Moving and rotating faces
Moving and rotating faces
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SOLID EDGE
VELOCITY SERIES

...with Synchronous Technology
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Welcome to self paced training for Solid Edge. This course is designed to educate you in the use of Solid Edge. The course is self-paced and contains instruction followed by activities.

**Solid Edge self-paced courses**
- **spse01510**—Sketching
- **spse01515**—Constructing base features
- **spse01520**—Moving and rotating faces
- **spse01525**—Working with face relationships
- **spse01530**—Constructing treatment features
- **spse01535**—Constructing procedural features
- **spse01536**—Modeling synchronous and ordered features
- **spse01540**—Modeling assemblies
- **spse01541**—Explode-Render-Animate
- **spse01545**—Creating detailed drawings
- **spse01546**—Sheet metal design
- **spse01550**—Practicing your skills with projects
- **spse01560**—Modeling a Part Using Surfaces
- **spse01610**—Solid Edge frame design
- **spse01640**—Assembly patterning
- **spse01645**—Assembly systems libraries
- **spse01650**—Working with large assemblies
- **spse01655**—Revising assemblies
- **spse01660**—Assembly reports
- **spse01665**—Replacing parts in an assembly
Lesson 1  

Introduction

- spse01670—Designing in the context of an assembly
- spse01675—Assembly features
- spse01680—Inspecting assemblies
- spse01685—Alternate assemblies
- spse01690—Virtual components in assemblies
- spse01695—XpresRoute (tubing)
- spse01696—Creating a Wire Harness with Harness Design
- spse01424—Working with Solid Edge Embedded Client

Start with the tutorials

Self-paced training begins where tutorials end. Tutorials are the quickest way for you to become familiar with the basics of using Solid Edge. If you do not have any experience with Solid Edge, please start by working through the tutorials for basic part modeling and editing before starting this self-paced training.
Lesson

2 Part modification by moving and rotating faces and planes

Overview

A synchronous solid model is defined as a set of connected facial topology that encompasses a volume. A synchronous solid model is modified by manipulating the facial topology. In this course, you will learn to modify a synchronous model by moving and rotating facial topology.

• Synchronous model faces and reference planes can be moved or rotated.

• When a face is selected, QuickBar displays the commands available for the selected face.

• **Move (1)** is the default command.

• **Move** includes both a linear direction movement and a rotational movement.

Note

**Extrude (2)** and **Revolve (3)** commands are covered in the Constructing base features course.

**Relate (4)** command is covered in the Working with face relationships course.
Lesson

3  *Moving synchronous faces*

When a synchronous face or reference plane is selected, a default graphic handle (1) appears at the select point. If the handle origin is selected, a different graphic handle (2) appears with more move options. Click the primary axis, secondary axis, or torus to start the Move command.
Graphic handle (3D steering wheel)

(1) secondary knob
(2) secondary axis
(3) cardinal points
(4) torus
(5) primary knob
(6) primary axis
(7) origin
(8) tool plane

Reorient the steering wheel

To learn how to use the 2D steering wheel, see the 2D steering wheel overview.

Swap the primary and secondary axes

1. Hold the Shift key down.
2. Click the steering wheel plane.

Change the direction of the primary axis at 90° increments
- Click a cardinal point on the steering wheel torus.

Change the direction of the primary axis at a user-defined angle
1. Hold the Shift key down.
2. Click the primary axis knob.
3. Move the cursor to define the angle or type an angular value in the dynamic edit box.
Lesson 3  Moving synchronous faces

4. Press the Tab key.

Change the direction of the primary axis using a geometric keypoint
1. Click the primary axis knob.
2. Move the cursor over the target keypoint and then click.

Change the direction of the secondary axis
1. Click the secondary axis knob.
2. As the cursor moves, the secondary axis automatically locks onto a 90° increment in either direction. Click to apply the new direction.
Change the secondary axis direction at a user-defined angle

1. Hold the Shift key down.

2. Click the secondary axis knob.

3. Move the cursor to define the angle or type an angular value in the dynamic edit box.

4. Press the Tab key.

Maintain a steering wheel orientation at a different location

- Hold the Shift key down and drag the steering wheel origin to the new location. If you drag the steering wheel origin near an edge midpoint, the origin snaps to the midpoint. Click to position the steering wheel at the midpoint or continue dragging the origin to another location.

Activity: Reorient the steering wheel

The activity guides you through the process of reorienting the steering wheel. The steering wheel orientation controls the movement direction of selected geometry during a synchronous operation.

Turn to Appendix J for the activity.
Lesson 3  

Moving synchronous faces

Moving a face

You can move a face in the following ways:

- Move a face in a direction along the primary axis or secondary axis by selecting either axis.
- Move a face freely along a plane where the graphic handle is connected by clicking the plane on the handle.
- Set the direction of the primary axis by dragging the handle origin to an edge or vertex. The primary knob also locks onto the edge to define the direction.
- Reposition the primary knob to change the direction of the primary axis.
- Reposition the secondary axis direction in 90° increments by selecting one of three cardinal points.
- The origin is the move from point. The origin can be moved prior to a move.

Rotating a face

Rotate a face by positioning the steering wheel secondary axis on an edge. The secondary axis becomes the axis of revolution. Select the torus to begin dynamic rotation or type a rotation angle in the dynamic input box.

Note

You can lock and drag a graphic handle orientation. Hold the Shift key, click the handle origin, and drag it to a desired edge or vertex.

Move face workflow

Single face move

1. Using the Select tool, select a face. The 3D steering wheel appears on the selected face. Initially you get the primary axis only. Click the 3D steering wheel origin to display entire steering wheel.

2. QuickBar appears with the available operations that can be performed on the selected face. Move is the default operation and thus does not need to be selected.

3. Click the primary axis on the handle to move the face in or out in a direction normal to the face.

4. Define the move to location by one of the following methods:
   - Dynamically drag the face to a new location and then click.
• Click a keypoint location. Choose the keypoint type on the Move QuickBar drop list.

![Move QuickBar](image)

• Type in a ± distance in the dynamic input box.

5. Press Esc key to end move.

**Note**

Workflow is the same for multiple faces in a select set.

**Single face rotate**

1. Using the Select tool, select a face. The 3D steering wheel is displayed on the selected face. Initially you get the primary axis only. Click the 3D steering wheel origin to display entire steering wheel.

2. Click and drag the origin of the steering wheel to an edge to rotate about.

3. Make sure the secondary axis of the steering wheel lies on the edge to rotate about. Click and drag the secondary knob to position if necessary.

4. Click the torus on the handle to rotate the face. Dynamically rotate the face by moving the cursor or by typing in a ± angle in the dynamic edit input box.

5. Press Esc key to end rotate.

**Note**

Workflow is the same for multiple faces in a select set.

**Activity: Move and rotate faces**

This activity guides you through a move and a rotate face process to reinforce the use of the 3D steering wheel.

Change the shape of part (1) to a modified part (2).
Activity: Copying a face and using keypoints to define movement

This activity guides you through the process of copying a face and using other geometry to define the movement direction and distance. The lower hole will be copied and positioned at the same angle and distance as the upper holes.
Turn to Appendix B for the activity.
Lesson

4 Selecting faces

Select faces using the Select tool.
A collection of selected faces to perform an action on is referred to as a select set.

Face selection methods

- Select and deselect faces manually (one face at a time).
- Select and deselect faces with the assistance of a selection manager.
  The selection manager uses the topological and attribute data of the face selected to add faces to a select set.

Selection mode

A selection mode symbol is located in the upper-right corner of the graphics window. Press the spacebar to change the select mode. The select mode selection is also available on the Home tab in the Selection group.
Normal mode

Normal mode is the default selection mode. Normal mode is a single selection. Select a face and the steering wheel displays on that face. Select another face and the steering wheel moves to that face. The face previously selected is deselected. Only one face selected per click.

Add/remove mode

The add/remove selection mode is used to build a select set. In the normal mode, select a face and then press the `spacebar` to switch to the add/remove mode. Each face selected in this mode is added to the select set. If a face is selected that has already been selected, it is deselected. The graphic handle remains on the first face selected. Both selected and unselected faces highlight as the cursor moves over them.

Add mode

The add mode only adds faces to the select set. Only unselected faces highlight as the cursor moves over faces. To set the mode to add, cycle through the select modes by pressing `spacebar`.

Remove mode

The remove mode only removes (deselects) faces from the select set. Only selected faces highlight as the cursor moves over faces. To set the mode to remove, cycle through the select modes by pressing `spacebar`.

Selection manager mode

When a face is selected, the selection manager is attached to cursor. Click to open Selection Manager.

Selection manager

Selection Manager is used to add or remove items from a select set using the topological and attribute data of a selected object.
A green dot displays when the cursor moves over a selected face.

As the cursor touches the green dot, it turns red.

Clicking the red dot displays the Selection Manager menu.

The topological relations relate only to the face where the green dot is selected.

The topological relations listed in the Select Manager menu are determined by the type of face selected (planar, non-planar, cylindrical, partial cylindrical).

You can also switch to a Selection Manager mode. On the Home tab, in the Select group, in the Select drop list, choose the Select Manager Mode command. You can also start the Select Manager mode by pressing Shift + spacebar. When in this mode Select Manager is active. Click on a face in the select set to display the select manager. To end the Select Manager mode, press the spacebar.

Note

If the green dot does not display, it is hidden by the graphic handle. You can either move the graphic handle and then the green dot can be seen or click the Selection Manager mode on the Select tool command drop list.

**Selection manager options**

The Selection Manager shortcut menu is available when valid elements are selected.

To display the Selection Manager shortcut menu, position the cursor over the green dot in the graphics window and left-click.

Drag the shortcut menu to a new location to better visualize the model.

**Connected**

Add faces which are connected to the focus element. Use the flyout options to specify what type of connected elements to add.

- **Connected** – Adds all faces which are connected to the focus element.
- **Interior Faces** – Adds all interior faces which are connected to the focus element.
Lesson 4  
*Selecting faces*

- **Exterior Faces** – Adds all exterior faces which are connected to the focus element.

**Related Items**

Adds elements that have a persistent relationship to the focus element.

**Sets**

Adds faces which are part of the same face set as the focus element.

**Recognize**

Adds all faces which are part of the same feature as the focus element. Use the flyout options to specify what feature type is recognized.

- **Feature** – Adds all faces which as part of the same feature as the focus element.
- **Rib/Boss** – Adds all faces which are part of the same rib/boss as the focus element.
- **Cutout** – Adds all faces which are part of the same cutout as the focus element.

**Parallel**

Add planar faces or reference planes which are parallel to the focus element. Use the flyout options to specify what type of parallel faces to add.

- **Faces** – Adds all planes which are parallel to the focus element, regardless of whether they are aligned or opposing. This option supports the Use Box Selection option.
- **Aligned** – Adds all planes which are parallel and face the same direction as the focus element. This option supports the Use Box Selection option.
- **Opposing** – Adds all planes which are parallel and face the opposite direction as the focus element. This option supports the Use Box Selection option.

**Perpendicular**

Adds all planes which are perpendicular to the focus element. This option supports the Use Box Selection option.

**Coplanar**

Adds all planes which are coplanar to the focus element. This option supports the Use Box Selection option.

**Concentric**

Adds all faces that are concentric to the focus element. This option is available only on faces that are cylinders, cones, and torii, both partial and full. This option supports the Use Box Selection option.
Blend Chain
Adds faces which are part of the same blend chain as the focus element to the select set.

Equal Radius
Adds faces which have a radius equal to the focus element to the select set. This option is available only on faces that are partial cylinders, partial cones, and partial tori. This option supports the Use Box Selection option.

Equal Diameter
Adds faces which have a diameter equal to the focus face to the select set. This option is available only on faces that are full cylinders, full cones, and full tori. This option supports the Use Box Selection option.

Tangent Faces
Adds faces which are tangent to the focus element.

Tangent Chain
Adds faces which are part of the same blend chain or tangent to the same blend chain as the focus element.

Symmetric About
Adds faces which are symmetric to the focus element about the same reference plane type specified. Use the flyout options to specify what type of reference plane to use as the symmetry plane.

- **Base XY Plane** – Adds faces which are symmetric to the focus element about the base XY plane.
- **Base ZX Plane** – Adds faces which are symmetric to the focus element about the base ZX plane.
- **Base YZ Plane** – Adds faces which are symmetric to the focus element about the base YZ plane.
- **Local Plane** – Adds faces which are symmetric to the focus element about a reference plane you select.

Axis
Adds faces which have an axis that is parallel or perpendicular to the focus element. This option is available only on faces that are cylinders, cones, and tori, both partial and full. Use the flyout to specify whether the axis must be parallel or perpendicular.

- **Parallel**–Adds faces which have an axis that is parallel to the focus element.
- **Perpendicular**–Adds faces which have an axis that is perpendicular to the focus element.
Lesson 4    Selecting faces

Use Box Selection

Defines a 3D box in the graphic window to add or remove items to the select set. When using box selection, the elements which are inside or overlapping the 3D box are included in the selection. This option is available only for a specific shortcut menu options.

When using the Use Box Selection option, there are two key in options to help define the location or area the selection box covers. The first option for box select is to define an area box. Use the C key to switch between a center or corner area box definition. Once the area of the box is defined, define the depth of the box. Use the S key to define a symmetric or non-symmetric box.

Use the Selection Manager shortcut menu as many times as required to build the select set.

Deselect Items

Deselects elements which match the focus element criteria when set.

Set the Deselect Items option and then define criteria to remove items from the select set.
Select tool menu

The Select Tool shortcut menu is available when valid elements are selected.

To display the Select Tool shortcut menu, position the cursor over the green dot in the graphics window and right-click.

Drag the shortcut menu to a new location to better visualize the model.

Select menu options

Deselect
Removes the focus element from the select set.

Clear Selection
Removes all elements from the select set.

3D Box Select
Specifies that you want to define a 3D box in the graphics window to add items to the select set. When using box selection, the elements which are inside or overlapping the 3D box are included in the selection.

Activity: Using the Selection Manager

Activity guides you through the process of using the selection manager.

Turn to Appendix C for the activity.

Activity: Modifying a part by moving select sets

This activity demonstrates how to move multiple faces in a single operation. You will modify part (1) to the shape of (2).
Lesson 4  
Selecting faces

Turn to Appendix D for the activity.
Lesson

5  QuickBar options for moving faces

You can control the results of the Move command with options that control the interaction of the select set and the rest of the model.

By setting these options, the resultant transformation can be altered within the command.

The options are Connected faces, Copy, Detach, and Precedence.

Connected face options

- **Extend/Trim**
  
  Default option. Selected face moves by extending and trimming the adjacent faces.

- **Tip**
  
  Selected face is rigid. Adjacent faces change to meet the movement of the rigid selected face.

- **Lift**
  
  Selected face is rigid. Adjacent connected faces are not affected. The selected face moves in a direction normal to the face to either add or remove material.
Lesson 5  
QuickBar options for moving faces

Copy

The *copy* option creates a copy of the faces in the select set. The faces are collected into a face set feature. The face set feature can be moved or rotated. This option is similar to a *copy and paste* operation. The original selected faces are unchanged.

Activity: Copying and attaching a feature (method 1)

The activity guides you through the process of copying a cutout feature and then attaching the copied feature at a new location on the model.

Turn to **Appendix E** for the activity.

Activity: Copying and attaching a feature (method 2)

This activity has the same goal as method 1, but uses a different approach. Turn to **Appendix F** for the activity.
Activity: Copying, rotating, and attaching a feature to a new location

This activity guides you through the process of copying a feature, aligning the feature to an angled face, and then positioning the feature on the model. Two methods are used in the activity.

Turn to Appendix H for the activity.

**Detach**

The detach option removes the select set from the part body. The removed select set can be moved or rotated. This option is similar to a cut and paste operation.
Lesson 5  
*QuickBar options for moving faces*

**Activity: Detaching and attaching a feature**

The activity guides you through the process of detaching an extruded feature and then attaching the copied feature at a new location on the model.

Turn to *Appendix G* for the activity.

**Precedence**

The precedence option is used to set which face(s) have priority during a synchronous move operation.

- **Select Set Priority**

  Selected and other moving faces have priority over non-moving faces.

- **Model Priority**

  Non-moving faces have priority over moving faces.
Lesson

6 Working with Live Sections

You use the Live Section command to create a 2D cross-section on a plane through a 3D part. For example, you can select one of the principal planes on the base coordinate system as the plane for a live section.

Live sections can make it easier to visualize and edit certain types of parts, such as parts that contain revolved features. You can then edit the 2D elements of the live section to modify 3D model geometry.
Lesson 6  Working with Live Sections

Creating live sections

You can select a planar face, reference plane, or principal plane on a coordinate system as the plane for the live section. When you select the plane, a live section is created, similar to a section view in a drawing. When the live section passes through a procedural feature, such as a hole, an edge set is created.

An entry for the live section is added to the Live Sections collector in PathFinder.

Editing live sections

You edit a live section using the Select tool and the 2D steering wheel editing handle. You can edit individual elements or you can edit the entire live section.

Editing 2D elements in a live section to modify the 3D model

When you select a 2D element in a live section, the 2D steering wheel editing tool is displayed. You can use the handles on the 2D steering wheel to move or rotate the live section element to modify 3D model geometry. If the live section element you select is an edge set created from a procedural feature, such as a hole, the editing handle for the procedural feature is also displayed.

You can also place PMI dimensions on the 2D elements of a live section and then edit the dimension value to modify the model.

Note

When moving a live section element using the 2D steering wheel edit tool or a PMI dimension, the current settings in Live Rules are used to control the edit behavior.

Editing the entire live section

You can select the entire live section using PathFinder or QuickPick. You can then use the steering wheel to move or rotate the entire live section. When you move or rotate the entire live section, 3D model geometry is not modified. The live section recalculates at its new position. This can be useful when you have modified the 3D model using other methods, such that the live section is no longer positioned where you want it.

Displaying live sections

You can use the check box adjacent to a live section entry in PathFinder to show or hide a live section in the graphics window. You can use the check box adjacent to the Live Sections collector to display or hide all the live sections.

You can use the Live Section Colors section on the Colors page of the Solid Edge Options dialog box to specify the colors you want to use for the edges, centerlines, and regions for live sections.
Model editing and live section update

The live section automatically updates when you add or remove features, or directly edit the 3D model. For example, if you add a pattern of holes to a synchronous model, the live section automatically updates.

Live sections in an assembly

You can edit a 2D element on a live section to modify a part in the context of an assembly. You can use the keypoints on adjacent parts to modify the live section element with respect to other parts in the assembly.

You can use commands on the shortcut menu to control the display of live sections on a selected part.

Activity: Live Section

This activity guides you through the process of creating a live section through a model. The model is modified by manipulating the live section edges.

Turn to Appendix I for the activity.
A Activity: Moving and rotating faces

Open activity file
- Open move_01.par.

Move a face

Move the back face of the boss a distance defined by a vertex on the back face of the lower base.
- Select the face shown. Use QuickPick if necessary.

- Click the primary axis to start the move command. By clicking the primary axis, the direction vector for the move is defined. All that is needed to complete the move is a distance to move.

The selected face is connected to the cursor and moves dynamically as the cursor moves.
Activity: Moving and rotating faces

- Use a keypoint locate to define the move to distance. On the Move QuickBar, select the All keypoints option.

- Move the cursor over the corner shown and click when the endpoint appears.

- Press the Esc key to end the move command.
Activity: Moving and rotating faces

Move faces

- Move the side faces on the boss a distance defined by a vertex on the side face of the lower base.

![Image of moving faces](image1)

Rotate a face

- Select the angled face.

![Image of rotating face](image2)
Activity: Moving and rotating faces

- To rotate the selected face, a rotation axis needs to be defined. Drag the steering wheel origin to the edge shown. The secondary axis must lie on an edge that the face will rotate about.

- Click the steering wheel torus to start the rotation. As the cursor moves, the rotation angle tracks with the cursor. Type 35 in the input box to define the rotation angle.

- Press the Esc key to end the command.

- This ends the activity. Exit the file and do not save.

Summary

In this activity you learned how to move and rotate faces. You can define distances to move by dragging and clicking, typing in a distance, or by using keypoints. To rotate a face, position the secondary axis of the steering wheel on an edge to rotate about. Click the torus and move the cursor to define the rotation angle or type a rotation angle in the edit box.
Activity: Copying a face and using keypoints to define movement

Open activity file
- Open move_02.par.

Select the hole to copy
- Select the cylindrical face shown.
- In QuickBar, click the copy option.
Define the move from point

At this point, the steering wheel origin is at the center of the selected cylindrical face. Move the origin to the top left hole.

- Click the steering wheel origin and then move the cursor to the upper left hole. Click when the origin locks to the center of the hole. You may have to zoom in if you have trouble locking to the center of the hole.
Define the move direction

- Click the primary knob shown. This controls the primary axis direction.

- Move the cursor over the cylindrical face shown, and click when the center point symbol appears.
Activity: Copying a face and using keypoints to define movement

- Notice that the primary axis now points to the center of the hole. Direction definition is complete.

Define the move distance

- Click the primary axis to start the Move command.
Activity: Copying a face and using keypoints to define movement

- Make sure the keypoints option in QuickBar is set to All or Center Point.
Activity: Copying a face and using keypoints to define movement

- Click the center of the hole shown. This defines the move distance. Click again to end the command.

Verify move distance

- Measure the copied distance. On the Inspect tab, in the 3D Measure group, choose the Measure Distance command.

- Measure the distance between the top two holes. Click when the center point highlights. Notice the minimum distance and then click Reset in the command bar. The distance is 14 mm.
Activity: Copying a face and using keypoints to define movement

- Measure the distance between the lower two holes. The distance between the holes should also be 14 mm.
- This ends the activity. Exit the file and do not save.

Summary

In this activity you learned how to use the 3D steering wheel to control a move or copy operation. You learned how to redefine an origin point (move from point) and how to modify the direction of a move. You used face keypoints to define the move/copy direction and distance.
C Activity: Using the Selection Manager

Open activity file
- Open select_b.par.

Select all rounds of equal radius
Use selection manager to select all rounds of equal radius and change the radius value for all the rounds selected.
- Select the round shown below.
- Move cursor over round to display the green dot.

Note
If the green dot does not display, move the steering wheel or zoom in.

- Click the green dot to display the selection manager. Make sure Use Box Selection is not checked.
Activity: Using the Selection Manager

- On the Selection Manager, click the **Equal Radius** option. Notice that all rounds that have the same radius (1.524) are added to the select set.

Change the round radius

- Select the PMI dimension on the round.
In the dimension box, type 2 and then press the **Enter** key. Press **Esc** to clear the select set. All rounds in the select set are now equal to 2.

**Use the selection box**

Add rounds to a select set using a selection box.

- Select the round shown.

- Bring up the Selection Manager and then click **Use Selection Box**.

- In Selection Manager, click **Equal Radius**.
Activity: Using the Selection Manager

Define selection box area

- The first step in defining the selection box is to define the area. Typing a C changes the area definition from a corner start point (1) to an area center start point (2). The start point is the point where the face was selected.
Use the center option and define an area as shown.

Define select box depth

- The next step is to define the select box depth. Typing an S changes the definition from a side definition (1) to a symmetric definition (2). Side step defines depth in either direction (1) normal to the defined area. The symmetric option defines the depth symmetric (2) about the defined area.

Define a symmetric depth as shown.

Note

You can rotate the view to better view the positioning of the area and the depth of the selection box.

Change radius of select set
Activity: Using the Selection Manager

- Edit the radius to 3.

- Press the **Esc** key to clear the select set.

- This completes the activity.

Summary

In this activity you learned how to use the Selection Manager to control the selection process. With practice you will master the use of the box selection.
Activity: Modifying a part by moving select sets

Open activity file
- Open select_a.par.

Suspend Live Rules
Since the Live Rules topic has not been covered, turn off live rules.
- A face must be selected for Live Rules to appear. Click on any face.
- On the Live Rules panel, click the Suspend Live Rules (1) check box.

Select feature to move
Move protrusion feature to other end of part.
- To select the feature to move, first select the face shown.

At this point, only the selected face would move.
Activity: Modifying a part by moving select sets

- Move the cursor over the face until the green dot displays.

- Move the cursor over the green dot and the red arrow displays. Click the red arrow to display the **Selection Manager**.

- In the Selection Manager, click **Sets**. This will find any sets that contain the selected face.

- QuickPick displays the sets found. Click the Protrusion entry displayed in QuickPick.

- The feature is selected and will participate in the move operation.
Move the feature

• Click the primary axis on the steering wheel and move the feature to the other side of the channel-shaped feature.

• Move the feature to the approximate location and click. The move from point is the origin on the graphic handle.

• Move complete. Press the Esc key to clear the select set.
Select the channel-shaped feature

- Select the face shown.

- In the Selection Manager, click Recognize → Rib/Boss.

The **Sets** option would work here also.
Move the channel-shaped feature

- Click the primary axis on the steering wheel and move the select set to the edge of the part. Use a keypoint on the edge of the part to define distance to move. The keypoint option is on QuickBar (3).

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Extend the legs of the channel-shaped feature

- Select the two faces shown.
Activity: Modifying a part by moving select sets

- Move the faces to the end of the protrusion feature as shown. Use a keypoint to define the distance.

Move top face of channel-shaped feature

- Select the top face.
Activity: Modifying a part by moving select sets

- Move the top face to the top of the protrusion feature.

Move protrusion feature to end of part

- Select the protrusion feature.
Activity: Modifying a part by moving select sets

- Move the select set to end of part.

- This ends the activity. Exit the file and do not save.

Summary

In this activity you learned how to create select sets for a move operation. Live Rules were suspended for the activity. After the live rules topic is covered, you will be able to perform the same part modifications in other ways.
Activity: Copying and attaching a feature (method 1)

Open activity file
- Open copy_a.par.

Select the feature to copy
- Select the cutout feature by clicking Cutout1 in PathFinder.
- Position the 3D steering wheel as shown.
Activity: Copying and attaching a feature (method 1)

Suspend Live Rules

Live Rules is covered in the Working with face relationships self-paced course. Suspend the Live Rules settings while moving the cutout feature. This ensures no other faces in the model participate in the move.

- Click the Suspend Live Rules (1) check box.

Set the copy option and move the feature

- On QuickBar, click the Copy option.
• Move the copied feature to the face shown in orange.

To begin the move, click the secondary axis shown. The move origin point is where the origin of the steering wheel is located.

Define the move distance and direction

• Move the feature to the edge of the part using a keypoint. On QuickBar, click the keypoints drop list arrow and select Endpoint.
Activity: Copying and attaching a feature (method 1)

- Select the keypoint location shown.

![Image of keypoint location]

**Change move direction**

- The primary axis should point down. If your steering wheel position is different, change the move direction by clicking the cardinal point (3) on the steering wheel and then click the primary axis.

![Image of move direction change]
Activity: Copying and attaching a feature (method 1)

- Select the keypoint.

**Move by keying in a distance**

- Click the secondary axis.
Activity: Copying and attaching a feature (method 1)

- Type **10** in dynamic edit box.
Attach the copied feature

The copied feature is now positioned but is detached from the model.

- Right-click in the part window and click Attach.

Summary

In this activity you learned how to copy a feature and then position the copied feature. There are other methods available to move the copied feature to a location other than what was shown in this activity.
F Activity: Copying and attaching a feature (method 2)

Open activity file

- Open copy_b.par.

Suspend Live Rules

Live Rules is covered in the Working with geometric relationships self-paced course. Suspend the Live Rules settings while moving the cutout feature. This ensures no other faces in the model participate in the move.

- Click the Suspend Live Rules (1) check box.

Select the feature to copy

- Select the cutout feature by clicking Cutout1 in PathFinder.
Activity: Copying and attaching a feature (method 2)

- Position the 3D steering wheel as shown.

Set the copy option and move the feature

- On QuickBar, click the Copy option.

- Move the copied feature to the face shown in orange.
Relocate the steering wheel origin

In this activity, the steering wheel plane is used to move the copied feature instead of the secondary axis used in the method 1 activity.

- To relocate the steering wheel origin, hold the Shift key down, and drag the steering wheel origin to the midpoint of the face edge (4). As you drag along the face edge (4), notice that the steering wheel origin jumps to the midpoint of edge (4). When you hold the Shift key down to move the steering wheel origin, the steering wheel orientation remains fixed.

The move is from the midpoint of edge (4) to the midpoint of edge (5).

Move the copied feature using the steering wheel plane

- Click the steering wheel tool plane.
Activity: Copying and attaching a feature (method 2)

- Drag the cursor over the edge shown and click when the midpoint symbol displays. You may have to turn on the midpoint option on QuickBar.

- Press the Esc key to end the move command.

Note
Since this copy operation was accomplished in one movement, the copied feature attaches automatically.

Summary
In this activity you learned how to copy a feature and then position the copied feature by moving the steering wheel origin and using the steering wheel plane to define the move vector.
Activity: Detaching and attaching a feature

Open activity file
- Open `detach_a.par`.

Suspend Live Rules

Live Rules is covered in the Working with geometric relationships self-paced course. Suspend the Live Rules settings while moving the feature. This ensures no other faces in the model participate in the move.

- Click the Suspend Live Rules (1) check box.
Activity: Detaching and attaching a feature

Select the feature to detach

- In PathFinder, click the feature named **Protrusion 2**.

![Diagram of a model showing a feature to detach.]

Position the steering wheel origin

- Drag the steering wheel origin to the midpoint of the edge shown.

![Diagram showing the steering wheel origin moved to the midpoint.]

Move the feature

- On QuickBar, click the **detach** option.

![Image of QuickBar with the detach option highlighted.]
Activity: Detaching and attaching a feature

- Click the steering wheel tool plane to start the move.

- Select the midpoint of the edge shown to complete the move.
Attach the feature

- Right-click in the part window and choose the Attach command.

Summary

In this activity you learned how to detach a feature, move it to a new location, and then attach the feature to the model. This process is similar to a cut and paste process.
Activity: Copying, rotating, and attaching a feature to a new location

Open activity file
- Open rotate.par.

Suspend Live Rules

Live Rules is covered in the Working with geometric relationships self-paced course. Suspend the Live Rules settings while moving the feature. This ensures no other faces in the model participate in the move.

- Click the Suspend Live Rules (1) check box.
Method one overview

You will use the copy option on QuickBar. Align the feature using the Relate command. Position the feature using the steering wheel. Copy feature (1) onto face (2). Center feature on face (2) with the feature holes aligned to the midpoint of edge (3).

Select the feature

- In PathFinder, select the feature named Protrusion 1.
- On QuickBar, click the copy option.
Move the copied feature

- Position the steering wheel as shown and click the primary axis to start the move command.

- In the dynamic edit box, type 125 and press the Enter key.
Align the feature to the angled face

- On QuickBar, choose the **Relate** command.

![Relate command](image1)

**Note**

The Relate command is covered in the Working with face relationships course. We will use the command to change the angle of the copied feature. You could use the steering wheel to rotate the feature but you need to know the angle of the face. The Relate command is an easier step.

Use the **Parallel** relationship.

![Parallel relationship](image2)

- Select the angled face.
Activity: Copying, rotating, and attaching a feature to a new location

- Click the **Accept** button on QuickBar and then press the **Esc** key.

The feature is now aligned with the angled face.

**Position the feature**

- Move the steering wheel origin to the center of one of the cylindrical faces as shown.

- Click the cardinal point shown to define the move direction.
Activity: Copying, rotating, and attaching a feature to a new location

- Click the secondary axis and then select the midpoint of the edge shown.

Note

If you cannot locate the midpoint on the edge, make sure the All keypoints option is set.
Activity: Copying, rotating, and attaching a feature to a new location

- Move the steering wheel origin to any point on the bottom of the feature.

- Click the primary axis and then select the endpoint shown.
Attach the feature

- Right-click in the part window and choose Attach.

This completes the first method of copying, aligning, and positioning a feature.
**Method two overview**

You will use the **Copy to and Paste from clipboard** commands. Use the short-cut keys Ctrl+C (copy) and Ctrl+V (paste). Align the feature using the F3 key. Position the feature using the steering wheel. Copy feature (1) onto face (4). Center feature on face (4) with the feature holes aligned to the midpoint of edge (3).

![Diagram](image)

To copy a selected feature to the clipboard, press Ctrl+C.

To paste a feature from the clipboard, press Ctrl+V.

You can also choose the commands from the **Clipboard** group.

**Select the feature to copy**

- In PathFinder, select the feature named **Protrusion 1**.

- Position the steering wheel origin at any point on the bottom of the feature. This comes into play when the feature is aligned to the target angled face. Make sure the secondary axis points in the direction shown.
Copy and paste feature

- Press **Ctrl+C** to copy the selected feature to the clipboard.
- Press **Ctrl+V** to paste the feature. The feature is attached to the cursor.
- Drag the cursor over the face shown.

- Press the **F3** key to coplanar align the steering wheel face to the angled face. This is the reason you positioned the steering wheel to a point on the bottom of the feature in an earlier step.
Activity: Copying, rotating, and attaching a feature to a new location

- Click to place the feature.

Rotate the feature

- Click the steering wheel torus.
Activity: Copying, rotating, and attaching a feature to a new location

- In the dynamic edit box, type 90 and then press the Enter key.
Activity: Copying, rotating, and attaching a feature to a new location

Center the feature on the face

- Move the steering wheel origin to the midpoint of a linear edge on the feature and click the secondary axis point shown to define the move direction.
Activity: Copying, rotating, and attaching a feature to a new location

- Click the midpoint on the edge shown.
Align the feature center to the midpoint on the edge

- Move the steering wheel origin to the center of a cylindrical face on the feature and then click the axis shown to define move direction.
Activity: Copying, rotating, and attaching a feature to a new location

- Click the midpoint of the edge.

Attach the feature to the model

- Right-click in the part window and choose Attach.

This completes the activity.
Summary

In this activity you learned how to copy, align, and position a feature. Two methods were shown to help you understand the available tools for copying geometry.
I Activity: Live Section

Open activity file

- Open live_section.par.

Create a section plane

- In the Planes group, choose the Coincident Plane command.

- Select the plane shown.
Activity: Live Section

- Move the coincident plane to the midpoint of the edge shown.

Create a Live Section

- In the Section group, choose the Live Section command.
• Select the plane created in the previous step to define the live section.

At this point you can use the steering wheel to move the live section if desired.

• Press the **Esc** key to end the live section command.
Activity: Live Section

- Notice in PathFinder that a **Live Sections** collector is created. The display of a live section can be controlled with the check box.

```
☐  ☑ Live Sections
☐  ☑ Live Section 2
```

Move a face

Instead of selecting a face to move, the edge created by the section through a face can be selected to move. Moving the edge is the same as moving the face.

- Select the edge shown and move it to observe the behavior.

Dynamically move the face but do not click. Press **Esc** to end the move. Press **Esc** again to clear the selected edge.

**Note**

The edge can take on all operations that its parent face can (for example: dimension, rotate, delete).

Modify the model shape by manipulating the live section

The model does not have to be displayed to manipulate a live section. Turn off the display of the model.
• Right-click in the part window and on the shortcut menu click **Hide All → Design Body**. Hide all reference planes too.

• Change the display to a front view. Press **Ctrl+F**.
Activity: Live Section

- Rotate the two arms on the left 15° about the center hole. Select the live section edges shown. Use a fence as shown.
- Move the steering wheel origin to the center of the hole as shown.

- Change the orientation of steering wheel. Click the cardinal point shown and then click the midpoint of the right edge.
• Click the torus. Type 15 and press the **Enter** key.

• Press the **Esc** key to end the move command.
Observe model changes

- Change to a dimetric view.

- On the shortcut menu, turn on the display of the Design Body. Notice that the model has changed to the modifications made to the live section.
Activity: Live Section

Delete a face

- Click the circular edge shown and press the Delete key.

![Delete a face][1]

Deleting the live section circular edge is the same as deleting the circular face.

Remove material to create a slot

- Turn off the design body display.

![Remove material to create a slot][2]
• Draw a sketch containing two lines and one arc. Choose the Line command and click the right section edge.
Activity: Live Section

- Align the circle center with the midpoint of the right edge.

- Select the region shown.

- On QuickBar, click (2) to set the Remove material option. Click (1) to set the Through All extent option.
• Click the origin on the handle to make sure the arrows point in both directions.

Change display to observe the changes

• Turn on the design body.
• In PathFinder, turn off the live section display.

This completes the activity.

Summary

In this activity you learned how to create a live section. The live section command creates edges where a user defined plane intersects the design body. Each live section edge represents a face in the model. You can select either the face or live section edge to modify the model.
Activity: Reorient the steering wheel

Overview
Examine the components used for reorienting the steering wheel. In this activity, a geometric feature is moved using the steering wheel. The steering wheel is oriented to define the move direction.

Open activity file
- Open steering_wheel.par.

Move geometry in the primary and secondary axes directions

Note
A move in the primary axis direction occurs on the steering wheel plane. A move in the secondary axis direction is normal to the steering wheel plane.

Note
A move occurs between the move from point to the move to point. The move from point is always the steering wheel origin. The move to point can be a keypoint, user-defined typed distance, or a drag and click location.

- In PathFinder, select the feature named feature A.
Activity: Reorient the steering wheel

- Click the primary axis. As you drag the cursor up and down, notice the feature moves in a direction defined by the primary axis.

At this point, you can drag and click to define the move distance, type a distance value in dynamic edit box, or click on a geometric keypoint.

- Press the **Esc** key to end the move.

- Click the secondary axis. As you drag the cursor, notice the feature moves in a direction defined by the secondary axis.

- Press the **Esc** key to end the move.
Change the direction of the primary axis

Note
You can change the primary axis direction in 90° increments by clicking a cardinal point.

Note
You can change the primary axis direction at a user-defined point or a drag and click point to define direction angle

- Hold the Shift key and click the primary knob.
Activity: Reorient the steering wheel

• Move feature A at a 45° direction angle. Once the angle is set, click the primary axis to start the move. The right portion of the image shows a front view to better visualize the 45° movement.

• Press the Esc key to end the move.

Move geometry in the steering wheel plane

You can perform a free move to any point on the steering wheel plane. You click the steering wheel plane and drag the selected geometry to a desired location and then click, or select a geometric keypoint.

• Select feature A.

• Click the steering wheel plane. Drag the feature around and notice that the movement is locked to the steering wheel plane.

• Press the Esc key to end the move.
Use the steering wheel torus to rotate geometry

Clicking the torus starts a rotate operation. The rotation axis is the secondary axis. You click the steering wheel torus and drag and then click to define the rotation angle. You can also type a rotation angle in the dynamic edit box.

- Select feature A.
- Click the torus and rotate feature A 90°.

- Click to end the rotate command.

Swap primary/secondary axes

You swap the primary and secondary axes by holding down the Shift key and clicking the steering wheel torus plane.

- Select feature A.
Activity: Reorient the steering wheel

- Hold down the **Shift** key and click the steering wheel plane.

![Diagram showing reorienting a steering wheel plane](image)

**Note**
This is a quick method of changing the rotation axis.

- Press the **Esc** key.

**Change the direction of the primary/secondary axes using a geometric keypoint**

You can change the direction of the primary or secondary axis by clicking the axis knob and then selecting a geometric keypoint.

- Select feature **A**.
• Reposition the move from point. Select the steering wheel origin and then drag the origin to the corner of the selected feature shown.

You want to move the feature to corner 1. Define the direction axis pointing to corner 1.

• Click the primary axis knob. Move the cursor over corner 1 and click when the endpoint displays.
Activity: Reorient the steering wheel

- Click the primary axis and notice the direction of movement.

At this point if you click the endpoint shown, the feature would move to that point.

- Press the **Esc** key twice to cancel the operation.

**Maintain a steering wheel orientation at a different location**

If you want to maintain a steering wheel orientation at a different location, you hold the **Shift** key down, click the steering wheel origin, and drag it to the desired location. If the origin is close to a keypoint, it will snap to that point. Click to position origin to that point.

- Select **feature A**.

- Hold the **Shift** key down and click the steering wheel origin.

- Drag the steering wheel origin over the model (at the corners and edge midpoints) and notice that the steering maintains the orientation.

If you repeat the same step without holding down the **Shift** key, the steering wheel orientation changes as it passes over the model edges, corners and faces.
Use the steering wheel to reorient and move a feature

Move the feature to the location shown. Make sure the feature has same orientation as shown.

• Turn off the display of feature A. In PathFinder, click the box in front of feature A.
• Turn on the display of feature B.
• Rotate feature B. Select feature B.
Activity: Reorient the steering wheel

- Click the torus, type 90 in the dynamic edit box, and then click.
Activity: Reorient the steering wheel

- Rotate the feature again to complete the orientation. Position the steering wheel origin as shown. Click the torus. Drag and type either 90 or -90 in the dynamic edit box (note the plus or minus value in the dynamic edit box to determine if 90 or -90 should be entered). Press the Tab key and then click.
Activity: Reorient the steering wheel

- Move the feature to new location. Select feature A. Position the steering wheel origin as shown (midpoint of edge). Define the primary axis direction to point to the edge midpoint on part. Click the primary axis to start the move. Move the cursor over the edge midpoint and click when the midpoint highlights. Press Esc to end the move operation.

- Try positioning the feature at other locations on the part.

Summary

In this activity you learned how to reorient the steering wheel to accomplish desired move and rotate operations.