Moving and rotating faces

Self_Paced Training

spse01520
Proprietary and restricted rights notice ................................. 7

Introduction ............................................................................. 9

Moving model faces ..................................................................... 1-1

Moving synchronous faces .......................................................... 2-1

Graphic handle (3D steering wheel) ............................................. 2-2
  Reorient the steering wheel ...................................................... 2-3
  Activity: Reorient the steering wheel ....................................... 2-5
Moving a face .............................................................................. 2-17
Rotating a face .......................................................................... 2-18
Move face workflow .................................................................... 2-19
Activity: Moving and rotating faces .......................................... 2-20
  Activity: Moving and rotating faces ....................................... 2-20
  Open activity file ....................................................................... 2-22
  Move a face ............................................................................... 2-23
  Move faces ............................................................................... 2-25
  Rotate a face ............................................................................. 2-26
  Summary ................................................................................... 2-27
Activity: Copying a face and using keypoints to define movement ... 2-27
  Activity: Copying a face and using keypoints to define movement 2-27
  Open activity file ........................................................................ 2-28
  Select the hole to copy .............................................................. 2-29
  Define the move from point .................................................... 2-30
  Define the move direction ....................................................... 2-31
  Define the move distance ....................................................... 2-33
  Verify move distance .............................................................. 2-35
  Summary ................................................................................... 2-36
Lesson review ................................................................. 2-37
Lesson summary ......................................................................... 2-38

Selecting faces ........................................................................... 3-1

Selection mode ........................................................................... 4-1

Normal mode ............................................................................. 4-2
Add/remove mode ................................................................. 4-3
Add mode .................................................................................. 4-4
Remove mode ............................................................................ 4-5
Selection Manager mode ........................................................... 4-6

Selection Manager ....................................................................... 5-1
Selection Manager options ................................................................. 5-2
Activity: Using the Selection Manager .............................................. 5-4
  Activity: Using the Selection Manager ........................................... 5-4
  Open activity file ........................................................................... 5-5
  Select all rounds of equal radius .................................................... 5-6
  Change the round radius ................................................................. 5-7
  Use the selection box ..................................................................... 5-8
  Define selection box area ............................................................... 5-9
  Define select box depth .................................................................. 5-11
  Change radius of select set ............................................................. 5-13
Summary .............................................................................................. 5-15
Activity: Modifying a part by moving select sets ................................. 5-15
  Activity: Modifying a part by moving select sets ............................ 5-15
  Open activity file ............................................................................ 5-16
  Select feature to move .................................................................... 5-17
  Move the feature ............................................................................ 5-18
  Select the channel-shaped feature .................................................. 5-19
  Move the channel-shaped feature ................................................... 5-20
  Extend the legs of the channel-shaped feature ................................. 5-21
  Move top face of channel-shaped feature ........................................ 5-22
  Move protrusion feature to end of part .......................................... 5-23
Summary .............................................................................................. 5-24

Lesson review ....................................................................................... 6-1

Lesson summary ................................................................................... 7-1

Move face command bar options ......................................................... 8-1
Connected face options ....................................................................... 8-2
Copy ..................................................................................................... 8-3
  Activity: Copying and attaching a feature (method 1) ...................... 8-3
  Activity: Copying and attaching a feature (method 2) ...................... 8-10
  Activity: Copying, rotating, and attaching a feature to a new location 8-16
Detach .................................................................................................. 8-32
  Activity: Detaching and attaching a feature .................................... 8-32
Precedence ........................................................................................... 8-38
Lesson review ....................................................................................... 8-39
Lesson summary ................................................................................... 8-40

Working with Live Sections ................................................................. 9-1
Activity: Live Section .......................................................................... 9-3
  Activity: Live Section .................................................................... 9-3
  Open activity file ............................................................................ 9-4
  Create a section plane ................................................................. 9-5
  Create a Live Section .................................................................... 9-7
  Move a face .................................................................................... 9-9
  Modify the model shape by manipulating the live section .............. 9-10
  Observe model changes ................................................................. 9-14
  Delete a face .................................................................................. 9-15
Remove material to create a slot ....................................................... 9-16
Change display to observe the changes ........................................... 9-20
Summary .......................................................................................... 9-21
Lesson review .................................................................................. 9-22
Lesson summary ............................................................................... 9-23
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Introduction

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

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 basic part modeling and editing tutorials before starting self-paced training.

Tutorials

Supported Browsers

- Windows:
  - Internet Explorer 8 or 9
  - Firefox 12 or higher
- UNIX/Linux
  - Firefox 9.x or higher*
- Mac: Safari 5.x or higher

Java Plug In Required for search

The search engine requires version 1.6.0 or higher of the Java Plug In installed to your browser. The plug-in is available (free) in the Java Runtime Environment (JRE) 6.0. If you need to install the JRE, or an equivalent Java environment, visit the Java download site at http://www.java.sun.com.

Adobe Flash Player required for videos and simulations

To watch videos and simulations, you must have the Adobe Flash Player version 10 or later installed as a plug-in to your browser. You can download the Flash Player (free) at the http://get.adobe.com/flashplayer

Adobe Acrobat Reader

Some portions of the help may be delivered as PDF which requires Adobe Acrobat Reader 7.0 or higher. You can download the reader (free) from http://get.adobe.com/reader/

Internet Explorer Caveats

- IE9 Compatibility View. The HTML deliverables are fully supported when launched with the http:// protocol or the file:/// protocol. However, if you are viewing the files from a local installation e.g, D:// , you may need to enable Compatibility View. In IE 9, do the following:

2. In the Compatibility View Settings dialog box, select the “Display all websites” in Compatibility View check box.

*Firefox Caveats

• Firefox recommends that users update the latest version for security issues surrounding Java. They do not recommend using older versions of Firefox due to these issues. See: http://support.mozilla.org/en-US/kb/latest-firefox-issues

• Most customers install and launch our deliverables via http:// protocol which is fully supported. However, Firefox has a default security setting that prevents the help from launching correctly from a UNC path (file:///). To change this setting, you need to change the value of the security.fileuri.strict_origin_policy preference:

  o In the address bar, type about:config.

  o In the Filter field, type security.fileuri, if the value of the security.fileuri.strict_origin_policy preference is set to true, set the value to false. (Double-clicked the value to toggle it.)

  o Restart the browser.
Lesson 1: Moving model faces

Overview

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

• Synchronous model faces and reference planes can move or rotate.

• When you select a face, command bar 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

The Constructing base features course covers the Extrude (2) and Revolve (3) commands.
Lesson 2: Moving synchronous faces

When you select a synchronous face or reference plane, a default graphic handle (1) appears at the select point in a minimal state. If you select the handle origin, the full 3D graphic handle (2) appears with more move options. Click an axis or torus to start the Move command.
Graphic handle (3D steering wheel)

(1) torus  |  (2) tool plane
(3) origin |  (4) X axis
(5) Y axis |  (6) knob
(7) Z axis (normal axis)

The steering wheel displays in a minimal state when selecting a face (1) and displays fully exposed when selecting a feature (2). In a minimal state, only the primary axis appears. To fully expose the steering wheel, click the origin and move it to an edge, keypoint or face of the model.
Reorient the steering wheel

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

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.
Lesson 2: Moving synchronous faces

Change the direction of the primary axis at a user-defined angle

1. Hold the Shift key down.
2. Click the 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.

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 axis direction at a user-defined angle

1. Hold the Shift key down.
2. Click the 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

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.
Lesson 2: Moving synchronous faces

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

- Open `steering_wheel.par`.

Move geometry in axes directions

**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 distance, or a drag and click location.

- In PathFinder, select the feature named feature A.

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

At this point, you can drag and click to define the move distance, type a distance value in dynamic edit box, or select a keypoint.
• Press the Escape key to end the move.

• Click the axis shown. As you drag the cursor, notice the feature moves in the direction defined by the selected axis.

• Press the Escape key to end the move.

**Change an axis direction**

**Note**

You can change an axis direction in 90° increments by clicking a knob on the torus.

**Note**

You can change the selected axis direction at a user-defined point or a drag and click point to define direction angle.
Lesson 2: Moving synchronous faces

- Hold the Shift key and click the axis knob.

- Move feature A at a 45° direction angle. Once the angle is set (1), click the axis (2) to start the move. Image (3) shows a front view to better visualize the 45° movement.

- Press the Escape key to end the move.
Move geometry in the steering wheel tool 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 keypoint.

- Select feature A.
- Click the steering wheel tool plane. Drag the feature around and notice that the movement is locked to the steering wheel plane.

- Press the Escape 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 axis normal to the tool plane. 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°.
Lesson 2: Moving synchronous faces

- Click to end the rotate command.

**Change direction of normal axis**

You change the direction of the normal axes by holding down the Shift key and clicking the steering wheel torus plane.

- Select *feature A*.

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

**Note**

This is a quick method of changing the rotation axis.

- Press the Escape key.

**Change the direction of an axis using a geometric keypoint**

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

- Select *feature A*. 
Reposition the point on the feature to move from. 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).
Lesson 2: Moving synchronous faces

- Click the primary axis knob. Move the cursor over corner (1) and click when the endpoint displays.

- Click the axis and notice the direction of movement.

At this point if you click the endpoint shown, the feature moves to that point.
• Press the Escape 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 snaps 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. The normal axis aligns with the edge it passes over.

**Use the steering wheel to reorient and move a feature**

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

• Turn off the display of *feature A*. In PathFinder, click the box in front of *feature A*.
Lesson 2: Moving synchronous faces

- Turn on the display of feature B.
- Rotate feature B. Select feature B.
- Click the torus, type 90 in the dynamic edit box, and then click.
- Rotate the feature again to complete the orientation. Position the steering wheel origin as shown. Click the torus. Drag and type 90 in the dynamic edit box. Press the Tab key and then click.
• Move the feature to new location. Select feature B. Position the steering wheel origin as shown (midpoint of edge). Define the axis direction to point to the edge midpoint on part. Click the axis to start the move. Move the cursor over the edge midpoint and click when the midpoint highlights. Press Escape 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.
**Moving a face**

You can move a face in the following ways:

- Move a face in a direction along an axis by selecting the axis.
- Move a face freely along a plane where the graphic handle connects by clicking the plane on the handle.
- Set the direction of the normal axis by dragging the handle origin to an edge or vertex. The primary knob also locks onto the edge to define the direction.
- Reposition an axis knob to change the direction of an axis.
- Reposition the axis direction in 90° increments by selecting a knob on the torus.
- The origin is the *move from* point. The origin can move prior to a move face operation.
Rotating a face

Rotate a face by positioning the steering wheel normal axis on an edge. The normal 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 steering wheel appears in a minimal state on the selected face. In a minimal state, you only get the normal axis of the steering wheel. Click the steering wheel origin to display entire steering wheel.

2. The command bar 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 normal 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 command bar list.
   - Type in a distance in the dynamic input box.

5. Press the Escape 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 steering wheel appears in a minimal state on the selected face. In a minimal state, you only get the normal axis of the steering wheel. Click the 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 normal axis of the steering wheel lies on the edge to rotate about. Click and drag the axis 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 an angle in the dynamic edit input box.

5. Press the Escape key to end rotate.

   **Note**

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

---

**Activity: Moving and rotating 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).
Moving synchronous 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 axis to start the move command. Clicking the axis defines the direction vector for the move. All you need to complete the move is a distance to move.

The selected face connects to the cursor and moves dynamically as the cursor moves.

- Use a keypoint locate to define the *move to* distance. On the Move command bar, choose the All keypoints option.
Lesson 2: Moving synchronous faces

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

- Press the Esc key to end the move command.
Move faces

* Move the side faces on the boss a distance defined by a vertex on the side face of the lower base.
Lesson 2: Moving synchronous faces

Rotate a face

• Select the angled face.

• To rotate the selected face, define a rotation axis. Drag the steering wheel origin to the edge shown. The normal axis must lie on an edge that the face rotates 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 Dynamic Edit 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 define distances to move by dragging and clicking, typing in a distance, or by using keypoints. To rotate a face, position the normaly 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 Dynamic Edit box.

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. Copy the lower hole (2) and positioned it at the same angle and distance as the upper holes (1).
Open activity file

- Open move_02.par.
Select the hole to copy

- Select the cylindrical face shown.

- On the Move command bar, choose 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 axis knob shown. This controls the axis direction.

- Move the cursor over the cylindrical face shown, and click when the center point symbol appears.
Lesson 2: Moving synchronous faces

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

- Click the axis to start the Move command.

- Make sure the keypoints option in command bar is set to All or Center Point.
Lesson 2: Moving synchronous faces

- 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—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.

- 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.
Lesson review

Answer the following questions:

1. How do you move a face?
2. How do you rotate a face?
3. How do you move a feature?
4. How do you rotate a feature?
5. What are the cardinal points on the steering wheel used for?
6. How do you copy a feature to a new location?
Lesson summary

Moving and rotating faces is how you modify synchronous models. You can move or rotate a single face, a select set of faces, features, and a combination of faces and features. Use the steering wheel to control how the selected faces move or rotate.
Lesson 3: 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 using the Selection mode.

• Select and deselect faces with the assistance of the Selection Manager. The Selection Manager uses the topological and attribute data of the face selected to add faces to a select set.
Lesson 4: Selection mode

A selection mode symbol appears 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→Selection group.
Normal mode

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. You can only select one face per click.
Add/remove mode

Use the add/remove selection mode 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 you select in this mode adds to the select set. If a you select face that is already selected, it is deselected. The graphic handle remains on the first face selected. Both selected and deselected faces highlight as the cursor moves over them.
Add mode

add mode

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

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

To activate Selection Manager mode, choose the Selection Manager mode button on the Home tab→Select group→Select command list. You can also activate by pressing Shift + Spacebar. To end the Selection Manager mode, press the Spacebar.
Lesson 5: Selection Manager

Use Selection Manager to add or remove items from a select set using the topological and attribute data of a selected object.

When in the Selection Manager mode, a green dot attaches to the cursor.

Clicking on a face 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→Select group, in the Select list, choose the Select Manager Mode command. You can also start the Select Manager mode by pressing Shift+Spacebar. To end the Select Manager mode, press the Spacebar.
Selection Manager options

The Selection Manager shortcut menu is available when you select valid elements.

To display the Selection Manager menu, click on a face.

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 connect to the focus element.
- Interior Faces – Adds all interior faces which connect to the focus element.
- Exterior Faces – Adds all exterior faces which connect 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 are 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 torii. 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.

**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 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: Using the Selection Manager**

Activity guides you through the process of 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.

▸ Activate the Selection Manager mode by either choosing on the Home tab→Select group, from the Select list or by pressing Shift+Spacebar.

▸ Select the round shown below.

▸ On the Selection Manager menu, make sure Use Box Selection is not checked.

▸ On the Selection Manager, click the Equal Radius option. Notice that all rounds that have the same radius (1.5 mm) add to the select set.

▸ Press the Spacebar to exit the Selection Manager mode.
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.

▸ Activate the Selection Manager.

▸ Select the round shown.

▸ On the Selection Manager menu, choose *Use Selection Box*.

▸ On the Selection Manager menu, choose *Equal Radius*.
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 you select the face.
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 (3) to a symmetric definition (4). Side step defines depth in either direction (3) normal to the defined area. The symmetric option defines the depth symmetric (4) 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.
Press the Spacebar to exit the Selection Manager mode.
Change radius of select set

- Change the selected rounds 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

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

- Open select_a.par.
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 moves.

- Activate the Selection Manager mode.

- Select the face shown.

- On the Selection Manager menu, choose Sets. This finds any sets that contain the selected face.

QuickPick displays the sets found. Click the Protrusion entry listed in QuickPick.

- Press the Spacebar to exit the Selection Manager mode.

- The selected protrusion feature participates in the move operation.
Move the feature

- Click the 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

- Activate the Selection Manager mode.
- Select the face shown.
- On the Selection Manager menu, choose Recognize → Rib/Boss.

The Sets option would work here also.
- Press Spacebar to exit the Selection Manager mode.
Move the channel-shaped feature

- Click the 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. Choose the keypoint option on command bar (1).
Extend the legs of the channel-shaped feature

- Select the face shown (1).

**Note**

Both red faces move together because they are coplanar. The Design Intent panel recognizes coplanar relationships and allows you to control the relationship between these faces.

- 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.

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

- Select the protrusion feature.

  Note

  You can select the protrusion from Pathfinder, QuickPick, or with Selection Manager. Make sure you select the protrusion shown.

- 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.
Lesson 6: Lesson review

Answer the following questions:

1. What is a select set?

2. What are the face selection methods?

3. What are the four selection modes and how do you change modes?

4. What is the Selection Manager?

5. How do you start the Selection Manager? How do you end Selection Manager?
Lesson 7: Lesson summary

You can build a select set by selecting faces to modify one at a time. This becomes cumbersome when models become large. The selection methods are available to ease the building of select sets. Selection Manager is a powerful tool to help define faces to modify. You can a combination of selection mode and selection manager to build the select set.
Lesson 8: Move face command bar options

You 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 alter within the command.

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

![Connected face options icons]

**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.
Copy

The Copy option creates a copy of the faces in the select set.
The faces collect into a face set feature.
The face set feature can move or rotate.
This option is similar to a *copy and paste* operation.
The original selected faces do not change.

**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 in a new location on the model.

Open activity file
- Open *copy_a.par*. 

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Move face command bar options

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Select the feature to copy

- Select the cutout feature by clicking Cutout1 in PathFinder.
Suspend the Design Intent option

How to use the Design Intent panel is presented in the *Working with geometric relationships* self-paced course. At this time just suspend the Design Intent setting in the Design Intent panel while moving the cutout feature. This ensures no other faces in the model participate in the move.

- On the Design Intent panel, uncheck the Design Intent (1) option.

![Design Intent panel before and after suspension](image)

Set the copy option and move the feature

- On command bar, choose the Copy option.

![Command bar with Copy option](image)
Lesson 8: Move face command bar options

- Move the copied feature to the orange face.

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

Define the move distance and direction

- Move the feature to the edge of the part using a keypoint. On command bar, click the keypoints list and choose Endpoint.
- Select the keypoint location shown.

**Move in a different direction**

- Move the selected feature downward. Click the axis shown.
Lesson 8: Move face command bar options

- Select the keypoint.

Move by keying in a distance
- Click the axis shown.
Move the cursor inward toward the part, type 10 in dynamic edit box, and press Enter.
Attach the copied feature

The copied feature is in position but is detached from the model.

- Right-click in the part window and choose 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.

Activity: Copying and attaching a feature (method 2)

This activity has the same goal as method 1, but uses a different approach.
Open activity file

- Open copy_b.par.

Select the feature to copy

- Select the cutout feature by clicking Cutout1 in PathFinder.
Suspend the Design Intent option

How to use the Design Intent panel is presented in the *Working with geometric relationships* self-paced course. At this time just suspend the Design Intent setting in the Design Intent panel while moving the cutout feature. This ensures no other faces in the model participate in the move.

- On the Design Intent panel, uncheck the Design Intent (1) option.

Set the copy option and move the feature

- On the command bar, choose the Copy option.

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

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

- The move is from the midpoint of edge (1) to the midpoint of edge (2).

As you drag the steering wheel origin over an edge, the normal axis aligns with the edge. When the edge midpoint indicator appears, click. Move the steering origin to the edge endpoint.
Move the copied feature using the steering wheel plane

- Click the steering wheel tool plane.

- Drag the cursor over the edge shown and click when the midpoint symbol displays. You may have to turn on the midpoint option on command bar.
• 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: 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. Use two methods in the activity.
Open activity file

- Open rotate.par.

Method one overview

Use the Copy option on command bar. Align the feature using the Parallel relationship 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.
Lesson 8: Move face command bar options

- On the command bar, choose the Copy option.

Suspend the Design Intent option

How to use the Design Intent panel is presented in the *Working with geometric relationships* self-paced course. At this time just suspend the Design Intent setting in the Design Intent panel while moving the cutout feature. This ensures no other faces in the model participate in the move.

- On the Design Intent panel, uncheck the Design Intent (1) option.
Move the copied feature

- Click the 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 the Home tab→Face Relate group, choose the Parallel relationship command.

**Note**

The Face Relationship commands are covered in the *Working with face relationships course*. 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 parallel relationship command is an easier step.
- When selecting the parallel command, face (1) is the seed face. Select the face (2) to redefine the seed face.

Select the angled face.

- Click the Accept button on command bar and then press the Esc key.
The feature aligns parallel with the angled face.

**Position the feature**

- The first move aligns the feature with a midpoint of an edge.
  Move the steering wheel origin to the center of one of the cylindrical faces as shown.
• Click the axis shown.

Select the midpoint of the edge shown.

**Note**

If you cannot locate the midpoint on the edge, make sure the All keypoints option is on.
• The second move aligns the bottom of the feature with the face.
  Move the steering wheel origin to any point on the bottom of the feature.

• Click the 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

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 (3). Center feature on face (3) with the feature holes aligned to the midpoint of edge (2).

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 Home tab→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 aligns to the target angled face. Make sure the normal axis points in the direction shown.

**Note**

The normal axis orients normal to the face pasted to.
Lesson 8: Move face command bar options

Copy and paste feature

- Press Ctrl+C to copy the selected feature to the clipboard.
- Press Ctrl+V to paste the feature. The feature attaches 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 position the steering wheel to a point on the bottom of the feature in an earlier step.
Click to place the feature.

**Rotate the feature**

- Position the steering as shown and click the steering wheel torus.
• In the dynamic edit box, type 90 and then press the Enter key.

Center the feature on the face

• Move the steering wheel origin to the midpoint of a linear edge on the feature.
• Click the axis shown to define the move direction.

• 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.

- 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.
**Detach**

The Detach option removes the select set from the part body. The removed select set can move or rotate. This option is similar to a cut and paste operation.

**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.
Open activity file

- Open `detach_a.par`.

Select the feature to detach

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

Suspend the Design Intent option

How to use the Design Intent panel is presented in the *Working with geometric relationships* self-paced course. At this time just suspend the Design Intent setting in the Design Intent panel while moving the cutout feature. This ensures no other faces in the model participate in the move.

- On the Design Intent panel, uncheck the Design Intent (1) option.
Lesson 8: Move face command bar options

Position the steering wheel origin

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

Move the feature

- On the command bar, choose the Detach option.
• 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.
**Precedence**

Use the precedence option to set which faces 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 review

Answer the following questions:

1. Name the three connected faces options and briefly describe the results of each option.

2. When you copy a set of faces or a feature, what happens to the original faces?

3. When using the Copy to (Ctrl+C) clipboard command, what is the importance of the steering wheel secondary axis direction?

4. What does the Detach option do?

5. Explain the Precedence option.
Lesson summary

Use the move command bar options to control the behavior of the select set during a move operation.
Lesson 9: 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.

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.
Automatic creation of live sections

When creating a revolved extrusion or cutout, use the Create Live Section option (1) to create a live section upon feature completion. The option is on by default.

All sketch dimensions migrate to the live section.

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 appears. 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 Design Intent settings 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.
Open activity file

- Open live_section.par.
Create a section plane

- On the Home tab→Planes group, choose the Coincident Plane command.

- Select the plane shown.
Move the coincident plane to the midpoint of the edge shown.
Create a Live Section

- On the Home tab→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.
Notice in PathFinder that a *Live Sections* collector appears. You control the display of a live section with the check box.

- Live Sections
- Live Section 2
Move a face

Instead of selecting a face to move, you can select the edge created by the section through a face 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 edge 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 display to manipulate a live section. Turn off the display of the model.

- Right-click in the part window and on the shortcut menu choose *Hide All → Design Body*. Hide all reference planes too.

- Change the display to a front view. Press Ctrl+F.
- Rotate the two arms on the left 15° about the center hole. Select the live section edges shown using a fence.
• 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.

**Note**

If you get an error during the rotation, uncheck the Design Intent option in the Design Intent panel.

- 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 changes to the modifications made to the live section.
Delete a face

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

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.
• Draw a sketch containing two lines and one arc. Choose the Line command and click the right section edge.
• Align the circle center with the midpoint of the right edge.

• Select the region shown.

• On the command bar, click (1) to set the Through All extent option, click (2) to set the Symmetric extent, and click (3) to set the Remove material option.
• Click on a direction handle and dynamically drag to exceed the width of the part.
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.
Lesson review

Answer the following questions:

1. How do you create a Live Section?

2. How do you edit a model with a Live Section?

3. When creating a revolved feature with a dimensioned sketch, what is the result when you choose Live Section option on the revolve command bar?

4. How do you redefine the live section?
Lesson summary

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.
Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

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