the rollout. By default, the* ***Angle Selections*** *field is activated. As a result, you can select faces of the components to limit the angle of rotation between them. Select two planar faces of components, see Figure 12.52. Next, specify the maximum and minimum angle values in the* ***Maximum Value*** *and* ***Minimum Value*** *fields, respectively and then click on the green tick mark in the PropertyManager.*

Gear Mate

The gear mate is used to create gear mechanism between components such that the

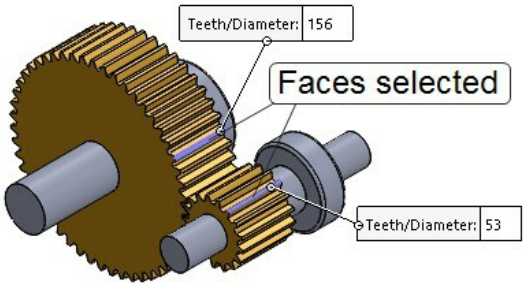
components can rotate relative to each other, see Figure 12.53.

Figure 12.53



To apply the gear mate, click on the **Gear** button in the **Mechanical Mates** rollout. Next, select a circular face of a gear teeth of the first gear component and then select a circular face of a gear teeth of the second gear component, see Figure 12.54. On selecting the circular faces of the gear components, the axis of rotations of the gears are defined automatically. You can select circular faces, conical faces, axes, or linear edges of the gear components to define the axes of rotation. Next, specify the gear ratio in the **Ratio** fields of the **Mechanical Mates** rollout. Notice that based on the relative size of the faces selected for defining the axis of rotation, the gear ratio is automatically calculated. After defining the gear ratio, click on the green tick mark in the PropertyManager. The gear mate is applied and now you can review the gear mechanism by dragging the gears.

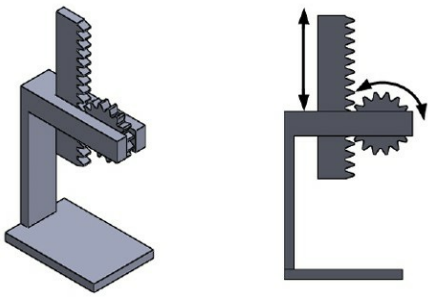
Figure 12.54



Rack Pinion Mate

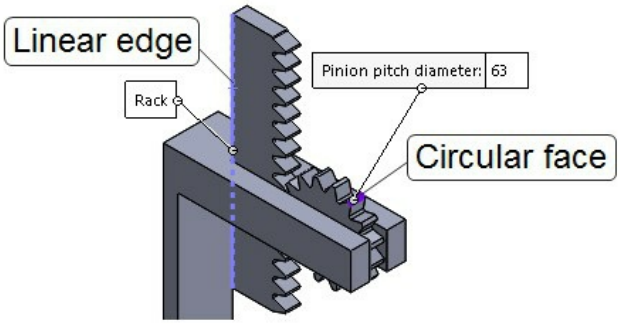
The rack pinion mate is used to translate linear motion of one component to the rotational motion of another component and vice versa. This mate creates the rack and pinion mechanism, see Figure 12.55, where the linear motion of the rack component creates rotatory motion in the pinion component and vice versa.

Figure 12.55



To apply the rack pinion mate, click on the **Rack Pinion** button, the **Rack** and **Pinion/Gear** fields are enabled in the **Mate Selections** rollout of the PropertyManager. Select a linear edge of the rack component, which defines the direction of movement of the rack component, see Figure 12.56. As soon as you select a linear edge of the rack component, the **Pinion/Gear** field gets activated. Select a circular face of the pinion/gear component, which defines the axis of rotation of the pinion component, see Figure 12.56. Next, click on the green tick mark in the PropertyManager. The rack pinion mate is applied between the components and now you can review the rack and pinion mechanism by dragging the components.

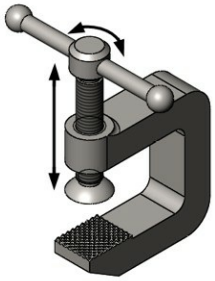
Figure 12.56



Screw Mate

The screw mate applies the pitch relationship between the rotation of one component and the translation of the other component such that it forms a screw mechanism, see Figure 12.57. On applying the screw mate, the translational motion of one component causes rotational motion in other component based on the specified pitch relationship. You can specify the pitch relationship between two components either by defining the number of revolutions of one component with respect to the per millimeter translation of another component or by defining the distance travelled by one component with respect to the per revolution of the other component.

Figure 12.57



To apply the screw mate, click on the **Screw** button in the **Mechanical Mates** rollout. Next, select a circular face of the first component and then select a circular face of the second component to define their axes of rotation, see Figure 12.58. On selecting the circular faces of both the components, an arrow appears in the graphics area which indicates the direction of revolution, see Figure 12.58. You can reverse the direction of revolution by using the **Reverse** check box of the **Mechanical Mates** rollout, see Figure

12.59. By default, the **Revolutions/mm** radio button is selected in the **Mechanical Mates** rollout, see Figure 12.59. As a result, you can specify the number of revolutions of one component with respect to the per millimeter translation of the other component in the **Revolutions/Distance** field. If you select the **Distance/revolution** radio button then you can specify the distance travelled by one component with respect to the per revolution of other component in the **Revolutions/Distance** field. Next, click on the green tick mark in the PropertyManager. The screw mate is applied between the components and now you can review the screw mechanism by dragging the components.

Figure 12.58

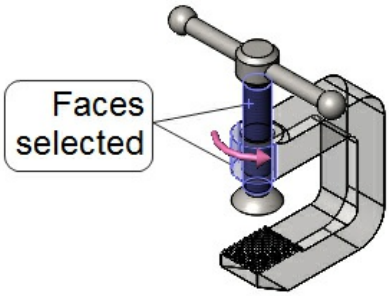
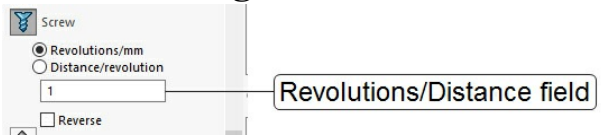


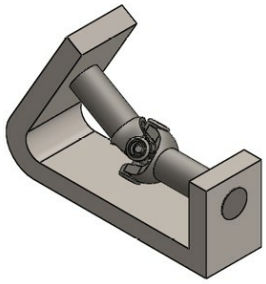
Figure 12.59



Universal Joint

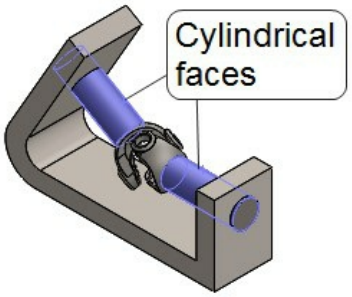
The universal joint mate is used to translate the rotational movement of one component to the rotational movement of another component about their axes of rotation, see Figure 12.60.

Figure 12.60



To apply the universal joint mate, click on the **Universal Joint** button in the **Mechanical Mates** rollout. Next, select two cylindrical faces of different components in the graphics area one by one, see Figure 12.61. After selecting the cylindrical faces, click on the green tick mark in the PropertyManager. The universal joint mate is applied between the components and now you can review the universal joint mechanism by dragging the components.

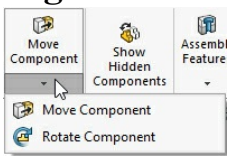
Figure 12.61



Moving and Rotating Individual Components

In SOLIDWORKS, you can move and rotate individual components of an assembly about their free degrees of freedoms by using the **Move Component** and **Rotate Component** tools of the **Assembly CommandManager**, see Figure 12.62. Both the tools are as follows:

Figure 12.62



Moving a Component by using the Move Component Tool

In SOLIDWORKS, you can move individual components of an assembly along their free degrees of freedoms by using the **Move Component** tool. To move a component of an assembly, click on the **Move Component** tool in the **Assembly CommandManager**. The **Move Component PropertyManager** appears, see Figure 12.63. By default, the **Free Drag** option is selected in the **Move** drop-down list of the **Move** rollout in the PropertyManager. As a result, you can move the component freely along its free degrees of freedom. Select the component to be moved and then drag the cursor by pressing and holding the left mouse button. The selected component starts moving along with the cursor. To stop the movement of the component, release the left mouse button. The other options in the **Move** drop-down list are **Along Assembly XYZ**, **Along Entity**, **By Delta XYZ**, and **To XYZ Position**, see Figure 12.64. All these options are as follows:

Figure 12.63

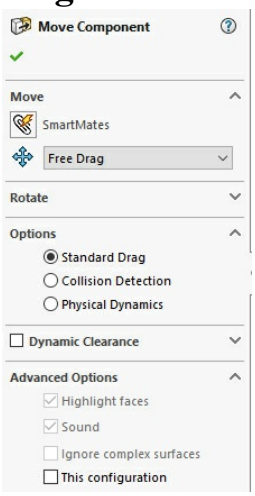
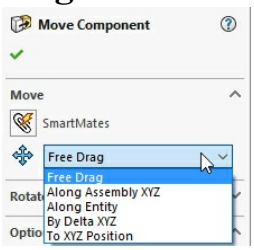


Figure 12.64



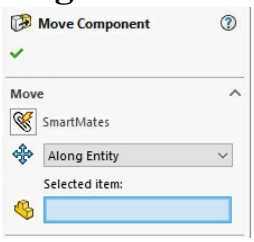
Along Assembly XYZ

On selecting the **Along Assembly XYZ** option, you can move a component along the X, Y, or Z axis of the assembly coordinate system.

Along Entity

On selecting the **Along Entity** option; the Selected item field appears in the **Move** rollout, see Figure 12.65. This field is used to select an entity as the direction along which the selected component has to be moved. You can select a linear edge, a sketch line, or an axis as the entity to define the direction along which the component has to be moved. After selecting the entity, select the component to be moved and then drag the cursor. The selected component starts moving along the direction of the entity selected.

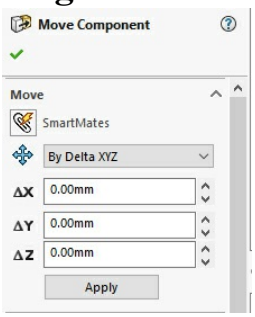
Figure 12.65



By Delta XYZ

On selecting the **By Delta XYZ** option, the **Delta X**, **Delta Y**, and **Delta Z** fields appear, see Figure 12.66. In these fields, you can specify the X, Y, and Z distance values for moving the component with respect to the current location of the component. After specifying the X, Y, and Z distance values in the respective fields, select the component to be moved and then click on the **Apply** button. The selected component moves with respect to the specified distance value.

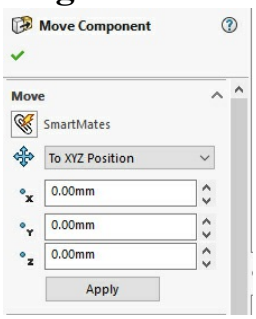
Figure 12.66



To XYZ Position

On selecting the **To XYZ Position** option, the **X Coordinate**, **Y Coordinate**, and **Z Coordinate** fields get enabled in the PropertyManager, see Figure 12.67. These fields are used to specify the X, Y, and Z coordinates of the location where you want to move the selected component. After specifying the coordinate, click on the **Apply** button. The origin of the selected component is moved to the specified coordinate. Note that if you select a vertex or a point of the component to be moved then after clicking on the **Apply** button, the selected vertex or point of the component is moved to the specified coordinate location.

Figure 12.67



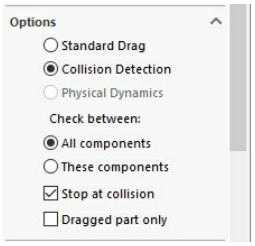
Note that by default the movement of the component is not prevented from any interference or collision occurred with other components of an assembly. It means the component moves continuously even if any other component comes across its way. This is because the **Standard Drag** radio button is selected in the **Options** rollout of the **Move Component PropertyManager**, see Figure 12.68. By selecting the **Collision Detection** or **Physical Dynamics** radio button, you can detect collisions or analyze the motion between components of an assembly. The methods of detecting collision and analyzing motion between components by using the **Collision Detection** and **Physical Dynamics** radio buttons are as follows:

Figure 12.68

Detecting Collision between Components

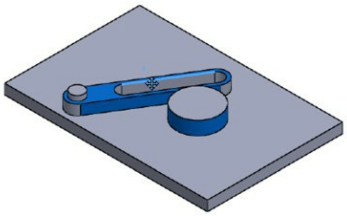
You can detect the collision between components of an assembly while moving a component by using the **Collision Detection** radio button. To detect collision between components, select the **Collision Detection** radio button in the **Options** rollout of the PropertyManager. The **All components** and **These components** radio buttons as well as the **S**top at collision **and Dragged part only** check boxes appear in the **Options** rollout of the PropertyManager, see Figure 12.69.

Figure 12.69



By default, the **All components** radio button is selected. As a result, collision is detected when the moveable component collides with any other component of the assembly. Note that as soon as collision is detected between the components, the components are highlighted in the graphics area, see Figure 12.70. If the **Stop at collision** check box is selected in the rollout then the movement of the component is stopped as soon as it collides with other component of the assembly.

Figure 12.70



On selecting the **These components** radio button, the **Components for Collision** Check field appears in the PropertyManager. This field is used to select components for the detection of collision. After selecting the components for the detection of collision, click on the **Resume Drag** button and then drag the component to be moved. Note that the collision is detected only when the component being moved touches any of the selected components of the assembly. Note that the components which are not listed in the **Components for Collision** Check field are ignored by the moveable component.

When the **Dragged part only** check box is selected in the **Options** rollout, collision is checked only for the component which is selected to be moved. However, if this check box is cleared then the collision is even checked for the components that move because of the mates with the moveable component.

Detecting Collision and Analyzing Motion between Components

Similar to the **Collision Detection** radio button, the **Physical Dynamics** radio button is also used to detect collision with the only difference that it forces the components to move along with the component, when the collision gets detected between them. In other words, by selecting the **Physical Dynamics** radio button, you can analyze the motion between the components of an assembly. Note that components can only move or rotate within their free degrees of freedom.

Note:

*In addition to moving individual components and detecting collision, you can also rotate components by using the **Rotate Component***

*PropertyManager. You can invoke the **Rotate Component PropertyManager** by expanding the **Rotate** rollout of the **Move Component PropertyManager** or by clicking on the **Rotate Component** tool in the CommandManager. The options of the **Rotate Component PropertyManager** are discussed next.*

Rotating a Component by using the Rotate Component Tool

Similar to moving individual components, you can rotate individual components of an assembly about their free degrees of freedom by using the **Rotate Component** tool. To rotate a component of an assembly, click on the **Rotate Component** tool in the **Assembly CommandManager**, see Figure 12.71. The **Rotate Component PropertyManager** appears, see Figure 12.72.

Figure 12.71

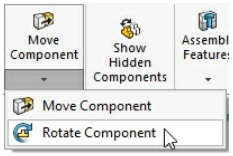
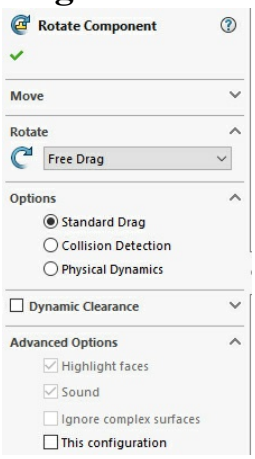


Figure 12.72



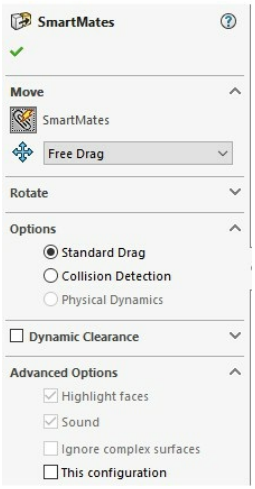
By default, the **Free Drag** option is selected in the **Rotate** drop-down list of the **Rotate** rollout in the PropertyManager. As a result, you can rotate a component freely about any axis by dragging the component. Select the component to be rotated in the graphics area and then drag the cursor about its degree of freedom by pressing and holding the left mouse button. The selected component starts rotating about the axis of rotation. Once you have rotated the component, release the left mouse button. All the options in the **Rotate Component PropertyManager** are the same as those discussed earlier while moving the component by using the **Move Component PropertyManager**.

Working with SmartMates

SmartMates is a smart method of applying standard mates such as coincident, parallel, and perpendicular between the components of an assembly. By using this method, you can save time and the designing process becomes faster. The SmartMates method of applying mates can be invoked by clicking on the **SmartMates** button in the **Move Component PropertyManager**. Note that as soon as you click on the **SmartMates**

button, the **SmartMates PropertyManager** appears, see Figure 12.73. You can invoke the **Move Component PropertyManager** by clicking on the **Move Components** tool in the **Assembly CommandManager**.

Figure 12.73



Once the **SmartMates PropertyManager** has been invoked, double-click on an entity of the component for applying a mate. The selected entity gets highlighted and the component becomes transparent in the graphics area, see Figure 12.74. Next, click on an entity of the other component of the assembly for applying the mate. The **Mate** Pop-up toolbar appears with the most suitable mate tool activated in it, by default. Also, the preview appears such that the most suitable mate is applied between the selected entities of the components, see Figure 12.75. You can apply mate by clicking the respective tool in the Pop-up tool. Next, click on the green tick mark in the toolbar. The selected mate is applied between the entities of the components.

Figure 12.74

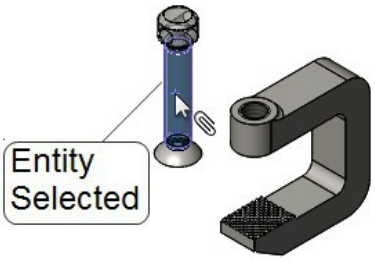
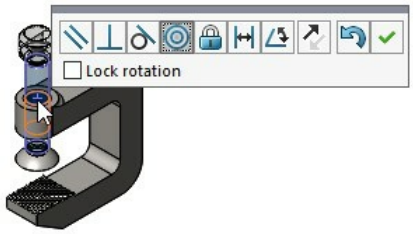


Figure 12.75



Note:

You can also press the ALT key and then drag a component toward the other component for applying a standard mate without invoking the ***SmartMates PropertyManager****.*

Tutorial 1

Create the assembly shown in Figure 12.76. For different views and dimensions of the components of the assembly, see Figures 12.77 through 12.83.

Figure 12.76

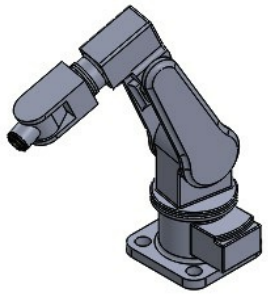


Figure 12.77

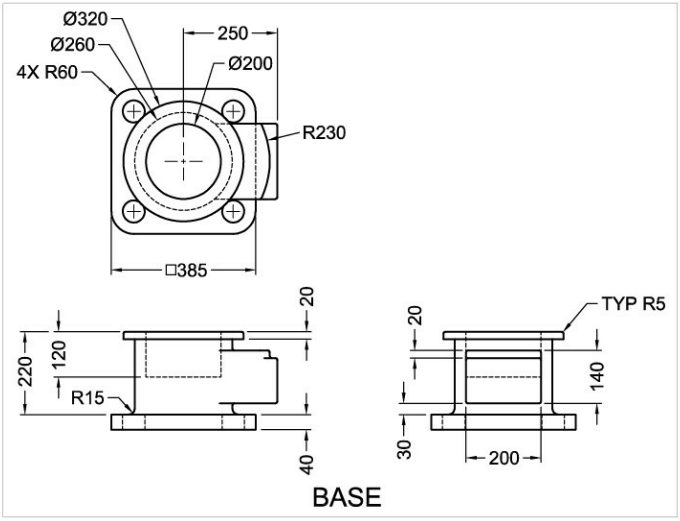


Figure 12.78

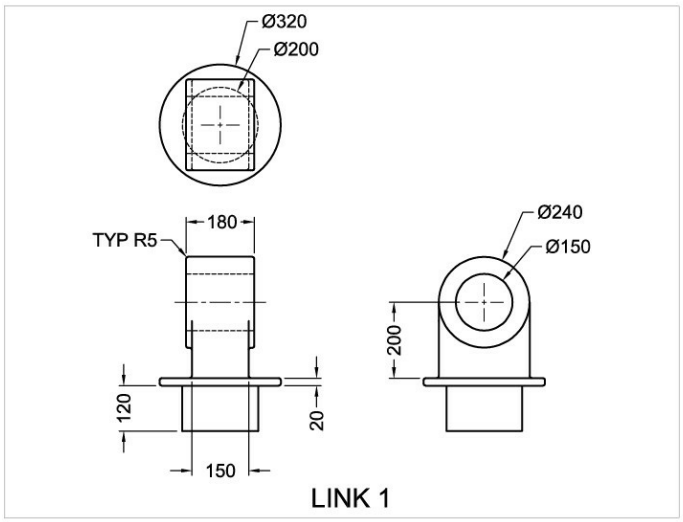


Figure 12.79

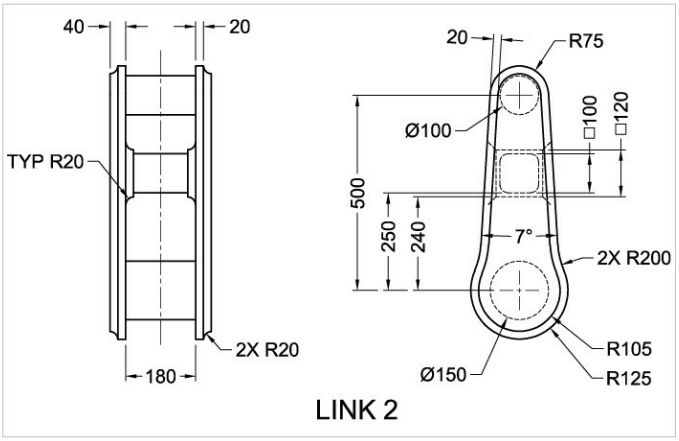


Figure 12.80

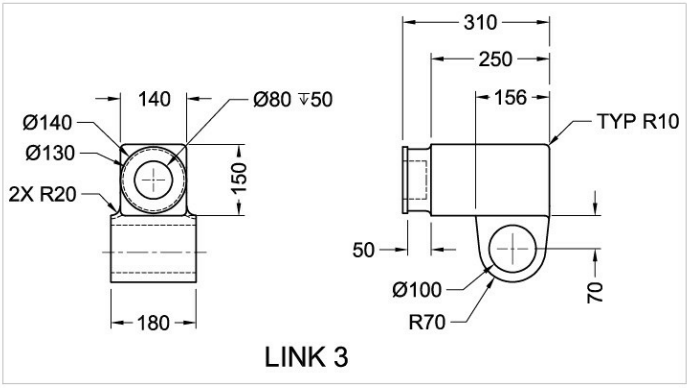


Figure 12.81

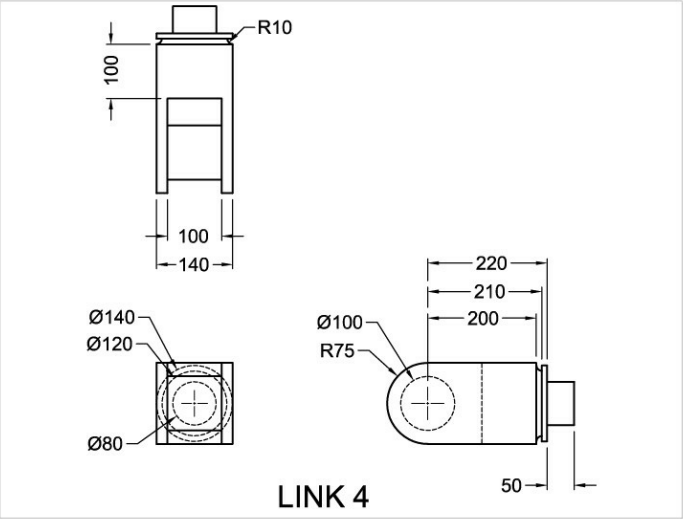


Figure 12.82

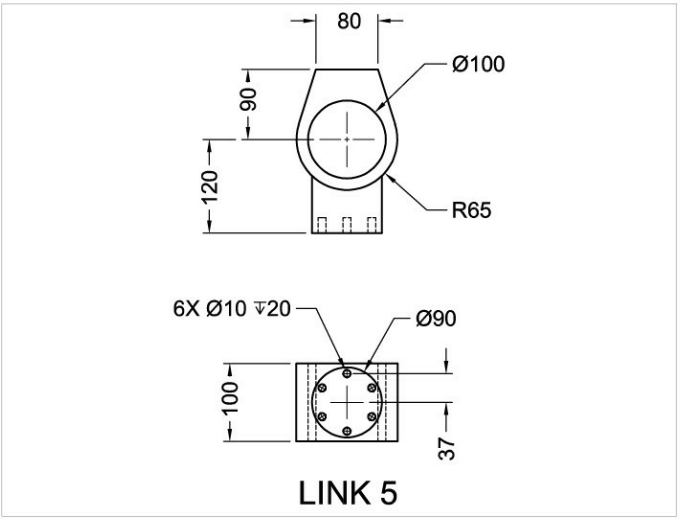
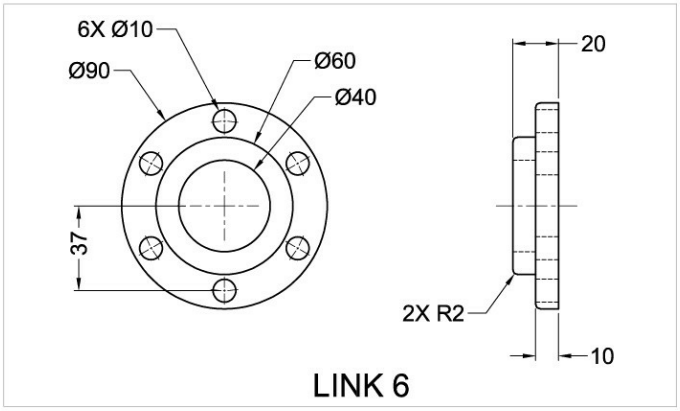


Figure 12.83



Section 1: Starting SOLIDWORKS and Creating Assembly Components

In this section, you will create all the components of the assembly in the Part modeling environment one by one.

1. Start SOLIDWORKS by double-clicking on the SOLIDWORKS icon on your desktop.

2. Create all components of the assembly one by one in the Part modeling environment. Refer to Figures 12.77 through 12.83 for the dimensions of each component. After creating components, save them in the *Tutorial 1* folder of the *Chapter 12* folder. You need to create these folders in the *SOLIDWORKS* folder.

Note:

You can also download all the components of the assembly from ***www.cadartifex.com****. If you are a first time user, you need to register yourself to access the online resources.*

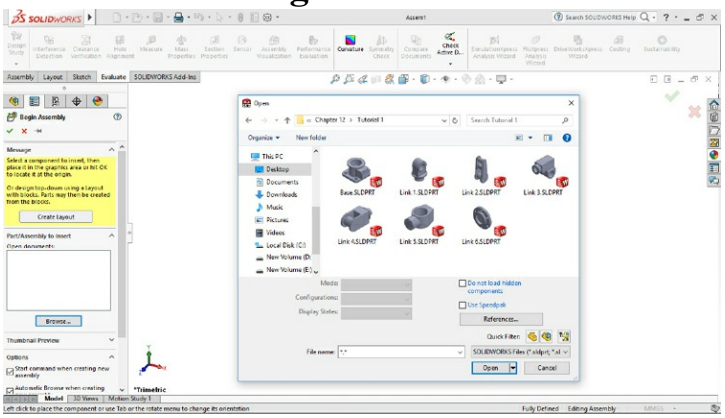
Section 2: Invoking the Assembly Environment

After creating all the components, you need to assemble them in the Assembly environment.

1. Click on the **New** tool in the **Standard** toolbar. The **New SOLIDWORKS Document** dialog box appears.

2. Click on the **Assembly** button and then click on the **OK** button in the dialog box. The Assembly environment is invoked and the **Open** dialog box appears along with the **Begin Assembly PropertyManager**, see Figure 12.84.

Figure 12.84



Note:

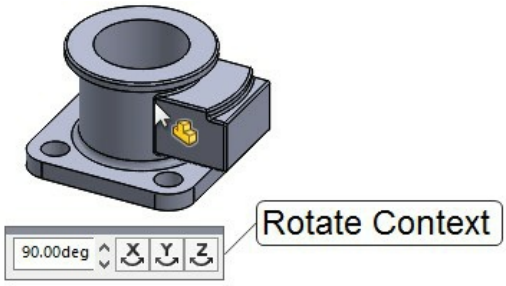
On invoking the Assembly environment, the ***Open*** *dialog box appears, automatically along with the* ***Begin Assembly PropertyManager****, if none of the components are opened in the current session of SOLIDWORKS. If the* ***Open*** *dialog box does not appear, then click on the* ***Browse*** *button in the* ***Begin Assembly PropertyManager*** *to open it.*

Section 3: Inserting the First Component

1. Browse to the location where all the components of the assembly are saved (*\SOLIDWORKS\Chapter 12\Tutorial 1*) by using the **Open** dialog box.

2. Select the *Base* component and then click on the **Open** button in the dialog box. The selected component is attached to the cursor, see Figure 12.85. Also, the **Rotate Context** toolbar appears in the graphics area, see Figure 12.85.

Figure 12.85



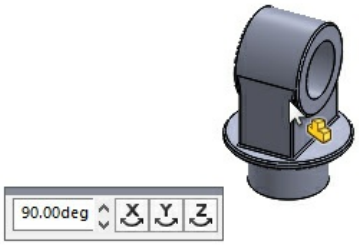
3. Click anywhere in the graphics area, the attached component moves toward the origin of the assembly and becomes the fixed component.

Section 4: Inserting the Second Component

1. Click on the **Insert Components** tool in the **Assembly CommandManager**. The **Open** dialog box appears, automatically along with the **Insert Component PropertyManager**. Note that the **Open** dialog box appears only if none of the components are opened in the current session of SOLIDWORKS. If the **Open** dialog box does not appear, automatically then click on the **Browse** button in the **Insert Component PropertyManager** to open the **Open** dialog box.

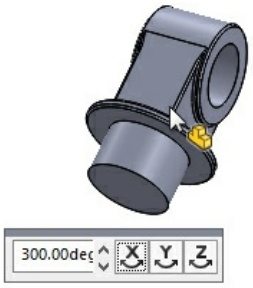
2. Browse to the location where all the components of the assembly are saved and then select the *LINK 1* component. Next, click on the **Open** button in the dialog box. The selected component is attached to the cursor, see Figure 12.86. Also, the **Rotate Context** toolbar appears in the graphics area, see Figure 12.86.

Figure 12.86



3. Enter **300** in the **Angle** field of the **Rotate Context** toolbar as the angle value to rotate the component. Next, click on the **X** tool in the **Rotate Context** toolbar. The component is rotated at 300-degree about the X axis in the graphics area, see Figure 12.87.

Figure 12.87



4. Click anywhere in the graphics area to specify the placement point for the second component (*LINK 1*). The component (*LINK 1*) is placed in the specified location, see Figure 12.88. Make sure that you specify the placement point such that the inserted component does not intersect the first component of the assembly.