

# *Solid Edge assembly*



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**SOLID EDGE**  
VELOCITY SERIES

*...with Synchronous Technology*

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**Lesson**

*1 Modeling assemblies*

## **Solid Edge Assembly**

An assembly is a collection of parts and subassemblies that are positioned in a meaningful way. The parts can be in their final orientation, or have freedom of movement in translation and rotation. Solid Edge Assembly provides the tools needed to lay out and position the parts relative to each other. Many methods exist to accomplish this task and these approaches to building assemblies will be covered in this text.

### **Objectives**

This lesson introduces the Solid Edge Assembly interface and discusses the different workflows for creating an assembly with the most commonly used part relationships.



## **Placing parts in assemblies**

You can place any of the following types of solid parts in Solid Edge assemblies using the Parts Library tab:

- A part constructed in the Solid Edge Part environment.
- A part constructed in the Solid Edge Sheet Metal environment.
- Another assembly constructed in the Solid Edge Assembly environment.
- Any file that is open in Solid Edge other than a draft file.

To place parts that were constructed in other CAD formats, you must first convert them to Solid Edge part files.

### **Note**

When you add an Insight-managed document to an assembly, Solid Edge uses the SearchScope.txt file to prevent you from creating links to documents with duplicate IDs. The SearchScope.txt file must list at least one managed workspace, or you will not be able to place managed parts in assemblies.

## **Sharing assemblies**

You can place parts in assemblies by selecting them from a local folder path or from a network share. If you use local folder paths, other Solid Edge users who access the assembly over a network will not be able to view its parts and subassemblies. If you intend to share an assembly over a network, you should always select the parts through a network share, even if they are stored on your computer.

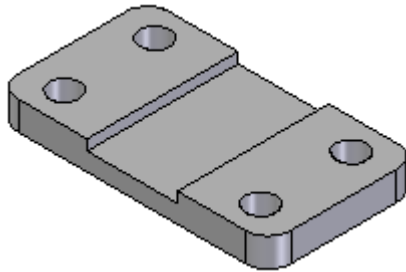
To do this, use the arrow on the right side of the Look In option on the Parts Library tab to browse to and select the folder on a network drive where the part or subassembly is stored.

The network share approach also allows you to build an assembly using parts that are stored on several computers on your network. For example, your company may have one or more computers that are used as servers, where commonly used parts are stored.

### Placing the first part in an assembly

To start the part placement process, in the Parts Library tab, select the part you want, then drag it in the assembly window. You can also start the part placement process by double-clicking the part in the Parts Library tab.

The first part you place into an assembly is important. It serves as the foundation upon which the rest of the assembly will be built. Therefore, the first part should represent a fundamental component of the assembly. Because the first part is placed grounded, you should pick a part with a known location, such as a frame or base.

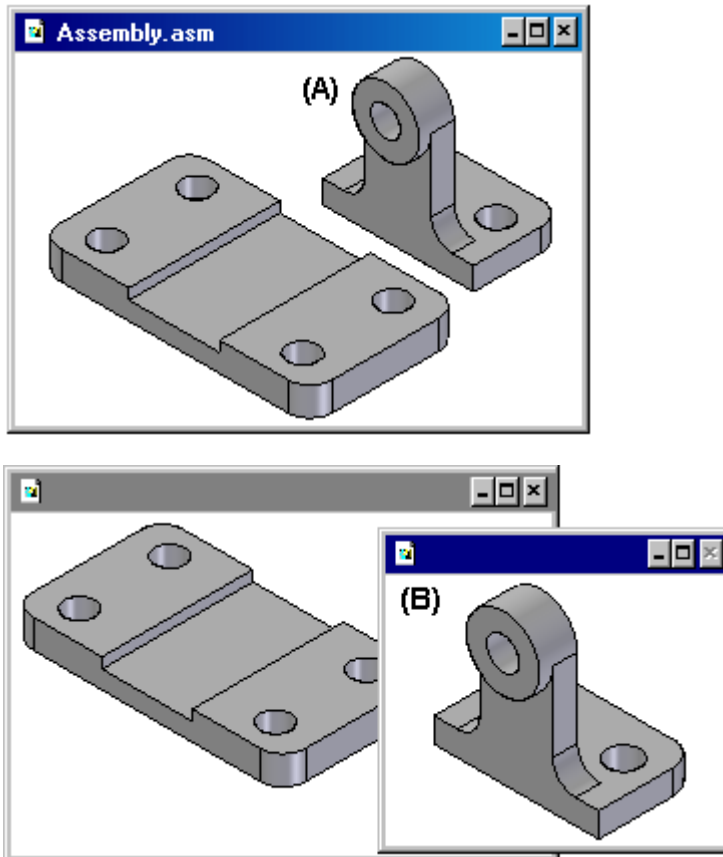


Although Solid Edge makes it easy to edit parts during the design cycle, the first part you place in the assembly should be as completely modeled as possible. In the same way, although it is easy to delete parts from assemblies and change assembly relationships, the first part you place should remain grounded and not be deleted.

To reposition the first part, you should first delete the ground relationship. You can then apply assembly relationships between the first part and the assembly reference planes or subsequent parts you place in the assembly.

## Placing additional parts in an assembly

You can use the Assembly tab on the Options dialog box to specify whether subsequent parts are temporarily placed in the assembly window (A), or displayed in a separate Place Part window (B).



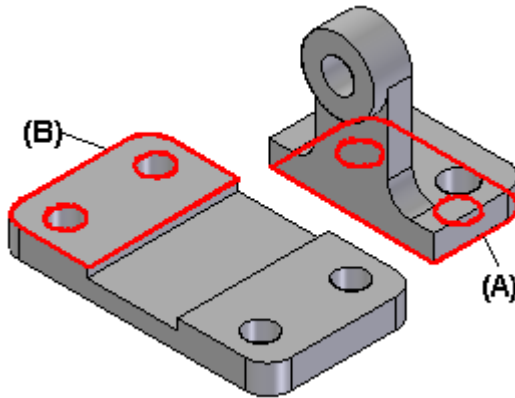
When you set the Do Not Create a New Window During Place Part option, the part is temporarily placed in the assembly window at the location where you dragged and dropped the part. To make the positioning process easier, drop the part in a location where it is easy to select the positioning elements you want to use. If you start the part placement process by double-clicking the part in the Parts Library tab, the display area of the assembly window is adjusted so you can see the new part.

When you clear the Do Not Create a New Window During Place Part option, the part is displayed in a separate Place Part window. If the active window is maximized, the Place Part window is also maximized, essentially hiding the assembly window from view. Due to this, beginning users should not maximize the active window. Let the windows overlap, and this will make placing parts into the assembly and applying relationships much easier.

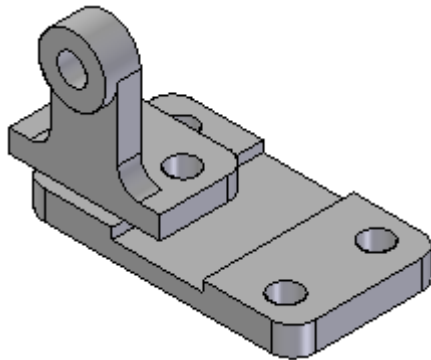
## Positioning parts

You use assembly relationships to position the new part relative to a part already in the assembly. The Relationships Types option on the Assemble command bar contains a wide range of assembly relationships for positioning parts relative to one another.

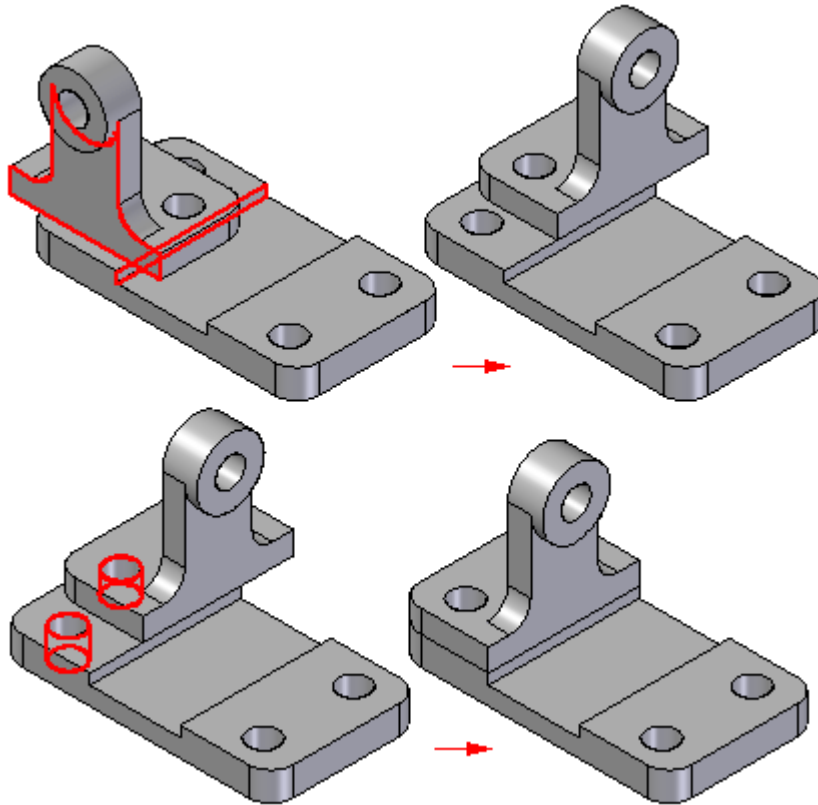
In addition to traditional assembly relationships, the FlashFit option reduces the steps required to position a part using the mate, planar align, or axial align relationships. This option is recommended in most situations. For example, you can use FlashFit to mate a face on the placement part (A) with a face on the target part (B).



After you apply the first assembly relationship, the new part is repositioned within the assembly.



As you apply the remaining assembly relationships, the software positions and reorients the part in the assembly.



Additional parts can be positioned relative to any part in the assembly, or even relative to more than one part in the assembly. You can also position a part relative to an assembly sketch.

For more information on positioning parts using assembly relationships, see the [Assembly Relationships](#) Help topic.

#### Note

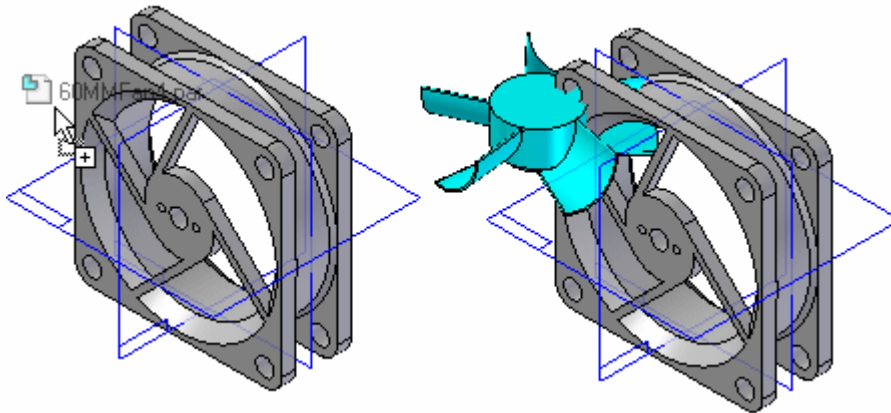
By default, Solid Edge maintains the relationships with which you position the part. If you clear the Maintain Relationships command on the Parts Library shortcut menu, the relationships will be used only for positioning, and the part will be grounded. Grounded parts do not update their positions when you make design changes.

## Placing parts that are not fully positioned

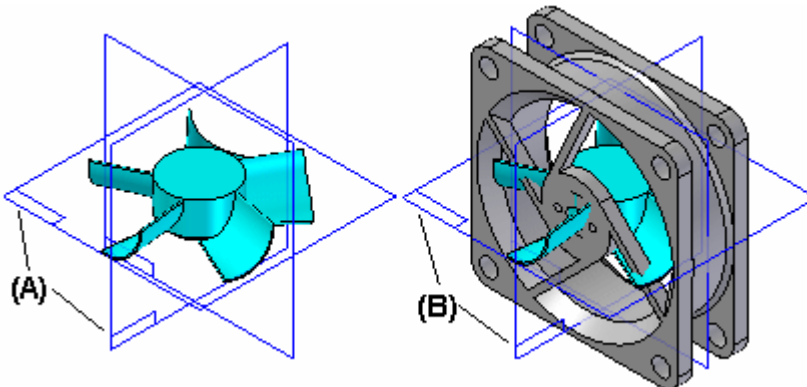
It is a good idea to fully position parts as you place them into assemblies. Fully-positioned parts will update their positions more predictably when changes are made. At times, though, you may want to place a part without fully positioning it. For example, you may be placing another part later that will be used to complete the positioning of both parts.

You can use the Esc key to interrupt the placement sequence at any time. If no relationships have been applied, the part is placed in the assembly at the same relative position it occupies in the part document. In other words, the part is placed in the assembly such that the base reference planes in the part document (A) are coincident with the base reference planes in the assembly (B).

If you work with the Do Not Create a New Window During Place Part option set, the part is placed in the assembly at the location you dragged and dropped it into the assembly.



If you work with the Do Not Create a New Window During Place Part option cleared, the part is placed in the assembly such that the base reference planes in the part document (A) are coincident with the base reference planes in the assembly (B).



You can also interrupt the placement process by selecting another command, for example, the Select tool.

## Placing the same part more than once

If you want to place the same part in an assembly more than once, you do not have to use the Parts Library tab every time. After you place the part the first time, you can select it, copy it to the Clipboard, and then paste it into the assembly.

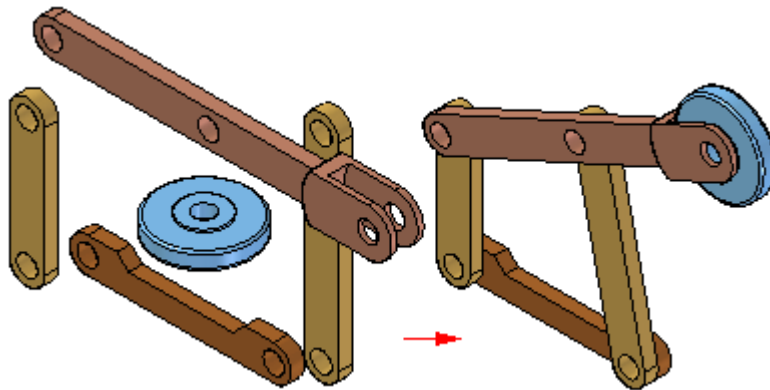
When you select the Paste command, the part is displayed in a separate window, as if you had selected it from the Parts Library tab. You can then apply assembly relationships between the new part and the other parts in the assembly.

You can also use PathFinder to place an existing part into an assembly again. Select the part in PathFinder, then drag and drop it into the assembly window.

If a part is being placed in the assembly several times using the same relationship scheme, you can use the Capture Fit command to store the relationships and faces used to position the part the first time. This reduces the number of steps required to define each relationship when you place the part again. When you place the part later, you do not need to define which relationship and face you want to use on the placement part. You only need to select a face on the target part in the assembly for each relationship.

## Positioning a set of parts

You can use the [Assemble command](#) to position a set of parts relative to each other without fully constraining each part in an ordered sequence. This type of workflow can make it easier to position a set of interrelated parts, such as when building a mechanism.



First, drag and drop the set of parts into the assembly. Then click the Assemble command and apply relationships between one part and the other parts. To position a different part, click the right mouse button.

## Finding parts

If you do not know the name and location of a part or subassembly, you can define search criteria using Search button on the Parts Library tab. You can then double-click the name of the part or subassembly from the list of search results to start the part placement process.

**Part placement properties**

When you place a part or subassembly into an assembly, Solid Edge sets properties that determine the following:

- The placement name of a part or subassembly.
- Whether the part is selectable or not selectable.
- The quantity of the part.
- The x, y, and z location for grounded parts or parts or not positioned using assembly relationships.
- Whether the part is displayed in a higher level assembly.
- Whether the part is displayed in a drawing of the assembly.
- Whether the part is considered a reference part in a drawing or parts list.
- Whether the part is used in a report, such as a Bill of Material.
- Whether the part is used in mass property calculations of the assembly.
- Whether a part is used in an interference analysis calculation.

You can also change these properties later using the Occurrence Properties button on the Place Part command bar or the Occurrence Properties command when selecting assembly components.

**Placing simplified parts**

The Use Simplified Parts command on the Parts Library shortcut menu allows you to specify whether you want to use the simplified or designed version of a part when placing it in the assembly. When you have the Use Simplified Parts command set (there is a check mark adjacent to the command), any faces that were deleted when simplifying the part will not be available for positioning purposes. To make these faces available, clear the Use Simplified Parts command.



## Placing subassemblies

You can place a Solid Edge assembly document into another assembly in much the same way you place an individual part. When placing an assembly, in the Place Part window, you must first select the placement part in the assembly you want to use for positioning purposes, then the face on the part.

If you are placing a large subassembly, you can save a display configuration in the subassembly first, then use this configuration to make placement easier. For example, you can hide all the parts, except those that will be used to position the subassembly. Before you place the subassembly, make sure the Use Configuration command on the shortcut menu is set. Then, when you place the subassembly, you can select the configuration name from the Configuration list on the Use Configuration dialog box. Subassemblies also place faster if parts have been hidden.

When placing a subassembly using FlashFit or the Reduced Steps mode, the placement part step is skipped. You specify the placement part by selecting a face on the placement part you want.

The placement part must be active before you can select a face. If the placement part is not already active, you can use the Activate Part button on the Place Part command bar to activate the placement part.

### Note

When placing parts into a subassembly, you can set an option that controls whether the part is displayed in higher level assemblies. If the Display When Assembly Is Attached As Subassembly option on the Properties dialog box is cleared for a part, that part will not be displayed in PathFinder or the graphics window in higher level assemblies.

## Locating parts

The parts and subassemblies used to build an assembly can be located on different computers on a network. To locate a part or subassembly, you can use the Assembly Statistics command on the Inspect tab to display the document name and location. When the Assembly Statistics dialog box is displayed, select any document in the list, then right-click, and then set the Show Document Name and Location option.

## Non-graphic parts

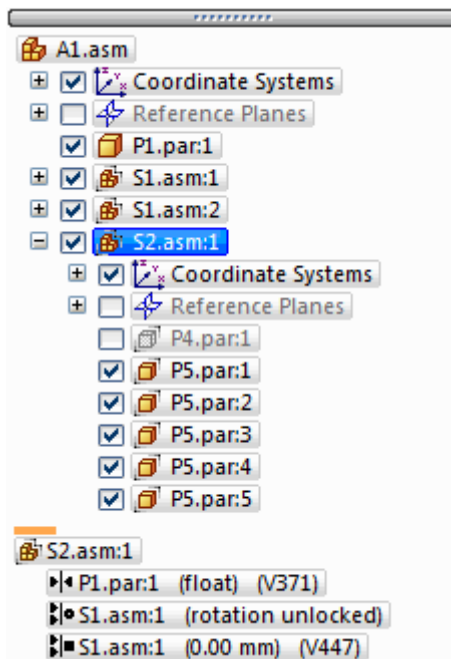
Sometimes you need to place parts in assemblies that have no model associated with them, so they can be documented in a parts list or bill of materials. For example, items such as paint, grease, oil, and so forth require no model, but a part must exist in the assembly for them to show up in the documentation for the assembly. Solid Edge allows you to create non-graphic parts by adding custom properties to an empty part document. You can then place the part in the assembly without positioning it with assembly relationships, by pressing the Shift key, then dragging the part into the assembly.

## PathFinder in assemblies

The PathFinder tab helps you work with the components that make up your assembly. It provides alternate ways to view the composition and arrangement of the assembly, besides looking at the graphics in a regular assembly window. You can also use PathFinder to in-place activate a part or subassembly so you can make edits to individual assembly components while viewing the entire assembly.

The PathFinder tab is available when you work in an assembly or a subassembly within the active assembly.

In the Assembly environment, you can also use PathFinder to view, modify and delete the assembly relationships used to position the parts and subassemblies, reorder parts in an assembly, and to help you diagnose problems in an assembly.



In the Assembly environment, PathFinder is divided into two panes. The top pane lists the components of the active assembly in a folder tree structure. Listed components can include: parts, subassemblies, assembly reference planes, and assembly sketches.

The bottom pane shows the assembly relationships applied to the part or subassembly selected in the top pane.

## Using the top pane

The top pane of PathFinder allows you to do the following:

- View components in collapsed or expanded form. For example, when you expand a subassembly, you can view all of its parts.
- Highlight, select and clear components for subsequent tasks.
- Determine the current status of the components within the assembly.
- Determine how the assembly was constructed.
- Reorder parts within an assembly.
- Rename reference planes, sketches, and coordinate systems.

When you pass your cursor over a component in the top pane of PathFinder, it is displayed in the graphics window using the Highlight color. When you click a component it is displayed using the Select color. This allows you to associate the component entry in PathFinder with the corresponding component in the graphics window.

### Note

When you pass your cursor over or click the top-level assembly in PathFinder, it does not display in the highlight or select color. This improves performance when working with large assemblies.

































Because the highlight and selection of components in large assemblies can impact performance, options are available on the Assembly tab on the Options dialog box that allow you to improve the performance when working with large assemblies. For example, options are available that allow you to simplify the display of highlighted and selected components in the graphics window and to disable the highlight of components in the graphics window when you pass your cursor over them in PathFinder.

For more information on improving performance in large assemblies, see the [Working with large assemblies efficiently](#) Help topic.

## Determining the status of a component

The symbols in PathFinder reflect the current status of the components in the assembly. The following table explains the symbols used in the top pane in PathFinder:

### Legend

	Active part
	Inactive part
	Hidden part
	Unloaded part
	Part that is not fully positioned
	Part that has conflicting relationships
	Linked part
	Simplified part
	Missing component
	Alternate components part
	Part position is driven by a 2D relationship in an assembly sketch
	Displayed assembly
	Adjustable Part
	Adjustable Assembly
	Driven Reference
	Fastener System
	Pattern group
	Pattern item
	Reference planes
	Reference plane
	Sketch
	Noncombinable sketch (synchronous only)
	Combinable sketch (synchronous only)
	Active sketch (synchronous only)
	Group of parts and subassemblies
	Motor
	Available
	In Work
	In Review
	Released
	Baselined
	Obsolete

**Note**

The symbols in PathFinder can also represent combinations of conditions. For example, a symbol can show that a part is hidden and not fully positioned:

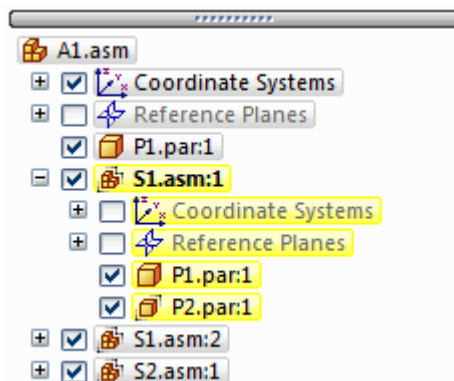
**Determining how the assembly was constructed**

The components in the top pane of PathFinder are listed in the order in which they were placed in the assembly. This can be useful when evaluating design changes. For example, if you delete a single assembly relationship from a part, the symbols for other parts could also change to indicate that the parts are no longer fully positioned. This occurs because the positioning of the other parts depended upon the part from which you removed the relationship. In this example, reapplying the single relationship should cause the other parts to also become fully positioned again.

**Making changes to assembly components**

You can use the top pane of PathFinder to open or in-place activate a part or subassembly so you can make design modifications. For example, you can select a part in PathFinder, then use the Edit command on the shortcut menu to in-place activate a part. You can then add, remove, or modify features on the part while viewing the other assembly components. You can also use geometry on the other assembly components to help you construct or modify features on the part. When you use the Open command to open an assembly component, you cannot view the other assembly components.

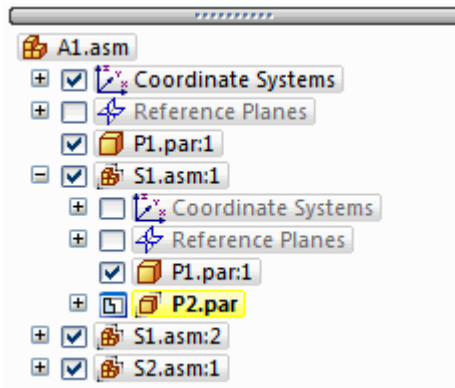
When you in-place activate a subassembly, the display of PathFinder changes to make it easier to determine your current position within the assembly structure. For example, while in the top-level assembly A1.asm, if you in-place activate into subassembly S1.asm:1, subassembly S1.asm:1 is displayed using bold text and a contrasting background color is used for the subassembly and its components.



When you in-place activate a part for editing, you do not need to return to the assembly first to in-place activate another part or subassembly in the assembly.

You can select another part or subassembly in PathFinder and use the Edit command on the shortcut menu to in-place activate the component for editing. When you are finished making the design changes, you can use the Close and Return command on the Home tab to return to the original assembly.

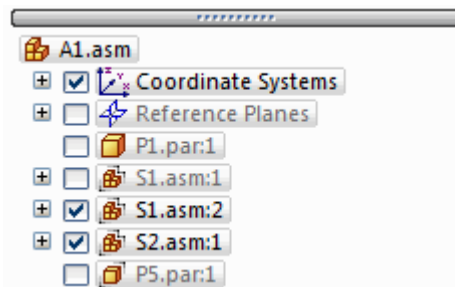
When you in-place activate a part or subassembly for editing, you cannot collapse the assembly structure to which the part or subassembly belongs within PathFinder. For example, in the following illustration, part P2.par:1 has been in-place activated and it is in subassembly S1.asm. If you click the minus (-) symbol adjacent to S1.asm to collapse its structure, it will remain expanded.



### Changing the display status of assembly components

You can use the top pane of PathFinder to control the display status of assembly components. For example, you can hide parts and subassemblies to make it easier to position a new part you are placing in an assembly. You can use the checkboxes adjacent to the assembly components in PathFinder to control component display or shortcut menu commands when one or more components are selected.

The color of the text in PathFinder also indicates whether a component is displayed or hidden.



### Reordering parts within an assembly

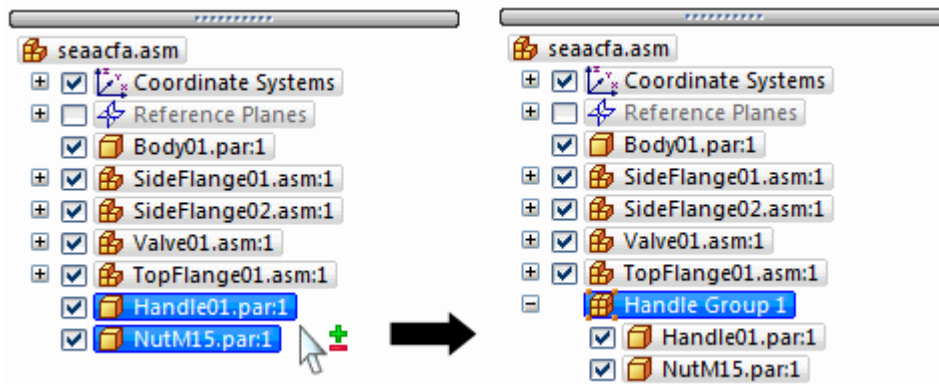
PathFinder allows you to drag a part to a different position within an assembly. As you drag the part, PathFinder displays a symbol to show where you can reposition the part in the assembly structure. The part will be positioned below the highlighted part occurrence in PathFinder.

## Grouping parts and subassemblies within an assembly

PathFinder allows you to select a set of parts or subassemblies in the active assembly, then specify that the selected components are a group using the Group command on the shortcut menu. The set of components is then collected into a group entry in PathFinder. You can then expand, collapse, or rename the group to a more logical name. Defining a group of parts reduces the space requirements for a set of parts, and allows you to gather together a set of similar parts into a logical group. This can make it easier to select the parts for other operations, such as displaying and hiding a set of parts.

You cannot select nested parts or nested subassemblies.

Grouping components is also useful when working with large assemblies that contain few or no subassemblies. You can select a set of parts, define them as a group with the Group command on the Pathfinder shortcut menu, then use the Rename command to rename the group to a more logical name.



### Note

Some assembly commands create a group of components automatically. For example, the Move Components command creates a group entry in PathFinder when you set the Copy option on the command bar.

You can ungroup a group using the Ungroup command on the shortcut menu when a group is selected in PathFinder.

The Select Components command, on the shortcut menu when you select a group entry in PathFinder, activates additional commands and options for manipulating groups that would otherwise be disabled. For example, after selecting a group with the Select Components command, you can then apply a face style to the group of parts, or transfer the group of parts to another assembly.

## Renaming PathFinder entries

You can use PathFinder to rename an entry for an assembly reference plane, sketch, group, or coordinate system. To rename an entry, select it in PathFinder, right-click and then click Rename. In the name box, type the new name for the entry.

## Finding parts

In a complex or unfamiliar assembly, it can sometimes be difficult to determine which subassembly a particular part is contained in. You can use the Scroll To Part command to quickly find a part in PathFinder. When you select a part in the assembly window, then click the Scroll To Part command on the shortcut menu, the display of PathFinder scrolls to the selected part. If the part is in a subassembly, the listing for the subassembly is expanded to display the part.

## Replacing the file name with the document name formula value

You can use the Document Name Formula dialog box to replace the file name displayed in PathFinder with a value composed of document properties. Refer to the Replace a file name with a property value help topic for instructions.

You can combine properties with additional characters to replace the file name. For example, you can separate two properties with dashes, such as Document Number–Revision Number.

If a property does not exist or does not have a value, the property name is displayed in place of the property value, and the file name is displayed in parentheses after the value.











### Note

The Property list displays the properties that you can use to replace the file name. You can add a property that is not in the active document by typing [property name] in the Formula field.

## Using the bottom pane

When you select a part or subassembly in the top pane of PathFinder, you can use the bottom pane to view and modify the assembly relationships between the selected part and the other parts in the assembly. The document name is also displayed, as well as a symbol that represents the type of relationship. The following table explains the symbols used in the bottom pane in PathFinder:

### Legend

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



When you select a relationship in the bottom pane you can do the following:

- View which elements were used to apply the relationship.
- Edit the fixed offset value of the relationship.
- Change the offset type of the relationship.
- Delete the relationship.
- Suppress the relationship

### **Viewing assembly relationships**

When you select a relationship in the bottom pane, the elements used to apply the relationship are highlighted in the assembly window. For example, if you select a planar align relationship, the planar faces or reference planes that were used to apply the relationship highlight in the assembly window. This can help you determine how design changes need to be applied.

### **Modifying assembly relationships**

When you select a relationship in the bottom pane, you can use the relationship command bar to edit the fixed offset value or change the offset type. For example, you may want to change a mate relationship from a fixed offset to a floating offset.

#### **Note**

If you change the offset type from fixed to floating, you may have to make other relationship edits to ensure that the part remains fully positioned.

### **Deleting assembly relationships**

If you delete an assembly relationship, the symbol next to the part in the top pane changes to show that the part is no longer fully positioned. The part is also placed on the Error Assistant dialog box list. It is good practice to apply a new relationship to the affected parts as soon as possible. If you delete too many relationships without applying new ones, it could become difficult to fully position the affected parts. If this occurs, you may have to delete the affected parts from the assembly and place them again.

### **Replacing relationships**

After you place a part in an assembly, you can replace any of its relationships. Select the part in PathFinder or in the graphics window, then click the Edit Definition button on the command bar. You can then select the relationship you want to replace from the Relationship List box on the command bar. Use the Relationship Types button to specify the new relationship you want to apply.

#### **Note**

You can also delete the current relationship in the bottom pane of PathFinder and apply a new one using the Assemble command bar.

## Conflicting relationships

If you change the design of parts in an assembly, some assembly relationships may no longer be applicable. When this occurs, the symbol next to the part or subassembly in the top pane of PathFinder will change to indicate that there are conflicting relationships and the part will be placed on the Error Assistant dialog box list.

When you select the conflicted part or subassembly, the symbols for the affected relationships in the bottom pane of PathFinder are displayed in red. You can then evaluate the relationship scheme to determine how to repair the assembly. For example, you can delete the affected relationships and apply new relationships to fully position the part.

## Suppressing assembly relationships

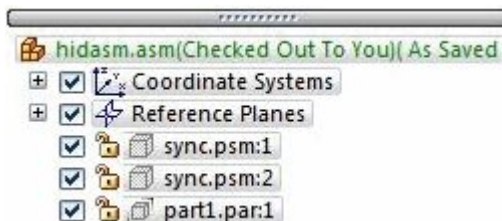
You can use the Suppress command on the shortcut menu to temporarily suppress an assembly relationship for a part. Suppressing an assembly relationship allows you to use the Drag Part command to evaluate how the part interacts with other parts in the assembly. When you suppress an assembly relationship, the symbol for the part in the top pane of PathFinder changes to indicate that the part is no longer fully positioned. Also, a symbol is added adjacent to the suppressed relationship in the bottom pane to indicate that the relationship is suppressed.

### Note

You can use the Unsuppress command on the shortcut menu to unsuppress the relationship.

## Displaying document status in PathFinder

You can display the document status for components in PathFinder. For example, in an Insight-managed document, the status can be Available, In Work, In Review, Released, Baseline or Obsolete. The Status@ Display Status command on the PathFinder shortcut menu turns on and off the display of symbols adjacent to the document names in PathFinder. For more information, see the Displaying and updating status for documents in assemblies Help topic.

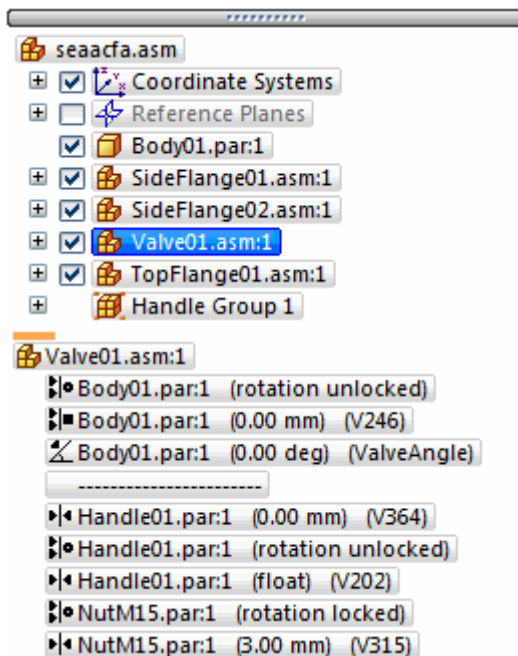


**Legend**

	Available
	In Work
	In Review
	Released
	Baselined
	Obsolete

**Dashed line in the bottom pane**

Often a dashed line is displayed between sets of relationships in the bottom pane of PathFinder. The relationships above the dashed line were applied to parts that are above the selected part in the top pane of PathFinder. The relationships below the dashed line were applied to parts that are below the selected part in the top pane of PathFinder. You can edit the relationships above the dashed line and below the dashed line. For example, when you select *Valve01.asm*, the relationships above the dashed line were applied to *Body01.par*, which is above *Valve01.asm* in the top pane of PathFinder. The relationships below the dashed line were applied to *Handle01.par* and *NutM15.par*, which are below *Valve01.asm* in the top pane of PathFinder.

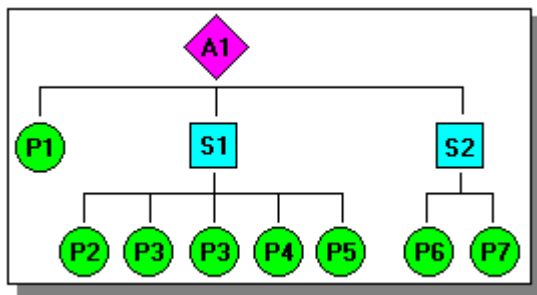


## Managing relationships in nested assemblies

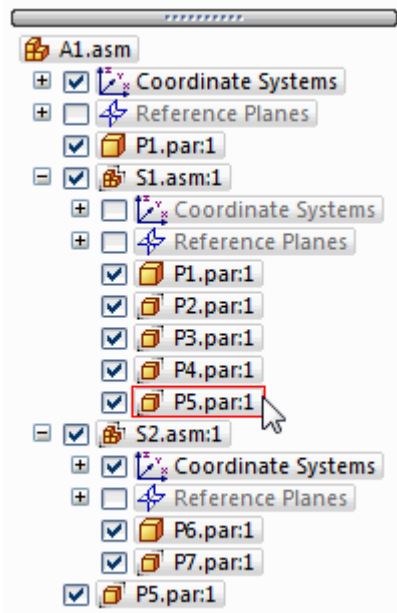
PathFinder does not display relationships applied outside the active assembly. Before you can view, modify, or delete an assembly relationship from a nested subassembly, you must first open or in-place activate the subassembly where the relationship was applied.

You can use the top pane of PathFinder to determine at which level in a multi-level assembly a particular part was placed. You can then select the subassembly in the top pane of PathFinder and use the Open or Edit commands on the shortcut menu to open or in-place activate the subassembly to modify or replace the relationship.

For example, assembly A1 in the next illustration was built using part P1 and subassemblies S1 and S2. Subassembly S1 was built using parts P2, P3, P4 and P5. Subassembly S2 was built using parts P6 and P7. If you want to change a relationship used to position part P5, you have to open or in-place activate subassembly S1.



You can find part P5 by viewing the assembly in the top pane of PathFinder, as shown in the next illustration. Since part P5 is indented under subassembly S1, you would have to open or in-place activate subassembly S1 in order to view, modify, or remove any relationships that control part P5.



## **Assembly relationships**

When placing a part or subassembly into an assembly, you must define how the part will be positioned with respect to the other parts in the assembly by applying assembly relationships. Available relationships include ground, mate, planar align, axial align, parallel, connect, angle, cam, gear, tangent, and center-plane.

In addition to the traditional assembly relationships listed above, the FlashFit option reduces the steps required to position a part using the mate, planar align, or axial align relationships.

The relationship options and FlashFit are located on the Relationship Types list on the Assemble command bar.

## **Part positioning workflows**

Solid Edge provides several workflows for positioning parts in an assembly:

- FlashFit
- Traditional Workflow
- Reduced Steps
- Capture Fit

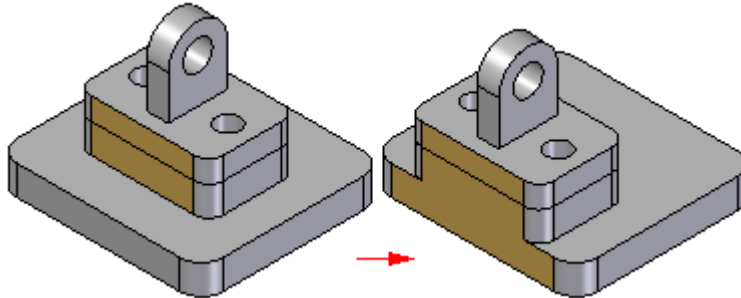
### **Note**

New users should focus on learning both FlashFit and the Traditional Workflow. As your expertise in building assemblies increases, you can explore the other workflows available. All the workflows are discussed in more depth later in this topic. The Slider tutorial demonstrates FlashFit capabilities.

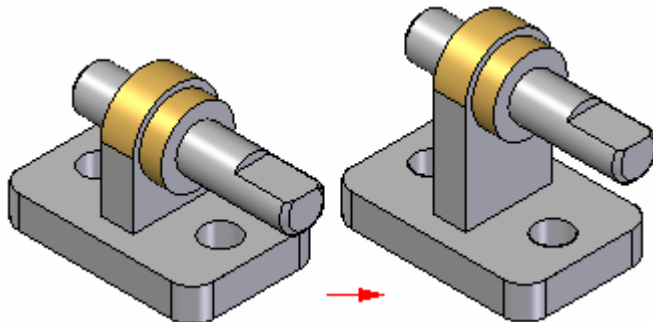
## Maintaining assembly relationships

By default, Solid Edge maintains the relationships with which you position the part. If the Maintain Relationships command is set on the Parts Library shortcut menu when you place a part, the relationships that you apply also control the behavior of the part when it is modified. For example:

- If you apply a planar align relationship between two parts, they remain aligned when either part is modified.



- If you apply an axial align relationship between two parts, they remain axially aligned when either part is modified.



### Note

You can view, modify, and delete assembly relationships using the Assembly PathFinder tab.

If the Maintain Relationships command is cleared when you place a part, you must still use assembly relationships to position the part in the assembly. However, instead of applying these relationships on the part, the software applies a ground relationship. Grounded parts do not update their positions in the assembly when other parts are modified.

## **Capturing design intent**

To fully control one part in relation to the other parts in an assembly, you must use a combination of assembly relationships. There is often more than one way to apply relationships that will position a part correctly. It is important to choose the way that best captures design intent, because this makes your assembly easier to understand and edit.

It may be helpful to keep in mind how the part will react to future modifications when positioning a part. Although the part may be positioned correctly using a particular set of assembly relationships, it may not behave as you expect when modifications are made.

As you gain experience placing parts in an assembly, you may find it useful to make minor design modifications and observe how the parts in your assembly react. If the assembly does not behave as you expect, you can delete the relationships and reapply them using a different approach. As you become more experienced, it will become easier to see which set of relationships correctly positions the parts, and gives you the behavior you want when design modifications are made.

## **Assembly relationships and part movement**

When a part is fully positioned in an assembly, it cannot move in any direction in relation to the assembly. The first assembly relationship you place controls some part movement, but the part is still free to move in some direction by sliding along or rotating around the X, Y, or Z axes.

Applying more relationships controls more movement until the part is fully positioned. The types of relationships you apply and the options you use determine how the relationships control part movement.

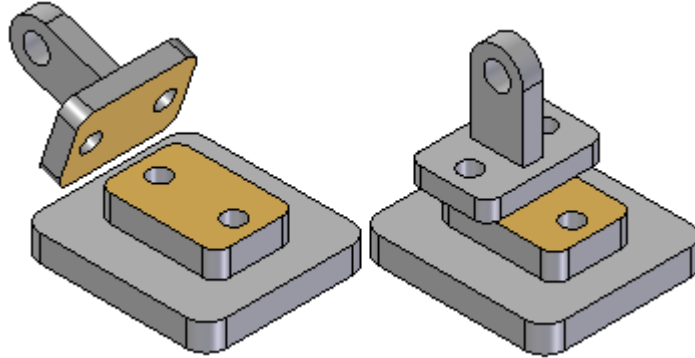
## **FlashFit**

As discussed earlier, the FlashFit option reduces the steps required to position parts using mate, planar align, and axial align relationships when compared to the traditional workflow. Because many parts are positioned using these relationships, FlashFit is appropriate in most situations.

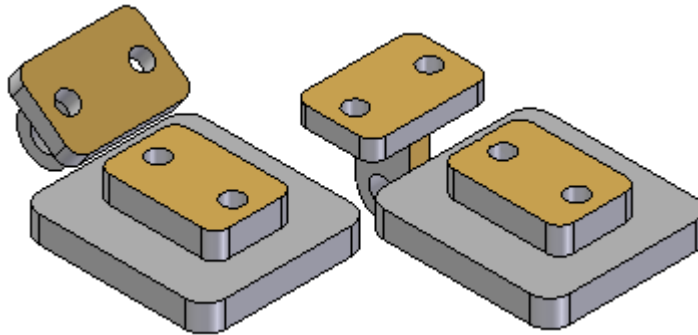
When you position a part using FlashFit, you first select a face or edge on the placement part. You then select a face or edge you want on the target part and let the inference logic built into Solid Edge determine the most likely relationship, based on the target part element.

For example, if you choose a planar face on the placement and target parts, the software assumes that you want to establish a mate or planar align relationship. When you select the target part element, the placement part is positioned in the assembly using the closest solution.

- If the two faces you select are closer to a mate solution, a mate relationship is applied.



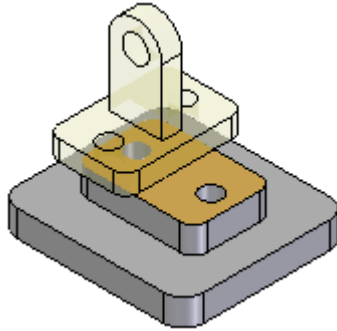
- If the two faces you select are closer to a planar align solution, a planar align relationship is applied.





A Flip button on the command bar allows you to select the alternate solution. You can also use the Tab key to select an alternate solution.

When using FlashFit to position a part, it is displayed translucent to make it easier to differentiate from the other parts in the assembly.



When possible, FlashFit moves the first part you select when applying the relationship, and the second part remains stationary. If the first part you select is fully constrained, the second part will move.

You can then use FlashFit to define the additional relationships required to fully position the part in the assembly, or select another relationship type.

**Note**

When placing a subassembly using FlashFit or the Reduced Steps mode, the parts in the subassembly must be active before you can select a face. If the subassembly is not already active, you can use the Activate Part button on the Assemble command bar to activate the placement part in the subassembly which contains the face you want to select.

FlashFit also allows more flexibility to use edges, in addition to faces, when positioning a part using mate, axial align, and planar align relationships.

This can be especially useful when positioning a fastener, such as a bolt into hole. For example, when positioning a part using an axial align relationship, you cannot use a circular edge to position a part. With FlashFit, you can use a circular edge on both the placement part and target part to completely position the part in two steps.

**FlashFit options**

The Options dialog box on the command bar allows you to set the FlashFit options you want to use. For example, you can specify the element types you want FlashFit to recognize when placing a part. This allows you to tailor the behavior of FlashFit for the part you are currently placing.

**Moving and rotating parts with FlashFit**

When using FlashFit, you can also move or rotate the placement part into a more convenient location. To move the part, position the cursor over the part and drag the cursor.

To rotate the part, press the Ctrl key while dragging the cursor. If any relationships have been applied to the placement part, the movement or rotation is limited to the available degrees of freedom.

## Traditional part positioning workflow

The traditional workflow walks you through every step required to position a part using assembly relationships. For new users, this allows you to gain a full understanding of the part positioning process. A command bar, which is unique to each relationship, guides you through the positioning process.

The traditional workflow is also preferred when positioning parts using relationships that FlashFit does not recognize, such as angle, cam, parallel, and tangent relationships.

## Reduced steps

The Reduced Steps option eliminates the part selection and accept steps in the traditional workflow. You can set this option using the Options dialog box on the Assemble command bar. When the Reduced Steps option is set, you specify the placement part and target part by selecting a face on each part. This reduces the number of steps from five to three for a typical mate relationship. There is some trade-off when using this option. Since the part in the assembly is no longer selected as a separate step, surfaces or cylinders on every active part are available for selection.

In large assemblies or in assemblies with numerous overlapping parts, positioning one part precisely to another can prove time consuming. In such cases, use QuickPick to filter the selection process.

### Note

When this option is set, you must specify the offset type and offset value before selecting the target face. If you want to use a reference plane on the target part to position the placement part, you must display the reference planes first.

## Capture fit

The **Capture Fit command** captures the assembly relationships and faces used to position a part or subassembly in the active assembly. When you place the part or subassembly again, you simply select the faces on a new target part already in the assembly to position the new part or subassembly. This reduces the number of steps required to position the part.

If you used the Insert option to position a part, the Capture Fit command will capture a mate and an axial align relationship, since these are the relationships that the Insert option actually places.

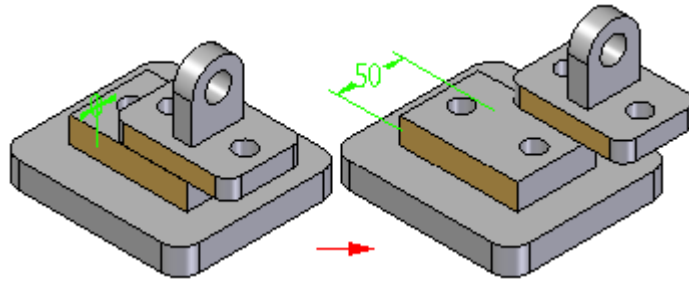
You can also capture relationships by setting the Automatically Capture Fit When Placing Parts option on the Options dialog box on the Assemble command bar.

### Note

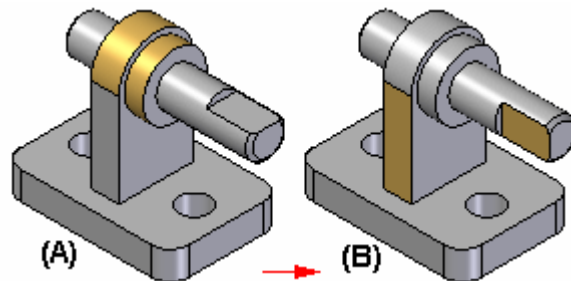
The Capture Fit command cannot capture Angular relationships.

## Defining offset values

Some relationships let you define fixed or floating offsets between the parts, such as the mate and align relationships. To specify an offset type, click one of the two offset buttons on the command bar. When you specify a fixed offset, you can type a dimensional value for the offset distance. For example, when you define a fixed offset for a planar align relationship, you can edit the value so that the parts are no longer co-planar.



A floating offset is useful when you need to control the orientation of a part with respect to another part, but it is impossible to define a fixed dimensional value. For example, you can use a floating offset to control the rotational orientation of a part. When you apply an axial align relationship using the Unlock Rotation option between a cylindrical shaft and the cylindrical face on another part (A), you can then use a planar align relationship, with a floating offset (B), to control the rotational orientation of the shaft.



If you try to apply a fixed offset for the planar align relationship, a message is displayed, saying that the fixed option conflicts with another relationship.

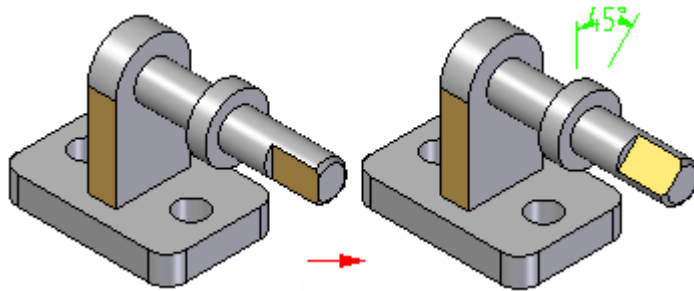
### Note

The range offset command is not intended to be used to for geometric tolerances. Depending on the relationships used to position the part are defined, this may result in an over constrained condition and cause errors.

### Locking and unlocking rotation on axial align relationships

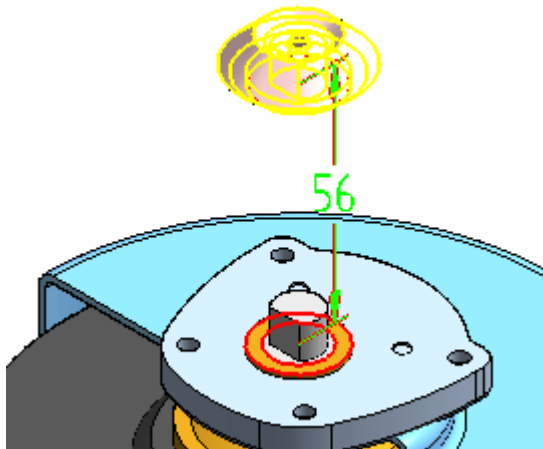
When you apply an axial align relationship, you can use the Lock Rotation and Unlock Rotation buttons on the command bar to specify whether the part is free to rotate around the axis of rotation. The Lock Rotation option is useful when the rotational orientation of the part is not important, such as placing a bolt into a hole. When you set the Lock Rotation option, the rotational orientation of the part is locked at a random position, but one less relationship is required to fully position the part.

When you set the Unlock Rotation option, you can define the rotational orientation you want by applying another relationship. For example, you can apply an angle relationship.



### Assembly relationship dimensions

When positioning parts using assembly relationships, driving or driven dimensions are created and displayed when appropriate. For example, when you position a part using a mate relationship with a fixed offset, a driving dimension is created.



When you position a part using a mate relationship with a floating offset (the offset value is controlled by another relationship), a driven dimension is created, which cannot be edited to reposition the part. Zero and negative value dimensions are supported.

When you apply or edit an assembly relationship, you can select and edit the driving dimension to change the offset value. You can use the Show All Dimensions option on the Options dialog box on the command bar to control whether the dimensions are displayed or hidden. When you set this option, dimensions display when you select a part and then click the Edit Definition button on the command bar. You can then select a dimension and edit its value using the command bar. When you clear this option, no dimensions are displayed, except for when you select a relationship in the bottom pane of PathFinder. When you select a relationship in the bottom pane of PathFinder, the dimension is displayed and selected, which allows you to edit its value using the command bar.

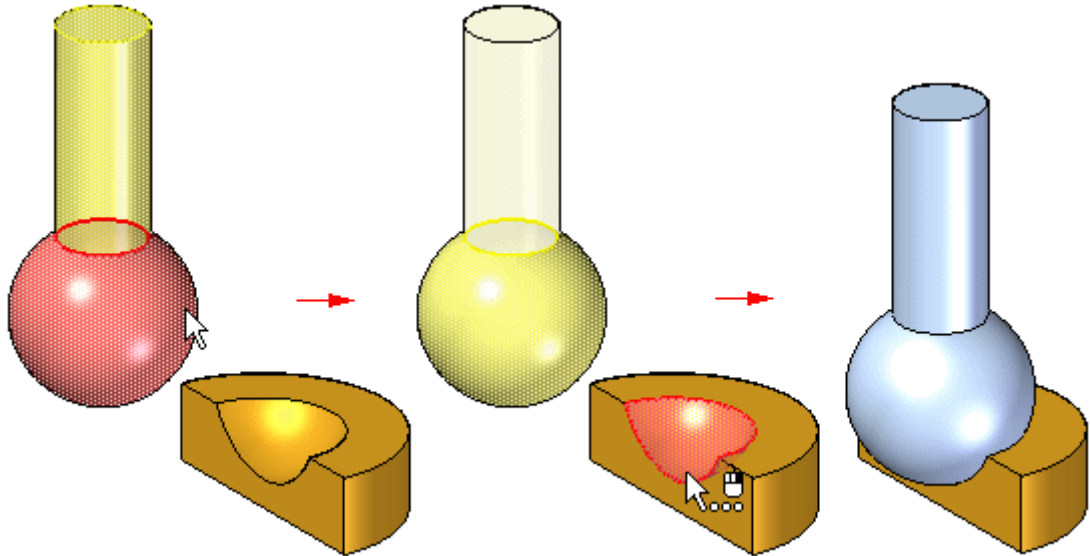
Dimensions are only created when appropriate for the relationship options you are using. Dimensions are created when using the mate, planar align, connect, angle, tangent, and parallel relationships. Dimensions are not created when using axial align, ground, or cam relationships.

### **Assembly Relationship Assistant command**

When working with assemblies whose parts are oriented correctly, but do not have assembly relationships, such as assemblies that were imported into Solid Edge from another CAD system, you can use the Assembly Relationship Assistant command to apply relationships between the parts and subassemblies. The relationships are applied based on their current geometric orientation. For more information, see the Assembly Relationship Assistant command Help topic.

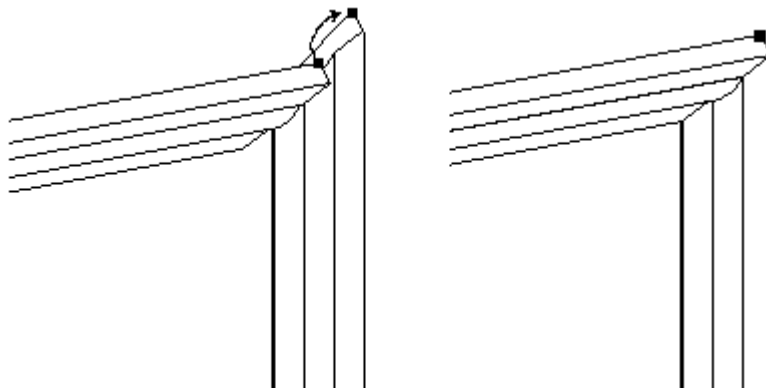
## Applying a connect relationship

When two parts in an assembly cannot be positioned properly by mate and align relationships, you can position them using connect relationships. A connect relationship positions a keypoint on one part with a keypoint, line, or face on another part. For example, you can apply a connect relationship to position the center of a spherical face on one part with respect to a spherical face on another part.

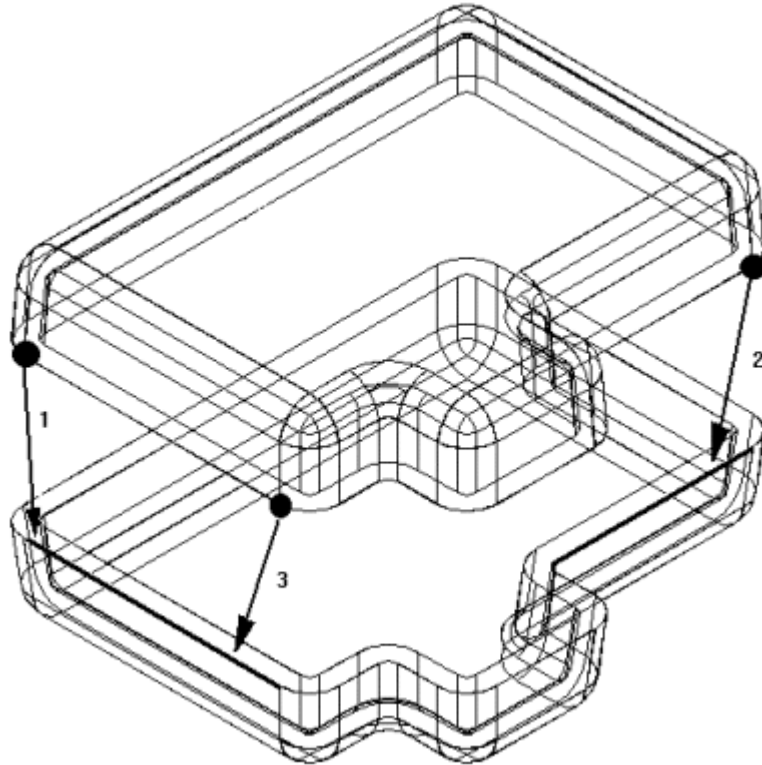


You can use the following methods to apply connect relationships:

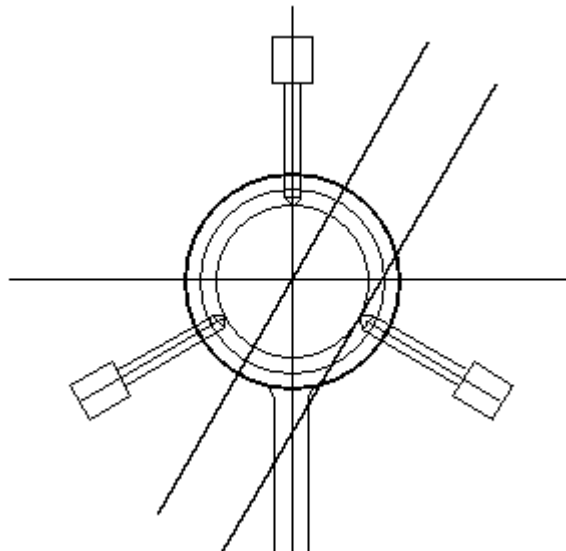
- **Point-to-Point:** In the following figure, a mate relationship is applied between the mitered corners of the parts. A connect relationship, which ties a point on one part to the appropriate point on another, connects the two corners properly. A floating align relationship between the back surfaces fully positions the part.



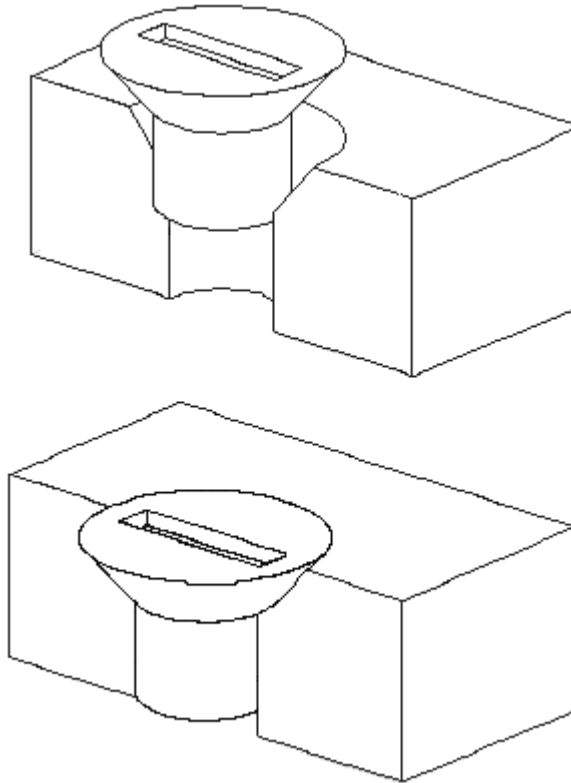
- **Point-to-Line:** In the following example, a mate relationship is applied between the faces of the two parts. Because the sides of each part are drafted, there are no part faces which you can use to apply a planar align relationship. You can apply three connect relationships between the keypoints on the top part and the linear edges on the bottom part.



- **Point-to-Plane:** In the following example, the lower right pin is positioned to a depth that just touches the surface of a reference plane.



- **Cone-to-Cone:** In the following example, the cone on the fastener is connected to the cone on the countersunk hole on the plate. When you add a connect relationship between two conical faces, the keypoint that represents the theoretical intersection of the individual cones are connected. You can also apply an offset value to a connect relationship between two conical faces.



### Recognizable elements for the connect relationship

#### Keypoints

- End points of lines, arcs, and ellipses
- Midpoint of a line (edge centerline)
- Arc center point
- Circle center point
- Elliptical element center point
- Spherical surface center point
- Conical surface center point



### **Lines**

- Linear edges (including tangent edges)
- Reference axis

### **Surfaces**

- Planar part surfaces
- Reference planes

### **Connect relationship combinations**

- Connect a point on the first part to a point on the second part
- Connect a point on the first part to a line on the second part
- Connect a point on the first part to a face on the second part
- Connect a line on the first part to a point on the second part
- Connect a face on the first part to a point on the second part

#### **Note**

You can also use an axial align or a connect relationship to position a part with respect to a keypoint or a line in an assembly sketch.

### **Differences between assembly relationships and sketch relationships**

The relationships that you apply between the parts and subassemblies of an assembly differ from the relationships that you apply while working with part sketches. For example:

- There are no relationship handles added to the assembly to show that a relationship has been applied. Instead, the relationships between parts are shown in PathFinder.
- Except for the ground relationship, all assembly relationships are defined between the part or subassembly you are placing and a part or subassembly already placed in the assembly.
- You cannot use dimensioning commands to place relationships between parts and subassemblies in an assembly.

**Positioning parts using coordinate systems**

You can also position parts in an assembly using coordinate systems. You do this by first defining coordinate systems in the part document on both the placement and target parts. You can then use the Planar Align relationship, Mate relationship, and Match Coordinate Systems option on the Assemble command bar to position the placement part.

For example, with the Match Coordinate Systems option, the placement part is positioned using planar align relationships that match the three principal axes on the coordinate system for the placement part and the target part. This allows you to position the placement part using fewer steps than applying three separate planar align relationships. This can be useful when working with a common part that is placed into an assembly multiple times in the same position relative to a target part.



## **Capture Fit command**

Captures the assembly relationships and faces used to position a part or subassembly already placed in the assembly. You can then place the part or subassembly again later using fewer steps. You can use the Capture Fit dialog box to specify which relationships you want to capture.

If you used the Insert option to position a part, the Capture Fit command will capture a mate and an axial align relationship, since these are the relationships that the Insert option actually places.


You can also capture relationships by setting the Automatically Capture Fit When Placing Parts option on the Options dialog box on the Place Part command bar.

### **Note**

The Capture Fit command cannot capture angular relationships.

Capture Fit dialog box

## Capture the assembly relationships for a part

1. In the assembly window, select a part for which you want to capture relationships.
2. Choose Home tab® Relate group® Capture Fit .
3. In the Capture Fit dialog box, use the Add and Remove buttons to specify which relationships you want to capture, and then click OK.

### Tip

- You can also select the part you want to use in PathFinder.
- When you use the Capture Fit command, the relationships and faces used to position the part or subassembly the first time are stored so you can place the part using fewer steps later.
- If you used the Insert option to position a part, the Capture Fit command will capture a mate and an axial align relationship, since these are the relationships that the Insert option actually places.
- The Capture Fit command cannot capture angular relationships.
- You can also capture relationships by setting the Automatically Capture Fit When Placing Parts option on the Options dialog box on the Place Part command bar.

## **Activity: Positioning parts with mate, axial align, planar align and insert**

### **Overview**

This activity shows the process of positioning parts using mate, axial align, planar align and insert. The parts will be positioned with the reduced steps option turned off to better understand the workflow options in the command bar. Then the same parts will be placed with the reduced steps option turned on to show how the process can be streamlined.

### **Note**

FlashFit is a preferred method of quickly positioning parts in an assembly and will be covered in another activity. This activity forces you to manually position parts so that you will understand what is occurring when parts are positioned using flashfit and how to change a single relationship to reposition a part if an edit is later required.

### **Objectives**

Parts will be added to an assembly using the commands mate, planar align, axial align and insert

In this activity you will:

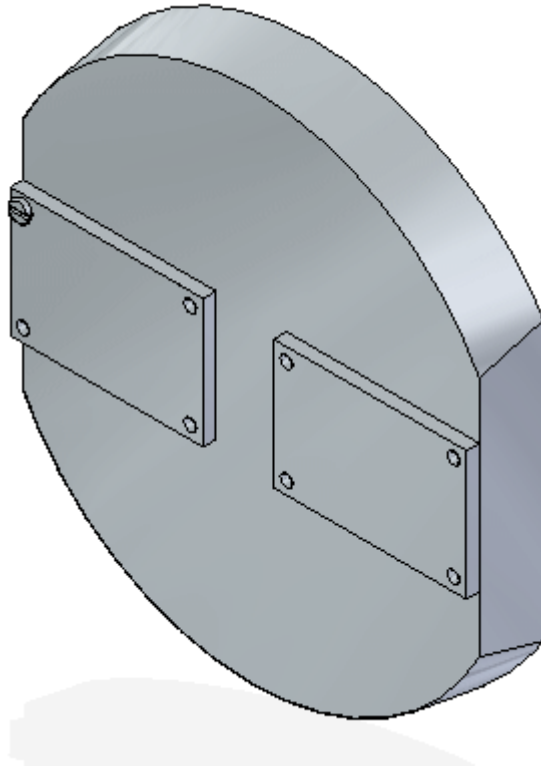
- Learn how to position parts using the commands mate, axial align, planar align and insert without using reduced steps.
- Learn how the command bar reflects the workflow during the positioning of parts.
- Use reduced steps to position parts with mate, planar align, axial align, and insert.

### **Activity**

In this activity you will learn the procedure for positioning parts in an assembly using the mate, planar align and axial align relationships.

**Activity: Placing parts using mate, planar align, and axial align**

This activity will position a part with reduced steps turned off to show the complete sequence of steps involved in positioning a part. The second part will be placed with reduced steps turned on to show a more efficient method of positioning parts.

*Create a new assembly and position the first part*

In this step, you will create a new assembly using the Synchronous ISO Assembly template. You will click the Parts Library on pathfinder and browse to the folder containing the assembly class files.

- ▶ Create a new assembly. After the assembly opens, click the Application button.



Choose Solid Edge Options, and then click the assembly tab. Check the box as shown.

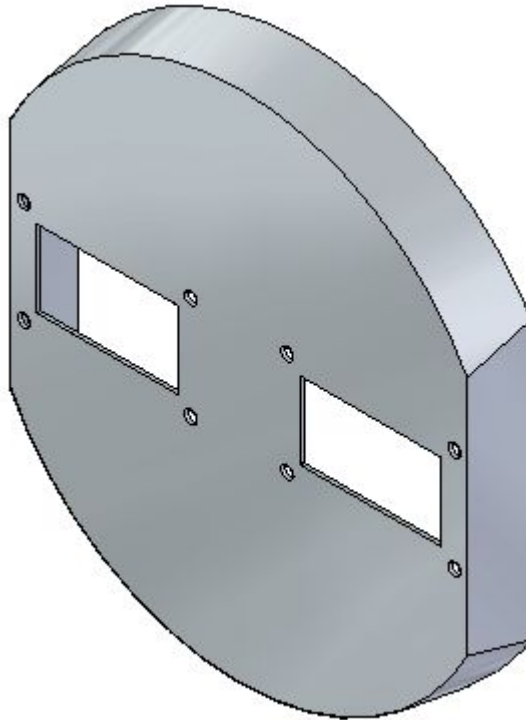
Do not create new window during Place Part

- ▶ From the Parts Library in PathFinder, drag the part *dome.par* in the assembly window.



**Note**

The first part placed in the assembly is placed as a grounded part.



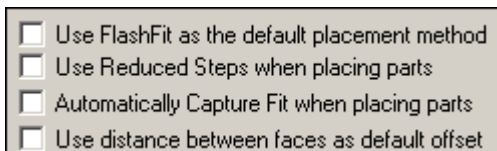
*Applying a mate relationship*

In this step you will drag the part *a1\_part.par* into the assembly window and apply a mate relationship.

- ▶ Click the Options button on the command bar.



- ▶ Set the options shown and then click OK. Make sure the reduced steps option is *off* and FlashFit as the default placement method is *off*.



- ▶ Select the Mate Relationship.

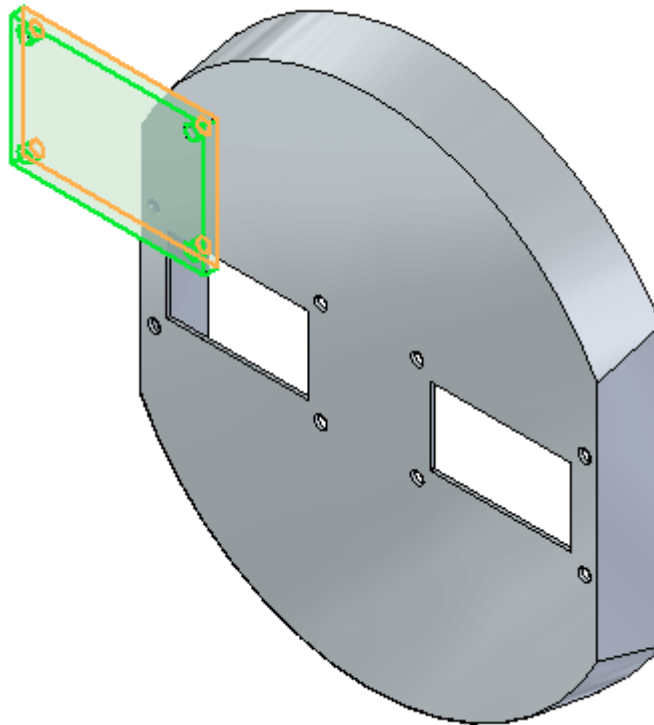


**Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. Notice the step is currently the Placement Part - Element step and you are being prompted to select an element of the placement part. For this relationship, select a face.



- ▶ Select the face shown.



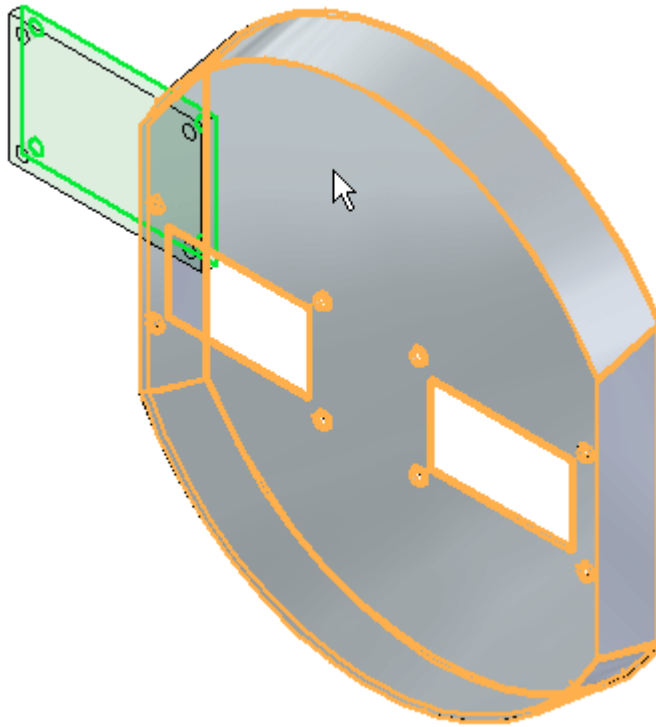
**Note**

The command bar shows the Target Part step is active, and you are being prompted to select the target part. This part has the face you will apply the mate relationship to. If you selected the wrong face in the previous step, you can back up by clicking the button corresponding to that step and then select the proper geometry again.



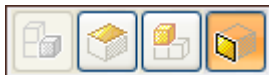


- ▶ Select the target part, *dome.par* as shown.

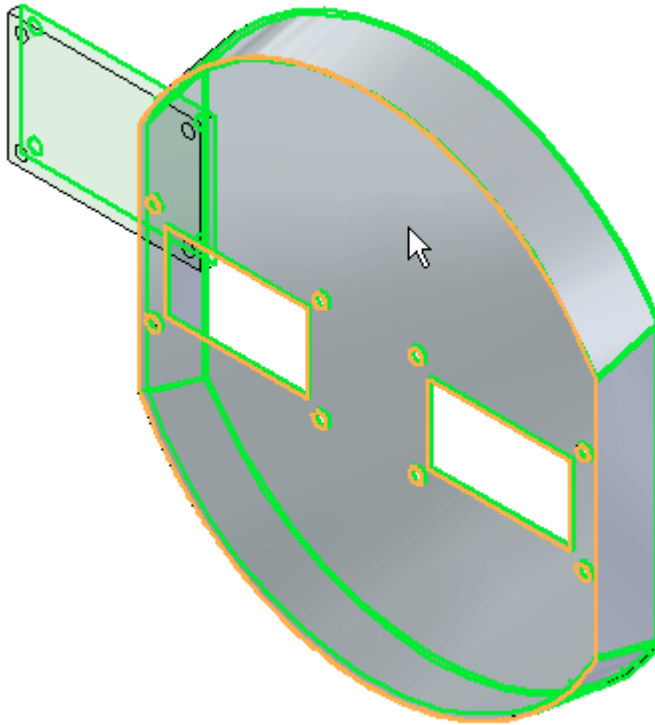


**Note**

Notice the Target Part - Element step is active, and you are being prompted to select the target part element. This element is the face the mate relationship will be applied to.



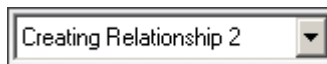
- ▶ Select the face shown on *dome.par*.



- ▶ Right-click or click the OK button to accept. The mate relationship is applied.

#### *Applying a planar align relationship*

Once this relationship is established, the relationship list increments to the next relationship. Relationship 2 will be a planar align.



- ▶ Set the relationship type to Planar Align.

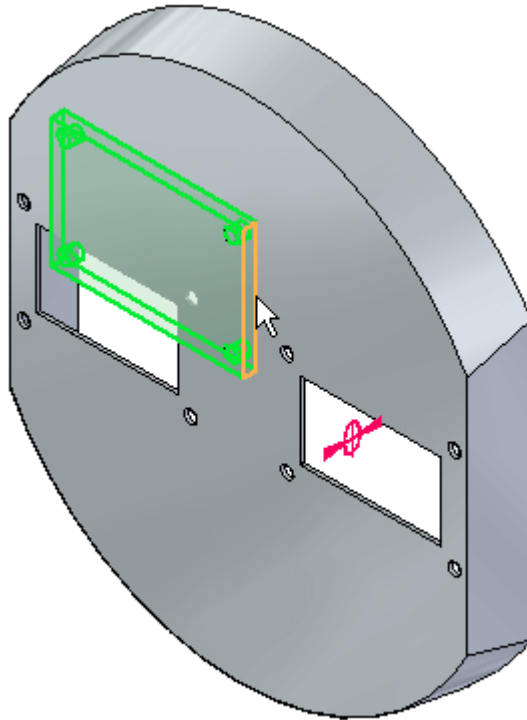


#### **Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. For this relationship, you will select a face.



- Select the face shown.

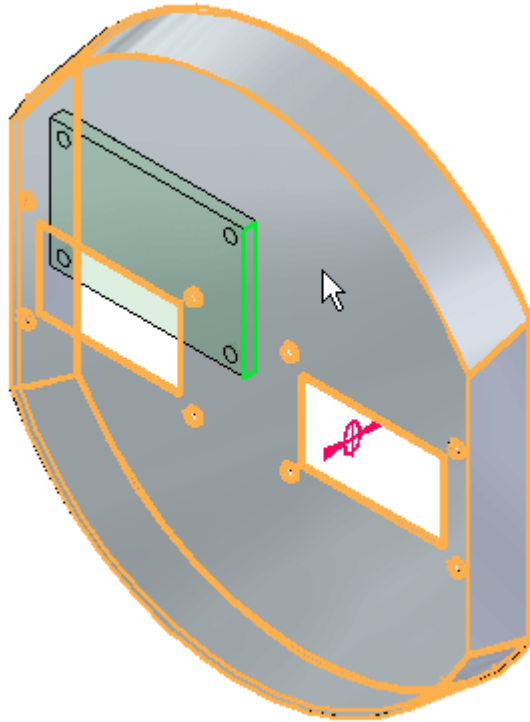


**Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. This part has the face you will apply the planar align relationship to.



- Select the target part shown.

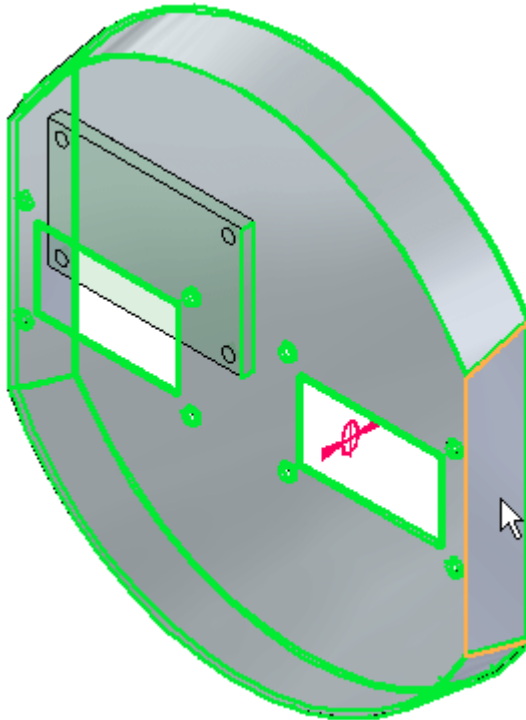


**Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. Select the element that is the face you will apply the planar align relationship to.



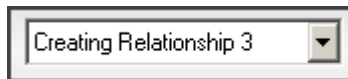
- ▶ Select the face shown.



- ▶ Right-click or click the OK button to accept. The planar align relationship is applied.

*Applying an axial align relationship*

The relationship list increments to the next relationship. Relationship 3 will be an axial align.



- ▶ Set the relationship type to Axial Align

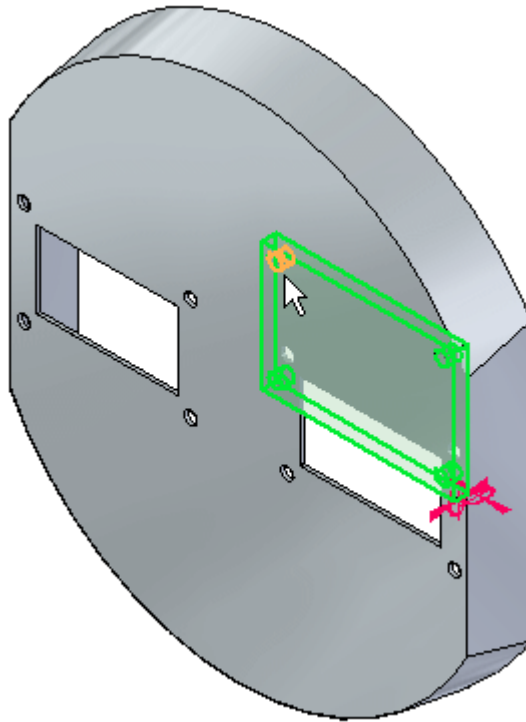


**Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. For this relationship, you will select a cylindrical face.



- ▶ Select the cylindrical face shown.

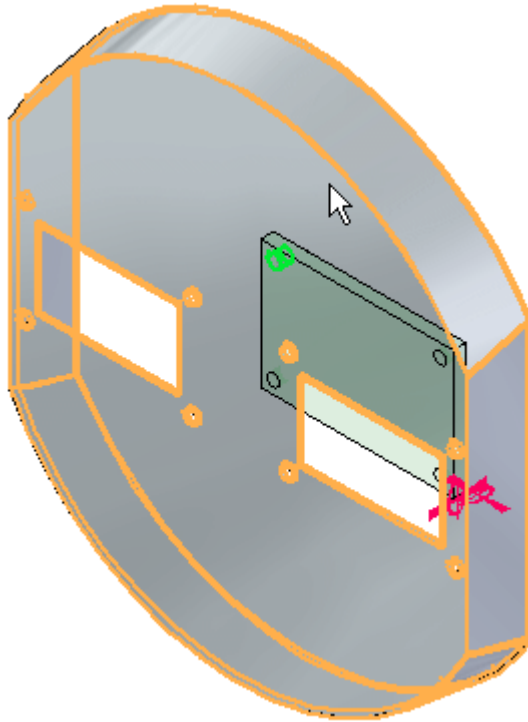


**Note**

The Locate Steps group on the command bar reflects the current placement step in the workflow. This part has the cylindrical face you will apply the axial align relationship to.

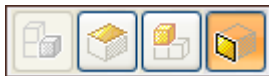


- Select the target part shown.

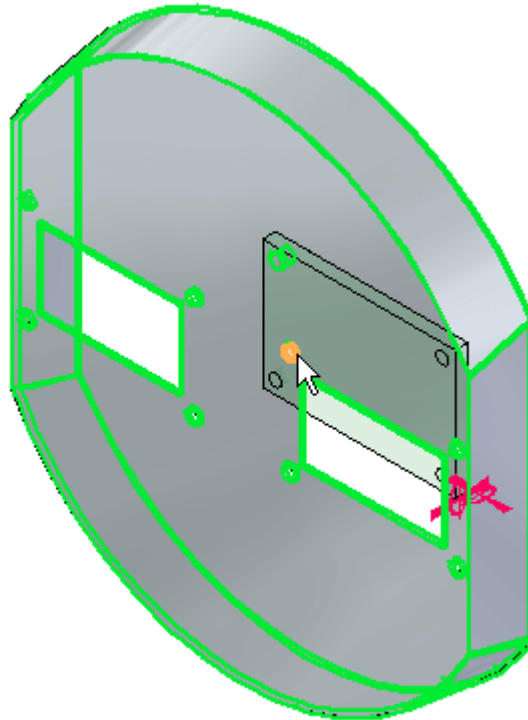


**Note**

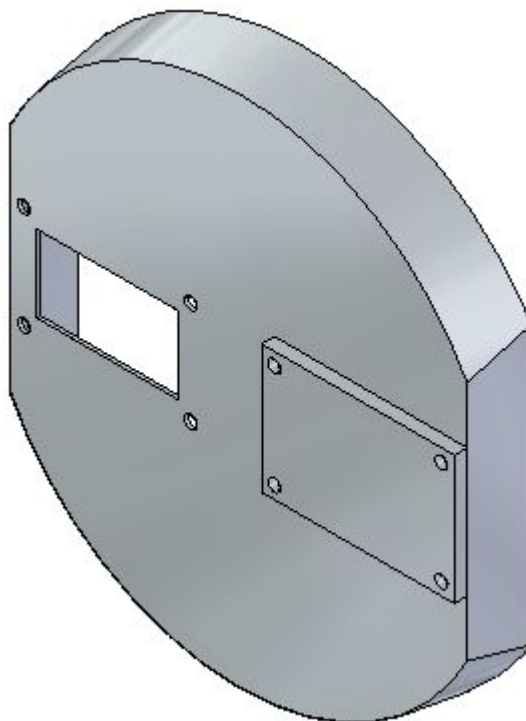
The Locate Steps group on the command bar reflects the current placement step in the workflow. This element is the cylindrical face you will apply the axial align relationship to.



- ▶ Select the cylindrical face shown.



- ▶ Right-click or click OK to accept. The axial align relationship is applied, and the part is fully positioned.





*Applying a mate relationship with reduced steps*

Another occurrence of the part *a1\_part.par* will be placed. The sequence of steps will be the same except reduced steps will be used.

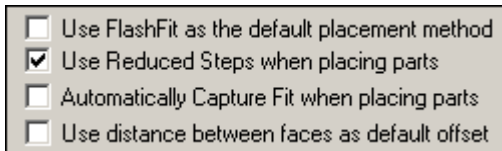
**Note**

When the reduced steps option is used, the step for selecting the target part is eliminated. Valid features on every part are available for selection and the target part is determined from the part containing the feature. This option is more efficient in most cases, however in large assemblies where an area can be congested with many parts, it is desirable to have more control by manually choosing the target part as was shown in the previous steps.

- ▶ From the Parts Library, drag the part *a1\_part.par* into the assembly window. You will apply a mate relationship.
- ▶ Click the Options button on the command bar.



- ▶ Set the options shown. Make sure the reduced steps option is *on* and FlashFit as the default placement method is *off*.



- ▶ Select the mate relationship.

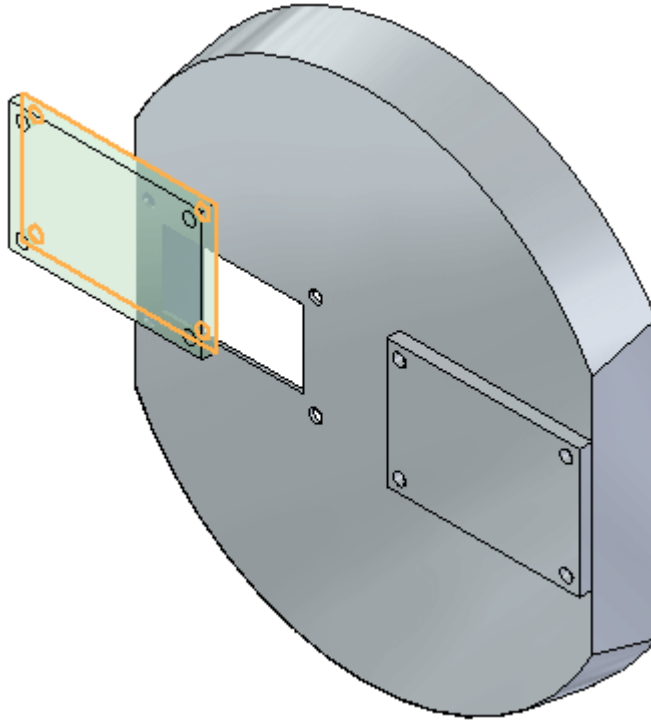


**Note**

The command bar reflects the placement step in the workflow. Notice the step is currently the element step and you are being prompted to select an element of the placement part. For this relationship, you will select a face.



- ▶ Select the face shown.

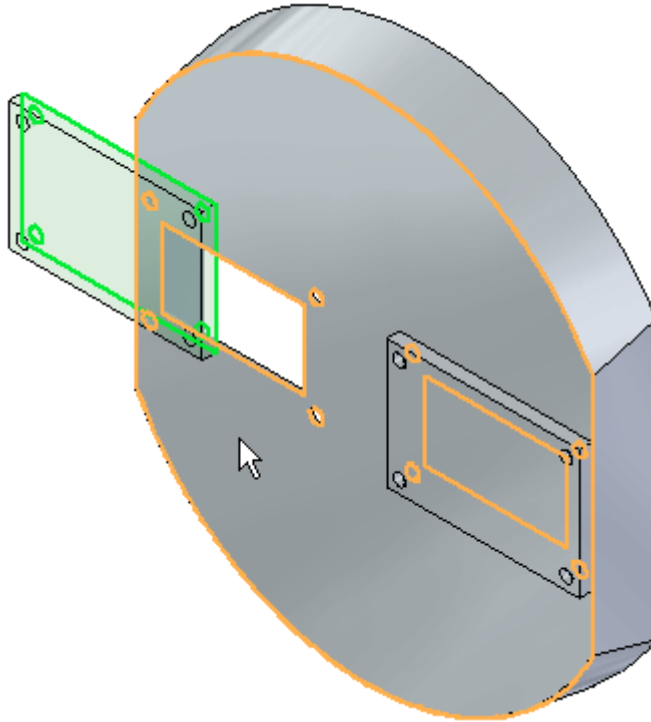


**Note**

Because the reduced steps option is set, the command bar reflects the placement step in the workflow. Notice the step is now the target part element, and you are being prompted to select the target part element. This element is the face the mate relationship will be applied to. The target part is automatically assigned and is the part on which the target element belongs.



- ▶ Select the face shown.



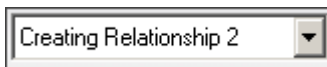
- ▶ The mate relationship is applied.

**Note**

Using reduced steps, there is no need to click OK to complete. Once the target element is selected, the relationship is established.

*Applying a planar align relationship with reduced steps*

Once this relationship is established, the relationship list increments to the next relationship. Relationship 2 will be a planar align.



- ▶ Set the relationship type to Planar Align.



**Note**

The command bar reflects the placement step in the workflow. Notice the step is currently the element step and you are being prompted to select an element of the placement part. For this relationship, you will select a face.



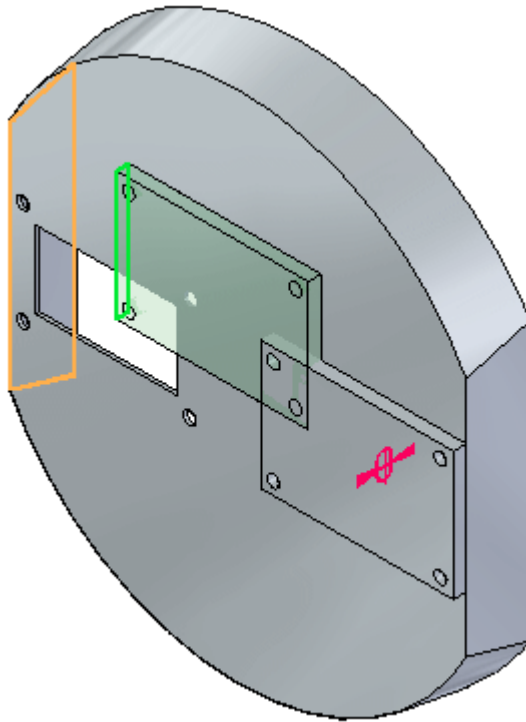
- ▶ Select the face shown.

**Note**

The command bar reflects the placement step in the workflow. Notice the step is now the target part element, and you are being prompted to select the target part element. This element is the face you will apply the planar align relationship to.



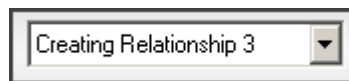
- ▶ Select the target part element shown.



The planar align relationship is applied.

*Applying a axial align relationship with reduced steps*

Once this relationship is established, the relationship list increments to the next relationship. Relationship 3 will be an axial align.



- ▶ Set the relationship type to Axial Align

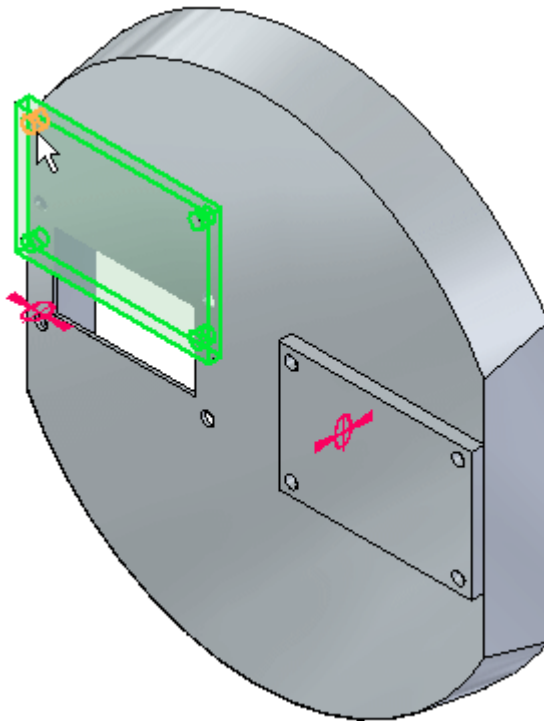


**Note**

The command bar reflects the placement step in the workflow. Notice the step is currently the element step and you are being prompted to select an element of the placement part. For this relationship, you will select a cylindrical face.

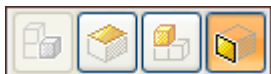


- ▶ Select the cylindrical face shown.

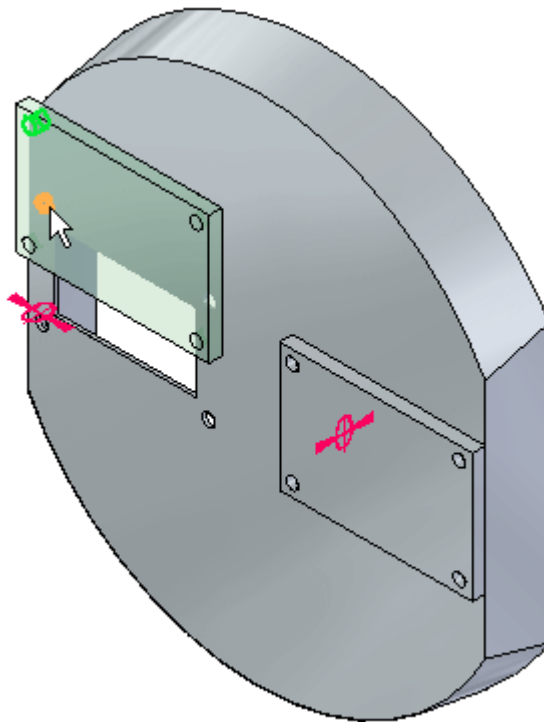


**Note**

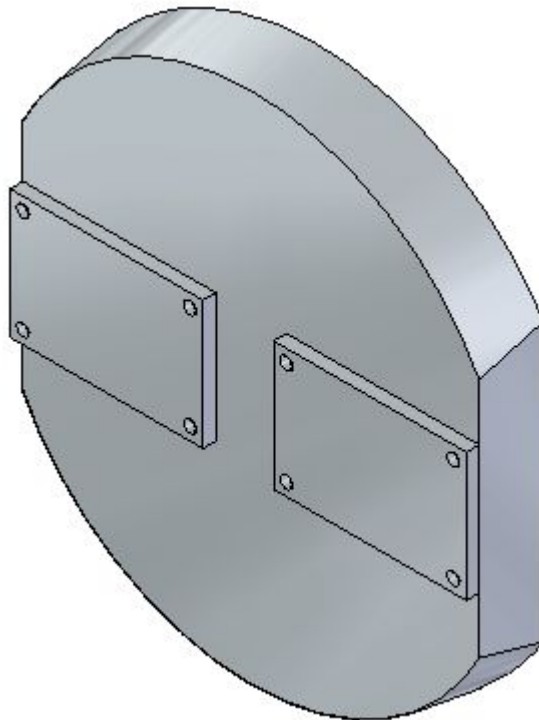
The command bar reflects the placement step in the workflow. Notice the step is now the target part element, and you are being prompted to select the target part element. This element is the cylindrical face you will apply the axial align relationship to.



- Select the cylindrical face shown.



The axial align relationship is applied, and the part is fully positioned.



*Placing a fastener with the Insert command*

Insert will be used to position a fastener in a hole.

**Note**

Insert requires a mate and an axial align. Once you have established these relationships, the rotation of the axial align is locked and the part is fully positioned.

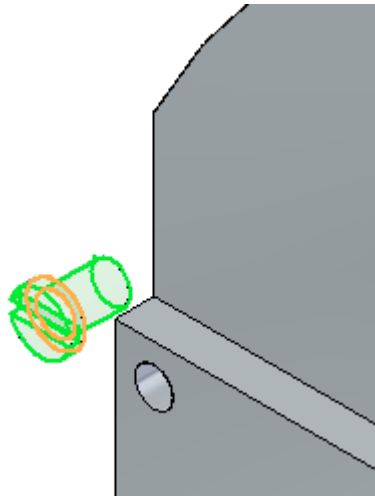
- ▶ Drag the part *10mm\_fastener.par* into the assembly window.
- ▶ Select the Insert Command.



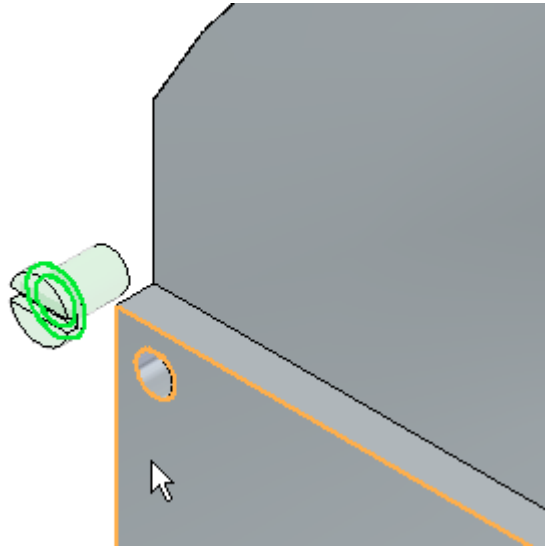
**Note**

The mate relationship will be established first, then the axial align. Because of the number of faces to choose from, Quickpick will be used to aid in the selection.

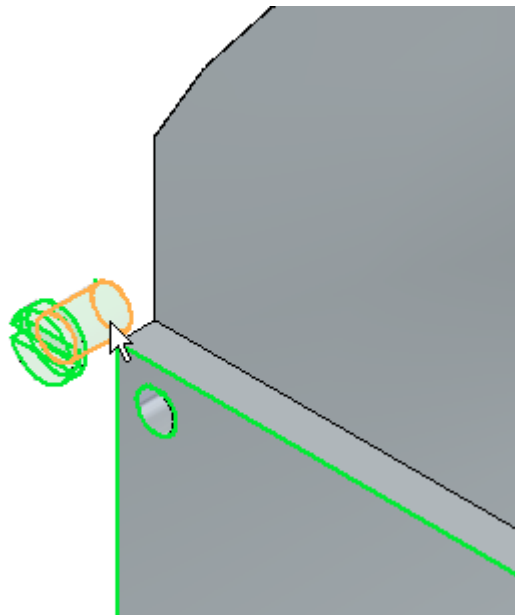
- ▶ For the mate relationship, select the face shown.



- ▶ Select the target face for the mate relationship as shown.

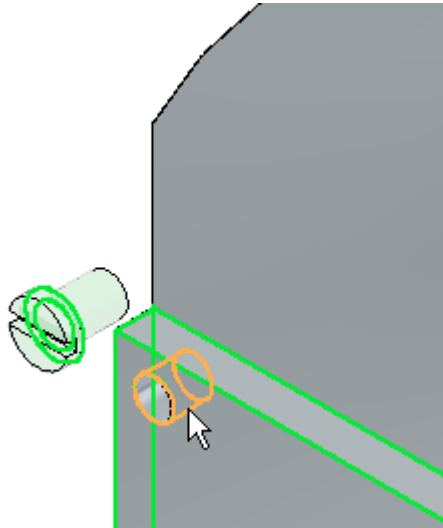


- ▶ For the axial align relationship, select the cylindrical face shown.

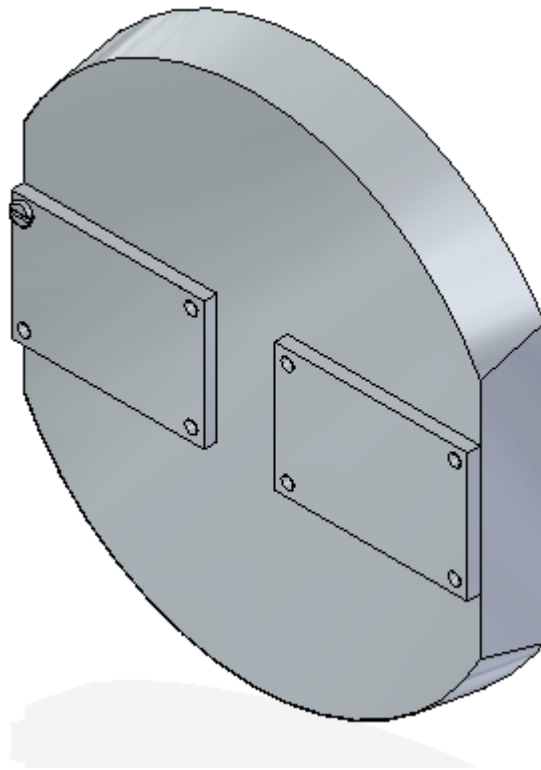




- ▶ For the target face of the axial align, select the face shown.



- ▶ The fastener is placed and fully positioned, with the rotation locked. Click the Select tool to exit. Close the assembly document without saving.



#### Activity summary

In this activity you learned the workflow for establishing relationships needed to position parts in an assembly. You also learned that using the reduced steps option streamlines the process of positioning parts.

**Lesson review**

Answer the following questions:

1. What are the steps required to apply a mate relationship without using flashfit or reduced steps?
2. What is the difference between mate and planar align?
3. What is a floating offset?
4. Are linear edges valid for the placement of an axial align relationship?
5. When using reduced steps, which step is eliminating when creating a relationship?

## **Answers**

1. What are the steps required to apply a mate relationship without using flashfit or reduced steps?
  - a. Select the face to mate on the placement part.
  - b. Select the target part.
  - c. Select the face to mate to on the target part.
  - d. Select the face to mate to on the target part.
  - e. Enter an offset distance (optional).
  - f. Click OK.
  
2. What is the difference between mate and planar align?

Mated faces face each other. Faces that are planar aligned face the same direction.
  
3. What is a floating offset?

A floating offset forces the faces to be parallel and the distance between the faces is determined by other relationships.
  
4. Are linear edges valid for the placement of an axial align relationship?

Linear edges can be used to define an axial align relationship, as well as cylindrical features, constant radius features such as rounds or fillets.
  
5. When using reduced steps, which step is eliminating when creating a relationship?

Valid features are found on all active parts and the selection of the target part is not required.

**Lesson summary**

In this lesson you learned the workflow for establishing relationships needed to position parts in an assembly. You also learned that using the reduced steps option streamlines the process of positioning parts.

## **Activity: Placing parts in an assembly using FlashFit**

### **Overview**

In this activity, FlashFit will be used to position parts in a valve assembly.

### **Objectives**

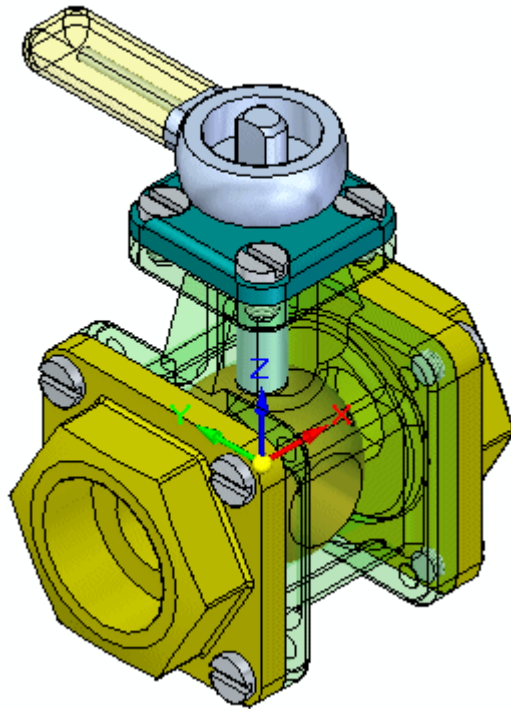
The objective of this activity is for you to be able to use appropriate relationships to position parts in an assembly.

### **Activity**

In this activity you will learn the procedure for positioning parts in an assembly using FlashFit to achieve the relationships of mate, planar align and axial align.

**Activity: Placing parts in an assembly with FlashFit**

You will use FlashFit to position parts and subassemblies in completing the valve assembly.

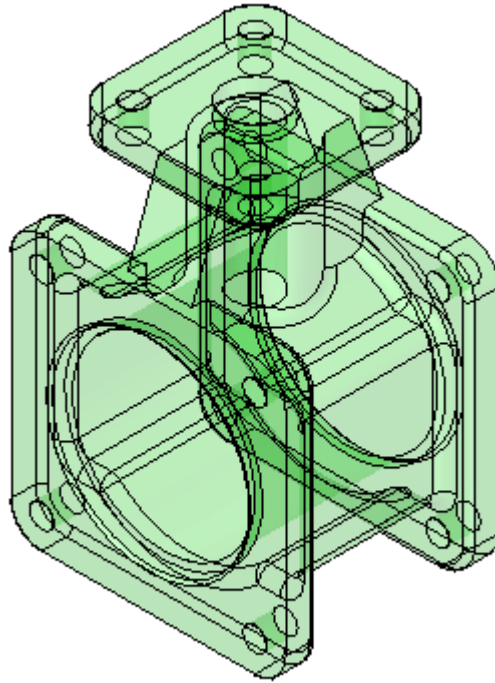


*Place the first part in the assembly.*

Create a new assembly and place the first part.

- Create a new assembly file.

- ▶ On the assembly PathFinder, click the Parts Library and drag *st\_v\_housing.par* into the assembly window. The first part placed in a new assembly file is grounded.



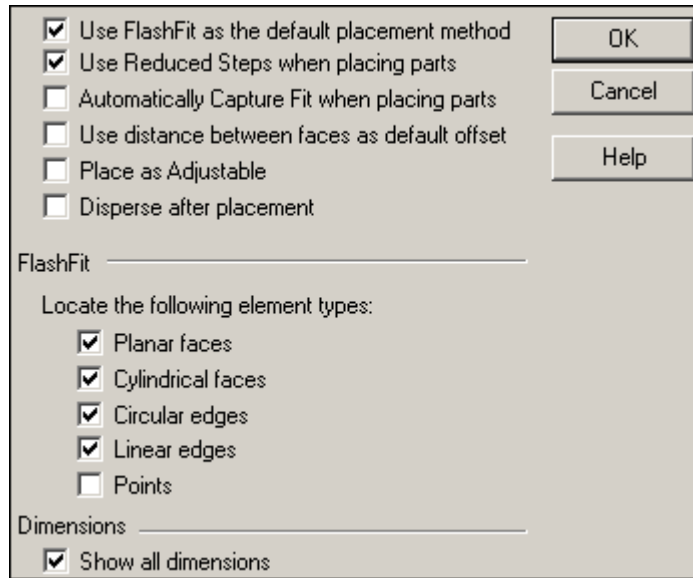
*Use FlashFit to position the valve parts and subassemblies*

Use FlashFit to position the valve parts. Before placing additional parts, set the FlashFit parameters. Once the parameters are set, the part will be positioned.

- ▶ From the Parts Library, drag the subassembly *st\_v\_handleball.asm* into the assembly window.
- ▶ Click the Options button on the command bar.



- ▶ Set the Options shown and then click OK.



- ▶ Set the relationship type to FlashFit.



- ▶ On the command bar, click the Activate Part button.

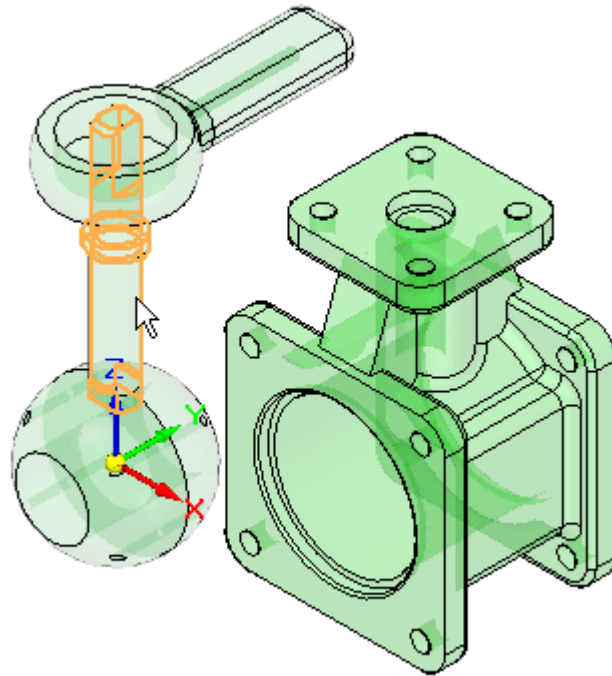


### Note

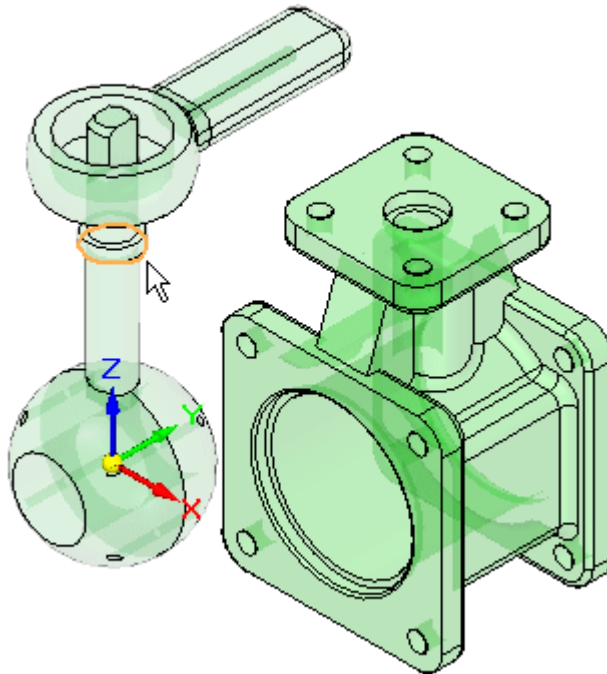
When positioning a subassembly using reduced steps, the parts making up the subassembly come in as inactive. The parts containing the geometry needed to position the subassembly need to be activated.



- Select *st\_v\_shaft.par* to activate it. Right-click to exit the Activate command and continue.



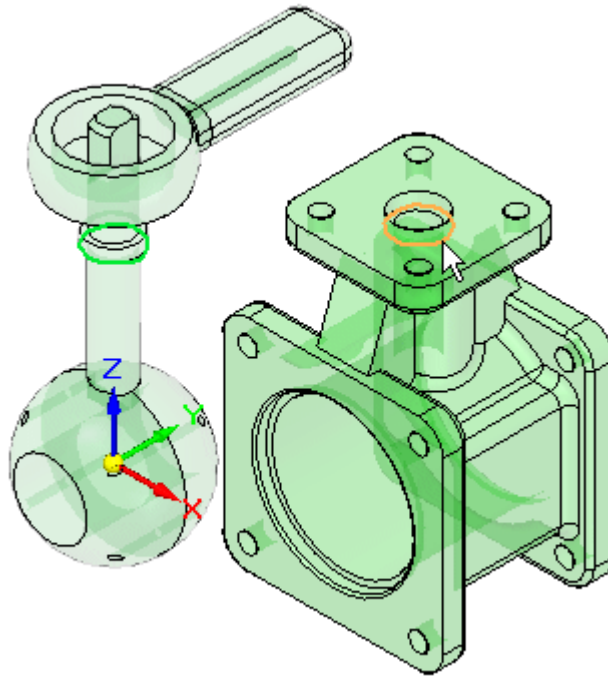
- ▶ Select the circular edge shown. Use QuickPick for an accurate selection.



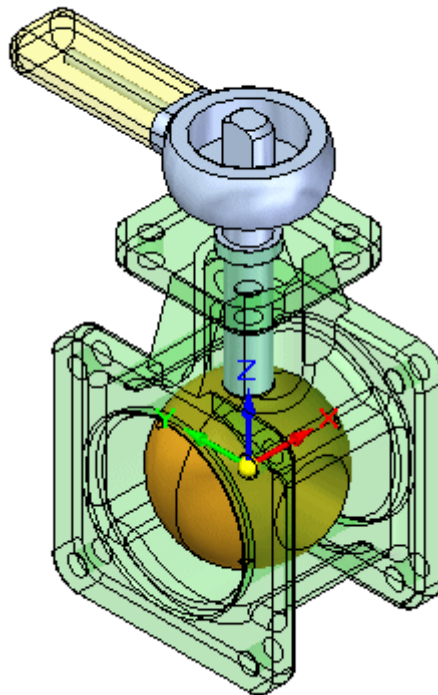
**Note**

Matching circular edges with FlashFit is the equivalent of using the Insert command. A Mate relationship, and an Axial Align relationship with locked rotation is created.

- ▶ Select the inner lip of the center hole of the housing.



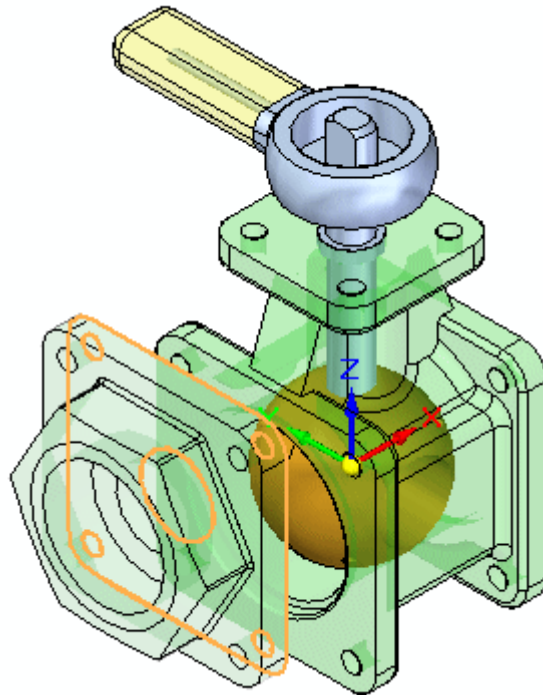
- ▶ The subassembly is positioned.



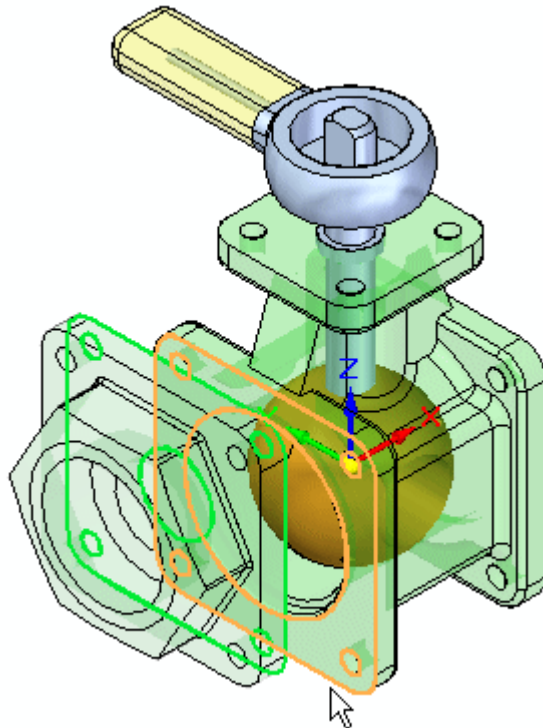
*Place the remaining parts*

Place additional parts in the assembly until finished.

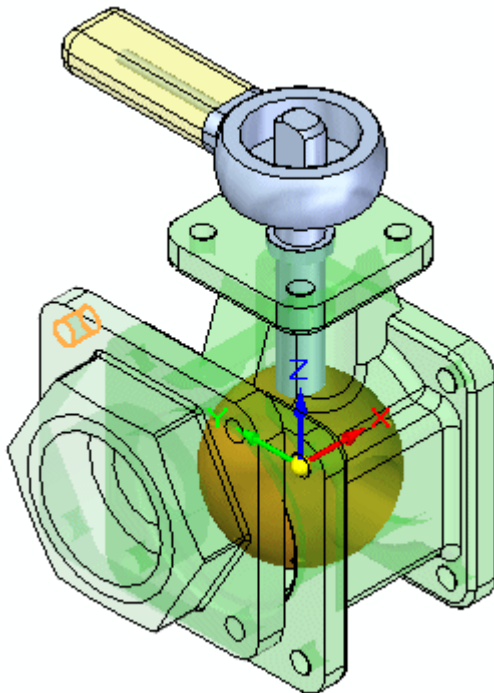
- Drag *st\_v\_endplate.par* into the assembly window.
- Use QuickPick to select the face shown.



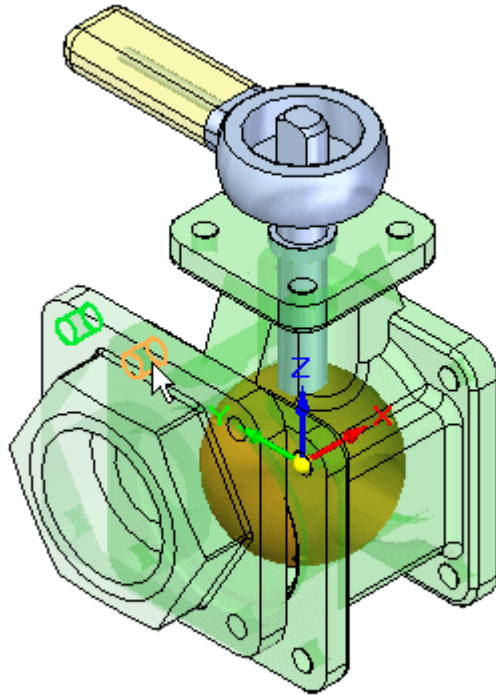
- ▶ Select the target face on the housing as shown. A Mate relationship is applied.



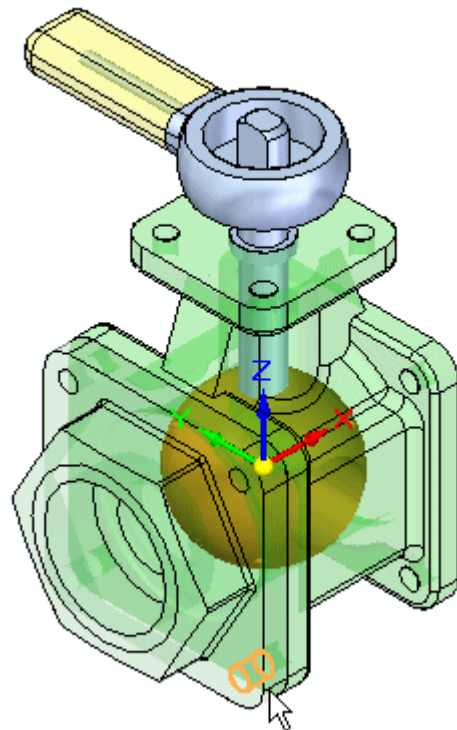
- ▶ The next two relationships will be established using alignment of holes in the parts. Select the cylindrical face on the *st\_v\_endplate.par* as shown.



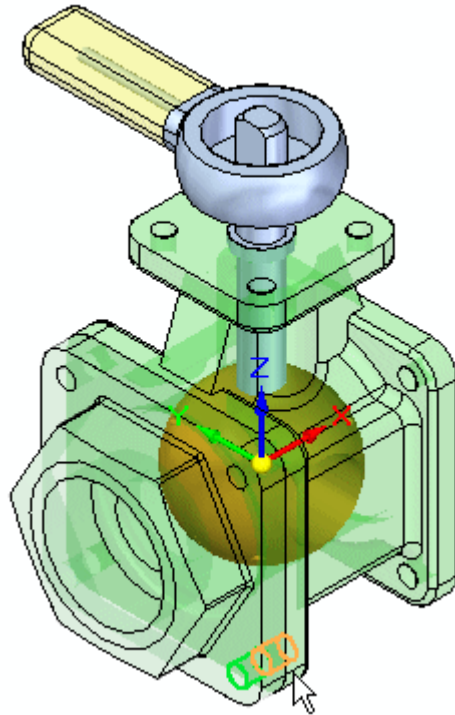
- ▶ For the target, select the cylindrical face shown. An Axial Align relationship is applied.



- ▶ For the last relationship needed to completely position the part, select the cylindrical face shown.



- ▶ For the target, select the cylindrical face shown. An Axial Align relationship is applied and the part is positioned.

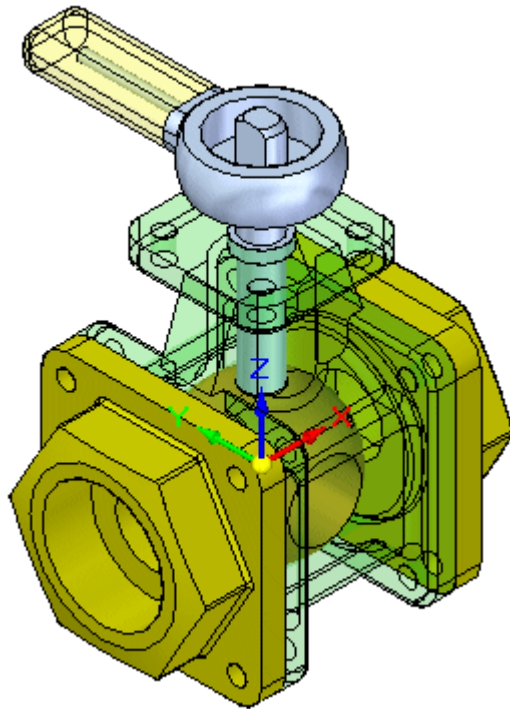
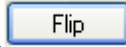


- ▶ Drag another occurrence of *st\_v\_endplate.par* into the assembly window.

- Place *st\_v\_endplate.par* on the opposite side of the housing from the one just place using the same procedure you used to place the previous part.

**Note**

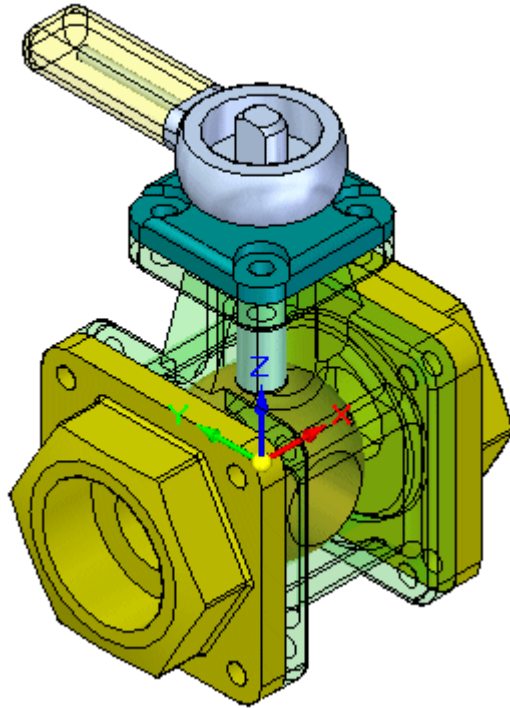
FlashFit will assign either a mate or a planar align to flat faces based on the closest orientation of the two faces being positioned. In this case, if a planar align is assigned rather than a mate, use the flip button to change the relationship type to a mate.



- Drag *st\_v\_top.par* into the assembly window.



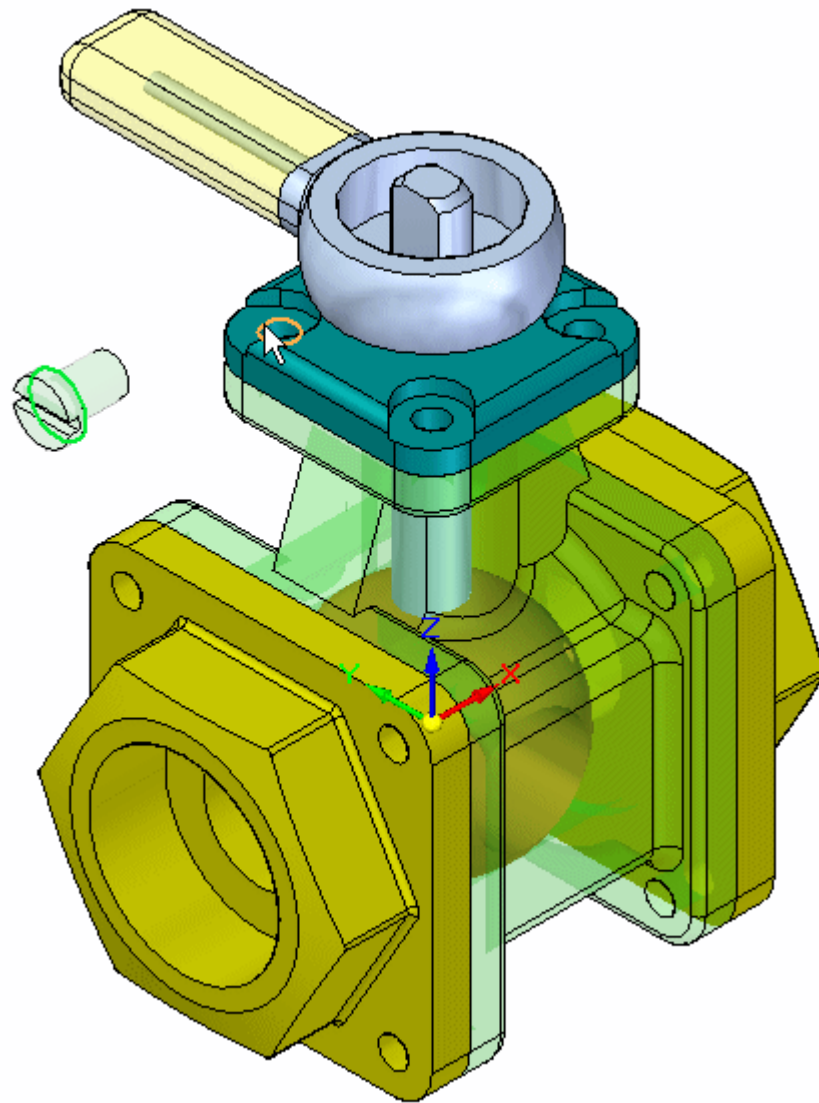
- ▶ Use FlashFit to position *st\_v\_top.par* as shown. The procedure is similar to the one used to place the previous two parts.



- ▶ Drag *10mm\_fastener.par* into the assembly window.
- ▶ Using QuickPick, select the circular edge shown.



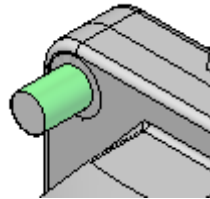
- For the target, select the circular edge shown on the top cap. The fastener is placed.



- Place additional occurrences of *10mm\_fastener.par* in the remaining holes on the valve using the same procedure.

**Note**

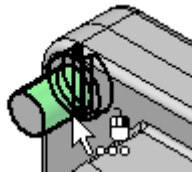
If FlashFit positions the fasteners incorrectly as shown, follow the steps outlined to correct the placement. The reason for the incorrect placement of the fastener is that FlashFit determines whether to apply a Planar Align or a Mate relationship based on the orientation of the face relative to the placement face. If the part faces are closer to a planar align relationship, then that is what is applied. Prior to selecting the circular edges in FlashFit, the fastener can be rotated into the approximate desired orientation by holding Ctrl while dragging it. This will result in correct placement and is easier than correcting the placement using Flip outlined below.



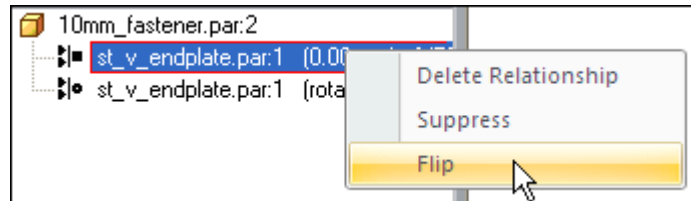
- Click the Select command.



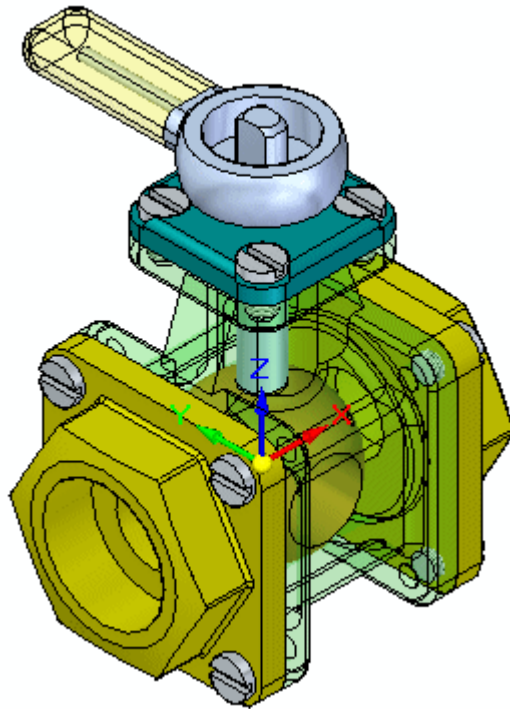
- Select the fastener.



- In the lower pane of PathFinder, right-click the Planar aAlign relationship, and then click Flip.



- This completes the activity. Close the assembly document without saving.



### *Activity Summary*

In this activity you learned how to place parts and subassemblies from a Parts Library and position them in an assembly. FlashFit consists of the Mate, Planar Align and Axial Align relationships and determines which is appropriate. When using FlashFit and selecting circular edges, fasteners can quickly be positioned because the rotation of the fastener is locked and the part becomes fully constrained.

## **Lesson review**

Answer the following questions:

1. Which relationships will flashfit attempt to create when positioning a part in an assembly?
2. When a circular edge is used on both the placement part and the target part, which relationships are created?
3. How does flashfit determine whether to place a mate or a planar align relationship when flat faces are selected?
4. If a planar align relationship is incorrectly created using flashfit, how can a mate relationship be assigned instead?
5. After a part is added from the parts library, how can it be approximately positioned to the target geometry?
6. If a relationship is created erroneously, how do you change or remove the relationship?

**Answers**

1. Which relationships will flashfit attempt to create when positioning a part in an assembly?

Flashfit creates a mate, a planar align or an axial align relationship.

2. When a circular edge is used on both the placement part and the target part, which relationships are created?

Using circular edges with flashfit is similar to the insert command and has fewer steps. Like the insert command, using circular edges with flashfit is ideal for the placement of fasteners. The edges are placed concentric creating a mate and axial align with the rotation locked.

3. How does flashfit determine whether to place a mate or a planar align relationship when flat faces are selected?

The closest orientation is chosen. If the faces are approximately facing each other, mate is selected. If the faces are oriented approximately in the same direction, a planar align relationship is created.

4. If a planar align relationship is incorrectly created using flashfit, how can a mate relationship be assigned instead?

Click the flip command before proceeding to the next positioning step.

5. After a part is added from the parts library, how can it be approximately positioned to the target geometry?

Drag the part to a new position with the left mouse button. Rotate the part by holding the CTRL key and dragging with the left mouse button.

6. If a relationship is created erroneously, how do you change or remove the relationship?

Edit the relationship by clicking on the part with the select tool. The relationships appear in the lower pane of assembly pathfinder and can be edited, suppressed or deleted from there.

## **Lesson summary**

In this lesson you learned how to place parts and subassemblies from a Parts Library and position them in an assembly. FlashFit consists of the Mate, Planar Align and Axial Align relationships and determines which is appropriate. When using FlashFit and selecting circular edges, fasteners can quickly be positioned because the rotation of the fastener is locked and the part becomes fully constrained.

## More Assembly Relationships

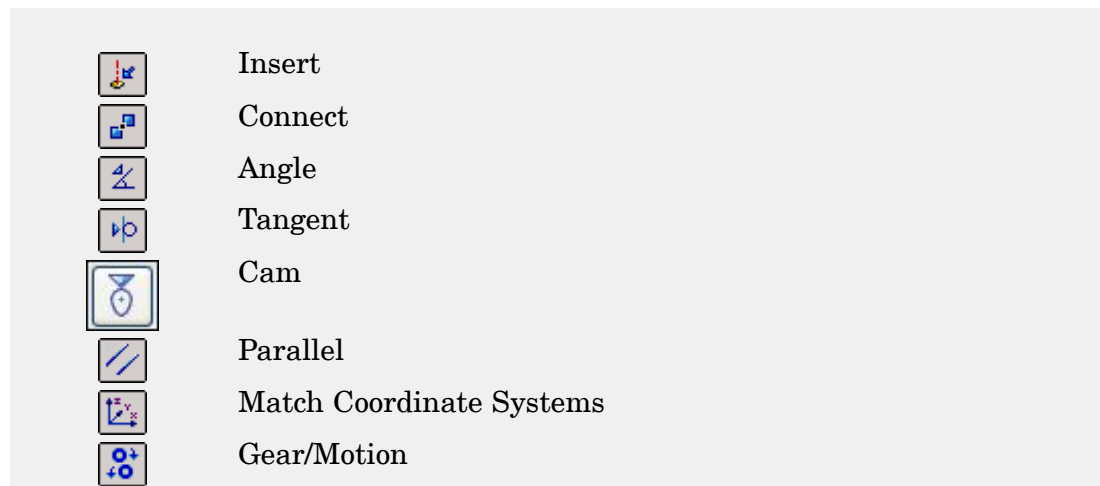
Positions the selected part or subassembly into the active assembly. Parts are positioned in assemblies using a combination of assembly relationships.

### Relationship List

Lists the relationships applied to the part. You can replace a previously applied relationship by selecting a relationship from the list and then selecting a new relationship from the Relationship Types list.

### Relationship Types

Lists the assembly relationship types you can apply. Parts can be positioned using the following relationship options:



### Note

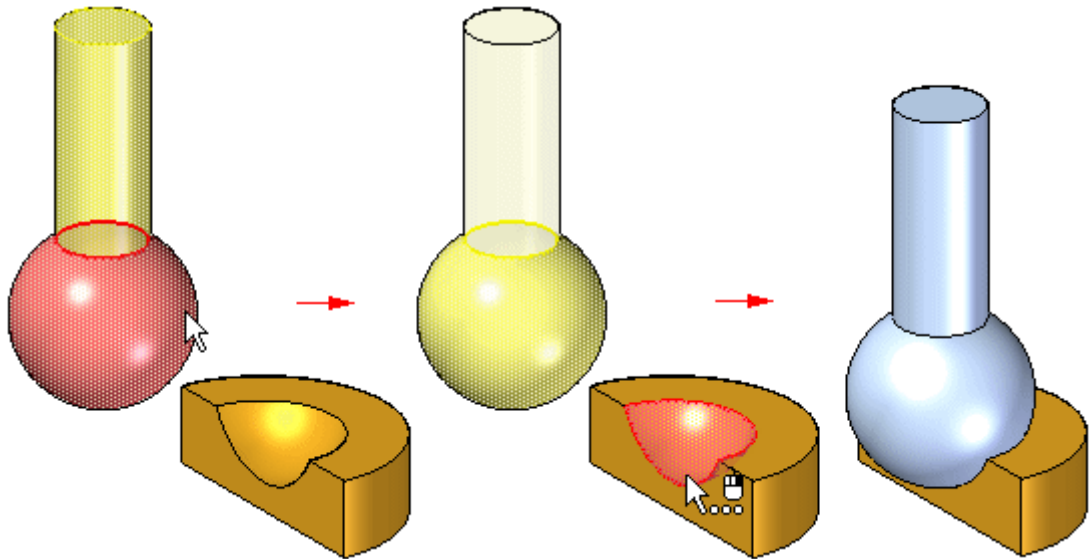
The Insert option applies a mate relationship with a fixed offset value and an axial align relationship with its rotational value fixed.



## Connect Relationship

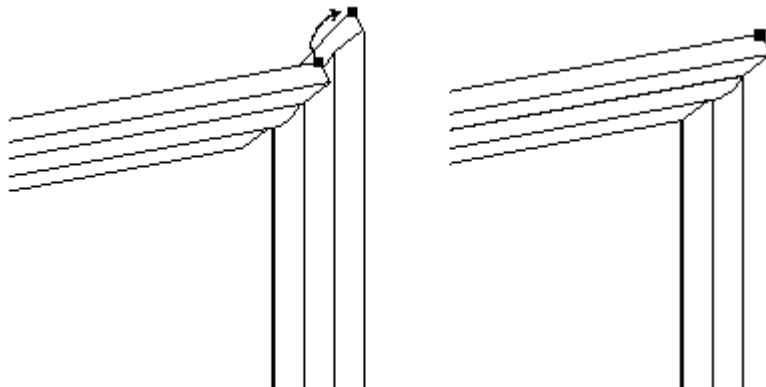
### Applying a connect relationship

When two parts in an assembly cannot be positioned properly by mate and align relationships, you can position them using connect relationships. A connect relationship positions a keypoint on one part with a keypoint, line, or face on another part. For example, you can apply a connect relationship to position the center of a spherical face on one part with respect to a spherical face on another part.

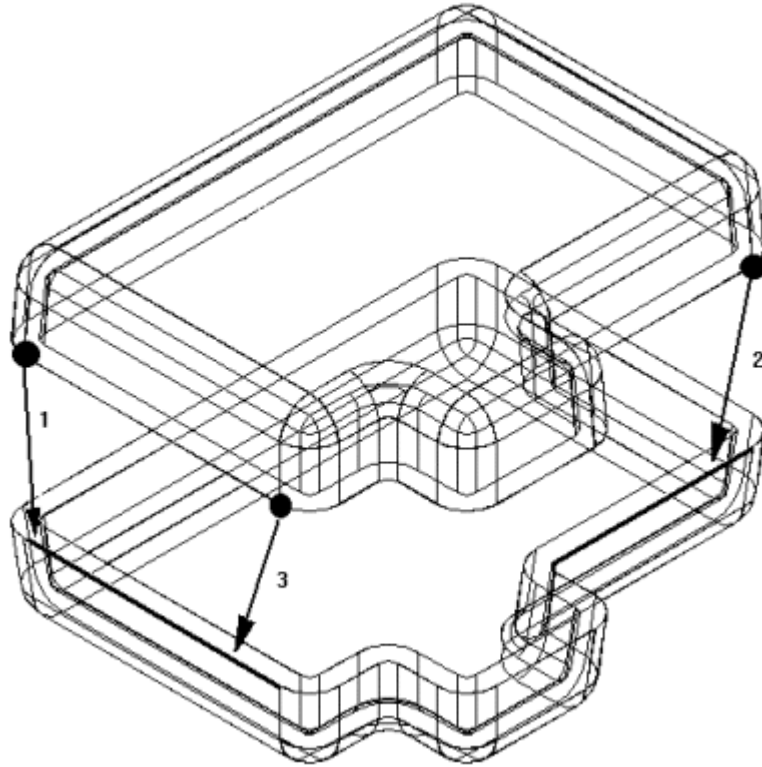


You can use the following methods to apply connect relationships:

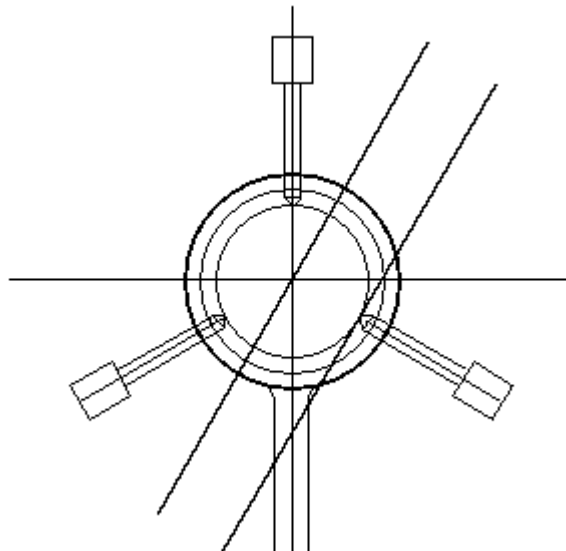
- **Point-to-Point:** In the following figure, a mate relationship is applied between the mitered corners of the parts. A connect relationship, which ties a point on one part to the appropriate point on another, connects the two corners properly. A floating align relationship between the back surfaces fully positions the part.



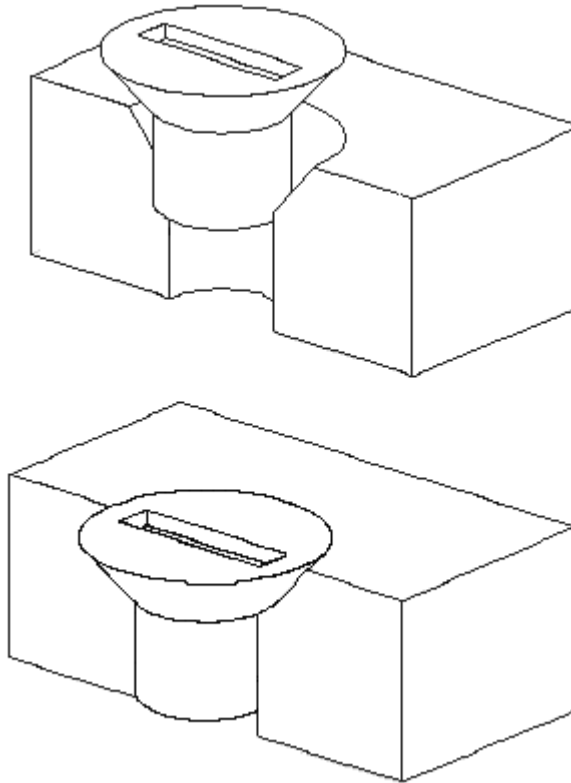
- **Point-to-Line:** In the following example, a mate relationship is applied between the faces of the two parts. Because the sides of each part are drafted, there are no part faces which you can use to apply a planar align relationship. You can apply three connect relationships between the keypoints on the top part and the linear edges on the bottom part.



- **Point-to-Plane:** In the following example, the lower right pin is positioned to a depth that just touches the surface of a reference plane.



- **Cone-to-Cone:** In the following example, the cone on the fastener is connected to the cone on the countersunk hole on the plate. When you add a connect relationship between two conical faces, the keypoint that represents the theoretical intersection of the individual cones are connected. You can also apply an offset value to a connect relationship between two conical faces.

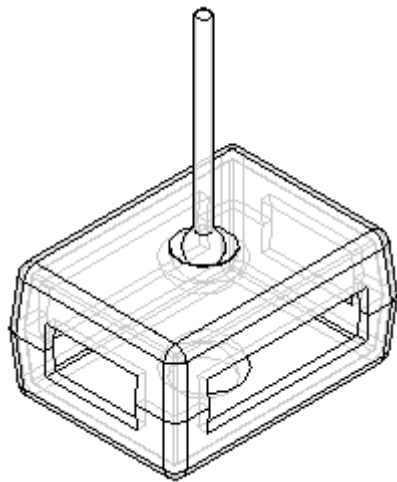


**Activity: Positioning assembly parts using the connect relationship****Overview**

The objective of this activity is to position a part in an assembly using the connect relationship.

**Activity**

In this activity you will use the connect relationship to position a part. The faces of the parts have draft angles and because of this the connect relationship will need to be used rather than the planar align relationship.



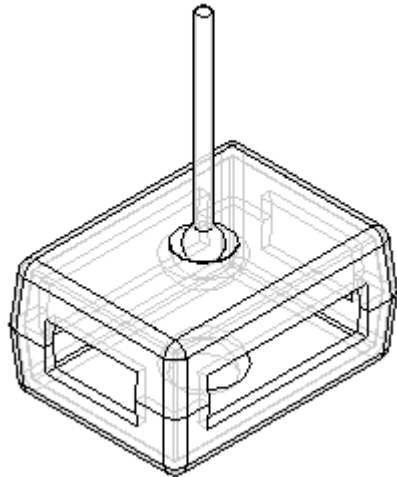
**Activity: Positioning assembly parts using the connect relationship**

**Overview**

This activity shows several options available that are used to position parts within an assembly using the connect relationship.

**Objectives**

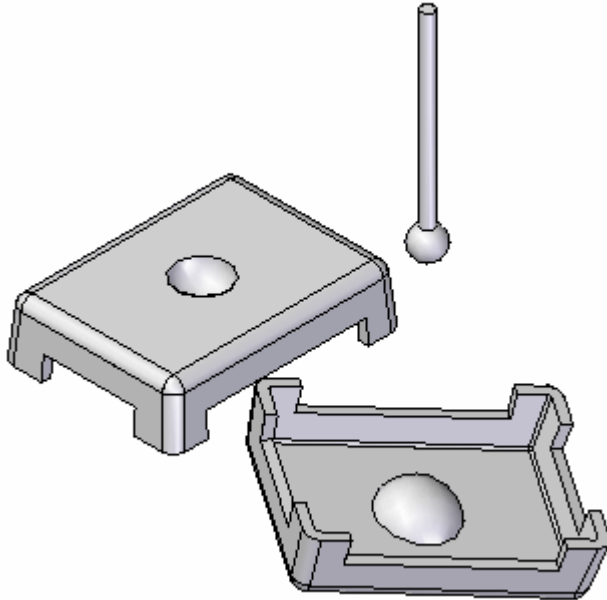
An assembly with several unconstrained parts will be opened. The Connect relationship will be used to position the parts.



*Open the assembly*

Open the assembly containing the parts to be positioned, and you will set the desired parameters.

- ▶ Open *Connect.asm* and activate all the parts.



- ▶ Click the Application button. Click Solid Edge Options, and then click the Assembly tab. Check the box as shown.

*Position the lid by connecting 3 points*

Use the Connect relationship to position the lid. Position the lid by connecting three of the corner arc centers together. This will completely position the lid.

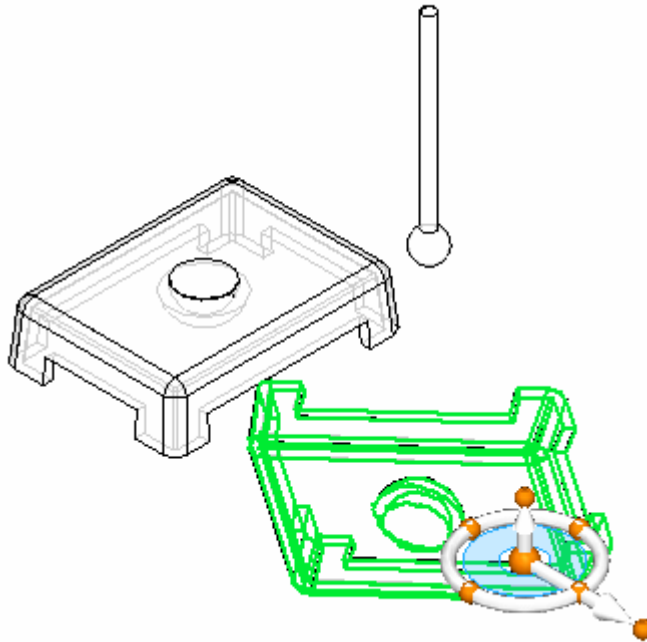
**Note**

The connect relationship recognizes key topological features to position parts. Like the axial align option, linear edges can be connected. Endpoints and midpoints of linear elements are valid for connecting, as well as arc and circle centers.

- ▶ Set the display to Visible and Hidden Edges. By exposing the hidden edges, locating desired geometry is more efficient.



- ▶ Click the Select command, and then select the lid shown.



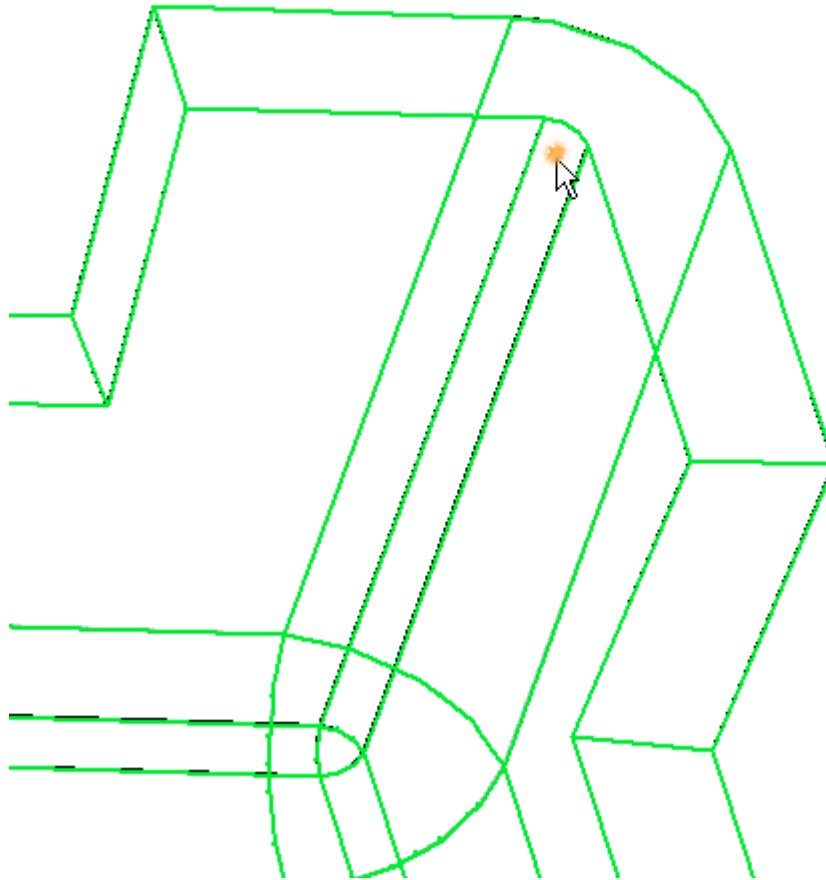
- ▶ To position the part, select the part with the select tool and then click the edit definition button.



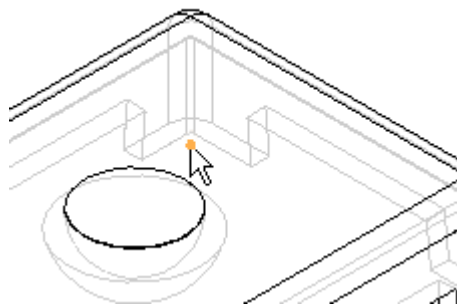
- ▶ Set the relationship type to Connect.



- ▶ Select the point on the arc center of the lid as shown.

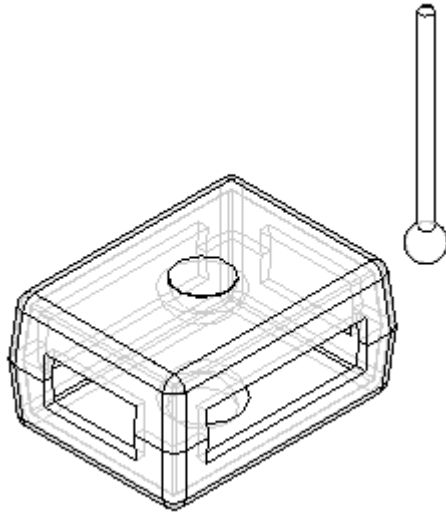


- ▶ Select the corner of the other lid shown as the target point for the first relationship.





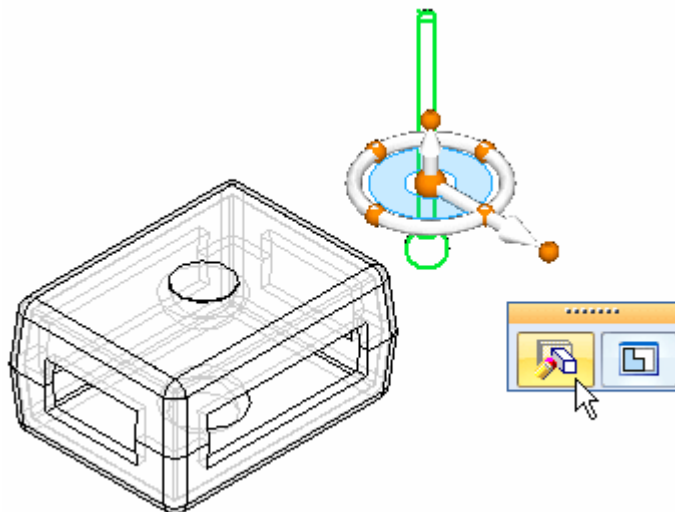
- ▶ Repeat these steps for any two of the remaining three corners. The lid is then completely positioned.



*Use spherical faces to define a connect relationship*

Position the center of the sphere on the knob to the center of the half sphere depression in the lid. This shows how spherical faces can be positioned using the Connect relationship.

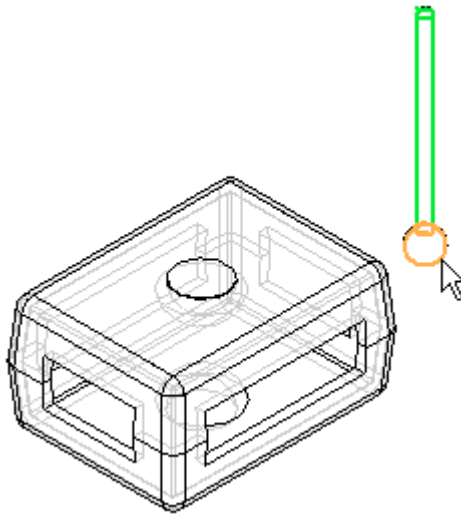
- ▶ Click the Select tool and select the knob. Then click the Edit Definition Command as shown.



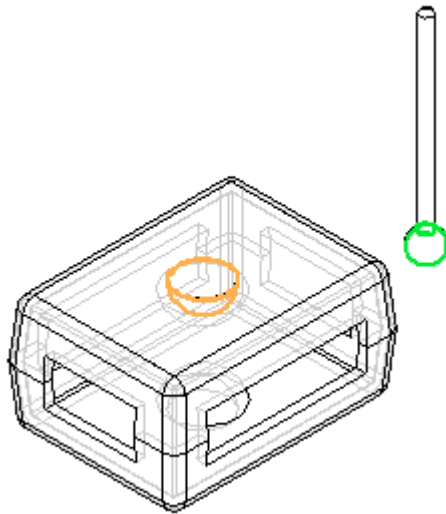
- ▶ Set the relationship type to Connect.



- ▶ Select the face of the sphere on the knob as shown.

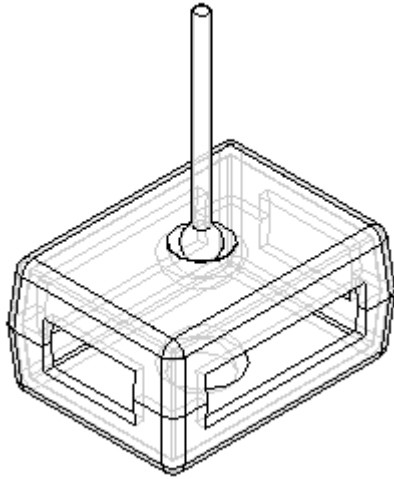


- ▶ As the target, select the face shown. You may need to use QuickPick to make the selection more efficient.

**Note**

The center of the sphere on the knob is now connected to the center of the spherical depression on the face. The knob has freedom to pivot about this point. Other relationships, such as Mate with a floating offset can be used to exactly position the knob.

- As an optional step in this activity, use the Mate relationship to completely position the knob as shown. You may also want to use the parts reference planes to help position the knob. Close the assembly without saving. This completes the activity.



*Activity summary*

In this activity you learned to use the Connect relationship to position a lid using points, and to position a knob by connecting sphere centers together.

This activity is complete.

**Lesson review**

Answer the following questions:

1. Give an example of why a connect relationship would be used to position a part rather than mate or planar align relationship.
2. Give some examples of valid geometry that can be used to create a connect relationship.
3. Give some examples of connect relationship combinations.

## **Answers**

1. Give an example of why a connect relationship would be used to position a part rather than mate or planar align relationship.

Molded parts where the drafted faces exist do not work well with planar align or mate relationships. Ball and socket scenarios where concentric spheres need to be positioned is another example of using the connect relationship. The connect relationship is useful in these situations.

2. Give some examples of valid geometry that can be used to create a connect relationship.

Examples of valid geometry for the connect relationship:

- End points of lines, arcs, and ellipses
  - Midpoint of a line (edge centerline)
  - Arc center point
  - Circle center point
  - Spherical surface center point
  - Conical surface center point
  - Linear edges
  - Reference planes
  - Reference axis
  - Planar part surfaces
3. Give some examples of connect relationship combinations.
    - Connect a point on the first part to a point on the second part
    - Connect a point on the first part to a line on the second part
    - Connect a point on the first part to a face on the second part
    - Connect a line on the first part to a point on the second part
    - Connect a face on the first part to a point on the second part

**Lesson summary**

In this lesson you learned to use the Connect relationship to position a lid using points, and to position a knob by connecting sphere centers together.

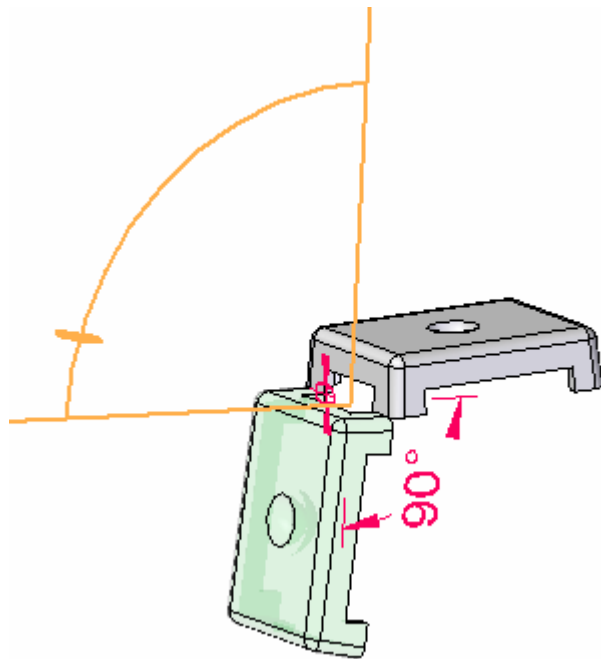
## Activity: Positioning assembly parts using the angle relationship

### Overview

The objective of this activity is to position a part using the angle relationship.

### Activity

In this activity you will position a part using the angle relationship, then modify the value of the angle and observe the change in position.

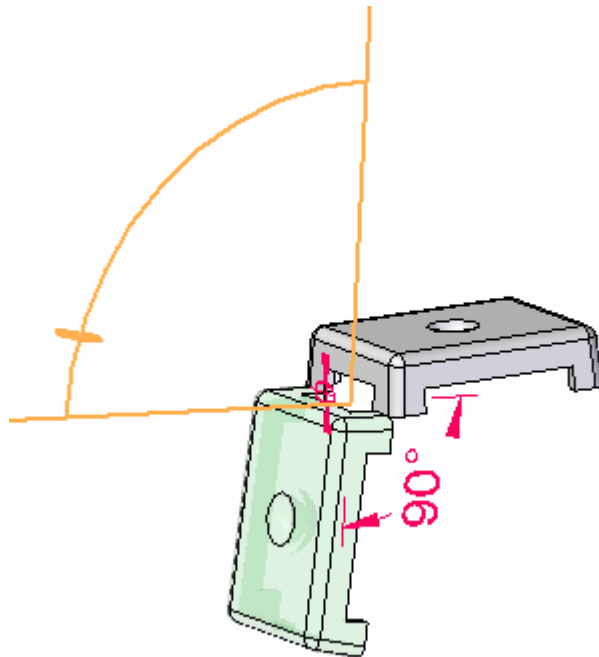


**Activity: Positioning assembly parts using the angle relationship****Overview**

This activity shows several options available that are used to position parts within an assembly using the Angle relationship.

**Objectives**

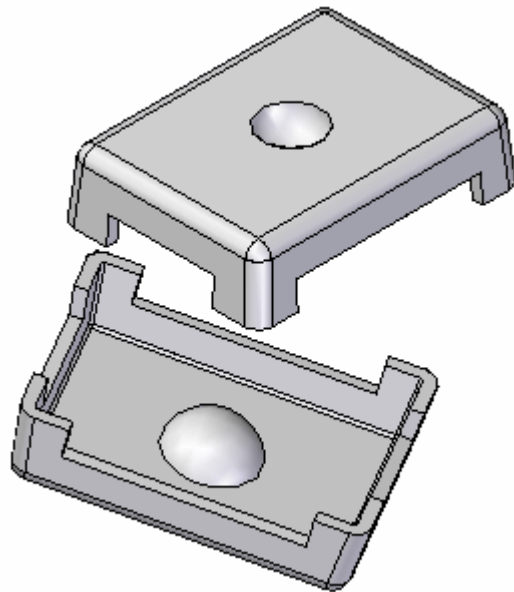
An assembly with unconstrained parts will be opened. The Angle relationship will be used to position the parts.

*Open the assembly*

Open the assembly containing the parts to be positioned, and then set the desired parameters.



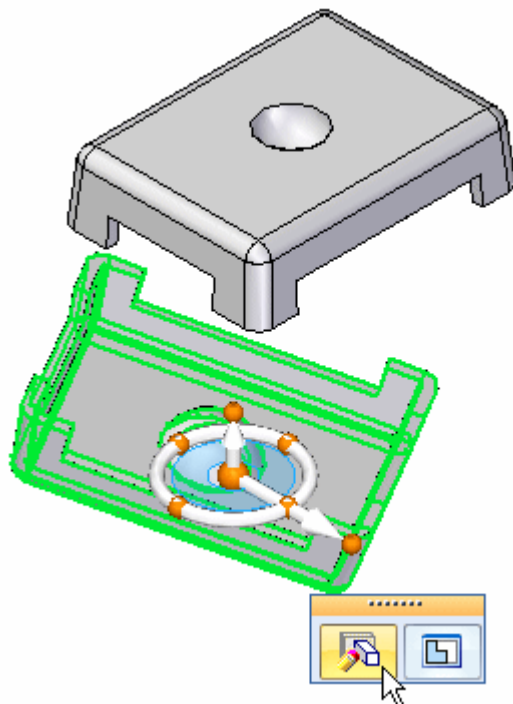
- Open *Angle.asm* with all the parts active.



*Create a connect relationship*

To position the lid, the first relationship you establish will be the Connect relationship.

- Click the Select command and select the part shown. Then click the Edit Definition button as shown.



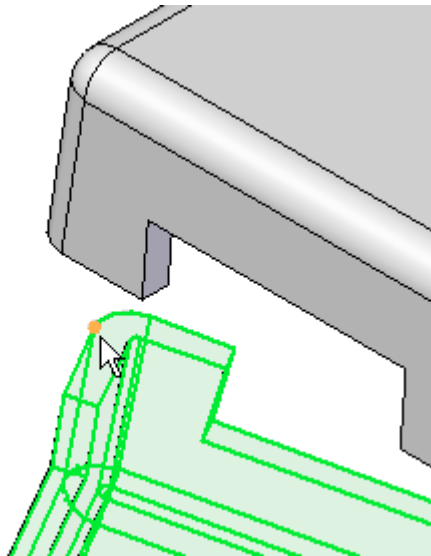
- ▶ Select the Connect relationship.



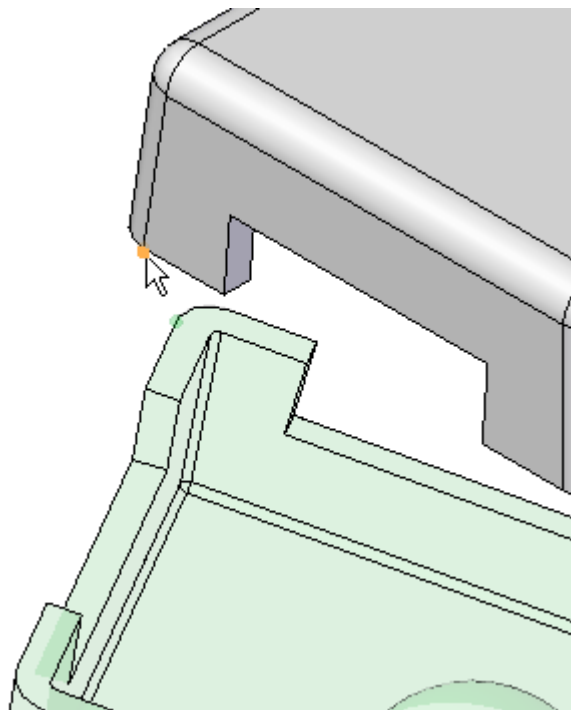
- ▶ Select the vertex point shown.

**Note**

You may have to rotate the view to better identify the point.



- ▶ For the target, select the vertex point shown.



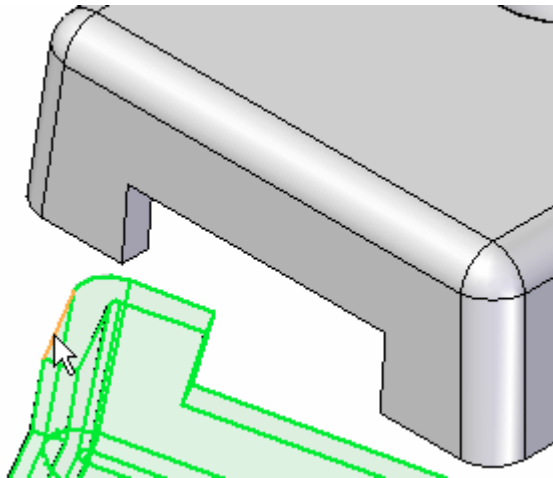
*Create an axial align relationship*

Use the Axial Align relationship for the second relationship.

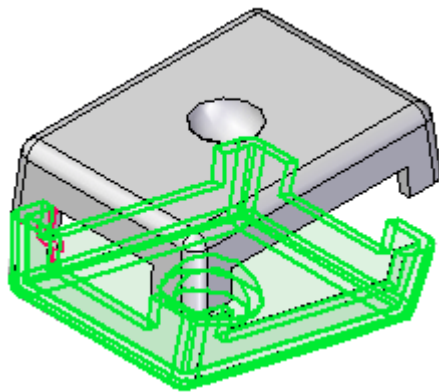
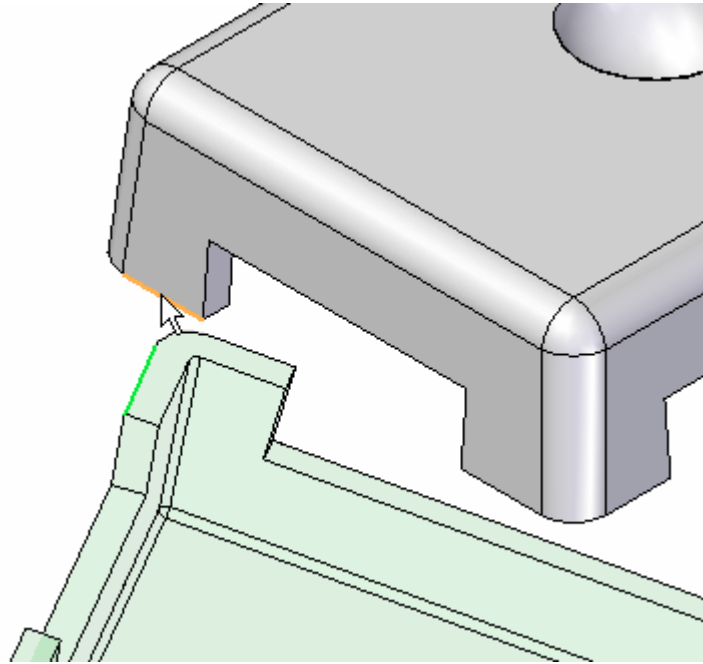
- ▶ Click the Axial Align relationship.



- ▶ Select the linear edge shown.



- For the target, select the linear edge shown.



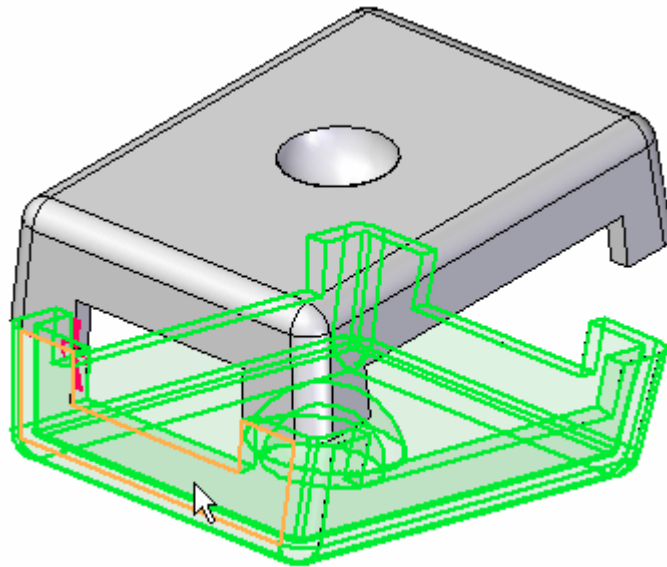
*Position the lid using the angle relationship*

Use the Angle relationship to position the lid. Once placed, the angular value will be able to be modified to reposition the orientation of the lid.

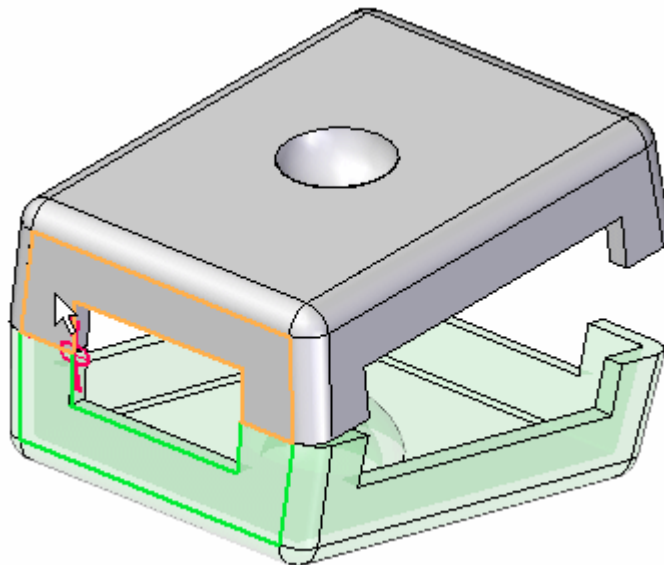
- Select the Angle Relationship.



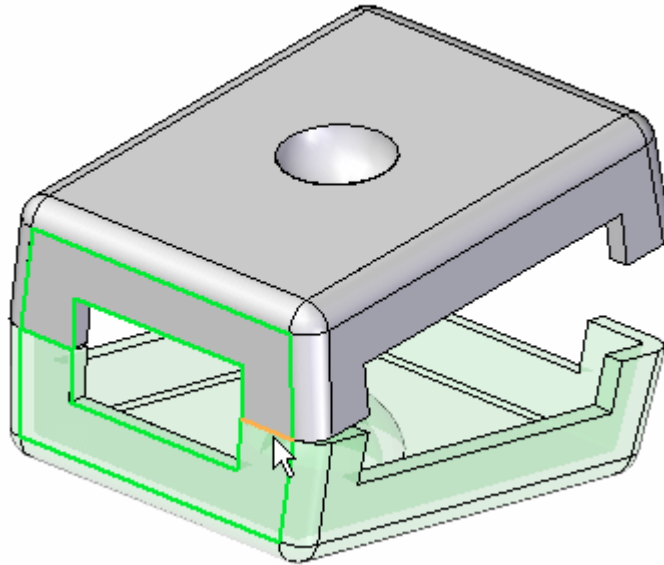
- ▶ Select the face shown as the face to measure to.



- ▶ Click the face shown as the face to measure from.



- ▶ When prompted to click on a plane in which the angle measurement will lie, click the edge shown.

**Note**

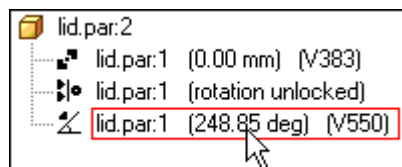
The Angle measurement is established.

- ▶ Click the Select tool.

*Edit the angle*

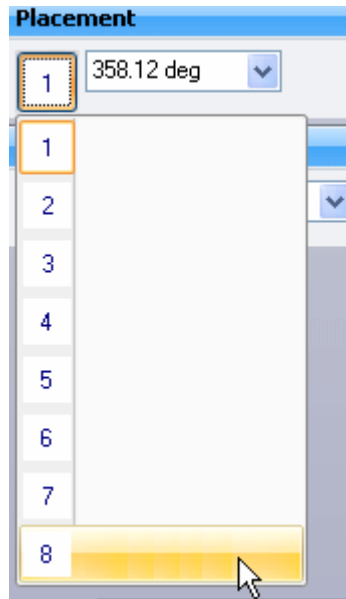
Edit the angle and the position of the lid will change.

- ▶ Press Ctrl+R on the keyboard to rotate the view to a right view.
- ▶ In PathFinder, select *lid.par:2* and then, in the lower pane, select the Angle relationship.

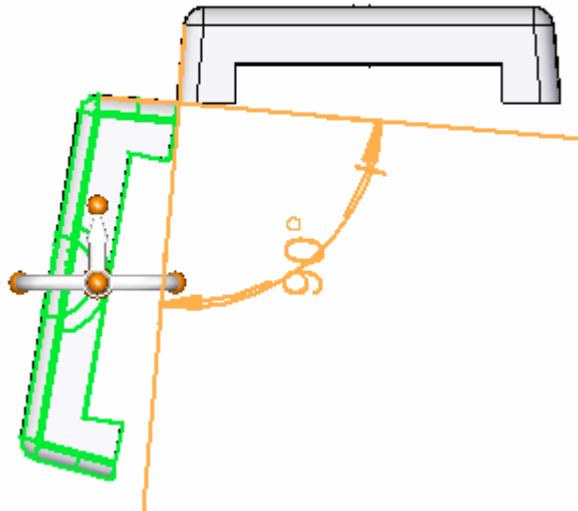
**Note**

Variable names and angle values may differ from the picture shown. This is not a problem.

- ▶ On the Placement command bar, click the Angle Format list, and then move the cursor over the eight options. Notice the difference in how the angle is measured in each of the different options.

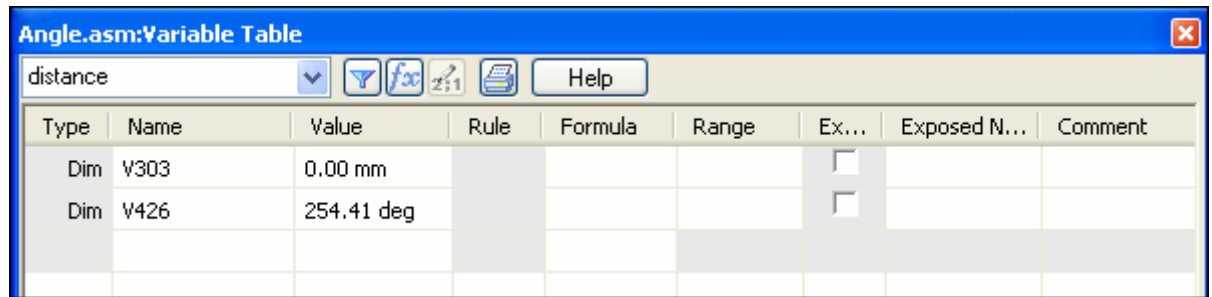


- ▶ Click the Angle format which gives the measurement shown below. Change the angle to 90°.



- ▶ Change the angle to different values and observe the behavior. Change the angle to 190°.

- ▶ On the ribbon, choose Tools tab® Variables to display the Variable Table. Notice this angular value is shown here and can be edited from the Variable Table. Also the angular value can be driven via a formula to other values within the table.



Type	Name	Value	Rule	Formula	Range	Ex...	Exposed N...	Comment
Dim	V303	0.00 mm				<input type="checkbox"/>		
Dim	V426	254.41 deg				<input type="checkbox"/>		

- ▶ This completes the activity. Close the assembly document without saving.

#### Activity summary

In this activity you learned to use the Angle relationship to position a lid, and modify the value of the angle to change the position of the lid.

This activity is complete.



## **Lesson review**

Answer the following questions:

1. What steps are used to place an angle relationship?

**Answers**

1. What steps are used to place an angle relationship?

The following steps define an angle relationship:

- Select the face on the placement part to measure the angle from.
- Select the target part.
- Select the face on the target part to measure the angle to.
- Select the axis or plane the dimension will be placed on.
- Enter the angular value.
- Select the angle format. This shows the 8 options on how the angle is defined.
- Click OK.

## **Lesson summary**

In this lesson you learned to use the Angle relationship to position a lid, and modify the value of the angle to change the position of the lid.

## **The Assemble command**

The Assemble command is an alternative method of positioning multiple parts in an assembly that have been placed in an assembly, but that are yet to be positioned. Utilizing Flashfit techniques, the Assemble command allows changing parts with a right mouse click.

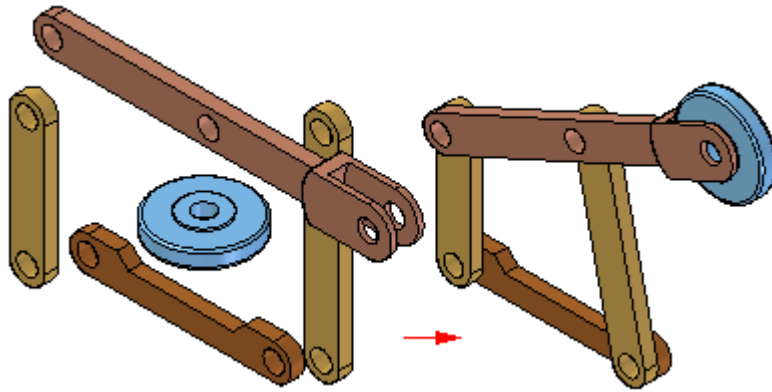


## Assemble command

Position parts in an assembly. You can use this command to position a single part in an assembly, or you can use this command to position several parts relative to each other without fully constraining each part in an ordered sequence.

This type of workflow can make it easier to position a set of interrelated parts, such as when building a mechanism.

After dragging and dropping a set of parts into an assembly, you can use the Assemble command to apply relationships between one of the parts and one or more target parts. To position a different part, click the right mouse button.



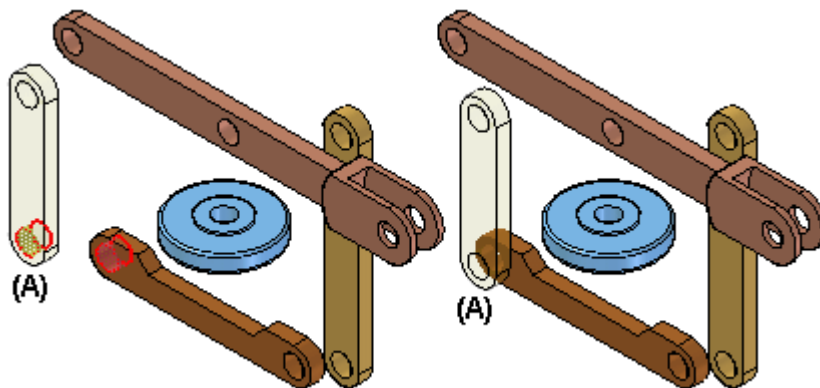
To position a series of parts using the Assemble command, you first drag and drop the set of parts into the assembly using the Parts Library tab.

If it is a new assembly, the first part is automatically grounded. When you drag and drop the second part into the assembly, the Assemble command bar is displayed, but you can continue to drag and drop parts into the assembly without positioning them.

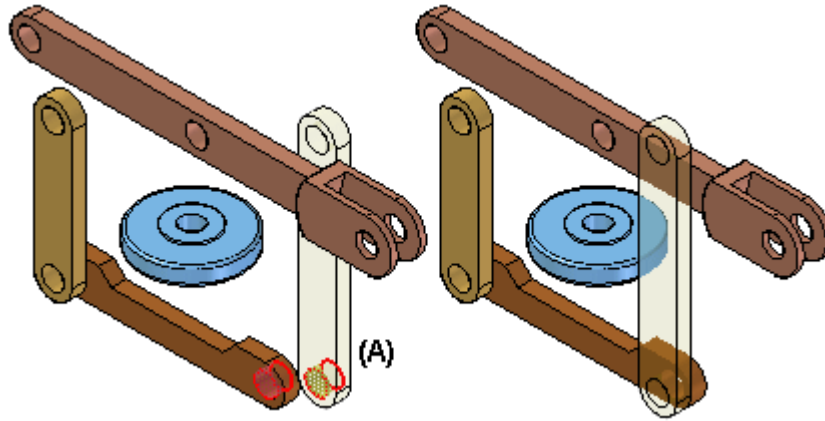
After the set of parts is placed in the assembly, you can use the Assemble command to position the parts.

When you click the Assemble command, the Assemble command bar is displayed. You can use the FlashFit option to apply a mate, planar align, or axial align relationship, or choose from the complete set of ordered relationships.

After you apply a relationship between two parts, the first part you select (A) remains selected so you can apply additional relationships to the part.



To position a different part, click the right mouse button. You can then select a different part (A) and apply the relationships you want.



The Assemble command is tightly integrated with the FlashFit positioning option. When you click the Assemble command, FlashFit is the default option. For more information on FlashFit, see the [Assembly Relationships](#) topic.

Assemble command bar

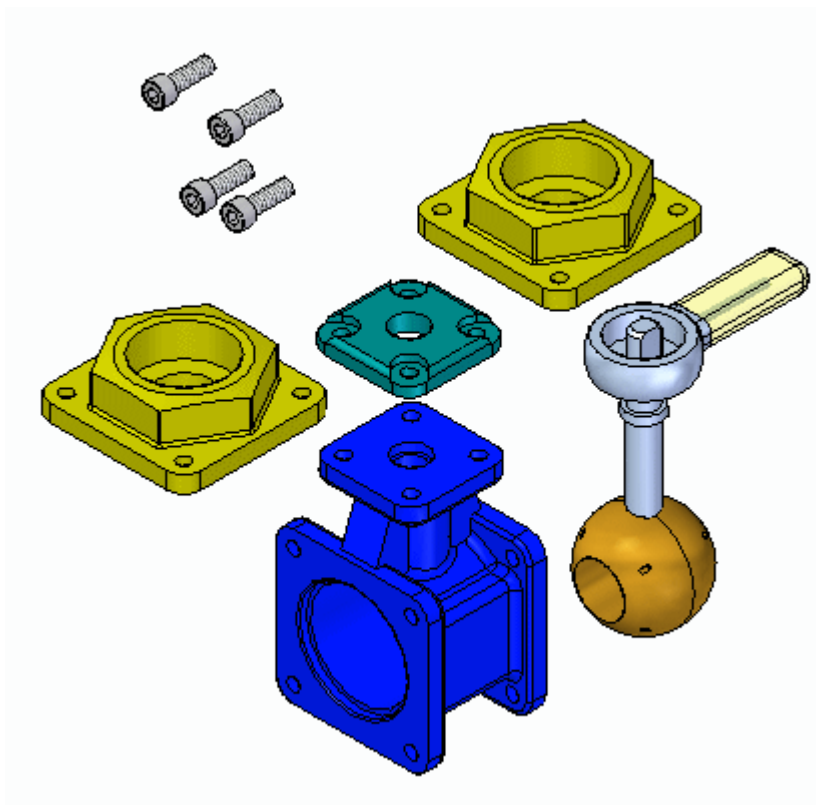
## **Activity: The Assemble command**

### **Overview**

The objective of this activity is to understand how to position parts using the Assemble command.

### **Activity**

In this activity you will learn how to use the Assemble command.



**Activity: Assemble command****Overview**

When you complete this activity, you will be able to position parts in an assembly using the Assemble command. This activity teaches how to use the Assemble command to manipulate and fully position parts in an assembly.

**Objectives**

The activity consists of the following:

- Settings that affect the Assemble Command
- Manipulation and positioning of parts using the Assemble Command
- Editing and error recovery

**Note**

*Positioning parts with the Assemble command:*

There are many ways to correctly assemble the parts and subassembly associated with this activity. You will not be given specific instructions on how to assemble these parts other than the order in which to assemble the parts. How a part is positioned using FlashFit is predictable. With the Assemble command however, parts can be positioned incorrectly or over constrained. This activity will purposely position a part incorrectly so that the steps to correct the positioning will be covered.

The rules on the behavior of the Assemble command are listed next. You will be instructed to use these rules where appropriate.



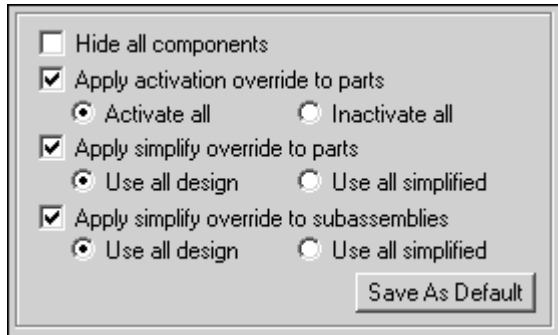
**Note***Assemble Command guidelines*

The following guidelines are used in positioning the parts in this assembly using the Assemble command.

- In the assembly you will be working with *valve\_housing.par* which was placed first and is grounded. The other parts will be positioned relative to this previously positioned part.
- FlashFit is the default assembly relationship creation mode and should be used.
- Once you select a part to position, it will become transparent. Once it is fully positioned, or another part is selected, it will no longer be transparent.
- If, in the middle of positioning a part, you decide to position another part, right-click to release the current part. It will no longer be transparent. The next part you select will then become transparent.
- If you work in wireframe, rather than shaded, you will not have the visual benefit of the selected part being transparent. For that reason, it is suggested that you use Shaded with Visible Edges display mode while using the Assemble command.
- Once a part is selected, it can be dragged to a new position with the left mouse button. The selected part is the part you are applying relationships to. Right-click to release the part.
- To rotate a selected part that is unconstrained, use Ctrl + left mouse button.
- FlashFit will determine whether to use a Mate or Planar Align relationship based on the closest orientation of the faces being matched. It is good practice to rotate the selected part into the approximate position before selecting the faces. After FlashFit, if the faces are 180° out of position, click the Flip button on the command bar.
- Matching circular edges will quickly position a part, such as a fastener, in one operation. The centerlines are superimposed and the rotation is locked. You can unlock the rotation by editing the relationships in Assembly PathFinder. Refer to the section of this activity that addresses editing and error recovery.

*Assemble command setup*

Open *assemble.asm* making all the parts active by using the settings shown in the open dialog box.



- ▶ Choose the Home tab->Assemble group->Assemble command..



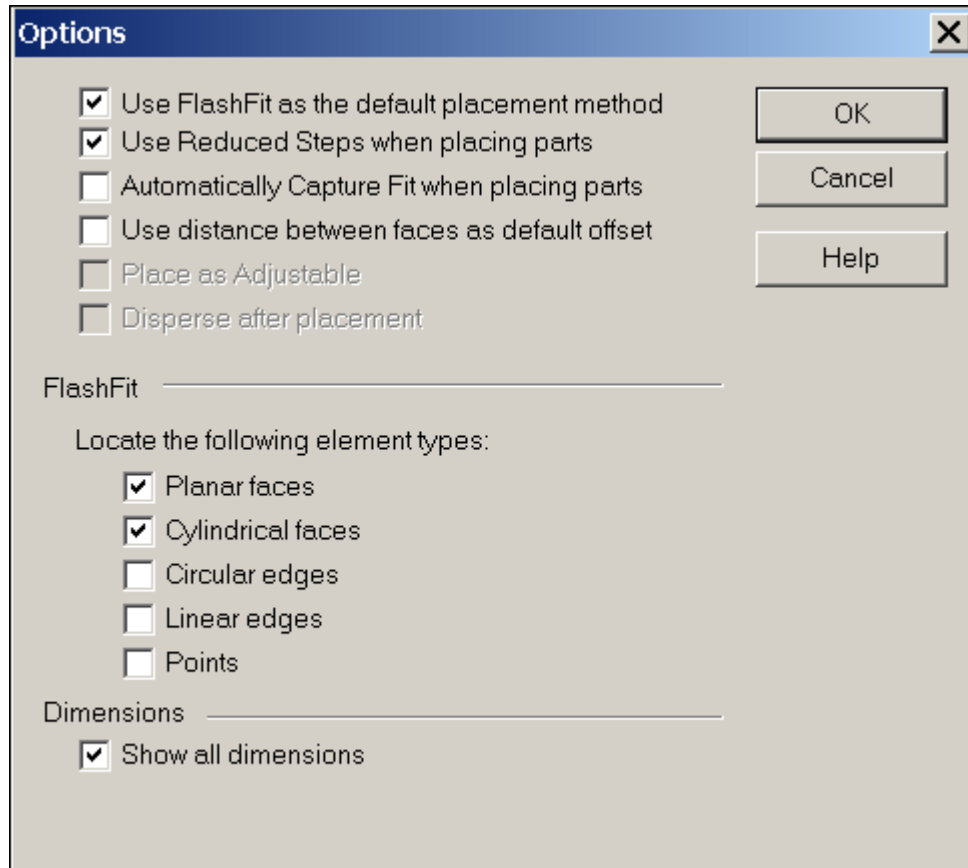
- ▶ Click the Options button on the command bar.



- Set the options shown below, then click OK.

**Note**

The behavior of positioning faces with FlashFit will first be shown. For other parts, edges will be used.



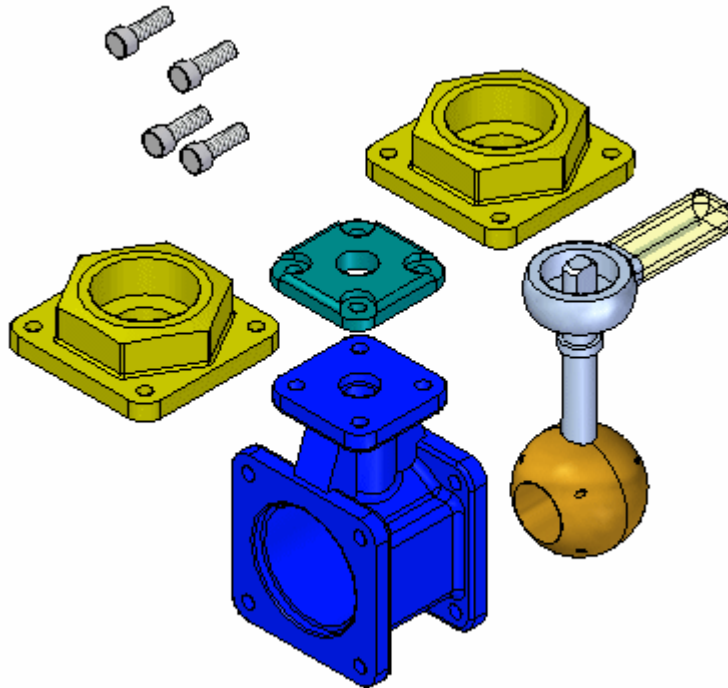
- Click Shaded with Visible Edges.



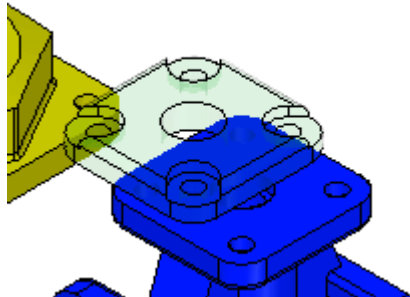
*Assemble the parts beginning with the top cap*

Move the parts to their approximate final position. Select a part to move. The part will become transparent. With the left mouse button, drag the part to the position shown. Right-click to release the part and left-click to select a different part to drag.

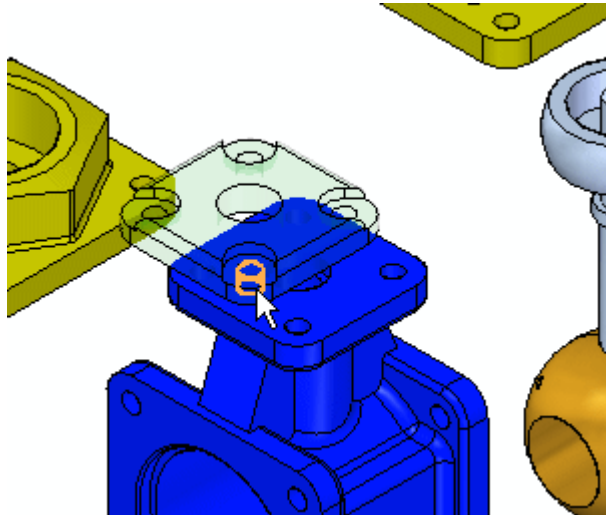
- ▶ Position all the parts approximately as shown.



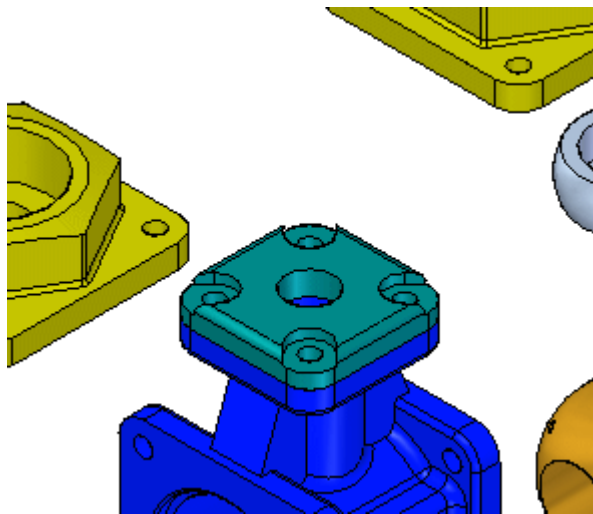
- ▶ Right-click to release the last part selected. Select the top cap. Zoom in on the top face of the valve housing. Using FlashFit, mate the bottom face of the top cap to the top face of the valve housing as shown.



- ▶ Select the cylindrical face of a hole on the top cap, and then select the cylindrical face of a hole in the top of the valve housing, as shown.



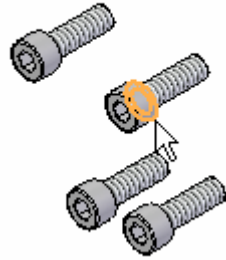
- ▶ To completely position the part, repeat the previous step beginning with a different cylindrical face in the top cap, and a corresponding cylindrical face in the valve housing. Once the part is completely positioned, it will become shaded and no longer be transparent.



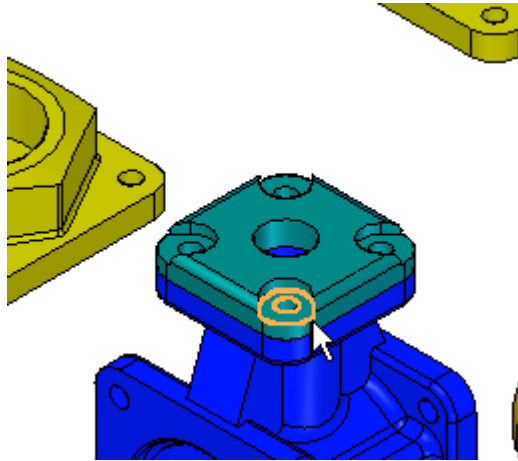
*Position the first fasteners on the top cap*

Position the first fastener on the top cap.

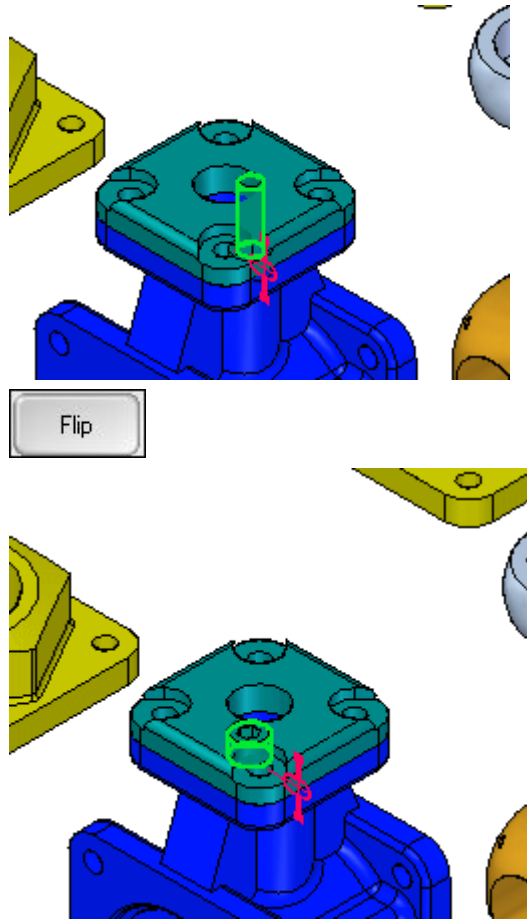
- ▶ To begin positioning the next part, select the face shown of one of the fasteners.



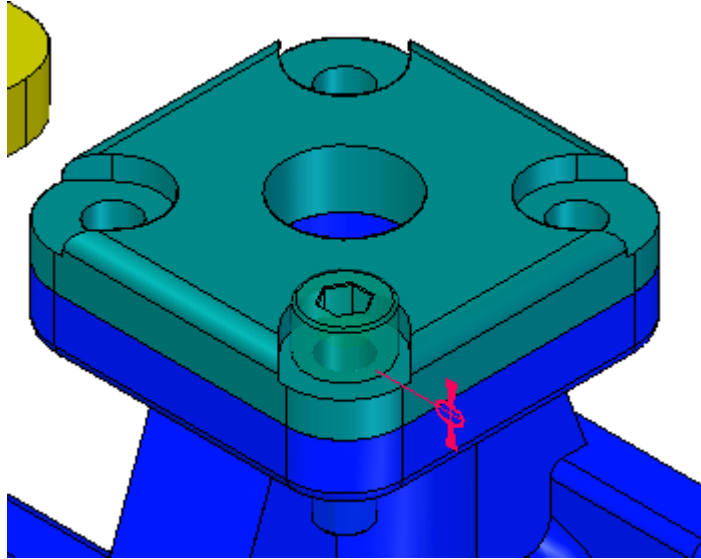
- ▶ Select the face of the top cap as shown.



- FlashFit determines whether to apply a mate or planar align relationship based on the orientation of each of the faces. If the fastener is placed reversed, as shown, click Flip on the command bar to correct.

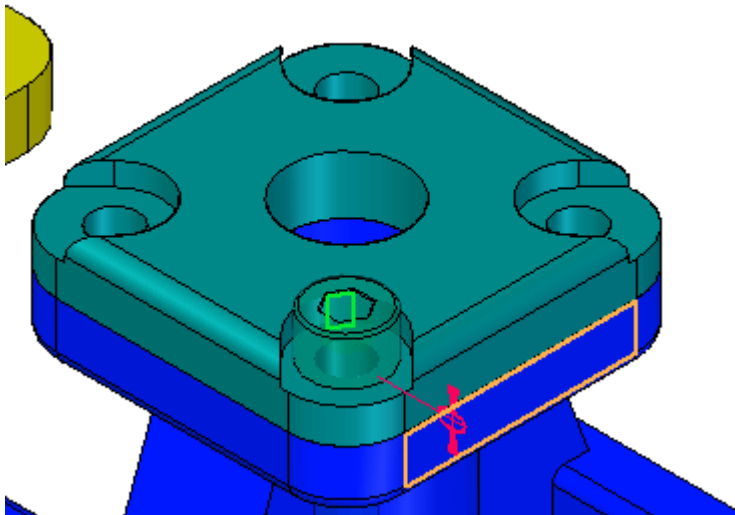
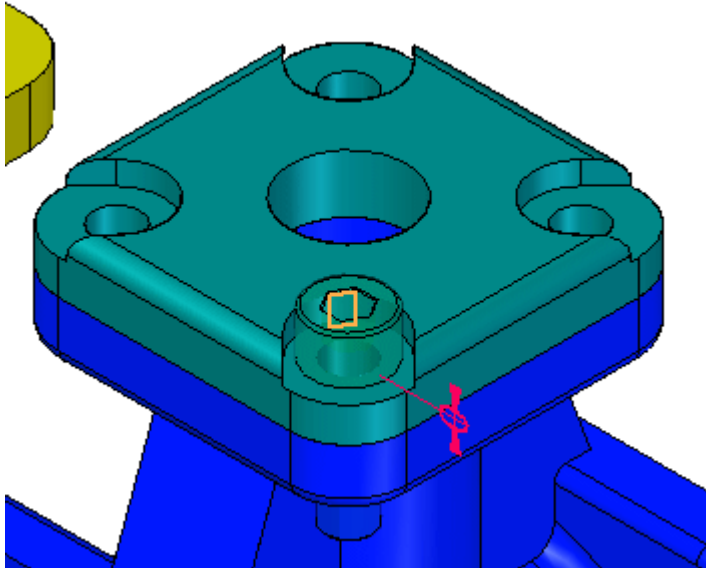


- Select the cylindrical shaft of the fastener and the cylindrical face of the corresponding hole. The fastener is positioned in the hole, but is transparent because there is freedom in the axis defined by the center of the shaft.





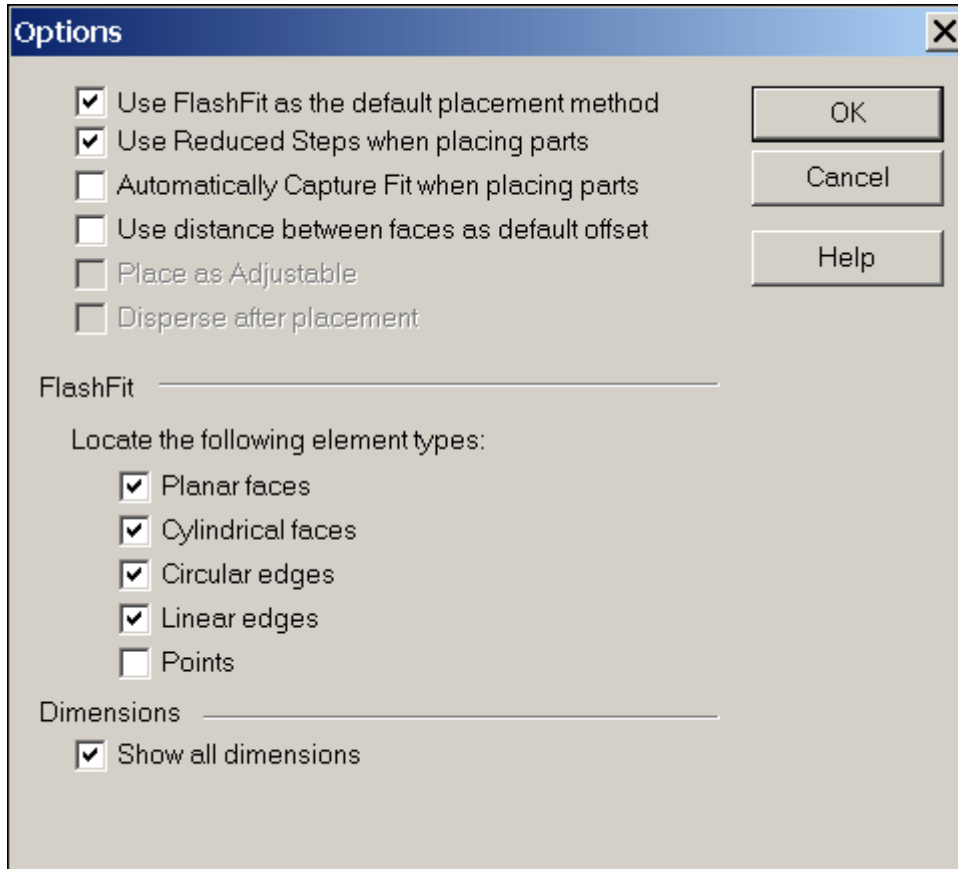
- ▶ Select the flat face on the head of the fastener as shown, and then select the face on the top cap as shown. The bolt rotates such that the planes become parallel and a floating offset is applied which locks the rotation of the bolt.



*Position the other fasteners using edge selection*

You will now position other fasteners by using FlashFit and selecting circular edges.

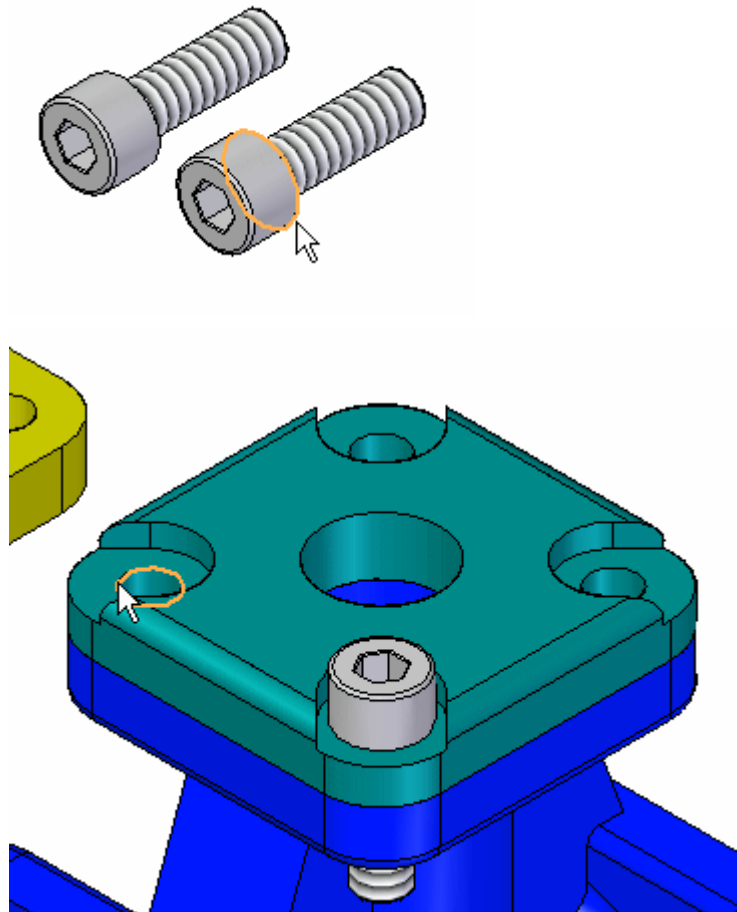
- ▶ Click the options button on the command bar and set the options as shown.



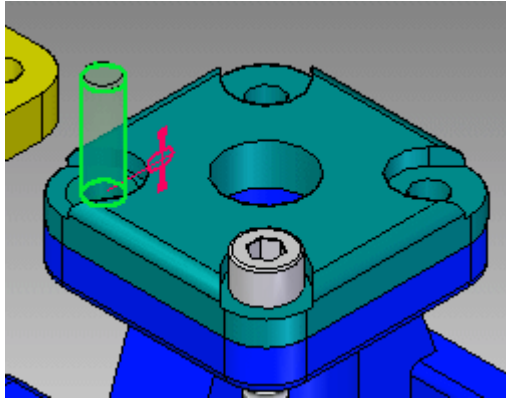
- Position the fastener using the edges shown below.

**Note**

Positioning by matching circular edges completely constrains the part by fixing the rotation. This is the preferred method.



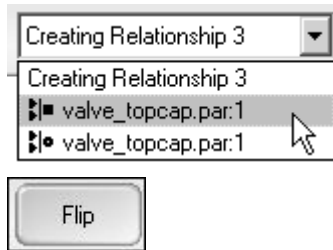
- ▶ If the fastener becomes oriented as shown, it is because the original orientation of the faces of the fastener was closer to a planar align relationship rather than a mate relationship.



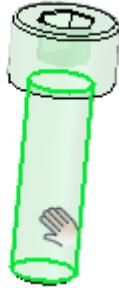
**Note**

Because the part is completely constrained, the Assemble command has released this fastener and is ready to position another part. The fastener has become shaded indicating it is fully positioned. To position this fastener correctly, you will need to temporarily exit the Assemble command by clicking the Select tool. Once the fastener is positioned correctly, click the Assemble command to continue positioning parts.

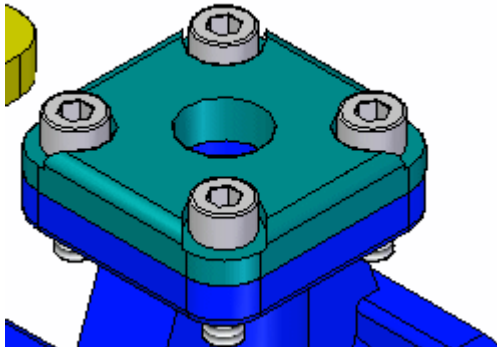
- ▶ If the fastener was placed upside down, select the fastener. It will become transparent. Select the mate relationship, click Flip, then click OK to correctly position the fastener.



- ▶ To position the remaining two fasteners, use these steps. Click the Assemble command and select one of the remaining fasteners. To rotate the fastener into the approximate vertical position, click and hold Ctrl while you left-click and then drag.



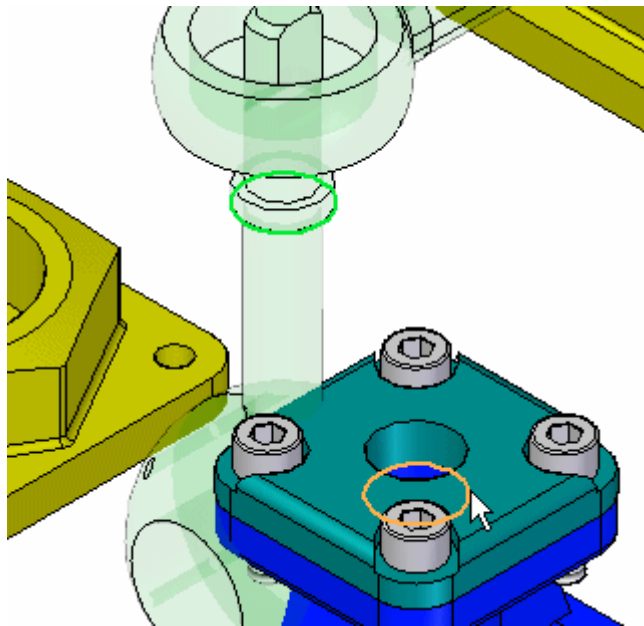
- ▶ Position the fastener by selecting the same circular edges as the previous part. The fastener will be oriented correctly because the orientation was close to the final position. Right-click to clear the selection when done.



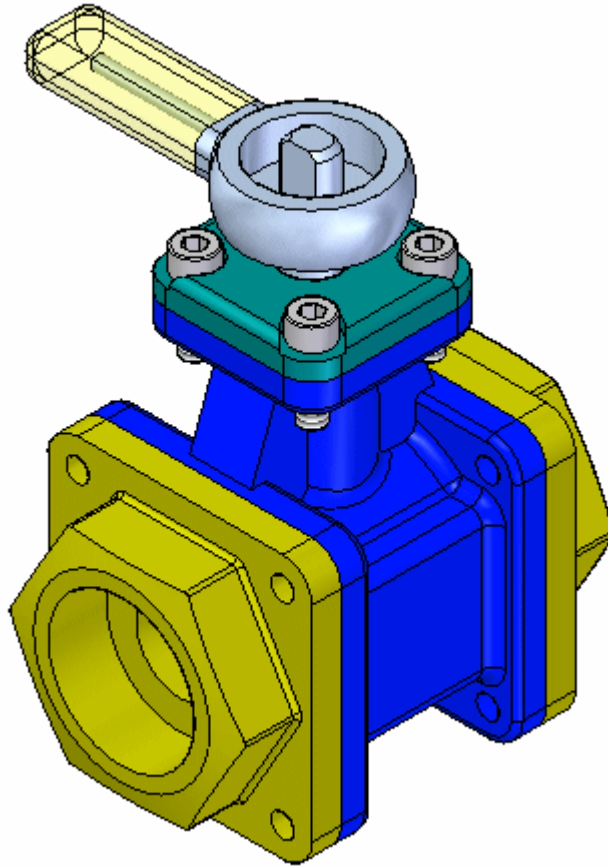
*Position the handle subassembly*

You will now position the handle subassembly by using FlashFit and selecting circular edges.

- Position the subassembly *handle and ball.asm* by selecting the circular edges as shown.



- Use the Assemble command and the techniques learned from the previous steps to position each endcap into the correct position on the valve housing. This completes this activity. Close the assembly document.



*Activity summary*

In this activity you learned how to use the Assemble command to quickly assemble a group of parts into an assembly. If all the parts making up an assembly are placed in an assembly window, the Assemble command can be used to complete the positioning of the parts into the final assembly.

This completes the activity.

**Lesson review**

Answer the following questions:

1. How do you move an unconstrained assembly component when using the assemble command?
2. How do you rotate an unconstrained assembly component when using the assemble command?
3. How do you select a different assembly component to position without exiting the assembly command?



## **Answers**

1. How do you move an unconstrained assembly component when using the assemble command?

Drag using the left mouse button.

2. How do you rotate an unconstrained assembly component when using the assemble command?

Hold the CTRL key and drag with the left mouse button.

3. How do you select a different assembly component to position without exiting the assembly command?

Right click to release the current assembly component, then select the next component to position.

**Lesson summary**

In this lesson you learned how to use the Assemble command to quickly assemble a group of parts into an assembly. If all the parts making up an assembly are placed in an assembly window, the Assemble command can be used to complete the positioning of the parts into the final assembly.

## **Designing in the context of an assembly**

When designing in the context of an assembly, you can construct the parts with either ordered or synchronous geometry based on which is appropriate for the desired outcome.

In the activity, you will experience designing parts in context of a top level assembly using the tools provided.

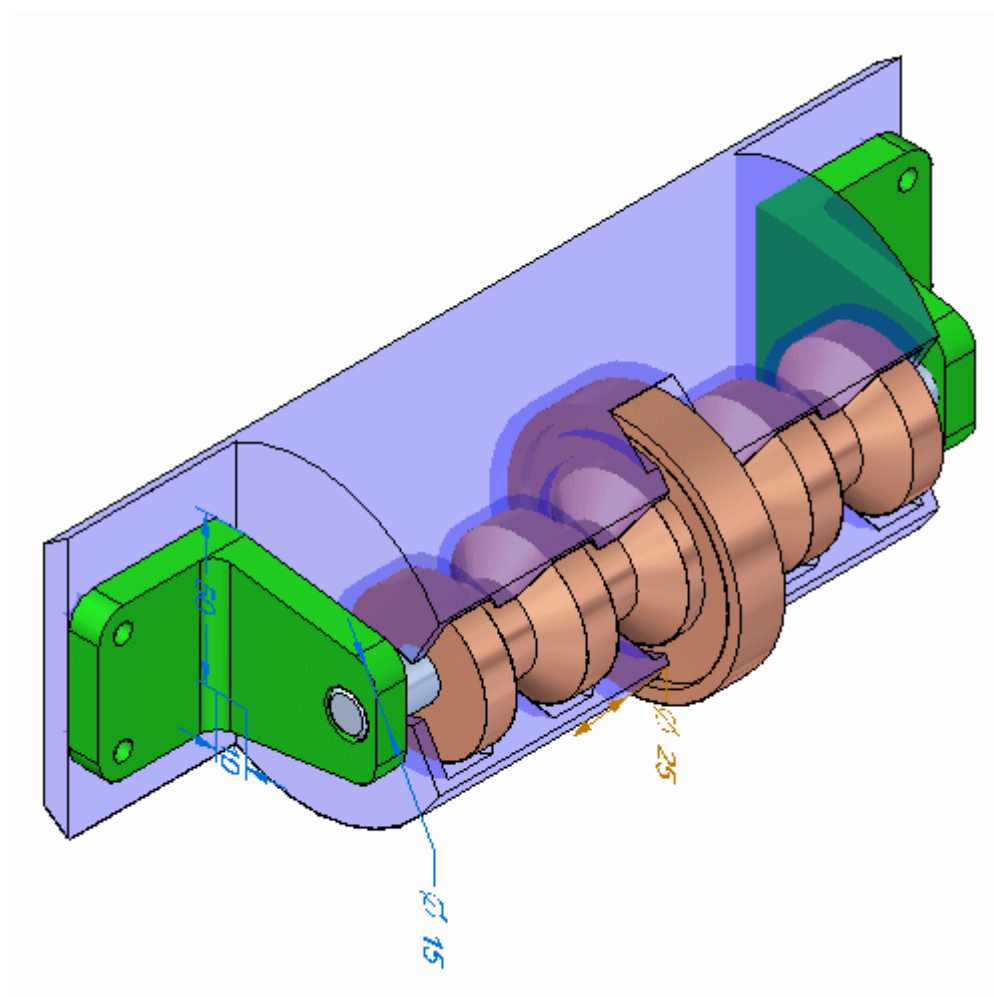
## Activity: Designing in the context of a synchronous assembly

### Overview

The objective of this activity is to explore designing in the context of an assembly with Solid Edge synchronous technology. You will open an existing assembly and use adjacent parts to refine the sizing and spacing of the faces and parts in the assembly. You will also use geometry from one part to create a cavity in an adjacent part.

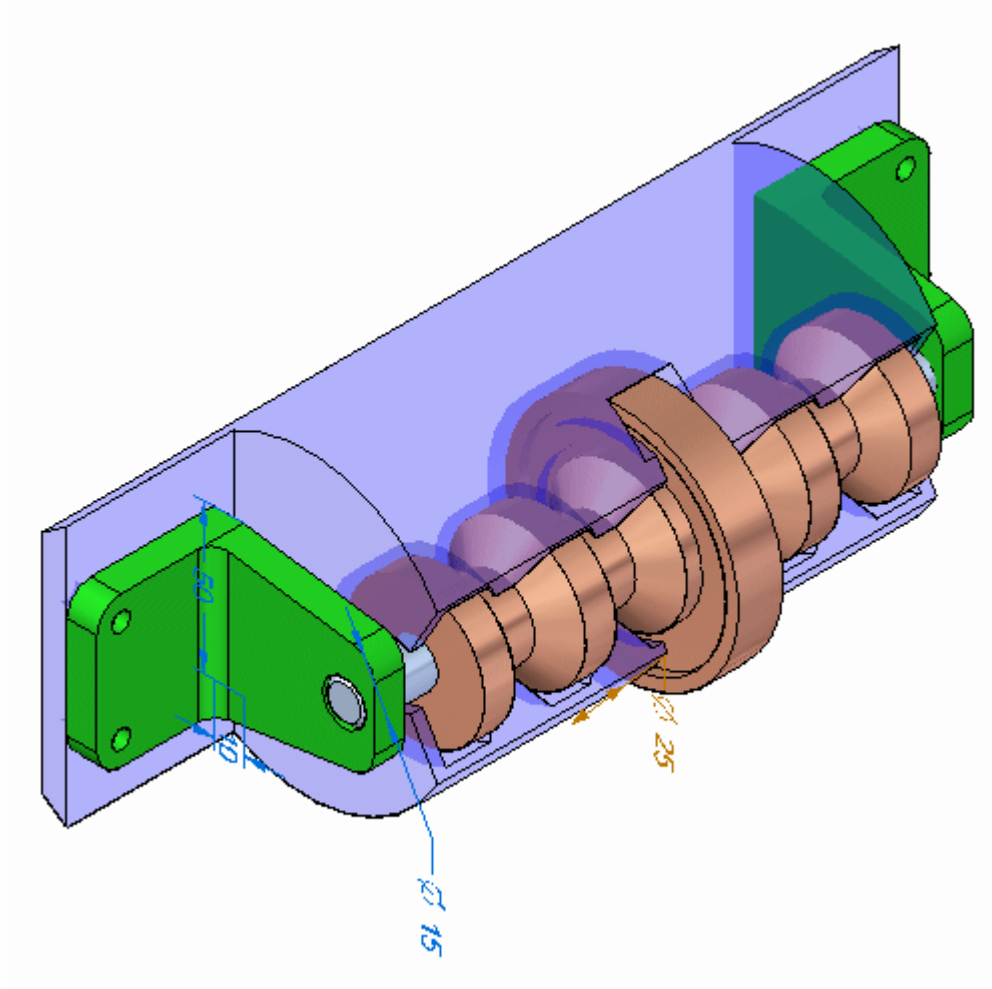
### Activity

In this activity you will learn how synchronous technology can be of benefit when designing in the context of an assembly.



**Activity: Designing in the context of a synchronous assembly**

In this activity, you will modify parts in the context of the assembly using Solid Edge.



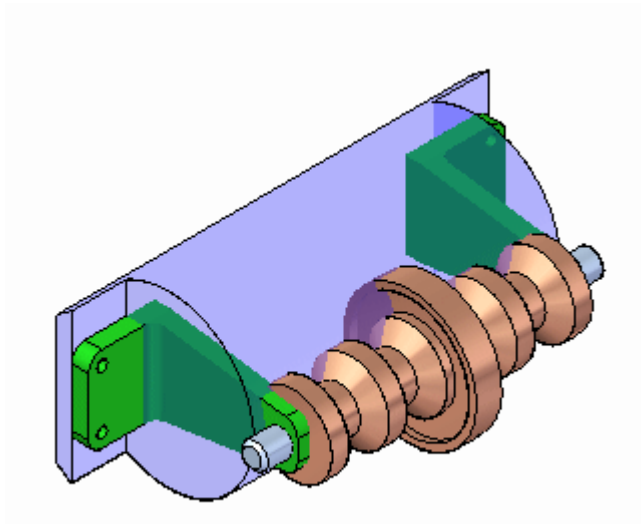
*Open the assembly*

Open *spindle\_cover.asm* with all the parts activated.

**Note**

Click the Select command and click on each part in PathFinder. Each relationship that has been used to position the assembly can be viewed by moving the cursor over the relationships in the lower pane of PathFinder. These existing assembly relationships as well as Live Rules are honored when manipulating geometry with Solid Edge.

- ▶ Open *spindle\_cover.asm*. Activate all the parts.



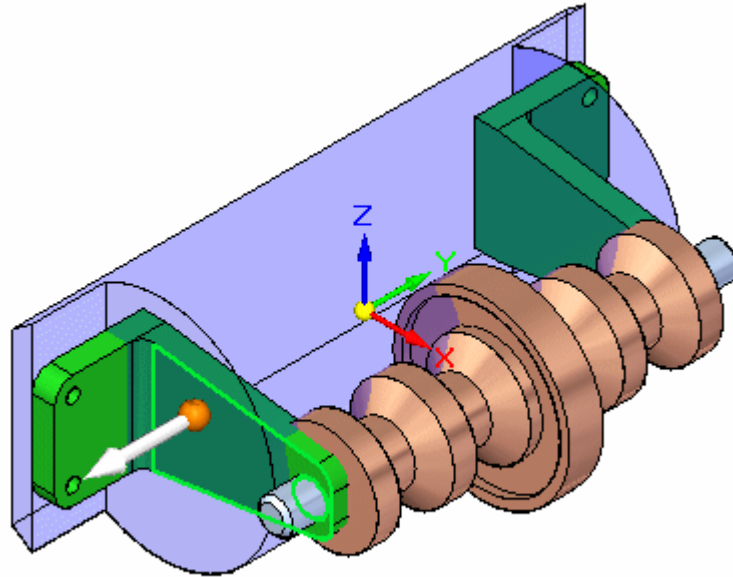
*Modify the plastic part to make the brackets fit*

The brackets holding the axle do not fit correctly on the plastic cylindrical part, and the face of the plastic part that the brackets attach to is not wide enough to accommodate the brackets. You will modify the parts in the context of the assembly to correct these problems.

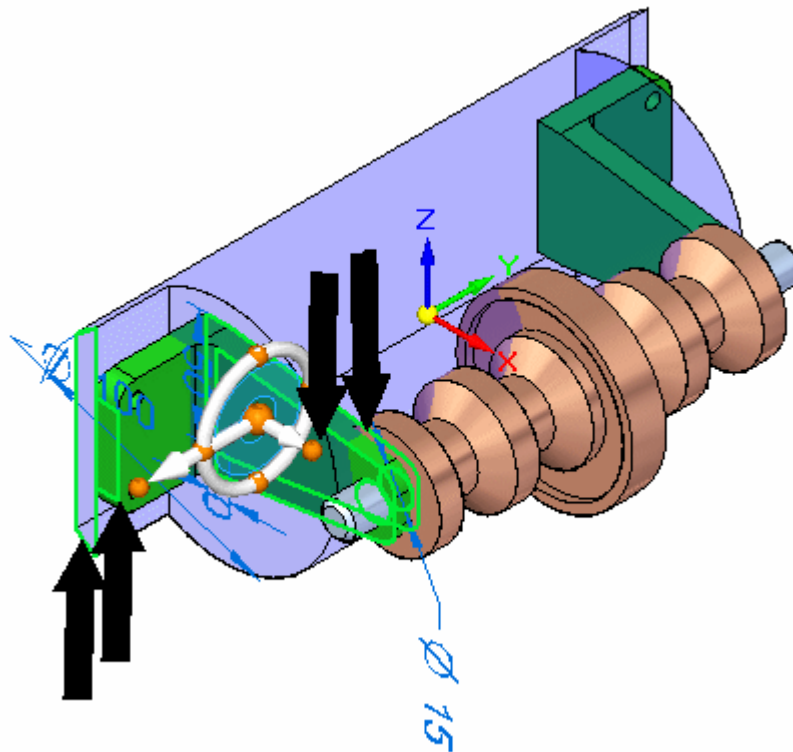
- ▶ Set the Selection Priority to Face.



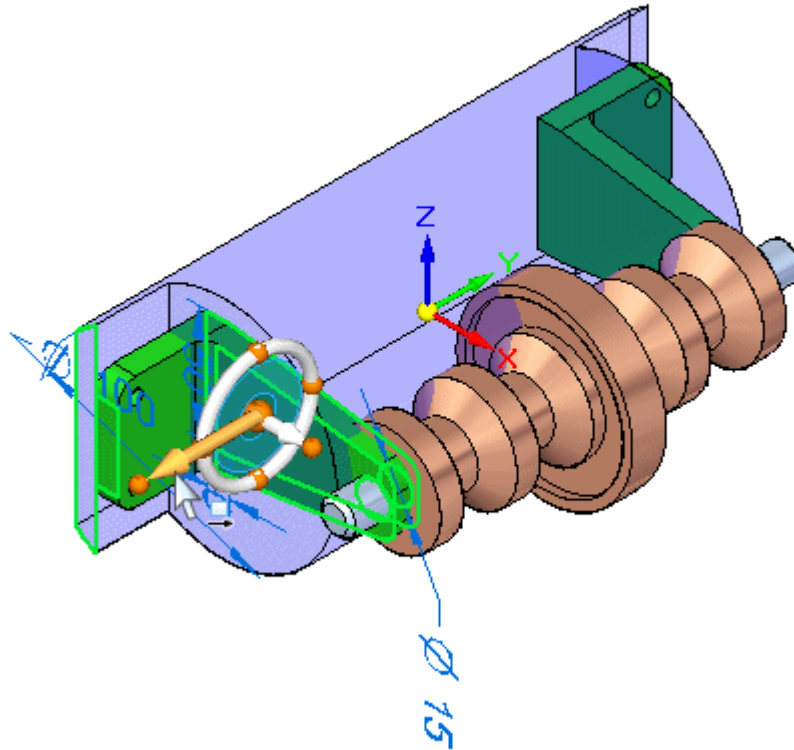
- ▶ Select the face shown with the steering wheel in the position shown.



- ▶ Select the additional faces shown. You should have a total of 4 faces selected.



- ▶ Select the axis of the steering wheel as shown.

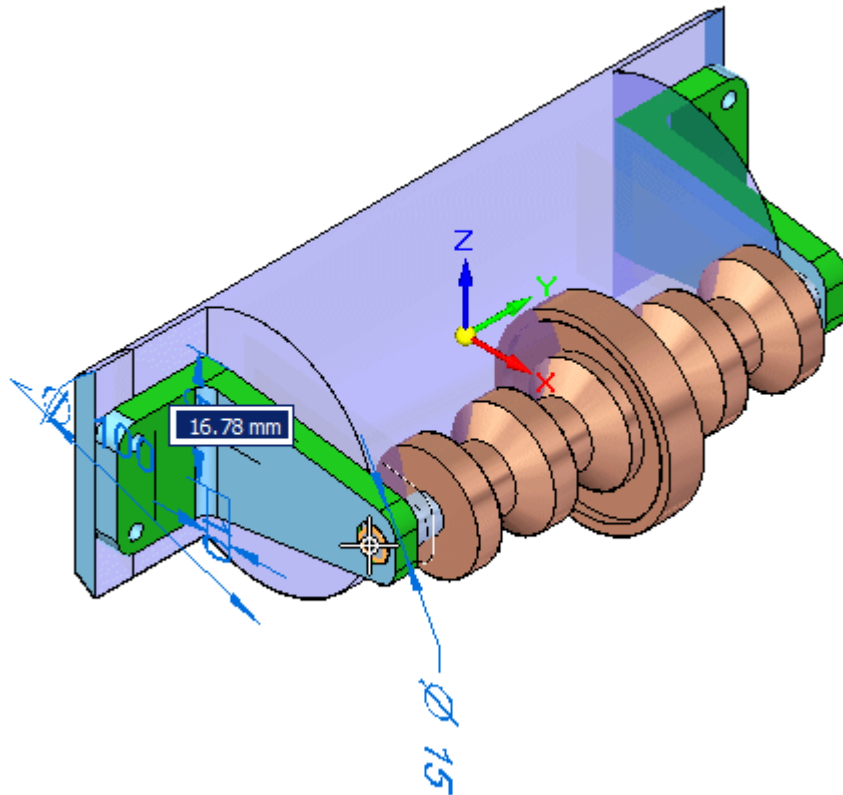


- ▶ On the QuickBar, select the circle center key point.





- ▶ Click the circular center of the end face of the axle. The 4 faces will move.



**Note**

Because symmetry about the base reference planes is set in live rules, the parts are modified on the opposite side as well.

- ▶ Clear the Selection.



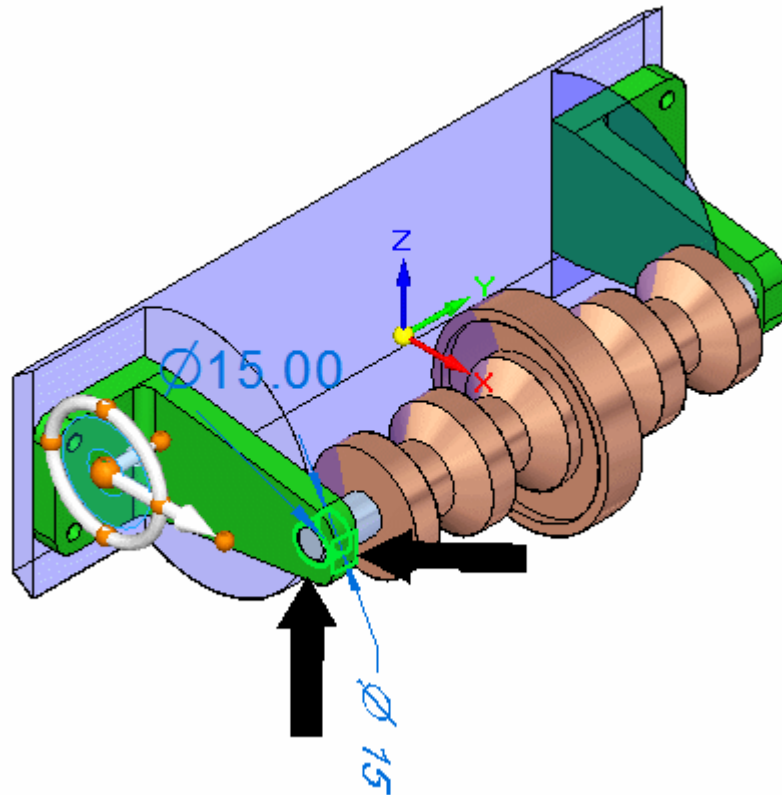
*Shorten the brackets*

Because the brackets are too tall, the axle is too far out from the plastic housing. You will shorten the bracket.

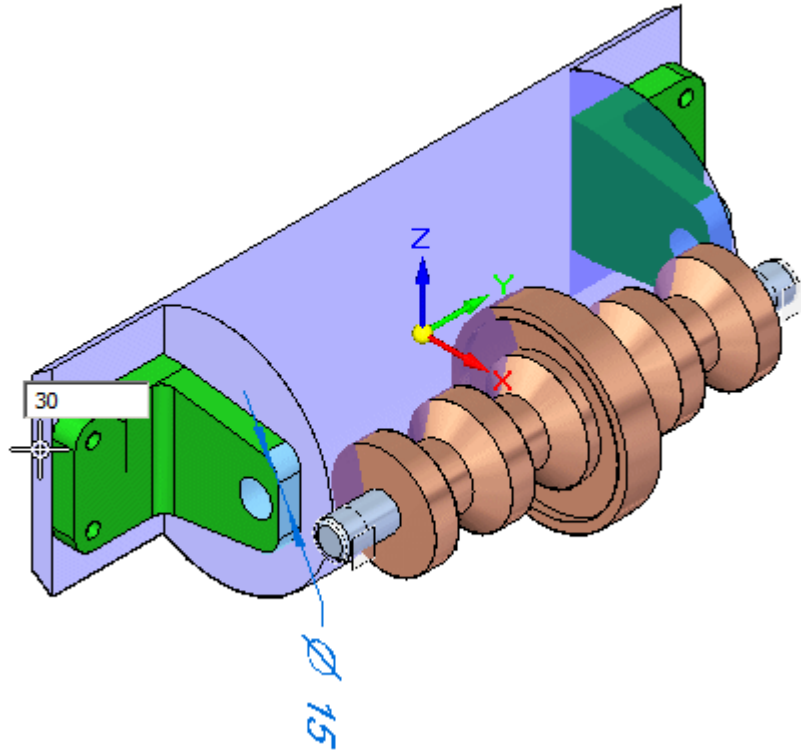
- Select the face and the cylinder shown. Move the origin of the steering wheel to the face shown.

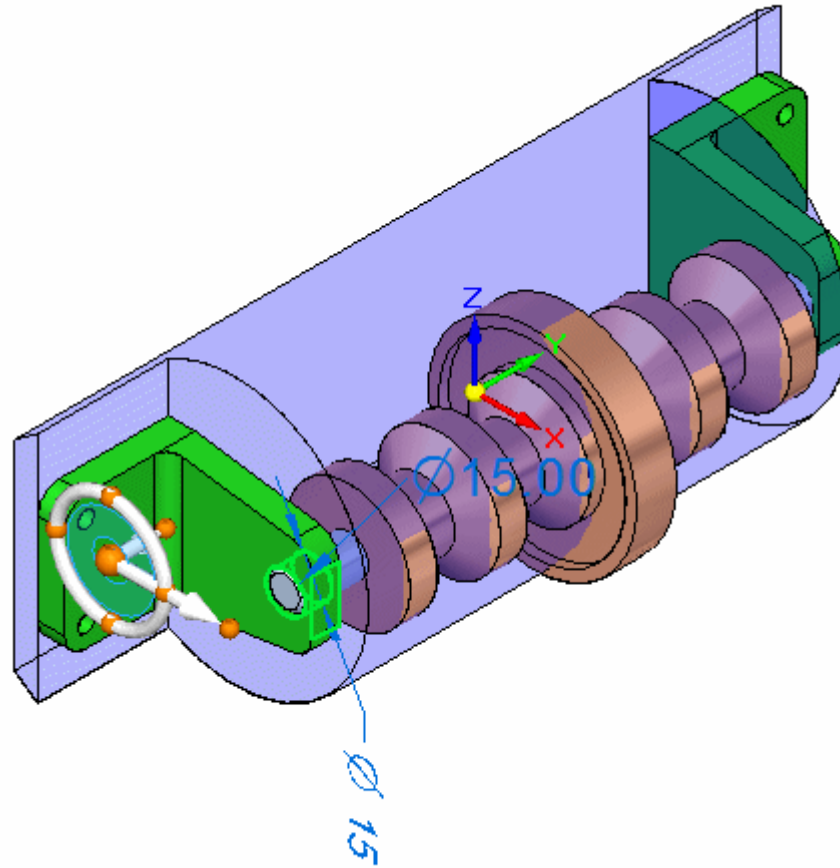
**Note**

The steering wheel can be relocated by dragging the origin knob which is the large sphere at the center.



- Drag the primary axis on the steering wheel so that the bracket becomes shorter. Enter 30.00 mm. The axial alignment between the bracket and the axle force the axle to stay aligned with the hole in the bracket.



**Note**

The axle moves with the bracket because of the axial align relationship used to place the axle.

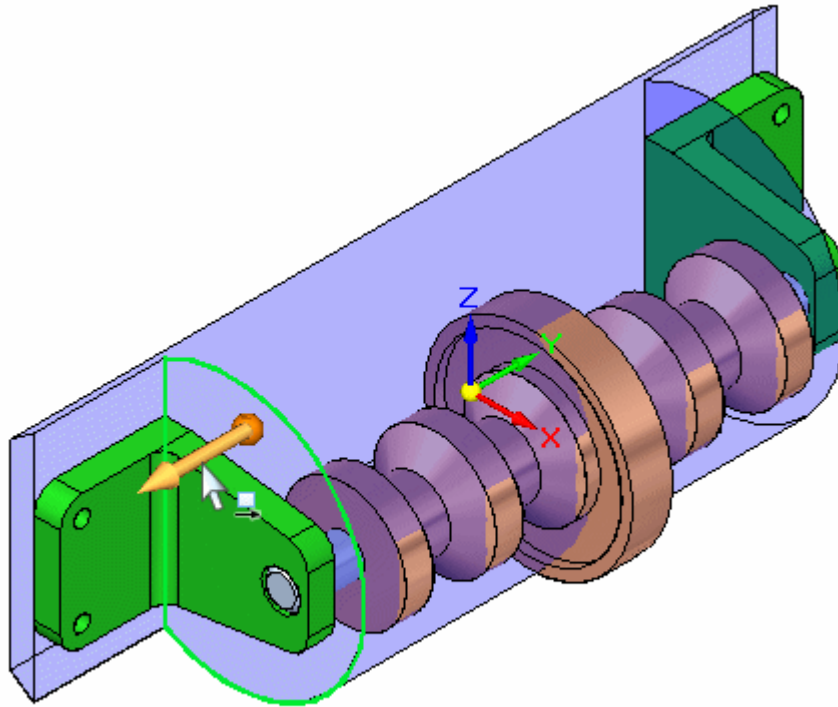
- Clear the Selection.



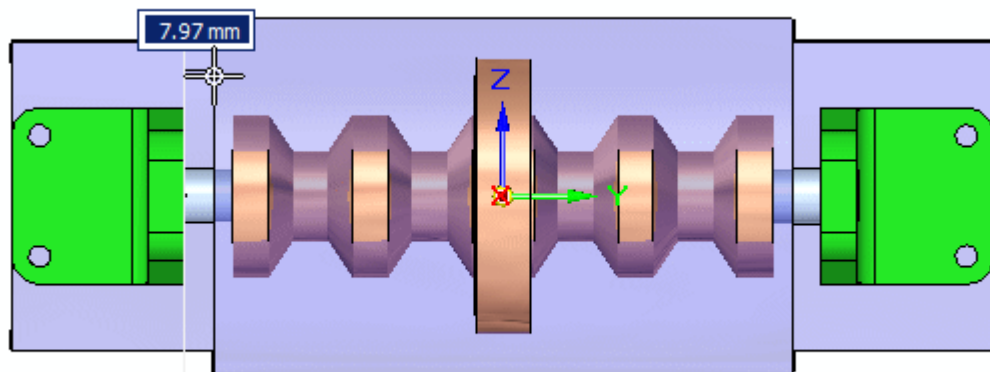
*Create clearance between the bracket and plastic housing*

The face of the plastic part is too close to the bracket. You will move the face inward.

- Select the face shown.



- Rotate the view to a right view and move the face somewhere between the bracket face and the next face on the shaft. Exact placement is not important. Because the part is symmetrical about the base, Live Rules are controlling the behavior such that the opposite face is also positioned correctly.



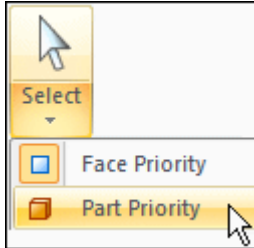
*In-place activate the plastic part and create inter-part geometry for cutting the part*

After in-place activating the plastic part, inter-part faces and an inter-part body will be created from other parts in the assembly.

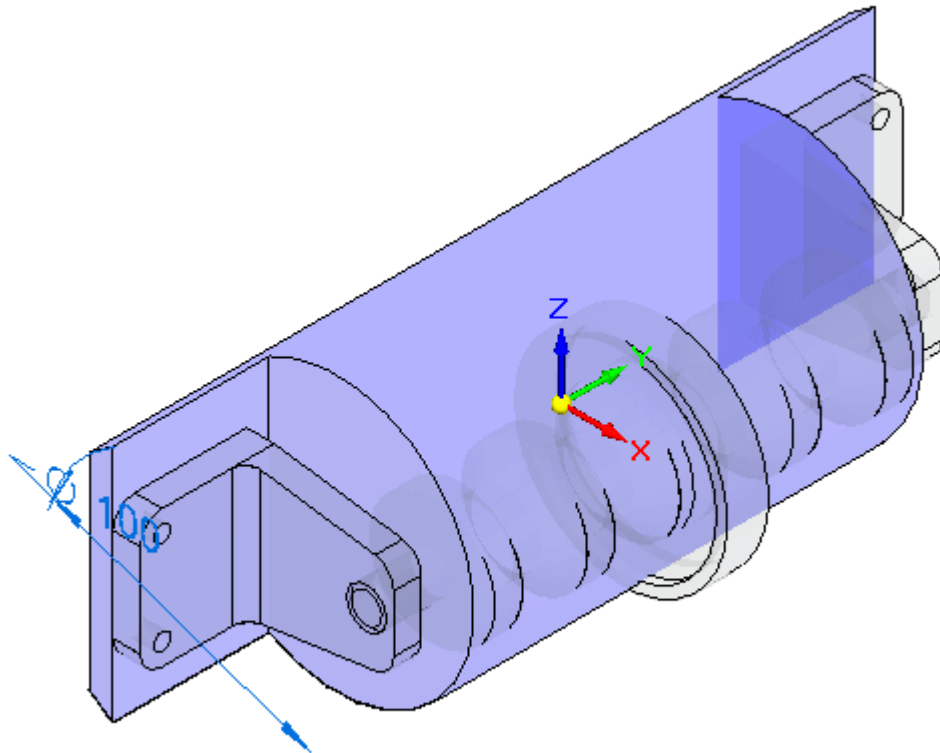
- ▶ Clear the select set.



- ▶ Set the selection criteria to Part.



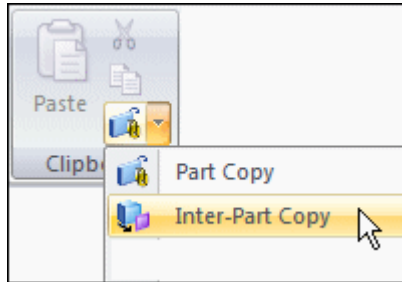
- ▶ Double-click the plastic part to in-place activate the part. You are now in the part environment but can still see the other parts in the assembly.



### Note

Using Inter-Part Copy, you will copy needed geometry from the assembly. You need two planar faces to create bolt holes in the plastic part to attach the brackets. You will also need the body of *beltdrive.par*.

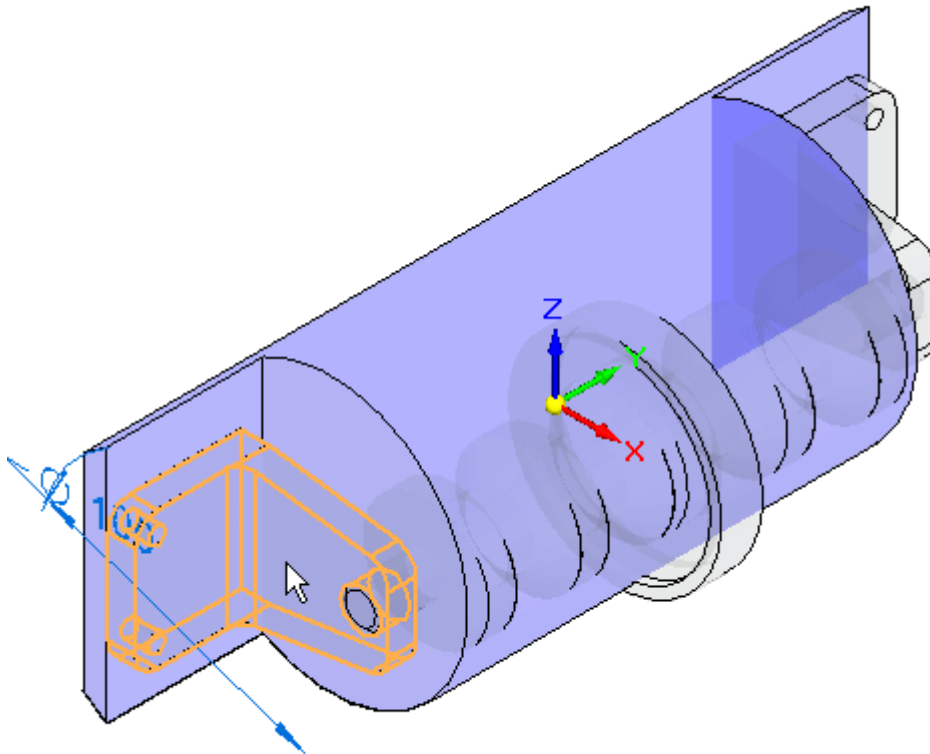
- ▶ Click the Inter-Part Copy command.



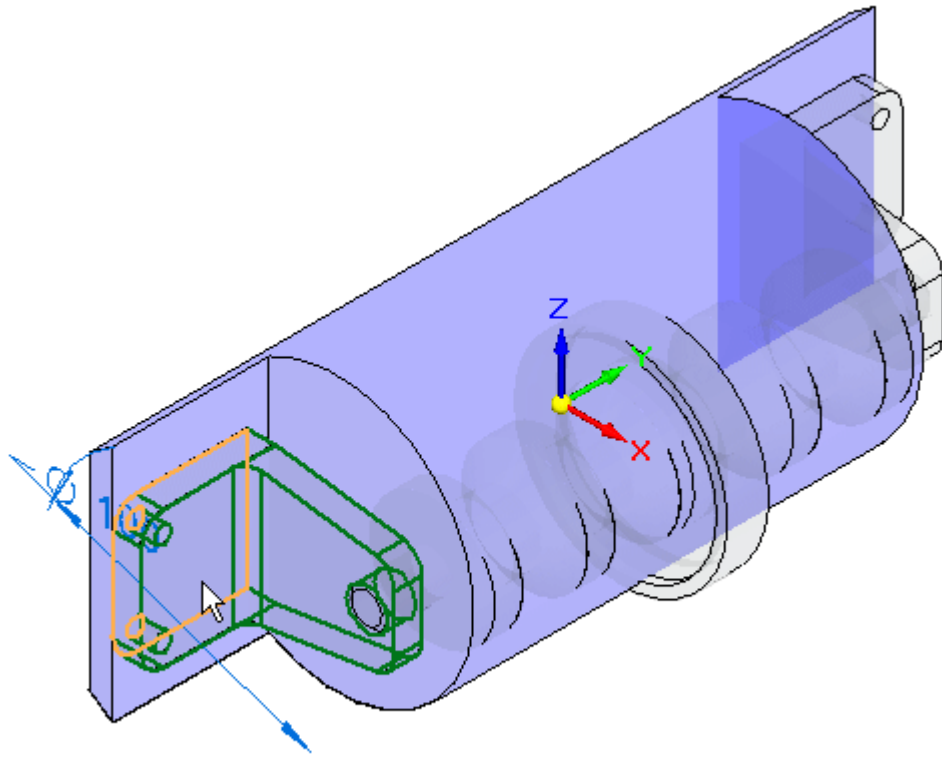
**Note**

Linked Inter-Part copies of surfaces can only be created in the ordered environment. The Inter-Part surface created for this exercise does not need to be linked and we will stay in the synchronous environment for this operation.

- ▶ Select the bracket.



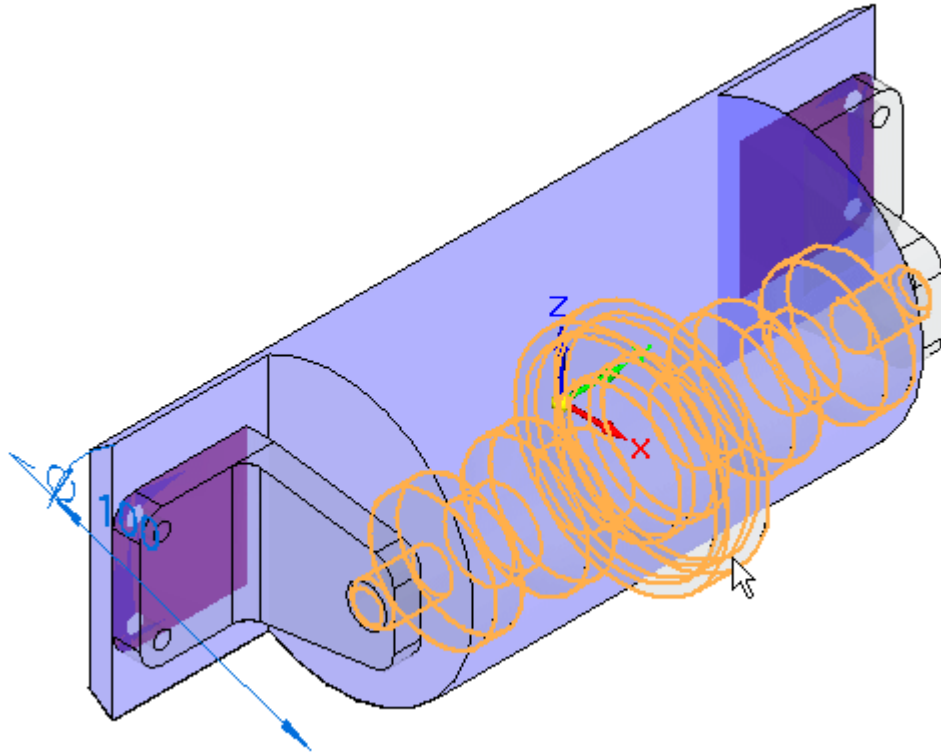
- ▶ In the Select Faces step in the command bar, select Face. Select the face shown.



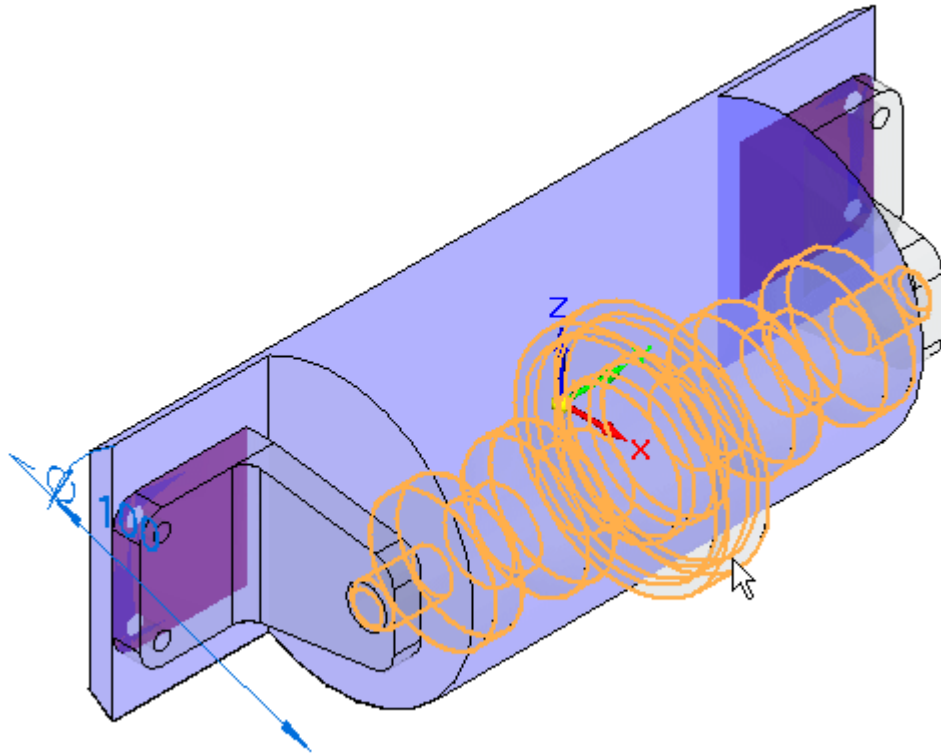
- ▶ Accept and then click Finish. Repeat for the opposite side.



- ▶ Click the Inter-Part Copy command and select the part shown.



- ▶ In the Select Faces step in the command bar, select Body. Select the whole body shown.



- ▶ Accept the body, then click finish. The body is created.

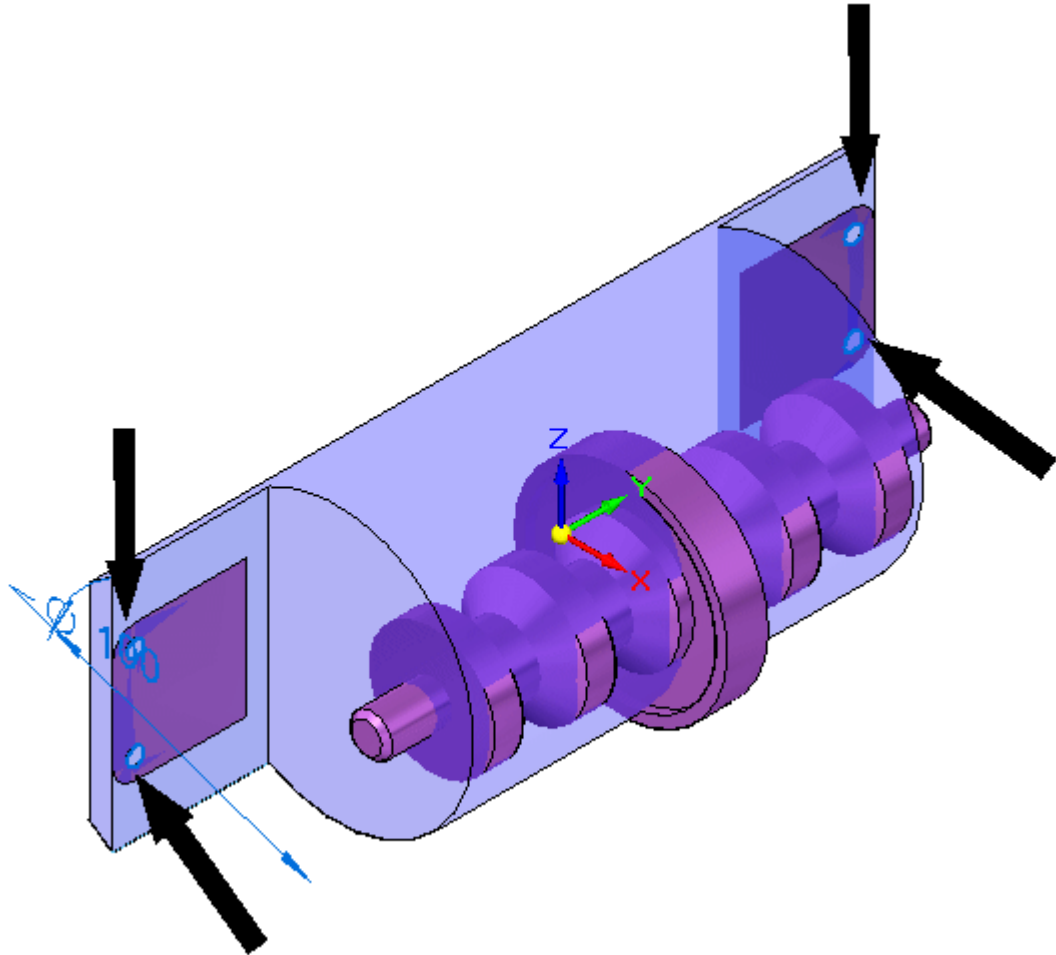
#### *Cut the plastic part using the inter-part faces*

The two inter-part faces and the inter-part body, and a cutout will be used to cut the plastic part.

- ▶ Click the View tab and in the Show group, click Hide Previous Level. This will turn off the display of the other parts in the assembly

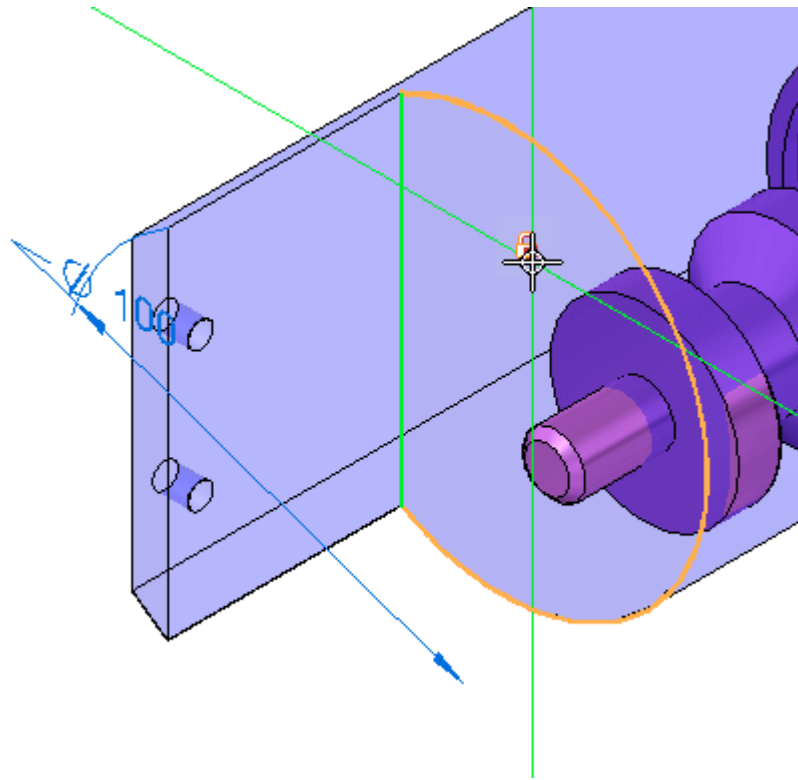


- ▶ Click the Sketching tab. Lock the sketch plane to the face containing the inter-part copies. Click Project to Sketch and select each of the 4 holes on the two Inter-Part faces.

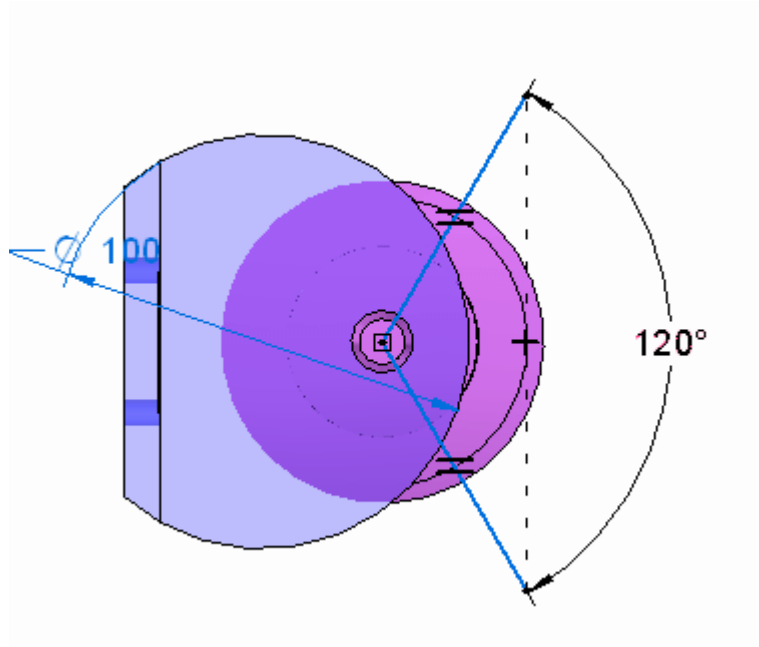


- ▶ Hide the Inter-Part Copy faces, used to create the holes, in PathFinder. Click the Extrude command. Create cutouts from each of the holes.

- ▶ Now you will draw a sketch for the first cutout in the housing. Select the sketch plane shown.

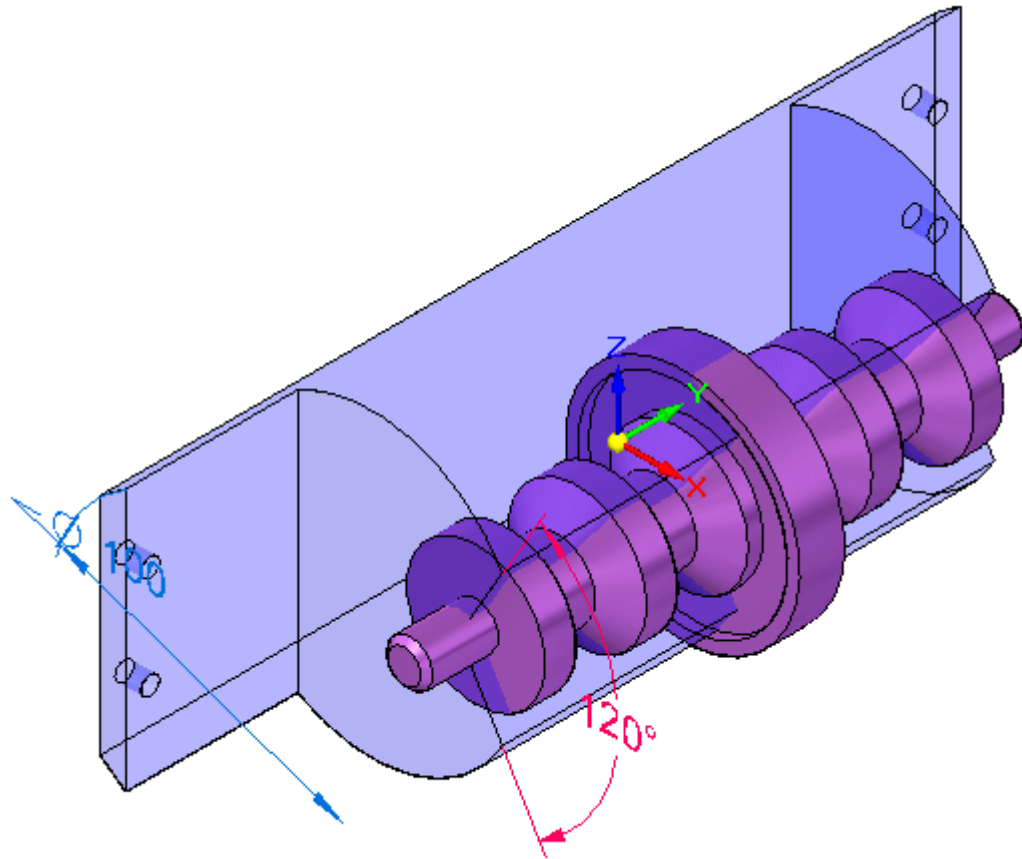


- ▶ Draw the sketch below and create an open cutout that extends the full length of the part.

**Note**

In the Relate group on the ribbon, use the Equal relationship to make the lines the same length. The angle between the lines is  $120^\circ$ . Use the Horizontal/Vertical relationship to line up the ends of the lines vertically. Intellisketch may put a perpendicular relationship at the intersection of the two lines. You will need to delete that relationship in order to get the  $120^\circ$  driving dimension placed.

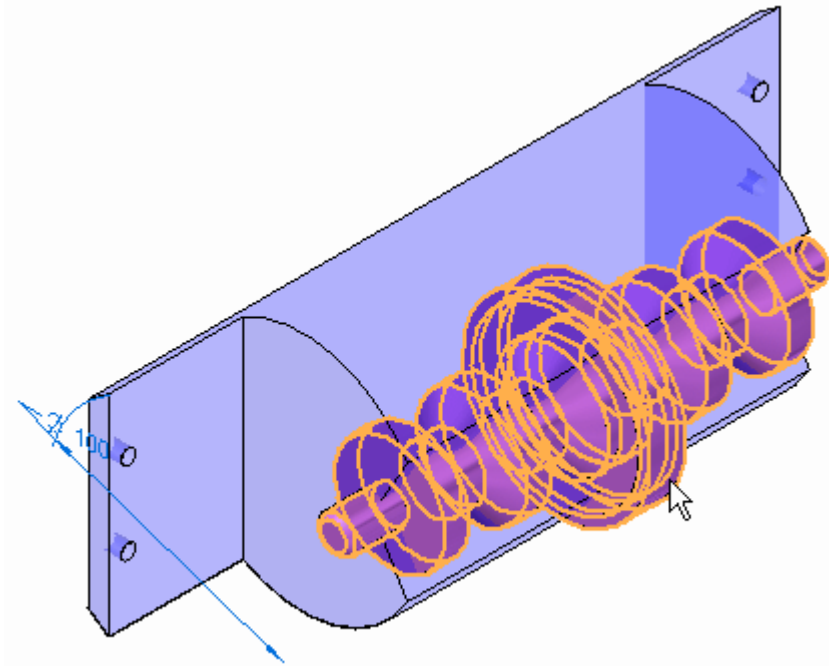
The part is shown.



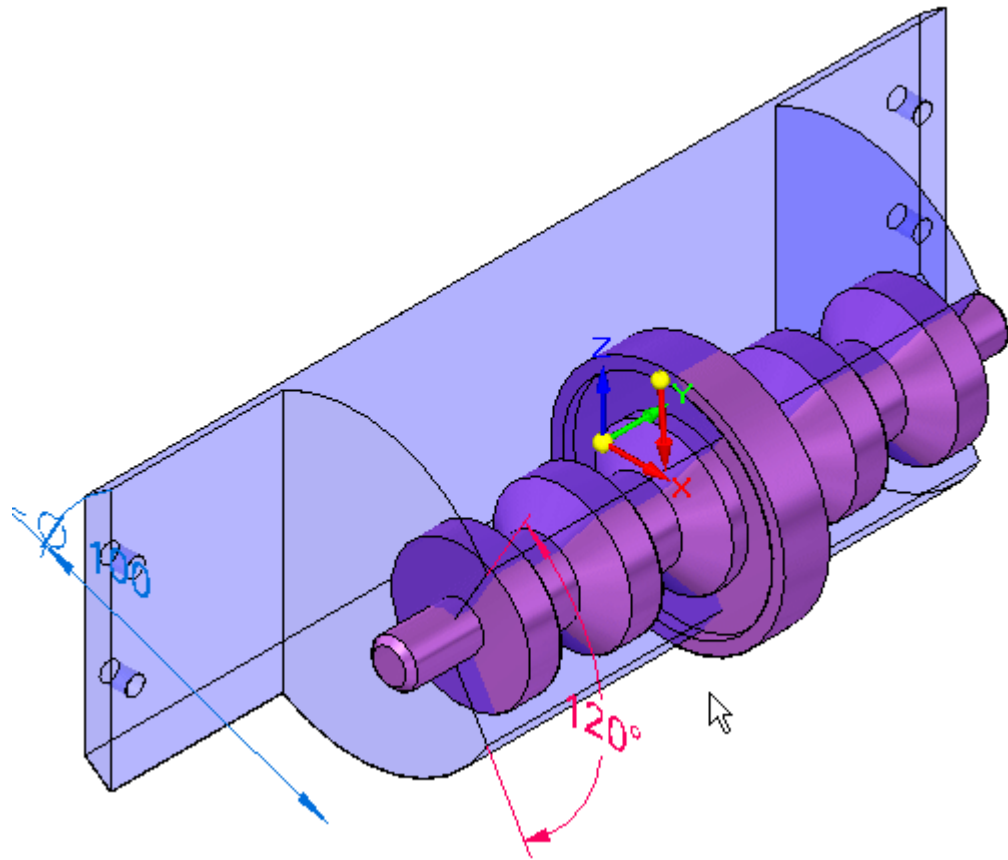
- ▶ Click the Surfacing tab. In the Surfaces group, click the Offset command.



- In the Select step in the command bar, set Select to Body. Select the Inter-Part Copy shown and accept.

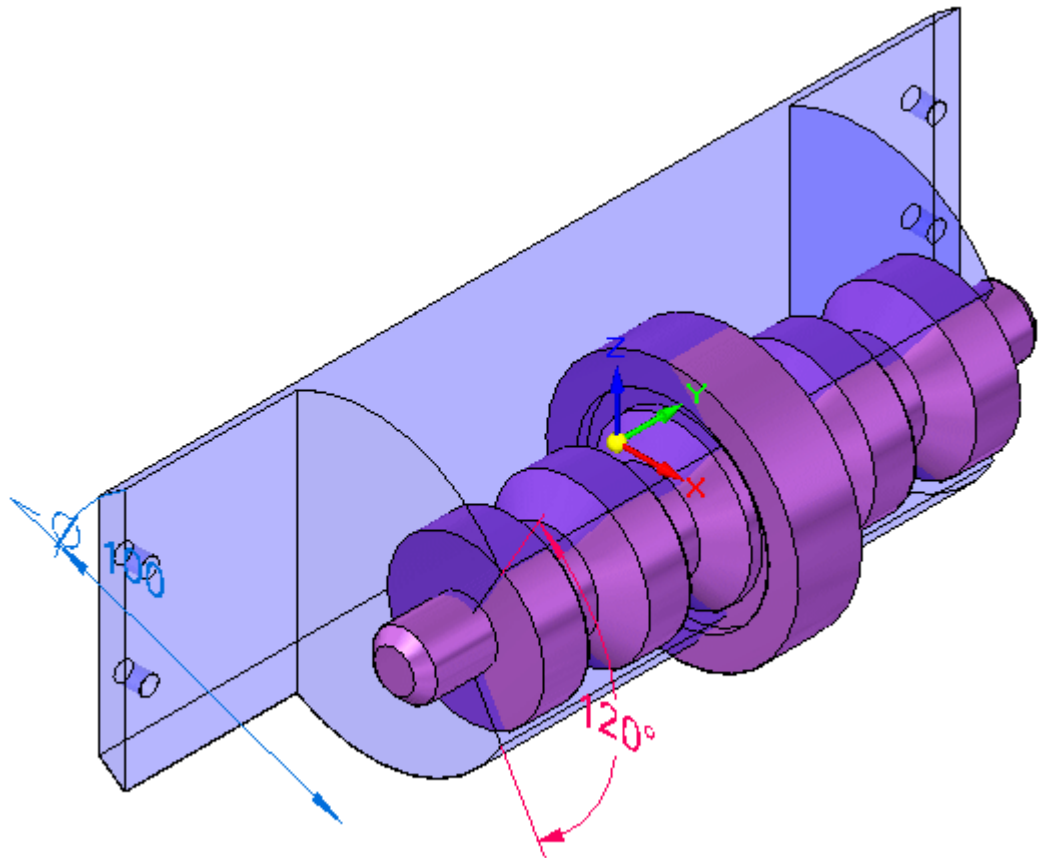


- ▶ Enter 3.00 mm for the offset distance. For the direction, click as shown. Then click Finish.





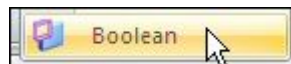
- ▶ The offset surface is shown. Notice it is bigger than the Inter-Part Copy.



**Note**

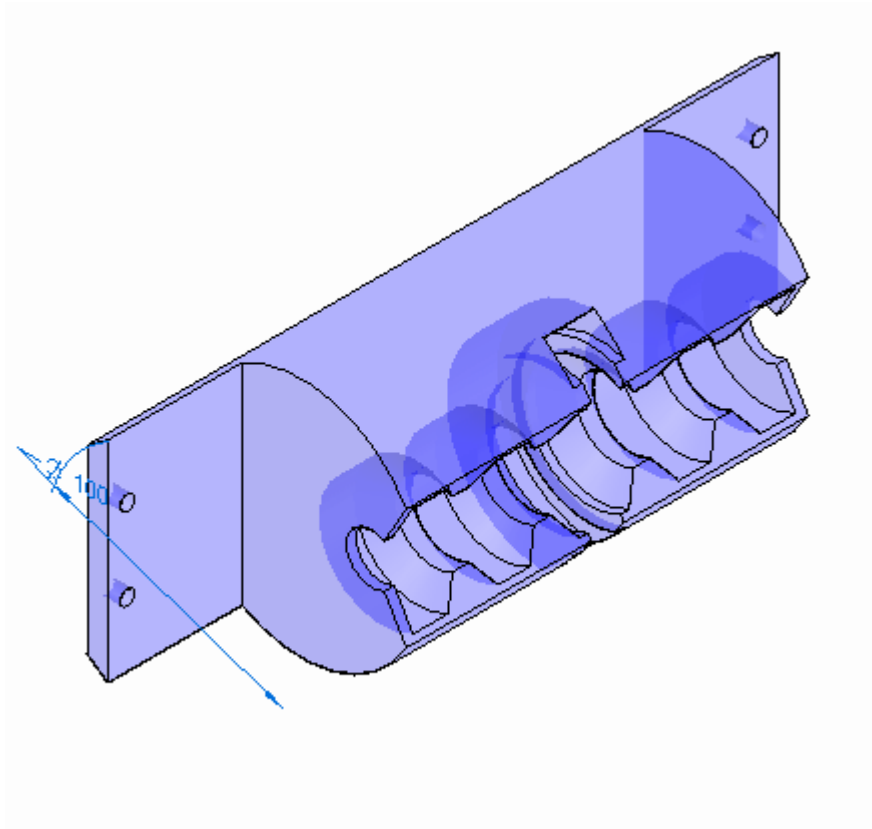
If the offset is smaller than the original part, you have chosen the wrong direction and will need to repeat the operation.

- ▶ Turn off the Inter-Part Copy of the body in PathFinder.
- ▶ Click the Surfacing tab. In the Surfaces group, click the Boolean command.

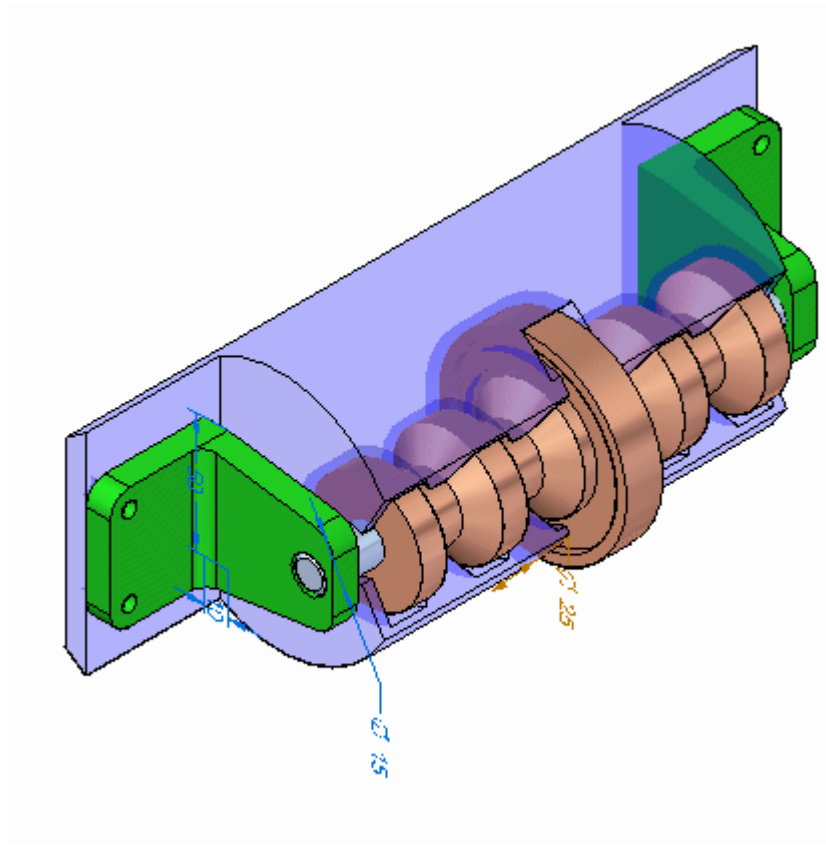


- ▶ In the command bar in the Tool Step, set the select to Body and click Subtract.
- ▶ Select the offset surface and accept. Then click Finish.

- ▶ Hide the offset surface in PathFinder.  
The part is as shown.



- ▶ Click the Home tab and then click Close and Return to return to the assembly. The assembly is as shown.

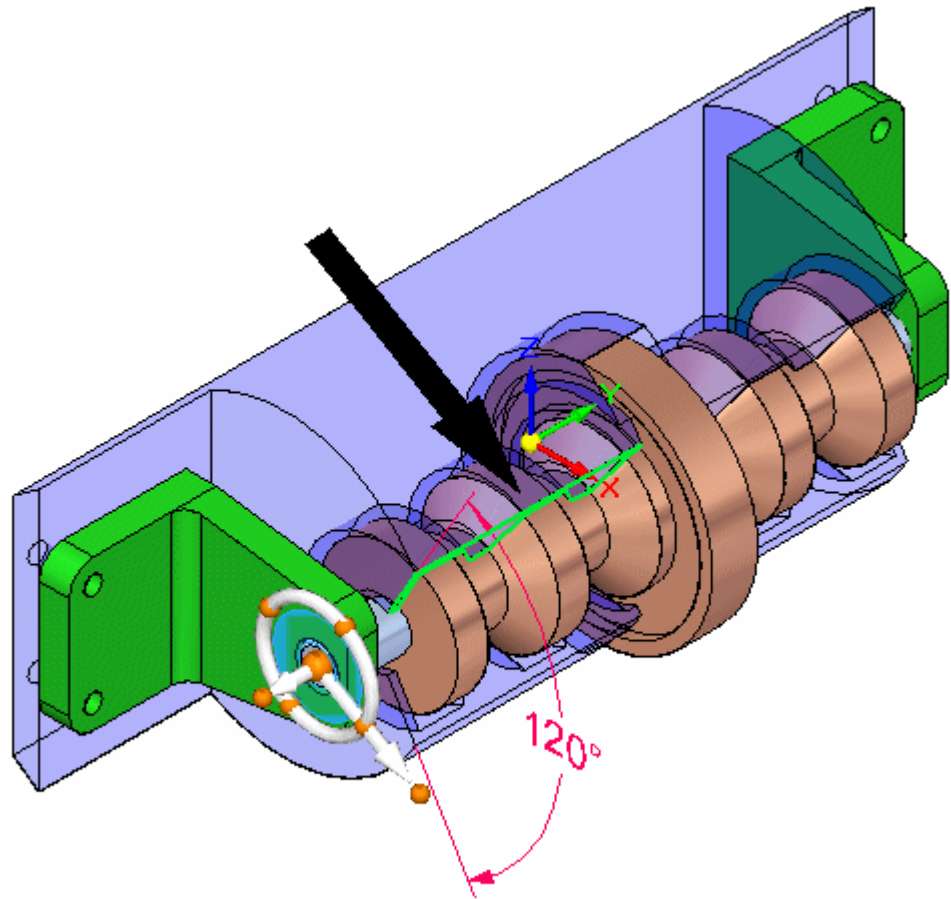


*Modify the angle of the opening in the plastic part*

Now you will modify the angle of the angular cut in the plastic face.

- ▶ Set the Selection Priority to Face.

- Select the face shown and move the steering wheel so that the primary axis aligns with the axis of the axle as shown.

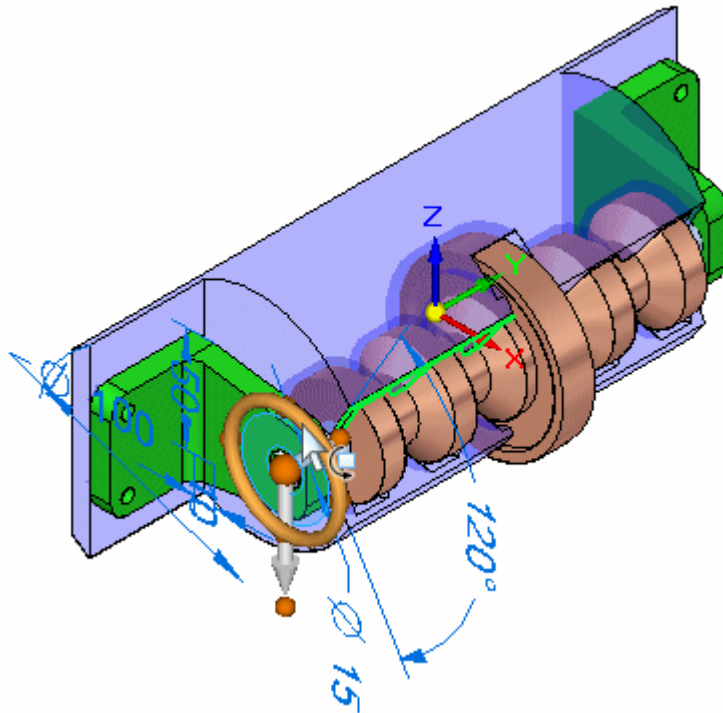


- Select the steering wheel torus as shown and rotate with the mouse. Notice the angle of the cut in the plastic part changes.

**Note**

Make sure the 120° dimension is unlocked so the face can move

This completes the activity.



*Activity summary*

In this activity you learned how to modify parts in the context of an assembly using Solid Edge.

**Lesson review**

Answer the following questions:

1. What is one advantage to modeling assemblies using synchronous components?
2. Can synchronous components in an assembly be copied using the steering wheel?
3. When using the steering wheel to move or copy an assembly component, when can live rules be used?

## **Answers**

1. What is one advantage to modeling assemblies using synchronous components?  
Faces and components can be selected and modified directly in the assembly environment using the steering wheel and live rules.
2. Can synchronous components in an assembly be copied using the steering wheel?  
Synchronous geometry can be moved or copied using the steering wheel. Solid Edge will attempt to repair relationships that become invalid after a move if directed to do so.
3. When using the steering wheel to move or copy an assembly component, when can live rules be used?  
Live rules can be used with the steering wheel in assembly when faces are selected, and the geometry is synchronous rather than ordered.

**Lesson summary**

In this lesson you learned how to modify parts in the context of an assembly using Solid Edge.



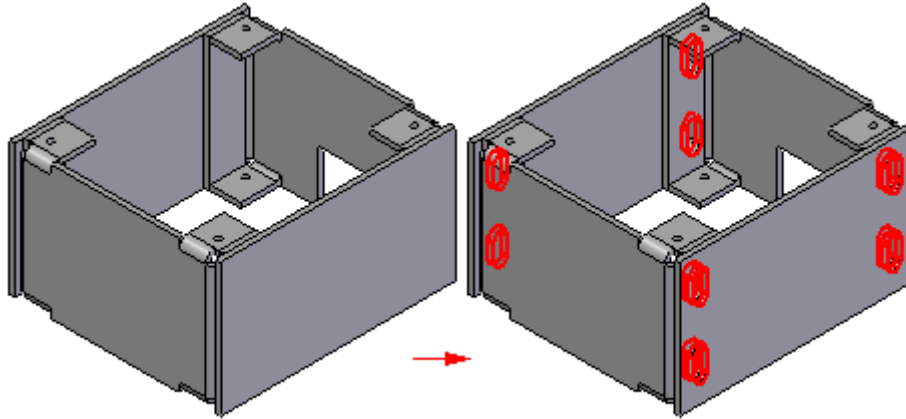
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**Lesson**

## **2**    *Assembly features*

## Assembly-based features

In some assemblies, you may want to construct a single feature that modifies more than one part. For example, a pattern of cutouts may extend through several parts.



In other cases, you may want to construct a feature within the context of the assembly, rather than in the Part or Sheet Metal document.

You can use the feature commands in the Assembly environment to construct features such as cutouts, revolved cutouts, holes, chamfers, and threads. You can also mirror and pattern these features.

In Solid Edge, you can construct two types of assembly-based features:

- Assembly Features
- Assembly-Driven Part Features

You use the Feature Options dialog box to specify whether the assembly-based feature you are constructing affects only the selected parts (Assembly Feature), or affects all parts with the same document names as the selected parts (Assembly-Driven Part Feature).

### Note

The Feature Options dialog box is not available until you set the Assembly-Driven Part Feature option on the Inter-Part tab of the Options dialog box. When this option is disabled, you can only construct assembly features.

You can use the assembly-based feature commands to modify the solid body, but not any surface bodies (construction bodies) that may reside in the part documents.

Some types of assembly-based features can only be constructed as assembly features. For example, the Chamfer and Thread commands do not support the Assembly-Driven Part Feature option.

## **Assembly Features**

When you set the Create Assembly Features option, the feature affects only the parts you select.

For profile-based features, the reference plane, profile, extent definition, and surface geometry resides in and can only be viewed from the assembly document where the feature was constructed.

This option is useful when you want to modify one or more commonly-used parts, but do not want to affect other instances of the same parts in the active assembly or other assemblies that reference those parts.

You do not need write access to the affected part files to construct an assembly feature.

## **Assembly-Driven Part Features**

When you set the Create Assembly-Driven Part Features option, the feature affects all parts with the same document name as the selected parts.

The reference plane, profile, and extent definition, resides in and can only be viewed from the assembly document where the feature was constructed. The surface geometry resides in the part documents. The surface geometry can be viewed in the part documents, and any other assemblies that reference the part documents.

This option is useful when you want to globally modify one or more parts while working in the assembly.

You need write access to the affected part files to construct an assembly-driven part feature.

A part document that contains an assembly-driven part feature is associatively linked to the assembly document where the feature definition (reference plane, profile, and extent definition) resides.

You can break the links that control an assembly-driven part feature by in-place activating the part document, selecting the feature in PathFinder, and clicking the Break Links command on the shortcut menu. After the link is broken, it is likely that some aspects of the feature will become undefined. For example, the reference plane may no longer be associated to its parent face. You can repair the broken links by selecting the feature, and using the Edit Definition option on the command bar to redefine the reference plane, feature extents, and so forth.

### **Note**

When you set the Assembly-Driven Part Feature option on the Feature Options dialog box, any existing assembly features are temporarily hidden while constructing the assembly-driven part feature. This prevents you from using edges on assembly-driven part features to construct the assembly feature.

When you finish constructing the assembly-driven part feature, the assembly features are redisplayed.

## Drawing the Profile

When constructing a profile-based assembly feature, you can draw the profile on one of the default assembly reference planes or use a part face to define a profile plane.

You can use the draw commands to draw the profile, or if you have created an assembly sketch, you can use the Select From Sketch option available when constructing a cutout or revolved cutout. The Text Profile command is not available when creating assembly features or assembly-driven part features.

## Dimensioning the Profile

You place dimensions to the profile elements in the same way as you would a profile-based features for a part. Before you can place dimensions or geometric relationships to part edges, you must set the Peer Edge Locate command on the Tools tab. You can then place dimensions or relationships between the assembly feature profile elements and the edges of other parts in the assembly.

## Defining the Extent

You can specify that the feature extends through all the parts, extends by a finite distance value, or define a from/to extent. When defining a from/to extent, you can select an assembly reference plane or a part face to define the feature extent. Associative links are created when you select a reference plane or part face to define the extent, and the links are displayed in the bottom pane of PathFinder when the feature is selected.

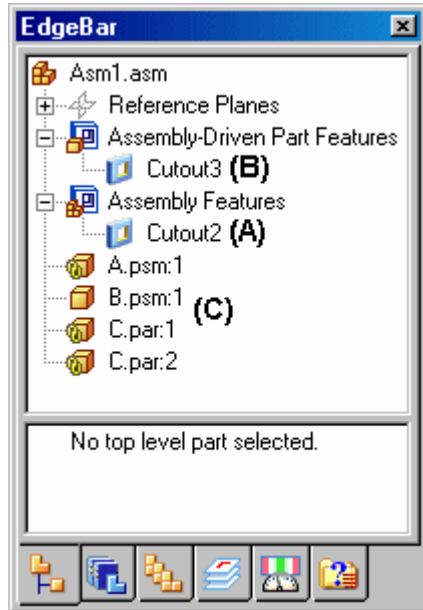
## Selecting the Parts

You do not have to apply the feature to every part that the profile geometry passes through. The Select Parts step allows you to specify which parts you want to modify. Parts that lie within the range of the profile and the feature extent are automatically selected. You can add parts to the select set by clicking on the part you want to add, and you can deselect parts by holding the SHIFT key and selecting parts that are highlighted.

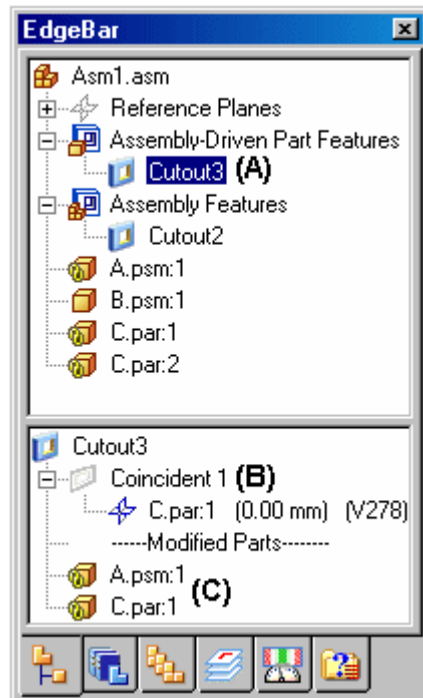
You can select a subassembly as the select set of parts to modify, and all parts that intersect the feature are modified. If parts are added to the subassembly later, the new parts are not added to the feature scope. If you want the new parts to participate in the feature, you must edit the feature and return to the Select Parts step and select the new parts to which you want to apply the feature.

### Assembly-based Features in PathFinder

Separate lists are maintained for assembly features (A) and assembly-driven part features (B) in PathFinder. When constructing assembly-driven part features, the symbols for the affected parts (C) are changed to show they have links to the assembly.



When you select an assembly-based feature in the top pane of PathFinder, the parents that control the feature are displayed in the bottom pane of PathFinder. For example, when you select Cutout3 (A) in the top pane, in the bottom pane it shows that C.PAR is the parent part for the coincident reference plane on which the profile was constructed. Also listed are the parts (C) that were modified by the feature.



## Editing Assembly-based Features

To edit an assembly-based feature, select the feature entry in PathFinder and click the Edit Definition button on the command bar. The command bar for the feature is displayed, so you can select the step you want to modify. For example, you can return to the Select Parts step and select additional parts to which you want to apply the feature.

The Select Feature Components command on the Pathfinder shortcut menu can be used to select the parent components for an assembly feature. This can be useful in determining the parent components for the feature, and for managing the component display. For example, after you select the parent components, you can use the Show, Show Only, and Hide commands to display or hide the parent components for editing purposes.

### Note

Assembly features and assembly-driven part features are not included in the undo list.

## Opening Part Documents with Assembly-Driven Part Features

When you open a part document that contains an assembly-driven part feature, a dialog box reminds you that the part document has links to the assembly where the feature was constructed. The dialog box allows you to specify whether you want to open the assembly document or the part document.

## Mirroring and Patterning Assembly Features

You can mirror and pattern only assembly features that were constructed in the current assembly. You cannot mirror or pattern assembly-driven part features. You can specify which parts are included in the mirror or pattern feature. If you add parts to the mirror or pattern feature which were not in the parent feature, a message is displayed to instruct you the parts must also be added to the parent feature.

## Suppressing Assembly-based Features

You can use the Suppress and Unsuppress commands on both types of assembly-based features, but the behavior differs somewhat.

When you suppress an assembly feature, a symbol is added adjacent to the feature in PathFinder to indicate that the feature is suppressed. When you suppress an assembly feature, the feature is suppressed for all the parts affected by the assembly feature. You cannot suppress an assembly feature for individual parts in an assembly feature.

When you suppress an assembly-driven part feature, no symbol is added to the feature in PathFinder to indicate that the feature is suppressed. Because the surface geometry for an assembly-driven part feature resides in the part document, the suppress symbol is added to the part documents in PathFinder.

You can also suppress an assembly-driven part feature on an individual part by in-place activating the part, and suppressing the feature on that part in PathFinder. When you return to the assembly where the feature was defined, no indication of the suppress operation is displayed in PathFinder, but the graphic display is updated to for the part that had the feature suppressed. You can redisplay the feature using the Unsuppress command in the assembly document or the part document.

## **Placing an Assembly that contains Assembly-based Features as a Subassembly**

Both assembly features and assembly-driven part features are viewable and can be used for positioning purposes when the assembly is placed as a subassembly in another assembly.

## **Assembly-based Features and Draft Documents**

When constructing drawings of an assembly that contain assembly features, you can specify whether the assembly features are displayed in a drawing view using the Drawing View Properties dialog box. This applies only to assembly features, not assembly-driven part features. Because assembly-driven part features physically modify the part document, they are always displayed in a drawing view.

## **Creating Stand-alone Documents for Components with Assembly Features**

You can save an individual part in a assembly to a new document using the Save Selected Model command. This is useful when you have used assembly features to modify a part.

Because assembly features are visible only within the context of the assembly, saving a part with assembly feature modifications to a new document allows you to create a drawing for that part, prior to creating an assembly drawing. You can also use the stand-alone document for manufacturing or analysis purposes.

The documents you create using the Save Selected Model command contain an associative part copy of the part in the assembly. Associative part copies do not contain a feature tree.

You can use the Save Selected Model dialog box to specify the new file type you want. You can save the component as a Solid Edge Part document (.PAR) or a Sheet Metal document (.PSM).

## **Assembly-based Features and Alternate Assemblies**

Assembly features, but not assembly-driven part features, are allowed when working with a family of assemblies, but certain restrictions apply. Inter-part links are not allowed when working with an alternate assembly, and both types of assembly-based features allow you to create inter-part links. When converting an assembly to a family of assemblies that contains inter-part links, a dialog box is displayed to warn you that the inter-part links will be deleted.

If you anticipate creating a family of assemblies that will contain assembly-based features, you should consider using only assembly features and not place dimensions or geometric relationships to part edges. This approach does not create inter-part links. For example, you can use an assembly sketch to define the profile for an assembly feature, and use assembly reference planes for positioning dimensions and geometric relationships.

### **Assembly Features and Simplified Parts**

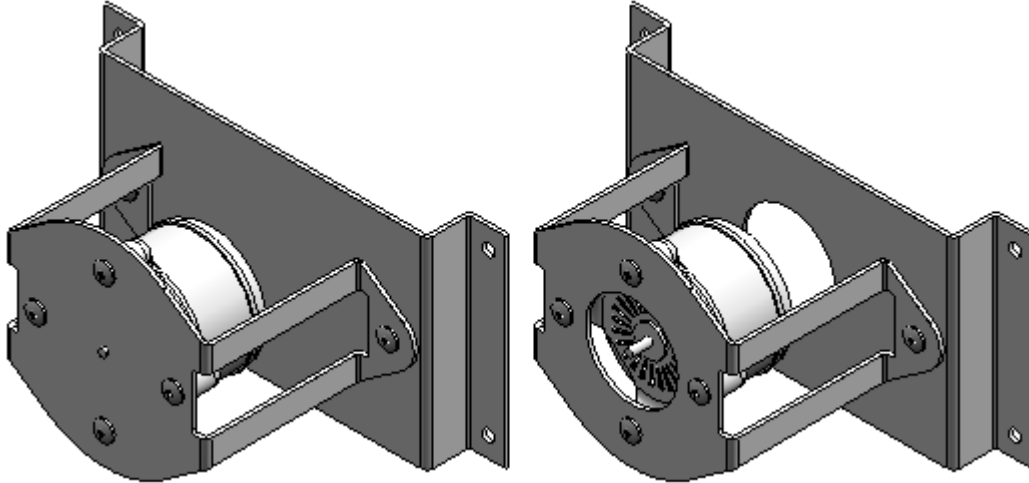
When you create an assembly feature (assembly-based feature or assembly-driven part feature), any simplified parts that are displayed in simplified mode in the assembly are redisplayed in design mode when you select them to be included in an assembly feature.

Any parts that were selected for an assembly feature cannot be displayed in simplified mode, whether the parts were modified by the assembly feature, or not modified. The simplified mode is also not available if you suppress the assembly feature.



## **Activity: Assembly features**

This activity guides you through creating a cutout traversing multiple parts in an assembly.



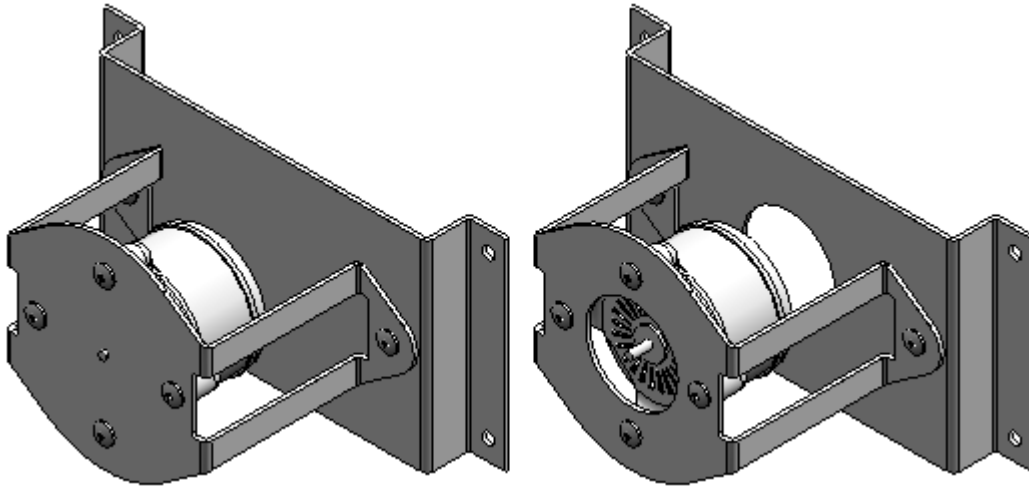
## Activity: Assembly features

### Overview

An assembly cutout allows you to cut multiple parts within the assembly environment. The resultant cutout can be applied only in the assembly (assembly feature) or it can be associatively linked back to the part or sheet metal files (assembly-driven part feature). This capability reduces the chance of error through misalignment that could happen when through holes spanning multiple parts are cut independently.

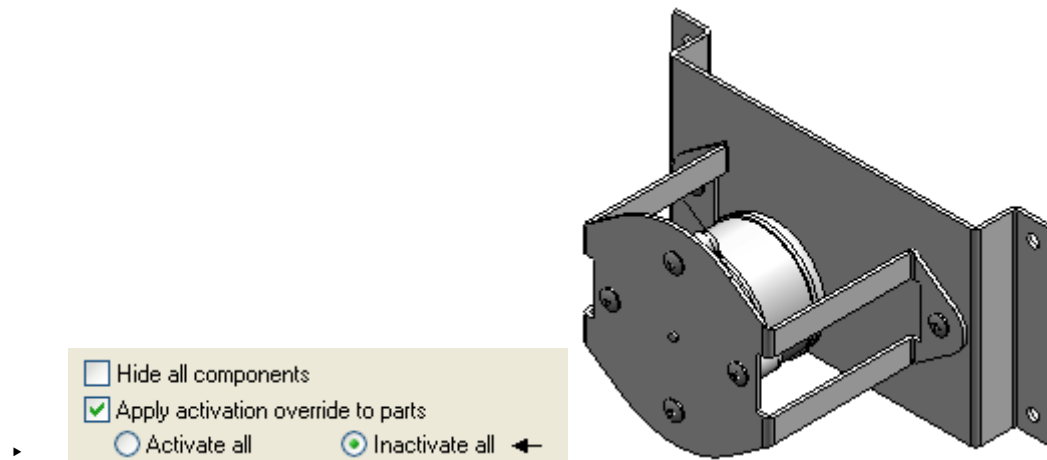
### Objectives

Learn to use the Assembly Cutout command. Assembly Cutout places a cutout in multiple parts from within the Assembly environment. To do this, draw a sketch on one of the assembly reference planes, and then select the parts to cut. For this activity, construct an assembly feature cutout in the assembly as shown in the following illustration.



### Open the assembly with inactive parts

- ▶ Open *assembly\_cut.asm*. In the Open File dialog box, select the Inactivate all option. Turn shading on or off as preferred.

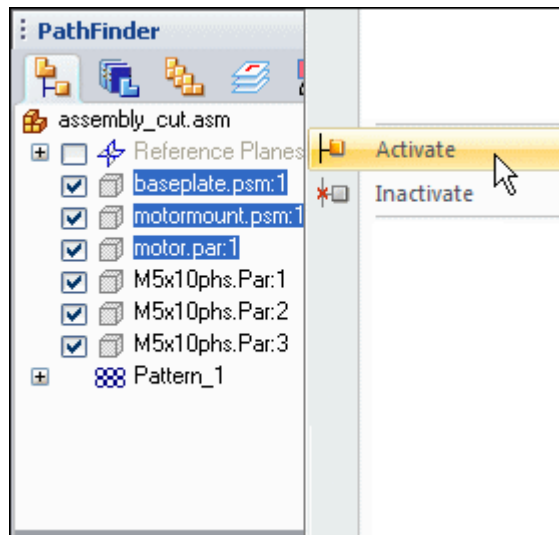


**Set the Inter-Part options**

- On the Application menu, click Solid Edge Options. In the Solid Edge Options dialog box, click the Inter-Part tab. Check all the boxes and then click OK.

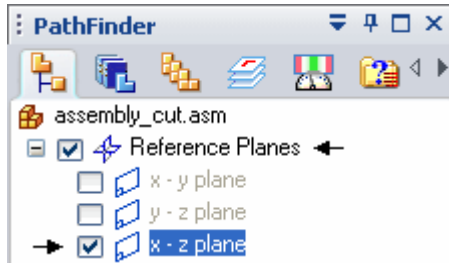
**Activate parts in the assembly in order to select them for modification**

- ▶ Activate the files *baseplate.psm*, *motormount.psm*, and *motor.par*. Select the three parts by holding down the Ctrl key. Right-click to display the shortcut menu, and then click Activate.



**Turn on the display of one of the reference planes**

- ▶ Turn on the display of the x-z plane. In PathFinder, click the check boxes for the x-z plane and the Reference Planes.

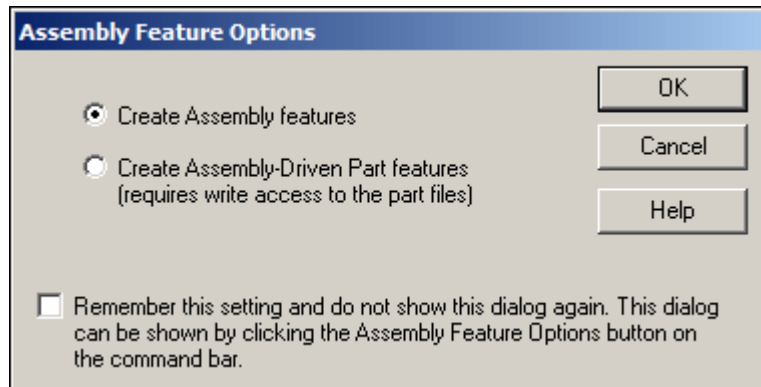


**Construct an assembly cutout through parts baseplate.psm and motormount.psm**

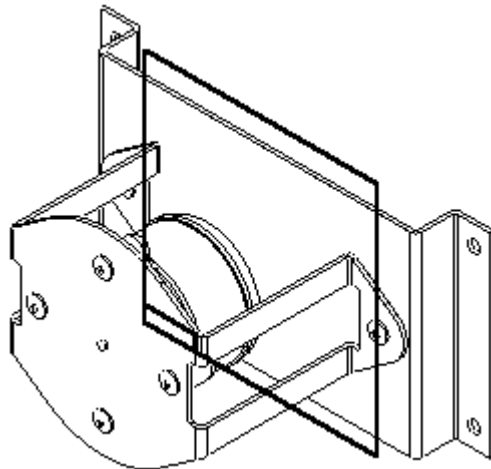
- ▶ On the Features tab, in the Assembly Features group, choose the Cut command



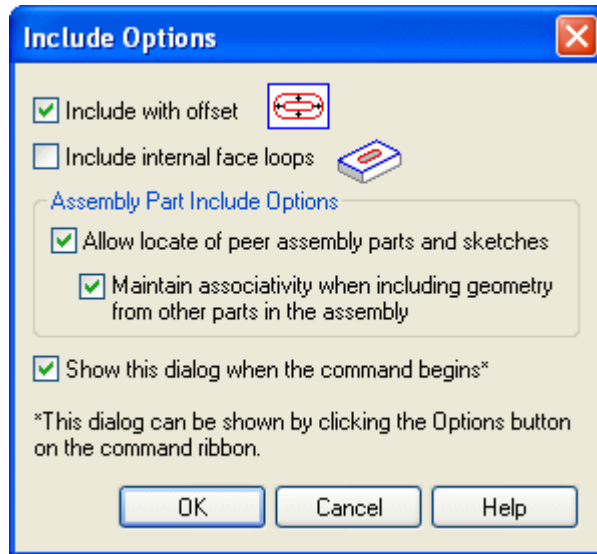
- ▶ In the Feature Options dialog box, select the Create Assembly features option and click OK.



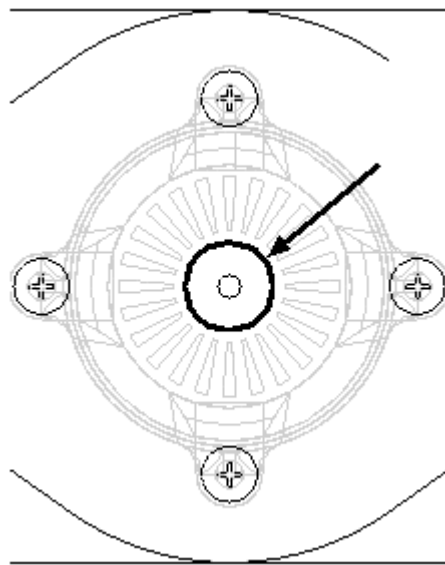
- ▶ Select the assembly reference plane as shown.



- ▶ On the Home tab, in the Draw group, choose the Include command. In the Include Options dialog box, select the options shown and click OK.



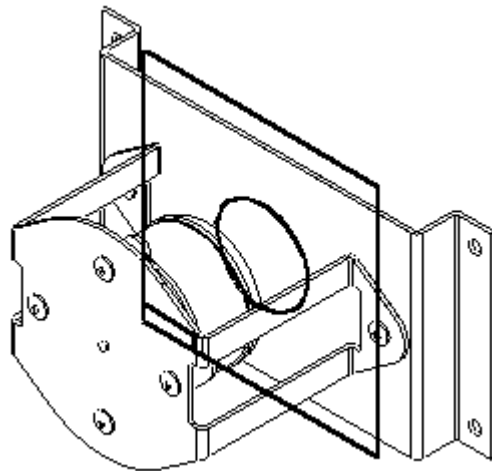
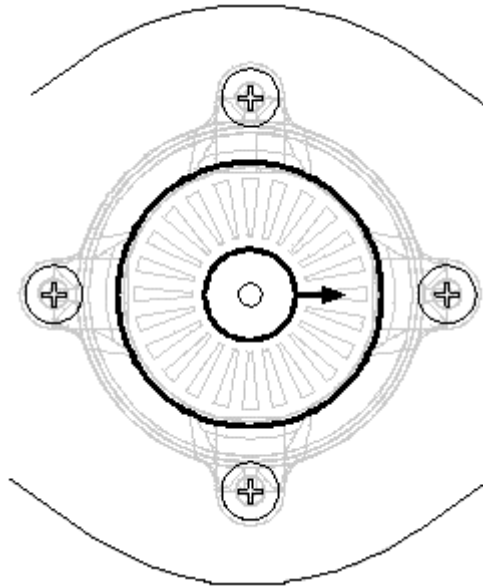
- ▶ Select the circle on the motor.



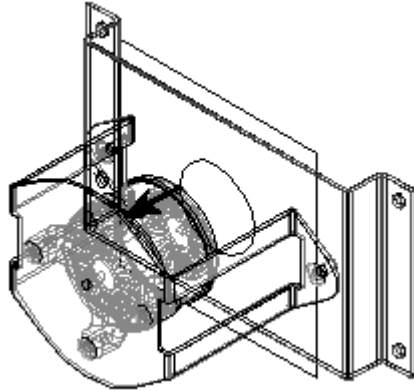
- ▶ Click the Accept button.



- Type 15 for the offset distance and then click to the outside direction of the selected circle. Choose Close Sketch.



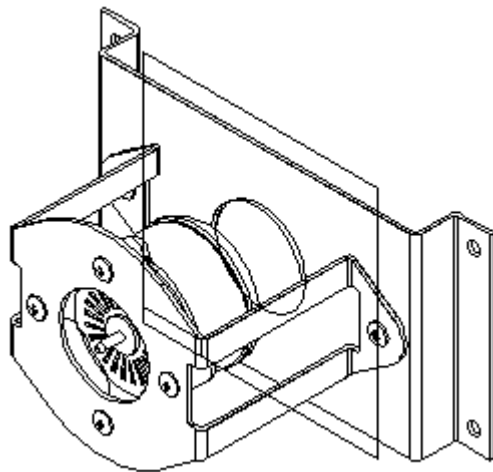
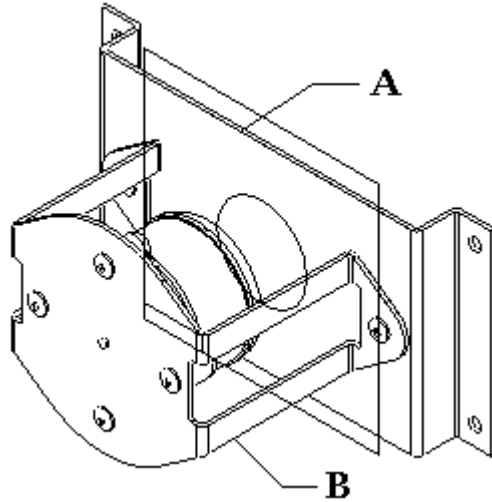
- ▶ On the command bar, click the Through All extent option. Position the cursor so the arrow points in the direction as shown, and click to accept.



- ▶ Click the Deselect button to deselect the parts highlighted in PathFinder. These are all the parts that the cutout affects.

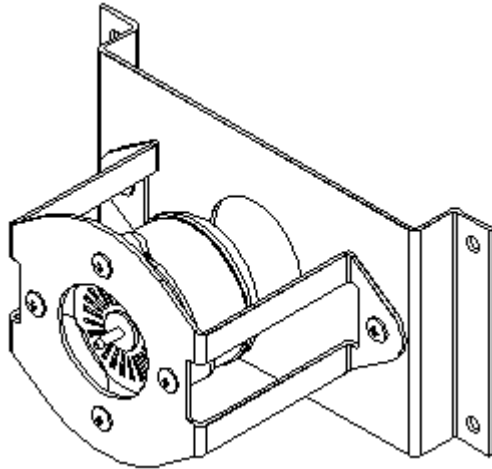


- In PathFinder, select the files *baseplate.psm* (A) and *motormount.psm* (B) for the parts to cut. Click the Accept button, and then click Finish. Both cutouts, driven by the circle, are placed in the assembly only.



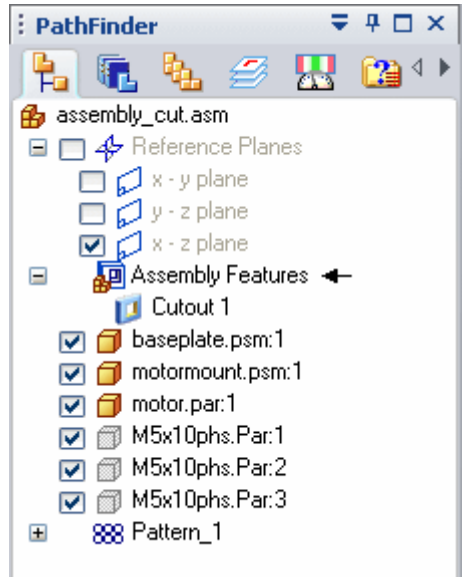
**Turn off the reference plane display**

- Click the Select tool. Right-click in the working window to display the shortcut menu, and then choose the Show/Hide All Components command. In the Show All/Hide All dialog box, check the Hide All box for Reference Planes.

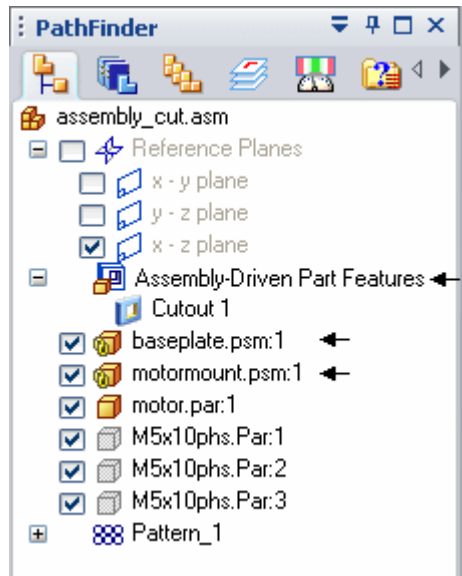


**Observe the changes in Pathfinder**

- ▶ Observe the changes in Pathfinder. Since the Create Assembly features option was selected, the cutout is shown as an Assembly Feature.



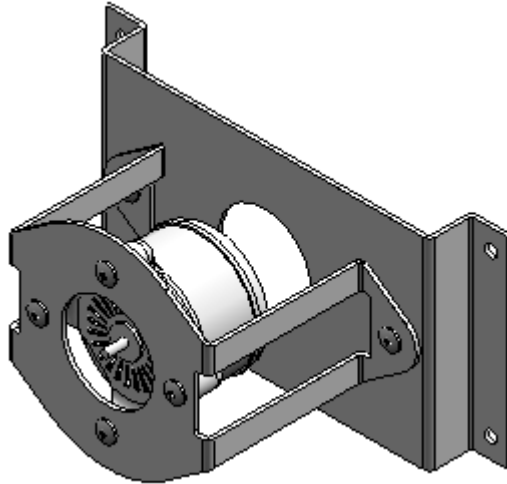
- ▶ If the Create Assembly-Driven Part features option was selected, then Pathfinder would show the following.



*baseplate.psm* and *motormount.psm* are now shown as associatively linked to the assembly sketch. The *baseplate.psm* and *motormount.psm* files are actually modified by the cutouts. Investigate this by saving and closing this file. Then open either *baseplate.psm* or *motormount.psm* to verify the modification.

**The activity is complete**

Save and close the assembly. This activity is complete.



### **Activity summary**

An Assembly Cutout can be applied in the assembly only (assembly feature) or the cutout can be applied directly to the part or sheet metal files (assembly-driven part feature). This activity only covered the assembly feature cutout option. You can go back later and use the assembly-driven part feature cutout option to observe how the part files have cutouts applied directly to them.

**Lesson review**

Answer the following questions:

1. What is the difference between an assembly feature and an assembly driven part feature?
2. Can an assembly feature, such as a cutout, operate on more than one assembly component?
3. How are assembly features displayed in a draft document?
4. Are assembly features valid when the assembly is placed as a subassembly?



## **Answers**

1. What is the difference between an assembly feature and an assembly driven part feature?

An assembly feature only exists within the assembly environment and does not add the feature to the part or sheet metal document. Other occurrences of those parts in other assemblies are not affected by the assembly feature.

An assembly driven part feature creates a linked feature in the part or sheet metal file and all occurrences of that part or sheet metal document, including other assemblies, are modified with that feature.

2. Can an assembly feature, such as a cutout, operate on more than one assembly component?

Components to which an assembly feature is applied to are selected prior to placing the feature.

3. How are assembly features displayed in a draft document?

When constructing drawings of an assembly that contain assembly features, you can specify whether the assembly features are displayed in a drawing view using the Drawing View Properties dialog box. This applies only to assembly features, not assembly-driven part features. Because assembly-driven part features physically modify the part document, they are always displayed in a drawing view.

4. Are assembly features valid when the assembly is placed as a subassembly?

Both assembly features and assembly-driven part features are viewable and can be used for positioning purposes when the assembly is placed as a subassembly in another assembly.

**Lesson summary**

An Assembly Cutout can be applied in the assembly only (assembly feature) or the cutout can be applied directly to the part or sheet metal files (assembly-driven part feature).

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Lesson

# 3 *Assembly patterning*

## **Assembly patterning**

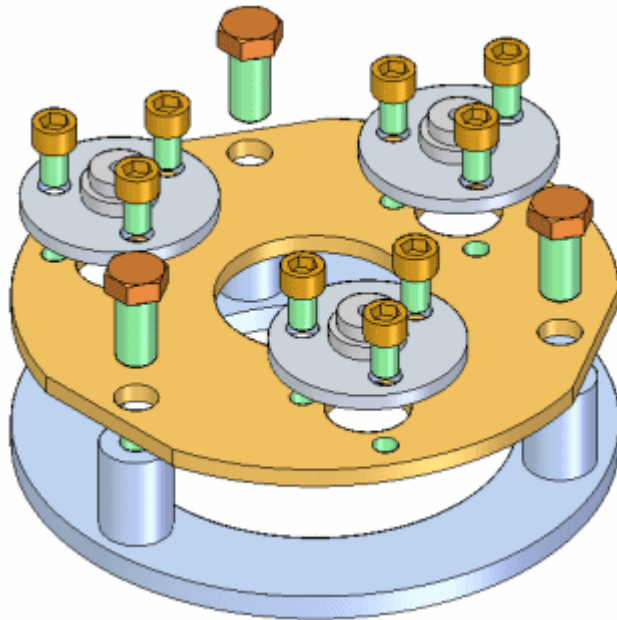
## **Patterning and mirroring components in assemblies**

When building assemblies, you often need to place parts and subassemblies multiple times in a pattern or mirror arrangement. For example, nuts, bolts, and other fasteners are placed in a rectangular or circular pattern on the parts they are fastening together.

### **Patterning parts**

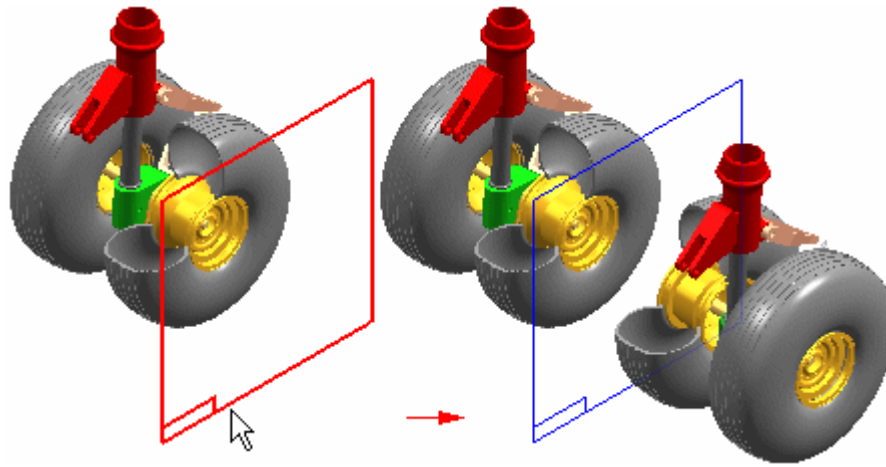
You can use the Pattern Parts command to quickly copy one or more parts and subassemblies into a pattern arrangement. You can also add an existing part pattern to a new part pattern.

The patterned parts are not positioned using assembly relationships, but according to a pattern feature on a part or assembly sketch.



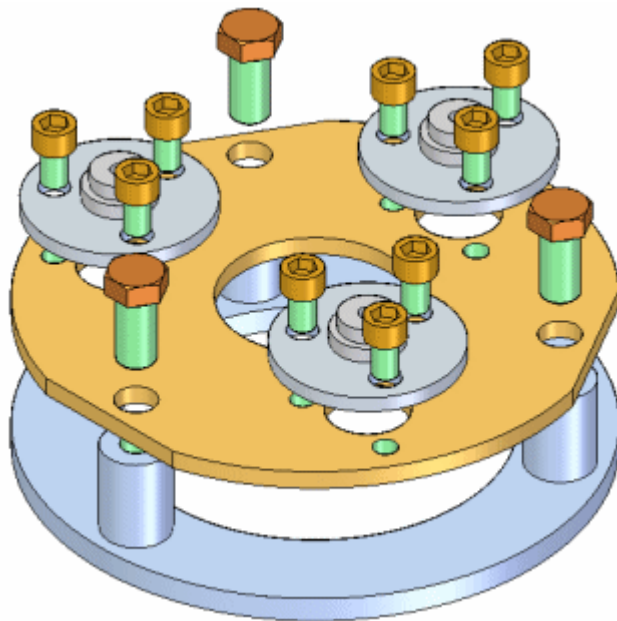
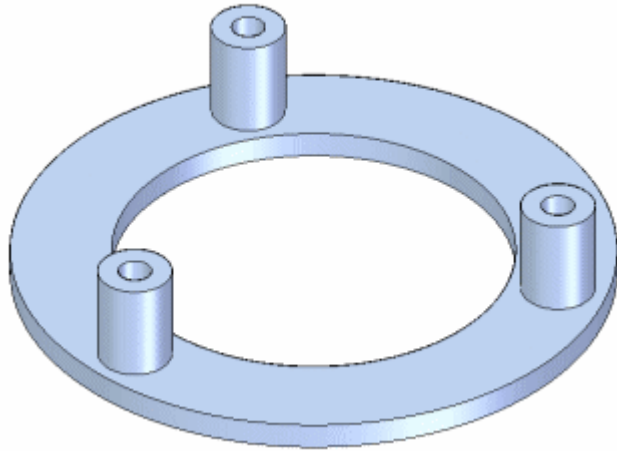
### **Mirroring parts**

You can use the Mirror Components command to quickly copy one or more parts and subassemblies into a mirror arrangement around a reference plane you select.



 **Pattern Parts command**

Copies one or more assembly components into a pattern.



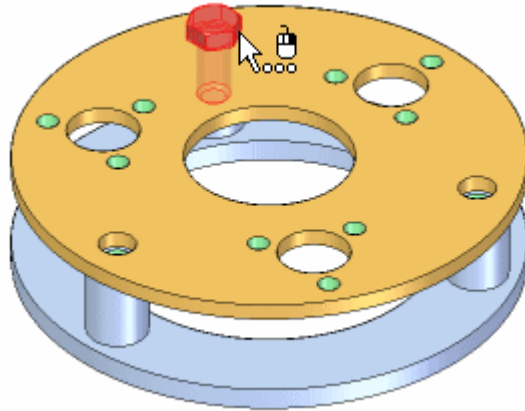
You can select the following types of components to define a pattern of parts:

- Parts in the active assembly.
- Subassemblies in the active assembly.
- Patterns of parts in the active assembly.

The parts are not positioned using assembly relationships. They are positioned using a pattern feature in a selected part or in an assembly sketch.

### Positioning the components to be patterned

Before creating a pattern of parts, you should properly position one copy of the components you want to pattern. For example, to pattern a bolt, use the Place Part command to place the bolt into one of the bolt holes in the part.

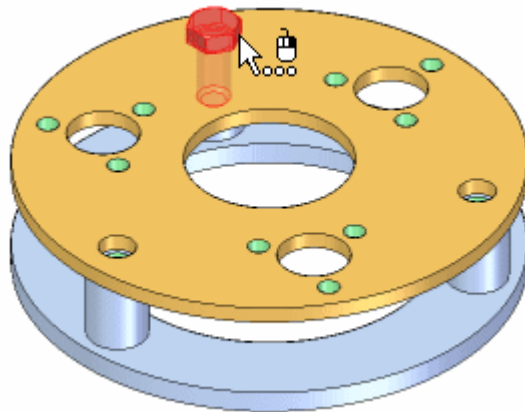


#### Note

If the pattern feature on the part was constructed using the Fast Pattern option, you should position the first bolt in the parent feature of the pattern. For example, if you used a hole feature to create the part pattern, position the first bolt in the hole feature, not one of the holes in the pattern feature.

### Selecting the components to be patterned

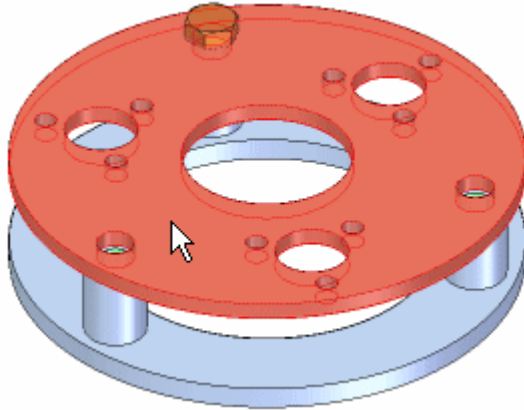
After you have positioned the components you want to pattern, use the Pattern Parts command to select them. The Select Parts step on the Pattern Parts command bar allows you to select the components you want to pattern. You can pattern multiple components in one operation. You can select parts in the active assembly, entire subassemblies, and existing patterns of parts.





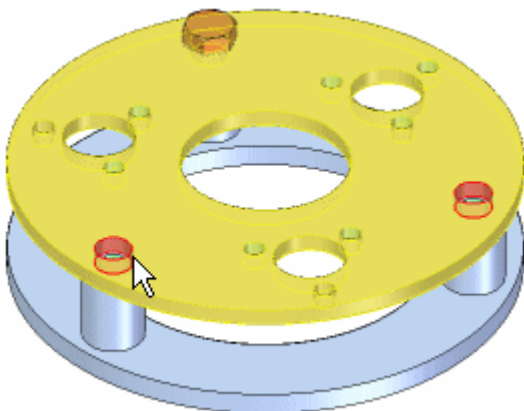
### **Defining the pattern**

After you have selected the components you want to pattern, the Define Pattern step on the command bar allows you to select the part that contains the pattern feature you want to use. You can also select an assembly sketch if the assembly sketch contains a 2D pattern profile.

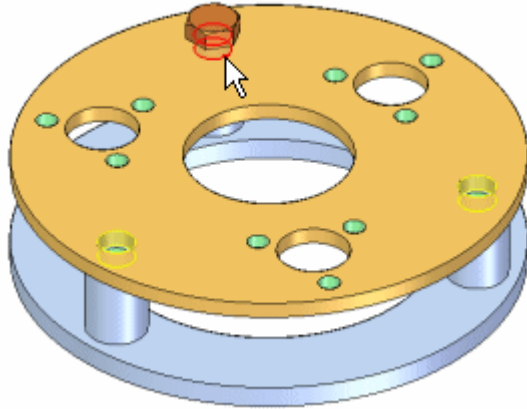


Next, you must select the pattern feature on the part or assembly sketch. The part you use as a pattern reference does not have to be the part in which you positioned the original components. You can select pattern features that were constructed with the following commands:

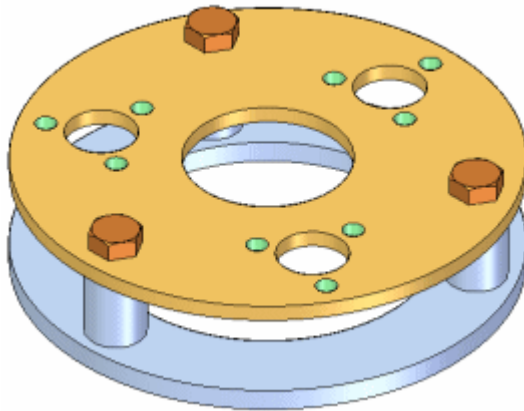
- Rectangular Pattern
- Circular Pattern
- Pattern Along Curve
- Hole



When the pattern feature highlights in the assembly window, select a reference position on the pattern feature. In most cases you should select the feature into which you placed the original copy of the part to be patterned.

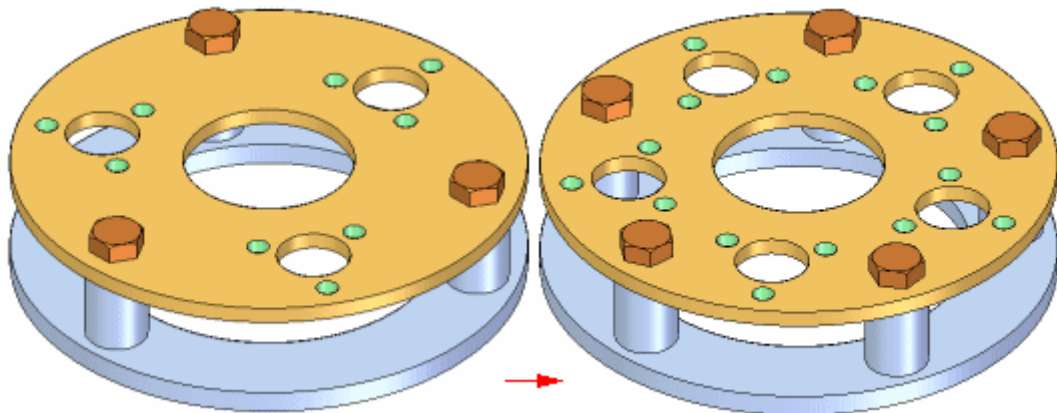


When you click the Finish button on the command bar, the original part is copied to each position in the pattern.



### Modifying the feature pattern

If you change the design of the part containing the feature pattern, the part pattern in the assembly updates. For example, if you increase the number of holes in the feature pattern, additional bolts are added to the part pattern in the assembly.



## **Deleting and suppressing patterned parts**

When you delete the original part in a pattern, all the patterned parts are also deleted. You cannot delete the individual Pattern Items in a pattern of parts, but you can suppress them. For example, you may have a pattern of 24 bolts where design considerations require one bolt to be shorter than the rest.

After you have placed the pattern of bolts, you can suppress the Pattern Item for that bolt in PathFinder. You can then use the Assemble command to place a shorter bolt in that location.

When you suppress a Pattern Item, the quantity value in an assembly report or a parts list in a drawing is adjusted accordingly. For example, if you suppress one Pattern Item in pattern of 24 bolts, the quantity value in the parts list would be 23 bolts.

## **Controlling occurrence properties of patterned parts**

You can control the occurrence properties of the individual parts in a pattern of parts. When creating a pattern of parts, the parent part's occurrence properties are applied to all of the patterned parts when you set the Patterned Parts Inherit Parent Part Occurrence Properties By Default option on the Assembly Tab on the Options dialog box.

After you create a pattern of parts, you can change the occurrence properties of individual parts in the pattern. For example, you may want to hide one of the patterned parts in the drawing, but still display it in the assembly. You can select the part in PathFinder or the graphics window, then use the Occurrence Properties command on the shortcut menu to set the occurrence property Display in Drawing Views to No for that part.

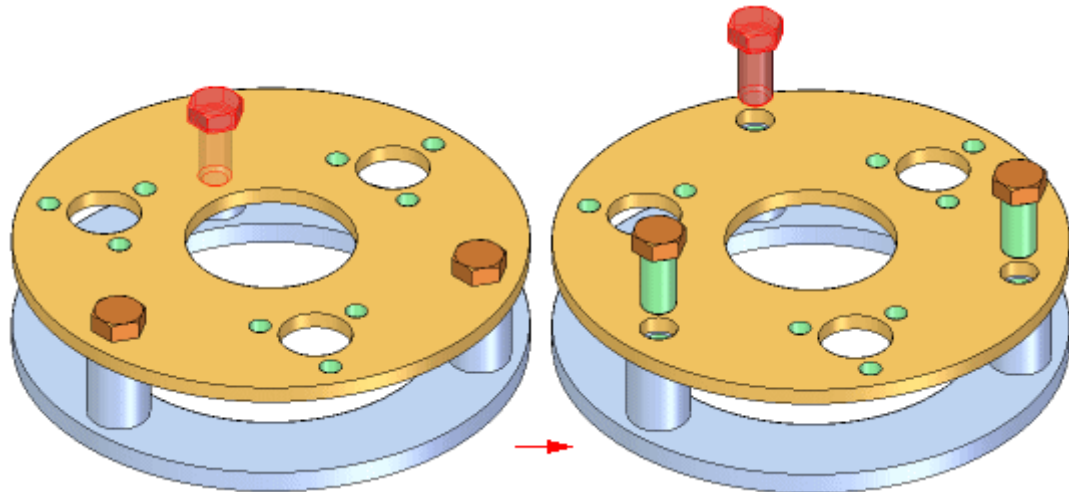
The part will be displayed in the assembly and counted in an assembly report, but will be hidden in the drawing and not counted in a parts list in the drawing.

## **Replacing patterned parts**

You can use the Replace Part command to replace the original part, but not the patterned parts. When you replace the original part, all the patterned parts are also replaced.

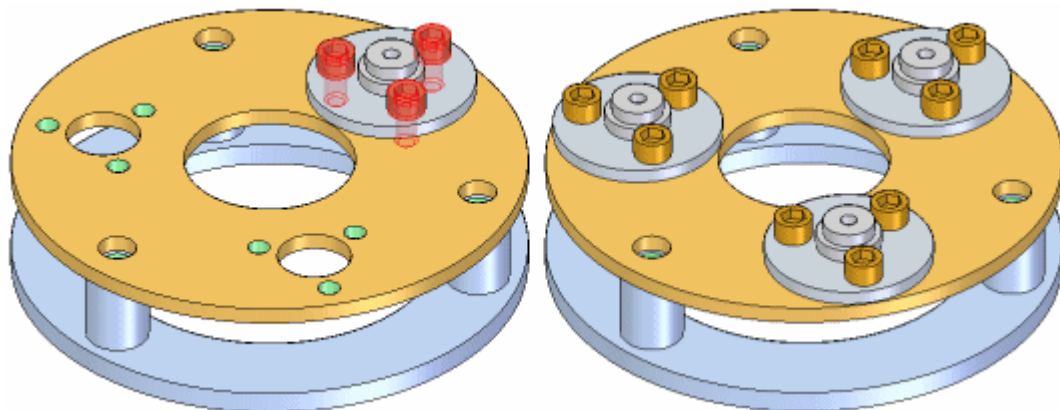
### Patterned parts and assembly relationships

Although the parts in a pattern are not positioned using assembly relationships, they do change position if a relationship controlling the original part is modified. For example, if you edit the offset value of the original part, all the patterned parts also update.



### Defining patterns productively

You can create patterns of parts that contain part patterns you defined earlier. For example, you can create a part pattern of socket head screws, then use that part pattern to construct another pattern.



### Assembly patterns and alternate assemblies

The Pattern Parts command is available when the Apply Edits to All Members option on the Alternate Assemblies tab is set (you are working globally) or cleared (you are working locally).

You can only modify the inputs for a pattern of parts when the Apply Edits to All Members option is set (you are working globally). For example, if an assembly pattern of parts originally included a bolt and nut, you cannot modify the pattern to add a washer to the pattern unless you are working globally.

For more information, see the Alternate Assemblies Impact on Solid Edge Functionality Help topic.

**Activity: Assembly patterning****Overview**

The objective of this activity is to show how to create a pattern of parts within an assembly.

**Activity**

In this activity you will use different options to assembly patterns.

## Activity: Generating a pattern in an assembly

### Overview

This activity demonstrates options to create patterns in an assembly.

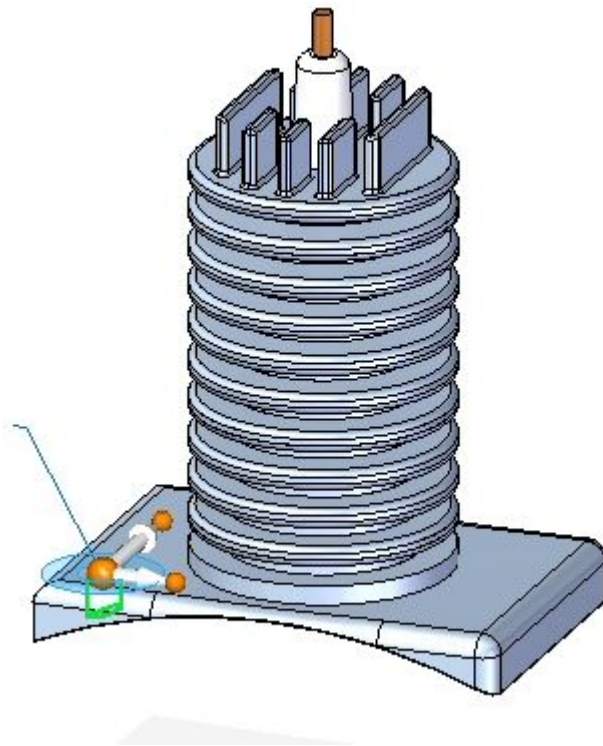
### Objectives

You will create different pattern types in an assembly.

#### *Pattern a hole feature in a part*

A pattern will be created in a part that will later be used to position fasteners in an assembly.

- ▶ Open the part *cylinder\_01.par* in the folder that contains the activities.
- ▶ In PathFinder, select the cutout shown.

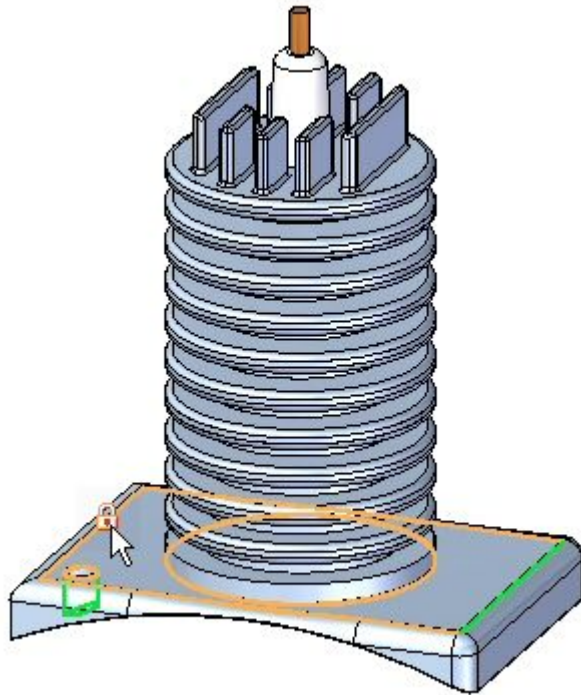


### Note

This feature is the reference feature of the pattern about to be created.

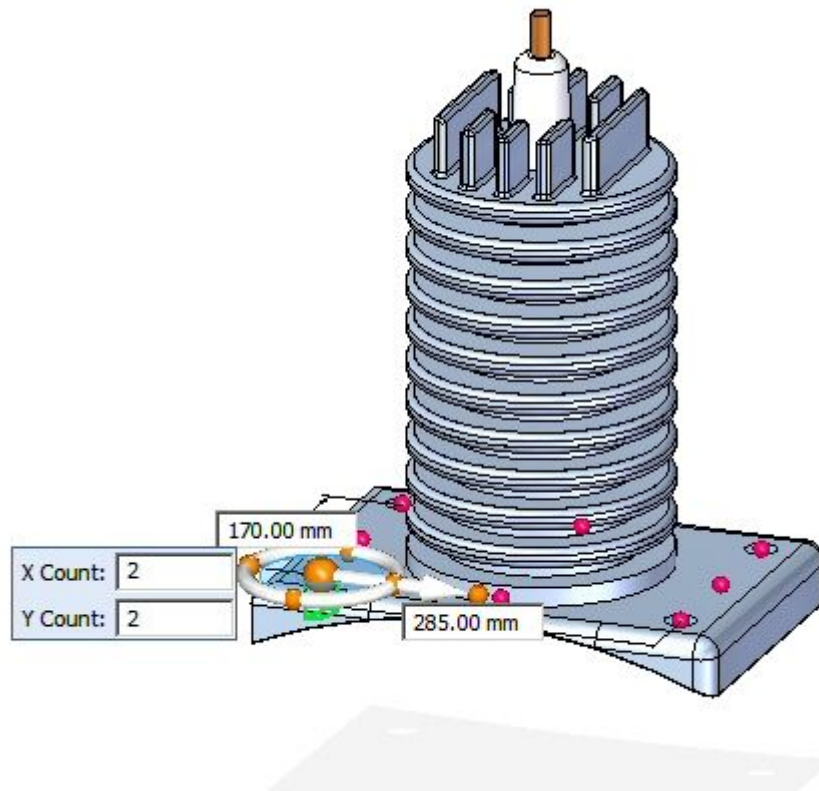
- ▶ On the home tab in the pattern group, click the rectangular pattern command.

- ▶ When prompted to select a face or reference plane, select the lock icon on the face shown.

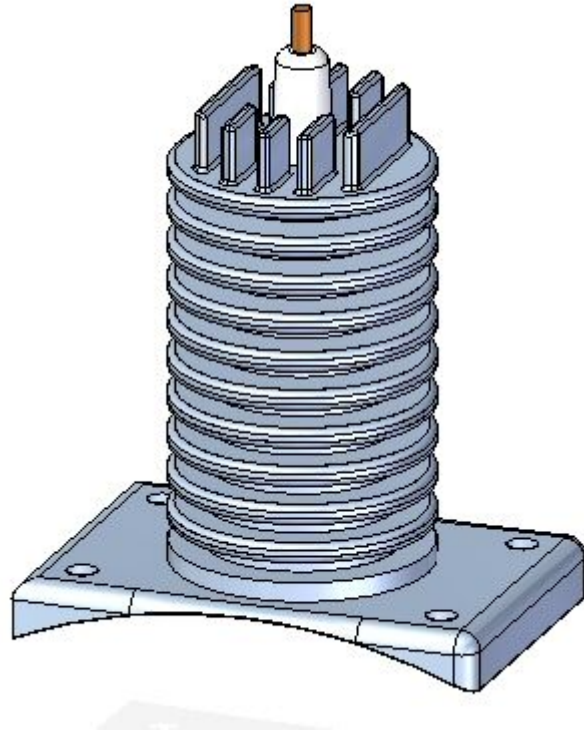




- Orient the steering wheel as shown and set the parameters to:
  - Pattern Type: Fit
  - X Count: 2
  - Y Count: 2
  - X Distance: 285.00
  - Y Distance: 170.00



- Accept the selection to complete the pattern.
- Save and close the *cylinder\_01.par*.

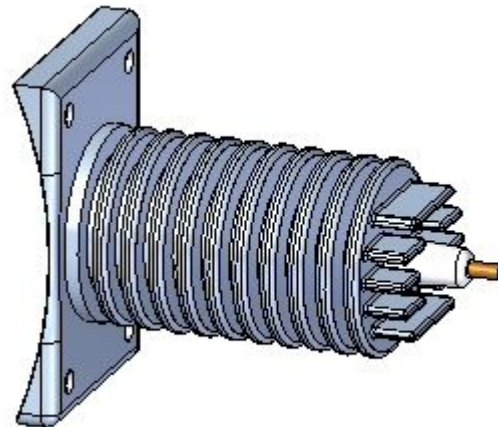
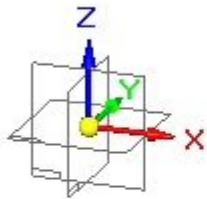
**Note**

It is important that the part be saved. The pattern created is used in the upcoming steps of the activity.

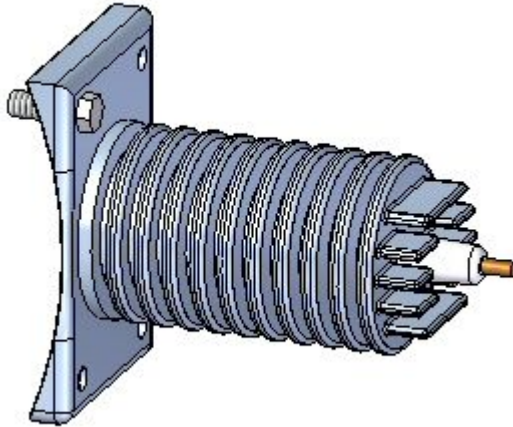
*Place a pattern of fasteners in an assembly*

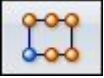
The fastener will be placed in using the reference feature of the pattern that resides in the part.

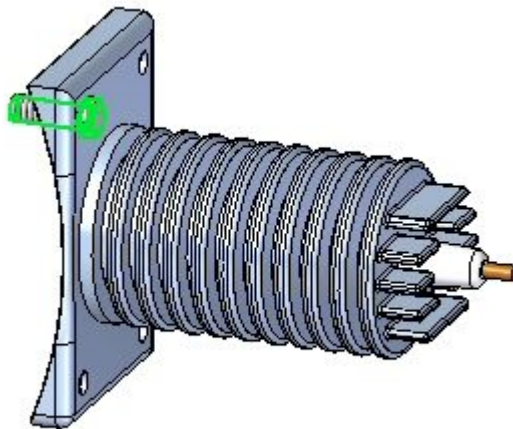
- Open the assembly *rotary\_engine.asm* with all the parts active.



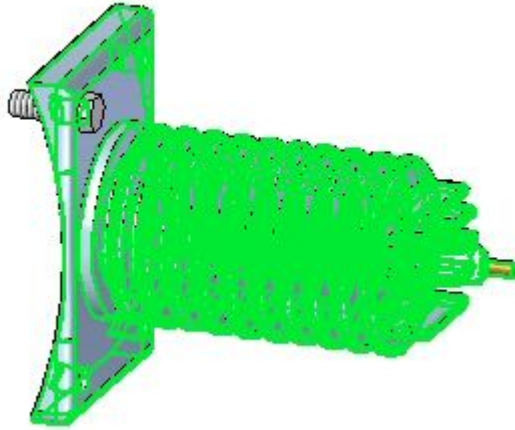
- ▶ Drag the fastener *25mm\_fastner.par* and place the fastener in the reference feature of the part pattern as shown.



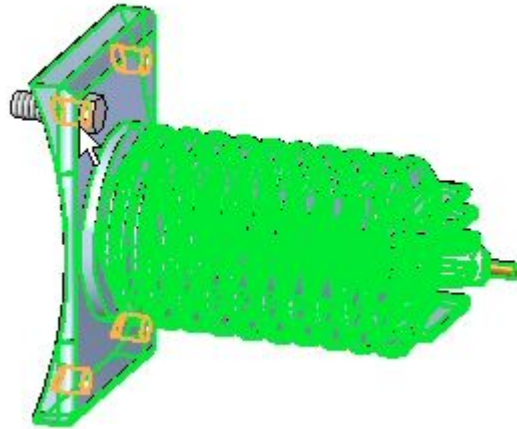
- ▶ On the Home tab in the Pattern group, click the Pattern command .
- ▶ When prompted to select the parts to be included in the pattern, select the fastener shown and accept.



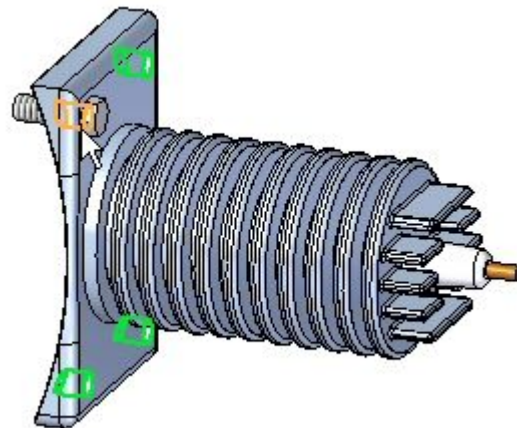
- ▶ When prompted to click on the part or sketch that contains the pattern, click the part *cylinder\_01.par* as shown.



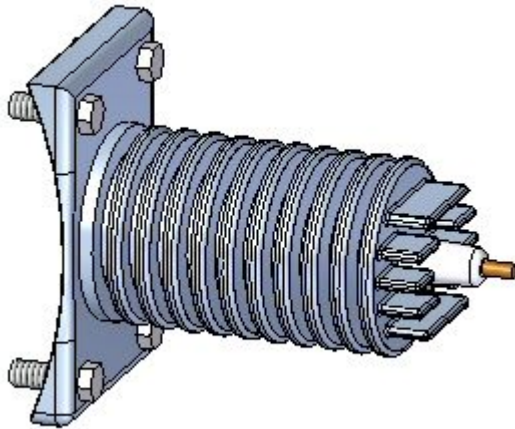
- ▶ When prompted to click on the pattern, select the whole pattern as shown.




- ▶ When prompted to click on the reference feature of the pattern, click the reference feature as shown.




- ▶ Click Finish. The pattern is placed.



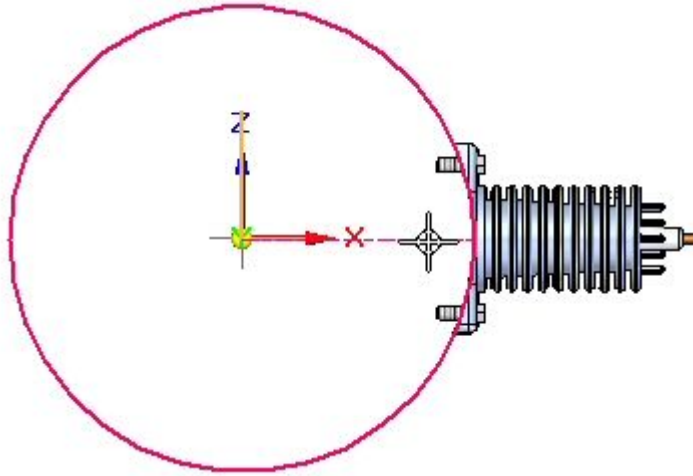
*Create an assembly pattern from an assembly layout sketch*

- ▶ On the Home tab in the Sketch group, click the Sketch command .
- ▶ Select the front reference plane to place the sketch.
- ▶ On the Home tab in the Features group, click the Circular Pattern command

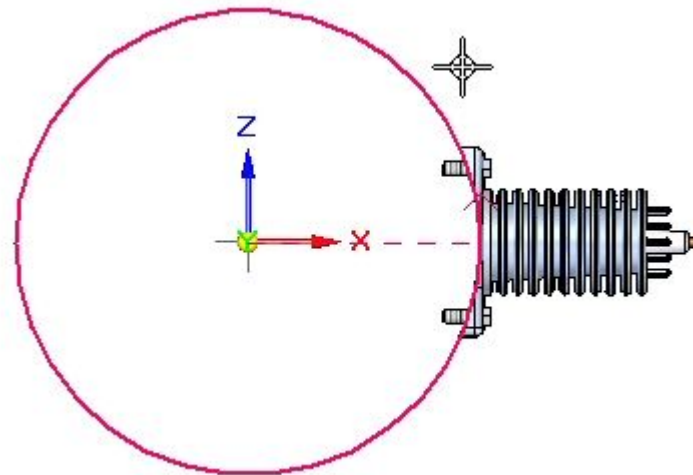


- ▶ Ensure that the full circle option  is set.
- ▶ Set the count to 4.

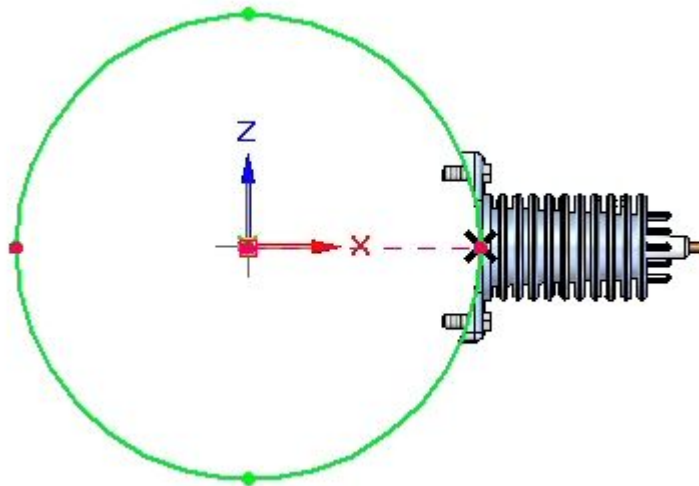
- ▶ Place a 450 mm radius circle centered on the origin of the reference planes oriented as shown.



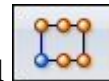
- ▶ When prompted to click on the arc direction, click above the part as shown.

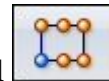


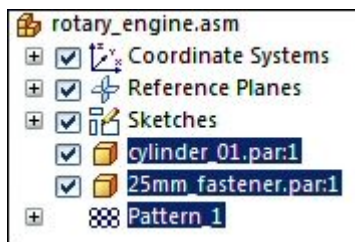
- ▶ The pattern circle is created.



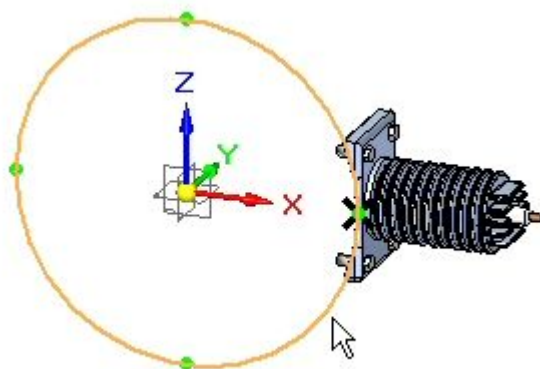
- ▶ Click Close Sketch , and then click Finish.



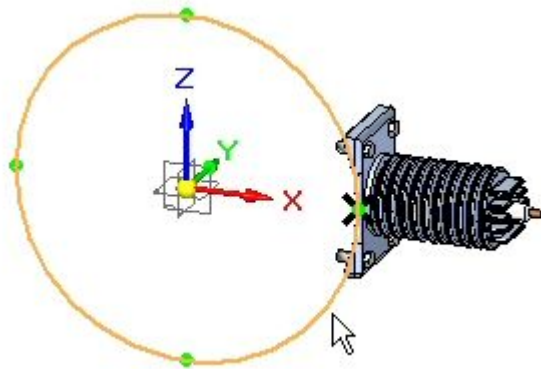
- ▶ On the Home tab in the Pattern group, click the Pattern command .
- ▶ When prompted to select the parts in the pattern, click on the parts in PathFinder as shown and accept.



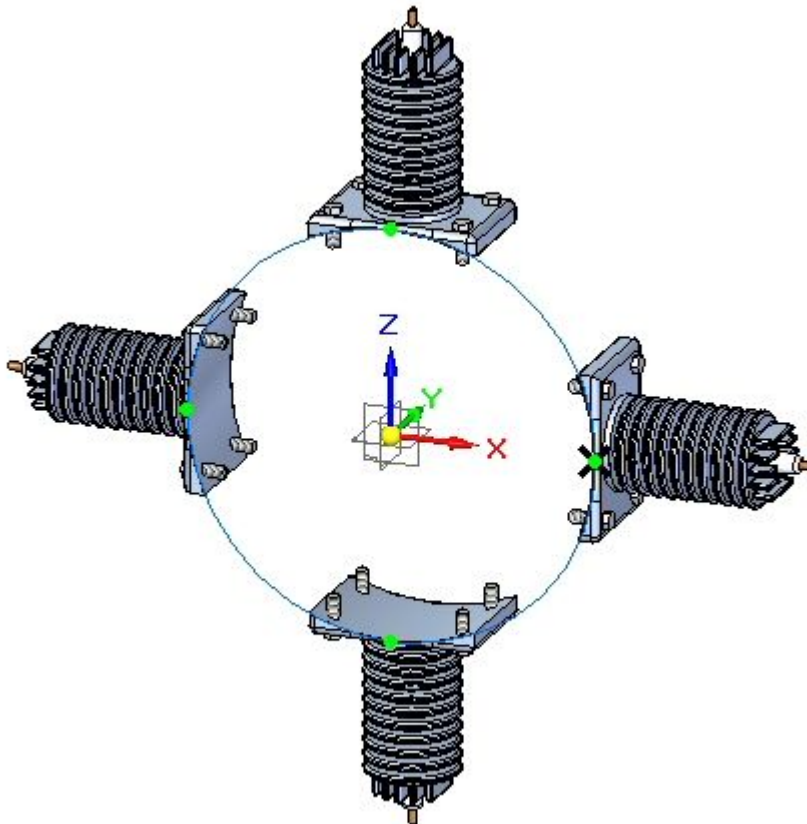
- ▶ When prompted to select the part or sketch that contains the pattern, select the sketch as shown.



- ▶ When prompted to click on the pattern, click on the circle as shown.



- ▶ Click Finish to create the pattern.



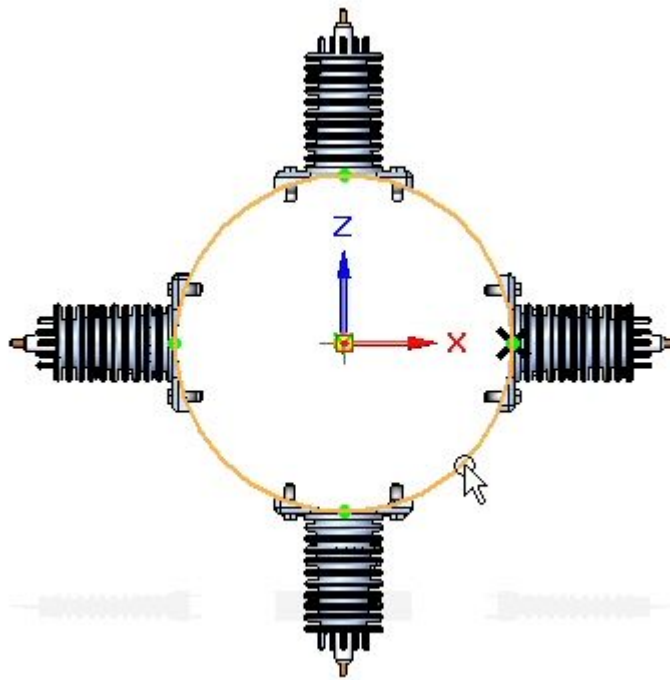
*Modify the pattern*

- ▶ Select the sketch and then click Edit Profile

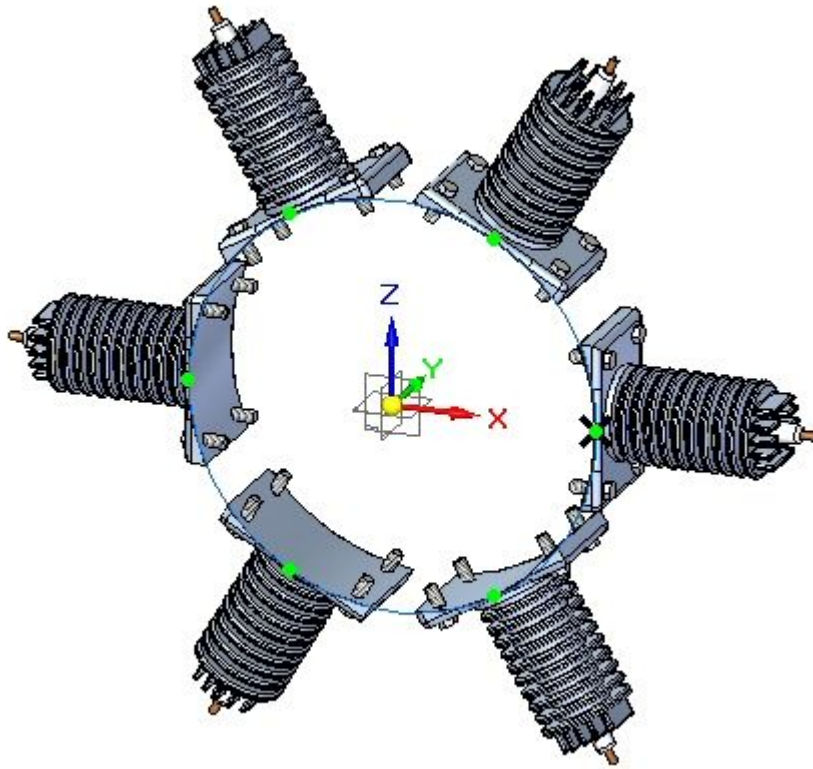




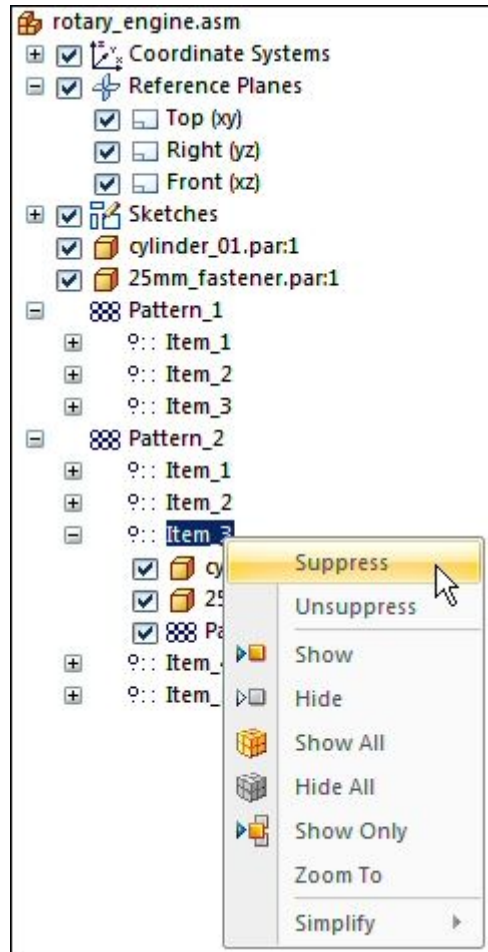
- Select the pattern circle as shown.



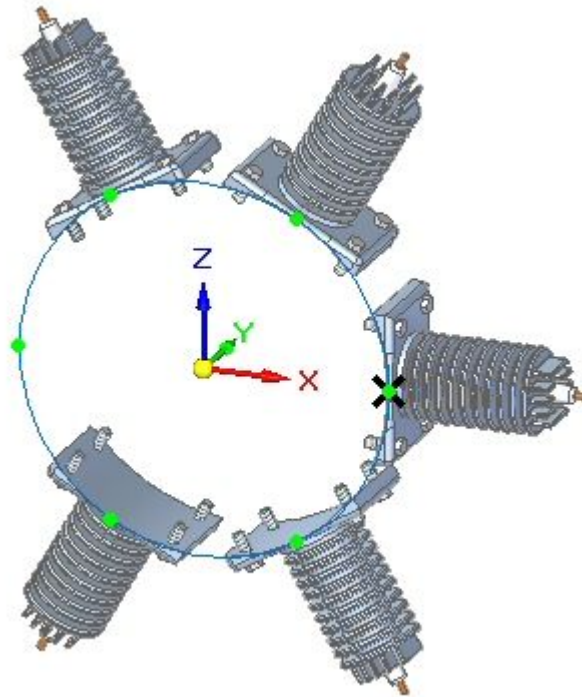
- Set the count to 6. Close the sketch and then click Finish.
- Select the pattern in PathFinder. Click Edit Definition, and then click Finish to recalculate the pattern.

*Suppress a pattern occurrence*

- In pathfinder, right click item\_3 in the pattern and then click suppress.



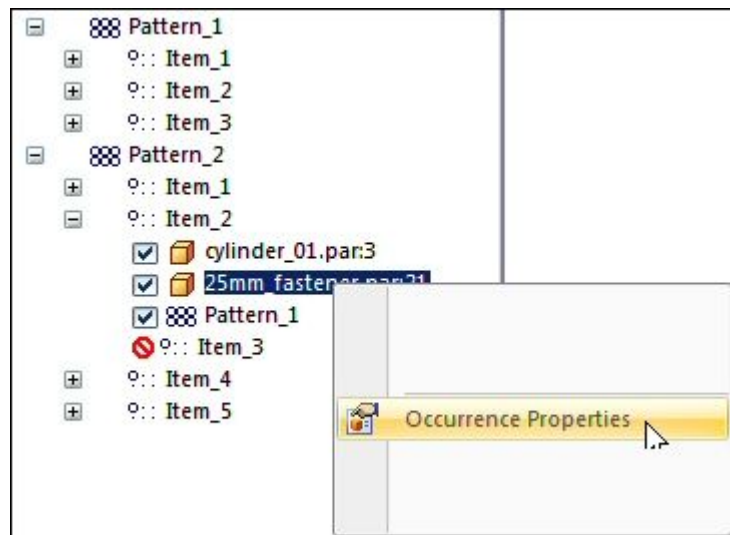
- Item\_3 is suppressed as shown.



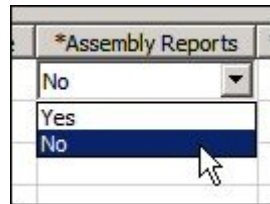
The occurrence is suppressed.

*Excluding a pattern occurrence from reports*

- In pathfinder, right click 25mm\_fastener.par in item\_2 in the pattern and then click occurrence properties.



- Set No in the assembly reports column.



The item will be removed from assembly reports and draft parts list.

### *Summary*

In this activity you learned some of the options available for generating assembly patterns.

**Lesson review**

Answer the following questions:

1. Name two entities that control the placement of a pattern of components in an assembly.
2. How do you suppress one of the occurrences of an assembly pattern?
3. How can a component that is part of an assembly pattern be excluded from assembly and draft reports?

## **Answers**

1. Name two entities that control the placement of a pattern of components in an assembly.

An pattern of assembly components can be controlled by either an assembly sketch, or by a pattern that exists within one of the components placed in the assembly.

2. How do you suppress one of the occurrences of an assembly pattern?

Right click on the item occurrence in pathfinder, then click suppress.

3. How can a component that is part of an assembly pattern be excluded from assembly and draft reports?

Right click on the occurrence in pathfinder and then click the occurrence properties.

**Lesson summary**

In this lesson you learned some of the options available for generating assembly patterns.



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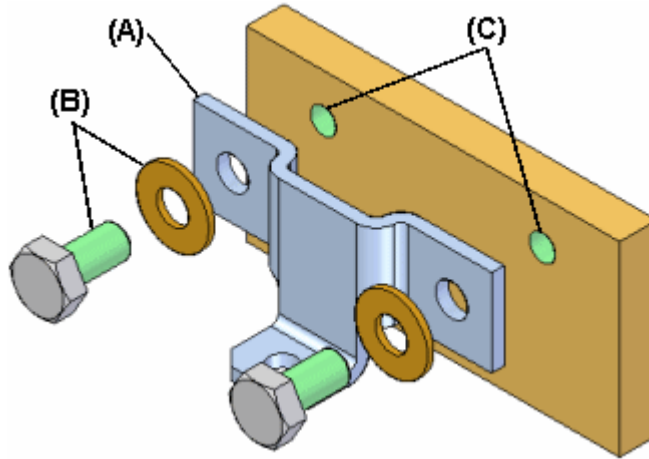
**Lesson**

# 4 *Assembly systems libraries*

## Working with Systems Libraries in assemblies

When building assemblies, you often need to use the same group of parts more than once. This group of parts may also need the same feature modification made to a mating part in the assembly to facilitate placement of the group of parts.

For example, you may need to place a bracket (A) and its mounting bolts and washers (B) in many different assemblies, or many times in one assembly. The bolts may also need mounting holes in the part (C) to which they are fastened.



You can use the systems library functionality in Solid Edge to define the group of parts, features, and assembly relationships so you can reuse them easily later.

The parts, features, and relationships for a systems library member are stored in an assembly document you define.

A systems library document can consist of the following:

- Complete subassembly documents
- Part documents
- Features that are associative children of one of the part documents

### Creating Systems Library members

When creating a new systems library member, you must adhere to the following rules:

- You can only select parts and entire subassemblies that are in the top level assembly.
- You cannot create a systems library member when working in an alternate assembly. You can use the Save Member As command to save a copy of the alternate assembly as a normal assembly to use this functionality.
- You cannot create a systems library member when you are in-place activated in a subassembly.

## **Creating a new Systems Library member**

You create a new systems library member using the Create Systems Library command on the Home tab in the Systems group. When you click the Create Systems Library button, a command bar guides you through the steps required to create the systems library member:

- Selecting the assembly components you want
- Selecting the features you want to capture (optional)
- Viewing the captured relationships
- Defining the new assembly document name, template, and location

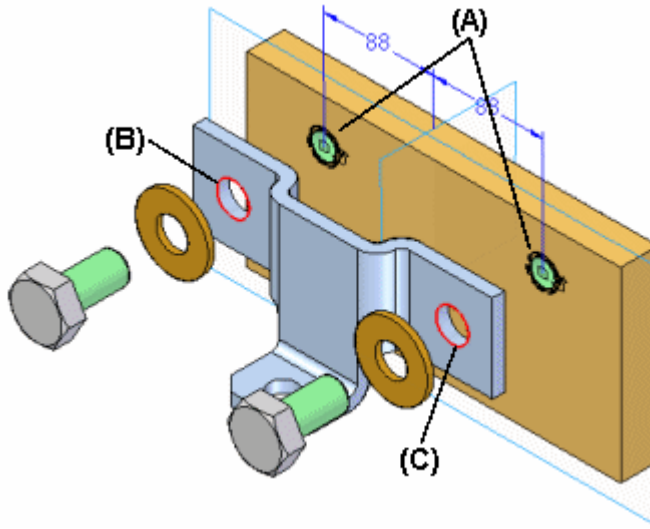
## **Selecting components**

Any parts and subassemblies that are in the active assembly can be added to a systems library document. When adding a subassembly, the entire subassembly must be added. You cannot select individual parts in the subassembly. You can select the parts you want to capture in the assembly window or in PathFinder. If you select an assembly pattern, the parts that were used to define the pattern are automatically captured.

The parts and subassemblies you select are listed in the Selected Components dialog box. When you have selected all the parts and subassemblies you want in the new systems library member, click the Accept button on the command bar to proceed to Captures Features Step. The relationships used to position the parts and subassemblies are automatically captured.

## Capturing features

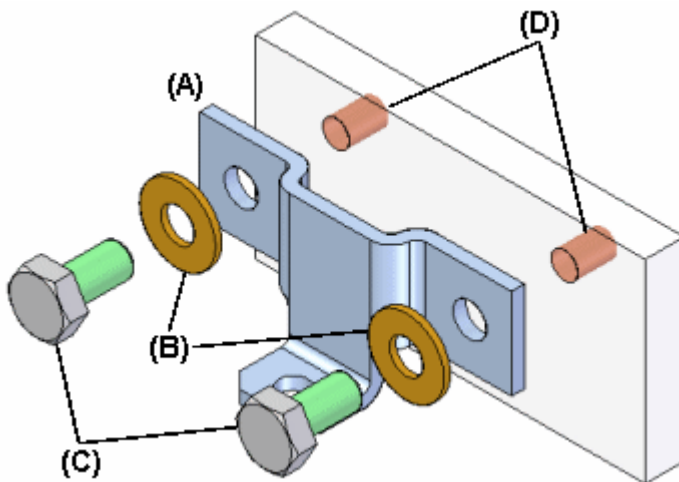
You can only capture features that were created using inter-part associativity techniques. If one of the parts you selected in the Select Components Step was used as a parent to create an associative feature on another part, the associative child feature on that other part is available for use in the systems library member. For example, if the hole feature (A) in the mounting plate was constructed using the Include command to associatively include edges (B) (C) from the bracket part, the hole feature (A) can be part of the systems library document.



### Note

For more information on inter-part associativity, see the [Inter-Part Associativity](#) Help topic.

In the previous example, the system library member will then consist of the bracket part (A), the bolts and washers (B) (C), and the hole feature (D). When you place the systems library document later, the plate part itself is not placed.



When you select parts that have been used as parents in an associative inter-part feature, the child features that are available for including in the new systems library

document are listed in the Captured Features dialog box and are highlighted in the assembly window so you can select only the features you want to capture.

You can also capture any treatment features that modified the associative inter-part feature. For example, if you added chamfers to the holes described earlier, you can add them to the captured features list by selecting the chamfer feature in the assembly window.

After you have selected the features you want to capture, click the Accept button on the command bar. If you do not want to capture features for the new systems library document, you can click the Next button on the command bar to bypass this step.

### **Capturing relationships**

When you finish defining the parts and features you want to capture, the Capture Relationships dialog box is displayed so you can review the parts, features, and positioning relationships you captured. You can click the relationships listed on the Capture Relationships dialog box to highlight the faces used to define the relationship you selected, or you can click the OK button on the dialog box to dismiss it.

### **Defining the Systems Library container document**

When you click the Create Group button on the command bar, the Create Systems Library dialog box is displayed so you can define the systems library document name and location, the assembly template you want to use, and rename any features you captured.

If you are going to share systems library documents with other users at your company, you should define file naming and location conventions to make it easier to use systems library documents.

### **Placing a Systems Library member document**

You place a systems library member document by dragging the assembly document for the systems library member from the Parts Library page in PathFinder, and dropping it into the assembly window. When you drop the systems document in the assembly window, a command bar and the Input dialog box are displayed to guide you through the steps required to place the systems document:

- Positioning the parts
- Positioning the features

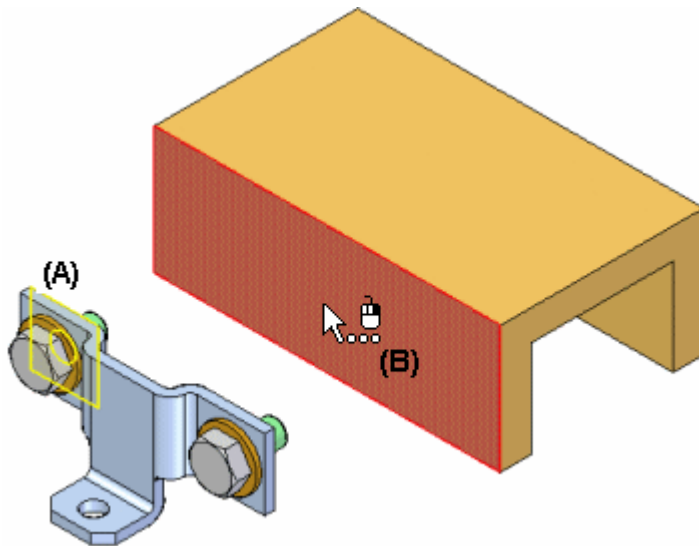
The parts that comprise the systems library document are also temporarily positioned in the assembly window.

The Input dialog box lists the assembly relationships required to position the systems library components. When placing the component parts and captured features, options are provided on the command bar to skip captured relationships or features. As you complete the steps required to define a relationship, the Input dialog box is updated to indicate the steps you have completed.

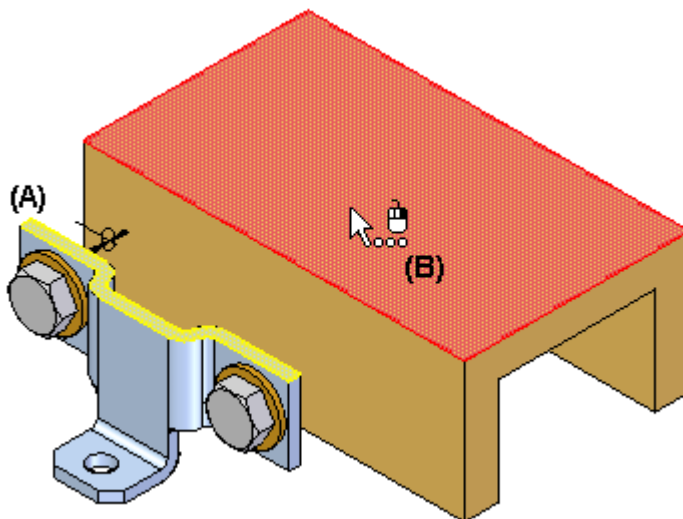
## Positioning parts

You position the parts similarly to the way you would normally position a part in an assembly, by selecting a part already in the assembly, and a face or element on that part to define the relationship. If the Use Reduced Steps When Positioning Parts option is set, then you will only need to select a face or element on an assembly part to define the relationship.

When you drop the systems library member into the assembly, the face on the placement part for the first positioning relationship is highlighted (A). You can then select the target face in the assembly to satisfy the first relationship (B). The Input dialog box displays the relationship type and default value for each relationship. You can also use the command bar to define a new offset value for the first relationship, or skip defining the relationship entirely.



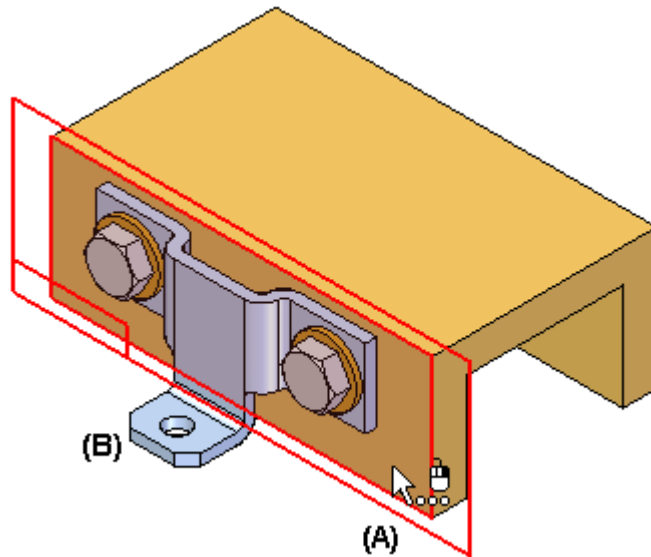
After you select the target face for the first relationship, the set of parts is repositioned, and the face on the placement part for the second positioning relationship is highlighted (A). You can then select the target face in the assembly to satisfy the second relationship (B).



You continue in this fashion until all the positioning relationships have been defined or skipped.

### Positioning features

After you define the assembly relationships, you specify the part that you want to modify with the captured features. For example, if the feature you are placing is a hole feature, you would select a planar face on the assembly part (A) on which you want to apply the hole feature. The hole feature is associatively linked as a child of the parent part used to define the feature. In this example, the parent part is the bracket part (B).

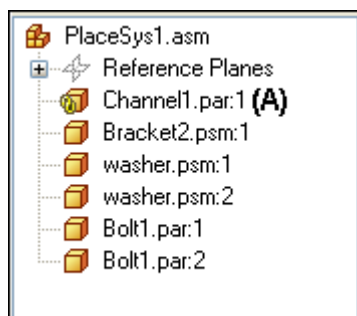


After placement, the parts behave exactly the same as any part in an assembly. The systems library document itself is not placed into the assembly. It is used as a container that defines the systems components.

### Reviewing inter-part relationships

The inter-part relationships that you use when creating a systems library member, and when you placing a systems library member are indicated in PathFinder, and can be further reviewed using the Inter-Part Manager dialog box.

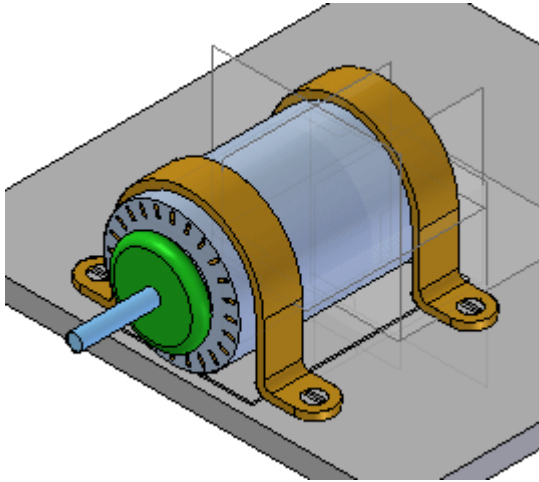
For example, a symbol is added to the child part (A) in PathFinder to indicate that it is associatively linked to another part in the assembly.



## Activity – Systems library

The objective of this activity is to teach the fundamentals of creating a systems library and then placing the library into an assembly so that the parts and features associated with that library are correctly positioned.

When you complete this activity, you will know how to use systems libraries to quickly create components and features, which can be used in many other assemblies. You will learn procedures used during systems library creation that make placement of system libraries easy and consistent.





## **Activity – Systems library**

In this activity you will create an assembly called a systems library that consists of parts and the underlying features needed to modify target geometry as the assembly is placed.

### Place the motor on the plate

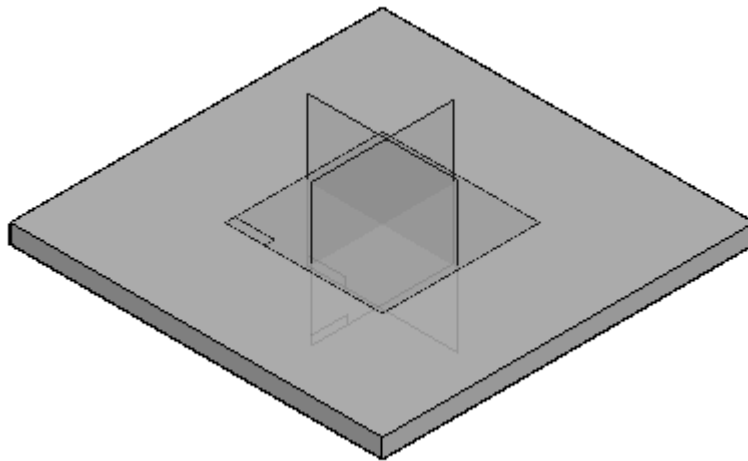
The motor is first placed on the plate. Parts and features that need to accompany the motor in a systems library are created on this plate.

The systems library consists of a group of parts and features that are meant to be placed many times. First, you assemble the components of the systems library in the manner you later wish them to be placed. Once the components are correctly positioned, then you define them as a systems library.

#### Note

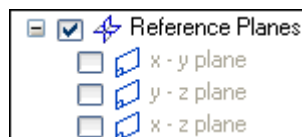
In creating a systems library, it is important to establish as many relationships as possible between components that are going to be a part of the systems library. Relationships to other parts need to be reestablished upon placement, so whenever possible establish relationships with components within the group to speed placement. Establish as many relationships as you can to the first part that is placed.

- ▶ Open *Create\_Sys\_Lib.asm* with all the parts activated.



- ▶ The right (y-z) plane of the part *base\_plate.par* will be used to position the motor on the plate. The assembly reference planes can be turned off.

Hide the assembly reference planes.

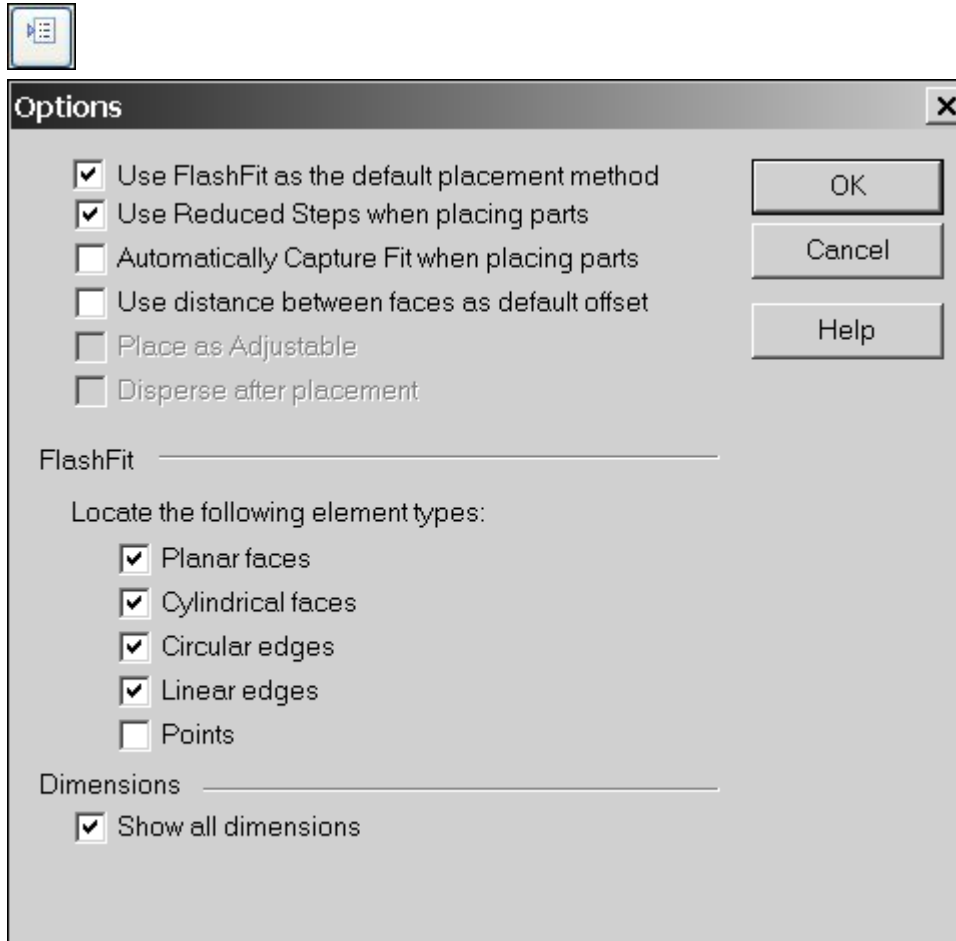


- ▶ Show the reference planes for the part file *base\_plate.par*. Later, these will be used to position the motor. In PathFinder, right-click *base\_plate.par* and then click Show/Hide Component and select the On check box for Reference Planes.

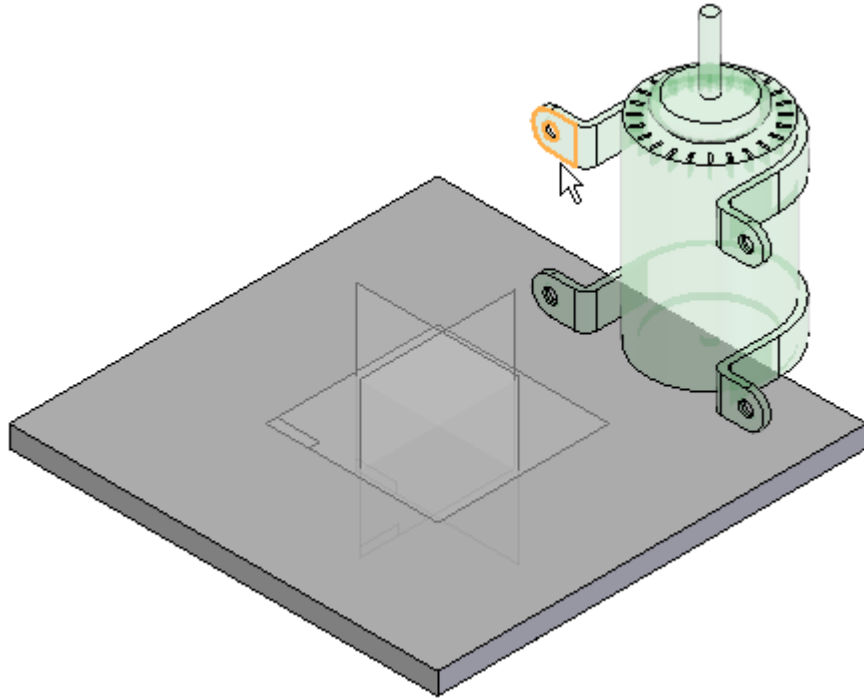
- ▶ From the Parts Library, drag *motor.par* into the assembly. Position the part using the following steps.
- ▶ Click Options on the command bar and set the options as shown.

**Note**

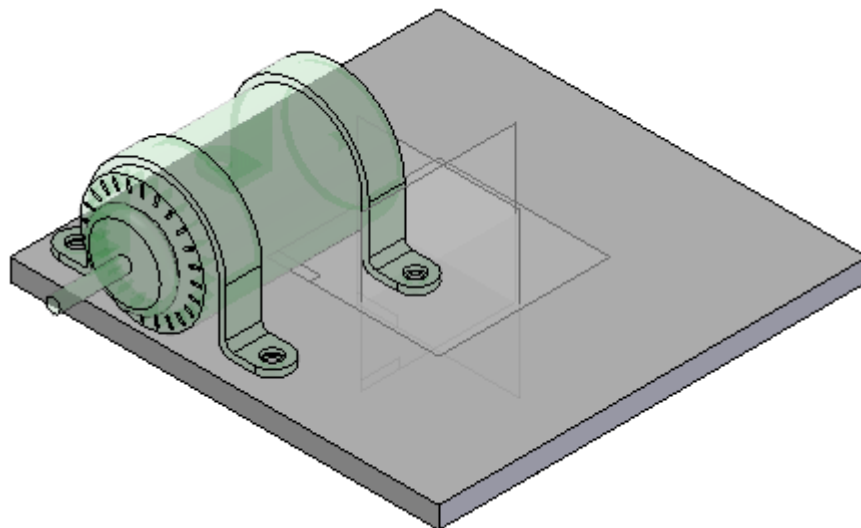
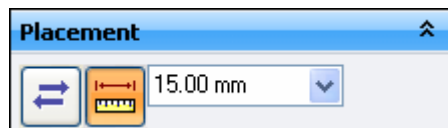
Even though the FlashFit option is set, choose the appropriate relationship and leave the default as FlashFit.



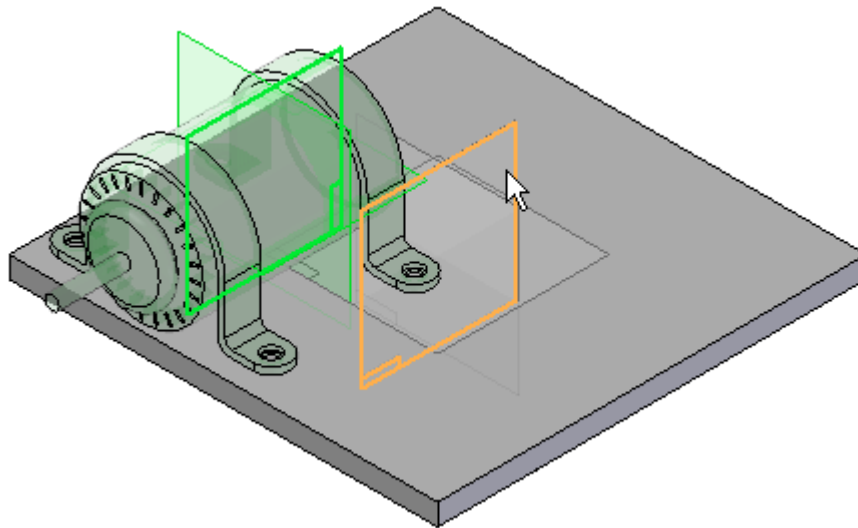
- ▶ On the command bar, select the Mate relationship, and mate the plane of the foot of *motor.par* to the top plane of *base\_plate.par*.



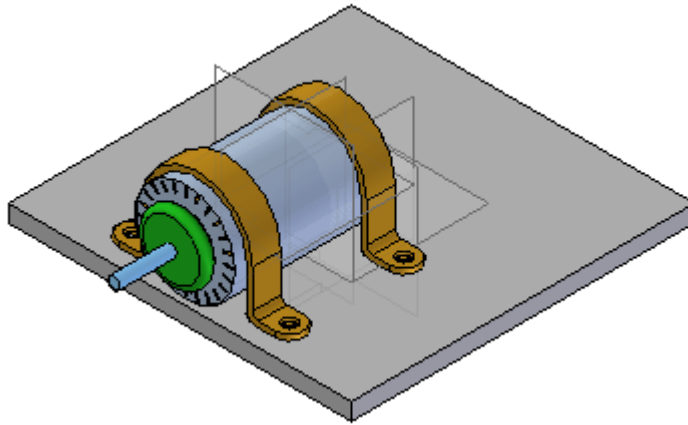
- ▶ Select the Planar Align relationship, and align the front face of the motor bracket on *motor.par* with the vertical face of *base\_plate.par*. Set the fixed offset to 15 mm.



- ▶ Use the Planar Align relationship to create a relationship between the reference plane of *motor.par* to the reference plane of *base\_plate.par*. Access the reference planes for the motor by turning on the construction display for reference planes as shown.



The motor is now centered on the base plate.



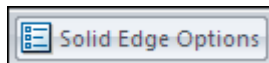
**Use Inter-Part Copy to associate the holes in the motor to the holes in the plate**

Next, add holes to the base plate. Use the Inter-Part Copy command to associate the holes with the corresponding geometry on the motor.

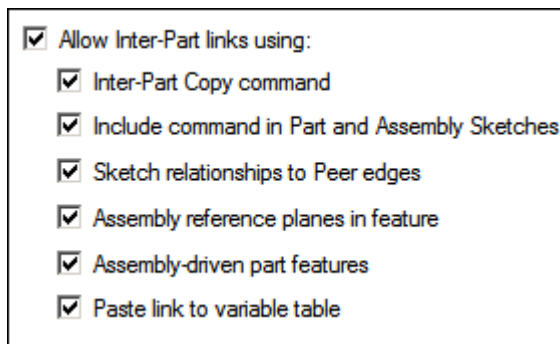
- ▶ Click the Application button.



- ▶ Click Solid Edge Options.



- ▶ Click the Inter-Part tab and set the options as shown. Then click OK.



- ▶ In Assembly PathFinder, right-click *base\_plate.par*, and then click Edit.

**Note**

When editing a part in the context of an assembly, you can turn off the view of the rest of the assembly. The command that does this is View tab® Show group® Hide Previous Level. For the upcoming steps, however, do not hide the assembly.

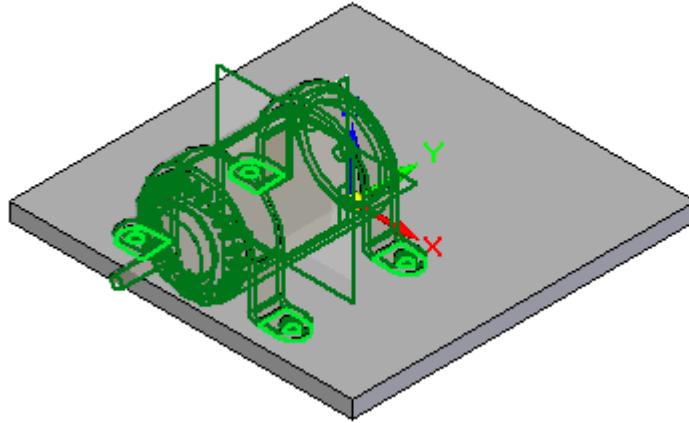
- ▶ If you cannot see the motor, select View tab® Show group® Hide Previous Level.



- ▶ Choose the Home tab® Clipboard group® Part Copy® Inter-Part Copy command



- Select *motor.par* as the assembly part to copy from. Select the bottom plane of each foot as the faces to copy.



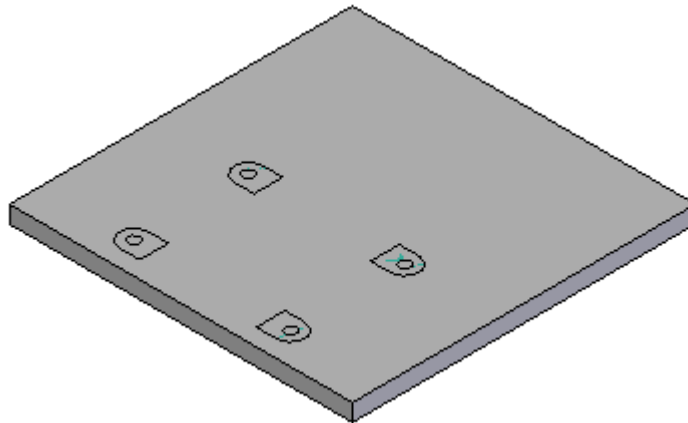
- Click the Accept button, and then click Finish to place the inter-part copy.



### Create holes in the plate from the Inter-Part copy

Place features on the plate. These features become part of the systems library and will be placed as features in the target part at the time of placement.

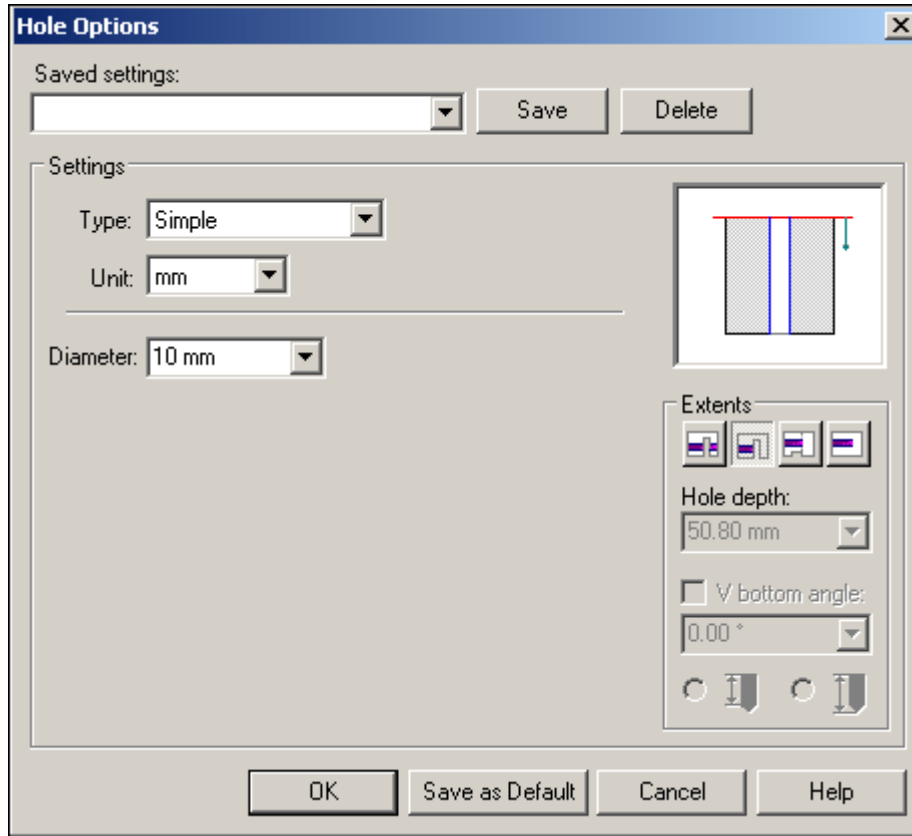
- ▶ Choose View tab@ Show group@ Hide Previous Level to turn off the display of *motor.par*.



- ▶ Choose Home tab@ Solids group@ Hole command



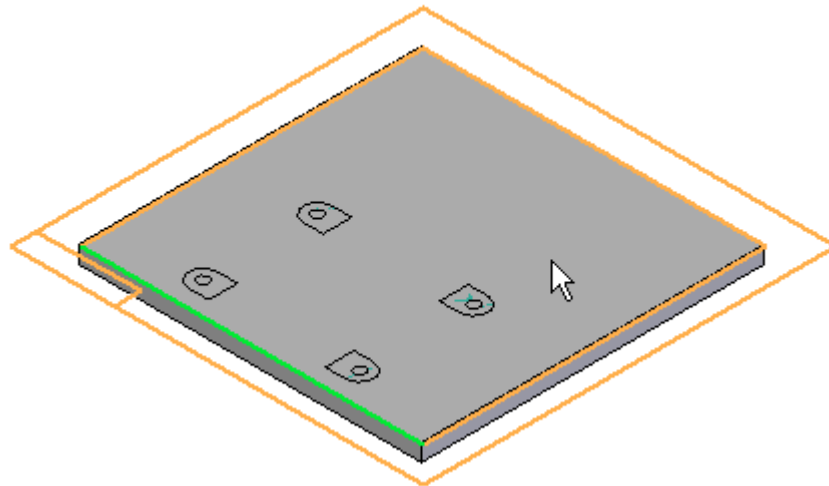
- ▶ Click Hole Options and set the values as shown, and then click OK.



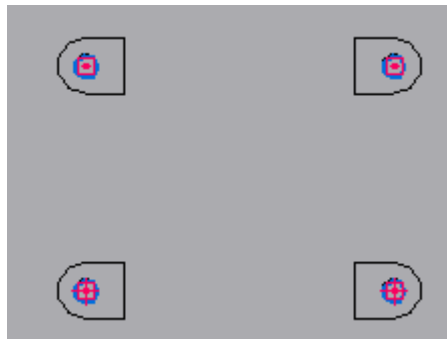
- ▶ Select the reference plane shown in the image. Use the N key on the keyboard to orient the reference plane as shown.

**Note**

When creating features for a systems library, it is good practice to orient the reference plane consistently for all the features.



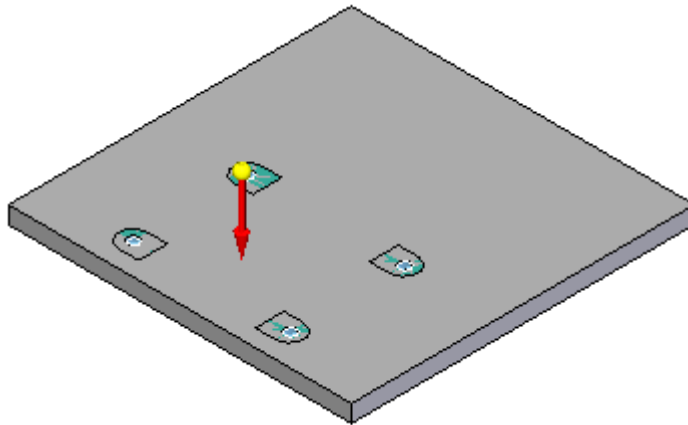
- ▶ Place four Through All holes using the center of the inter-part surfaces to place the holes.



- ▶ Choose Home tab@ Close group@ Close Sketch.



- ▶ Click with the arrow in the direction shown to define the direction of the hole extent.



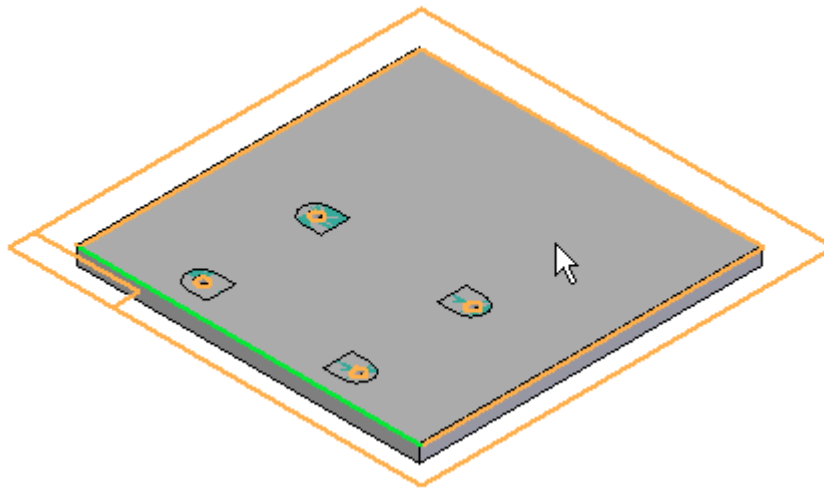
- ▶ On the command bar, click Finish.

### Create the pocket on the plate

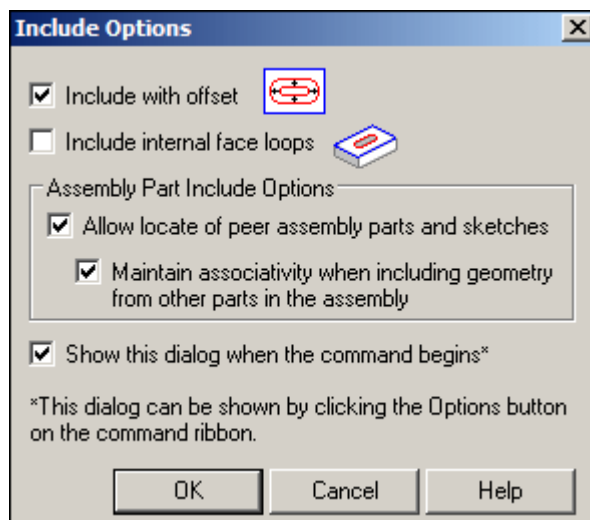
- ▶ Click the Cut command.



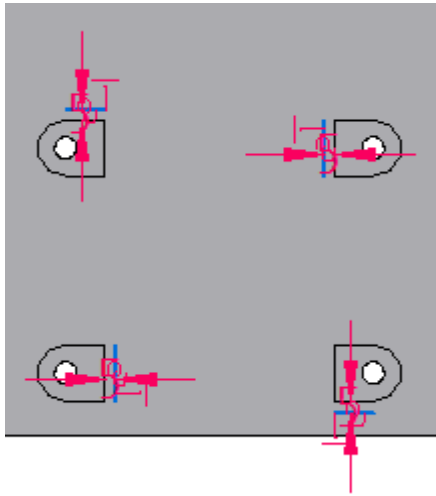
- ▶ The reference plane is created from the top face of the plate. Use the N key to orient the reference plane as shown.



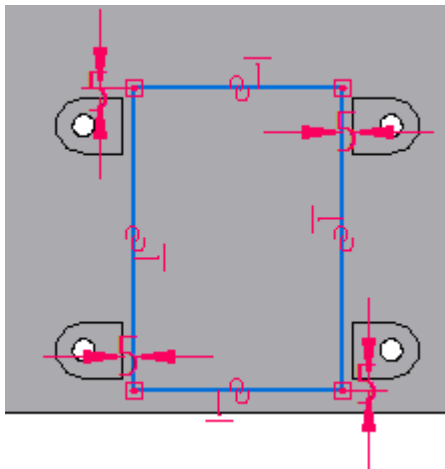
- ▶ Click the Include command and set the options as shown, and then click OK.



- ▶ As shown below, offset the bold lines a distance of 5 mm from the inter-part construction surfaces.



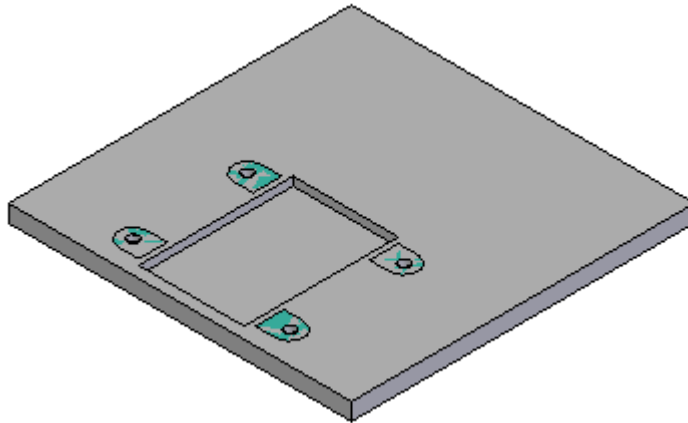
- ▶ Click Trim, and trim the 4 offset lines as shown.



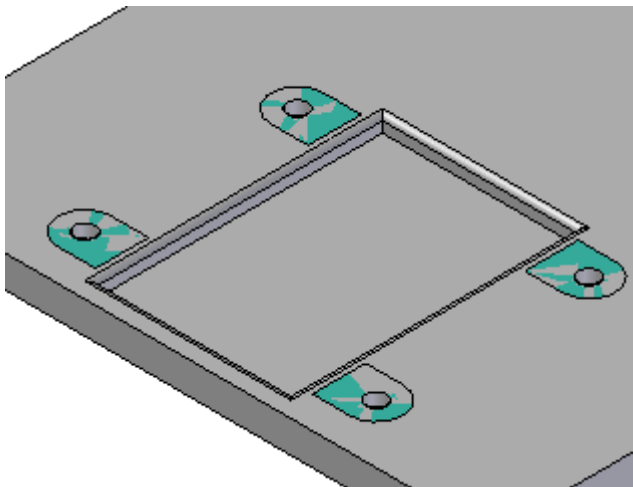
- ▶ Click Close Sketch and on the command bar, enter 7.5 mm for the extent of the cutout.



- ▶ Click to define the direction of the cutout as into the part. Click Finish to complete the cutout.



- ▶ Click the Round command. Select the top edges of the cutout, and enter a radius of 2.5 mm. Preview and finish the round.

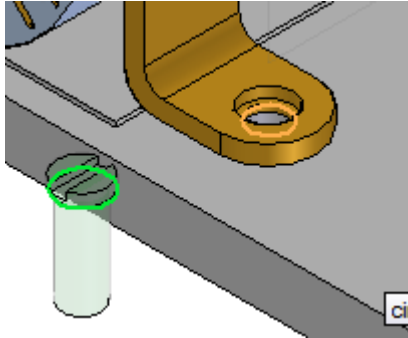


- ▶ Click Close and Return to return to the assembly.



**Place the first fastener**

- ▶ Drag *10mm\_cs\_screw.par* into the assembly.
- ▶ Using FlashFit, select the circular edge of the screw, and then select the circular edge of the hole in the foot of the motor as shown.



- ▶ The screw is placed.



- ▶ Drag *10mm\_nut.par* into the assembly.

**Note**

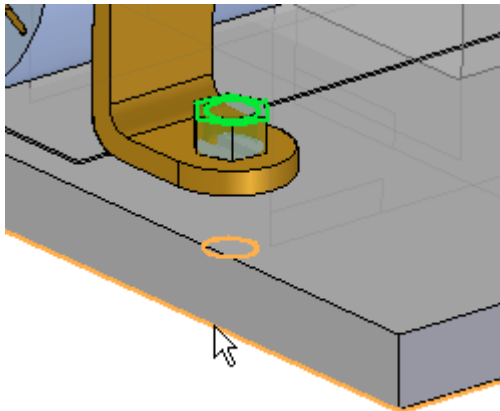
It is good design practice when creating a systems library to position subsequent parts relative to a single part that will be included in the systems library. In this case, the plate will not be a part of the systems library, but the features on the plate will be. Because all the features need to be placed relative to the motor, the motor will be used to establish as many relationships to as possible.



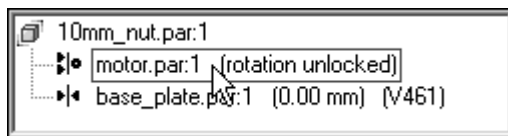
- ▶ To establish relationship 1, axial align the cylindrical face of *10mm\_nut.par* with the cylindrical face of the hole in *motor.par*.



- ▶ Mate the top face of the hex face of *10mm\_nut.par* to the bottom face of *base\_plate.par*.



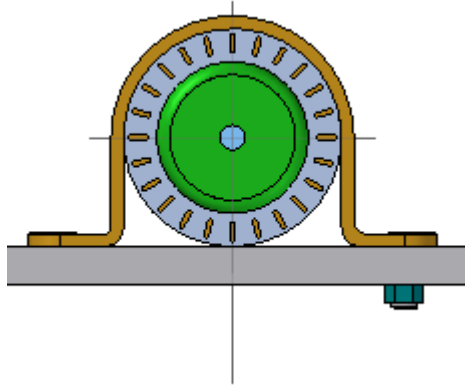
- ▶ Click the Select tool to exit the Place Part command. Select *10mm\_nut.par* in Assembly PathFinder, and in the lower plane, click the axial align relationship.



- Lock the rotation.



The nut is positioned.

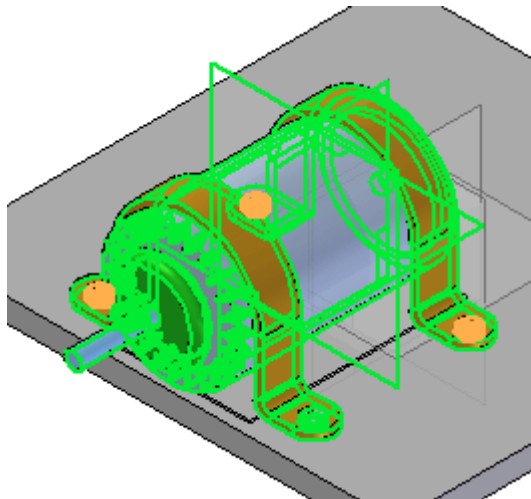


### Pattern the fasteners

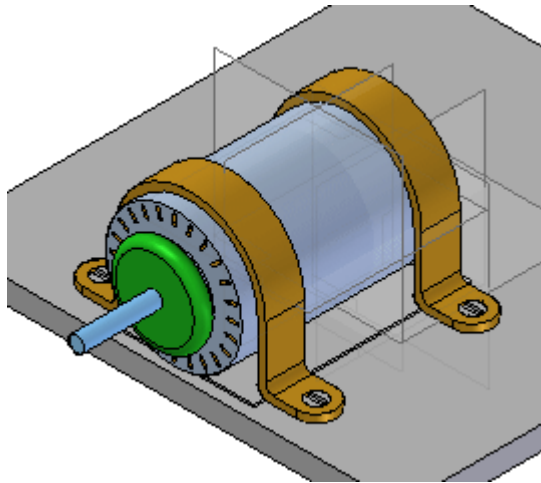
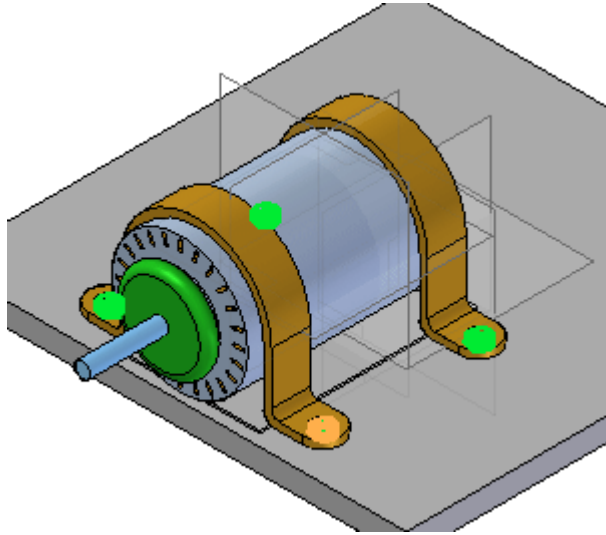
The pattern in *motor.par* will be used to pattern the fasteners.



- ▶ Click the Pattern command
- ▶ In Assembly PathFinder, select *10mm\_cs\_screw.par* and *10mm\_nut.par* to be included in the pattern, and then click the Accept button.
- ▶ When prompted to select the part that contains the pattern, select the motor.
- ▶ Select the pattern shown.




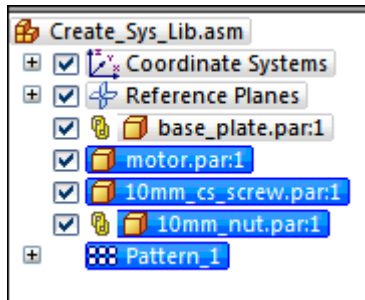
- ▶ Select the reference feature shown, and then click **Finish** to place the pattern.



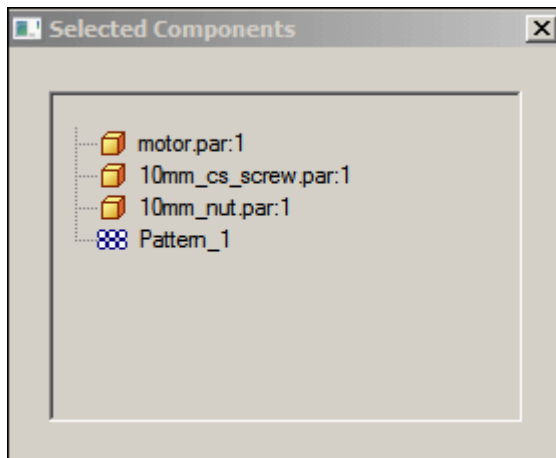
### Create the systems library

All the components of the systems library are in place and can now be stored. Create the systems library.

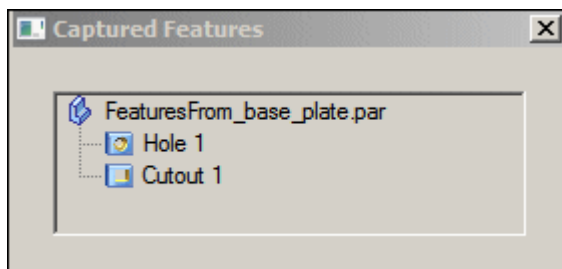
- ▶ Choose the Home tab® Assemble group® create Systems Library command .
- ▶ In the Assembly PathFinder, select all parts except *base\_plate.par*.



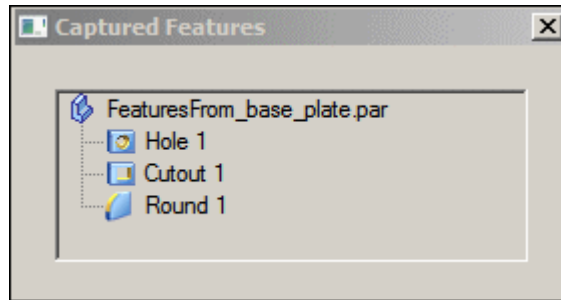
- ▶ The dialog box showing the set of parts to be added to the systems library is displayed.



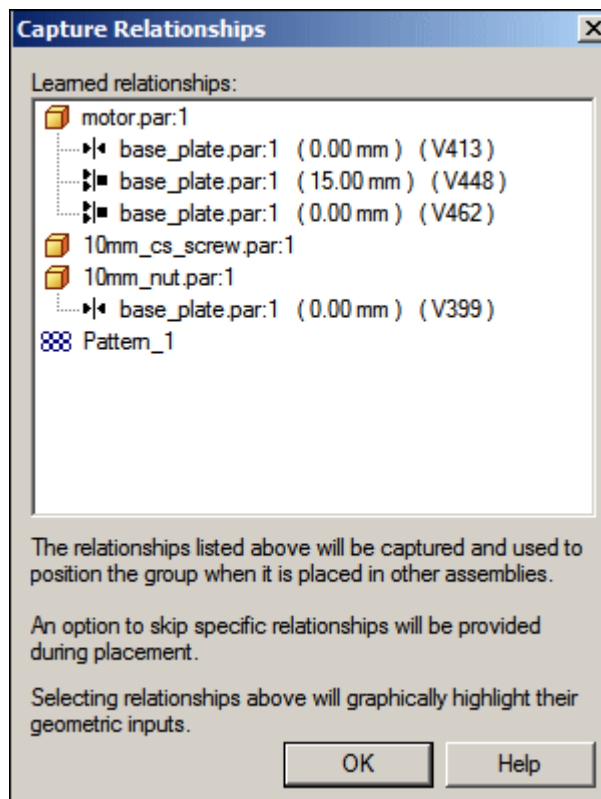
- ▶ Click Next on the command bar.
- ▶ The dialog box showing which features are to be included in the systems library is displayed.



- ▶ Add the round feature to the selection set, by clicking on the feature in the assembly window.

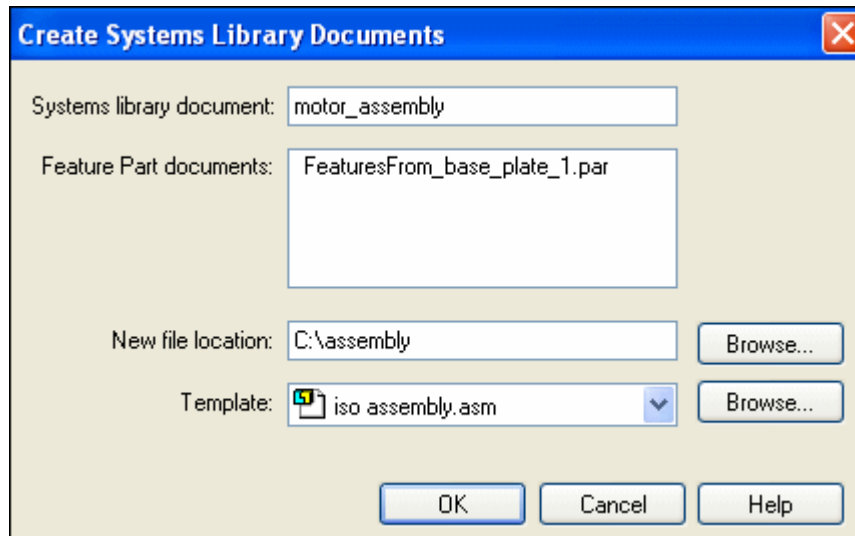


- ▶ Click Next on the command bar.
- ▶ The dialog box showing the Captured Relationships is displayed. Click OK.



- ▶ Click Create on the command bar.

- ▶ Name the new systems library *motor\_assembly.asm*. Fill in the appropriate boxes in the dialog box. Make sure the new file location is the folder containing the lab files for this activity.



- ▶ Click OK.
- ▶ The systems library is now created. Save and close the assembly.

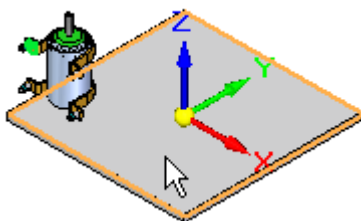
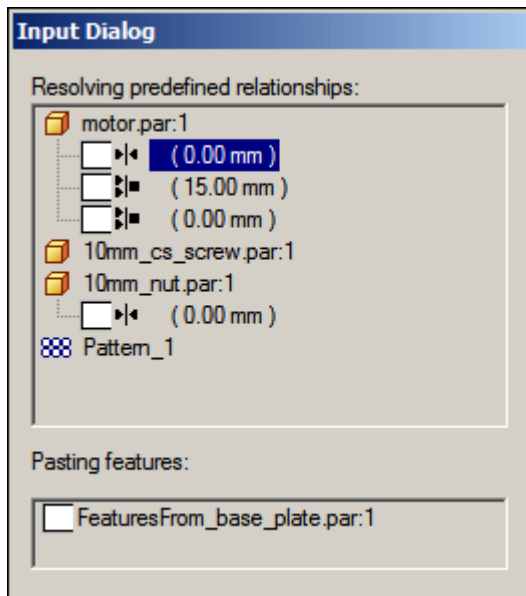
### Place the systems library into an assembly

To place the systems library, you will create a new assembly with a plate. You will place four occurrences of the systems library on the plate.

#### Note

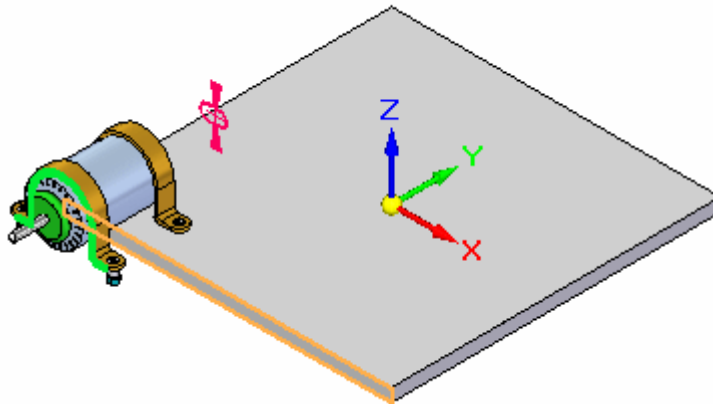
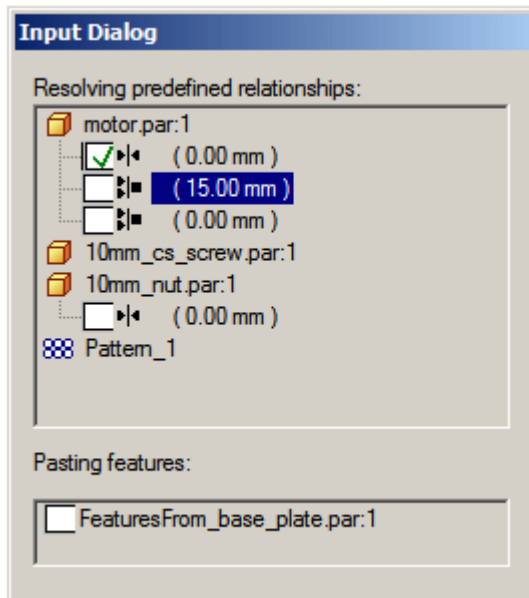
During creation of the systems library, care and consideration was given in orienting the reference planes consistently for each feature, and as many relationships as possible were established to *motor.par*. This makes accurate placement easier.

- ▶ Create a new assembly file. Save the assembly file as *newplate.asm*.
- ▶ Drag *newplate.par* into the assembly window.
- ▶ Hide the assembly reference planes.
- ▶ Right-click *newplate.par* and click Show/Hide Component, and then select the check box for Reference planes. These will be used to position the systems library on the plate.
- ▶ Drag *motor\_assembly.asm* into the assembly window.
- ▶ The first relationship to be established is the mate between the foot of the motor and the top face of the plate. Select the top face of *newplate.par*

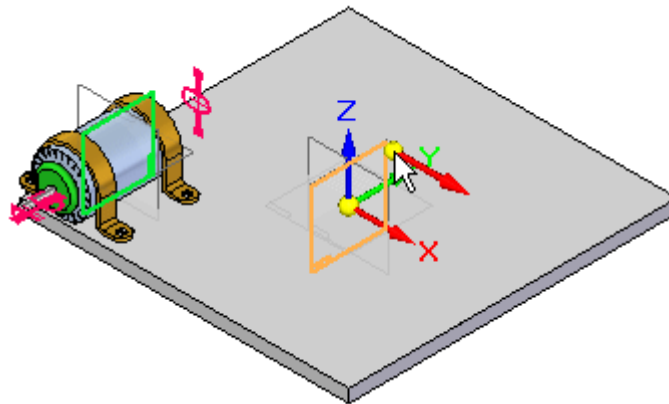
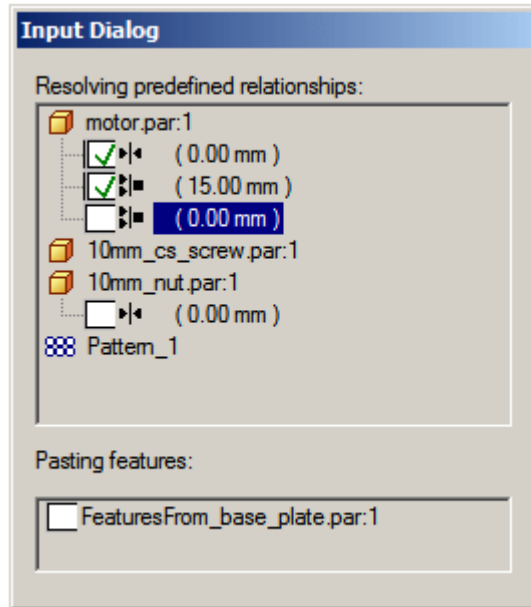




- ▶ Establish the planar align relationship with the 15 mm offset. Click the front face of *newplate.par*.



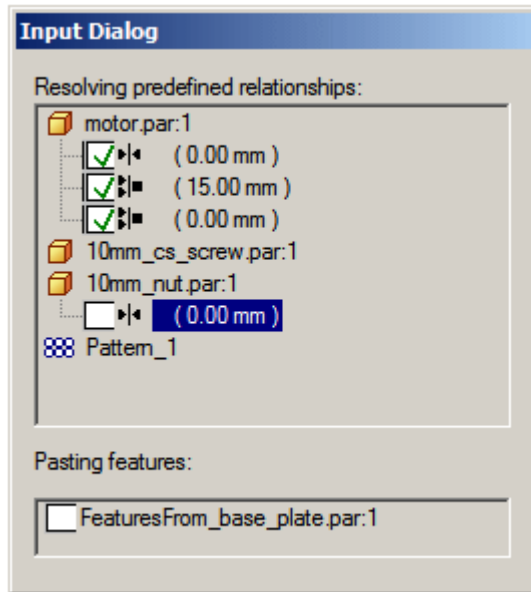
- Now the Mate relationship between the reference plane of the motor and the reference plane of the plate is to be established. Click the reference plane shown.



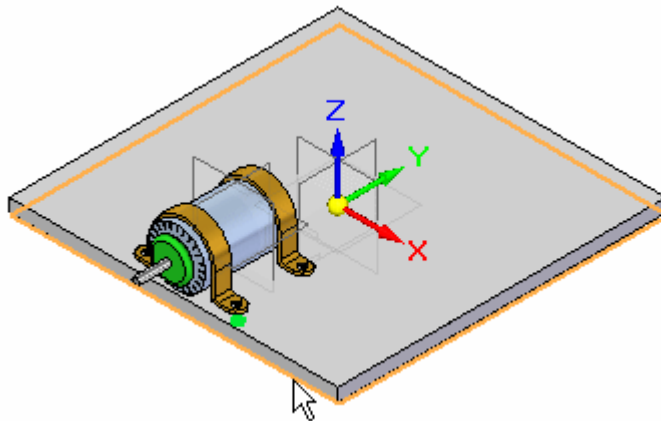
- Place the *10mm\_nut.par*. Establish the Mate relationship with the bottom plane of the plate.

**Note**

Notice that no action is required to place the screws. This is because all the relationships needed to position the screws were established relative to the motor.



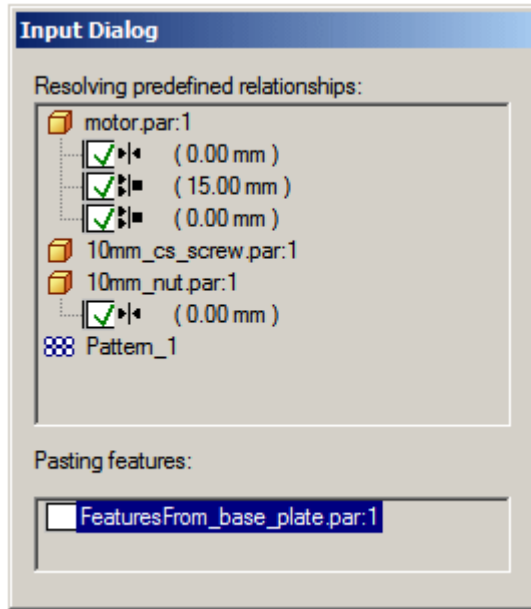
- Click the bottom face of *newplate.par*.



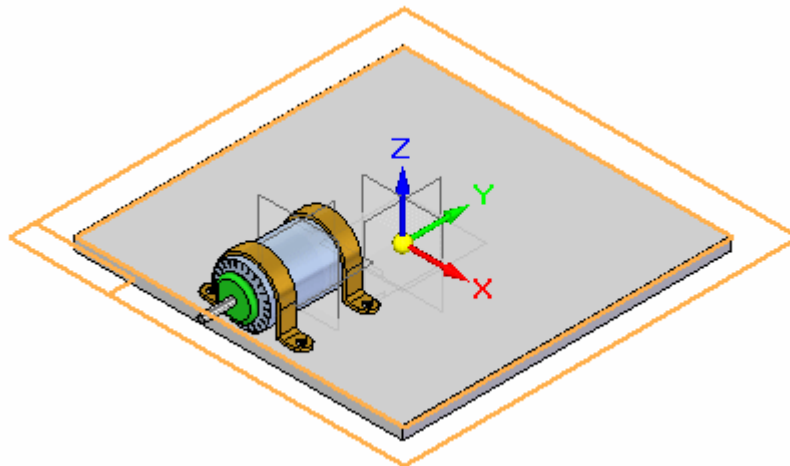
- Place the Cutout feature and the Round on the plate.

**Note**

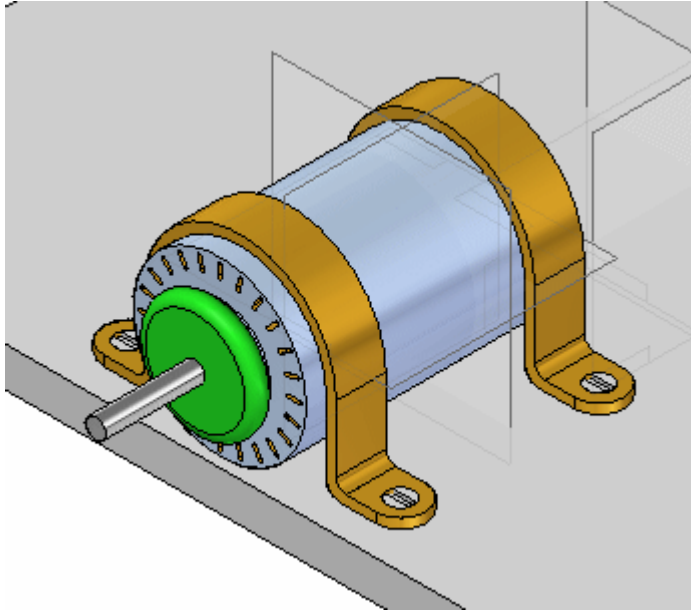
The orientation of the reference plane selected to place the features will be consistent with the orientation at the time of their creation.



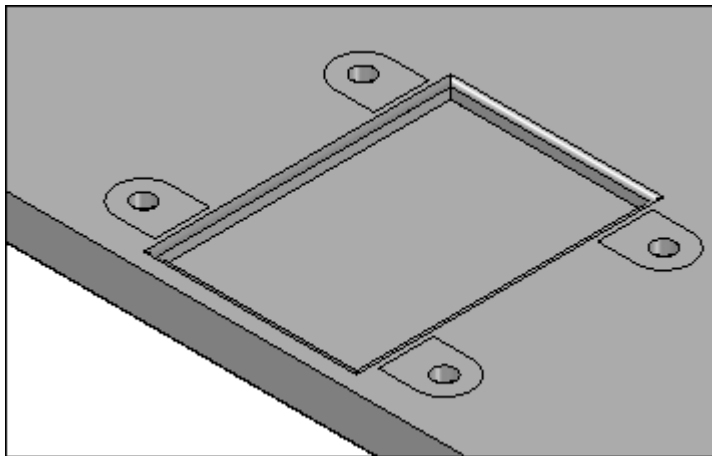
- Position the mouse so that the reference plane associated with the top face of *newplate.par* highlights. If needed, use the N key to orient the reference plane as shown, then click.



- ▶ The systems library has been placed.

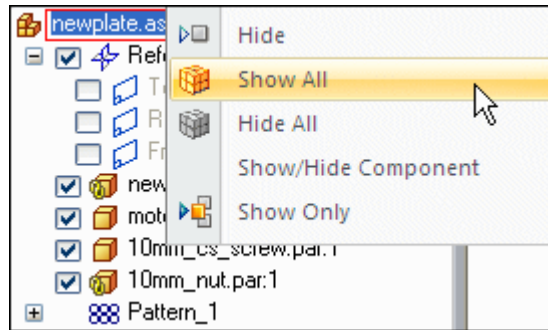


- ▶ In Assembly PathFinder, right click *newplate.par* and then click Show Only. You will see the systems library placed holes for the screws as well as the cutout and rounds under the motor.



- ▶ Save, but do not close, this assembly as *newplate.asm*.

- ▶ In Assembly PathFinder, right-click the *newplate.asm* and then click Show All.



**Place more occurrences of the system library**

- Repeat the placement of *motor\_assembly.asm* on the other 3 sides of *newplate.par* as shown. Remember to orient the reference plane correctly when placing the features.
- This completes the activity.

**Activity summary**

In this activity you learned how to place components into an assembly and define them as a systems library. Relationships were, when possible, established to the first part in the features library. Care was taken in creating profile based features such that the reference plane was oriented consistently for each feature. When the systems library was placed, the reference plane orientation for placement was consistent with the orientation used to create the feature.



## **Lesson review**

Answer the following questions:

1. What is the definition of a systems library?
2. How are features, such as holes and cutouts linked to the system library?
3. When placing assembly components in an assembly to be used in a systems library, is it good practice to define as many relationships as possible to a single component that is part of the systems library?
4. When creating features that will be placed on target components of systems library, is it good practice to keep the reference plane orientation the same for each feature?

## Answers

1. What is the definition of a systems library?

A systems library is a group of assembly components and features that can be repeatedly placed. Features that typically be needed on target parts, such as cutouts and holes can be stored and placed as the systems library is placed.

2. How are features, such as holes and cutouts linked to the system library?

When creating features to be placed on the target assembly component, the features are linked by inter-part geometry.

3. When placing assembly components in an assembly to be used in a systems library, is it good practice to define as many relationships as possible to a single component that is part of the systems library?

It is good practice to have as many relationships as possible to the one of the components in the systems library. Relationships that need to vary because of differences in target part geometry (such as varying plate thickness) do not need to be defined by the relationships to another member in the systems library.

4. When creating features that will be placed on target components of systems library, is it good practice to keep the reference plane orientation the same for each feature?

When creating features that will be placed on target components of systems library, it is good practice to orient the reference plane the same for each feature.

## **Lesson summary**

In this lesson you learned how to place components into an assembly and define them as a systems library. Relationships were, when possible, established to the first part in the features library. Care was taken in creating profile based features such that the reference plane was oriented consistently for each feature. When the systems library was placed, the reference plane orientation for placement was consistent with the orientation used to create the feature.



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**Lesson**

# *5 Working with large assemblies*

## **Working with large assemblies**

When working with large assemblies, there are tools available to speed up the process and make work more efficient. This activity will explore those techniques.

## **Working with large assemblies efficiently**

There are many ways you can improve interactive performance when working with large assemblies in Solid Edge. This Help topic describes actions you can take to improve performance when working with large assemblies.

### **How available memory affects performance**

The amount of physical memory available on your computer affects the performance of all your Windows applications, not just Solid Edge. When the physical memory is completely allocated, some operations are swapped to virtual memory. Virtual memory is disk space on your hard drive allocated for use when physical memory resources are not available.

Virtual memory is much slower than physical memory. When any application has to swap information between virtual memory and physical memory to complete a task, system performance slows down considerably. You can improve performance by increasing available physical memory in the following ways:

- Reduce the demand for physical memory
- Install additional physical memory in your computer

#### **Note**

See the *readme.htm* file in the Solid Edge folder for additional information on memory recommendations for Solid Edge.

### **Reducing the demand for physical memory**

The easiest way to reduce the demand for physical memory is to close any applications you are not using. Doing this can speed up any application, not just Solid Edge.

#### **Note**

You can use the Windows Task Manager to evaluate both physical and virtual memory usage.

## Improving display performance

Display performance in Solid Edge is improved if your computer is equipped with a graphics board that supports OpenGL acceleration. The shaded display data in Solid Edge is manipulated directly by OpenGL during dynamic view operations. The amount of physical memory on the graphics board also affects display performance.

The hidden line display mode in Solid Edge supports multiple processors. If your computer contains multiple processors, hidden line performance is improved.

The Process Hidden Edge During View Manipulations option in the View tab in the Options dialog box affects interactive performance during dynamic view manipulations, such as when dynamically rotating a view. Processing edges during dynamic view manipulations can negatively impact interactive performance when working with complex parts or large assemblies. When this option is set, the display status of edges (whether they are visible, hidden, or silhouette) is continuously calculated during dynamic view manipulations. When you clear this option, the display status of edges is suspended during a dynamic view manipulation. When the dynamic view manipulation is completed, the edges are processed. This can significantly improve performance.

You can also improve display performance from within Solid Edge Assembly by controlling which parts in the assembly use physical memory resources. Hiding and unloading parts frees up physical memory and improves display performance. Hidden and unloaded parts place the least demand on computer resources.

## Using display configurations to hide, unload, and inactive assembly components

When you are building or modifying a large assembly, you often work in a localized area, or on a limited number of parts for a period of time. You can use the display configuration control commands in Solid Edge to make it easier to work and to improve performance in large assemblies.

For example, you can use the [Display Configurations command](#) to capture the current display status of the assembly components to a name you define. After you define a display configuration, you can use the Assembly Configuration list on the Select tool command bar to apply a display configuration. This allows you to quickly display, hide, inactivate, and unload specific parts and subassemblies.

Each of these display states places a different demand on physical or graphic memory. Additional configuration control commands are located in several places in Solid Edge. For example, they are available on the PathFinder shortcut menu and the Home tab in the Configurations group.

You can determine the current display status for parts and subassemblies using PathFinder. Unique symbols indicate whether an assembly component is displayed, hidden, inactive, or unloaded. For a list of these and other symbols used in PathFinder, see the [PathFinder in assemblies](#) Help topic.

### Note

If the inactive part or subassembly contains construction surfaces, the construction surfaces will be automatically hidden. You can redisplay the construction surfaces by activating the part or subassembly.



### **Hiding components**

You can hide parts and subassemblies to make it easier to visualize the area you are working on. This makes your display less cluttered and helps you work more efficiently because you can locate and select the proper parts quickly. Hiding components also reduces physical memory demands, which improves performance.

### **Unloading components**

Once the unneeded parts and subassemblies are hidden, you can also unload them from physical memory with the Unload Hidden Parts command on the Tools menu. Unloading hidden parts from memory frees up memory for other operations, which speeds up Solid Edge Assembly functions.

It also allows you to work with assemblies with higher component counts and more unique components. In some cases, unloading hidden parts from memory allows you to open and work with an assembly that would otherwise exceed the memory limits on your computer.

If you hide and then unload all the parts in a subassembly, the component structure within the subassembly listing in PathFinder is collapsed. You can use the plus symbol (+) in PathFinder, the Expand, and the Expand All commands to redisplay the assembly structure within PathFinder.

### **Inactivating components**

You also can improve performance without hiding parts and subassemblies by inactivating them. When you inactivate components with the Inactivate command, they remain displayed but use less physical memory.

Inactivating parts unloads all the feature history and mathematical part definition and leaves only the graphical representation of the part loaded into memory. This significantly reduces physical memory requirements, which can be useful when working with large assemblies.

An inactive part is automatically activated when you use it to position another part, or when you use the Edit or Open commands to open it in the Part environment.

You also can inactivate components automatically by setting the Inactivate Hidden and Unused Components Every XXX Minutes option, which is on the Assembly tab on the Options dialog box.

### **Expanding assemblies**

The Expand command expands only next level of references for a selected subassembly. If the subassembly also contains nested subassemblies, the structure for the nested subassemblies is not expanded.

The Expand All command expands the entire structure for a selected subassembly, including any nested subassemblies.

You can also use the plus (+) and minus (-) symbols in PathFinder to expand or collapse subassembly listings in PathFinder.

### Using simplified assemblies

When working with a large assemblies, you can create and use simplified assembly representations using the commands in the Simplify group on the Tools tab in the Assembly environment.

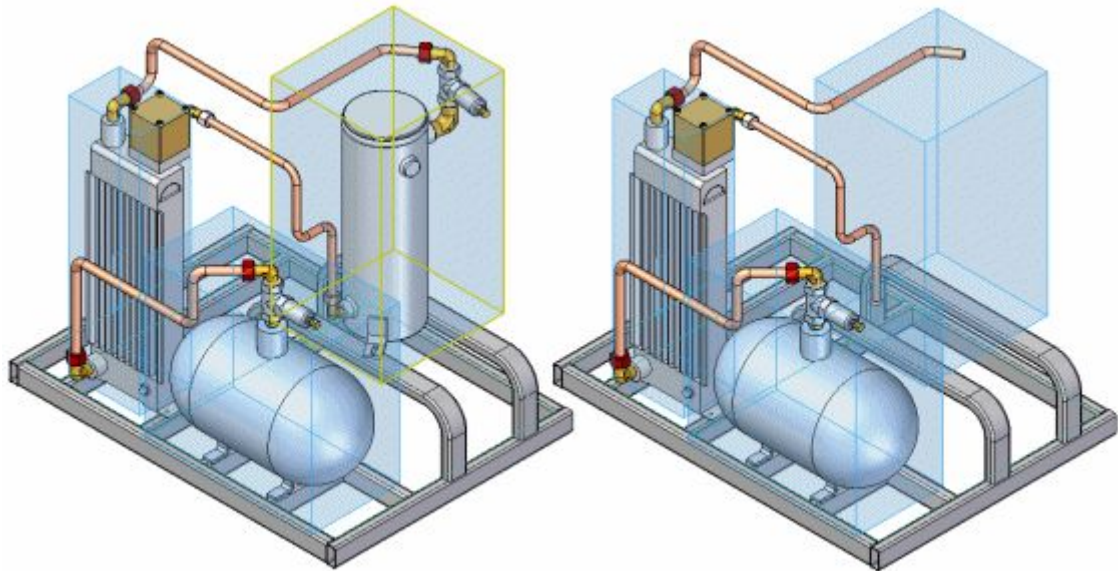
A simplified representation of assembly only contains the external shell faces of the parts in the assembly, which can significantly reduce the memory requirements required when working with a large, nested assembly. You can then specify that the simplified representation is used in higher level assemblies, in drawings, and when opening assemblies.

When working with large, nested assemblies that contain simplified subassemblies, you should work with the simplified subassemblies inactive whenever possible. This can significantly reduce memory requirements.

For more information, see the [Simplifying assemblies](#) Help topic.

### Using zones

You can use the Zone command on the Select Tools tab on PathFinder to define a named rectangular volume of space. You can then display, hide, and select the assembly components which are contained within the zone. For example, you can select a zone in the Select Tools tab, then use the Hide command on the shortcut menu to hide the assembly components within the zone.



When you add new components to the assembly, they are automatically added to the zone(s) they fall within.

For more information on zones, see the [Using zones in assemblies](#) Help topic.

## Improving performance when opening assemblies

You can use the options on the Open File dialog box to improve performance when opening a specific assembly. You can also use the Save As Default option on the Open File dialog box to specify that assemblies you open using Windows Explorer are opened with the options you set.

- You can open the assembly as *read-only*, then edit only the portion of the document where you have responsibility.
- You can set the Hide All Components option to open the assembly with all parts hidden. You can then display only the parts you want to work with. The fastest way to open an assembly is with all parts hidden.
- You can select a display configuration from the Configuration list when opening an assembly. This can improve performance when the display configuration specifies that only a subset of the assembly components are displayed.
- You can select a zone from the Zones list when opening an assembly. This can improve performance when the zone definition specifies that only a subset of the assembly components are displayed.
- You can set the activation status of assembly components with the Apply Activation Override option on the Open File dialog box. You can then use the Activate command to activate only the parts you want to work with.
- You can also open the assembly with all parts active, or, by clearing the Apply Activation Override option, open it with the active/inactive statuses that were in use when the assembly was last saved.

When you open an assembly with all parts hidden, only the assembly document you are opening is loaded into physical memory.

When working with an assembly with thousands of parts and dozens of subassemblies, this can dramatically reduce the time required to open the assembly. Because these types of large assemblies are typically shared by many users who only work on specific subassemblies within the top-level assembly, hiding all parts when opening the assembly allows you to work efficiently with a very large assembly.

When you open an assembly with all components hidden, the subassembly listings in PathFinder are collapsed. You can open the next level of documents using the "+" symbol, or you can expand the subassembly listings using the Expand and Expand All commands on the shortcut menu.

You can expand the listings for only those subassemblies you need, then display or activate only the parts and subassemblies you need.

Options on the Open File dialog box also enable you to specify whether the simplified representation of assemblies are used when opening an assembly. When a simplified representation of an assembly exists, using these options improves file open times.

## Highlighting parts

You can improve large assembly performance by setting the Fast Locate Using Box Display option on the Assembly tab on the Options dialog box. When you pause your cursor over a part in the assembly, it will highlight using a rectangular range box, instead of all of the graphic display elements of the part.

Setting the Fast Locate When Over PathFinder option on the Assembly tab on the Options dialog box also allows you to improve performance. When you set this option, the name of the assembly component is displayed in the message field when you pass the cursor over the component name in PathFinder, but it does not highlight in the graphics window. When you clear this option, the assembly component highlights in the graphics window when you pass the cursor over the component name in PathFinder.

## Selecting parts

Solid Edge provides different options for selecting parts in the Assembly environment using the Select Tool command. Following are short descriptions of the selection methods and related forms.

### Selecting parts with the mouse

The mouse click is the fastest method for selecting single parts. Parts can be selected either in the graphics window or from PathFinder. To select a part with the mouse, simply click the part in the graphics window or the part name in PathFinder. To create a select set of parts, hold down the Shift key and select multiple parts with the mouse.

### Selection Box

This command lets you select a group of parts by drawing a dynamic 3D box in the graphics window. This button is available only when a single part is selected. This command is used to quickly select parts that are in the proximity of another selected part. After creating select sets, you can manipulate them as a group. For example, the Show Only command on the PathFinder shortcut menu can be used to hide all parts not selected, leaving on the select set in the immediate display.

### Select All Identical Parts

Selects all the parts in the assembly which are identical to the selected part.

### Select Subassembly Parts

Selects all the parts in multiple occurrences of the same subassembly as the selected part.

### Select Small Parts

This command selects parts uses a spin box to select parts dynamically by their respective size as a percentage of the entire assembly size. Use the up and down arrows on the box to increase or decrease the size criteria.

### Select Visible Parts

This option selects only those parts that are fully or partially visible in the graphic window.

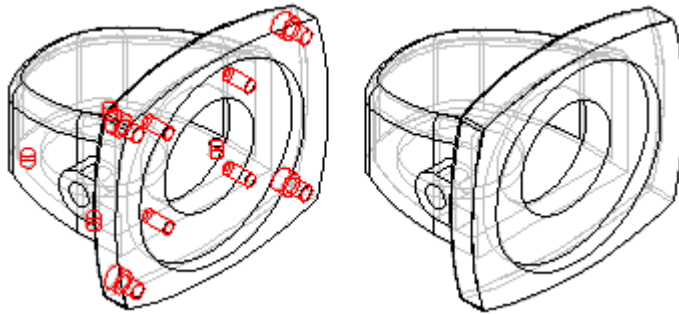
### **Select Parts Constrained To**

Selects parts that are constrained to one or more previously selected parts. This option is available after you select one or more parts. This can make it easier to perform operations on a related set of parts. For example, you can select a set of parts constrained to a part, then use the Show Only option to hide the remaining parts in the assembly.

**Simplifying parts**

When working with an assembly, it can be useful to work with a simplified version of a complex part. For example, a part that contains numerous rounds, chamfers, and holes will process more slowly than a part from which these features have been removed.

The commands in the Simplify Model environment allow you to reduce the complexity of a part so that it processes more quickly when used in an assembly. The ultimate goal of part simplification is to reduce the total number of surfaces that make up the part.



You can also control whether the simplified version or the designed version of the part is displayed in the assembly.

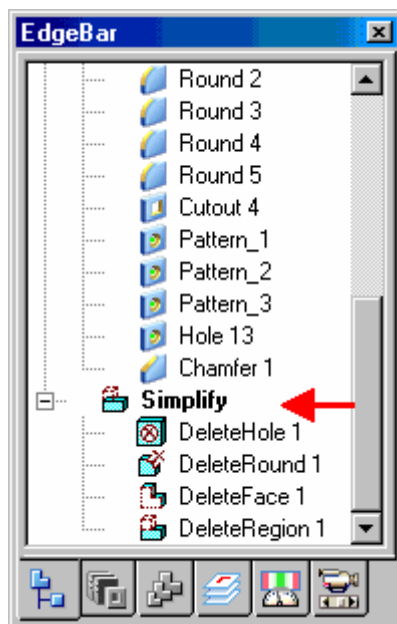
## Simplifying parts

You access the commands for simplifying a part using the Simplify command on the Tools tab in the Part and Sheet Metal environments. When you set the Simplify command, commands that can be used for simplifying a part are activated, and commands that are not appropriate for simplifying a part are disabled. For example, the commands on the Delete list on the Home tab in the Modify group are available for deleting faces, regions, and so forth.

After you have simplified a part, you can return to the Part or Sheet Metal environment with the Design command on the Tools tab.

You can also simplify a part by adding extruded and revolved protrusions, and extruded and revolved cutouts. Feature construction commands are included in the Simplify Model environment because sometimes it can be easier to simplify a part by adding one new feature than deleting many features. For example, you can construct one protrusion that obstructs several features, which then eliminates dozens of surfaces in one operation.

The features you create in the Simplify Model environment are added to the Simplify section of the PathFinder tab in the part document. You can also use the commands on the shortcut menu within PathFinder to manipulate the simplified features you construct.



## Direct editing and simplifying compared

Many of the commands available when directly editing a part are also available when simplifying a part in the Simplify Model environment. Deciding whether to directly edit the model or simplify the model is determined by whether you want to have access to a simplified version of the part in an assembly or when creating a drawing.

If you want to use a simplified version of the part in an assembly or a drawing, you must set the Simplify command. No simplified version of the model is created when you directly edit a model.

### **Saving simplified parts to a separate file**

You can save the simplified representation of the part out to a separate file using the Save Model As command on the Application menu. The Save Model As dialog box allows you to specify a file name, folder location and file format. You can save the new document as a Solid Edge document or as a Parasolid body, and it is not associative to the original model.

### **Simplified parts in assemblies**

When placing a part in an assembly, you can place it using the simplified version of the part, or the designed version of the part. When you set the Use Simplified Parts command on the Parts Library shortcut menu, the simplified version of the part is displayed when placing the part in an assembly. Any faces that you deleted when simplifying the part will not be available for positioning the part in the assembly. To make these faces available for positioning, clear the Use Simplified Part command before placing the part.

When working with simplified parts in an assembly, you can control whether the simplified version or as-designed version of the part is displayed. If you place the same part in an assembly more than once, you can control the display for each instance of the part individually. When you select a part in the assembly, you can use the Use Designed Part and Use Simplified Part commands on the shortcut menu to control which version of a selected part is displayed.

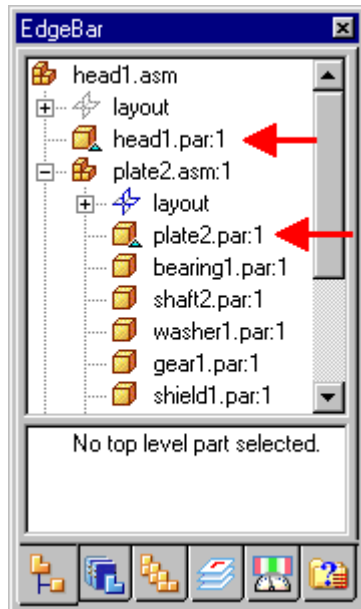
#### **Note**

The Use Simplified Part command is not available for parts that were selected for an assembly feature, whether the parts were modified by the assembly feature, or not modified.



## Simplified parts in PathFinder

The symbols adjacent to each part in the PathFinder tab in an assembly change to indicate whether the simplified version or designed version of the part is currently displayed.



## Locating edges in simplified models

In some cases there may be edges in a simplified part that are not locatable. This occurs when the simplified version of the part creates edges that do not have corresponding design edges.

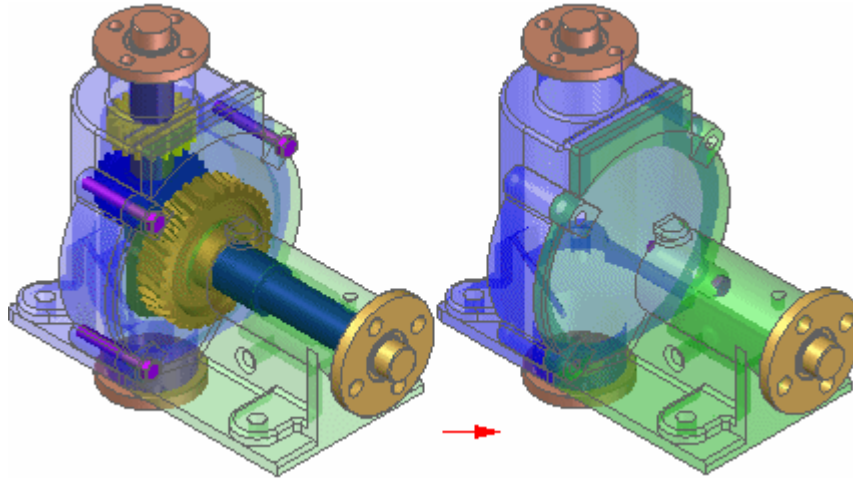
When this occurs you will not be able to locate an edge on a simplified part when placing a dimension or relationship, or when including an edge.

Location of these edges is purposely prevented to ensure stability of the model in downstream operations.

### Simplifying assemblies

When working with a large, complex assembly, it can be useful to work with a simplified version of the assembly. For example, a large assembly with many subassemblies can process slowly.

The commands in the Simplify group on the Tools tab in the Assembly environment allow you to create an exterior shell of faces that represents the assembly envelope, and exclude selected parts.



A simplified assembly processes more quickly when used in a higher level assembly or a drawing. You can also control whether the as designed or simplified assembly representation is used in other documents and when opening an assembly. This allows you to work with larger data sets more efficiently.

You can also save the simplified representation of the assembly to a new document name. This can make it easier to share or protect proprietary information when exchanging data with other companies that need access to your data.

### Simplified assemblies and memory usage

When you create a simplified representation of an assembly, the data storage requirements for the assembly document increase because the surface data for the simplified representation is stored in the assembly document.

The size increase required to support the simplified representation is small when compared to the size requirements of all the documents that make up the assembly.

When you place a simplified assembly document as a subassembly into another assembly, the memory requirements required to display the higher level assembly drop dramatically. This improves performance and also allows you to work with larger data sets more effectively.

This performance improvement also applies when creating a drawing of a simplified assembly. Because less memory is required to support the simplified data set, the drawing views will process quicker.

For more information on working with simplified assemblies in Solid Edge, see the [Simplified assemblies best practices](#) Help topic.

## **Simplifying an assembly**

You access the commands for simplifying an assembly using the Simplify Assembly command on the tools tab in the Model group. The Simplify commands allow you to create and update the simplified representation and save the simplified representation as a separate document. After you have simplified the assembly, you can return to the Assembly environment using the Design Assembly command on the Tools tab in the Model group.

When you create a simplified representation of an assembly, an entry is added to PathFinder to indicate that a simplified representation of the assembly exists.

## **Creating the simplified representation**

The Create Simplified Assembly command allows you to process the assembly to show only the exterior envelope of faces and to exclude parts, such as small parts, which reduces the total number surfaces that make up the assembly. The simplified assembly representation is associative to the components in the assembly.

## **Updating simplified assemblies**

When you make design changes to assembly components, you must use the Update Simplified Assembly command to update the simplified representation before the design changes are displayed in a higher level assembly or drawing that uses the simplified representation.

## **Saving the simplified representation as a document**

You can use the Save Model As command on the Application menu to save the simplified representation of the assembly as a new Solid Edge Part document (\*.PAR) or as a Parasolid document.

This can be useful when another company uses your assembly as a part in their assemblies. This reduces the data-management and transfer requirements to a single document, and it can also protect any proprietary information that a complete assembly might reveal.

## **Using simplified assemblies**

You can specify whether the as designed version or the simplified representation of the assembly is used in many down-stream operations. In some cases, other Solid Edge functionality requires that the as designed version is used. For example, you cannot create a simplified assembly representation of an alternate assembly.

If you have already created a simplified assembly representation, then try to convert the assembly to an alternate assembly, a message is displayed to warn you that the simplified representation will be deleted.

## **Placing simplified assemblies in other assemblies**

When you place an assembly as a subassembly in another assembly, you can specify whether the assembly is placed using the as designed or simplified version. The Use Simplified Assemblies command on the Parts Library shortcut menu allows you to specify how the assembly is placed. When you place the simplified version of the assembly, only the faces that comprise the simplified representation of the assembly are available for positioning the assembly using assembly relationships.

You can use the commands on the PathFinder shortcut menu to specify whether the as designed or simplified version of a subassembly is used. You can control each subassembly in an assembly individually. This allows you to display the as designed version of a subassembly when needed, and then switch to the simplified version of the subassembly later to improve performance.

## **Inactivating Simplified Subassemblies**

When working with large, nested assemblies that contain simplified subassemblies, you should work with the simplified subassemblies inactive whenever possible. This can significantly reduce memory requirements.

You can use the Inactivate and Activate commands on the PathFinder shortcut menu to inactivate and activate a simplified subassembly. When placing or editing relationships, you can use the Activate button on the Assemble command bar to activate a simplified subassembly.

## **Creating drawings of simplified assemblies**

When creating or modifying drawing views of an assembly, options on the Drawing View Wizard and Drawing View Properties dialog box allow you to control how simplified assembly representations are applied.

## **Opening simplified assemblies**

When opening an assembly, options on the Open File dialog box allow you to control how simplified assembly representations are applied.

## **Moving simplified assemblies**

Because the simplified representation is considered a construction body, collision detection is not available using the simplified representation.

## **Checking interference on simplified assemblies**

Because the simplified representation is considered a construction body, interference detection is not available using the simplified representation.

### *Simplified assemblies best practices*

When considering whether to simplify an assembly, there are several factors that can determine whether you receive the maximum benefit from assembly simplification. This Help topic discusses these factors.

## Simplify both parts and assemblies

Using simplified parts in conjunction with a simplified assembly improves simplified assembly performance in the following ways:

- Faster creation of the simplified assembly representation because there are fewer total surfaces to evaluate.
- Reduced assembly document size, which reduces memory demand.

When you simplify parts before simplifying the assembly, you are reducing the total number of surfaces which must be evaluated during assembly simplification. This allows the assembly simplification process to complete faster.

The wide range of commands available for part simplification also allow you better control over which part surfaces are removed prior to assembly simplification. For example, there may be holes and cutouts on an exterior part that are not necessary in the simplified assembly. If these holes and cutouts expose interior faces, the interior faces will be included in the simplified assembly.

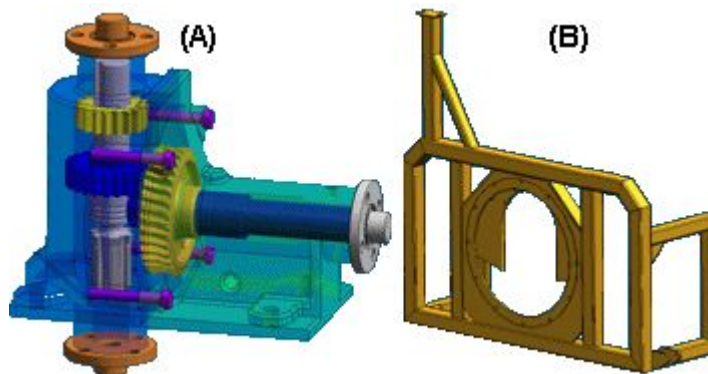
By simplifying the exterior part to remove the holes and cutouts, the simplified assembly representation will contain fewer surfaces, which reduces file size and memory demands.

## Simplify assemblies that contain enclosures

Some assemblies are better suited to assembly simplification than others. In general, assemblies that have many interior components are better candidates for assembly simplification. This is because assembly simplification processes the assembly to show only the exterior envelope of faces and by excluding small parts.

For example, assembly (A) is an ideal candidate for assembly simplification because there are many complex components enclosed by the exterior housing. The exterior housing itself also contains many interior surfaces that would be excluded by assembly simplification.

Assembly (B) is a poor candidate for assembly simplification because there are few interior surfaces and no interior components to exclude. If assembly (B) is also used in a higher level assembly, where assembly (B) is enclosed, that higher level assembly may be a better candidate for simplification.



### **Open assemblies with simplified subassemblies**

When you open an assembly that contains simplified subassemblies, you can specify whether the assembly is opened with the subassemblies simplified or as designed. When you open an assembly with the subassemblies simplified, file open times are improved.

### **Simplify deeply nested assemblies**

The more subassemblies and parts an assembly has, the more likely the assembly is a good candidate for simplification. When you open a large assembly with many subassemblies that have been simplified, file open performance is improved.

### **Inactivate simplified subassemblies**

When working with large, nested assemblies that contain simplified subassemblies, you should work with the simplified subassemblies inactive whenever possible. This can significantly reduce memory requirements.

### **Avoid simplifying top-level assemblies**

The simplified assembly representation is stored in the assembly document in which it was created, which increases the file size. An assembly that has been simplified will open slower than an identical assembly which has not been simplified.

This means that the top-level assembly itself does not benefit from assembly simplification.

Although performance is improved when creating drawing views of a top-level assembly that has been simplified, in most cases this performance improvement does not offset the performance impact when opening the top-level assembly.

## **Activity: Using Simplified Assemblies**

### **Overview**

The objective of this activity is to show how a large assembly can be simplified.

### **Activity**

In this activity you will create a simplified assembly.

**Activity: Creating a simplified assembly****Overview**

This activity demonstrates the creation of a simplified assembly.

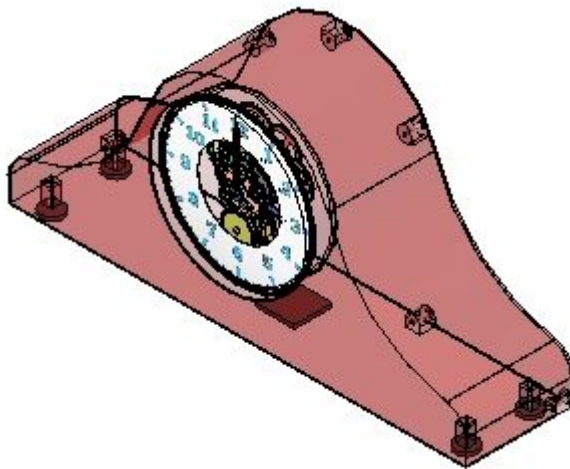
**Objectives**

The objective of this activity is to take a large assembly and create a simplified version to be used in a higher level assembly to make processing more efficient.

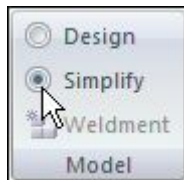
*Open an existing assembly*




The clock assembly has many internal parts that do not need to be represented in a higher level assembly.

- ▶ From the Solid Edge start screen, click Open Existing Document. Browse for *clock.asm* in the folder where the activity files are located.



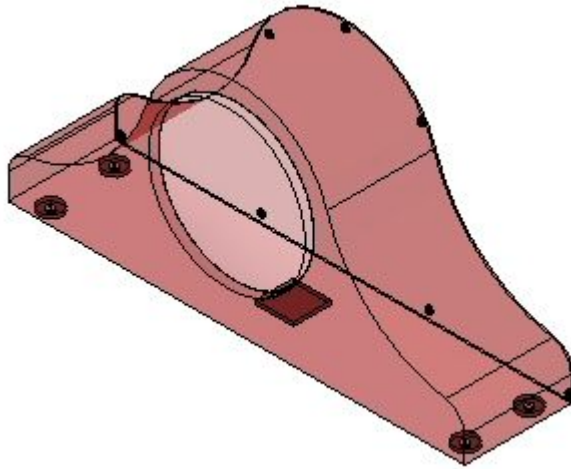
- ▶ Click Tools tab@ Model group@ Simplify command.



- ▶ Click Tools tab@ Simplify group@ Create command .
- ▶ On the Create Simplified Assembly command bar, select the Options button  and review the colors that are assigned to the simplified assembly.
- ▶ Click the Analyze Assembly button , and then click Process.
- ▶ Click Preview.



- ▶ Click Finish. The simplified assembly is created and ready to be used.



- ▶ Click Tools tab@ Model group@ Design command .  
This restores the assembly to the design state.
- ▶ Save and close the assembly. This completes this activity.

### Summary

In this activity you learned how to simplify an assembly. The simplified version can be used in a higher level large assembly, and is more efficient because the interior details do not have to be shown.

**Lesson review**

Answer the following questions:

1. What is a simplified assembly?
2. Why is a simplified assembly desirable?
3. Can a simplified assembly be saved as a Solid Edge Part Document? Why is this desirable?
4. Generally, what type of assembly is better suited as a candidate for simplification?

## **Answers**

1. What is a simplified assembly?

A simplified assembly shows only the exterior envelope of faces and excludes parts, such as small parts, which reduces the total number surfaces that make up the assembly. The simplified assembly representation is associative to the components in the assembly.

2. Why is a simplified assembly desirable?

When you place a simplified assembly document as a subassembly into another assembly, the memory requirements required to display the higher level assembly drop dramatically. This improves performance and also allows you to work with larger data sets more effectively.

3. Can a simplified assembly be saved as a Solid Edge Part Document? Why is this desirable?

Yes the simplified assembly can be saved as a Solid Edge Part Document.

This can be useful when another company uses your assembly as a part in their assemblies. This reduces the data-management and transfer requirements to a single document, and it can also protect any proprietary information that a complete assembly might reveal.

4. Generally, what type of assembly is better suited as a candidate for simplification?

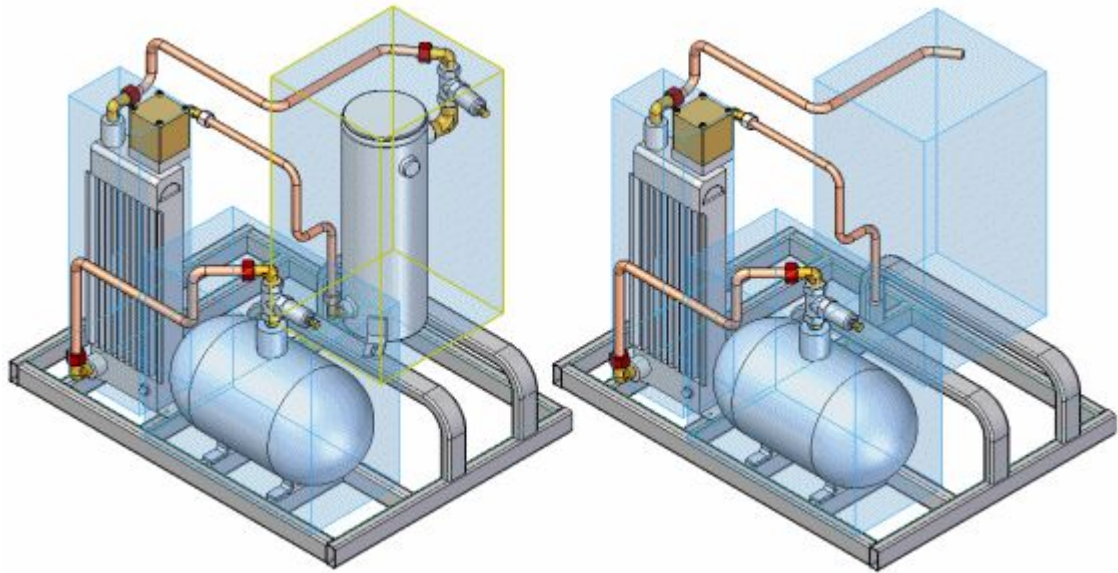
Assemblies that are enclosures with many internal parts are generally good candidates for simplification.

**Lesson summary**

In this lesson you learned how to simplify an assembly. The simplified version can be used in a higher level large assembly, and is more efficient because the interior details do not have to be shown.

## Using zones in assemblies

It can be useful to define a set of components in an assembly based on the volume of space the parts occupy. The Zones functionality on the Select Tools page in Solid Edge allows you to define a rectangular volume of space based on one or more assembly components you select. You can then use that named zone to select, display, or hide all the assembly components that are contained within the boundary of the zone. For example, you can select a named zone on the Select Tools page, then click the Hide Components command on the shortcut menu to hide all the assembly components in the zone.



### Note

You can use a zone to show, hide, and select parts, assemblies, assembly sketches, weld beads, coordinate systems, and reference planes in an assembly. Defining and using zones is especially useful when working with large assembly data sets. For more information on working with large assemblies, see the [Working with large assemblies efficiently](#) Help topic.

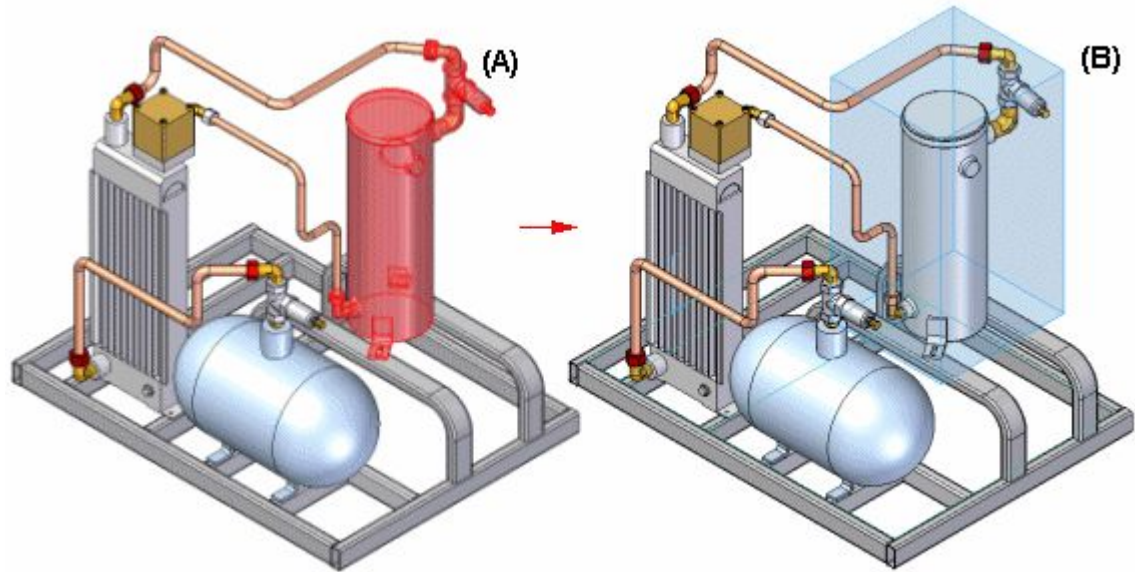
## Defining zones

You define a zone using the Create Zone command on the Select Tools page. When you click the Create Zone button, a command bar is displayed, with the Origin and Size Step active, which allows you to define the initial volume of the zone by selecting one or more assembly components. The range box of the design body of each selected component is used to calculate the zone box volume.

### Note

Construction bodies and simplified bodies are ignored when calculating zone box volume.

You can select the components in the PathFinder page or the graphics window. For example, you can select a subassembly (A) using PathFinder. When you click the Accept button, a zone boundary box is displayed in the graphics window (B). The command bar then advances to the Modify Size Step, which allows you to further refine the size of the zone.



To change the size of the zone during the Modify Size Step, you can select a face on the zone box, and then select a keypoint on an adjacent part or a point in free space.

To control which components are included in the zone, you can specify whether the zone definition is inside or overlapping using the options on the command bar.

When you finish defining a zone, a zone boundary box is displayed in the graphics window (B), and a zone name is added to the Select Tools page. You can also edit the size of a zone later.

The zone boundary box volume is not associative to the parts used to define the zone. If you move, edit, or delete the parts used to define the zone, the size of the zone boundary box remains the same.

### Note

When you create the first zone in an assembly, the assembly structure must be updated. A message is displayed to warn you that updating the assembly structure may take several minutes, depending on the size of the assembly. For more information, see the [Updating the assembly structure](#) section of this Help topic.

### Modifying zones later

You can edit the size of a zone later using the Edit Definition command on the shortcut menu when you select a zone entry in the Select Tools page, or a zone box in the graphics window. If you click the Origin and Size Step on the command bar, you can select new parts to redefine the zone boundary. If you click the Modify Size Step, you can edit the size of the zone boundary as described earlier.

## Displaying zone boundary boxes

The zone box can be displayed or hidden in the graphics window.

- You can use the zone shortcut menu commands to show and hide a zone boundary box display when you select a zone entry on the Select Tools tab or in the graphics window.
- You can show or hide all the zone boundary boxes using the View tab@ Show group@ Zones command.

## Using zones

Shortcut menu commands in the Select Tools page allow you to show only, show, hide, and select the components in a zone. You can also show all the parts in a zone by double-clicking a zone box in the graphic window.

A part can belong to one or more zones. As you add new components to an assembly, the components are automatically added to the zone(s) they fall within.

## Drawing views of assembly zones

You can use the Drawing View Wizard to select an assembly zone volume to be shown in a drawing view. The assembly zone captures any part components that exist within the predefined volume.

You can choose an assembly zone from the .cfg, PMI model view, or Zone list on the Drawing View Options page of the Drawing View Wizard.

## Zones and display configurations

The functionality of zones and display configurations differ in a number of ways. For more information, see the [Comparing display configuration and zones section](#) of the [Displaying parts in assemblies](#) Help topic.

## Updating the assembly structure

When working with zones, assembly components that are considered to be inside or outside a zone is accurate as long as the assembly structure is up to date. The assembly structure can become out of date when multiple people are working on different subassemblies, and you then open a higher level assembly with some of those changed subassemblies hidden. For example, on the Open File dialog box, there are options that allow you to open an assembly using a display configuration, a zone, or with all components hidden.

You update the assembly structure using the Update Assembly Structure button on the Select Tools page. When you click the Update Assembly Structure button, the entire assembly structure is loaded into memory to ensure that the assembly structure is up to date.

### Note

Updating the assembly structure can take several minutes, depending on the size of the assembly.

**Activity: Working with zones****Overview**

The objective of this activity is to demonstrate how to use zones to make moving around in assemblies more efficient.

**Activity**

In this activity you will open an assembly with a zone, modify a zone and create a new zone.



**Activity: Working with zones**

**Overview**

This activity demonstrates the use of zones in a large assembly.

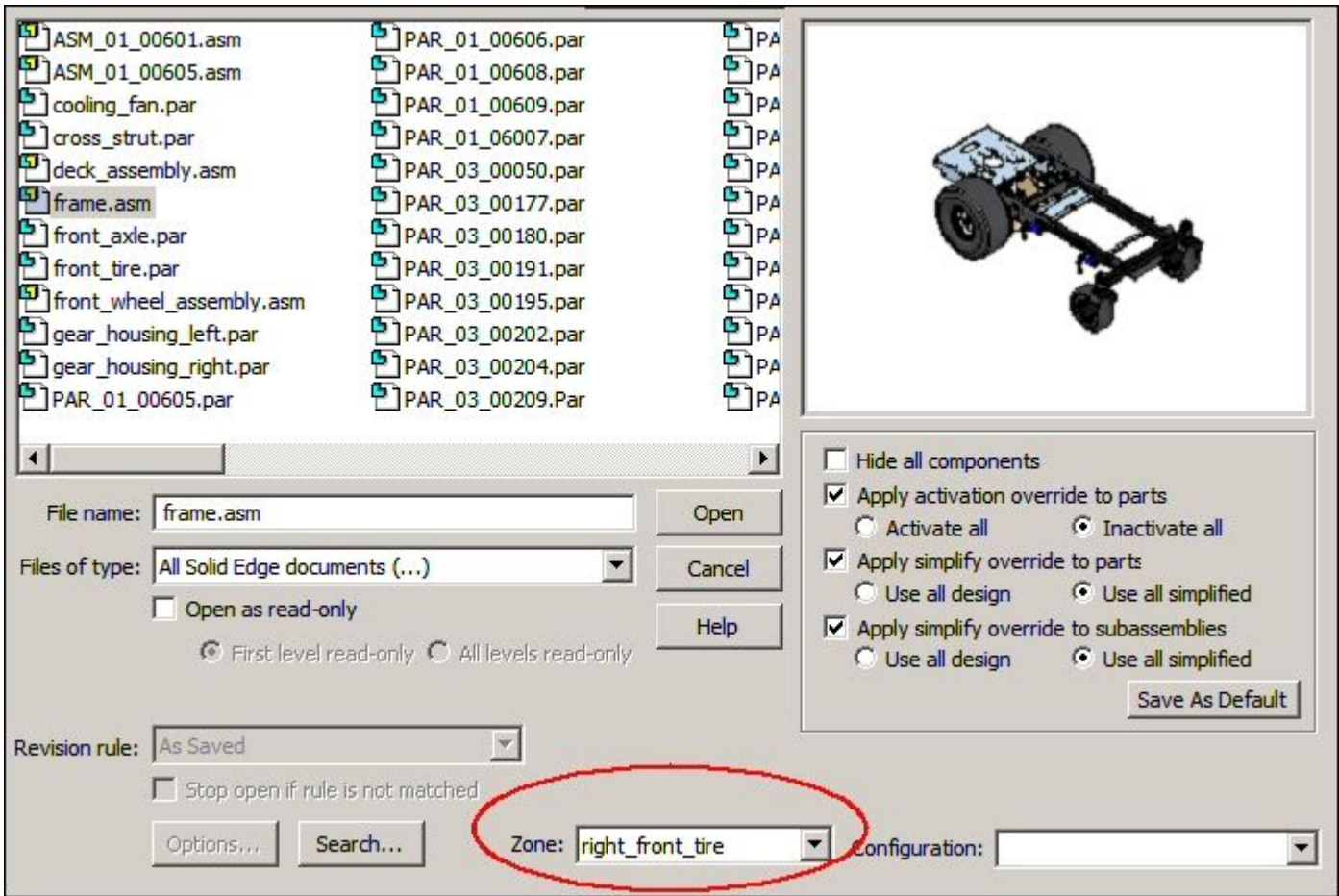
**Objectives**

You will create and manipulate zones in a large assembly.

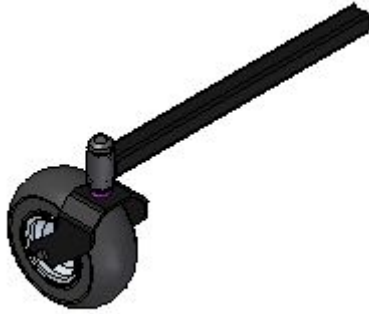
*Open an existing assembly using zones*

The assembly you will open has two zones defined. You will open the assembly using a zone, and then add a new zone.

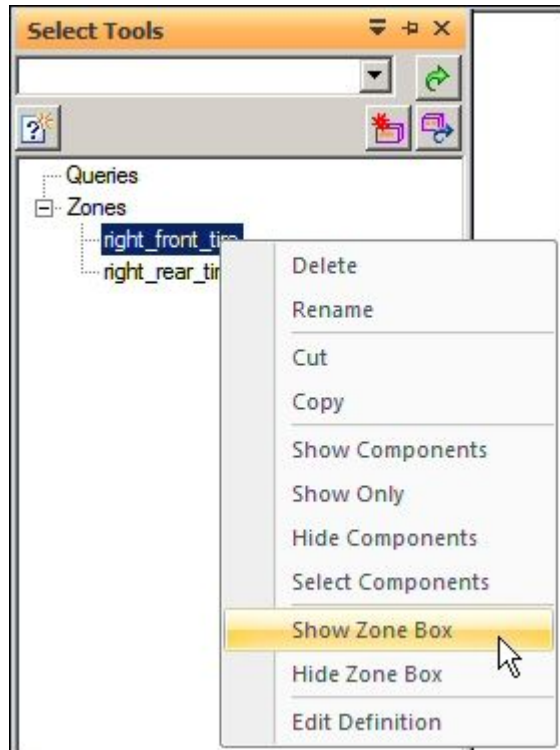
- From the Solid Edge start screen, click Open Existing Document. Browse for *frame.asm* in the folder where the activity files are located. In the Open File dialog box, set the zone to *right\_front\_tire* as shown.



- Observe the assembly showing the zone `right_front_tire`.





- In the PathFinder tab set, on the Select Tools tab, right-click the zone `right_front_tire`, and then click Show Zone Box.

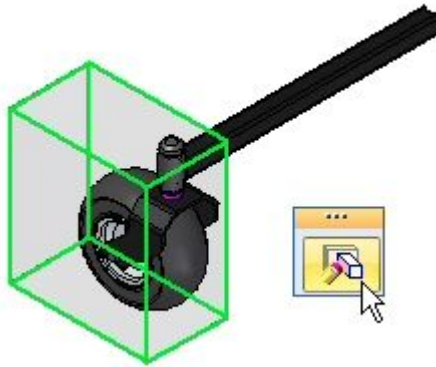


### Note

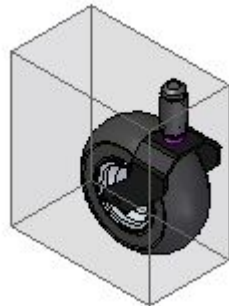
If PathFinder does not contain the tab you are looking for, such as Select Tools, Parts Library, or Alternate Assemblies, you can display it by doing either of the following:

- Choose View tab® Show group® Panes , and then select the tab name from the menu.
- In any of the other open docking windows, such as the Layers tab or the Sensors tab, click the Display Docking Window Menu button , and then select the tab name.

- ▶ Select the zone box and then click Edit Definition as shown.



- ▶ On the command bar, change the Zone Filter from Overlapping to Inside, and then click Show Components. Observe the results. The parts that are not fully contained in the zone are not shown.

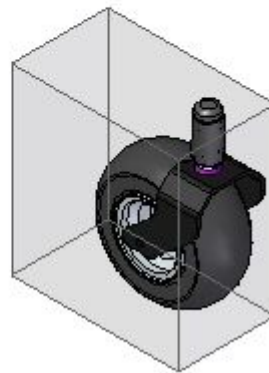
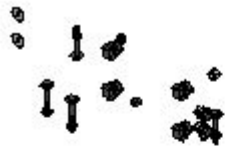
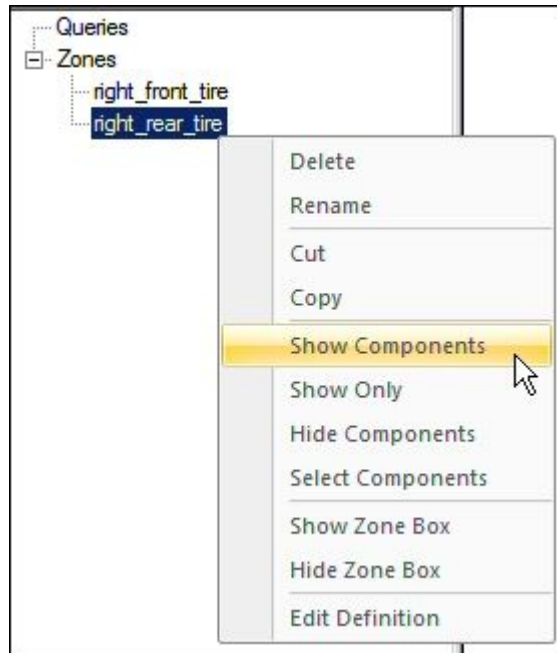


- ▶ Click Finish to complete the editing of the zone.

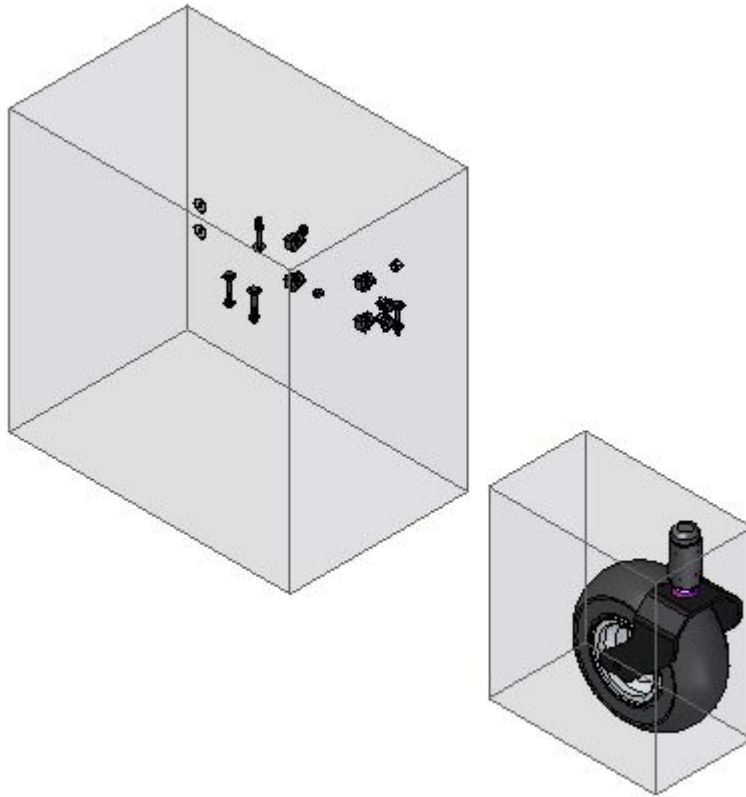
### *Modifying the extent of a zone*

The extent of an existing zone will be modified.

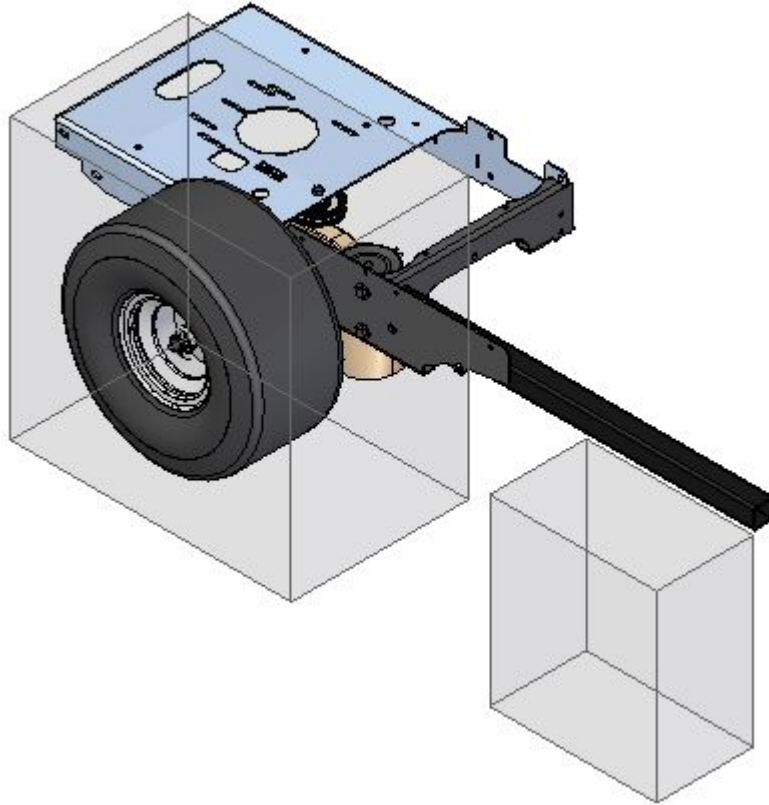
- ▶ On the Select Tools tab, right-click the zone `right_rear_tire` and then click Show Components.




- On the Select Tools tab, right-click the zone `right_rear_tire` and then click Show Zone Box.

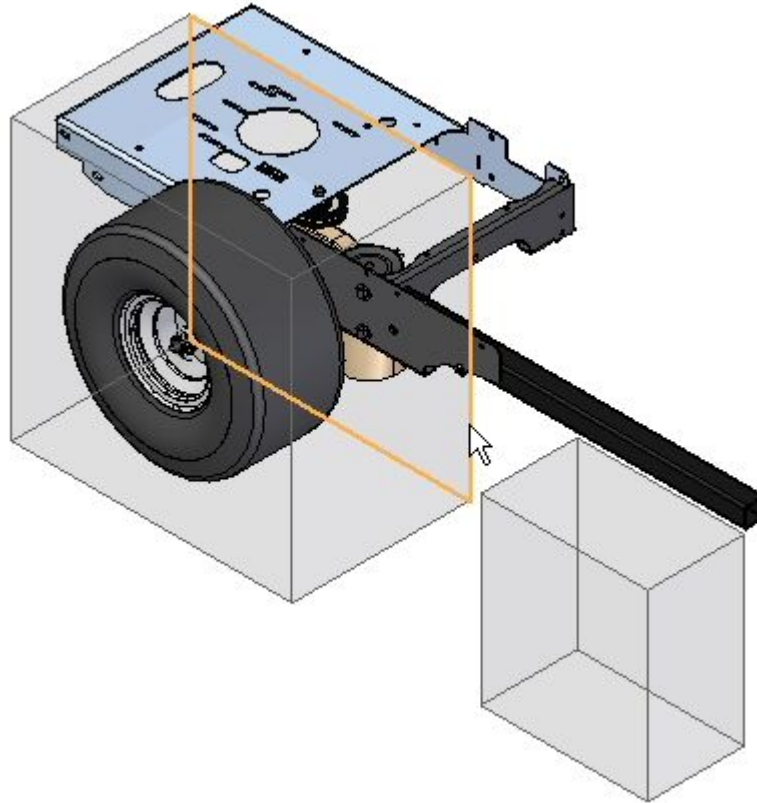


- ▶ Edit the definition of the zone `right_rear_tire` just as you did for the `right_front_tire` zone. Set the Zone Filter to Overlapping and click Show Components.



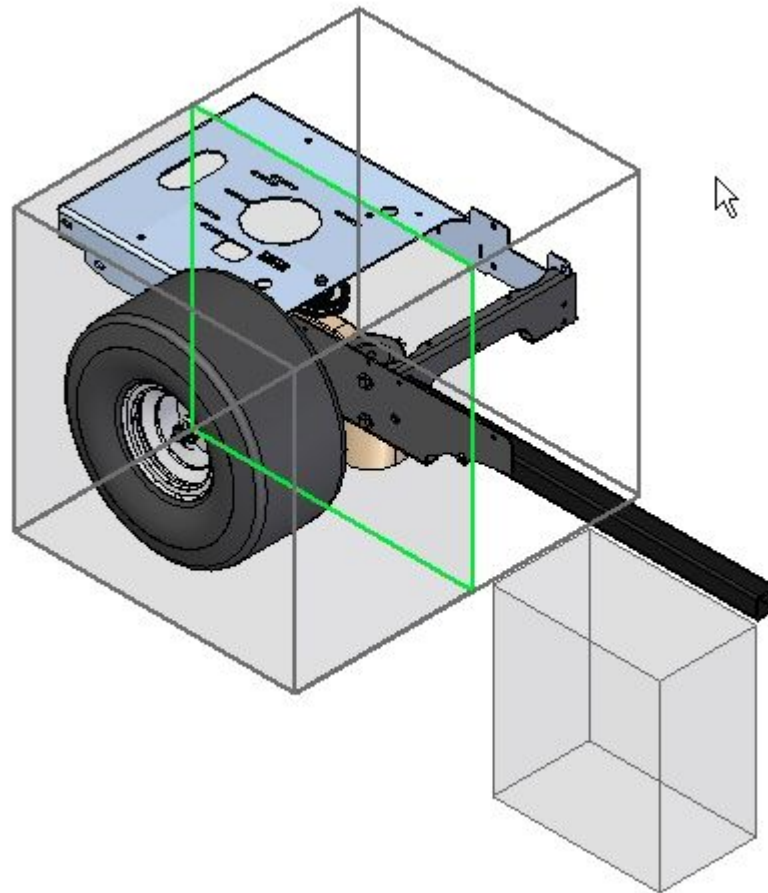
- ▶ On the Edit Definition command bar, click Modify Size Step .

- Select the face shown.





- Drag the face to the approximate position shown.



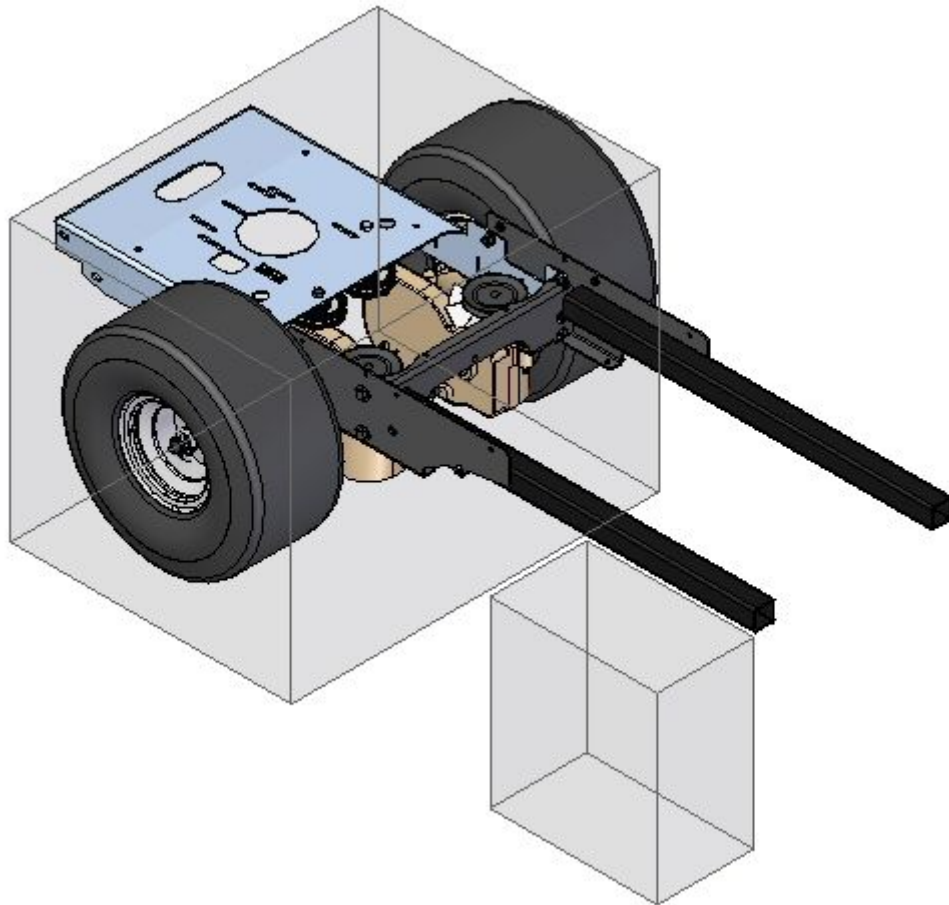
**Note**

To precisely position the face, use a keypoint on some existing geometry. The part has to be active for the keypoint to be available to use.

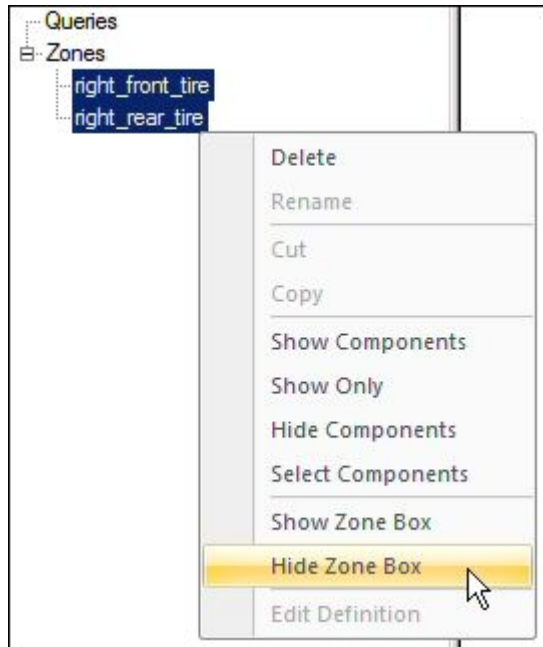
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**Lesson 5**     *Working with large assemblies*

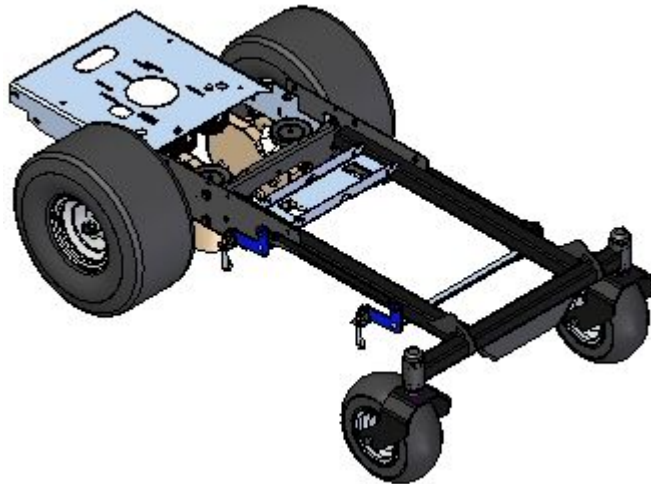
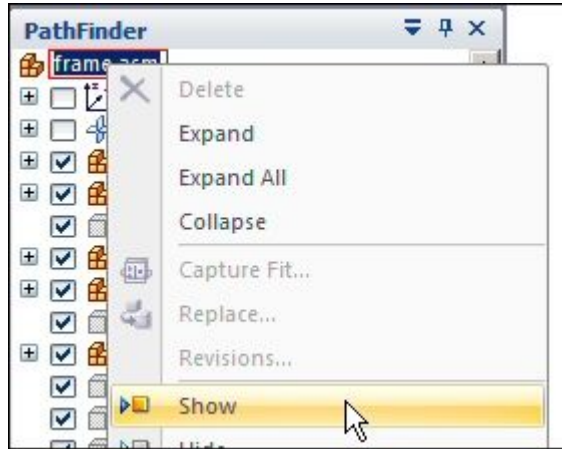
- ▶ Click Next.
- ▶ Set the zone filter to Overlapping and then click Show Components. Click Finish.



- ▶ On the Select Tools tab, select both zones. Right-click and select Hide Zone Box.



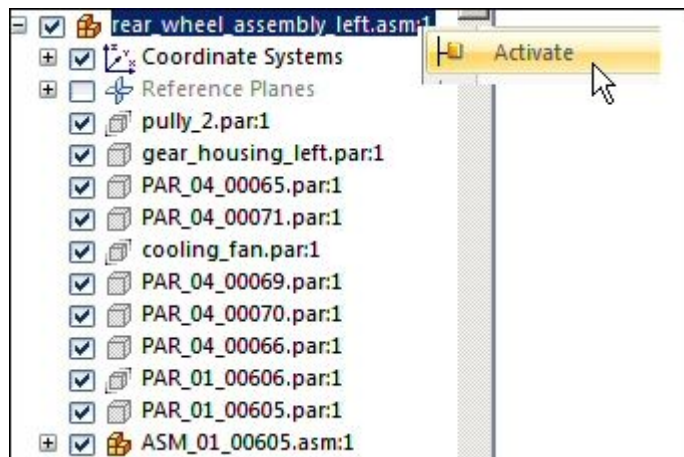
- In Assembly PathFinder, right-click frame.asm, and then click Show.



*Create a new zone*


Create a new zone for the left rear tire assembly.

- ▶ In PathFinder, right-click the subassembly *rear\_wheel\_assembly\_left.asm*, and then click Activate.

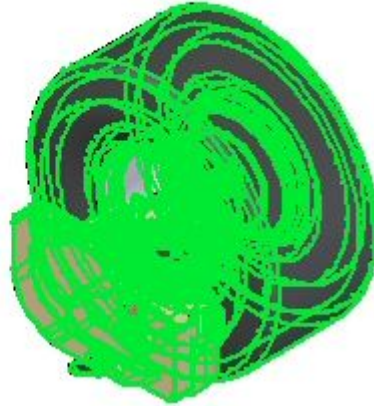


- ▶ In Pathfinder, right-click the subassembly *rear\_wheel\_assembly\_left.asm*, and then Show Only.

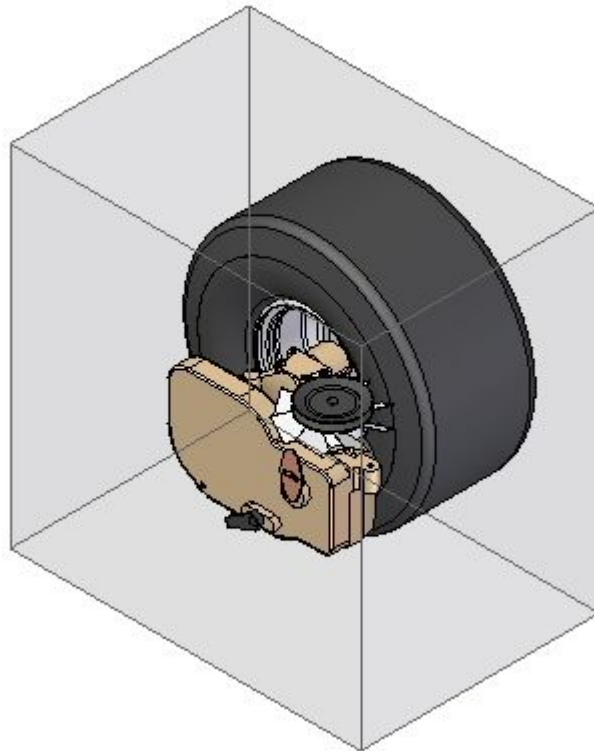


- ▶ On the Select Tools tab, click the Create Zone command .

- ▶ In the Select Parts Step, select *rear\_wheel\_assembly\_left.asm* in Pathfinder, and then click Accept.



- ▶ Click Next. Name the zone *rear\_wheel\_assembly*, and then click Finish. The new zone is created.



- ▶ Save and close the assembly. This completes the activity.

*Summary*

In this activity you learned how to use zones to narrow down a specific volume of an assembly to focus your work on.

**Lesson review**

Answer the following questions:

1. What is a zone?
2. Where are the zone commands located?
3. Can the extent of a zone be modified?
4. What options determine which components are displayed within the volume of a zone?
5. Is there an option to direct an assembly to be open using a zone?
6. Can zones be used to display components using the display configuration dialog box?



## **Answers**

1. What is a zone?

A zone defines a set of components in an assembly based on the volume of space the parts occupy.

2. Where are the zone commands located?

Zones can be created and maintained on the Select Tools tab in Pathfinder.

3. Can the extent of a zone be modified?

The extent of a zone can be modified by moving one of the rectangular faces. Component key points can be used to position the zone face.

4. What options determine which components are displayed within the volume of a zone?

Two options exist for displaying components that reside within the volume of a zone. These are:

- Inside, which displays only those components 100% contained within the volume defined by the zone.
- Overlapping, which displays all the components that have any portion contained within the volume defined by the zone.

5. Is there an option to direct an assembly to be open using a zone?

In when opening an existing assembly, there is an option which specifies which existing zone will control the display of components after the assembly is opened.

6. Can zones be used to display components using the display configuration dialog box?

Zones can be chosen from the display configuration dialog box.

**Lesson summary**

In this lesson you learned how to use zones to narrow down a specific volume of an assembly to focus your work on.

## Using display configurations

You can save display configurations of an assembly and exploded views of an assembly. When you save a display configuration with the Display Configurations command, the current display status is stored so you can use it later.

### Note

Display configurations store both the show/hide and the simplified/designed status of the parts in the assembly.

For example, if you hide several parts in an assembly, you can save the display configuration to a name you define. Later, if you want to quickly hide the same parts, you can apply the display configuration using the Configuration drop list on the Home tab in the Configurations group. You can also save, apply, edit, and delete display configurations using the Display Configurations dialog box.

You can also apply a saved display configuration when doing the following:

- Opening an assembly document
- Placing a subassembly in another assembly
- Creating a drawing of an assembly
- Creating a drawing of an exploded assembly

Display configurations are stored in a document that has the same name as the assembly document, but with a (.CFG) extension. The configuration document is stored in the same folder as the assembly document.

### Note

To support concurrent design, the configuration file allows multiple users to simultaneously add, delete, and edit display configurations.

## Using display configurations effectively

Both assembly configurations and exploded view configurations are saved to the same configuration file. To use display configurations effectively, your company should define a naming convention so all users can easily distinguish between types of configurations.

### Note

You should avoid using special characters in configuration names. For example, special characters such as \ / : ! are not allowed.

You can apply an assembly configuration to an exploded view to control the show/hide status of the parts and subassemblies. You cannot apply an exploded view configuration to a regular assembly window. For this reason, configuration names of exploded views are not displayed in the Configuration list on the Home tab when you are working in a regular assembly window.

### **Using a display configuration to open a document**

When working with large assemblies, you can open the document faster if you use a display configuration where some parts and subassemblies were hidden or where simplified versions of the parts were defined. To apply a display configuration when you open an assembly, use the Configuration list on the Open File dialog box.

### **Using a display configuration to place a subassembly**

You can select an assembly configuration name in the Use Configurations dialog box when placing a subassembly into an assembly. Subassemblies will place faster and can be easier to visualize if you apply a configuration where parts not necessary for placement are hidden. Exploded view configuration names are not displayed in the configuration list on the Use Configurations dialog box.

To use a display configuration when placing a subassembly, you must first set the Use Configurations command on the Parts Library shortcut menu. When you set this option and then drag and drop a subassembly into the assembly, the Use Configuration dialog box is displayed so you can select the configuration name you want to use.

### **Using display configurations in the Draft environment**

You can use both assembly and exploded view configurations when creating drawings in the Draft environment. When you click the Drawing View Wizard command and select an assembly document from the Select Model dialog box, you can also select an assembly configuration or an exploded view configuration. When you select either configuration type, the drawing view will be placed with the parts displayed or hidden as they were when the configuration was saved.

You can also use display configurations to control the display of weld beads and material addition features in an assembly drawing view. For example, you can hide the display of weld beads and material addition features, then save a display configuration. You can then use the display configuration to place a drawing view of the assembly with these features hidden.

### **Using display configurations in alternate assemblies**

Display configurations are available when working with alternate assemblies. The behavior of display configurations based on whether you specify that the alternate assembly is a family of assemblies or an alternate position assembly.

#### **Family of assemblies**

For a family of assemblies, a display configuration is member-specific. In other words, the family of assembly member which is active when you create the display configuration is the only member in which you can use the display configuration later. The Configuration list on the Home tab filters the available display configurations automatically.

#### **Alternate position assemblies**

For alternate position assemblies, display configurations are not member-specific. In other words, you can use any display configuration for any active member. The Configuration list on the Home tab displays all the display configurations.

## **Zones and display configurations**

The functionality of zones and display configurations differ in a number of ways. For more information, see the [Comparing display configuration and zones section](#) of the [Displaying parts in assemblies](#) Help topic.

## Displaying parts in assemblies

When you work with complex or unfamiliar assemblies, it is often useful to change the display of the parts and subassemblies. Solid Edge makes it easy to hide and display the components in your assembly so you can work more efficiently.

### Displaying and hiding assembly components

The display commands in the Assembly environment are especially useful when you work with large assemblies. For example, you may want to hide most of the parts in an assembly to make it easier to place a new part. You can select the parts to display using the PathFinder tab, then use the Show Only command to hide all other parts.

For more information, see the [Working with large assemblies efficiently](#) Help topic.

#### Note

When you hide parts or subassemblies in the active assembly, the symbols in PathFinder change to indicate that the components are hidden.

### Saving and applying display configurations

After you hide parts or subassemblies, you can save the display configuration with the [Display Configuration command](#). You can quickly apply a saved configuration to a view using the Configuration list on the Home tab in the Configurations group. Avoid using special characters in configuration names. For example, special characters such as \ / : ! are not allowed.

For more information, see the [Using display configurations](#) Help topic.

### Using zones

You can also use zones to show, hide, and select assembly components, such as parts, assemblies, layouts, weld beads, and reference planes.

For more information, see the [Using zones in assemblies](#) Help topic.

### Comparing display configurations and zones

Both display configurations and zones are useful tools to manage component display in assemblies. The following information compares and contrasts display configurations and zones.

#### Display configurations

A display configuration allows you to control the display status of assembly components regardless of the components physical location in the design space.

If you add components to the assembly, they are not added to an existing display configuration. You must apply the configuration, display the components you want to add to the configuration, then resave the configuration. When working in large, nested assemblies, shared by multiple users, it can sometimes be difficult to keep track of efficiently.

When working with a family of assemblies, you can create member-specific display configurations.

### Zones

A zone allows you to show, hide, and select assembly components based on a rectangular volume of space you define when you create the zone. As you add new components to the assembly, they are automatically added to any existing zones they fall within.

You can edit the physical size of a zone later to include or exclude components.

When working with a family of assemblies, the zones functionality is not available.

### Part display and system performance

When you work with large assemblies, you can improve system performance by hiding parts and subassemblies you are not using. For example, display commands such as Zoom, Zoom Area, and Pan work faster if you hide parts. Assembly documents also open faster if parts have been hidden.

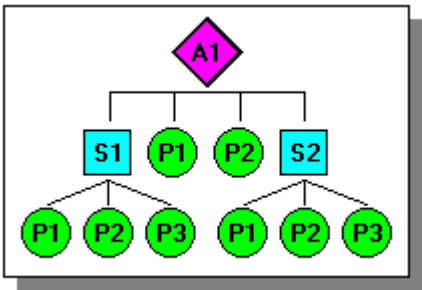
#### Note

You can improve performance by using the Unload Hidden Parts command to unload the hidden parts from memory.

### Displaying and hiding parts during editing

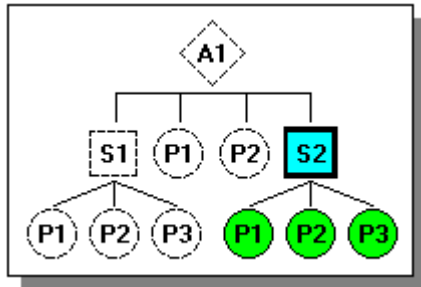
When you in-place activate a part or subassembly to make changes, the remaining parts and subassemblies are still displayed, but they change color to make it easier to focus on the part you are editing. You can use the Hide Previous Level command on the View tab to hide other parts and subassemblies while you edit the part.

For example, suppose you are working in assembly A1 and need to make changes in subassembly S2.

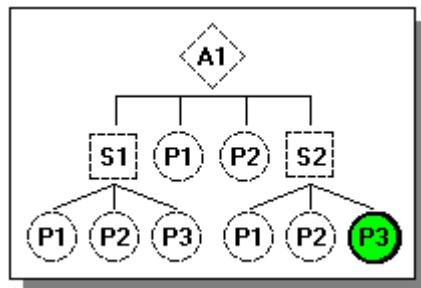


You can select subassembly S2 in PathFinder, then click the Edit command on the shortcut menu to in-place activate subassembly S2. In subassembly S2, you can still see all the parts and subassemblies, but the parts and subassemblies that are not in subassembly S2 are dimmed.

You can use the Hide Previous Level command on the View tab to hide the other parts and subassemblies as shown below.



If you then in-place activate part P3 in subassembly S2, you also can hide the remaining parts in subassembly S2 using the Hide Previous Level command. The result would be similar to the following illustration.



To redisplay the hidden parts, select the Hide Previous Level command again.



## **Displaying and hiding specific geometry for assembly components**

You can also control the display of a selected part or subassembly's reference planes, sketches, coordinate systems, reference axes, construction surfaces, and construction curves. You can use the Show/Hide Component commands on the shortcut menu to display and hide these element types for one or more selected parts. For example, you can select a part, then on the shortcut menu, point to Show/Hide Component, then click Construction Surfaces to display or hide all the construction surfaces for a part. A checkmark is displayed adjacent to the command name when a component is displayed. You can also display an assembly component based on how it was saved using the As Last Saved command on the Show/Hide Component shortcut menu.

Similarly, when nothing is selected you can also hide the reference planes, sketches, reference axes, construction surfaces, or construction curves for all the parts in an assembly using the commands on the Hide All shortcut menu. For example, to hide the sketches for all parts in the assembly, on the shortcut menu point to Hide All, then click Sketches.

### **Note**

When you display or hide reference planes, sketches, coordinate systems, reference axes, construction surfaces, and construction curves for a part or subassembly, the current display status for these element types is not captured in a saved display configuration.

You can also control the display of part geometry while you place a part or edit its position in the assembly. The Construction Display option on the Assemble command bar allows you to display and hide reference planes, coordinate systems, and so forth for the placement part or the target part.



### **Configuration Manager command**

Configuration Manager is used to manage changes in configurations. With the Configuration Manager command, when parts are added to or deleted from subassemblies, the configurations in the top level assembly can easily find the changes in the subassemblies and allow the current configurations to incorporate these changes.

Configuration Manager is useful when using large assemblies, nested assemblies, or assemblies being shared by several designers.

For more information, see the [Using display configurations](#) and [Working with large assemblies efficiently](#) Help topics.

#### **Note**

Exploded configurations can only be edited in the ERA environment.

## **Configuration Manager dialog box**

Edits a display configuration in the active assembly and searches for changes in subassemblies.

### **Dialog Box Options**

#### **Configuration file**

Displays the name of the current configuration file, which contains configuration names and display information. You can type a name or use the Browse button to find another configuration file.

#### **Find Previous New Component**

Moves the selection up the list of components and stops at a new component not currently listed in the display configuration. Use the check box to include or exclude the component into the display configuration.

#### **Find Next New Component**

Moves the selection down the list of components and stops at a new component not currently listed in the display configuration. Use the check box to include or exclude the component into the display configuration.

#### **Clean Up Deleted Components**

Finds Components that were deleted and removes them from the display configuration.

#### **Browse**

Accesses a dialog box that allows you to search for an existing configuration.

#### **Save**

Saves the configuration to the current configuration file.

#### **Cancel**

Cancels changes made before saving.

**Activity: Using configurations in assembly****Overview**

The objective of this activity show how configurations can be used to make working in an assembly more efficient.

**Activity**

In this activity you will open an assembly with predefined configurations. You will use these to change the display and assembly parameters of the assembly.

**Activity: Using configurations in assembly**

**Overview**

This activity demonstrates the use of configurations in a large assembly.

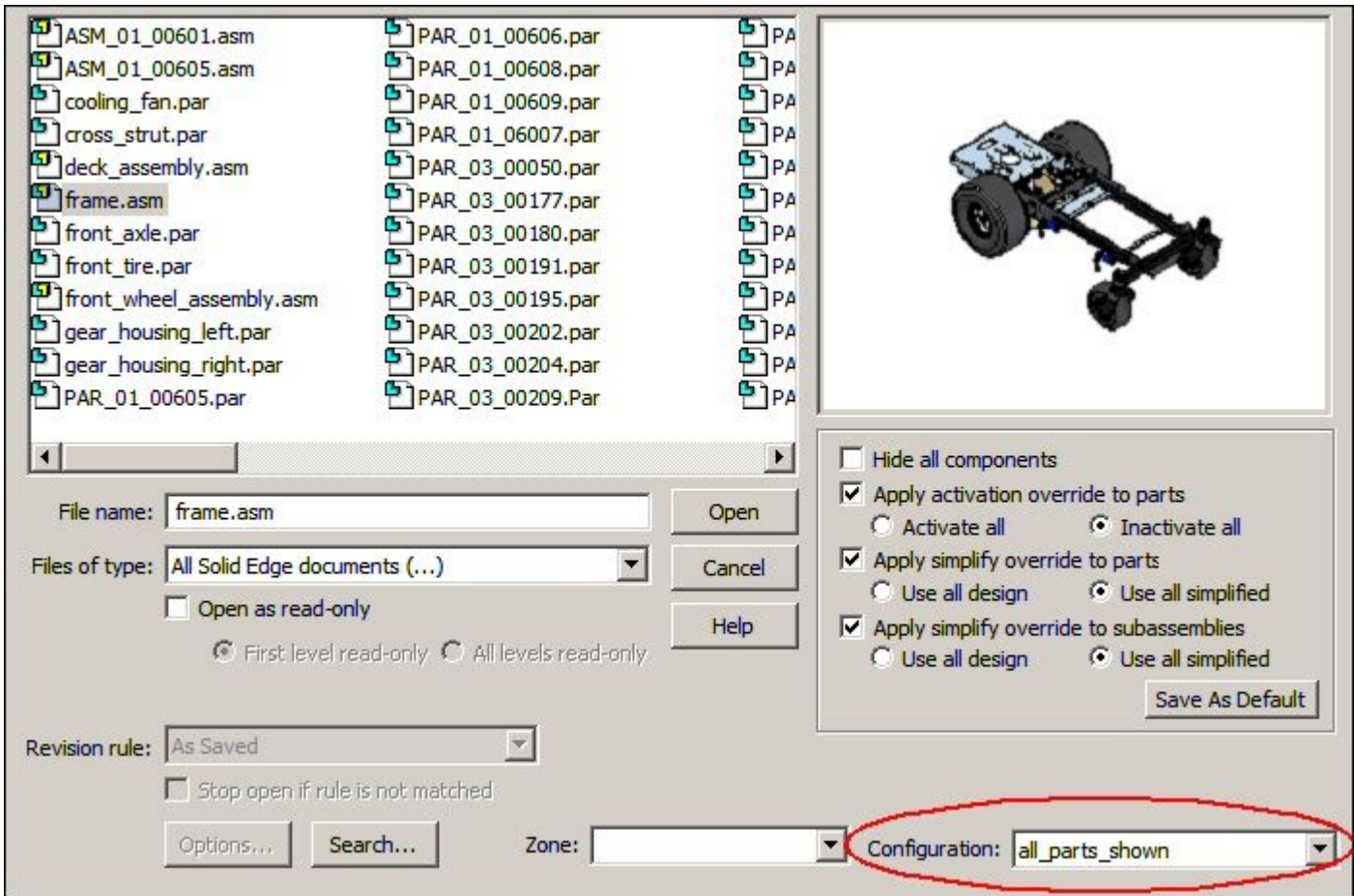
**Objectives**

You will create and manipulate configurations in a large assembly.

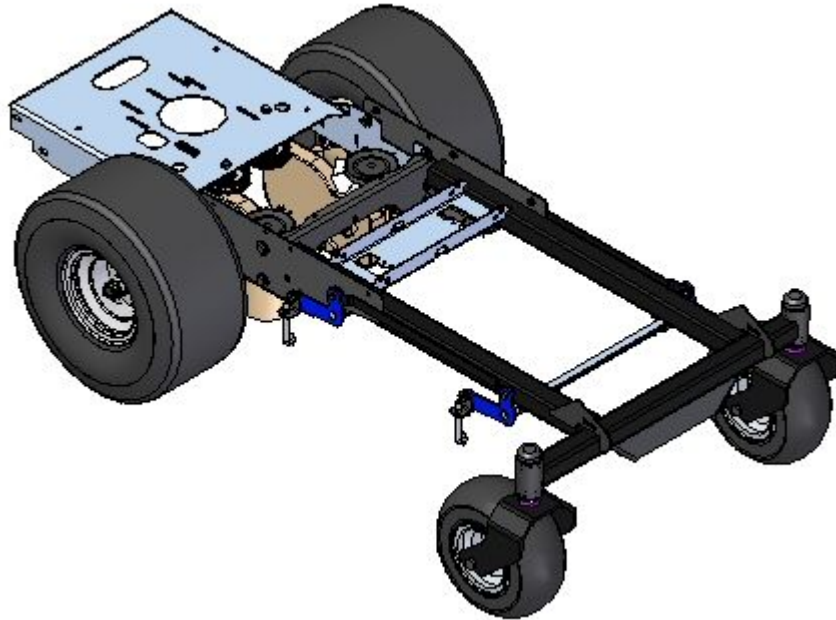
*Open an existing assembly using configurations*

The assembly has several display configurations defined for it. You will open the assembly using one configuration, and then add a new configuration.

- ▶ On the Solid Edge start screen, click Open Existing Document. Browse for *frame.asm* in the folder where the activity files are located. In the Open File dialog box, from the Configuration list, select *all\_parts\_shown*.



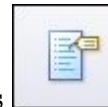
- ▶ Observe in Assembly PathFinder that all the subassemblies and parts are displayed. Not all the parts are active.



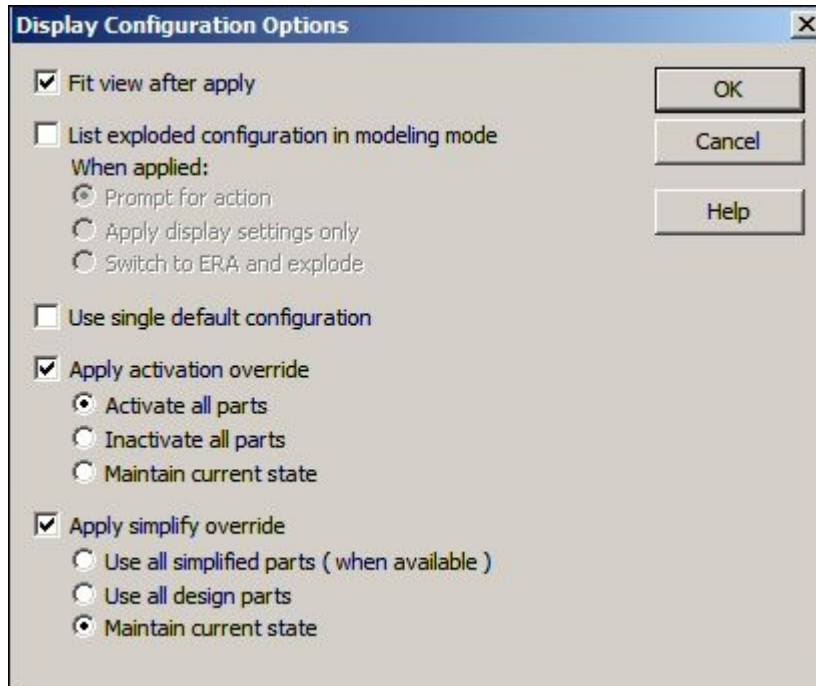
### *Changing configurations*


There are several existing configurations in the assembly. You will change the configuration and observe the result.

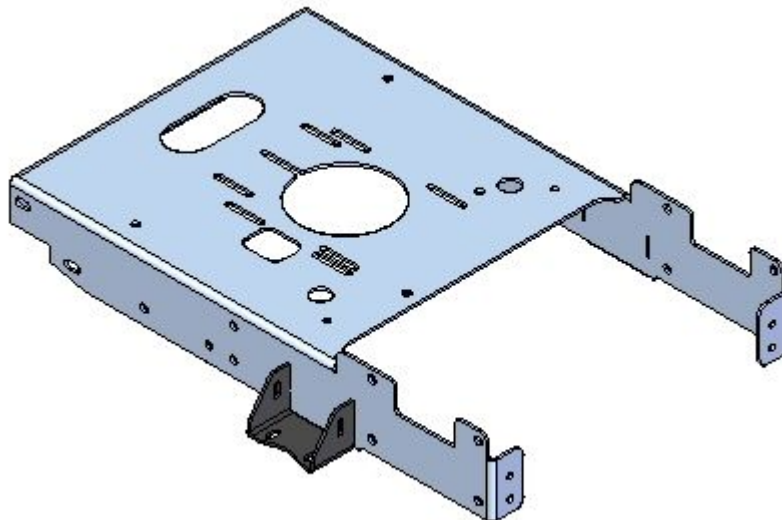
- ▶ Click Home tab® Configurations group® Configuration Options



- ▶ In the Display Configuration Options dialog box, select Apply activation override and Activate all parts. This activates all the parts when selecting a configuration.



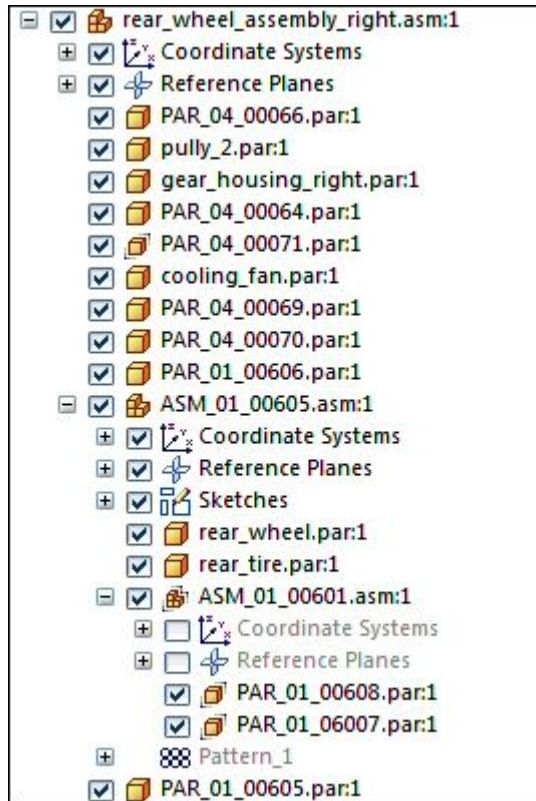
- ▶ Click Home tab@ Configurations group@ Display Configuration ..
- ▶ In the Display Configurations dialog box, select deck\_assembly, click Apply, and then click Close. Observe PathFinder and notice just the assembly making up the deck is shown. Because of the activation override, all the parts in the assembly are active.




*Change the configuration*

Add a few more parts to the current configuration.

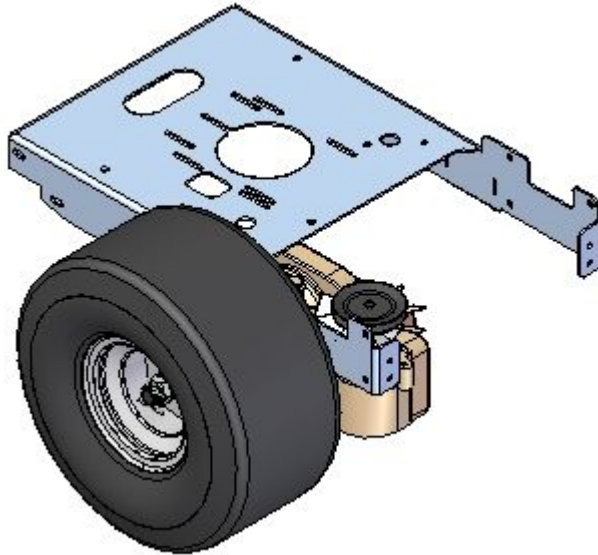
- ▶ In PathFinder, expand the subassembly *rear\_wheel\_assembly\_right.asm* and all the subassemblies contained within it. Display all the parts in the subassembly.



- ▶ Click Home tab@ Configurations group@ Display Configuration .



- In the Display Configurations dialog box, select `deck_assembly`, click Update, and then click Close. The configuration now has the display of the subassembly `rear_wheel_assembly_right.asm`.

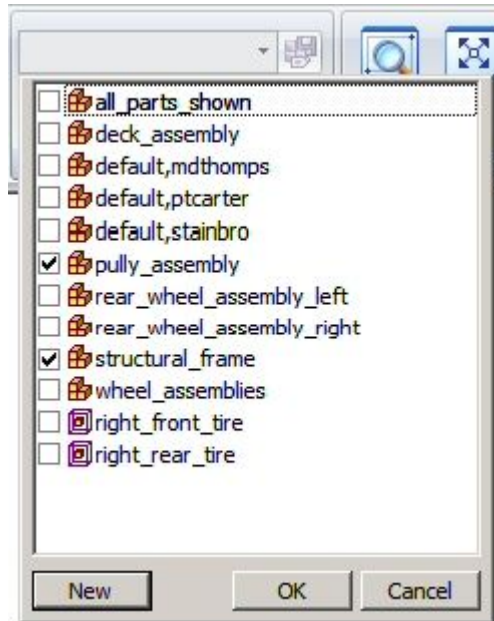


**Note**

The update can quickly be applied using the Save Display Configuration button.



- ▶ On the Home tab in the Configurations group, click the arrow beside the Current Configuration list to see a menu of all the available display configurations. Select the two shown.



### Note

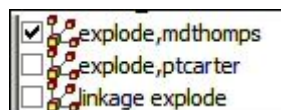
Using the Current Configuration list, you can combine display configurations, and you can select zones.

- ▶ Practice turning on and off configurations using this menu. Also, display some parts in PathFinder, and update a configuration to include those parts.

### *Configurations in the Explode-Render-Animate environment.*

Configurations used to explode a part are not visible until you display the Explode-Render-Animate environment.

- ▶ Choose the Tools tab® Environs group® ERA command.
- ▶ Click Home tab® Configurations group® Current Configuration list, notice that now the explode configurations are available. Select the one shown.



### Note

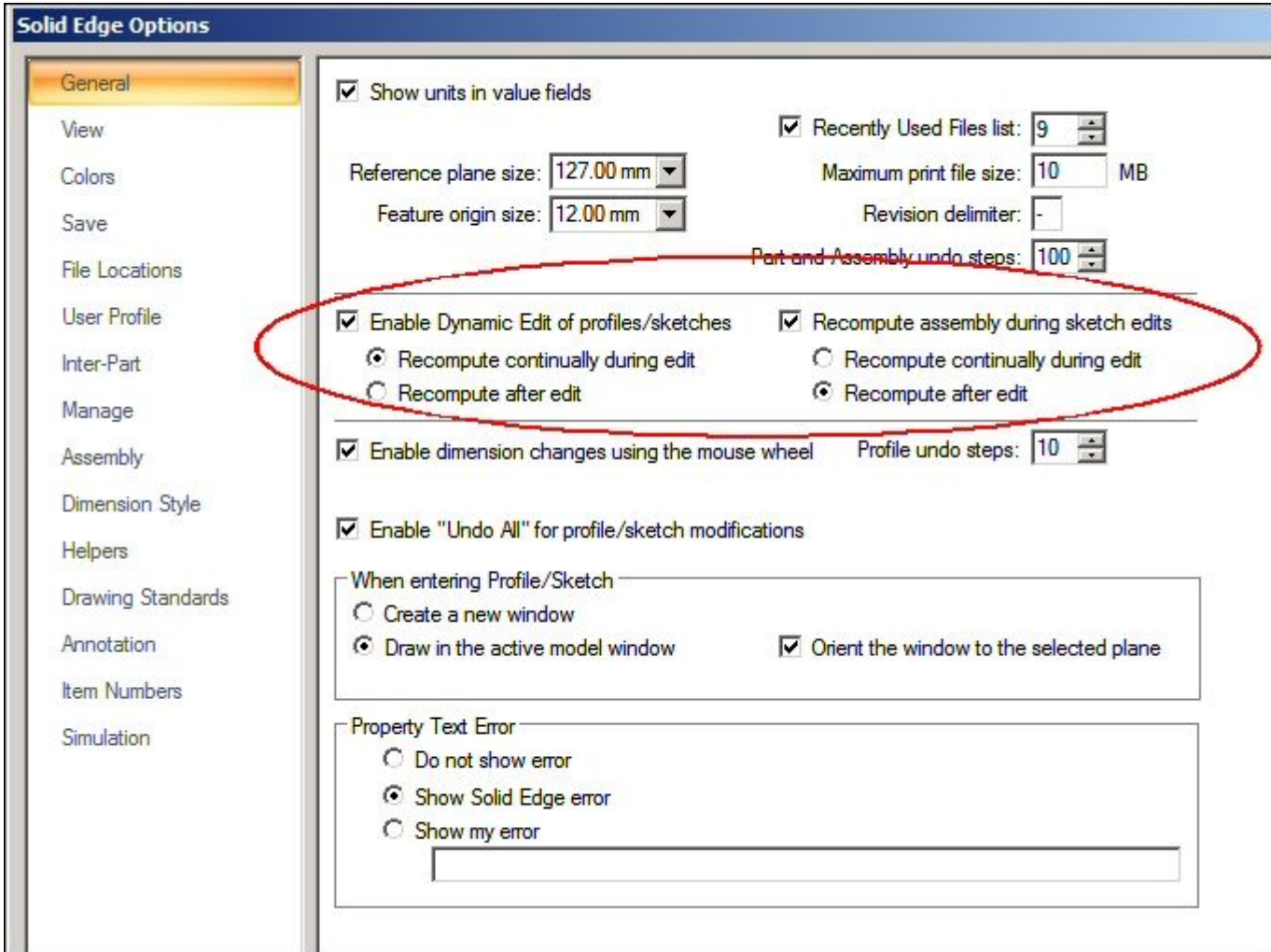
Notice the exploded view stored in the configuration is displayed.

- ▶ Click Close ERA to exit the ERA environment.

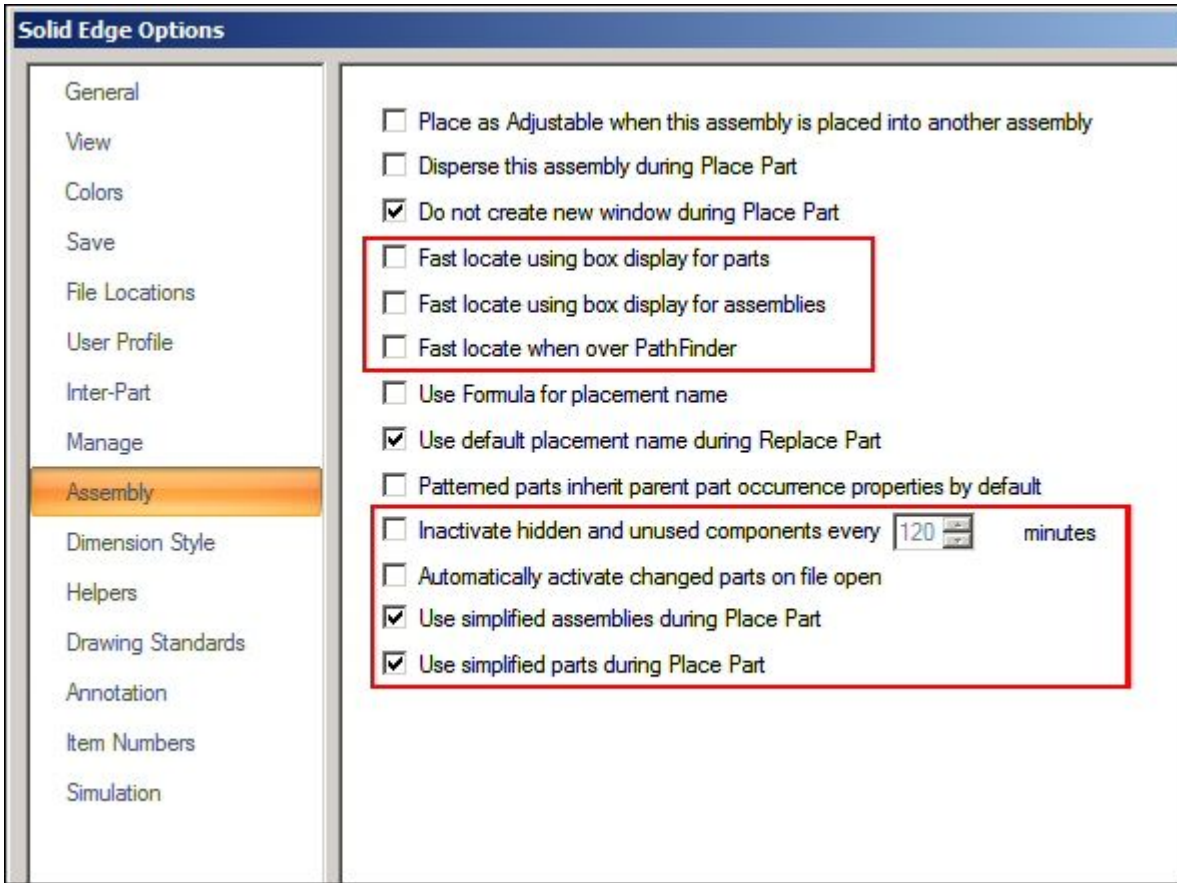
More tips for working with large assemblies

There are more tools available to speed up working in large assemblies. Some of these are there to enhance performance. Others are there to prevent you from loading in more information that you really need to perform certain tasks. Here we will look at these tools.

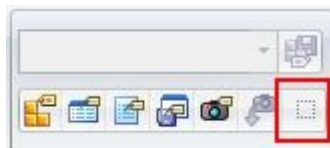
- Click the Application button, and then click Solid Edge Options. On the General tab, review the circled items, which can be used to improve performance.



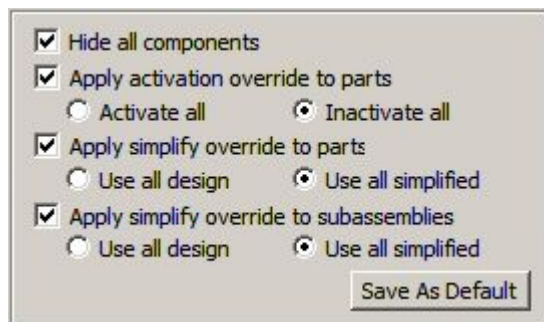
- On the Assembly tab, the boxed items are options that can speed up performance.



- On the Home tab, in the Configurations group, notice the Unload command. Use this command to inactivate any hidden parts.



- Save and close the assembly.
- On the Solid Edge startup screen, on the Open File menu, there are several options available for speeding up the opening of large assemblies. These can be set as the default so that this happens each time an assembly is opened.



- This completes the activity.

*Summary*

In this activity you learned the following:

- How to use configurations to narrow down a specific volume of an assembly to focus your work on.
- Options available to enhance performance when opening large assemblies, or working in them.

**Lesson review**

Answer the following questions:

1. What is a display configuration in an assembly?
2. Where are display configurations stored?
3. Are new parts added to an assembly automatically added to a display configuration?
4. If a subassembly has parts removed from it, how can a display configuration be updated to reflect these changes?
5. When creating an exploded view to place on a drawing sheet, where is the information about how to display the exploded view stored?

## **Answers**

1. What is a display configuration in an assembly?

A display configuration allows you to control the display status of assembly components regardless of the components physical location in the design space.

2. Where are display configurations stored?

Display configurations are stored in a .cfg file external to the assembly.

3. Are new parts added to an assembly automatically added to a display configuration?

If you add components to the assembly, they are not added to an existing display configuration. You must apply the configuration, display the components you want to add to the configuration, then resave the configuration. When working in large, nested assemblies, shared by multiple users, it can sometimes be difficult to keep track of efficiently.

4. If a subassembly has parts removed from it, how can a display configuration be updated to reflect these changes?

Configuration manager contains tools that remove deleted parts in subassemblies from a display configuration.

5. When creating an exploded view to place on a drawing sheet, where is the information about how to display the exploded view stored?

Exploded view display parameters are saved in a display configuration. These displays can be viewed in the explode, render and animate environment.

**Lesson summary**

In this lesson you learned the following:

- How to use configurations to narrow down a specific volume of an assembly to focus your work on.
- Options available to enhance performance when opening large assemblies, or working in them.



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Lesson

# 6 *Inspecting assemblies*

## **Inspecting assemblies**

## **Introduction**

Solid Edge supports a variety of tools to inspect and analyze assemblies. Covered here are:

- Motion
- Collision detection
- Sensors
- Mass properties

## **Objectives**

After completing this lesson, you should be able to:

- Assign material densities to different parts and calculate a center of mass and weight of an assembly.
- Use motion to analyze and detect collisions in unconstrained parts.
- Establish sensors to warn if an unconstrained part is violating a set parameter as it moves.

## Physical properties of parts and assemblies

You can calculate the following physical properties of parts and assemblies:

- Volume
- Mass
- Center of volume
- Center of mass
- Surface area
- Orientation of the principal axis
- Mass moments of inertia
- Radii of gyration

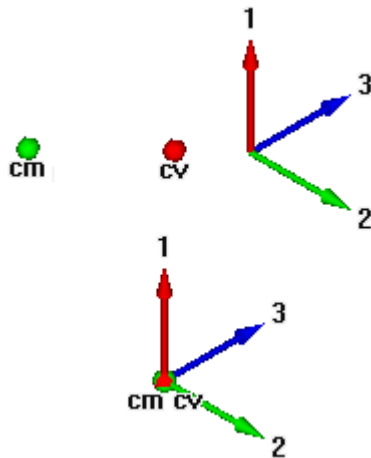
### Note

You can calculate surface area only for parts.

Center of mass, center of volume, mass moment of inertia, and principal axis coordinates are output with respect to the global coordinate system. Principal moments of inertia and radii of gyration are output with respect to the principal axes.

## Physical property symbols

The Physical Properties command places symbols on the part or assembly to show the center of mass location, center of volume location, and principal axis orientation. You can display and hide the symbols individually, or display them all at once using the Physical Properties dialog box.



## Calculating and storing physical properties

When you calculate physical properties for a part, the Physical Properties command calculates and stores the properties for the individual part. In the Assembly

environment, the Physical Properties command calculates and stores the properties for the entire assembly.

If you want to calculate the properties for a portion of an assembly, you can select the individual parts in the assembly before selecting the command. The software places property symbols indicating the center of mass, center of volume, and principal directions for the selection set.

Physical properties for a selection set are not stored with the assembly. When you close the Physical Properties dialog box, the property data for the entire assembly is restored. To store physical properties of a selection set, click the Save As button before dismissing the Physical Properties dialog box.

### **Using coordinate systems**

You can calculate physical properties relative to a user-defined coordinate system. A list of all user-defined coordinate systems plus an entry for the Model Space coordinate system, which is the default, is available on the Physical Properties dialog box. Calculating physical properties relative to a user-defined coordinate system will affect all properties other than Mass, Volume, and Surface Area.

### **Defining the density**

Before calculating the physical properties for a part or the weld beads in a weldment assembly you must specify the density of the part or weld bead.

For a part, you can define the density on the Material tab of the Solid Edge Material Table dialog box, the Physical Properties dialog box, or the Variable Table. The Change button on the Physical Properties dialog box displays the Solid Edge Material Table dialog box so you can edit the material type and/or the density for the part.

For weldment assemblies, you can define the weld bead material density using the Weldment Assembly command, or the Variable Table.

When you have defined the density, click the Update button. You must provide a positive density value. If you do not specify a density, Solid Edge uses a density of zero and produces the error message: Density Must Be A Positive Numeric Value.

#### **Note**

You can create material-specific template files by defining the material and density in the files you use as templates.

In the Assembly environment, the software checks each part in the assembly to determine if a density was defined for each part in the Part or Sheet Metal environments. If you have not specified a density for a particular part, you can supply a density value for calculating the part's physical properties.

When you save an assembly, part, or sheet metal document, you can also save the density information with the document. The information can be used later to update the physical properties.

### **User-defined properties**

You can override physical properties calculated by the software by setting the User Defined Properties option on the Physical Properties dialog box in the Part and Sheet Metal environments. For example, if you know the mass for a particular part,

you can supply the value, and it will be treated as computed data. However, if you then update values, the software will recalculate and override any user-defined values. The physical properties will not update using a mix of user-defined and system calculated properties.

### Note

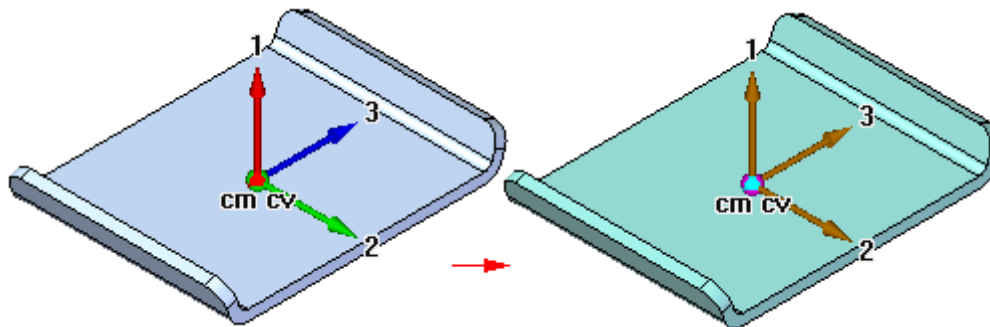
Density, Mass, and Volume cannot be zero.

## Failed parts or assemblies

Physical properties cannot be calculated for parts with features that have not recalculated correctly. In the Part and Sheet Metal environments, you can use the Error Assistant dialog box to determine which parts have features that have not recalculated correctly and why.

## Updating physical properties

If you change a part or assembly so that its physical properties change, the colors of the physical properties symbols change to show that the last calculated physical properties are out-of-date and should be updated. For example, if you change the material density for a part, the physical properties symbols can go out of date.



A part can also become out-of-date if you add, delete, or modify a feature. An assembly can become out-of-date if you add or delete a part from the assembly, or if you add or remove assembly features.

In the Part and Sheet Metal environments, you can set or clear the Update on File Save option to specify whether the physical properties are automatically updated when you save the document. You can also update the physical properties of a part or assembly by selecting the Update button on the Physical Properties dialog box. A message is displayed at the bottom of the dialog box that indicates whether the physical properties are up-to-date or out-of-date.

## Physical properties manager

In the Assembly environment, you can use the Physical Properties Manager command to view, edit, and manage the physical properties for all the parts in the active assembly. This can be useful because you can view and edit the physical properties for all the parts at once, rather opening each part document to view and edit its physical properties.

**Mapping physical properties for synchronization with Teamcenter**

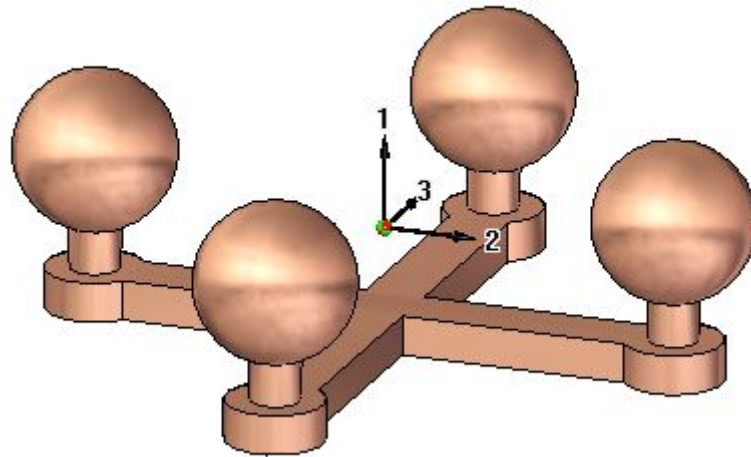
By using mapping definition files, physical properties such as density, mass, and volume can be stored in the Teamcenter database and displayed and modified in both Solid Edge and Teamcenter. This synchronization allows an attribute in one application to be updated automatically when a modification is made to the corresponding attribute in another application. Refer to the *Solid Edge Embedded Client Administrator's Guide*, for attribute mapping syntax and examples.



## **Activity: Computing mass properties of an assembly**

### **Overview**

This activity shows how to change materials, and how to compute mass properties and center of gravity of an assembly.



## **Activity: Computing mass properties of an assembly**

### **Overview**

When you complete this activity, you will be able to change material types in parts, and then compute the center of mass in an assembly.

### **Objectives**

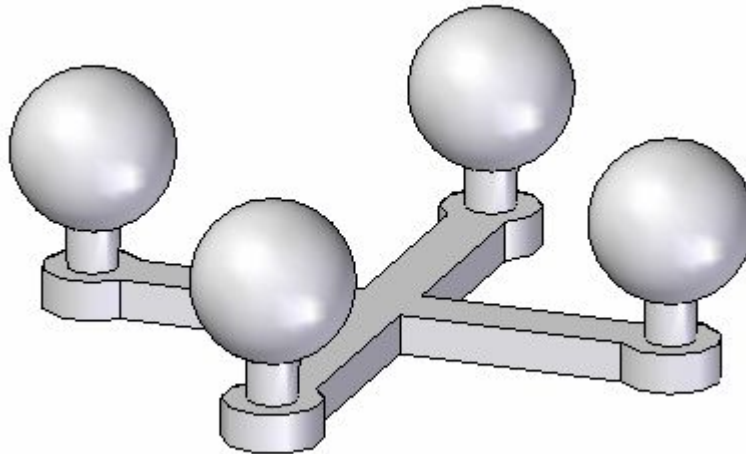
The activity consists of the following:

- Editing a part in the context of an assembly.
- Changing the materials for each part.
- Computing the mass properties for the assembly.

### **Activity**

**Open the assembly and change the material**

- Open and activate all the parts in the assembly *mass.asm* located in the folder where the activity files are located.



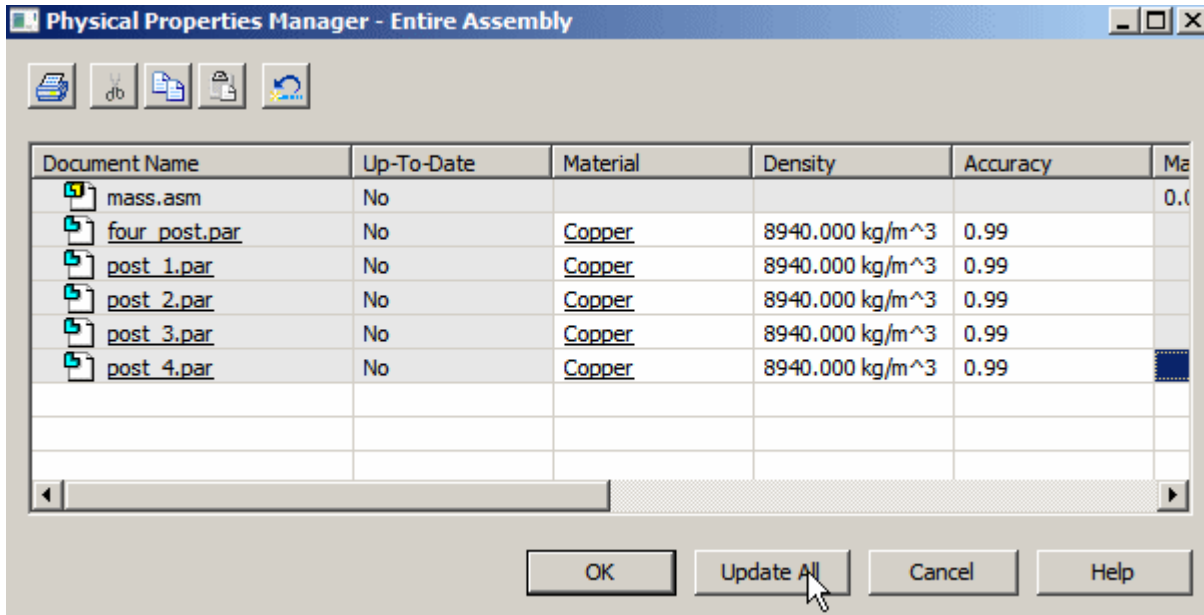
**Note**

The assembly consists of five parts: the base and the four posts with spheres. If the posts were identical, the assembly could have been made with just two parts, and the post placed four times. In this activity, the posts are different because the material and density will be altered for each post.

- Choose the Inspect tab@ Physical Properties group@ Properties Manager

command .

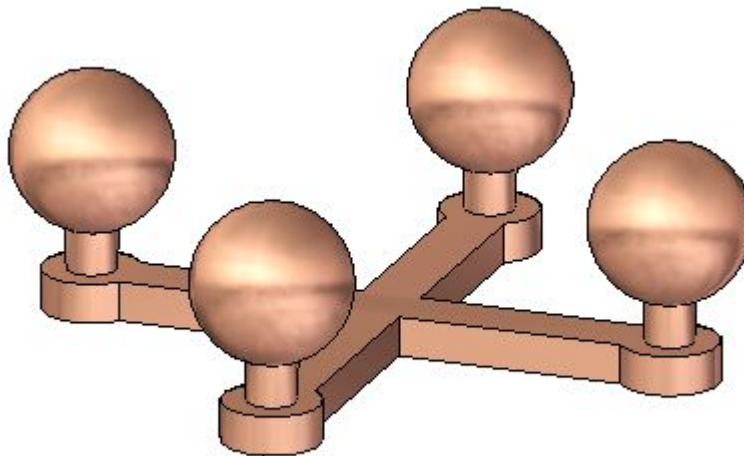
- ▶ In the dialog box, set the material for all the parts to be copper, and then click Update All.



**Note**

Another way to change the material for each of the posts is to in-place activate and change the material through File Properties® Material Table. With the Physical Properties Manager, this task is much simpler.

- ▶ Notice the color of the parts have changed to match the material. Click OK to dismiss the dialog box.



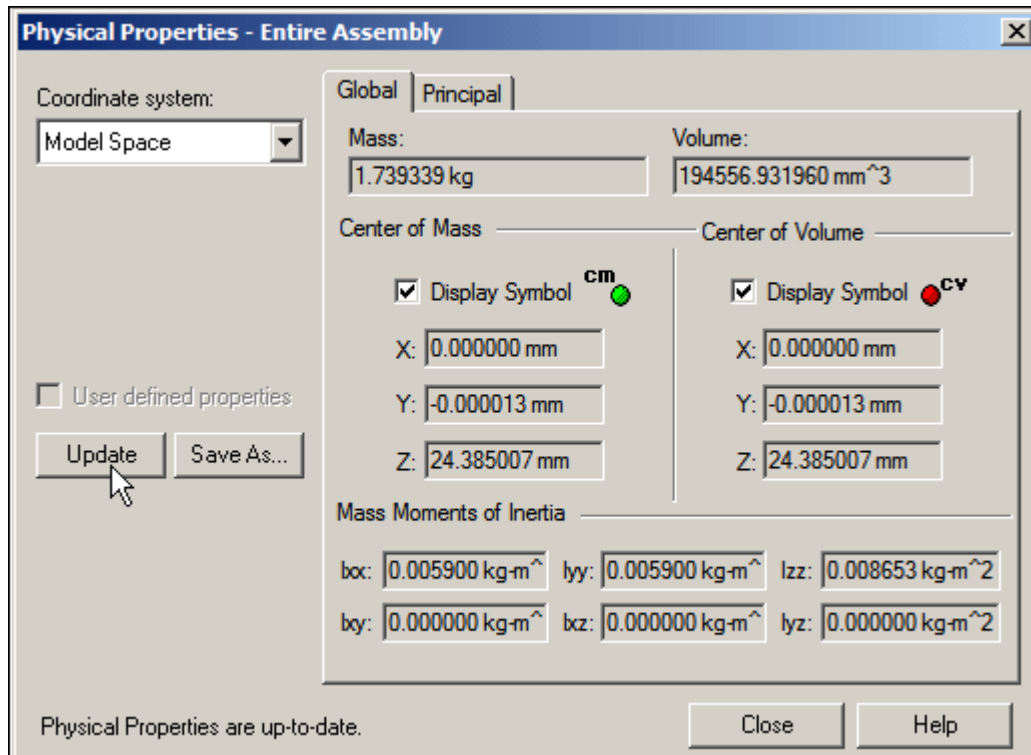
### Examine the physical properties

Since the whole assembly is symmetric about the front and right planes, you would expect the center of gravity to be centered in the top view.

- Choose the Inspect tab@ Physical Properties group@ Properties

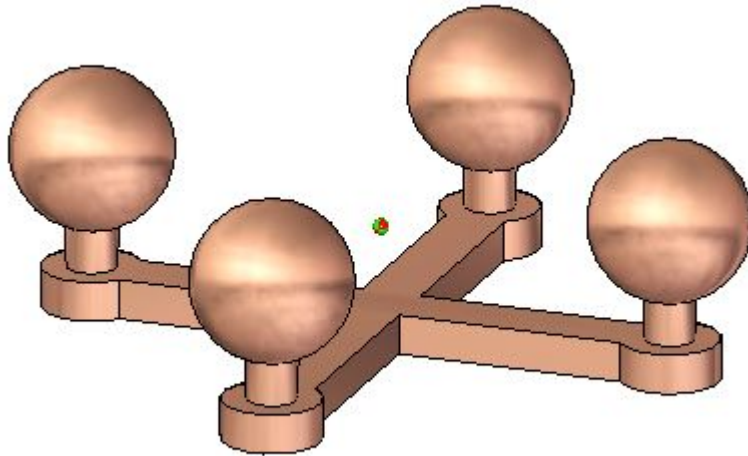


- Turn on the Display Symbol for the center of mass and the center of volume. Notice that the mass is 1.74 kilograms. Click Update, and then click Close.



**Note**

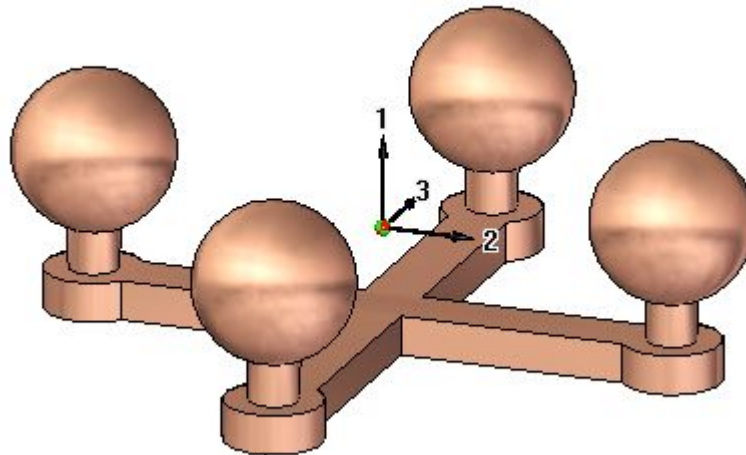
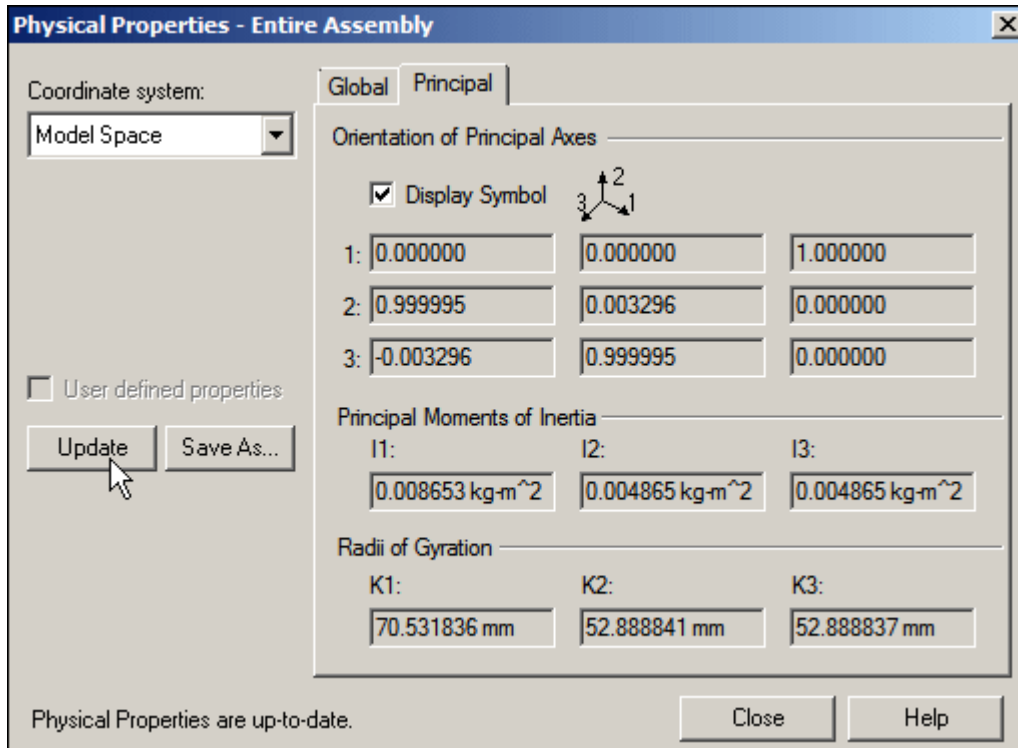
Rotate the view and notice that the center of mass and the center of volume are located in the same position. The center of volume should not change for this assembly for the remainder of the activity. The center of mass will change.



- ▶ Choose the **Inspect** tab@ **Physical Properties** group@ **Properties**



- Click the Principal tab to calculate the principal axes. Turn on the Display Symbol for the principal axes. Click Update, and then click Close.

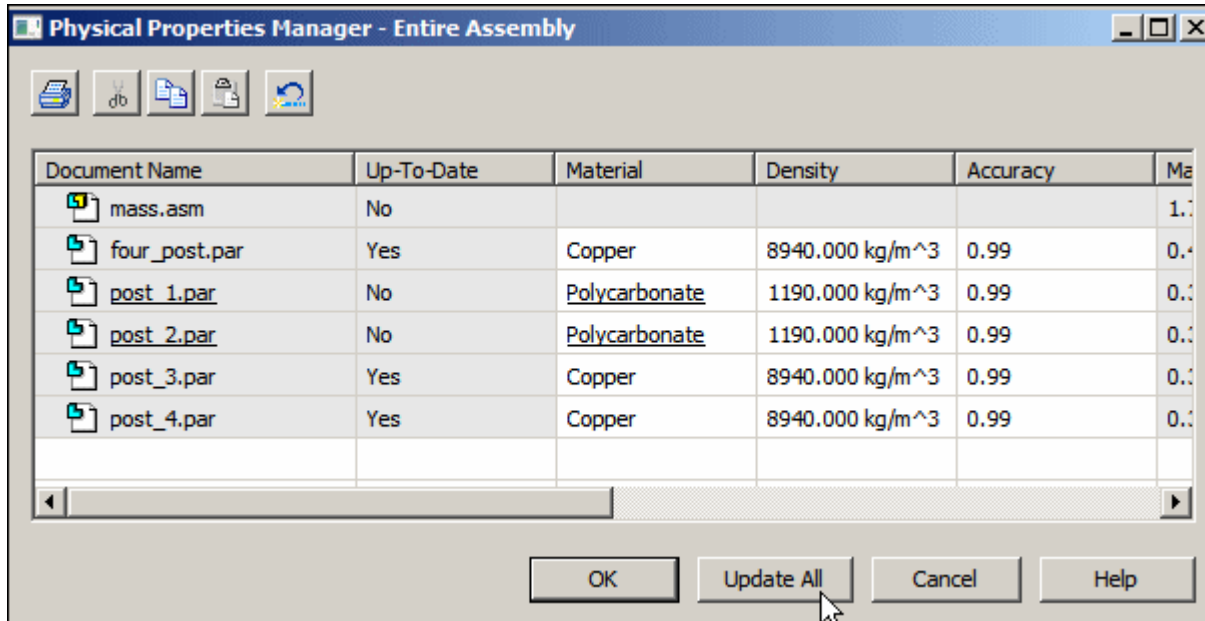


**Change the material**

- ▶ Choose the Inspect tab® Physical Properties group® Properties Manager

command .

- ▶ In the dialog box, set the material for all the parts as shown, and then click Update All. Click OK to dismiss the dialog box.

**Note**

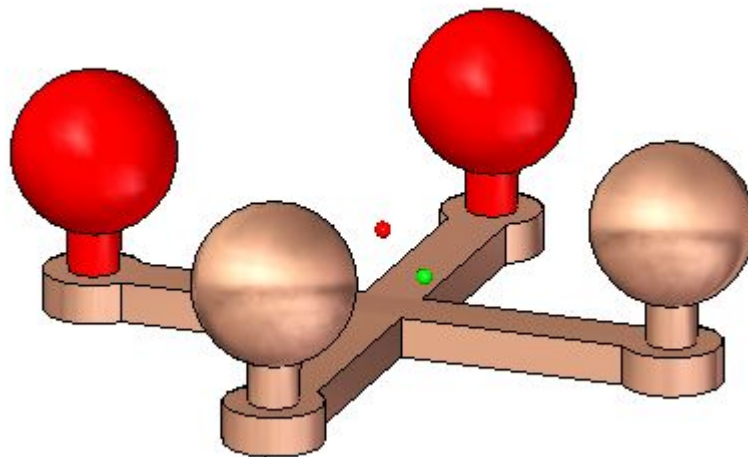
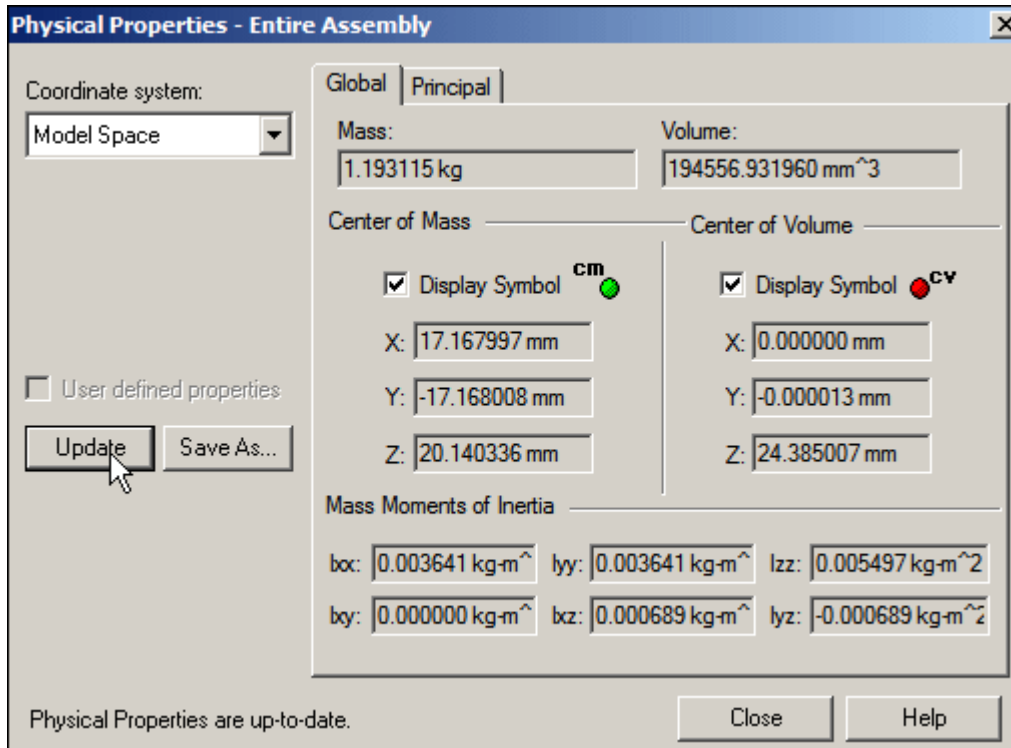
Notice that Polycarbonate is much less dense than copper. This means the center of gravity will be closer to the two copper posts.

- ▶ Choose the Inspect tab® Physical Properties group® Properties

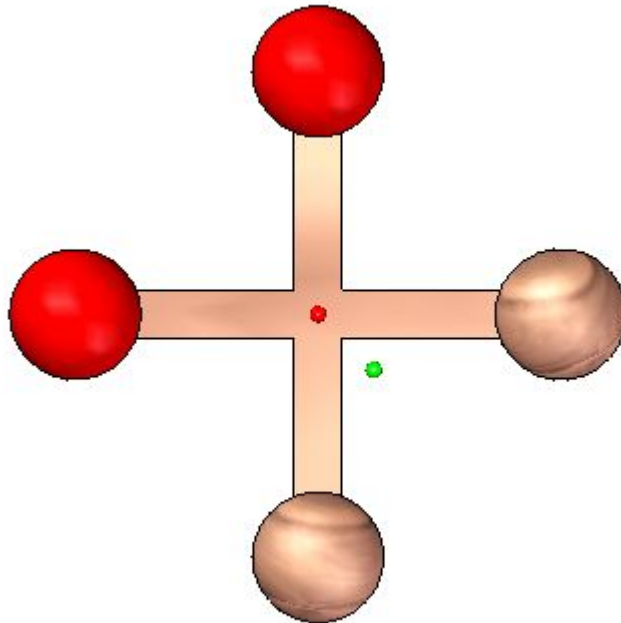
command .



- ▶ Click the Principal tab to calculate the principal axes. Turn off the display symbol for the principal axes. Click Update, and then click Close.



- Rotate the view to the top and observe the location of the center of mass.



- Continue using the Physical Properties Manager. Change the material values of the parts in the assembly and observe how the mass and center of gravity change.
- Save and close the assembly. This completes this activity.

**Activity Summary**

In this activity you learned out to change the material of a part using the Physical Property Manager, and how to calculate the mass properties for an assembly.

**Lesson review**

Answer the following questions:

1. In a part or sheet metal document, how is the density of the material defined?
2. In an assembly document, how is the density of the material defined?
3. Where is the radii of gyration calculated and the values displayed in an assembly document?

## **Answers**

1. In a part or sheet metal document, how is the density of the material defined?

The material table allows you to define the material and density of a part or sheet metal document.

2. In an assembly document, how is the density of the material defined?

The properties manager shows and allows edits to the material properties for the all the assembly components.

3. Where is the radii of gyration calculated and the values displayed in an assembly document?

The radii of gyration is calculated and displayed on the principal tab of the physical properties dialog box.

**Lesson summary**

In this lesson you learned out to change the material of a part using the Physical Property Manager, and how to calculate the mass properties for an assembly.

## Analyzing movement and detecting collisions in assemblies

You can use the [Drag Component command](#) in the Assembly environment to analyze physical motion and detect collisions between parts in an assembly.

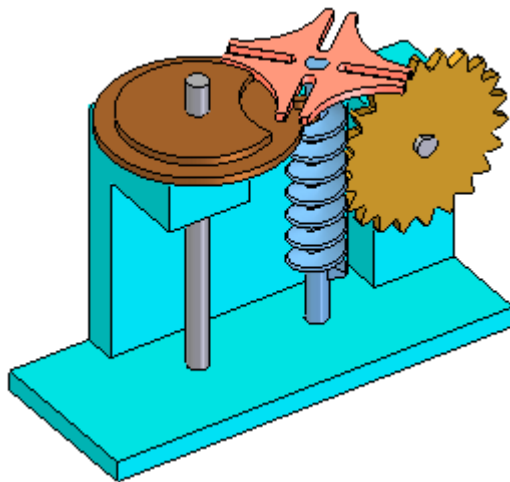
The Options button on the Drag Component command bar displays the Analyze Options dialog box so you can define the analysis options you want to use. To reduce the performance impact when working with large assemblies, you should limit the analysis to as few parts as possible.

The Analyze (Shown Parts Only) options allow you to specify whether only active parts or both active and inactive parts are analyzed. With large assemblies, you can activate only the parts you want to analyze, which can improve performance. You can also hide the parts you do not want to analyze. These options are used when you set the Physical Motion option on the command bar.

The Collision Options settings allow you to specify whether only the parts that contact the selected part are analyzed or that all parts that move in tandem with the selected part are analyzed. These options are used when you set the Detect Collisions option on the command bar.

### Analyzing motion

The Physical Motion option on the command bar allows you to simulate motion in an assembly. This option detects contact between parts and applies temporary constraints between the contacting parts to simulate motion. When you set this option, you must select a part and define a movement type and value. Also with this option, the Distance and Angle values you specify are applied incrementally, rather than all at once. This makes it possible to analyze motion in mechanisms that contain gears and other forms of sliding or intermittent contact.



### **Collision detection**

The Detect Collisions option on the command bar allows you to detect collisions between parts. You perform collision detection by dragging a part to simulate its movement in the assembly. The part you are dragging must be free to move in the assembly or be grounded. You can only select grounded parts and subassemblies when the Locate Grounded Components option on the Analysis Options dialog box is set.

If the part is not free to move, you can use the Suppress command on the shortcut menu to temporarily suppress one or more of its assembly relationships. When collisions are detected, the display changes to temporarily highlight the faces that are involved in the collision. You can also specify that the part you are dragging stops temporarily at the point of collision or that an audible warning is given.

If you are working in a shaded view, and a hidden face is involved in a collision, you will not be able to see the hidden face highlight. To see these faces highlight, set the view to Vector Hidden Line.

#### **Note**

The current zoom level impacts the accuracy of collision detection. For more accurate results, zoom in closer to the area where collisions are suspected.

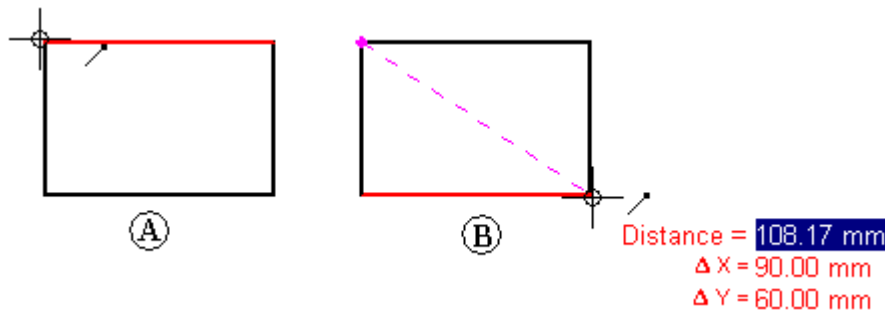


## Distance and area measurement

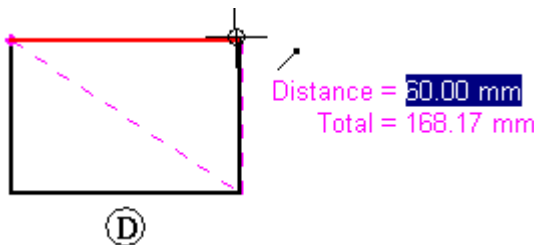
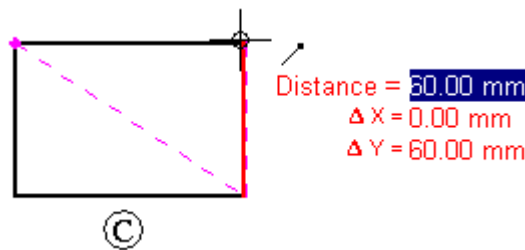
You can measure distances or areas, even when you are in the middle of another task. To set the units for measuring distances or areas, use the Properties command on the Application menu.

### Measuring distances in 2D

In the Draft environment, you can measure distance using the Measure Distance command. These commands measure linear distances or measure the cumulative linear distance along a series of points. The first point you click establishes the origin of the measurement (A). After that, you can select any keypoint to see the distance between it and the origin, as well as the delta distance along each principal axis (B).

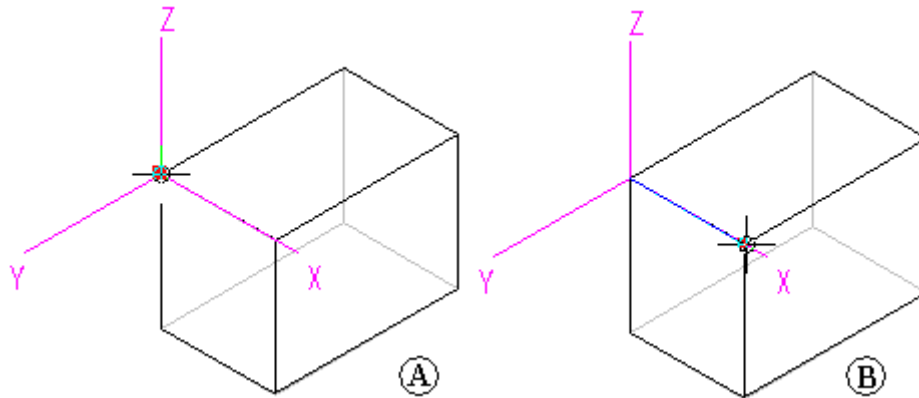


Clicking the keypoint adds it to a series of measurement points. Then you can select another point to see the new linear distance and deltas (C), or click it to see the distance between the last two points and the total cumulative distance from the origin to the last point (D). Click the right button to reset the command.

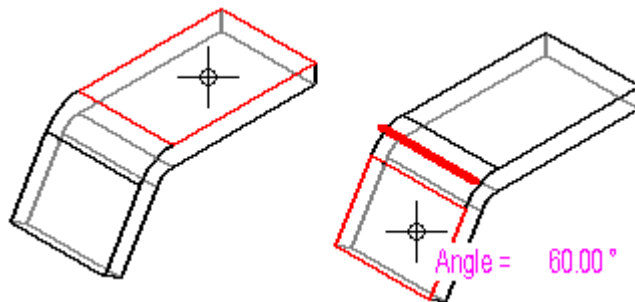


### Measuring distances and angles in 3D

In the Part, Sheet Metal, and Assembly environments, the Measure Distance command measures linear distances. The first point you click establishes the origin of the measurement (A). After that, you can select any keypoint (B) to display the Measure Distance dialog box which displays the keypoint select type, the true distance, the apparent screen view distance, and the delta distance along each principal axis.



In the Part, Sheet Metal, and Assembly environments, the Measure Angle command measures angles. You can measure between any two faces or between any three points.



### Measuring minimum distances

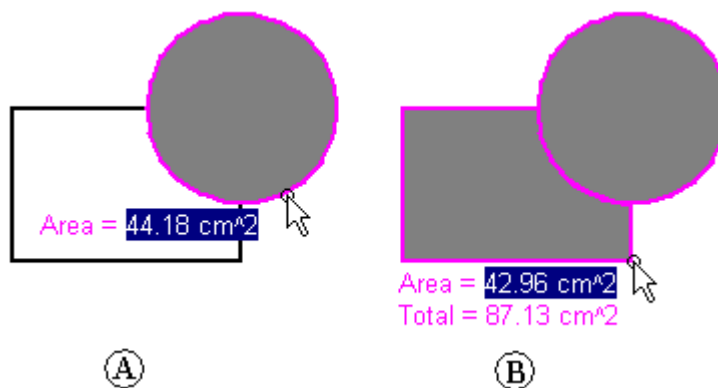
In the Part, Sheet Metal, and Assembly environments, you can use the Measure Minimum Distance command to measure the minimum distance between any two elements or keypoints. You can use the Select Type option on the Minimum Distance command bar to filter which type of elements you want to select. When working in the context of an assembly, you can also use the Activate Part option to activate the parts you want to measure.

## Measuring normal distances

In the Part, Sheet Metal, and Assembly environments, the Measure Normal Distance command measures normal distances between a planar element or line and a keypoint. You can use the Element Types option on the Measure Normal Distance command bar to filter which type of elements you want to select. You can use the Key Point option to specify the type of keypoint you want to identify when measuring the distance. You can use the Coordinate System option to select a user-defined coordinate system to define one of the points. If you use a coordinate system, the returned values will be relative to the specified coordinate system. When working in the context of an assembly, you can also use the Activate Part option to activate the parts you want to measure.

## Measuring areas

The Measure Area command, available only in the Draft environment and in 2D profiles and sketches, measures the area inside a closed boundary (A). You can also measure the cumulative area inside more than one closed boundary by holding the Shift key as you click elements (B). Each time you click, the area of the last element is displayed, along with the total area. Click another element without holding the Shift key to reset the command.



## Measuring lengths

The Measure Total Length command measures the cumulative length of a select set of 2D geometry.

## Measuring automatically

In addition to the individual distance, area, length, and angle commands, you can use the Smart Measure command in 2D and 3D environments to measure automatically based on what you select:

- Select a single 2D element or 3D object to measure its length or its angle or radius.
- Select two or more 2D elements or 3D objects to measure the distance or angle between them.

The Smart Measure command works like the Smart Dimension command, except that it does not place a dimension as a result.

### **Copying measurement values**

You can copy the highlighted measurement value to the Clipboard by pressing Ctrl+C. You can then use the copied value as input for another command. For example, you can paste the copied value into the Line command bar to define the length of a line. Use the Tab key if you want to highlight a different value.

### **Measuring drawing view geometry**

When you measure model geometry within a drawing view, or when you measure distances between model edges in two drawing views, you can select the Use Drawing View Scale check box on the command bar to specify that the measured value is displayed using the equivalent of the model distance.

Alternatively, you can apply a user-defined scale value by selecting it from the Scale list on the command bar.

#### **Note**

- You can show the scale of a drawing view using the General page (Drawing View Properties dialog box).
- User-defined scale values are defined in the Drawing View Scales section of the Custom.xml file, in the Solid Edge Program folder. See the Help topic, Add custom drawing view scales to Solid Edge.

## Drag Component command

Translates or rotates parts in an assembly. The part being moved must be grounded or not fully positioned. You can use this command to do the following:

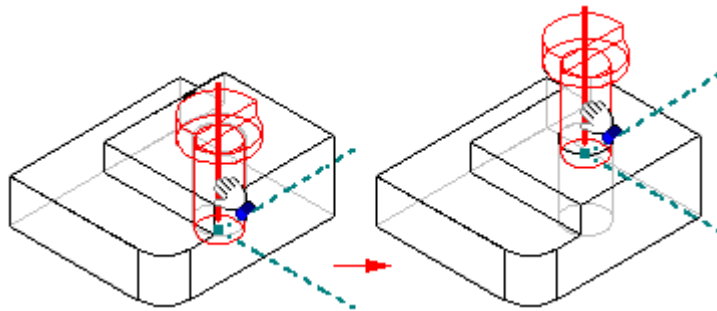
- Reposition parts dynamically along the x, y, or z axis.
- Analyze physical motion in mechanisms.
- Detect collisions between parts.

Any relationships applied to the part will prevent movement along the axis whose position is controlled by a relationship. A part will move only in an under-constrained direction. You can use the Suppress command on the shortcut menu to temporarily suppress one or more assembly relationships to allow the part to move.

You can only select grounded parts and subassemblies when the Locate Grounded Components option on the Analysis Options dialog box is set.

After you select the part you want to move, you can use the options on the command bar to specify the type of movement you want. For example, you can click the Move button, then position the cursor over one of the principal axes and drag the part to a new position.

You can restart the command by clicking the right mouse button.



### Note

Patterned parts do not participate in collision detection or motion analysis.

When you use the Drag Component command to move an adjustable part, the assembly relationships positioning the adjustable part are temporarily suppressed. When you complete a move cycle, the relationships are reapplied. If the movement you define conflicts with the adjustable part's relationship structure, the adjustable part is returned to its original position or a position that is consistent with the relationship structure.

**Activity: Analyzing motion****Overview**

This activity shows how to use motion to analyze collisions in moving parts.

## Activity: Analyzing motion

### Overview

This activity shows how to use the Motion command to analyze collisions in moving parts.

### Objectives

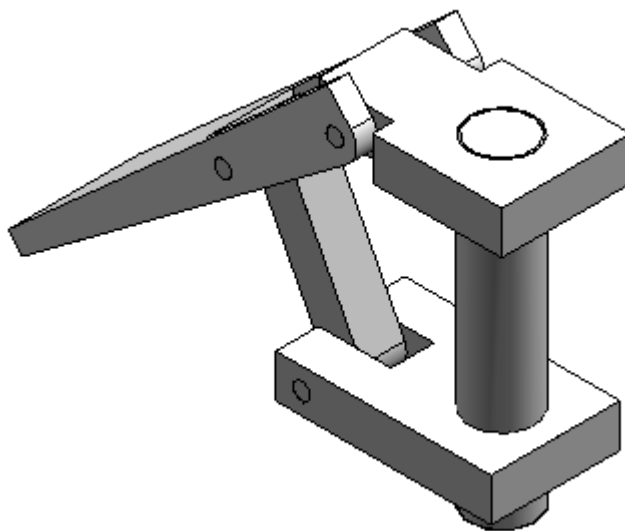
You will use the Motion command to analyze the motion of a simple bar linkage.

### Activity

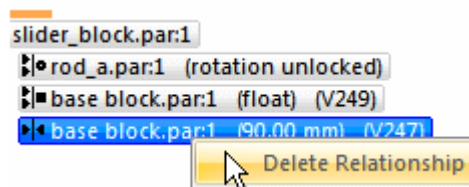
*Open the assembly*

Open the assembly and activate all the parts in the assembly.

- Open and activate all the parts in the assembly *slider\_linkage.asm* located in the folder where the activity files are located.




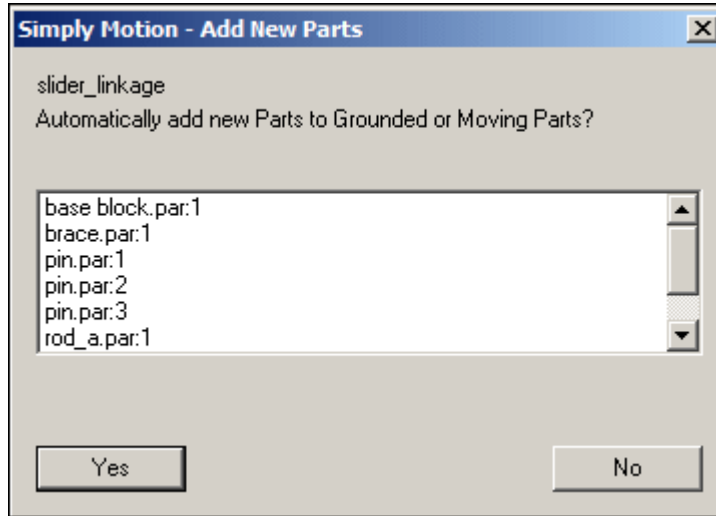
- In Assembly PathFinder, select *slider\_block.par*. Right-click the mate relationship, and click Delete Relationship.



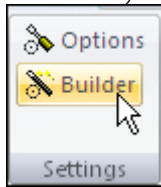
*Motion environment*

Start the Motion application.

- ▶ Choose the Tools tab® Environs group® Motion command .
- ▶ When prompted to automatically add the parts, click No in the dialog box.



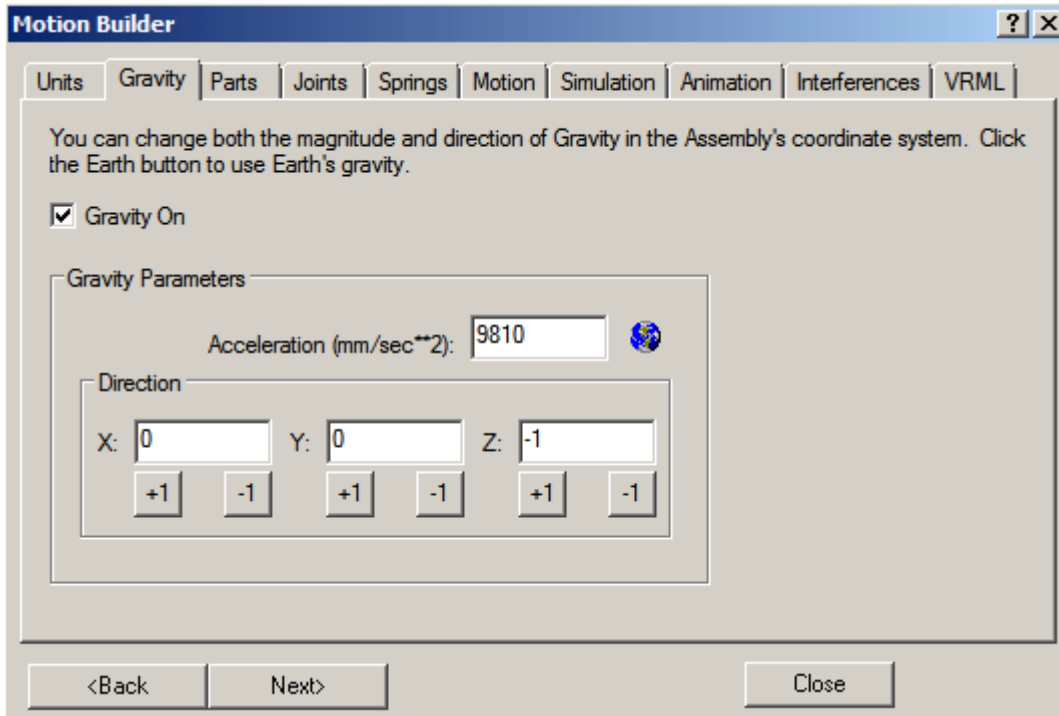
- ▶ In the Motion environment, in the Settings group, select the Display Motion



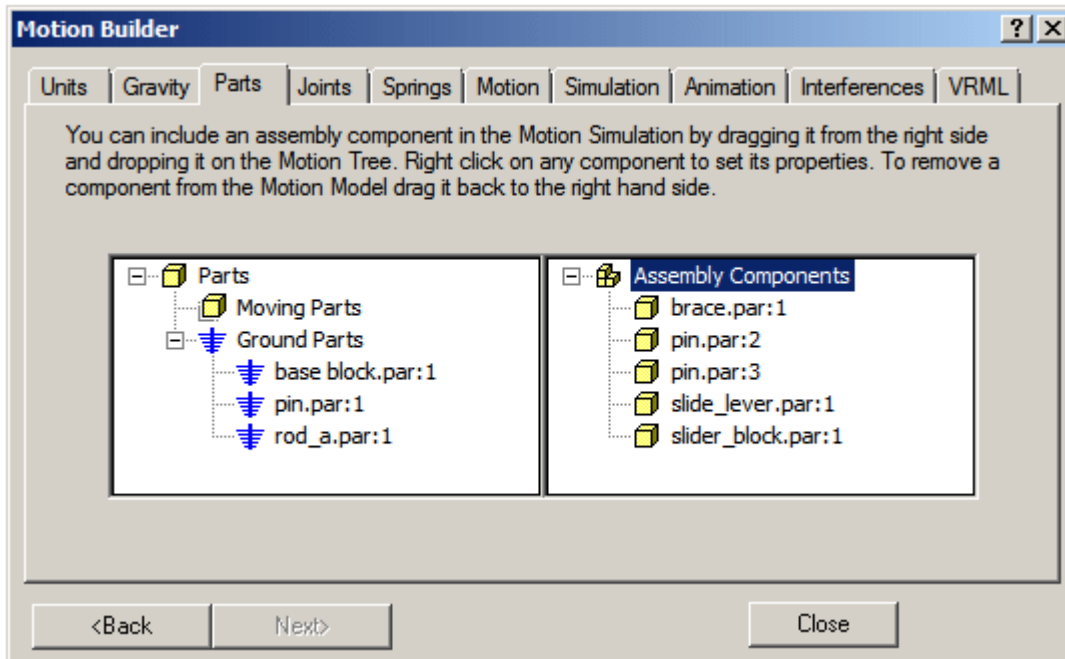
Builder command



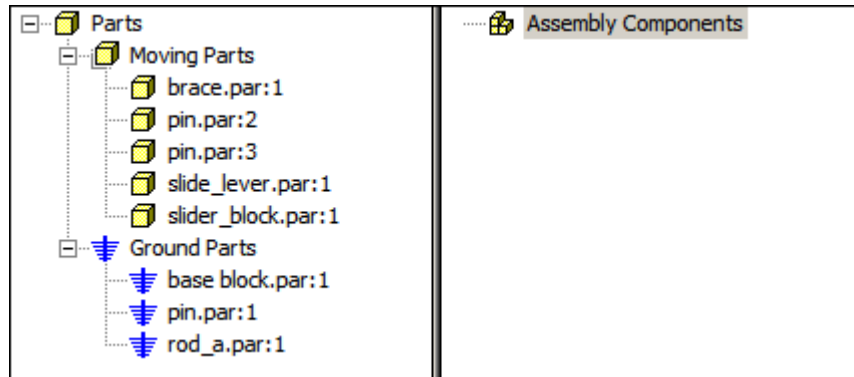
- ▶ In the Motion Builder wizard, click the Gravity tab. Verify the Gravity On box is selected, and then click Next.



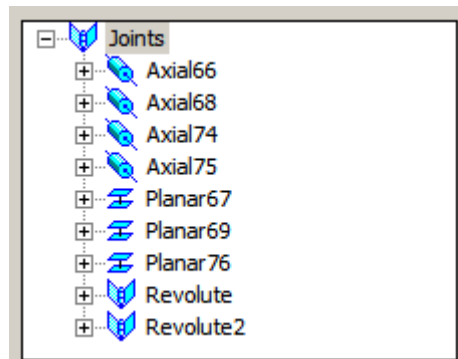
- ▶ On the Parts tab, drag *base block.par;1*, *rod\_a.par;1*, and *pin.par;1* to Ground Parts.



- ▶ Drag the remaining parts to Moving Parts. Dismiss the Simply Motion Messages dialog box.



- ▶ Click Next to proceed to the Joints tab.
- ▶ Click the Joints plus sign to expand the joints list.



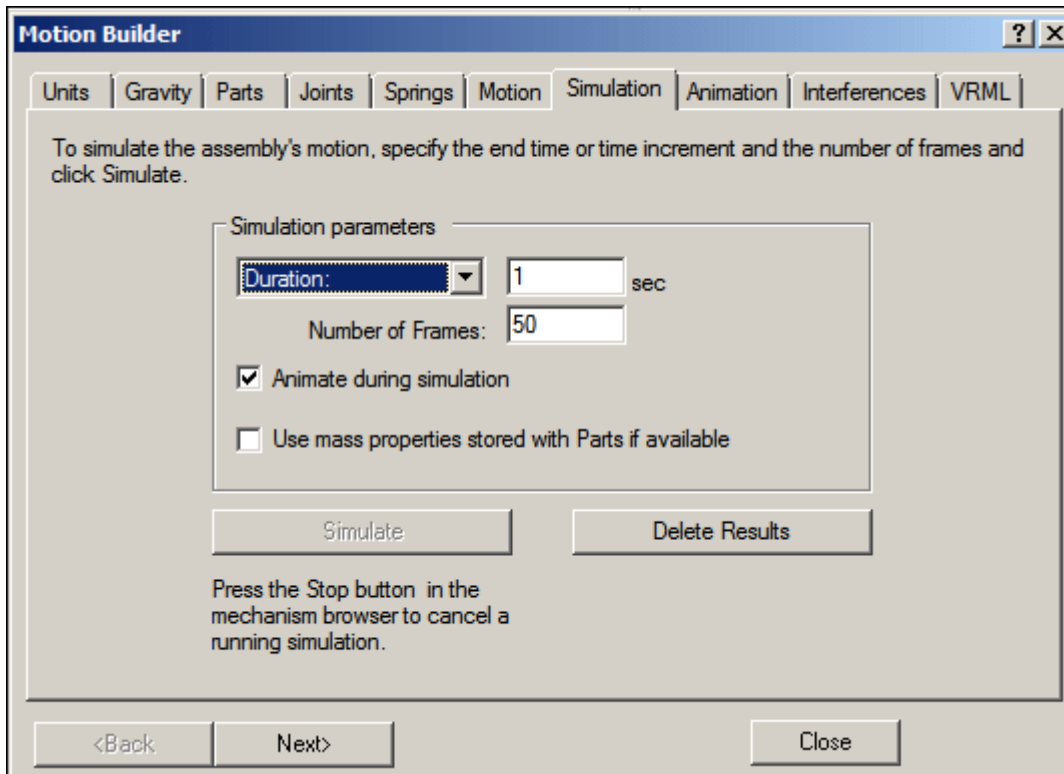
### Note

Because you deleted the mate relationship before activating the motion command, Solid Edge did not recognize or create a joint from the mate relationship.

### *Find the initial collision*

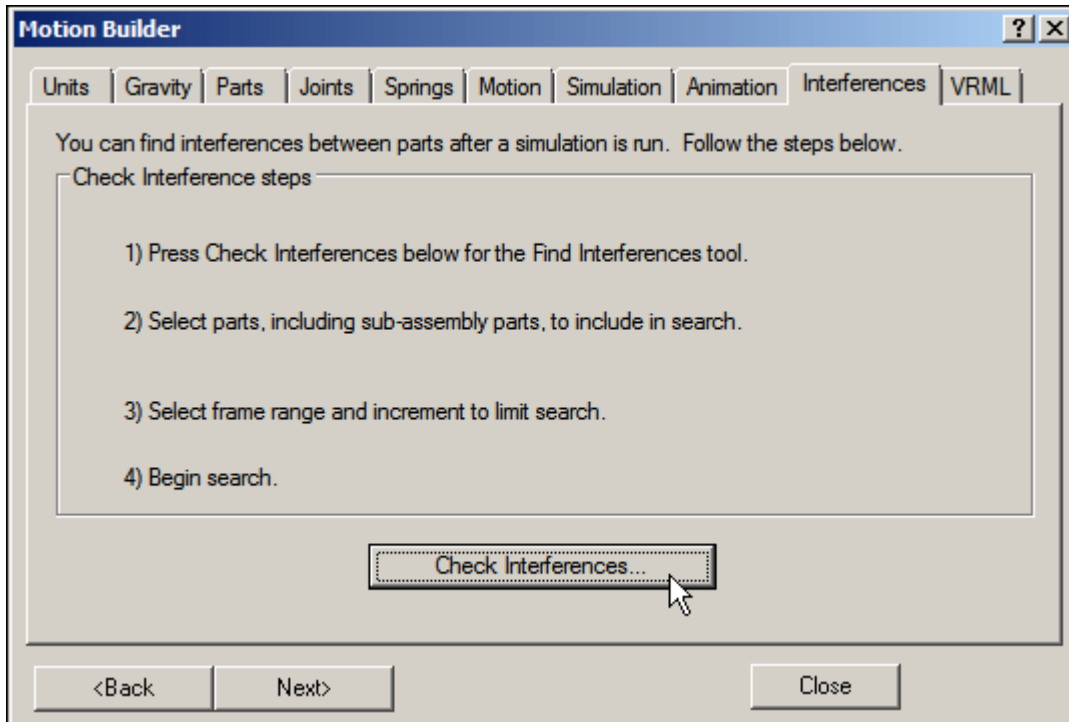
Simulate motion of the linkage and find the initial collision of the parts.

- ▶ In the Motion Builder wizard, click the Simulation tab.

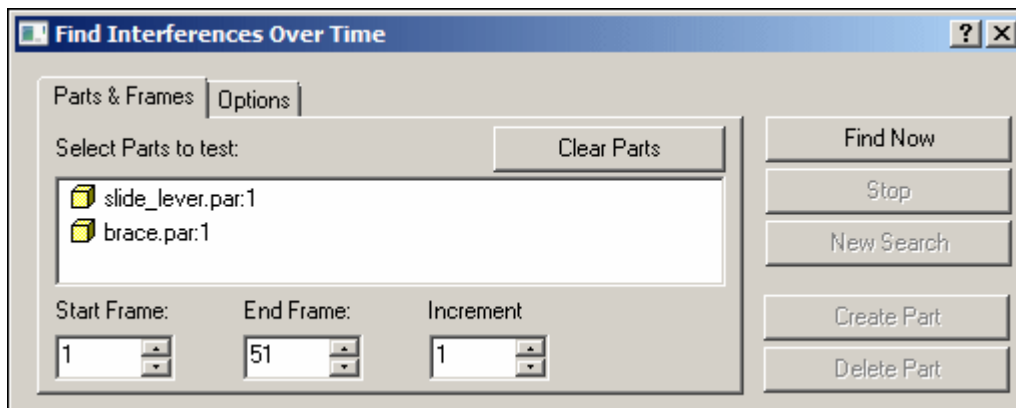
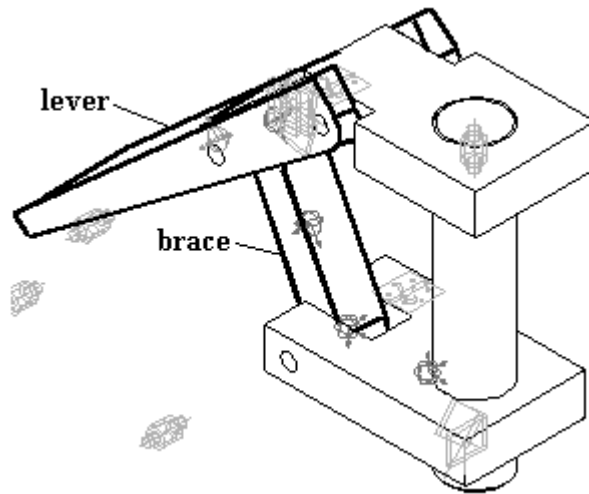


- ▶ Click Simulate to set the parts in motion, and watch the motion display.
- ▶ Click the Interferences tab.

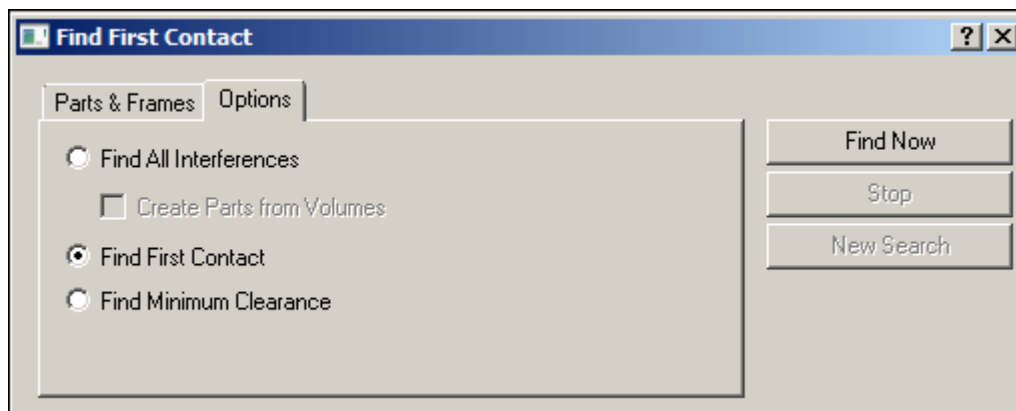
- If the wizard blocks the view of the assembly, drag the wizard window to the right of the assembly window. Click Check Interferences.



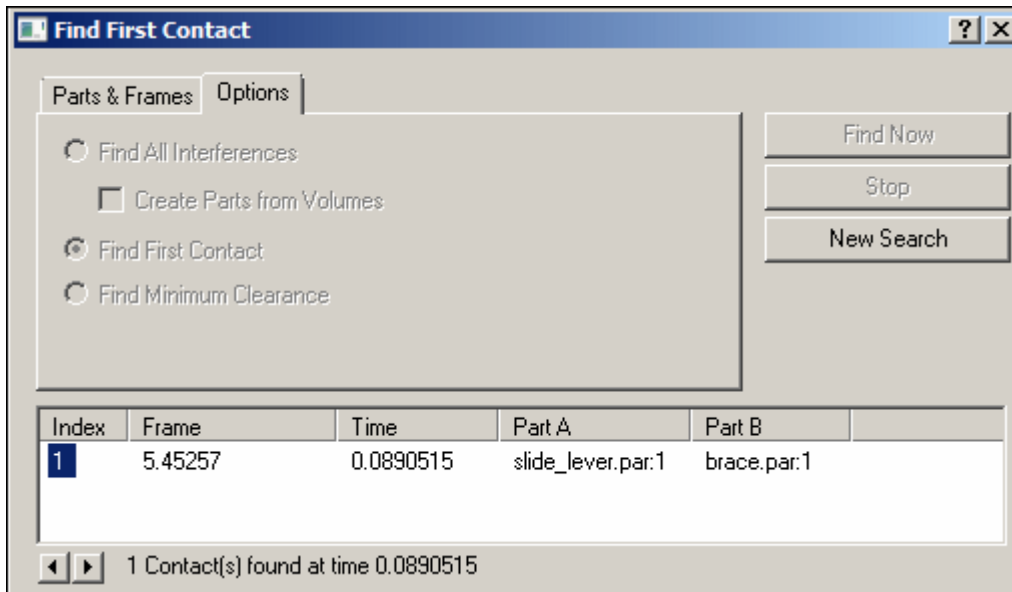
- ▶ In the assembly graphics window, click *brace.par* and *slide\_lever.par* to add them to the list of parts to test in the Find Interferences Over Time dialog box.



- ▶ Click the Options tab. Click Find First Contact, and then click Find Now.



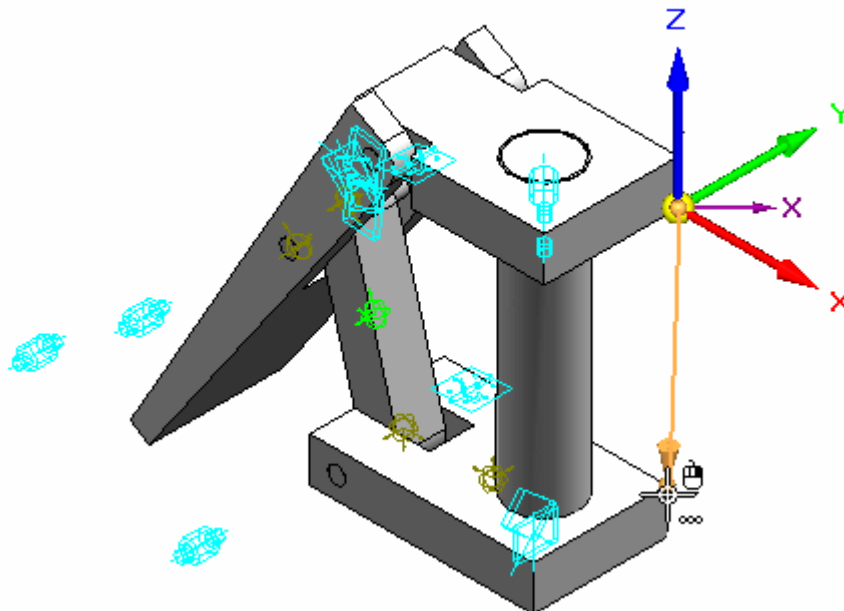
- ▶ Watch the assembly move and then stop when the first interference between the two parts is detected.



**Note**

Part interference begins at frame 5.45257.

- ▶ Close the Find First Contact dialog box.
- ▶ On the Inspect tab, click Measure Distance. Set the element type to All elements.
- ▶ For the points to measure between, select the lower-right corner of the base and the upper-right corner of the slider block.



- Notice that the distance in delta Z is -110.14 mm. These parts interfere if the distance between the base block and the slider block is 110.14 mm or greater.
- On the Measure command bar, click Close.
- Close any open dialog boxes. Click Close and Return to return to the assembly.
- Save and close the file. This completes the activity.

### *Activity Summary*

In this activity you learned how to use the Motion command to analyze the movement of a simple linkage. Parts that did not move were grounded and parts that did move had no constraints in their direction of travel. When motion was applied, interference was detected and measured.

**Lesson review**

Answer the following questions:

1. What is Simply motion?
2. How can you detect the initial point of a collision between moving parts?
3. Once an interference is detected how can the range of motion be limited so that interference does not occur?



## **Answers**

1. What is Simply motion?

The Simply Motion module allows you to perform motion simulations on Solid Edge assemblies. Simply Motion creates moving parts and motion joints directly from assembly constraints. Additional joints, springs, and motion generators can be added through an easy to use "Wizard" style interface. Simply Motion contains a 3D dynamic motion engine that allows you to simulate problems that are far beyond simple linkages or kinematic type problems. The simulation results can be used to generate animations of your moving assembly, or to check for interference as the assembly moves through its full range of simulated motion

Simply Motion is a subset of Dynamic Designer from Design Simulation Technologies. Simply Motion is included in Solid Edge at no charge. Upgrades to Dynamic Designer can be purchased from Design Simulation Technologies.

2. How can you detect the initial point of a collision between moving parts?

Using the interference tab in Simply motion will analyze and show where the initial interference occurs. The drag component command also checks for collisions and interference.

3. Once an interference is detected how can the range of motion be limited so that interference does not occur?


The range offset values can be set for valid assembly relationships.

**Lesson summary**

In this lesson you learned how to use the Motion command to analyze the movement of a simple linkage. Parts that did not move were grounded and parts that did move had no constraints in their direction of travel. When motion was applied, interference was detected and measured.

## Sensors

When constructing parts and assemblies, you often need to keep track of critical design parameters. For example, when designing a shield or shroud that encloses a rotating part, you must maintain enough clearance for maintenance and operational

purposes. You can use the Sensors tab  on PathFinder to define and keep track of design parameters for your parts and assemblies.

You can define the following types of sensors:

- Minimum distance sensors
- General variable sensors
- Sheet metal sensors
- Surface area sensors
- Custom sensors

### Creating a sensor

Although there are several types of sensors, you follow the same basic steps when creating any sensor:

1. From the Sensors tab on PathFinder, select the sensor type you want.
2. Define what you want to track in the design.
3. Define the operating limits for the sensor.

Every sensor you define in the document is displayed and managed on the Sensors tab on PathFinder.



#### Minimum distance sensors

Minimum distance sensors are used to track the minimum distance between any two elements. For example, you can track the minimum distance between two part faces in an assembly. You define a minimum distance sensor similar to how you measure the minimum distance between two elements with the Minimum Distance command.

When you click the Minimum Distance Sensor button on the Sensors tab, the Minimum Distance command bar is displayed so you can select the two elements you want to measure between. After you select the two measurement elements, the Measure Distance box displays the current minimum distance value. When you click the Close button on the command bar, the Minimum Distance Sensor Parameters dialog box is displayed so you can define the sensor parameters you want, such as the sensor name, display type, threshold value, sensor range, and so forth. When you click OK, the new sensor is displayed on the Sensors tab.



#### General variable sensors

You can use a general variable sensor to track variables, such as driving and driven dimensions. To create a general variable sensor, select the Variable Sensor button on the Sensors tab, then select the variable you want in the variable table. When

you click the Add Variable button on the Variable Sensor Parameters dialog box, the variable value is added to the Current Value box. You can then define the remaining sensor parameters.



### Sheet metal sensors

You can use sheet metal sensors to track design parameters, such as the minimum distance between particular types of sheet metal features and part edges. You can create your own sheet metal sensors from scratch, or you can select from a list of predefined examples.

Sheet metal sensors are available only in sheet metal documents.

To create a sheet metal sensor, select the Sheet Metal Sensor button on the Sensors tab, then use the Sheet Metal Sensor command bar to define the faces and edge sets you want to track. When you click the Finish button, the Sheet Metal Sensor Parameters dialog box displays so you can define the remaining sensor parameters.

#### Note

Sheet metal sensors are not available in 64-bit systems.



### Surface area sensors

You can use a surface area sensor to monitor a surface or a set of surfaces. You can monitor both positive and negative surface area. A negative surface area sensor monitors the "holes" or internal boundaries in a surface. For example, you may need to track the total area for a series of ventilation holes and cutouts in a surface.

The Surface Area Sensor command bar is where you define the faces you want to track as a sensor. When you click the Accept (checkmark) button on the command bar, the Surface Area Sensor Parameters dialog box displays so you can define the remaining sensor parameters. This sensor type is available only for part and sheet metal documents.



### Custom sensors

You can use a custom sensor to monitor any numeric result that is calculated from a custom program. For example, you could create a custom program that assigns a manufacturing cost to each feature type used for creating sheet metal parts. The program would then monitor the part features and give you the part cost of the completed model.

When you click the Custom Sensor button, it displays the Custom Sensor DLL dialog box so you can define the DLL and user function you want. After you define the DLL and user function, and click OK, the Custom Sensor Parameters dialog box displays so you can define the sensor parameters.

#### Note


For more in-depth information on how to use custom sensors, see the *readme.doc* file delivered to the Solid Edge\Custom\CustomSensor folder.

## Sensor alarms

When a change to the model exceeds the defined sensor threshold limit, the Sensors tab displays a special notification symbol and a sensor violation alarm is displayed in the upper-right corner of the graphic window.

Violation alarms


On the Sensors tab: 

In the graphics window: 

When one of the elements a sensor tracks is deleted, a warning notification symbol is displayed to call attention to it.

Warning alarms

On the Sensors tab: 

In the graphics window: 

When you see an alarm symbol in the upper-right corner of the graphics window, you can click it to respond to the alarm. This displays the Sensor Assistant, which contains hyperlinks to all the sensor violations and warnings in the document.

You can activate or deactivate the Sensor Assistant and alarm notification in the graphics window using the Show Sensor Indicator option on the Helpers page of the Solid Edge Options dialog box. This does not affect the operation of the sensors themselves.

To learn how to use the Sensor Assistant, see [Responding to sensor alarms](#).

**Activity: Creating a sensor****Overview**

When you complete this activity, you will be able to create a distance sensor to track the motion of a simple bar linkage.

**Activity: Creating a sensor**

**Objectives**

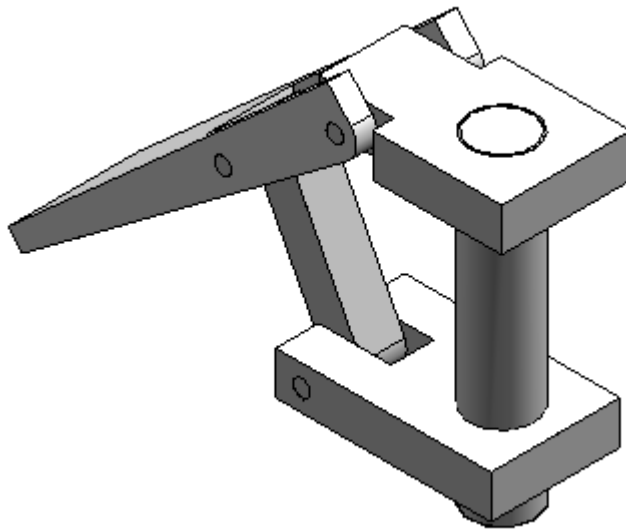
When you complete this activity, you will be able to create a distance sensor to track the motion of a simple bar linkage.

**Activity**

*Open the assembly with all parts active*

Open the assembly and activate all the parts in the assembly.

- Open and activate all the parts in the assembly *slider\_linkage.asm* located in the folder where the activity files are located.



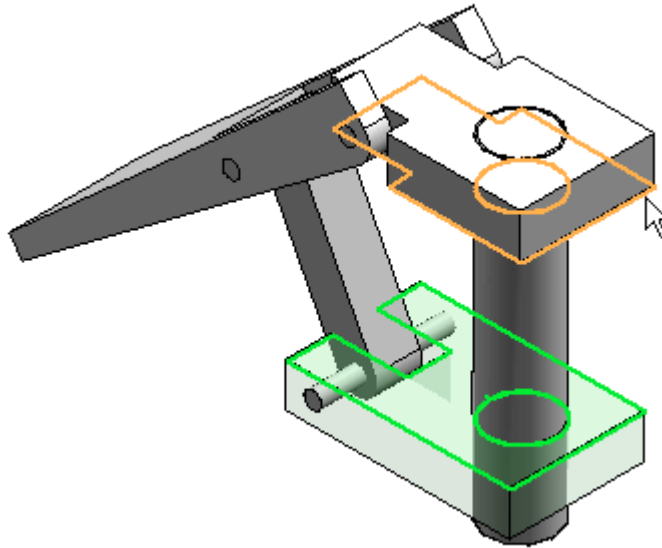
*Reestablish the mate relationship*

Reestablish the previously deleted mate relationship.

- In Assembly PathFinder, select *slider\_block.par* and then click Edit Definition





- ▶ For the next relationship, create a mate relationship with an 80 mm offset value between the top of the slider block and the bottom of the base block.



### *Create sensor*

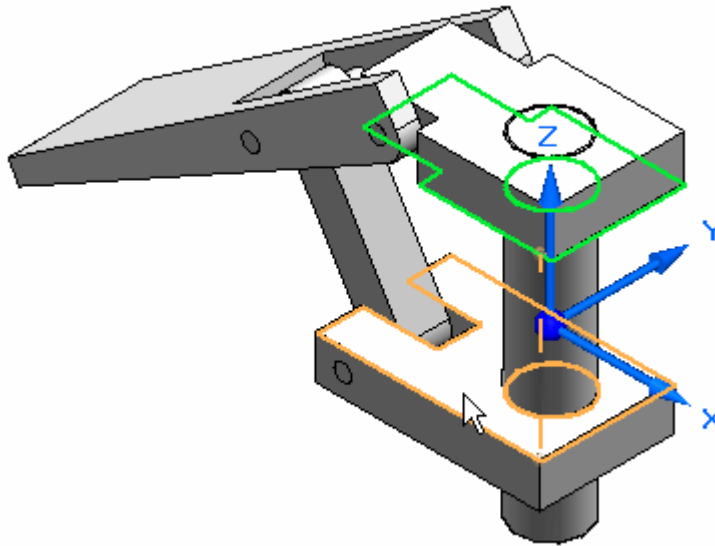
Create sensors.

- ▶ In PathFinder, click the Sensors tab .
- ▶ Click the Minimum Distance Sensor .
- ▶ From the Element Types list, select Surfaces.





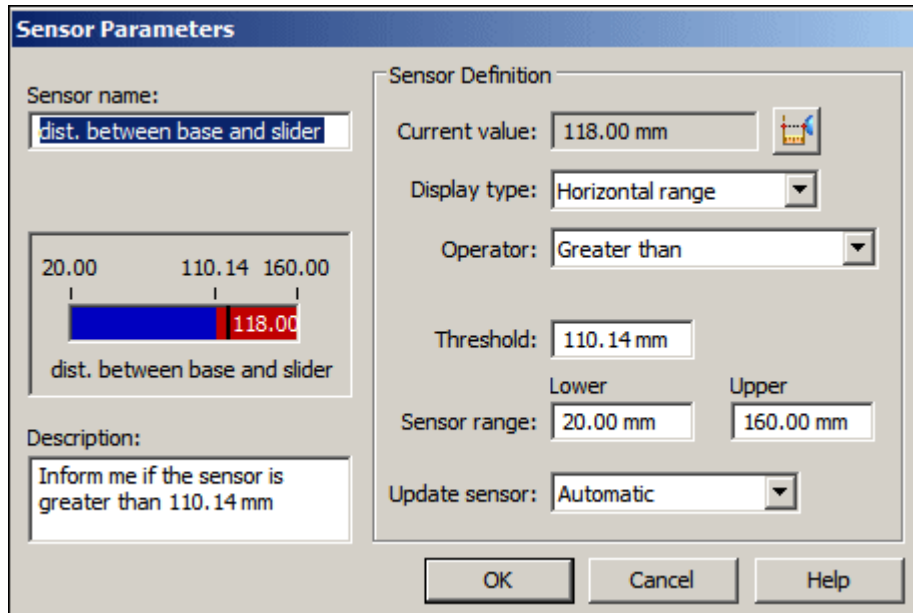
- ▶ To measure between the top of the slider block and the bottom of the base block, select the two surfaces.



Minimum Distance
Minimum Distance: 80.00 mm
Delta X: 0.00 mm
Delta Y: 0.00 mm
Delta Z: -80.00 mm
Point1: 25.00 -30.00 55.00
Point2: 25.00 -30.00 -25.00

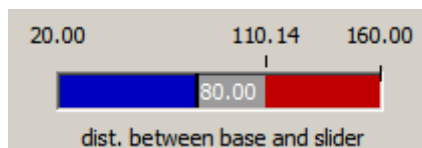
- ▶ Click Close.

- ▶ In the Sensor Parameters dialog box, enter the information as shown, and click OK. In the previous activity, when interference was found using the Motion command, the software determined a threshold value of 110.14. Click OK.



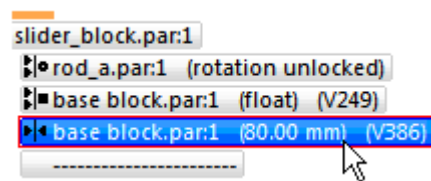
**Note**

Notice that the sensor is added to PathFinder.

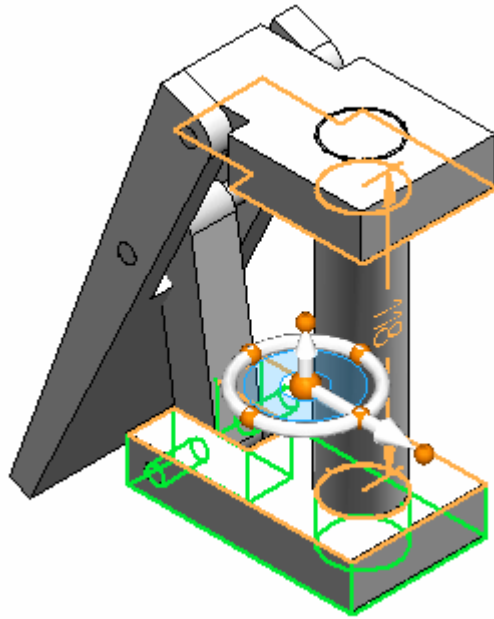


*Test the sensor*

- ▶ In Assembly PathFinder, click *slider block.par*, and then click the mate relationship with the 80 mm offset in the lower pane of Assembly PathFinder.



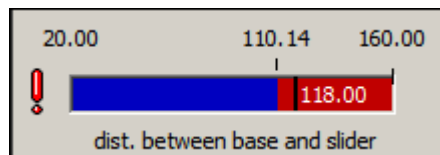
- ▶ Edit the slider block offset value to 118 mm.



- ▶ In PathFinder, click the Sensors tab.

**Note**

Notice that the sensor shows the range is no longer valid.

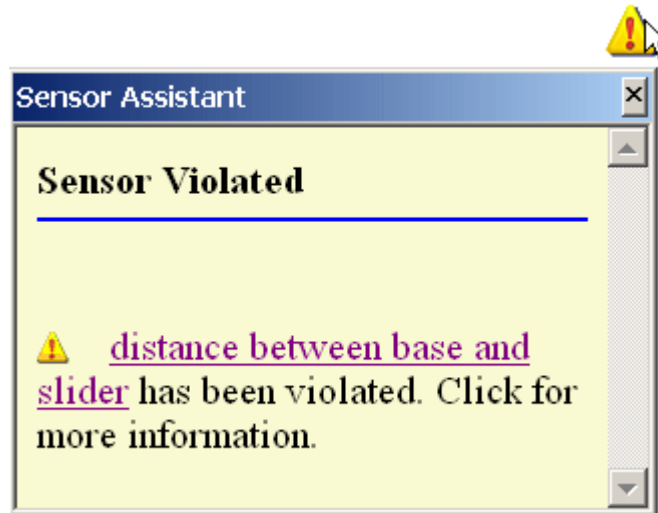


**Note**

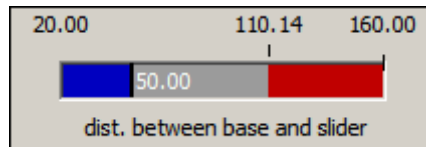
When a sensor is violated, an icon appears on the upper right corner of the screen.



- ▶ Click the icon in the corner. Information is displayed. Click for more information. The sensors document in PathFinder is displayed.



- ▶ Set the offset value of the slider block back to 50.00 mm. Notice that the sensor returns to a valid state.



### Activity Summary

In this activity you learned how to create a distance sensor and establish a valid range of travel. Parameters on the distance sensor were set so that when the travel of the mechanism passed the valid range, alerts were triggered.

## **Lesson review**

Answer the following questions:

1. Name three types of sensors.
2. Is the interface for sensors a tab in pathfinder?
3. How can you tell when a sensor has been violated?

**Answers**

1. Name three types of sensors.

The types of sensors are:

- a. Minimum distance sensor
- b. General variable sensors
- c. Surface area sensors

Custom sensors and sheet metal specific sensors can also be created.

2. Is the interface for sensors a tab in pathfinder?

The sensors tab is located in pathfinder?

3. How can you tell when a sensor has been violated?

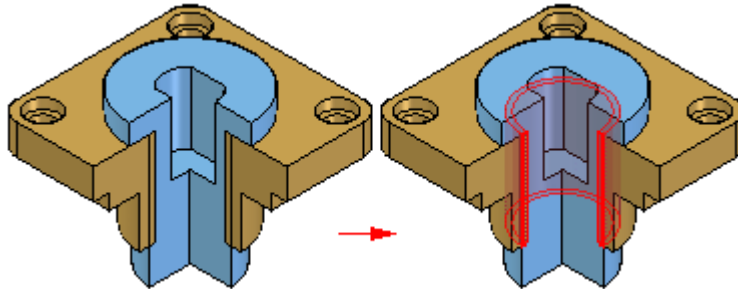
An attention icon will display. It must be turned on in Solid Edge Options>Helpers tab.

## **Lesson summary**

In this lesson you learned how to create a distance sensor and establish a valid range of travel. Parameters on the distance sensor were set so that when the travel of the mechanism passed the valid range, alerts were triggered.

## Checking interference on parts

Solid Edge allows you to check for interference between parts in assemblies using the Check Interference command on the Inspect tab.



## Creating sets of parts

To run an interference analysis, you must create one or two sets of parts. A set of parts typically contains many parts; however, each set can contain any number of parts. If a set of parts will include all the parts in a subassembly, you can select the subassembly using the PathFinder.

### Note

You can also use the Select Tools tab on PathFinder to define a select set using a query.

## Checking sets of parts for interference

You can check parts in the following ways during the interference analysis:

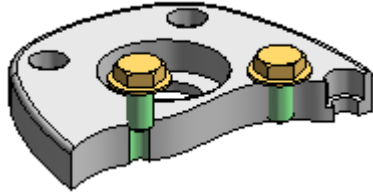
- All parts of set one against all parts of set two.
- All parts of set one against all other parts in the active assembly.
- All parts of set one against all currently displayed parts.
- All parts of set one against themselves.

Unless you indicate otherwise, the software uses the first option, checking all the parts of set one against all the parts of set two.



## Interference and threaded features

You can use the settings on the Options Tab (Interference Options dialog box) to control how threaded fastener interference is checked.



Check interference can report or ignore interferences when the threaded portion of a bolt interferes with a non-threaded hole in a mating part. For example, you would typically want to ignore interferences when using a self-tapping bolt in a non-threaded hole, but would want to report interferences when a standard bolt extends past the thread depth in a threaded mating part.

The Ignore Threaded Fasteners Interfering With Non-threaded Holes option allows you to control whether interference is detected in these situations.

Check interference also can report or ignore interferences between a threaded cylinder and a threaded hole whose nominal diameters match. When you set the Ignore Interferences of the Same Nominal Diameter option, if the thread pitch does not match between a bolt and a threaded hole with the same nominal diameters, no interference is detected.

## Analyzing results

Before running the interference analysis, you can set options for analyzing results. The following methods are available:

- Output a report to a text file.
- Display the interfering volumes.
- Save the interfering volumes as parts.
- Highlight the interfering parts.
- Dim the display of parts that do not interfere.
- Hide parts not selected for an interference check.

If no interferences were detected during the interference analysis, the software displays a message box telling you that no interferences were detected.

**Activity: Interference checking****Overview**

In this activity, you learn how to:

- Use the interference checking and measure tools to determine where parts interfere with each other.
- Edit the part shape to correct the offending part.
- Edit part relationships.

## Activity: Interference checking

### Overview

When you complete this activity, you will know how to use the interference checking and measure tools to determine where parts interfere with each other, how to edit the part shape to correct the offending part, and how to edit part relationships.

### Objectives

The activity consists of the following:

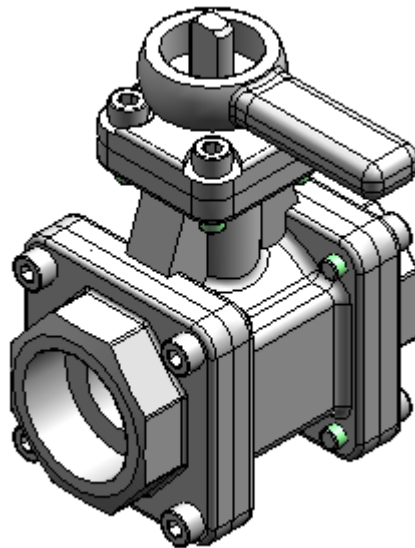
- Inspecting parts and subassemblies to determine interference and to measure distances required to correct the interference.
- Editing parts in the context of the assembly to correct the interference.


### Activity

*Open the assembly and analyze the interference*

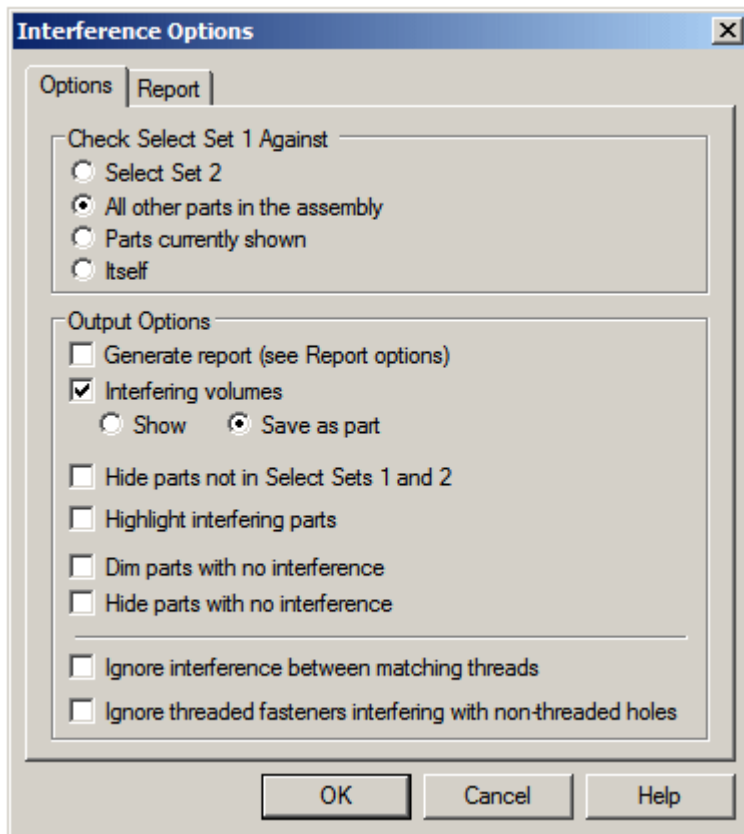
Open an assembly and check for interference between adjacent parts. Once the interference is found, the volume representing the interference is created as surface features.

- Open and activate all the parts in the assembly *ball\_valve.asm* located in the folder where the activity files are located.



- Choose the Inspect tab® Evaluate group® Interference command .
- Click Interference Options.

- ▶ On the Options tab, set the options as shown and click OK.



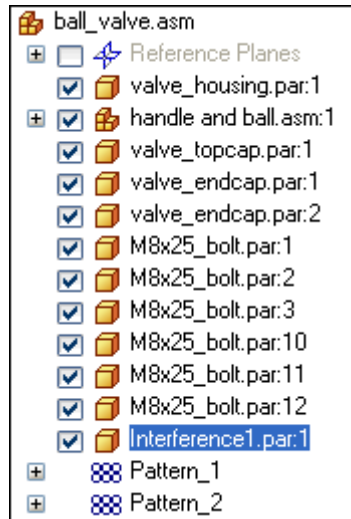
- ▶ Select the subassembly *handle and ball.asm* in PathFinder. Accept the selection.
- ▶ Click Process.

**Note**

A new part is placed at the bottom of the list in PathFinder. This part represents the volume of the interference. The interference is represented as a construction surface. Right-click the new part and ensure that


the construction surfaces are displayed by selecting Show Construction Surfaces.

- ▶ Click the Select tool, and then select the interference part to highlight the interference volume in the assembly window.

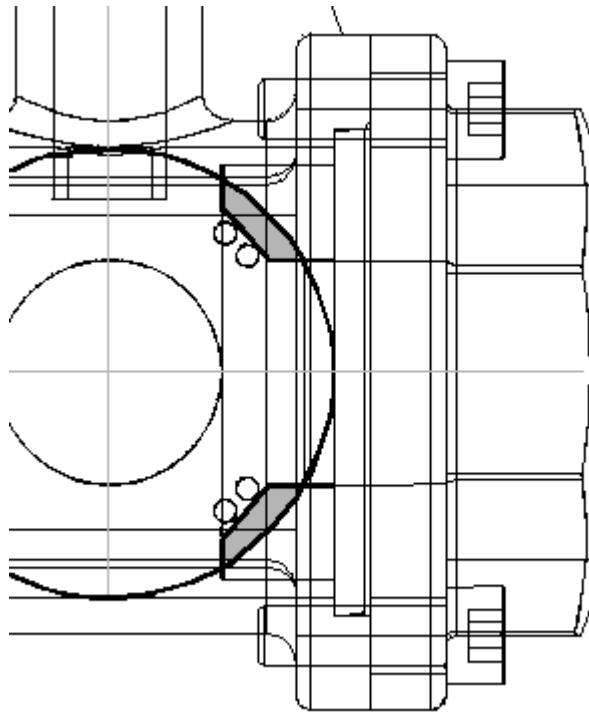


*Find the interference*

The highlighted interference is between the ball and the end caps. Modify the end cap to remove the interference.

- ▶ Shade the assembly using the Visible and Hidden Edges command .

- ▶ Press **Ctrl+R** on the keyboard to rotate the assembly to a right view. Use the Zoom Area command for a better view of the interference.



### Measure the interference

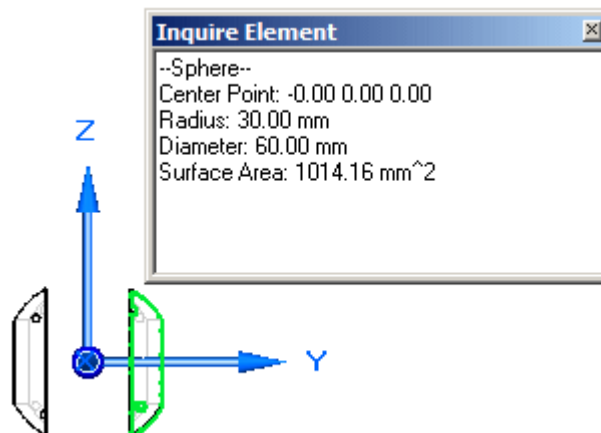
Measure the elements to help decide how to modify one of the parts.

- ▶ Right-click *Interference1.par* in PathFinder and choose Show Only.

- ▶ Choose the Inspect tab® 3D Measure group® Inquire Element command

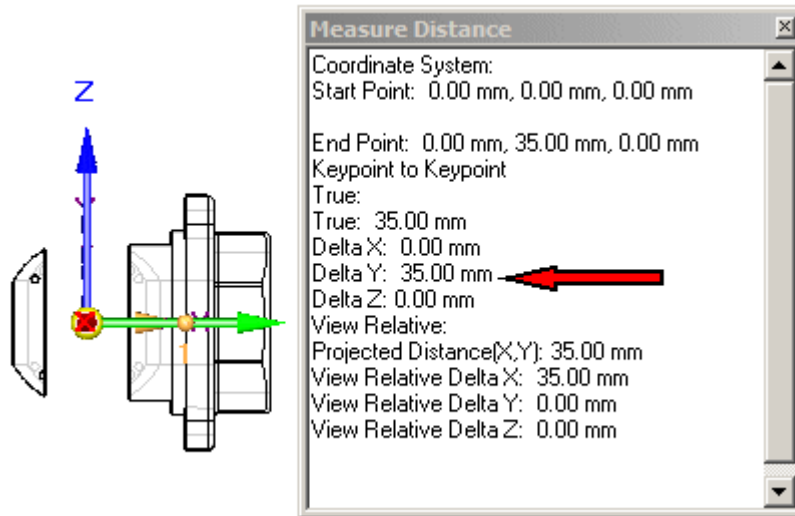



- ▶ Select the outside radius of the ball.

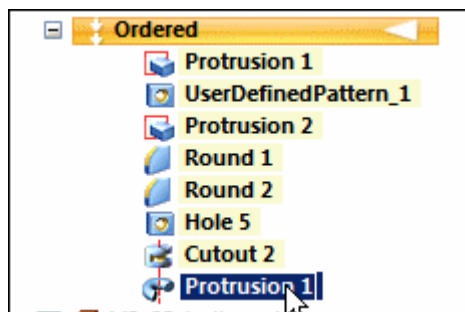



- ▶ Notice the radius is 30 mm. Remember this value when the end cap is modified.

- ▶ Right-click *valve\_endcap.par:2* in PathFinder and choose Show.
- ▶ Choose the Inspect tab@ 3D Measure group@ Measure Distance command. Using keypoints, measure from the radius of the ball to the midpoint on the mating surface of the end cap. To select the center of the sphere, select the circular edge shown, then select the face of the end cap shown.

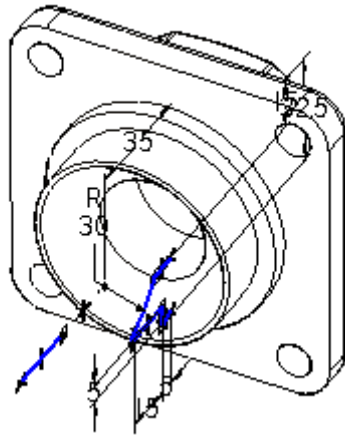
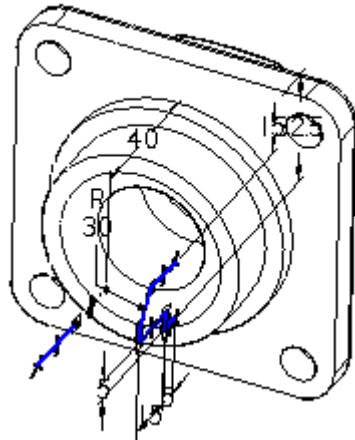



- ▶ Notice the distance value is 35 mm. Remember this value when editing the end cap. Click Close to exit the measuring command.
- ▶ Click the Select tool command, right-click the right end cap named *valve\_endcap.par:2* in PathFinder, and then click Edit.
- ▶ Press **CTRL+I** on the keyboard to return to the isometric view.
- ▶ Choose the View tab@ Show group@ hide Previous Level command .
- ▶ At the bottom of the list in feature PathFinder, click the revolved protrusion.



- ▶ On the command bar, click the Dynamic Edit button .

- ▶ Select the 40 mm dimension on the profile, and change it to 35 mm.



- ▶ On the Home tab, click Close and Return .
- ▶ Display all the parts in the assembly.
- ▶ Click the Select tool command, and in Assembly PathFinder highlight the interference part. Press the Delete key.
- ▶ Run the interference checking against the handle subassembly and all other parts. No interference should be found.
- ▶ Save the file.

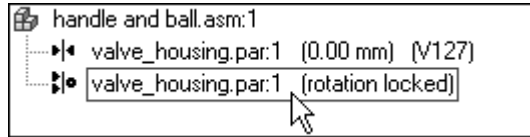
#### *Unlock the handle and reposition*


Now that there is no interference between the ball and the housing, rotate the handle to a new position. Unlock the axial relationship locking the rotation, and then reposition the handle.

- ▶ In Assembly PathFinder, select the *handle and ball.asm* subassembly.





- ▶ In the bottom pane of Assembly PathFinder, click the axial align relationship.



- ▶ On the command bar, click Unlock Rotation , and then click OK. Notice that the fixed offset used by the original insert relationship is gone.

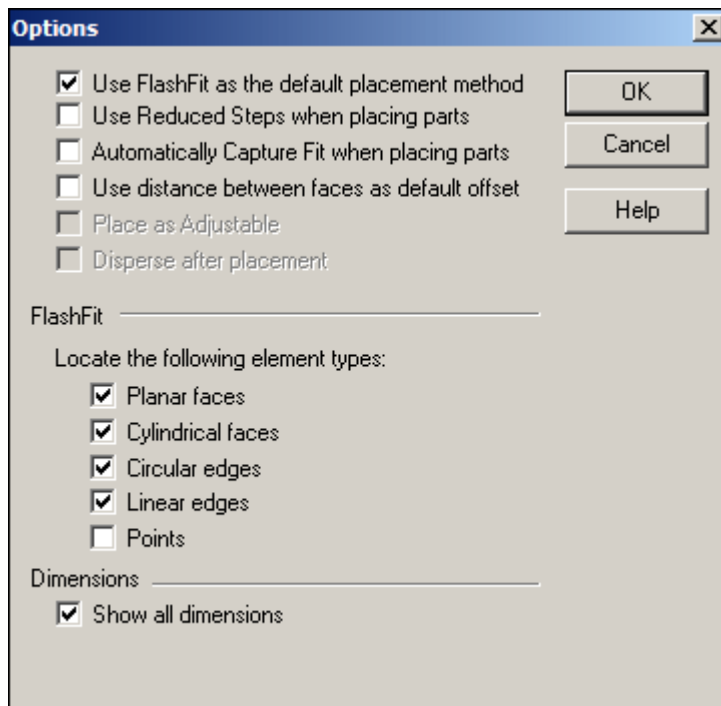
- ▶ Click Edit Definition .

- ▶ On the command bar, set the relationship type to the Angle relationship .

- ▶ Click the Options button .

**Note**

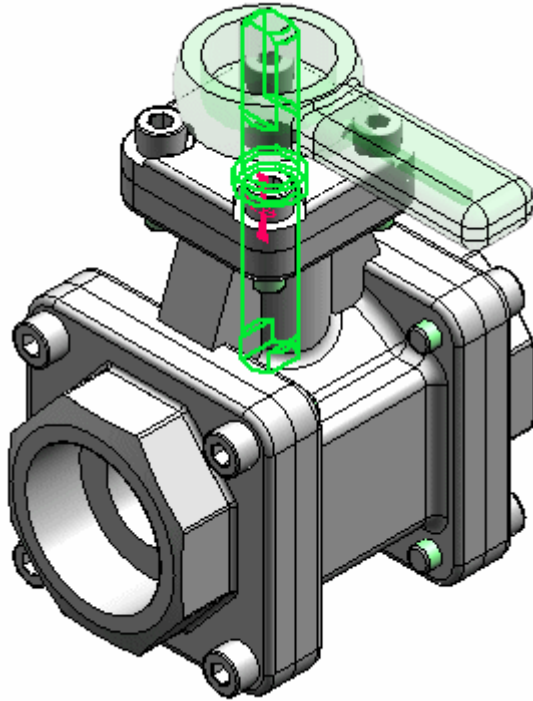
Make sure the reduced steps option is cleared.



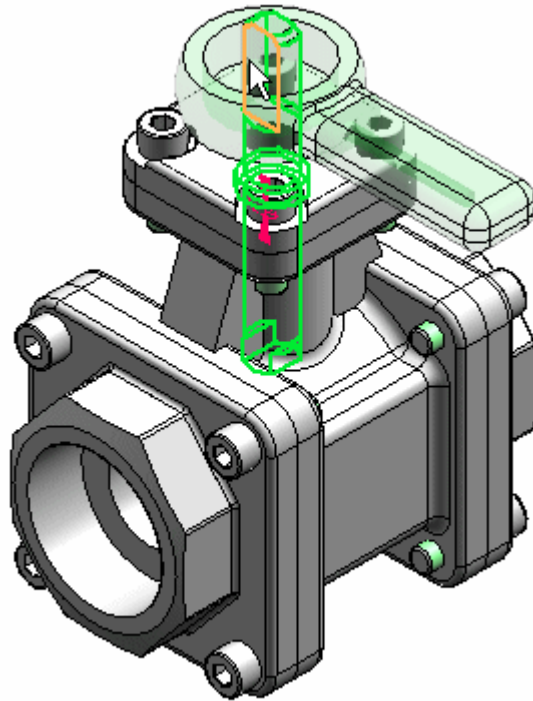
- Using QuickPick, select *valve\_shaft.par:1*.

**Note**

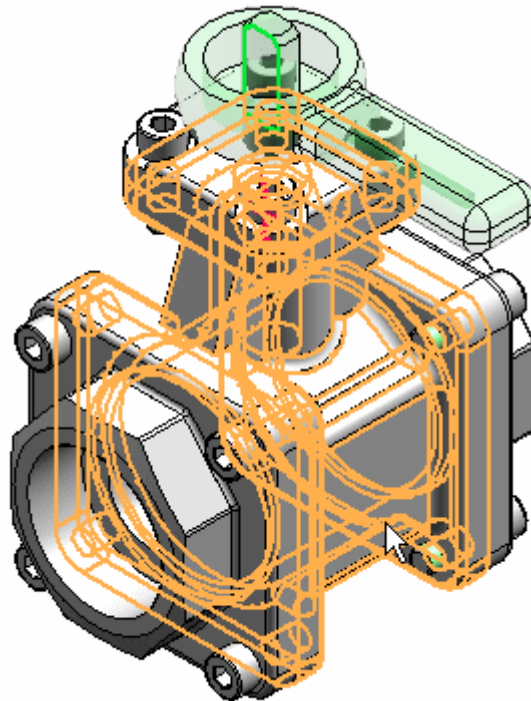
If you have trouble picking *valve\_shaft.par:1* it may not be active in the subassembly. Use the Activate button on the command bar.



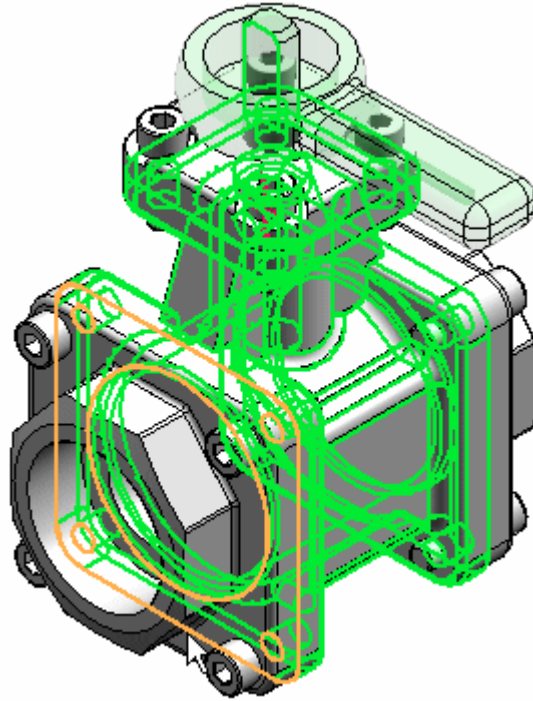
- ▶ Select the left planar side of the shaft.



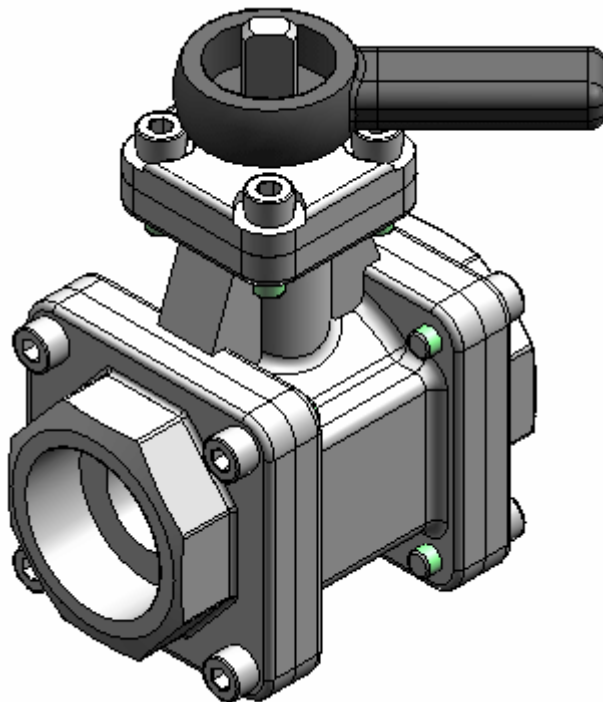
- ▶ Select the main housing.



- ▶ Select the left side of the housing.



- ▶ Type 45° for the rotation angle.



- ▶ Change the angle value and watch the handle and ball subassembly rotate.
- ▶ Save and close the file. This completes the activity.

*Activity Summary*

In this activity you learned how to check for interference between parts and subassemblies. You learned how to determine the extent of the interference and make the modifications necessary to correct the interference.

**Lesson review**

Answer the following questions:

1. After selecting a set of assembly components to check for interference, what options are available to perform the interference detection?
2. After the interference is detected, what options exist for analyzing the interference?

## **Answers**

1. After selecting a set of assembly components to check for interference, what options are available to perform the interference detection?

The options are:

- a. Select set 2
- b. All other parts in the assembly
- c. Parts currently shown
- d. Itself

2. After the interference is detected, what options exist for analyzing the interference?

The options are:

- a. Generate Report
- b. Create interfering volumes and show them or save them as a part.
- c. Hide parts not in the select sets
- d. Highlight interfering parts
- e. Dim parts with no interference
- f. Hide parts with no interference
- g. Ignore interference between matching threads
- h. Ignore threaded fasteners interfering with non-threaded holes

**Lesson summary**

In this lesson you learned how to check for interference between parts and subassemblies. You learned how to determine the extent of the interference and make the modifications necessary to correct the interference.



## **Lesson review**

1. What do the check interference options control?
2. Where are motion tools located in Solid Edge assembly?
3. When setting up the assembly for motion, which command steps you through the motion process?
4. What are the two types of sensors available in the assembly ?
5. Describe the difference between true/false range and horizontal range sensors.

## Lesson summary

- When creating an assembly, at some point you must test the design to see if it is valid. Solid Edge provides tools to calculate physical properties, analyze part motion, and check for minimum distance. Using these tools, you can quickly identify potential problem areas and correct your design.
- The physical properties dialog box, located under the inspect tab, activates graphics in the assembly view to illustrate physical properties.
- The motion commands in Solid Edge assembly set parts in motion so you can analyze the motion and check for part interference. Motion is found under the tools tab.
- Sensors help monitor critical design elements in assembly design applications where parts must maintain a minimal clearance from others. Sensors can monitor required conditions and warn when a condition is not met. The sensors tab is located on PathFinder.
- Interference can be checked against a subset of parts or the whole of the assembly. A part file representing the common volume defining the interference can be produced to see the shape of the interference, and to measure overlapping volumes in order to fix the problem.

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**Lesson**

**7** *Replacing parts in an assembly*

## **Replacing parts in an assembly**

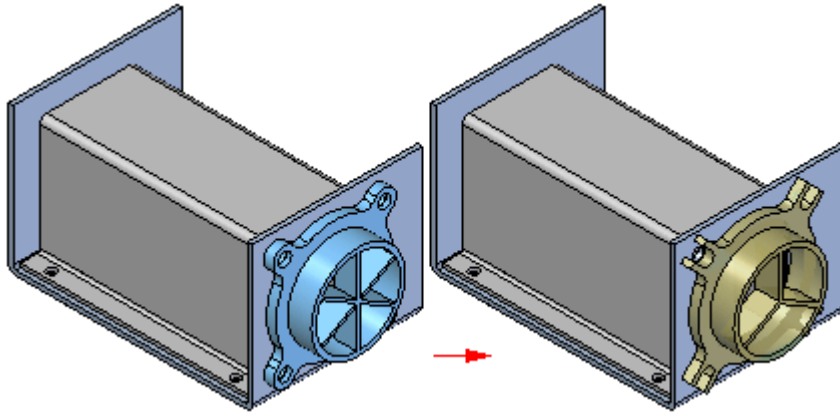
It is possible to quickly replace a part in an assembly with another part or subassembly that has similar geometry.

## Replacing parts in assemblies

At times you may need to replace a part or subassembly in an assembly with a new part or subassembly. For example, a part may be placed in multiple assemblies, and at a later time, the part needs to be redesigned for one assembly only. After creating a new version of the part, you can replace the existing part using the Replace command.

### Tip

In the Teamcenter-managed environment, you should use the Revisions command to replace an item with a different revision of the same item.



## Replacing similar parts

When you replace a part with a new version of the same part, Solid Edge attempts to use the existing assembly relationships to position the new version. However, if any faces used to position the old part were consumed when the part was modified; assembly relationships can fail. If this occurs, you can delete the affected assembly relationships using PathFinder, and then apply new relationships to completely position the new part.

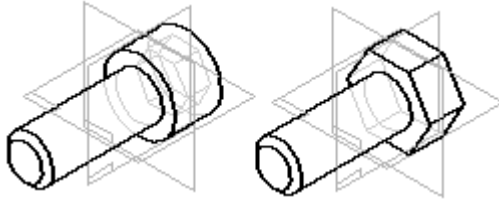
You can also use a member of a family of parts as a replacement part. If the current part is not a family of parts member, and you want to replace it with a family of parts member, use the Replacement Part dialog box to select a family of parts member as the replacement part.

If the current part is a member of a family of parts, the Family of Parts Member dialog box is automatically displayed when you select the part you want to replace. You can then specify another family member you want to use as the replacement part. If you want to replace the family of parts member with any other part, click the Browse button on the Family of Parts Member dialog box to display the Replacement Part dialog box. This allows you to select any part as the replacement part.

### Replacing dissimilar parts

When you replace a part with a different part, one that was created independently of the part being replaced, Solid Edge compares the geometry of the two parts. If the geometry matches sufficiently, the replacement part is positioned properly.

When replacing dissimilar parts, the original part and the replacement part are required to be in the same relative orientation in their respective part files.



### Replacing one occurrence of a part

If the part you are replacing has been placed more than once in the active assembly, you can specify whether to replace all occurrences of the part, or only the selected one.

### Replacing parts in subassemblies

The Replace command can be used to replace parts in the active assembly only. If the part you want to replace is in a subassembly, you will have to in-place activate the subassembly before replacing the part.

### Replacing subassemblies

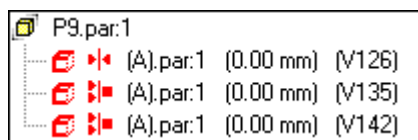
When you replace a subassembly, you can replace it with another subassembly or you can replace it with a part. If the current or replacement subassembly is also a family of assemblies, you can use the Assembly Member dialog box to specify which family member you want to use as the replacement.

#### Note

For more information on working with families of assemblies, see the Family of Assemblies and Alternate Position Assemblies topic in online Help.

### Failed relationships during part replacement

Assembly relationships can fail when you are replacing parts. Normally, this happens because the assembly cannot locate the faces in the new part that it was connected to in the old part. When this occurs, a symbol displays to the left of the failed relationship in the lower pane of the PathFinder.



### Replacing renamed occurrences

When you replace an assembly occurrence whose default name in PathFinder has been overridden using the Rename or Occurrence Properties commands on the

PathFinder shortcut menu, you can control whether the replacement occurrence uses the new document name or the overridden name in PathFinder.

When you set the Use Default Placement Name During Replace Part option on the Assembly tab on the Options dialog box, the document name of the replacement occurrence is used. When you clear this option, the overridden name is maintained.

When you override the name of an assembly occurrence, the actual document name is not changed.

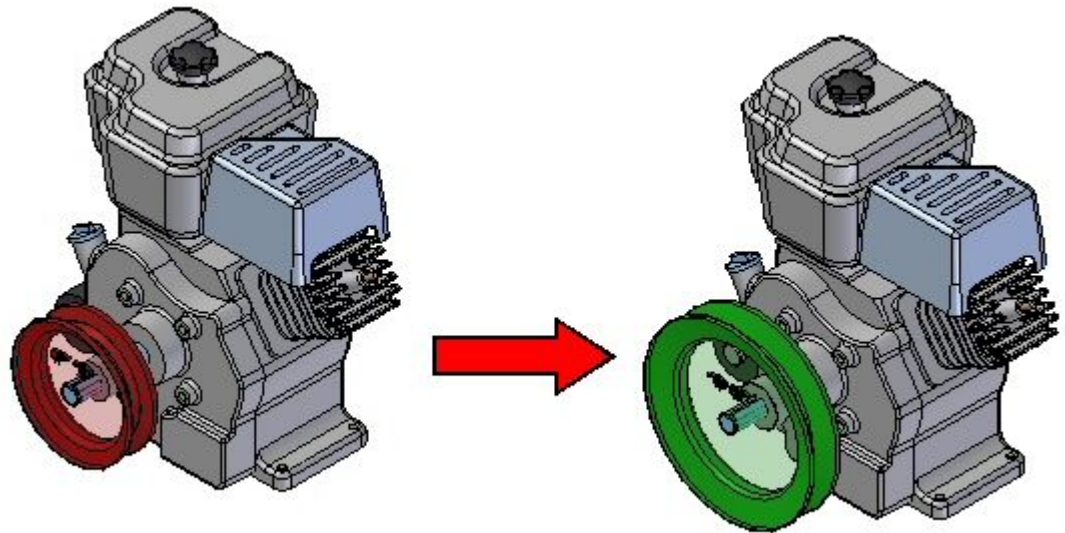
**Activity: Replacing parts in an assembly**

**Overview**

The objective of this activity is to show how to replace a part or subassembly in an assembly.

**Activity**

In this activity you will replace a subassembly with a similar subassembly inside of the top level assembly.





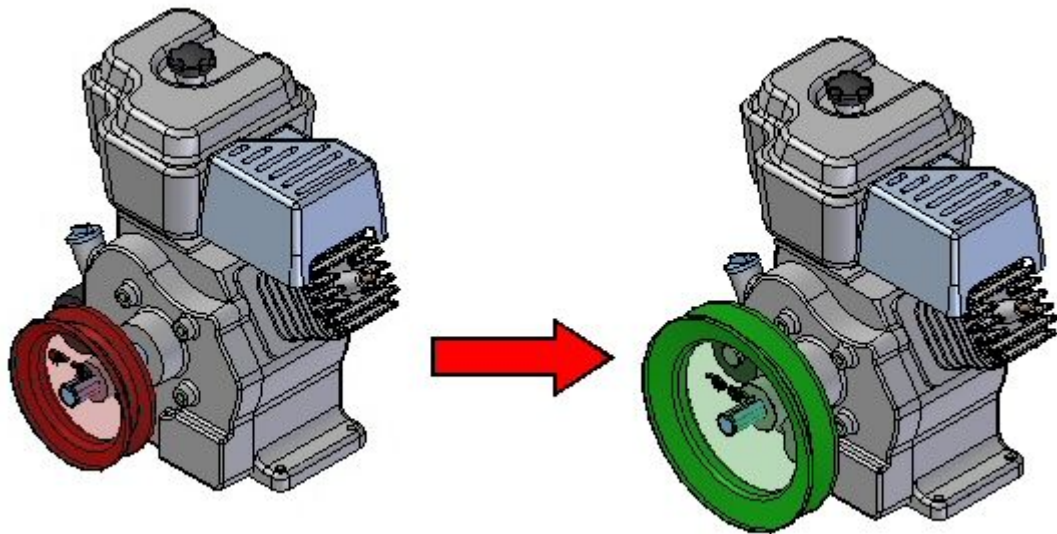
**Activity: Replacing parts in an assembly**

**Overview**

This activity demonstrates how to replace a part in an assembly.

**Objectives**

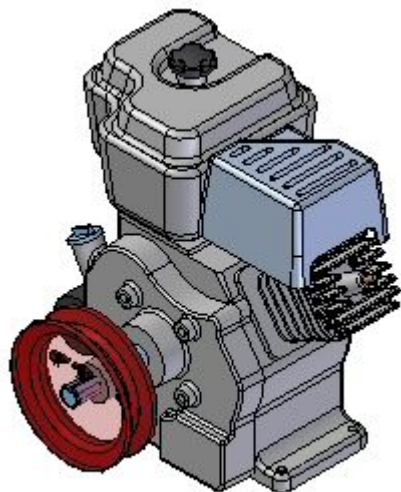
You will replace a subassembly in an assembly.



*Open an existing assembly with all the parts active*


The assembly you will open has a small pulley subassembly that you will replace with a large pulley subassembly.

- From the Solid Edge start screen, click Open Existing Document. Browse for *Engine.asm* in the folder where the activity files are located.

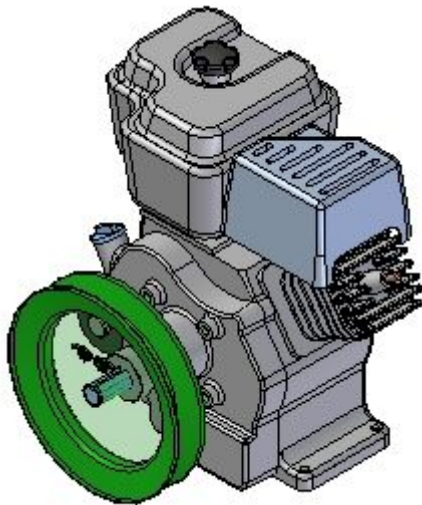


### Replacing the subassembly

The subassembly *small.asm* will be replaced with *large.asm*.

- In the Home tab, in the Modify group, choose the Replace Part command .
- In Pathfinder, select *small.asm* as the subassembly to replace.
- In the replacement part dialog box, browse for the assembly *large.asm* in the folder where the activity files are located, and then click open.

The subassembly has been replaced.



- Save and close the assembly. This completes the activity.

### Summary

In this activity you learned how to replace a subassembly within a top level assembly.

## **Lesson review**

Answer the following questions:

1. Can you replace subassemblies within an assembly?
2. Can parts with dissimilar geometry be replaced?
3. Can you replace parts in a subassembly, and if so, how?
4. Can you replace members when creating a family of assemblies?

**Answers**

1. Can you replace subassemblies within an assembly?

Subassemblies as well as parts can be replaced in a top level assembly.

2. Can parts with dissimilar geometry be replaced?

Parts with dissimilar geometry can be replaced, however failed relationships are likely to occur. These would have to be repaired.

3. Can you replace parts in a subassembly, and if so, how?

Parts can be replaced in a subassembly by in place activating the subassembly and then replacing the part.

4. Can you replace members when creating a family of assemblies?

When creating a member of a family of assemblies, replacing a part as an occurrence override.

## **Lesson summary**

In this lesson you learned how to replace a subassembly within a top level assembly.



---

**Lesson**

**8** *Designing in the context of  
an assembly*

## Top-Down and Bottom-Up Design in Solid Edge

Mechanical design and engineering requires careful planning and an analytical approach to developing new products. The purpose of CAD/CAE/CAM is to reduce the design cycle length, and eliminate fit errors by taking better advantage of common part geometry. When designing assemblies using computer-aided design tools, there are two basic ways of categorizing assembly design: top-down design and bottom-up design.

### Top-Down assembly modeling

Top-down assembly modeling is an assembly-centric modeling method where the assembly design is started at the highest level possible, and individual parts and subassemblies are defined within the context of the overall assembly. With this approach, an assembly layout is typically created first, and this assembly layout is used to define individual part geometry and position.

This approach is often used at companies where the product being designed is large enough that it requires many people to complete the design. A senior-level designer might create the initial assembly layout, then divide the assembly layout into logical subassemblies and parts for the remainder of the organization to complete.

### Bottom-Up assembly modeling

Bottom-up assembly modeling is a part-centric modeling method where the assembly design is started with a principal structural or functional element, and individual parts are designed in relative isolation from the overall assembly. Component parts and subassemblies are defined as the process moves up towards the top-level assembly. With this approach, as the design of a key component is completed, its geometry may or may not be used to aid the design of related mating components.

This approach is often used at companies where the product being designed is small enough that one or only a few people are needed to complete the design.

### Combining both approaches

Solid Edge provides tools that allow you to take advantage of the benefits of both approaches as needed. Many organizations use a combination of both methods, using the method which best suits the immediate requirements. For example, you can use the top-down approach to create the initial assembly layout and to define the document structure needed. You can then copy the assembly layout geometry to subassembly and part documents to divide the work among the organization.

You can shift to the bottom-up approach in areas of the design that use purchased parts, existing parts from an earlier project, or where you are modeling standard parts in 3D that were created on an earlier 2D CAD system.

The suite of commands and tools in Solid Edge also allow you to use either approach associatively or non-associatively, as you see fit.



## **Top-Down tools**

The primary top-down assembly design tool within Solid Edge is virtual component modeling, available in the Assembly environment. The [Creating and Publishing Virtual Components](#) Help topic discusses this functionality in depth.

If you choose not to use virtual component modeling, you can use the Copy Sketch command to copy layout graphics to parts and subassemblies, either associatively or non-associatively.

The Create In-Place option on the Parts Library tab of PathFinder allows you to create new parts associatively or non-associatively within the context of the assembly. The Constructing New Parts within an Assembly Help topic discusses this functionality.

You can control whether the new parts are associative to existing parts using the suite of [Inter-Part Associativity](#) tools available. The options on the Inter-Part tab on the Options dialog box allow you to control what types of associativity you want to use.

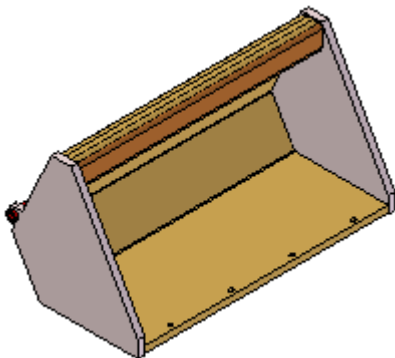
The Include command also allows you to associatively or non-associatively copy geometry between documents when working in the context of an assembly.

Many feature commands allow you to define the extent of the feature associatively by selecting a keypoint on another part in the assembly.

## **Bottom-Up tools**

With the bottom-up approach, you can create new 3D models in relative isolation from the assembly by referencing existing paper or electronic drawings, or by designing entirely new components.

A variation of the bottom-up approach involves using the Part Copy command on the Insert menu to associatively or non-associatively copy surface geometry from one 3D model to another 3D model outside the context of an assembly. This approach is especially useful when working with tightly related components that share common characteristics, such as parts that make up a weldment assembly.



When used associatively, the Part Copy command allows you to control common geometry on several child parts from one or more parent parts. You can edit the parent document, then open and update the child documents without creating or referencing an assembly document.

When used non-associatively, part copies allow you to quickly reuse existing geometry in another document.

**Bringing it together**

With either approach, you usually will want to view the components within one or more assemblies. You can use assembly relationships to position the components in the assembly, or you can use coordinate systems to define the position of each component in the assembly structure.

## **Transferring and dispersing assemblies**

The ability to transfer and create subassemblies within the tree structure of an assembly or to push the parts out of a subassembly into a higher level assembly is a powerful tool when managing and manipulating large assemblies.

## **Restructuring assemblies**

Solid Edge contains commands that allow you to change the structure of an existing assembly.

The Transfer command transfers parts and subassemblies from one assembly to another. You can transfer these parts and subassemblies to any level of the assembly that can be seen from the top level assembly that is open. You can also use the Create New Subassembly dialog box to create a new subassembly for the transferred files. To access the Create New Subassembly dialog box, on the Transfer to Assembly Level dialog box, click the New Subassembly button.

The Disperse command transfers the parts in a subassembly to the next highest subassembly and deletes the reference to the subassembly. The command disperses only the top-level occurrence of a subassembly. For example, if a subassembly exists as an occurrence within the assembly being dispersed, the subassembly remains unchanged, but is moved up to the next higher assembly level.

To change the order of the files within an assembly, you can drag and drop parts in the PathFinder.

## **Transferring parts between assemblies**

You can use the Transfer command to transfer assembly files, parts containing inter-part relationships, tube parts, and pattern parts. It is important that you understand how Solid Edge handles these transfers so you can avoid possible problems.

When transferring parts, it is very important that you have write access to all of the part and assembly files involved in the transfer.

Solid Edge handles relationships during transfers just like it would if you deleted a part from one subassembly and added it to another. It attempts to re-establish each positioning relationship exactly like it was before the transfer, with reference to the same reference part. If the reference part remains in the assembly tree below its new location, the relationship should be successfully re-established. If the reference part is not below the transferred part's new location, it will not be converted into a non-positioning relationship and the relationship is removed. You will not receive warnings for affected relationships during transfers, so you should be very careful when transferring parts with relationships. You might choose to add positioning relationships to any occurrence that becomes underconstrained after its transfer.

If you transfer a part that is a parent of an inter-part relationship, the link is broken, but not deleted. Inter-part Manager will show the inter-part link as broken, just as if the parent part was deleted from the assembly. You will not receive a warning when the parent part is deleted. You must understand the relationship dependencies within your assembly so that you can avoid breaking links when you transfer parts. You will be warned if you transfer a part that is inter-part child. If you continue with the transfer, the link will be broken and automatically deleted by the command.

Pasted variable links will remain intact during transfer as long as both the parent and child remain anywhere in the assembly.

If you transfer a tube part containing the port that defines a path, the link will be deleted and you will not receive a warning message. If you transfer a part containing a port to a level above the assembly containing the tube path, the tube path becomes non-associative to the part and you will not receive a warning message.

If you transfer a part containing a feature pattern that drives the assembly pattern, the pattern disappears and you not receive a warning message.

### **Transferring parts to a new subassembly**

Solid Edge allows you to create a new subassembly for parts you want to transfer. The New Subassembly button on the Transfer to Assembly Level dialog box accesses the Create New Subassembly dialog box. You can use this dialog box to specify a template, file name and location for the new database. You can also use the dialog box to define the position of the transferred parts in the new subassembly.

You have two options when defining the part position.

- Position First Selected Part at Origin and Others Relative to It
- Maintain Current Offsets From Assembly Origin

The first option specifies that if the new subassembly is opened outside of the parent assembly, the parts will be positioned relative to the global reference planes so that when you fit the view, the parts will not be remote from the reference planes. This options provides results similar to creating a new assembly with existing parts. For example, when you create a new assembly and drag the first part in from Parts Library, it is grounded at the origin of the assembly file. The subassembly is then positioned as a whole within the upper level assembly.

The second option specifies that you want to position everything relative to a single global origin. After the new subassembly is created, if you open the subassembly outside of the parent assembly and the fit the view, the parts might be remotely located from the global reference planes.

### **Transferring part occurrences between subassemblies**

If you transfer a part from one subassembly to another and there are multiple occurrences of one or both of the subassemblies within the assembly structure, it is very likely that the instances of the transferred occurrence will change. For example, if a part in subassembly A, which occurs only once, is transferred into subassembly B that occurs five times, the effect is that four instances of the transferred occurrence is added. Likewise, if there are more occurrences of the source subassembly than there are of the target subassembly, the number of occurrences instead could be reduced.

### **Things to consider when transferring parts**

There are several things you need to consider when transferring parts. It is important that you understand how Solid Edge handles these situations so you get the desired results from your transfer.

#### **Occurrence numbers**

The occurrence number of a occurrence after its transfer into the target assembly is the next consecutive number available for the file name that is transferred. If you transfer more than one of the same file name occurrences at the same time, the number that is assigned to each occurrence in the target assembly is determined by the order in which they are numbered in the source assembly.

#### **Display configurations**

Existing display configurations become invalid as parts are removed or added during part transfer.

### **Face styles of transferred parts**

If you transfer a part into a target assembly that contains a style that is assigned to the part in the source assembly, you must reapply the style after the transfer. If the target assembly does not contain the style assigned to the part in the source assembly, the part is assigned the Aluminum style.

### **Explode configurations**

Explode configurations become invalid as parts are removed or added during part transfer. In the Draft environment, drawing views will go out-of-date when parts are removed or added from the configuration.

### **Groups**

Groups are not maintained during part transfer. Solid Edge handles the transfer of groups the same as if the part was manually deleted from or added to the source assembly.

### **3D section views**

Since 3D section views contain a list of parts that are cut, they are affected during transfer. Solid Edge handles the transfer of 3D section views the same as if the part was manually deleted from or added to the source assembly.

### **Sensors**

Sensors are not maintained during part transfer. Solid Edge handles the transfer of sensors the same as if the part was manually deleted from or added to the source assembly.

### **Motion joints**

Motion joints are not maintained during part transfer. Solid Edge handles the transfer of motion joints the same as if the part was manually deleted from or added to the source assembly.

### **Physical properties**

Physical Properties are not maintained during part transfer. Solid Edge handles the transfer of physical properties the same as if the part was manually deleted from or added to the source assembly.

## **Dispersing subassemblies**

You can use the Disperse command to disperse a subassembly by reassigning the parts to the next highest subassembly and removing the reference to the existing subassembly. The command will disperse only the top-level occurrence of a subassembly. For example, if a subassembly exists as an occurrence within the assembly being dispersed, the subassembly remains unchanged, but is moved up to the next higher assembly level.

The command does not modify the dispersed subassembly on the disk. The part occurrences are copied to the next higher level and the reference to the subassembly

is deleted. When you save the top-level assembly, since it is no longer in the assembly structure, the dispersed subassembly occurrence is not saved.

If the subassembly being dispersed contains a pattern, the parts of the pattern are placed at the proper location in the next higher level and a ground constraint is placed on each of the parts. The parts will not be grouped in the PathFinder under a pattern node, but will be ordered the same in the next higher assembly.

If you disperse a subassembly containing a tube part, the tube part and other parts are transferred to the next higher level, but the dispersed subassembly on the disk is not affected. Therefore, the tube part is still associative to the path when you open the subassembly stored on the disk.

**Activity: Transferring and dispersing assemblies**

The objective of this activity is to demonstrate how the assembly structure can be altered without having to delete and replace parts and subassemblies manually.

In this activity you used the disperse and transfer commands to change the organizational structure of an assembly.



**Activity: Transferring and dispersing in assembly**

**Overview**

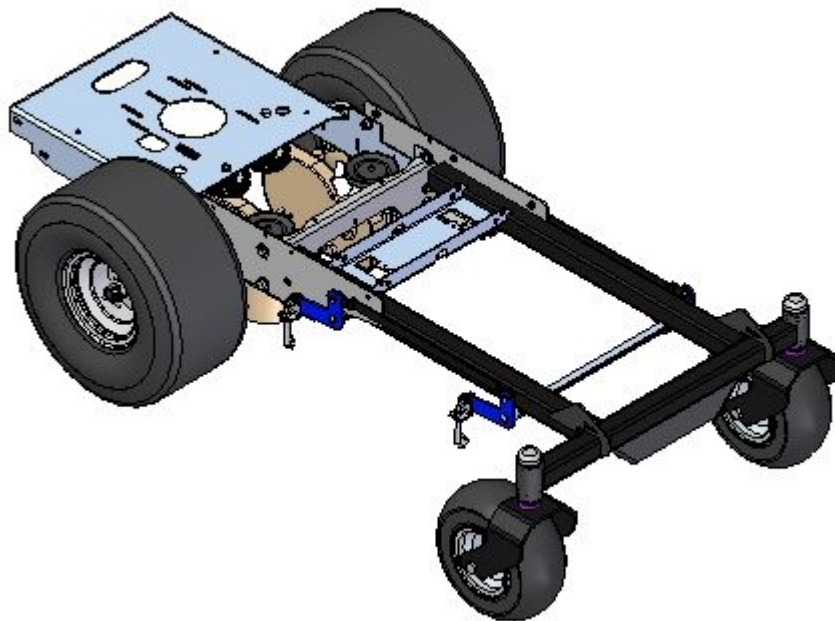
This activity demonstrates the process reorganizing an assembly structure using the disperse and transfer commands.

**Objectives**

You will manipulate an existing assembly to become familiar the disperse and transfer commands and some of the options associated with them.

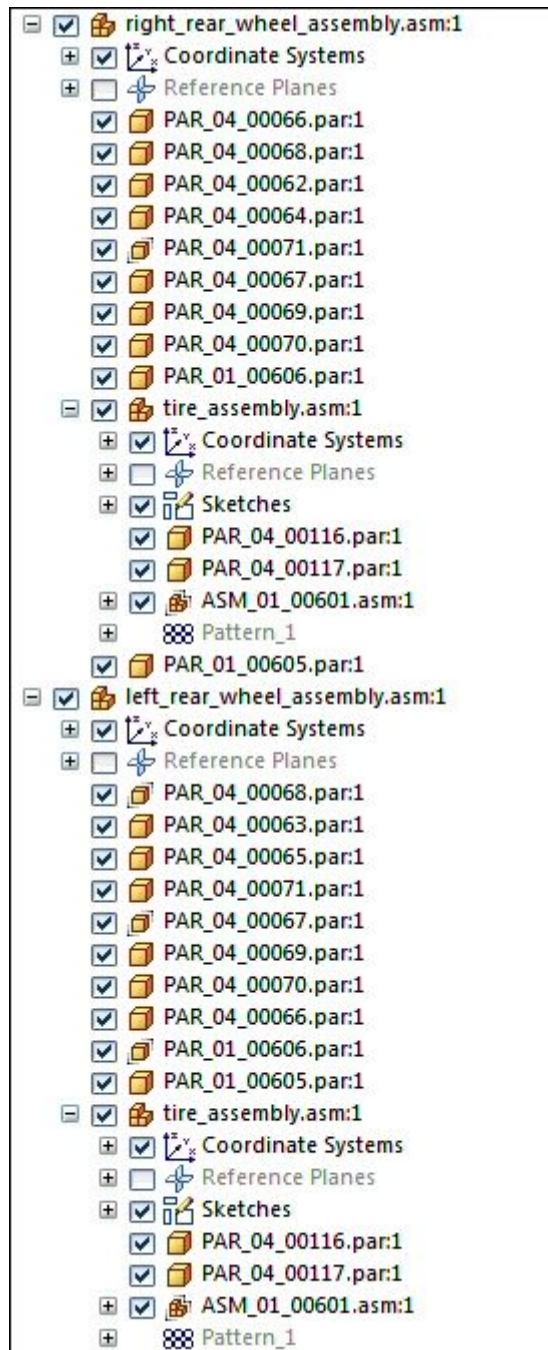
*Open the assembly*

- Open the assembly *Wheel\_Base.asm* with all the parts active.



Examine the assembly structure

- In PathFinder, expand the subassemblies *right\_rear\_wheel\_assembly.asm* and *left\_rear\_wheel\_assembly.asm* as shown.

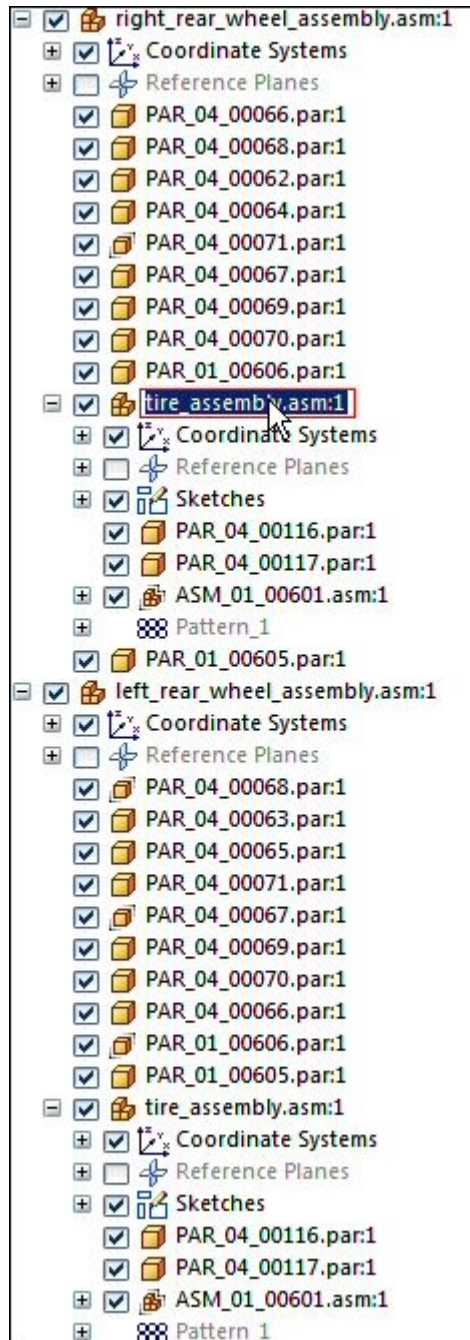


### Note

Notice an occurrence of the subassembly *tire\_assembly.asm* is present in both *right\_rear\_wheel\_assembly.asm* and *left\_rear\_wheel\_assembly.asm*

Disperse one of the subassemblies

- In the subassembly *right\_rear\_wheel\_assembly.asm* select *tire\_assembly.asm* as shown.



- ▶ Click the Home tab® Modify group® Disperse command.



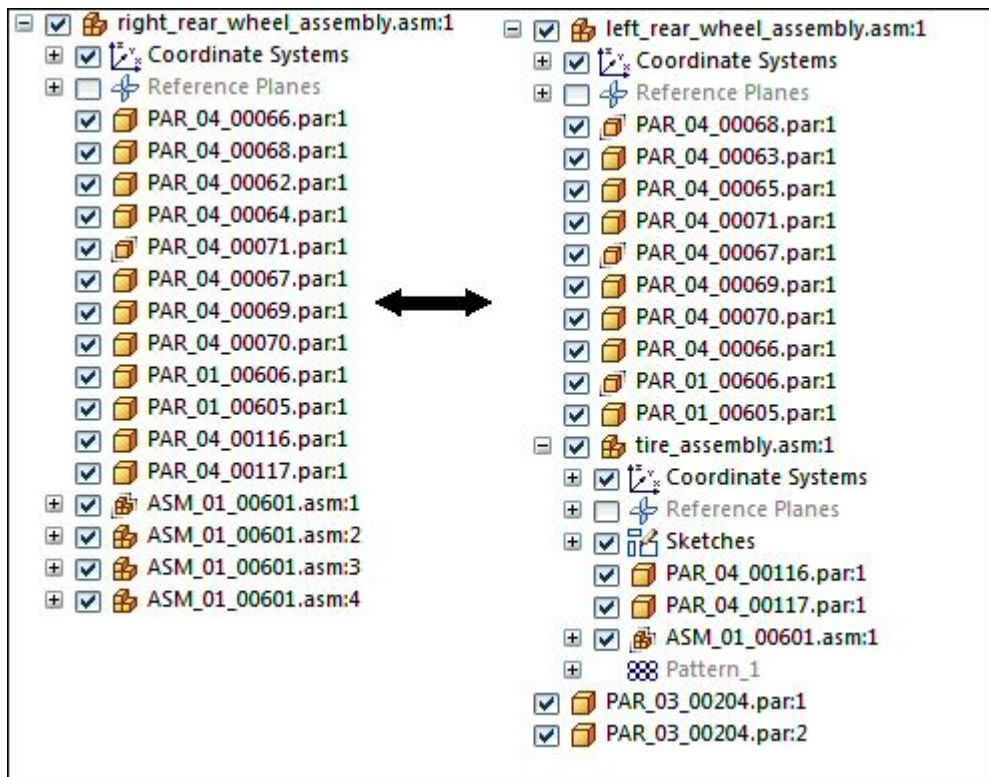
- ▶ When prompted: *Transfer the parts in the selected assembly to the next higher level, and delete the selected assembly occurrence?* Click Yes.
- ▶ When prompted: *The assembly that you are dispersing contains interpart relationships. If you continue, the interpart relationships could be broken. Continue?* Click Yes.

**Note**

When using Inter-part relationships, the changes in one part can control the size and shape of geometry in another part. When changes are made to the parent part. If this type of behavior is still desired in the assembly after executing the disperse command, check to see if the links were broken. If they were, you will need to reestablish them.

Examine the results of the disperse command

- Compare the differences between *right\_rear\_wheel\_assembly.asm* and *left\_rear\_wheel\_assembly.asm*. Notice:
  - That *tire\_assembly.asm* only exists in *left\_rear\_wheel\_assembly.asm* now.
  - That the parts that once existed in *tire\_assembly.asm* have been placed in the *right\_rear\_wheel\_assembly.asm*.
  - The pattern that had four occurrences of *ASM\_01\_00601.asm* is no longer a pattern, and those four occurrences have been placed in *right\_rear\_wheel\_assembly.asm*.



Transfer parts into a subassembly

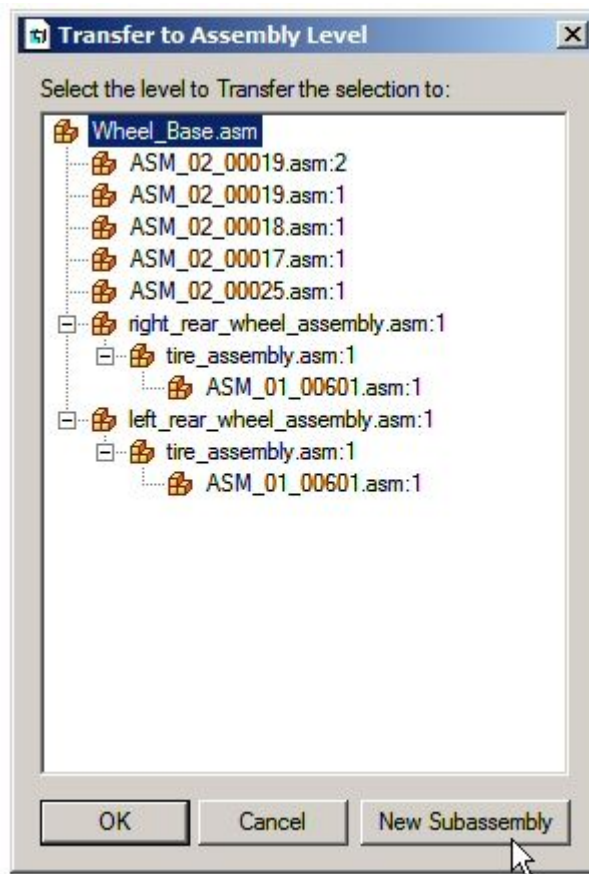
- Select the parts *hub.par* and *shaft.par* in Assembly PathFinder.
- Right-click on the selection in pathfinder, and then click Show Only to hide the rest of the assembly.
- Click Fit to fit the view.
- Select the parts *hub.par* and *shaft.par* in Assembly Pathfinder.

- Click the Home tab® Modify group® Transfer command.

**Note**

The Transfer command either moves the selected parts to a new location in the assembly structure, or combines the selected parts into a subassembly.

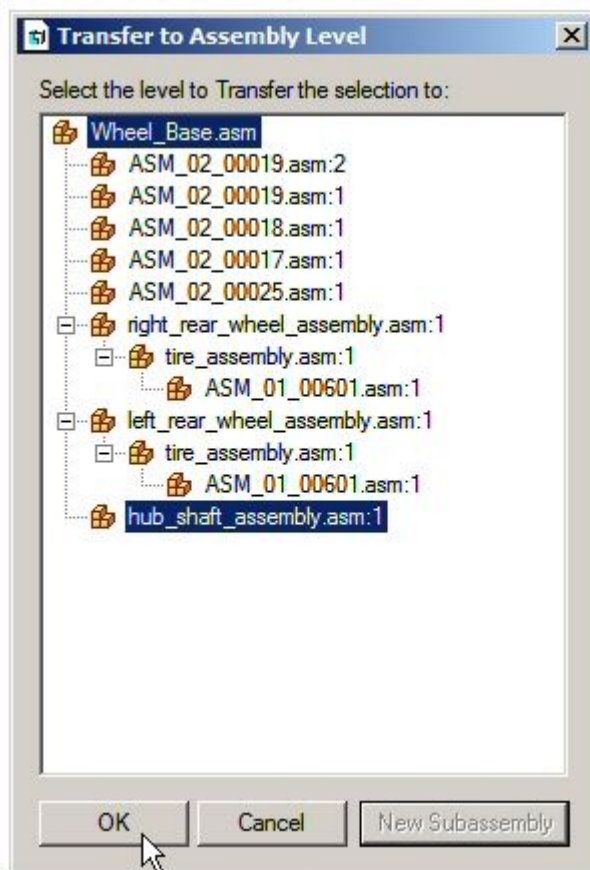
- Select the top level assembly, *Wheel\_Base.asm*, as the destination for the subassembly being created, then click New Subassembly.



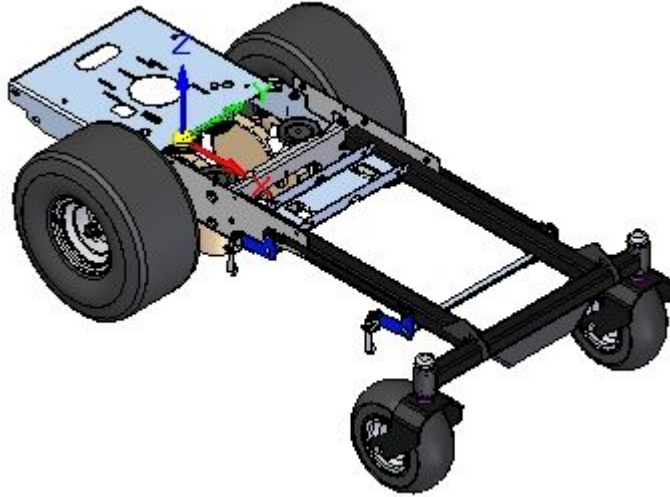
- ▶ Name the new subassembly hub\_shaft\_assembly.asm. Direct the subassembly to the folder where the remainder of the assembly resides as shown. Then click OK.



- ▶ View the destination of the new subassembly. If satisfactory, as shown, then click OK.



- ▶ Notice the new subassembly consisting of the two selected parts in the assembly pathfinder.
- ▶ Right-click on Wheel\_Base.asm in PathFinder and then click show to display the complete assembly, then fit the view.



- ▶ Save and close the assembly. This completes the activity.

### *Summary*

In this activity you learned the following

- How to move parts in a subassembly into a new location in the assembly structure with the disperse command.
- How to create a new subassembly from parts within an assembly using the transfer command, and how to locate the position in the assembly structure for the new subassembly to reside.



## **Lesson review**

Answer the following questions:

1. What does the transfer command do in an assembly document?
2. What does the disperse command do in an assembly document?
3. Can you disperse a subassembly?

## Answers

1. What does the transfer command do in an assembly document?

The Transfer command transfers parts and subassemblies from one assembly to another. You can transfer these parts and subassemblies to any level of the assembly that can be seen from the top level assembly that is open. You can also use the Create New Subassembly dialog box to create a new subassembly for the transferred files. To access the Create New Subassembly dialog box, on the Transfer to Assembly Level dialog box, click the New Subassembly button.

To change the order of the files within an assembly, you can drag and drop parts in the PathFinder.

2. What does the disperse command do in an assembly document?

The Disperse command transfers the parts in a subassembly to the next highest subassembly and deletes the reference to the subassembly. The command disperses only the top-level occurrence of a subassembly. For example, if a subassembly exists as an occurrence within the assembly being dispersed, the subassembly remains unchanged, but is moved up to the next higher assembly level.

3. Can you disperse a subassembly?

You can use the Disperse command to disperse a subassembly by reassigning the parts to the next highest subassembly and removing the reference to the existing subassembly. The command will disperse only the top-level occurrence of a subassembly. For example, if a subassembly exists as an occurrence within the assembly being dispersed, the subassembly remains unchanged, but is moved up to the next higher assembly level.

## **Lesson summary**

In this lesson you learned the following

- How to move parts in a subassembly into a new location in the assembly structure with the disperse command.
- How to create a new subassembly from parts within an assembly using the transfer command, and how to locate the position in the assembly structure for the new subassembly to reside.

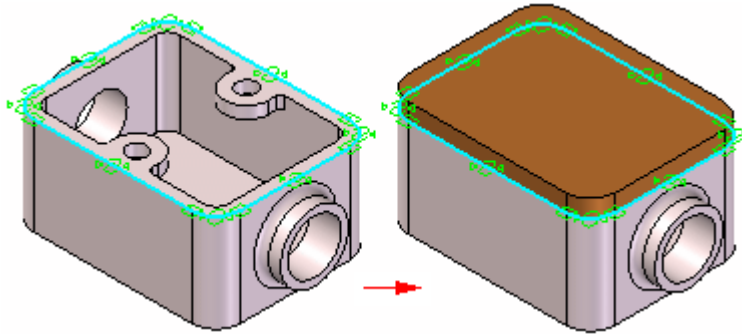
## **Inter-Part**

### **Overview**

Creating parts in an assembly whose size and shape can be controlled based on the size and shape of geometry in another part in the assembly is accomplished through Inter-part links. When the geometry of the parent part changes, the Inter-part links in the linked part alter the geometry accordingly.

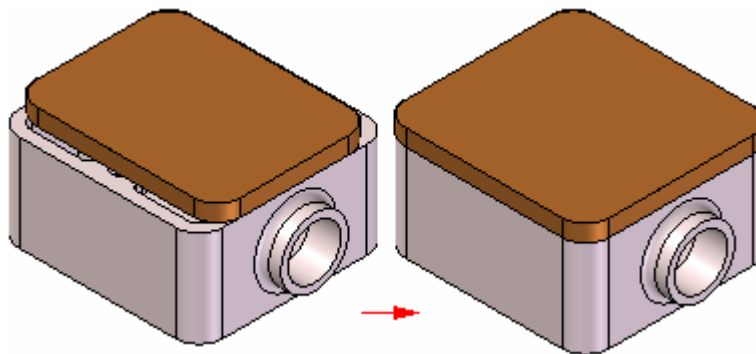
## Inter-Part associativity

When constructing the parts and assemblies for a design project, you can use the geometry on other parts in the assembly to help you construct a new part or subassembly. For example, you can use the Include command to create 2D geometry for the base feature of a new part by copying edges on an existing part.



Depending on the approach you use, the included geometry can be associative or non-associative to the original edges.

When you create new geometry associatively, then modify the original, or parent geometry; the child geometry also updates. If you change the size of the parent part, the included child geometry for the base feature also updates.



### Note

An Inter-Part Associativity tutorial is available that demonstrates how to create associative inter-part features.

### Note

When designing in the context of an assembly, only ordered features can be linked using Inter-part relationships. In the synchronous environment faces can be copied, but they are not linked to geometry contained in the part that the face was copied from.

The following commands and functions in Solid Edge allow you to use existing geometry associatively:

- Include command
- Inter-Part Copy command
- Assembly-Driven Part Features
- Reference Plane Definition
- Feature Extent Definition
- Variable Table

**Note**

Many of these options are available only when you set the proper inter-part associativity option on the Inter-Part tab on the Options dialog box.

When one part is associatively linked to another part in Solid Edge, special symbols are used to indicate the associative link. For example, when you include an edge from another part in the assembly to define the profile for cutout feature in the active part, a link symbol is displayed adjacent to the part feature in PathFinder and adjacent to the part entry in PathFinder.



These associative links between parts are called inter-part links to indicate that one part is dependent on another part for the definition of some of its geometry. The link information is added to the highest level assembly that is common to both parts, based on the assembly you opened.

For more information about managing inter-part links, see the Managing Inter-Parts Links section of this Help topic.

## **Including elements**

You can use the Include command to include edges from the active part, an assembly sketch, or the other parts in the assembly. When including edges from an assembly sketch or another part in the assembly, you can control whether the included edges are associative to the parent element using the Inter-Part Locate options on the Include dialog box.

You can only include elements from an assembly sketch or the other parts in an assembly when you are editing a part in the context of an assembly (you have in-place activated a part or you are creating a part in place).

When the Allow Locate of Peer Assembly Parts and Assembly Sketches option is set, you can locate and select elements in other parts and assembly sketches. To copy the elements associatively, you must also set the Maintain Associativity When Including Geometry From Other Parts in the Assembly option. When this option is cleared you can copy elements from other parts and assembly sketches non-associatively.

### **Note**

To include elements associatively between documents, set the Allow Inter-Part Links Using: Include Command in Part and Assembly Sketches option on the Inter-Part tab on the Options dialog box.

## **Inter-Part copies**

You can use the Inter-Part Copy command in the Part and Sheet Metal environments to associatively copy faces, features, and entire parts into another part document as construction geometry. You can then use the Include command to associatively copy edges from the construction geometry into a profile for a feature. To ensure that the Include command only copies edges from the associative construction geometry, turn off the assembly display using the Hide Previous Level command on the View tab.

The Inter-Part Copy command is only available when you are editing a part in the context of an assembly (you have in-place activated a part or you are creating a part in place).

### **Note**

To copy elements associatively between documents, set the Allow Inter-Part Links Using: Inter-Part Copy Command option on the Inter-Part tab on the Options dialog box.

## Assembly-Driven part features

You can use the assembly feature commands such as Cutout, Hole, and Revolved Cutout in the Assembly environment to construct assembly-driven part features in an assembly. You can specify which parts in the assembly you want to cut. Assembly-driven part features are added as a linked feature to each part document.

For more information on assembly-driven part features, see the [Assembly-based Features](#) Help topic.

### Note

To construct assembly-driven part features in an assembly, set the Allow Inter-Part Links Using: Assembly-Driven Part Features option on the Inter-Part tab on the Options dialog box.

## Reference planes

When constructing a feature for a part, you can use an assembly reference plane to define the new feature. If the assembly reference plane is modified, the feature associatively updates. To select an assembly reference plane, press the SHIFT key, then select the assembly reference plane.

You can only use an assembly reference plane when you are editing a part in the context of an assembly (you have in-place activated a part or you are creating a part in place).

### Note

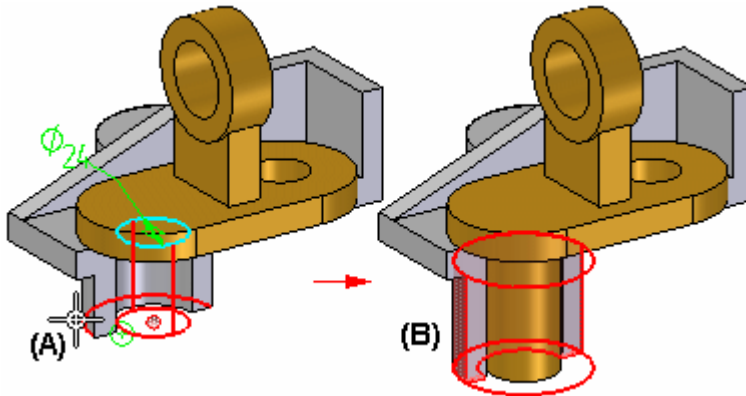
To use an assembly reference plane while constructing a part feature, set the Allow Inter-Part Links Using: Assembly Reference Planes In Feature option on the Inter-Part tab on the Options dialog box.



## Feature extents

When working within the context of an assembly, many feature commands allow you to select a keypoint on another part in the assembly to define the extent for a feature. For example, when constructing a protrusion in the Part environment, you can select a keypoint (A) on another part in the assembly during the Extent Step.

The extent for the feature is associative to the keypoint on the part you select. If the other part is modified (B) such that the keypoint location changes, the extent for the linked feature also updates.



You can also use an assembly sketch to define the extent for a feature.

## Variables

You can use the Variable Table in Solid Edge to associatively paste an assembly variable into a part or subassembly. This allows you to control several parts at once with one variable. For example, you can create an assembly variable to control the size of a hole in several parts in the assembly.

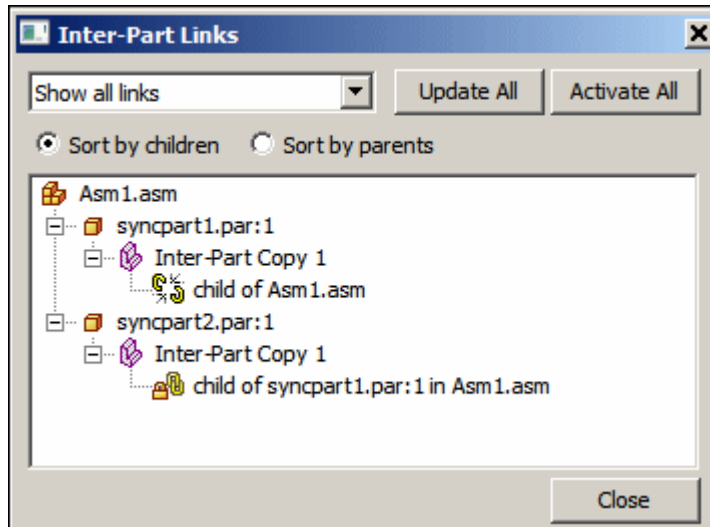
For more information on pasting variables between documents, see the Linking Variables Between Parts in an Assembly section of the Variables Help topic.

### Note














To associatively paste variables between documents, set the Allow Inter-Part Links Using: Paste Link To Variable Table option on the Inter-Part tab on the Options dialog box.

## Managing Inter-Part Links

You can use the Inter-Part Links dialog box to view and manage the inter-part links you create between the parts and assemblies for a design project. You display the Inter-Parts Links dialog box using the Inter-Part Manager command on the Tools tab.









### Legend

	Part
	Assembly
	Inter-part copy
	Variable
	Reference plane
	Sketch or Profile
	Feature Extent
	Link is intact
	Link status cannot be determined because parent is inactive
	Parent is not found
	Link to parent is broken
	Link has multiple solutions
	Link is out-of-context with its container assembly

The Inter-Part Links dialog box allows you to sort the inter-part links by parents or by children. When set to Children, each part or assembly that is a child in an inter-part relationship is displayed. Below each child is a description of the type of inter-part relationship and the name of the parent document. When set to Parents, each part or assembly that is a parent in an inter-part relationship is listed. Below each parent is the name of each of its children.

In the Inter-Part Links dialog box, special symbols are used to indicate the status of the associative links:

-  The link is intact and if design changes are made, the link should update properly.
-  The status of the link cannot be determined because the parent document is currently inactive. You can resolve this with the Activate All command on the shortcut menu to activate all the parts in the assembly with associative links.
-  Parent is not found. This can occur if, for example, you rename the parent file outside of [Revision Manager](#). This can be fixed by renaming the file back to its original name.
-  Link to parent is broken. This can occur if a feature that another part is dependent on is deleted. For example, a cutout feature in Part P1 is used to create an inter-part copy in Part P2. If you then delete the parent cutout feature in Part P1, a broken link symbol would be displayed adjacent to the inter-part copy listing for Part P2.
-  Link has multiple solutions. The current assembly relationship or inter-part link contains multiple solutions. To correct the problem, you can adjust the assembly relationships or use Inter-Part Manager to determine which links are affected and delete those links.
-  Linked document is out-of-context with its container assembly. When you create inter-part links between documents, the link information is contained in the highest level assembly that is common to both the child and parent documents, based on the assembly you opened. When you open a child document out-of-context with its container assembly, this symbol is displayed. See the Understanding In-Context Container Assembly section for more details.

This symbol does not indicate a problem, only that inter-part links in the child document may not update properly if the parent document is modified outside the context of the container assembly. You can do one of the following:

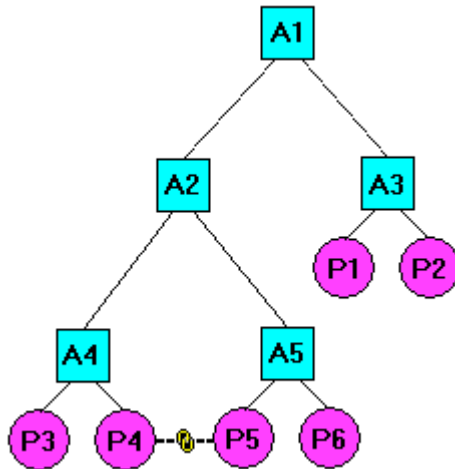
- Open the assembly listed in the Tooltip, then in-place activate the child document.
- Ignore the symbol.
- Break the link between the child document and parent document.

## **Understanding the In-Context Container Assembly**

When you create the first inter-part link for a part, the link information is stored in the highest level assembly possible that is common to both parts, based on the assembly you opened (using the Open dialog box or from Windows Explorer). A document can have only one in-context container assembly.

For example, you start a design session by opening assembly A2. You then in-place activate part P4. You then create a cutout feature using included geometry from part P5, which is in assembly A5. An inter-part link is created, with part P5 being the parent part, and part P4 being the child part.

Because the assembly you opened at the start of the design session was A2, A2 becomes the in-context container assembly for part P4. To create inter-part links for part P4 in the future, you must open assembly A2, then in-place activate part P4.

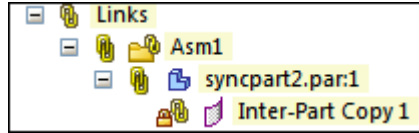


In a future design session, if you were to open assembly A1, the inter-part associativity options would be disabled. For example, on the Include Options dialog box, the Maintain Associativity When Including Geometry From Other Parts in the Assembly option would not be available.

You can determine the in-context container assembly for a part using the Inter-Part Manager dialog box. When you position the cursor over the inter-part link entry in the Inter-Part Manager dialog box, a Tooltip is displayed which lists the in-context container assembly for the part.

## Links Collection

The links collection in pathfinder can be expanded to show the links in the current open document.



RMB Links in pathfinder gives the following options:

- Expand All
- Collapse All
- Activate All
- Update All
- Freeze All
- Unfreeze All
- Break All
- Inter-part Manager

All links can be broken or froze from the top level in the links collector, or individually by RMB on the expanded items in the collection.

## Inter-Part Associativity and Inactive Parts

Before you make a design change to a part or assembly that is involved in an inter-part relationship, you should first use the Activate All command on the shortcut menu in the Inter-Part Links dialog box to activate the parts that contain inter-part relationships. The Activate All command only activates those parts that contain inter-parts links in the assembly.

## **Breaking Inter-Part Links**

If you want to delete any associative links between a child element and its parent, you can select the child element in the Inter-Part Links dialog box, then use the Break Links command on the shortcut menu. For example, you may want to break the associative links to a part that used to be unique to the assembly, but now will be used in other unrelated assemblies.

When you break the associative links to a part, you can still make design changes to the individual parts, but you may need to add dimensions or edit the feature to redefine the inputs for the feature. For example, if you define the extent for a feature using a keypoint on another part, then break the link later, you can still edit the extent for the feature.

After you break the associative link, you can select the feature, then use the Edit Definition option to access the Extent Step for the feature, then type a dimension value on the command bar to define a new extent for the feature.

### **Note**

When you define the extent for a feature associatively using a keypoint on another part, no driving dimension for the feature extent is created.

If you break the link and then edit the feature extent by typing an extent value, a driving dimension for the extent is created.

## **Updating Assembly Documents with Inter-Part Relationships**

The methodology by which inter-part relationships (inter-part copies, associative include from assembly sketch, associative reference planes, and multi-part cutouts) are managed has been enhanced in version 14. The new methodology is more reliable and flexible. Because of this enhancement, pre-version 14 assemblies with inter-part relationships should be opened in-context and saved before new design work is performed.

To do this, you must open the in-context assembly with write access to all affected documents (the child documents with inter-part relationships and the assembly) and then save the assembly.

## **Determining the In-Context Assembly**

In many situations, it can be difficult to determine which assembly is the proper in-context assembly. To make the update process easier, an Update dialog box is displayed when all the inter-part documents within an assembly cannot be updated. The Update dialog box explains the actions you need to take to successfully update all inter-part relationships.

There are three situations that can cause an inter-part occurrence to not be updated to the new compute methodology when you save the assembly:

- The inter-part child document is out of context with the assembly that controls the inter-part relationship.
- Either the child document or the in-context assembly is write-protected.
- Another unknown reason. This is usually due to a problem that existed in the previous version of the file (version 12 or earlier). One possibility is that the child part was replaced in the assembly and became out of context in V12. To resolve this particular problem, you can open the assembly and use Inter-part Manager to break any unresolved links.

If any of the three situations occurs when you open the assembly, an Update dialog box is displayed. If the Update dialog box is not displayed, the assembly and all inter-part occurrences within the assembly were updated successfully.

The inter-part relationships in an assembly are updated on an occurrence by occurrence basis. This means that, in some cases, only some of the inter-part child documents in the current assembly are updated, while others are not updated.

If the Update dialog box is displayed, this indicates there were some child documents in the current assembly that could not be updated. When this occurs, you should still save the current assembly. The Update dialog box will display messages that explain the actions you need to take to correct the situation.

Any child documents and related assemblies that were not updated will need to be addressed separately. This is usually done by opening the correct in-context assembly with write access to the child documents, and then saving that assembly.

**Note**

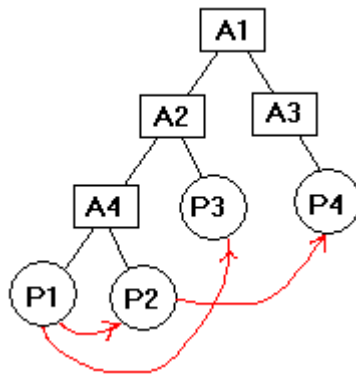
It is very important that you update all inter-part relationships before proceeding with further design work on the assembly and the inter-part occurrences. Any inter-part relationships that are not updated will not function properly, which can result in incorrect part positions and incorrect geometry on new design work.

The in-context assembly might be in a higher-level assembly than the open assembly in which the message is seen, or it could be a different assembly that is unrelated to the open assembly except for the presence of the out-of-context occurrence in both assemblies.

If a single inter-part relationship in an occurrence cannot be updated, no other inter-part relationships in that occurrence will be updated. A child document must be in-context of all of its inter-part relationships before the child document can be updated. This does not prevent other child documents in the assembly from being updated.

**Examples**

The following examples assume write access to all affected documents. Consider the assembly represented in the following illustration. When an update is attempted, the in-context assembly is the lowest assembly that is common to both the child and parent. (A = assembly, P = Part, the arrow points from child document to parent document)



If A4 is opened, no inter-part documents can be updated. In this example, two messages are displayed in the dialog box:

- File. P1 / Reason. The in-context assembly is A2
- File. P2 / Reason. The in-context assembly is A1

If A2 is opened, P1 will be updated, but there will be a message for P2:

- File: P2 / Reason. The in-context assembly is A1

If A1 is opened, all inter-part documents are updated, as this is the lowest level assembly that is common to all child and parent documents.

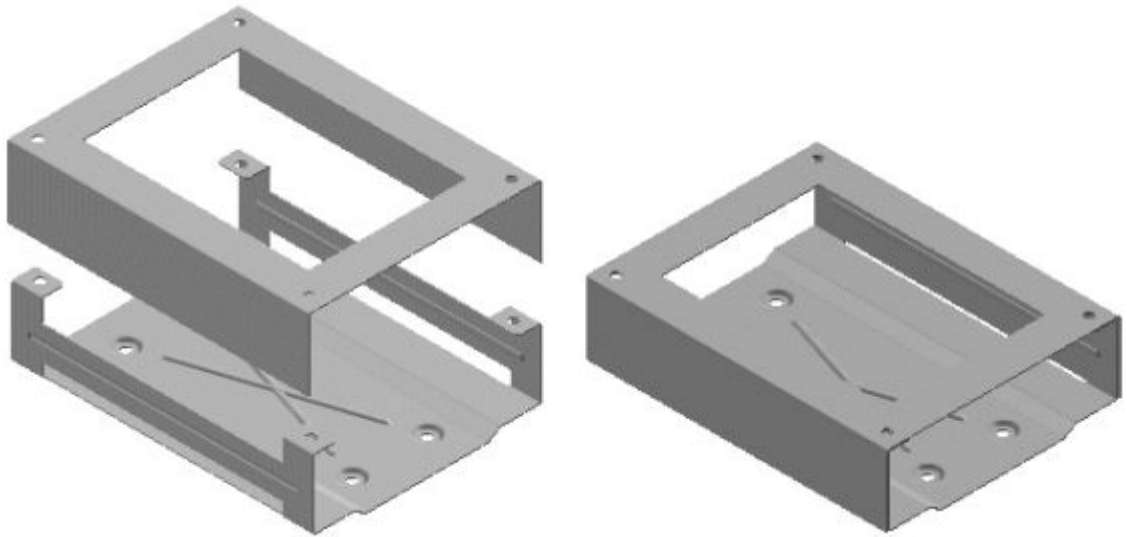


### **Activity: Inter-part assembly modeling**

When you complete this activity, you will be able to use inter-part modeling techniques to create new parts in an assembly.

For this activity, you will model a mating cover for the sheet metal chassis shown in the following illustration. The activity emphasizes the use of inter-part modeling in Solid Edge Assembly. Inter-part enables the modeling and mating of parts within an assembly to maintain like design characteristics between the parts. In other words, when one part is changed, the mating parts change accordingly. This is a top-down assembly modeling method.

Activity guides you through the process of associatively linking parts with Inter-part copies, so that changes in one part will be reflected in another part.



**Activity: Inter-part assembly modeling****Overview**

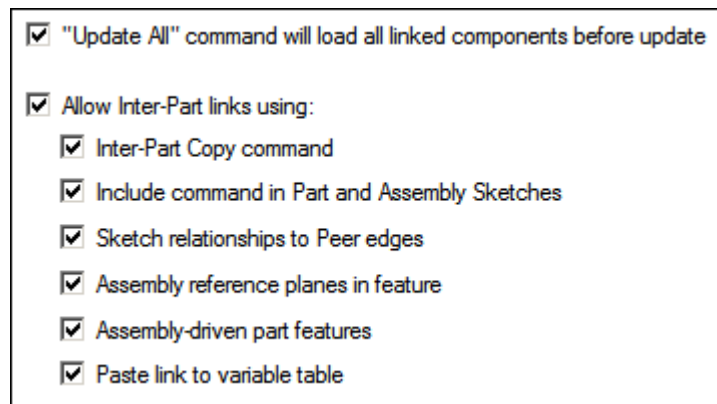
In this activity, an assembly will be created and a sheet metal part placed in the assembly. Another sheet metal part will be created from within the assembly and the geometry of that part will be controlled by the first part based on inter-part links

**Objectives**

The objective of this activity is to show how inter-part links can be used to link geometry so that changes in one part are reflected in another part via inter-part links. By doing this, the create in place command shows how to design in the context of the assembly. Having access to the geometry in adjacent parts in an assembly is a powerful design tool.

*Create a new assembly file and set the parameters for inter-part*

- ▶ Create a new metric assembly file.
- ▶ Hide all the assembly reference planes if they are shown.
- ▶ To allow inter-part links using inter-part copy, set the option in the Solid Edge Options dialog box. Click the Application button. Click Solid Edge Options. Click the Inter-Part tab, and select the options as shown. Click OK.



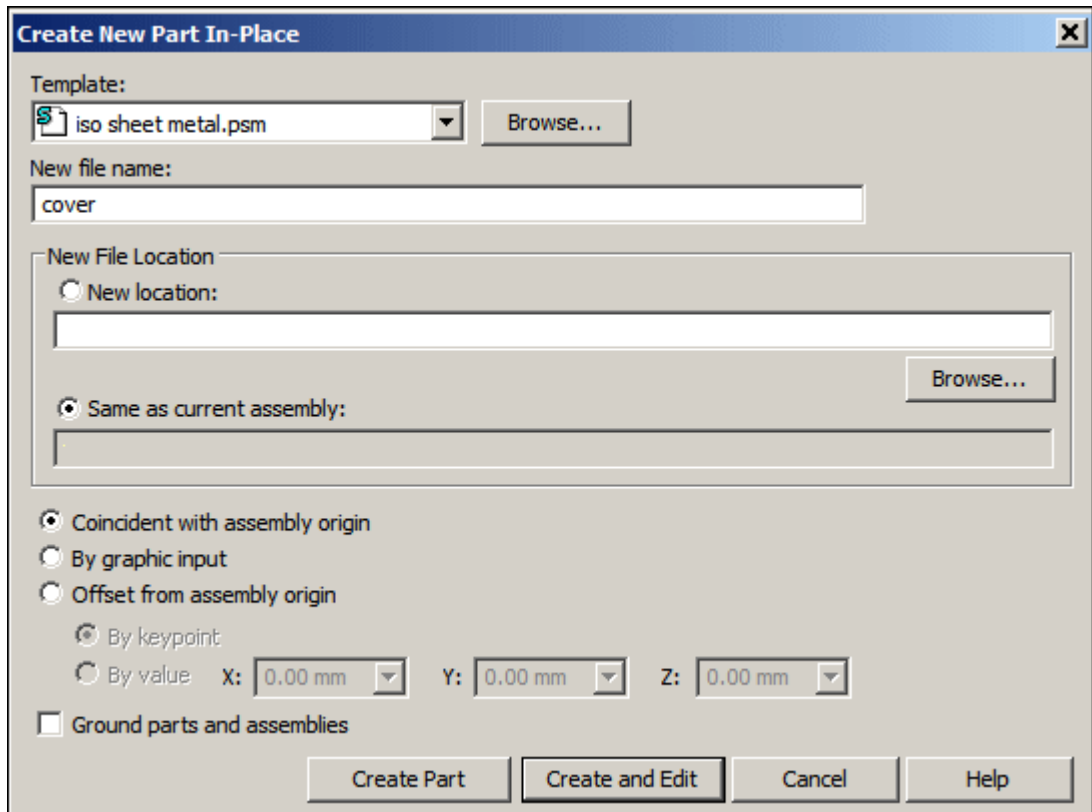
- ▶ Insert a part into the assembly. This part will be used as a reference part to construct the mating cover. In Parts Library tab, drag *Chassis.psm* into the main assembly window.



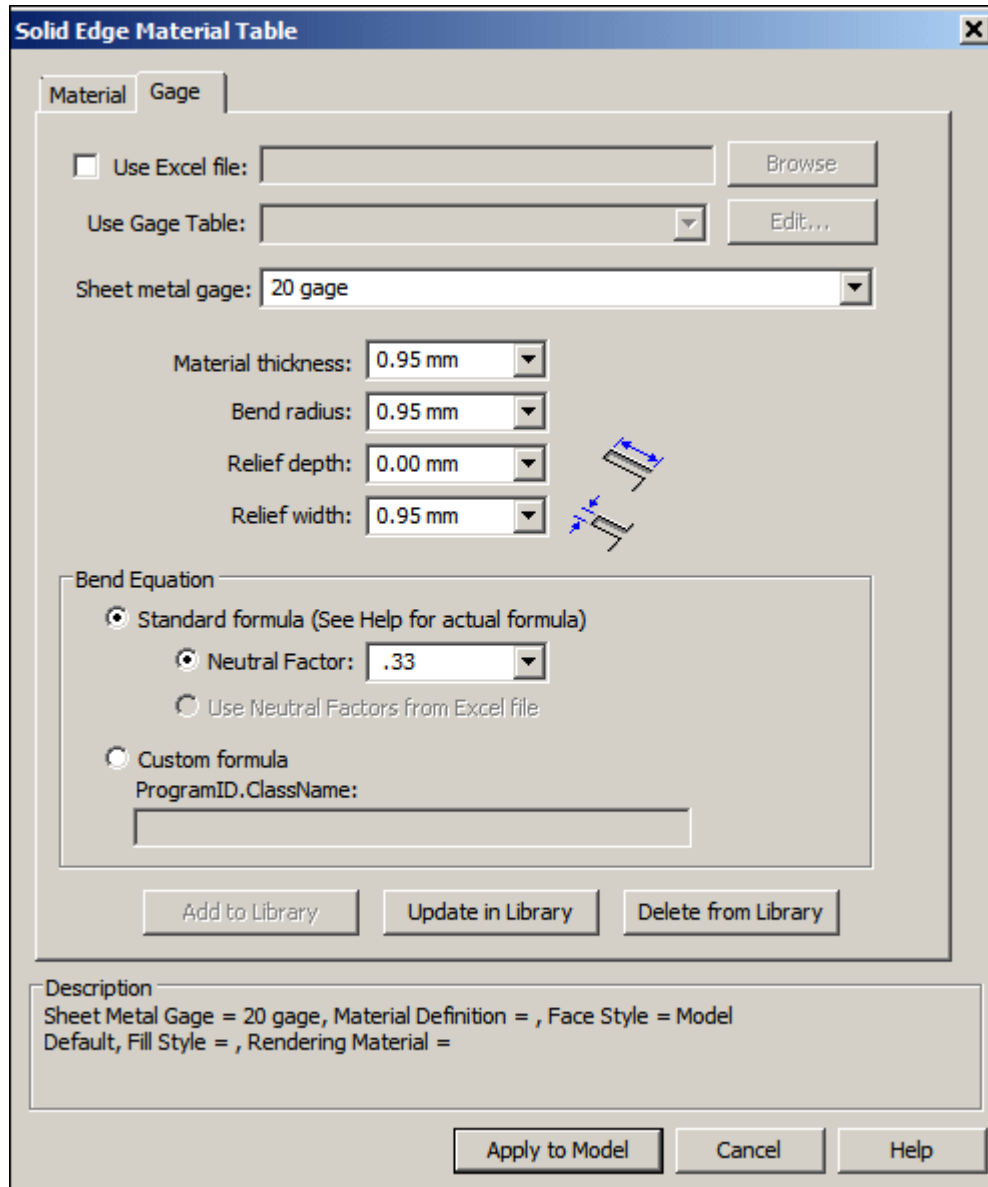
- ▶ Save the file as *interpartassy.asm* in the folder being used for the activity.

Create a new part in-place

- Click the Home® Assemble® Create Part In-Place command. Fill out the Create New Part In-Place dialog box as shown, and then click Create and Edit. Make sure the template option is set to the sheet metal template, *iso sheet metal.psm*. The name of the new part will be *cover.psm*. The Coincident with assembly origin option assures proper orientation of the new part. Make sure the folder for the New file location is the folder containing the activity files.



- ▶ On the Application menu, choose Properties® Material Table. Set the gage to 20, then click Apply to Model.



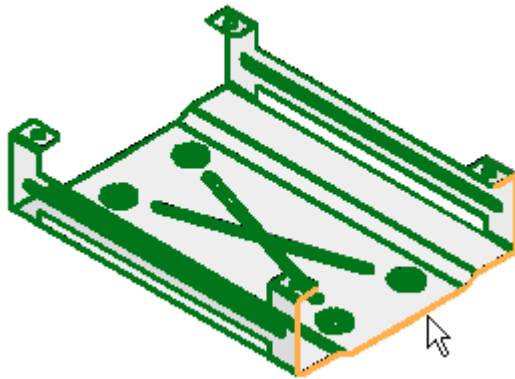
*Insert an inter-part copy that will be used as a construction surface to create a sheet metal cover*

- ▶ Right-click in PathFinder and then click Transition to Ordered.

- ▶ Click Home@ Clipboard@ Inter-part Copy.



- ▶ Click *Chassis.psm* as the assembly part to copy from, and then select the thickness face as shown.



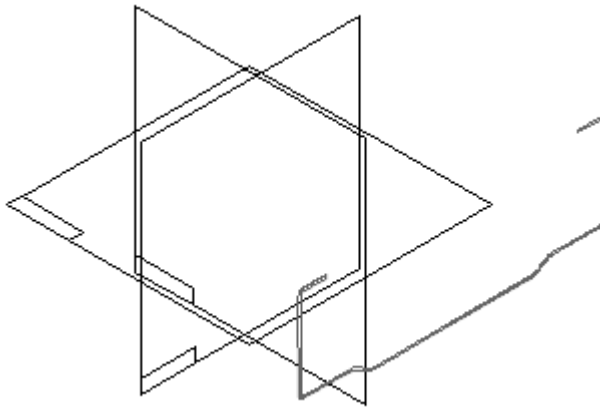
- ▶ Click the Accept button and then click Finish. This face serves as a construction surface within the sheet metal part file.

- ▶ To turn off the display of *Chassis.psm*, click View® Show® Hide Previous Level. Ensure the base reference planes are shown.

**Note**

When you select a part for modification (in-place activation) through the edit command within an assembly, hide previous level turns off the display of the remaining parts within the assembly for the sake of clarity.

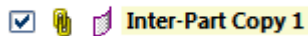
- ▶ Ensure the base reference planes are shown.



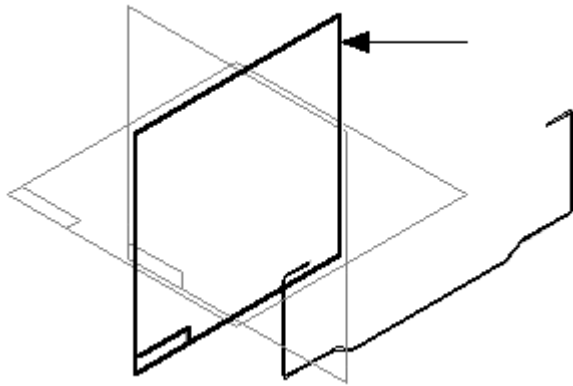
Use the *Contour Flange* command to construct the sheet metal cover

**Note**

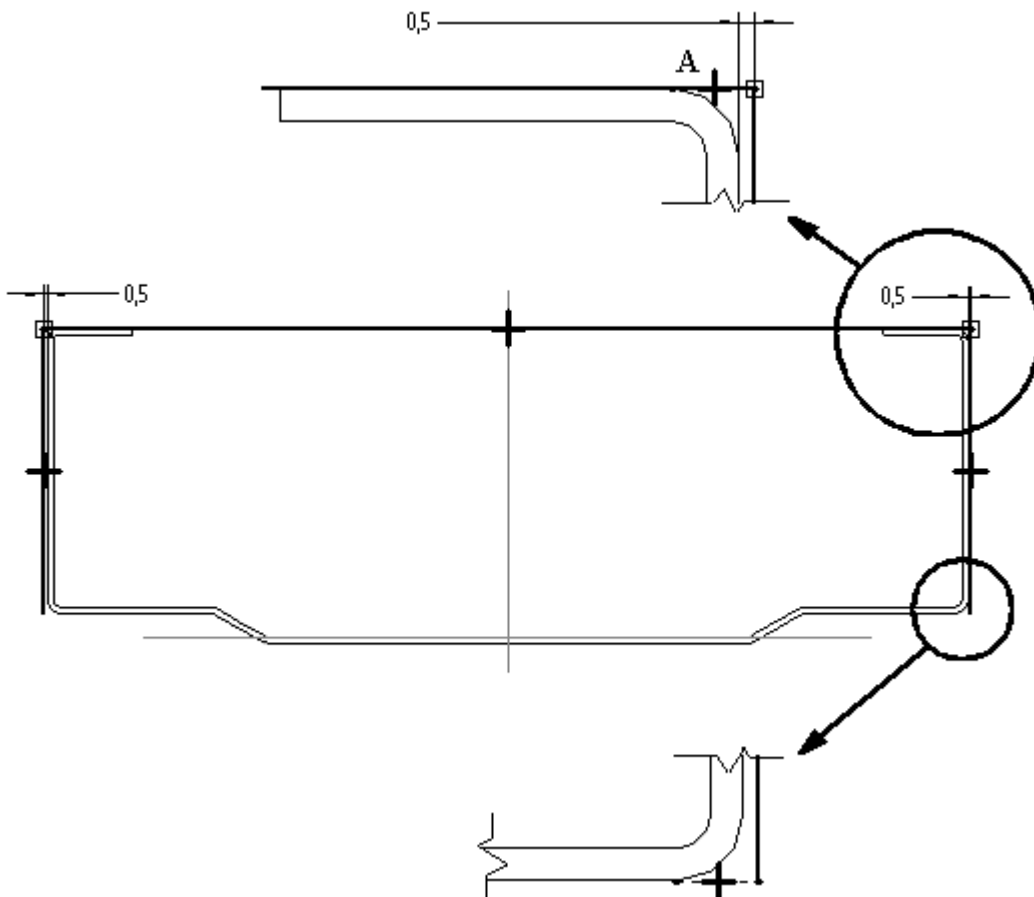
The modeling strategy for this activity is to use the construction surface to control the size and shape of the cover. PathFinder shows the thickness face as a construction surface. Constructions will need to be displayed in PathFinder to see the linked inter-part copy.



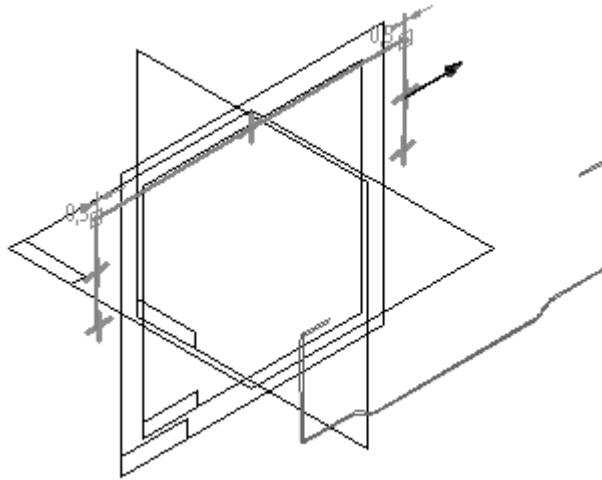
- ▶ Click the Contour Flange command. Select the reference plane as shown.



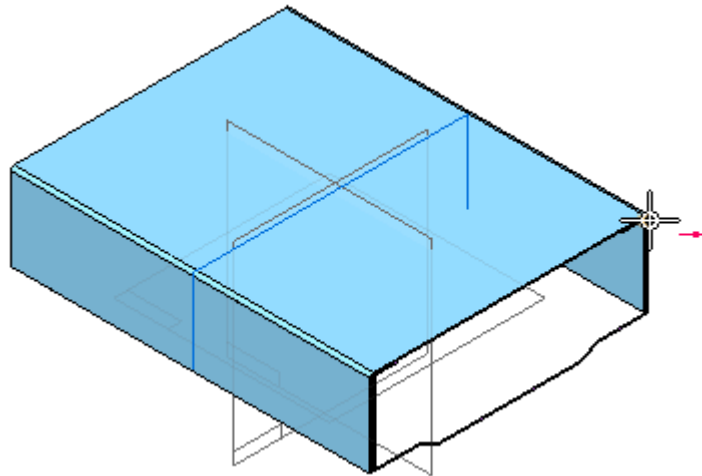
- ▶ Draw the profile as shown. Dimension and constrain the left side the same as shown for the right side. Apply a horizontal/vertical relationship as shown (A). Click Close Sketch.



- ▶ Click outside the profile to accept the side step as shown.



- ▶ Click the Symmetric Extent button, and then select any key point on the construction surface for the cover extent. Click Finish to complete the cover.



#### Note

The new cover is now in place. If an edit is made to *Chassis.psm* (from which the construction surface was copied), the new cover will update when the update links button is clicked, and then the construction surface will update. You will do this later in the activity.

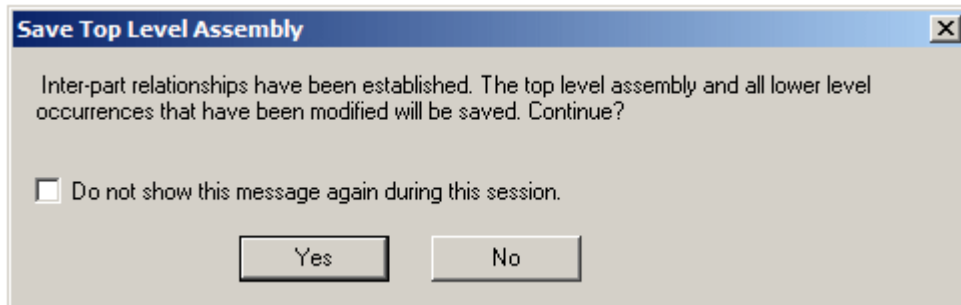
- ▶ Click View® Show® Hide Previous Level. This turns on the display of *Chassis.psm*.



- ▶ Hide the reference planes.

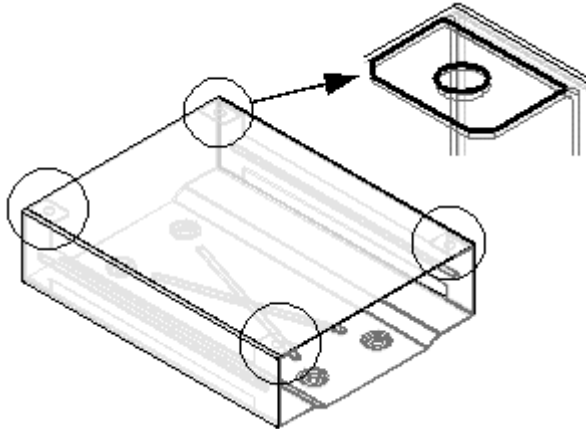


- ▶ Save, but do not exit the file. Click Yes if you see the dialog box shown.

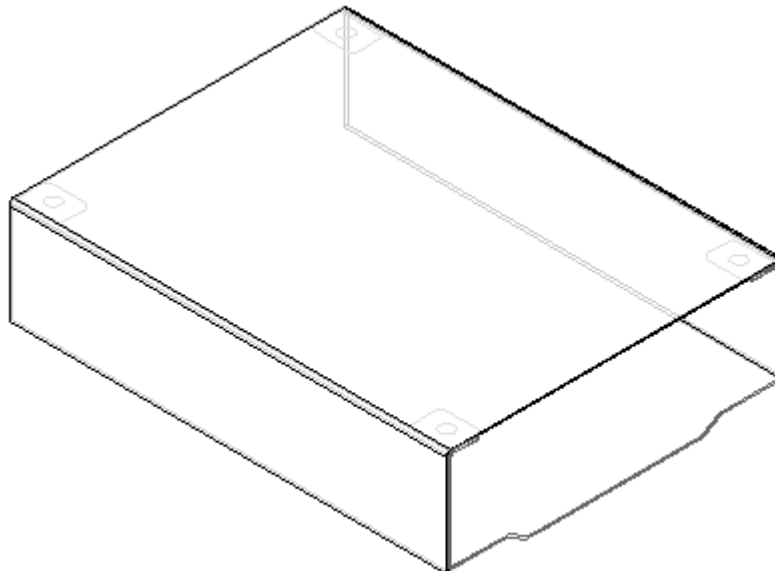


*Place a cutout and four holes on the top face of the cover linked via Inter-part*

- ▶ Click Home® Clipboard® Inter-part Copy. Copy the four top faces of the tabs as shown.



- ▶ Click View® Show® Hide Previous Level to turn off the display of *Chassis.psm*. The cover and the four copied faces display.

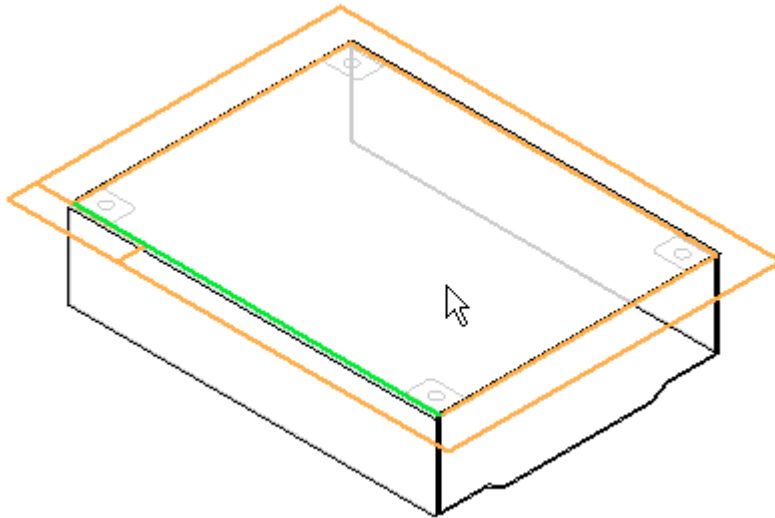


*Place four holes on the cover using the inter-part copied faces as reference*

- ▶ Click the Home® Sheet Metal® Hole command.



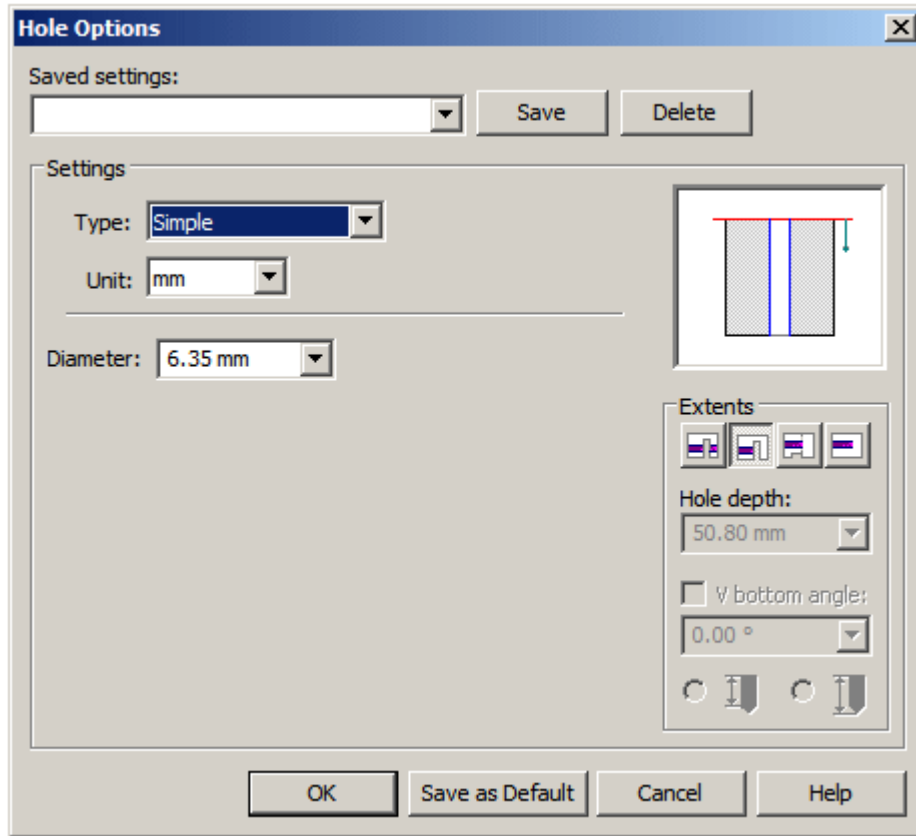
- ▶ Highlight the top face of the sheet metal cover for the reference plane and orient the plane as shown. Press N to toggle reference plane orientations, then click to select this orientation.



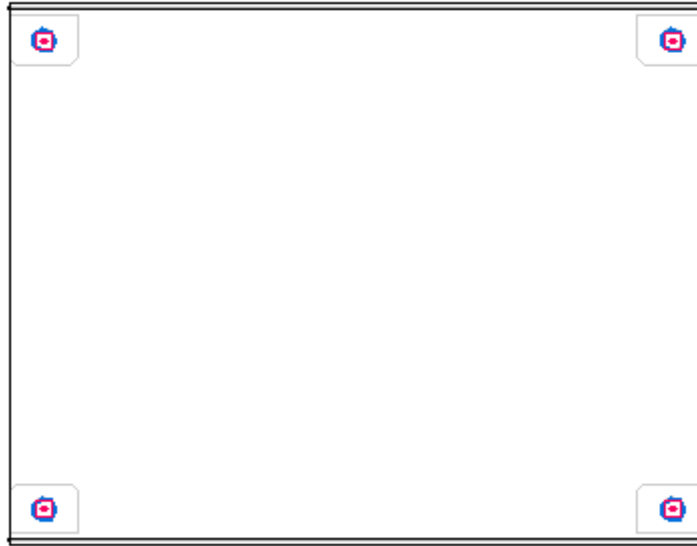
- ▶ Click the Hole Options button.



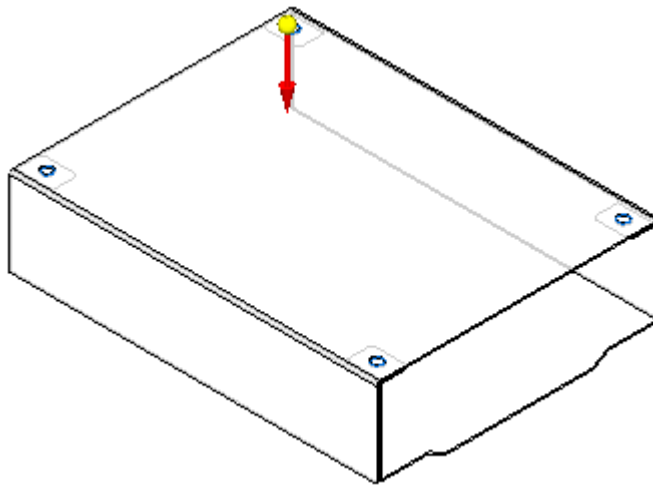
- ▶ Set the options as shown.



- ▶ Place four 6.35 mm Simple holes centered on the circles of each inter-part copied face as shown.

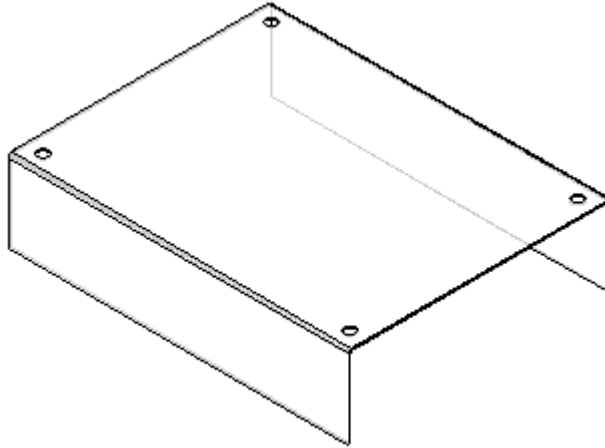


- ▶ Close the sketch and select the direction as shown, then click Finish.



- ▶ In PathFinder, turn off the surfaces created by inter-part copy.

- ▶ Save the file. The result is shown.

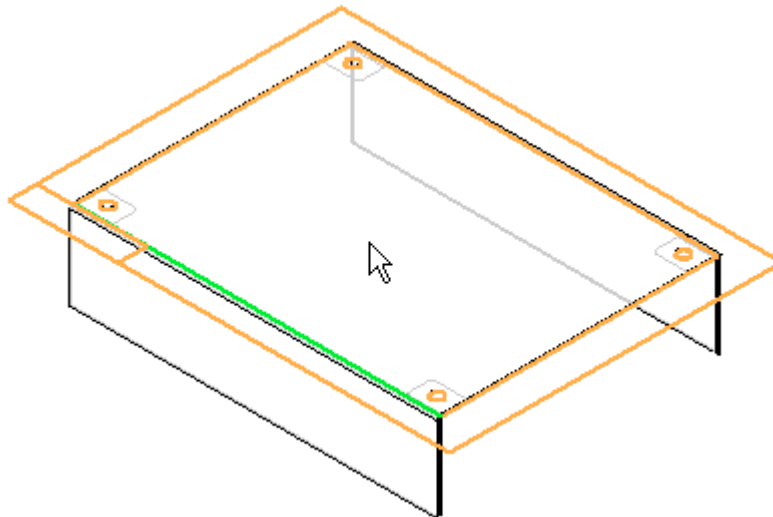


*Add a cutout to the cover*

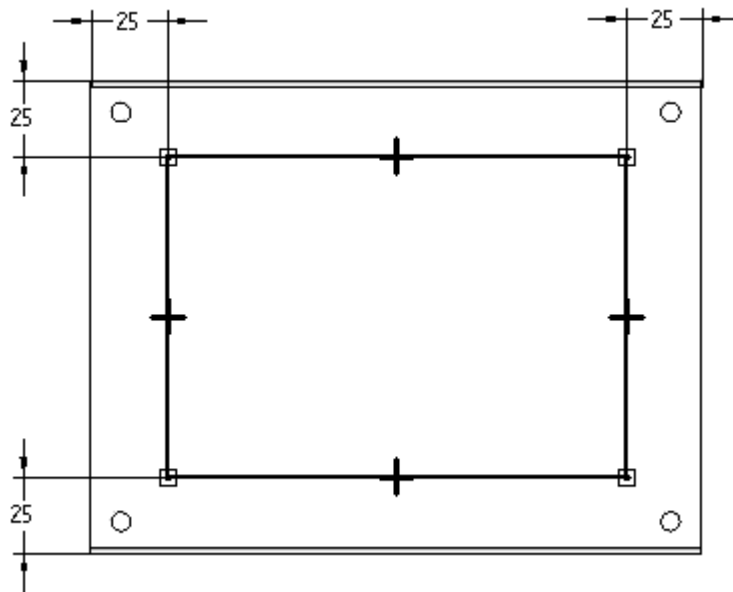
- ▶ Click the Home® Sheet Metal® Hole® Cut.



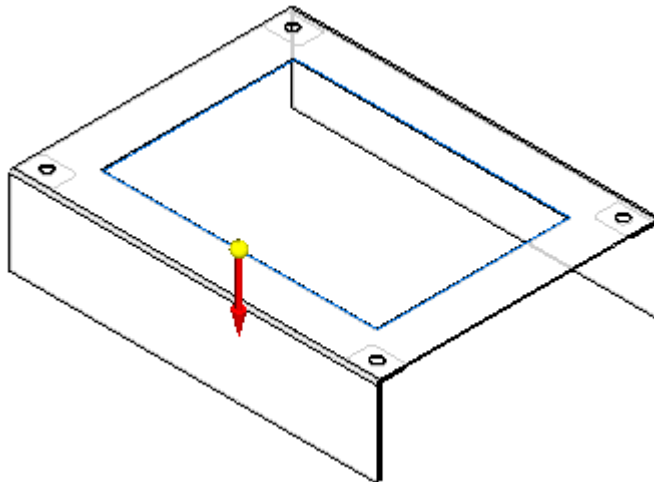
- ▶ Select the top face of the sheet metal cover for the reference plane and orient as shown.



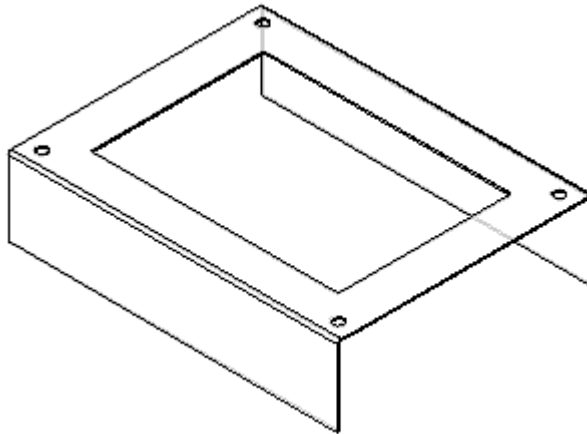
- ▶ Draw the profile for the cutout as shown.



- ▶ Click Close Sketch.
- ▶ Click Through Next and position the arrow as shown.



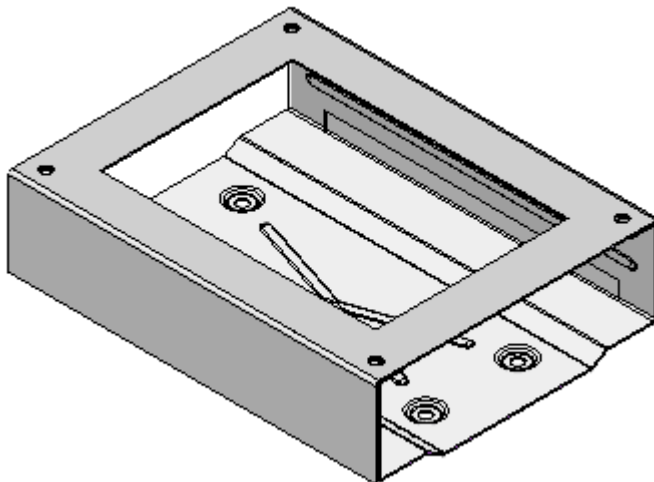
- ▶ Click Finish.



- ▶ Save the file, and click Yes in the Save Top Level Assembly dialog box.

*Turn on the display of Chassis.psm and then return to the assembly file*

- ▶ Click Hide Previous Level to turn on the display of *Chassis.psm*. The two-part assembly displays.

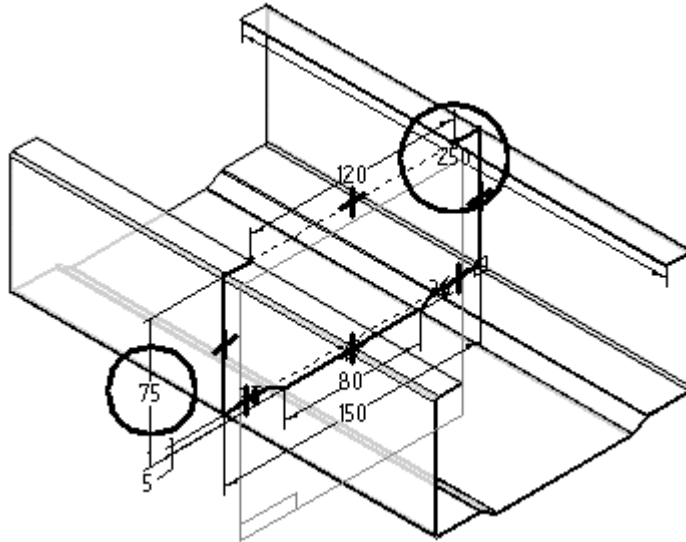


- ▶ Click Close and Return. This returns you to the assembly.

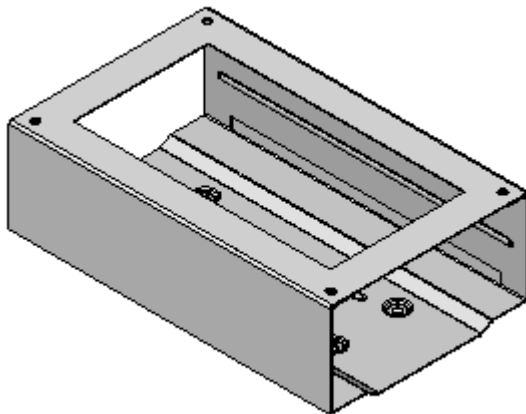
*Make an edit and update the links to update the new sheet metal cover*

- ▶ On Assembly PathFinder, right-click *Chassis.psm*, and then click Edit on the shortcut menu. This opens the *Chassis.psm* file for editing.
- ▶ Right-click the first contour flange feature listed in PathFinder and then click Edit Definition. This is the base feature for the model.

- ▶ Select the following dimensions and edit their values:  
200 mm (part length) to 250 mm  
50 mm (part height) to 75 mm



- ▶ After *Chassis.psm* recomputes, click Close and Return to return to the assembly file. If the cover has not updated at this time, click Tools® Links® Update All Links to force the update.



- ▶ Repeat the editing procedure, and change dimensions on *Chassis.psm* back to their original values:  
250 mm (part length) to 200 mm  
75 mm (part height) to 50 mm
- ▶ Save the file. This completes the activity.



*Activity Summary*

In this activity you learned how to use the create in place command to create a new sheet metal file in the context of an existing assembly. Inter-part functionality was used to link geometry in the new sheet metal file to geometry in other files in the assembly. You learned that because the faces used to create the new sheet metal file were linked to the other parts of the assembly, changes made to those other parts result in a change of size and shape of the new part, which responds to the changes made via the inter-part link.

## Lesson review

Answer the following questions:

1. Name the two methods of building an assembly in Solid Edge.
2. Describe the bottom up approach to assembly modeling.
3. Describe the top down approach to assembly modeling.
4. Is it possible to combine top down and bottom up assembly modeling?
5. Describe the use of the include command when using the top down assembly modeling approach.

## Answers

1. Name the two methods of building an assembly in Solid Edge.

The two methods are the top down approach and the bottom up approach.

2. Describe the bottom up approach to assembly modeling.

Bottom-up assembly modeling is a part-centric modeling method where the assembly design is started with a principal structural or functional element, and individual parts are designed in relative isolation from the overall assembly. Component parts and subassemblies are defined as the process moves up towards the top-level assembly. With this approach, as the design of a key component is completed, its geometry may or may not be used to aid the design of related mating components.

This approach is often used at companies where the product being designed is small enough that one or only a few people are needed to complete the design.

3. Describe the top down approach to assembly modeling.

Top-down assembly modeling is an assembly-centric modeling method where the assembly design is started at the highest level possible, and individual parts and subassemblies are defined within the context of the overall assembly. With this approach, an assembly layout is typically created first, and this assembly layout is used to define individual part geometry and position.

This approach is often used at companies where the product being designed is large enough that it requires many people to complete the design. A senior-level designer might create the initial assembly layout, then divide the assembly layout into logical subassemblies and parts for the remainder of the organization to complete.

4. Is it possible to combine top down and bottom up assembly modeling?

Yes, both top down and bottom up assembly modeling can be used together during assembly modeling.

Solid Edge provides tools that allow you to take advantage of the benefits of both approaches as needed. Many organizations use a combination of both methods, using the method which best suits the immediate requirements. For example, you can use the top-down approach to create the initial assembly layout and to define the document structure needed. You can then copy the assembly layout geometry to subassembly and part documents to divide the work among the organization.

You can shift to the bottom-up approach in areas of the design that use purchased parts, existing parts from an earlier project, or where you are modeling standard parts in 3D that were created on an earlier 2D CAD system.

The suite of commands and tools in Solid Edge also allow you to use either approach associatively or non-associatively, as you see fit.

5. Describe the use of the include command when using the top down assembly modeling approach.

The Include command also allows you to associatively or non-associatively copy geometry between documents when working in the context of an assembly.

Many feature commands allow you to define the extent of the feature associatively by selecting a keypoint on another part in the assembly.

## **Lesson summary**

In this lesson you learned how to use the create in place command to create a new sheet metal file in the context of an existing assembly. Inter-part functionality was used to link geometry in the new sheet metal file to geometry in other files in the assembly. You learned that because the faces used to create the new sheet metal file were linked to the other parts of the assembly, changes made to those other parts result in a change of size and shape of the new part, which responds to the changes made via the inter-part link.

## **Assembly layouts**

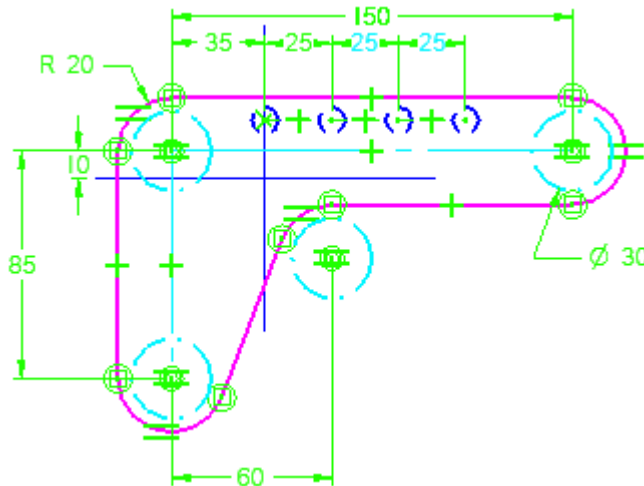
### **Overview**

Creating parts in an assembly whose size and shape can be controlled based on the size and shape of geometry in a sketch in the assembly is accomplished through Inter-part links. When the geometry of the sketch changes, the Inter-part links in the linked part alter the geometry accordingly.

## Assembly layouts

As you develop the design concepts for a new assembly, it is useful to create a layout of the preliminary design. The Sketch command in the Assembly environment allows you to draw 2D sketch geometry on part or assembly reference planes.

You can draw assembly sketches on the three default assembly reference planes or you can create new assembly reference planes to draw sketches on.



You can use assembly sketches to do the following:

- Create 3D ordered geometry within parts.
- Create assembly features.
- Position 3D parts relative to the sketch geometry.
- Position an assembly sketch relative to a 3D part.

### Note

For more information about 2D drawing in Solid Edge, see the Drawing in Solid Edge and Drawing Profiles topics.

## **Drawing an assembly sketch**

Drawing an assembly sketch is similar to drawing a sketch in the Part environment. When you click the Sketch button and then select or define a reference plane, a sketch view is displayed. You can position the new reference plane relative to an assembly reference plane or relative to a face or reference plane on a part in the assembly.

You can use the 2D element creation commands to draw sketches that represent the basic shapes of the parts. You can set style, color, type, and width options for the 2D elements to make it easier to interpret the assembly sketch.

The 2D elements you draw are assigned to the active layer. For example, when working with a complex assembly sketch that will be used to construct several parts, you may want to arrange the elements on multiple layers.

You can add dimensions and relationships to control the position and size of the sketches. You can also define functional relationships using the Variables command. You can use the PathFinder shortcut menu commands to control the display of the sketch elements.

You can also control the display of elements in an assembly sketch by assigning the elements to a logical set of layers, and then display or hide the layers to control the display of the elements.

You can use the Save and Save All commands to save an assembly sketch. When you have finished drawing, close the sketch view using the Close Sketch command on the Home tab.

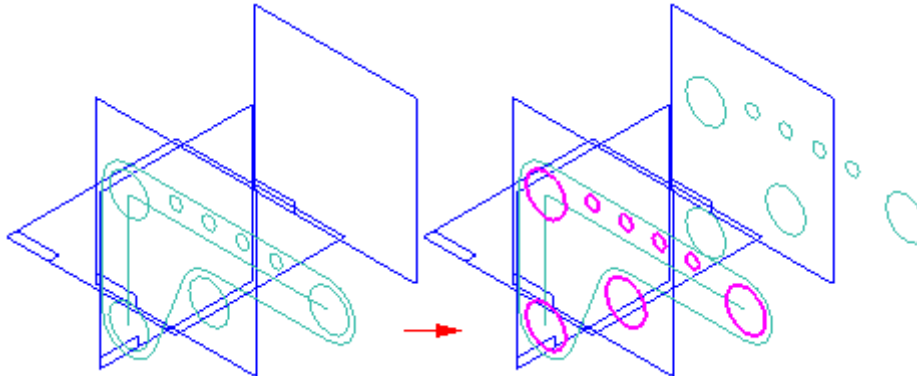
## **Copying sketches**

Your assembly layout might contain representations of many parts and subassemblies, and during the course of design you might choose to copy sketches between documents. You can use the Copy Sketch command to copy assembly and part sketches between documents when working in the context of an assembly. When copying sketches, you use the Copy Sketch To Target dialog box to specify the target document, and whether the copied sketch is associatively linked to the original sketch.



## **Tearing-off sketches**

You can use the Tear-Off Sketch command to move or copy elements in an assembly sketch to new reference plane you define.



When you copy sketch elements, you can use the options on the Tear-Off Sketch Options dialog box to specify whether the copied elements are associative to the existing elements.

## **Using an assembly sketch to construct parts and subassemblies**

### **Note**

Assembly sketch geometry can be associatively linked to drive the creation of ordered part geometry only.

When you create or modify a part or subassembly in the context of the assembly, you can use your assembly sketches to construct part profiles and subassembly sketches. You can also use the sketch elements to construct assembly features. For example, you can use the Include command to include an element from an assembly sketch to a part profile or to include an edge from a part to the assembly sketch.

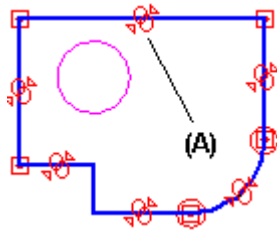
To associatively include an assembly sketch element, you must first set an option on the Inter-Part tab on the Options dialog box: **Allow Inter-Part Links Using: Include Command In Part and Assembly Sketches**. Then, on the Include Options dialog box, set the **Allow Locate of Peer Assembly Parts and Assembly Sketches** option and the **Maintain Associativity When Including Geometry From Other Parts** in the Assembly option.

### **Note**

For more information about working with associative links between parts and assemblies in Solid Edge, see the Inter-Part Associativity topic.

### Link relationship handle

A special relationship handle is added to any linked profile element (A) to indicate that it is linked to an element in another document. You can break a link by deleting the relationship handle for that link.



### Positioning 3D components using assembly sketches

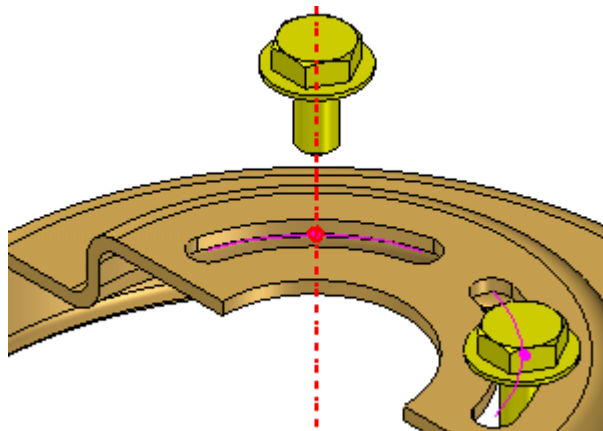
You can position parts and subassemblies with respect to a part or assembly sketch. You can position 3D components using assembly relationships, such as mate and planar align; or using 2D dimensions and relationships, such as distance between and connect.

While editing an assembly sketch, you can use the Parts Library tab to add new components to the assembly.

#### Positioning 3D components using assembly relationships

You can use assembly relationships, such as mate, planar align, connect and axial align, to position a part to a keypoint or a line on a part, subassembly, or top-level assembly sketch.

For example, it can be difficult to position a bolt in a slot using faces. In this example, a bolt part is positioned to a point element on a part sketch.



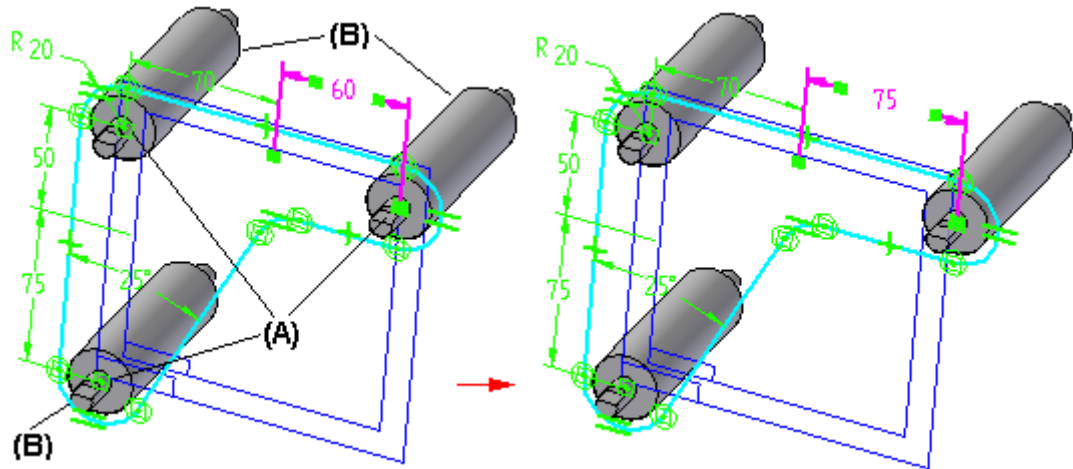
An inferred axis that is perpendicular to the reference plane of the sketch on which the point lies defines the axis for the axial align relationship. The arc and point were drawn such that if the size or position of the slot changes, the arc and point will also update properly.

#### Positioning 3D components using 2D dimensions and relationships

While you are editing an assembly sketch, you can use 2D dimensions and relationships to position a 3D component relative to elements in the sketch. For example, you can use connect relationships (A) to position roller parts (B)

relative to an assembly sketch. When you edit the assembly sketch, the position of the roller parts update. This can be useful when creating a new assembly that uses existing components.

In this example, the value of the 60 millimeter dimension was edited in the assembly sketch to 75 millimeters. Because the roller part was constrained to the assembly sketch using a 2D connect relationship, the position of the roller updated when the sketch dimension was edited.



When a sketch window is active, you can use the Select tool and options on the Position 3D Component command bar to specify whether the sketch drives the position of the 3D component or the 3D component drives the position of the sketch. In the previous example, the Sketch Driving option was set for the roller parts to specify that the sketch drives the position of the 3D components.

**Note**

When you set the Sketch Driving option, the component symbol in PathFinder indicates that the component is driven by the assembly sketch.

You can use the Select tool to select the 3D component in PathFinder or you can use the Component Select Tool command to select the 3D component in the graphics window. In either case, when you select the 3D component, the Position 3D Component command bar is displayed.

As you choose which relationships and techniques to use when you position components in assemblies, keep in mind the following:

- That you cannot use 2D dimensions and relationships to position 3D components that conflict with existing 3D relationships.
- When you set the Sketch Driving option for a part, it is with respect to the current sketch. If relationships are available, the part can be driven by or drive another sketch.
- You can set the Sketch Driving option for a part in more than one sketch and apply 2D relationships in each sketch until the part is fully positioned. This can make it easy to fully position a part using two or more sketches.
- When you set the Lock Alignment option on the Position 3D Component command bar, the part is locked parallel with respect to the sketch plane and the part face you selected. The part can still move and rotate. This can make it easy to fully position a part using one sketch.

### **Positioning assembly sketches using 3D components**

You can also constrain elements on an assembly sketch to a 3D assembly component such that if the size, shape or position of the 3D component changes, the assembly sketch updates. When you are editing an assembly sketch, you can select a 3D component and set the Component Driving option on the Position 3D Component command bar to specify that the 3D component drives the size, shape, and position of the assembly sketch.

### **Assembly sketches and alternate assemblies**

The Sketch and Copy Sketch commands are available only when the Apply Edits to All Members option on the Alternate Assemblies tab is set (you are working globally).

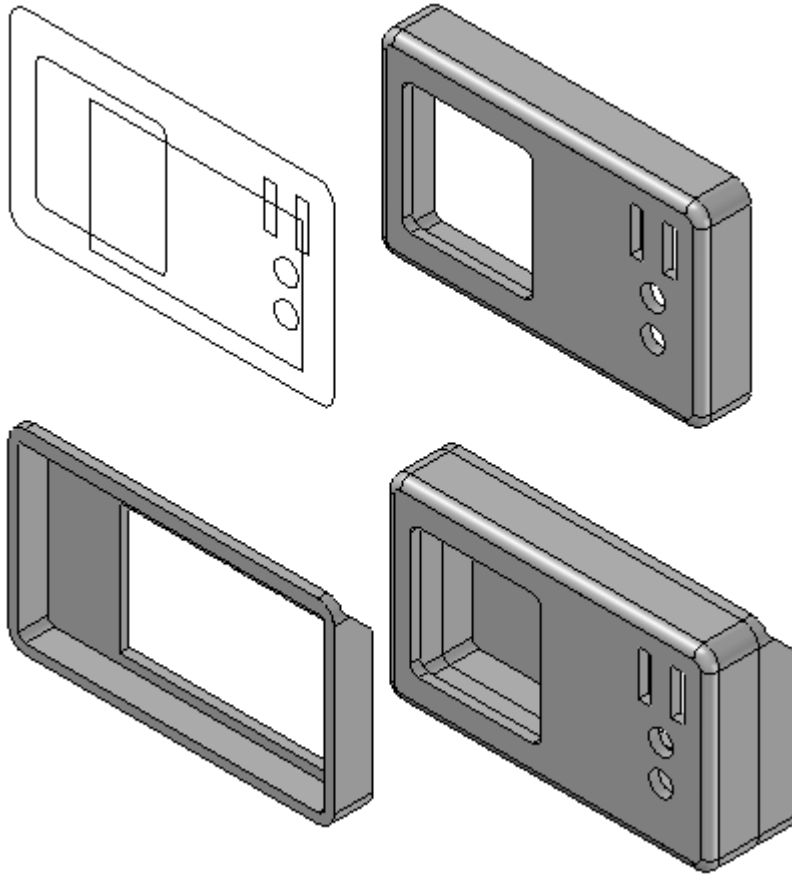
For more information, see the Alternate Assemblies Impact on Solid Edge Functionality Help topic.

### **Activity: Layout sketches in assembly**

When you complete this activity, you will be able to use inter-part modeling techniques to create new parts in an assembly using sketch geometry in the top level assembly.

For this activity, you will use sketch geometry to control the size and shape of two halves of a radio housing.

Activity guides you through the process of associatively linking parts with Inter-part links to sketch geometry, so that changes in the sketch will be reflected in the parts in the assembly.



**Activity: Creating parts from assembly sketches**

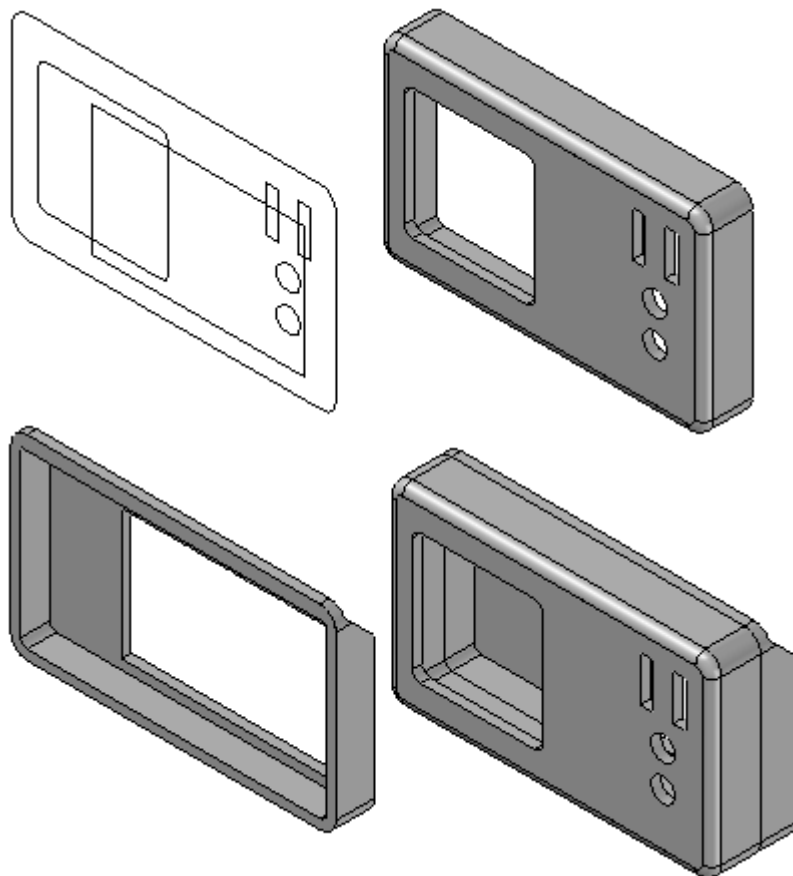
This activity shows how part geometry can be driven from assembly sketches that are associatively linked to the part.

*Activity – Creating parts from assembly sketches***Overview**

When you complete this activity, you will be able to use inter-part modeling techniques to create new parts in an assembly using sketch geometry in the top level assembly.

**Objectives**

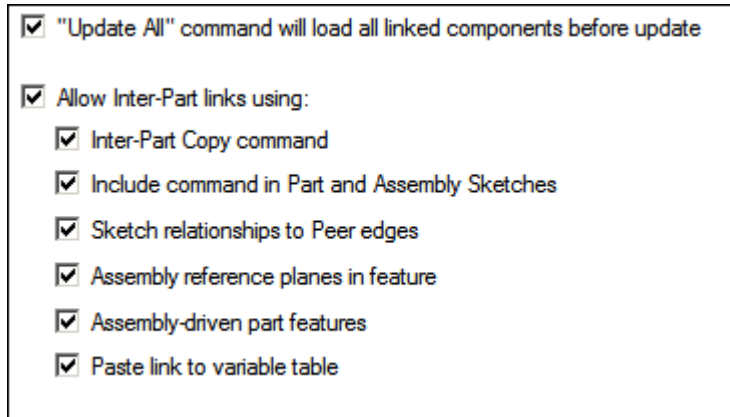
When you complete this activity, you will be able to model the shell of a radio housing shown in the following illustration. This activity teaches the Include from assembly sketches modeling capability of Solid Edge inter-part assembly. The Include from Assembly sketches option allows you to model mating parts within an assembly that maintains like design characteristics between its parts. Sketch sharing is the key to maintaining like characteristics between parts. When you change a sketch, the mating part(s) change. This is a top-down assembly modeling method.



*Set associativity parameters*

Set the parameters needed to have the sketch control the geometry of the parts created in the assembly.

- Open *radio.asm* located in the folder where you placed your activity files.
- Click the Application button, then click Solid Edge Options. In the Solid Edge Options dialog box, click the inter-part tab. Select the options as shown and click OK.



**Note**

For this activity, an assembly sketch is provided. This sketch will be used to drive the creation of two parts that make up the front and back of a radio housing.

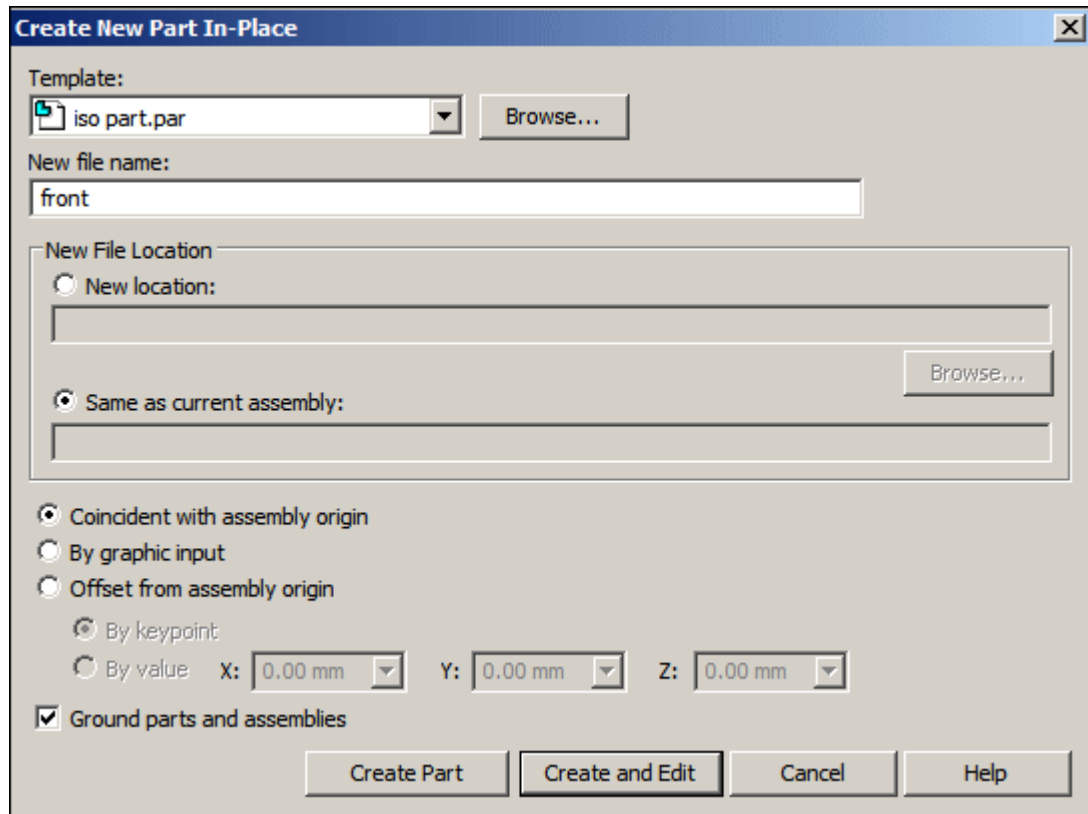
*Create a part that is the front half of the radio housing*

Create a part that is the front-half of the radio housing.

- Click Home® Assemble® Create Part In-Place.



- ▶ Type the new part file name as front. The new file location will be the same location as the current assembly. Click Create and Edit. Solid Edge opens the new part file and displays the sketch contained in the assembly file *radio.asm*.



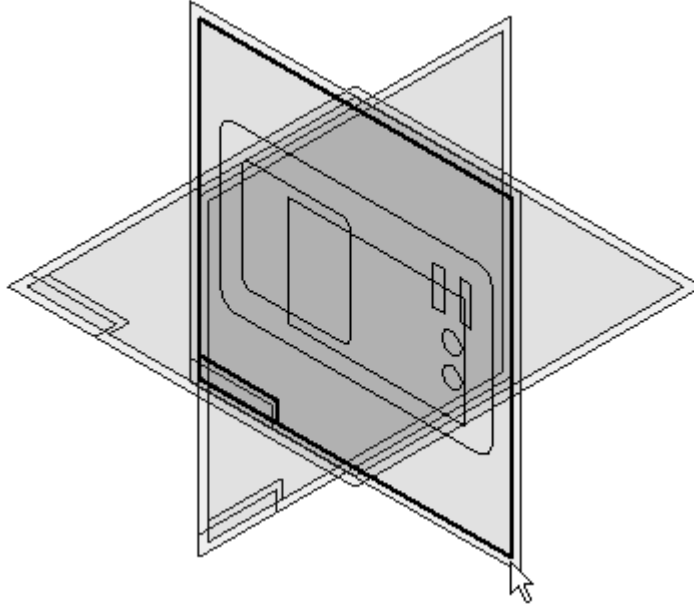
### *Construct an extrusion*

Construct an extrusion using the assembly sketch.

- ▶ If you are in the Synchronous environment, right-click in PathFinder and then click Transition to Ordered.
- ▶ In PathFinder, display the base reference planes.
- ▶ Click the Extrude command.



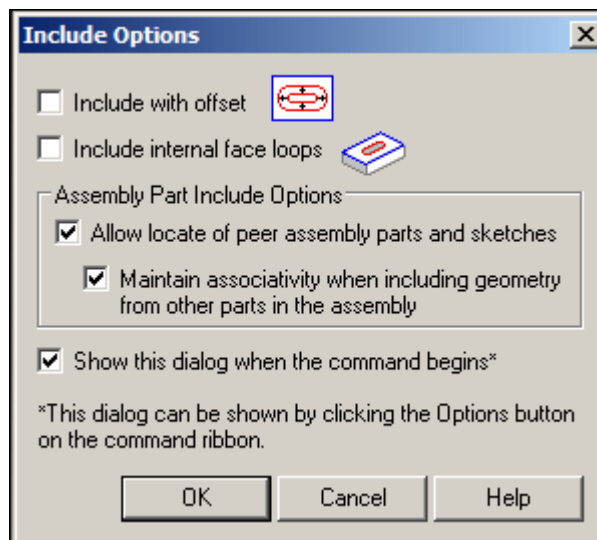
- ▶ Select the reference plane shown.



- ▶ Click the Include command.

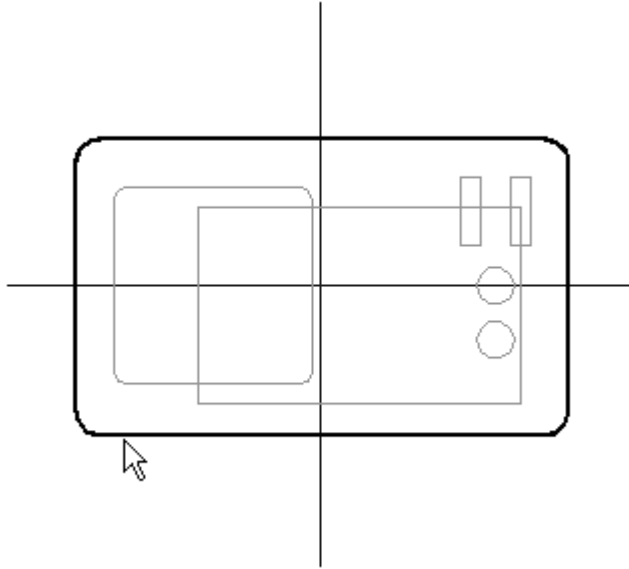


- ▶ In the Include Options dialog box, make the selections as shown. Any elements you select to include will link to the assembly sketch for associative edits. Click OK.

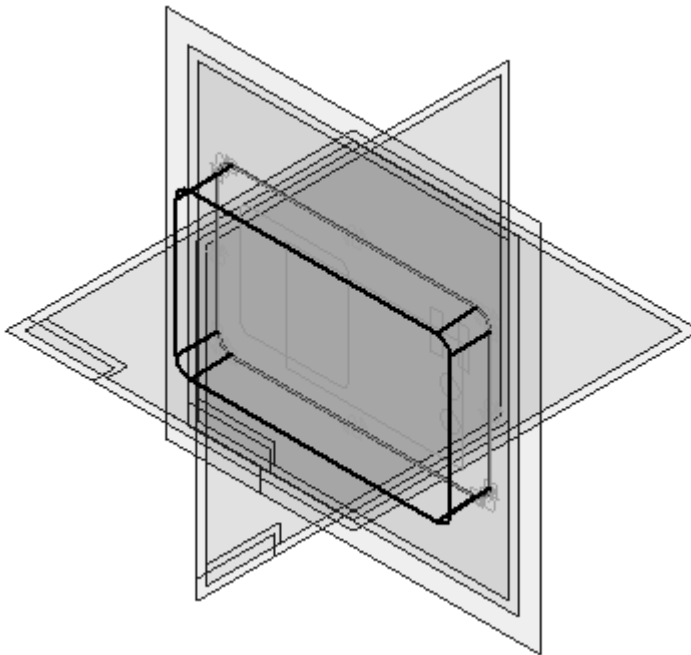


- ▶ In the command bar, set the select box to Wireframe chain.

- ▶ Select the sketch chain shown.

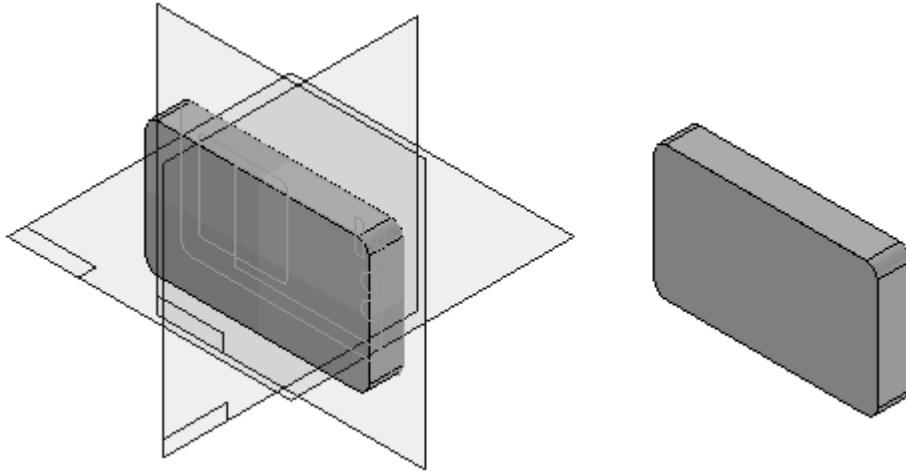


- ▶ Click the Accept button and then Close Sketch.
- ▶ Type 15 for the finite extent distance and select the extent direction shown.



- ▶ Click Finish.
- ▶ In Feature PathFinder, turn off all reference planes in the current part file. The reference planes and sketch from the assembly file continue to display.

- ▶ Click View® Show® Hide Previous Level. This turns off the display of the reference planes and the sketch.



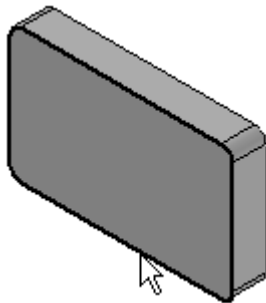
*Add a round*

Add a round to an edge of the part.

- ▶ Click the Round command.

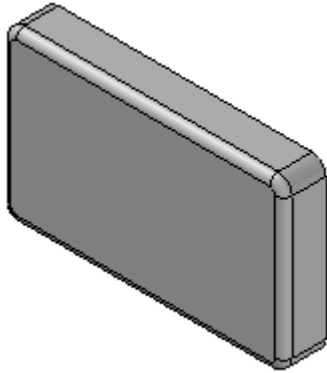


- ▶ Select the edge shown.



- ▶ Type 3 for the radius and click the Accept button.

- ▶ Click Preview and then Finish.



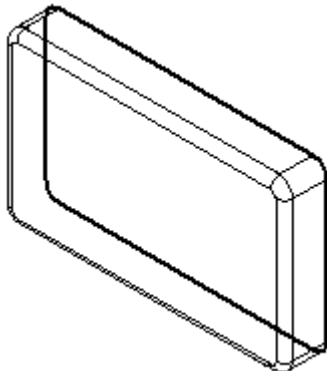
*Apply a thickness to the part*

Apply a thickness to the part using the thin wall command.

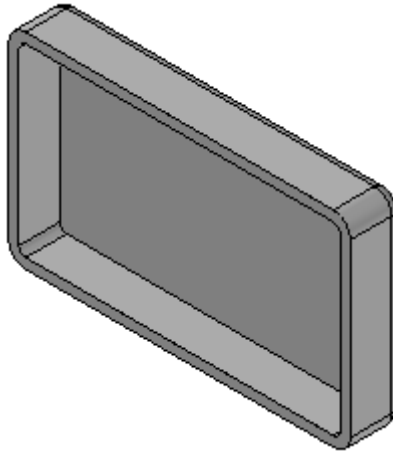
- ▶ Click the Thin Wall command.



- ▶ Type 3 for the common thickness. Select the face shown to be the open face.



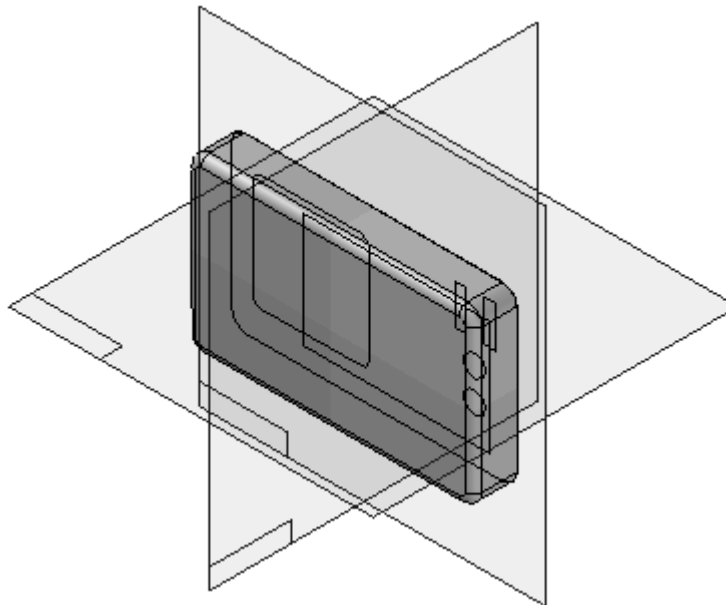
- ▶ Click Preview and then Finish. In the image below, the part was rotated 180° to show the thin wall feature.



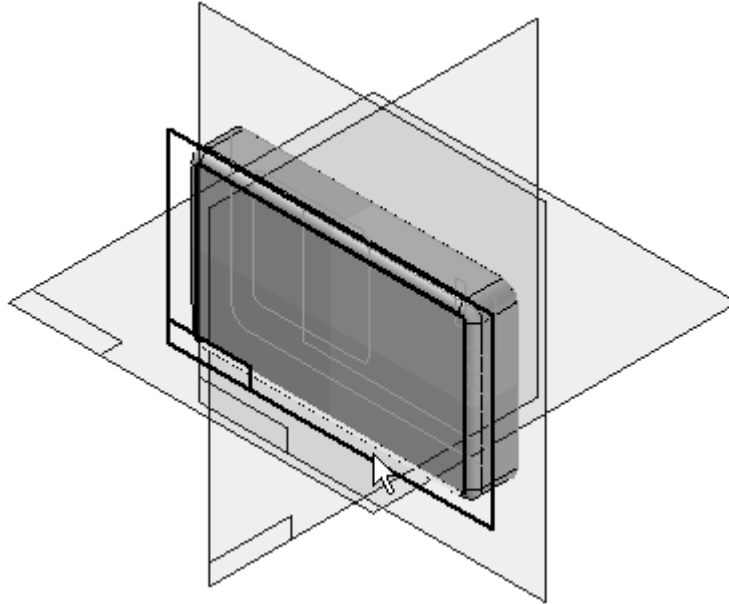
#### *Add cutouts*

Add several cutout features to the part in one step.

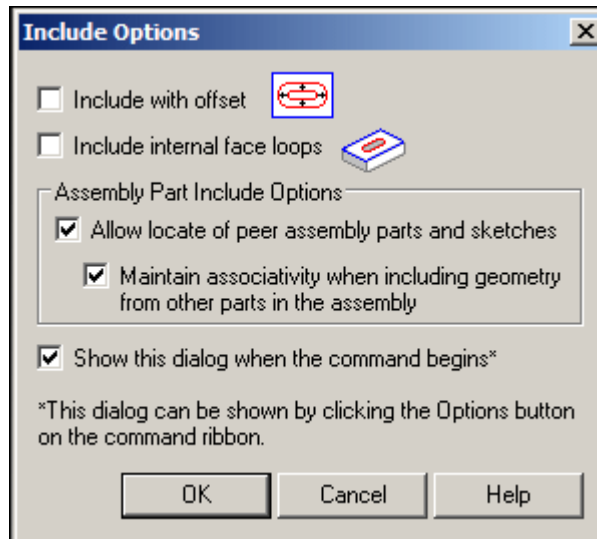
- ▶ Click the Cut command.
- ▶ Click View® Show® Hide Previous Level. This turns the display of the reference planes and the sketch back on again.



- ▶ Select the front face as the profile plane as shown.

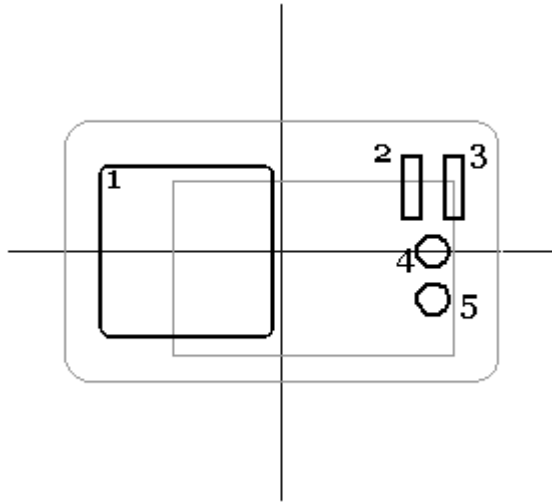


- ▶ Click the Include command. Set the options as shown and click OK.

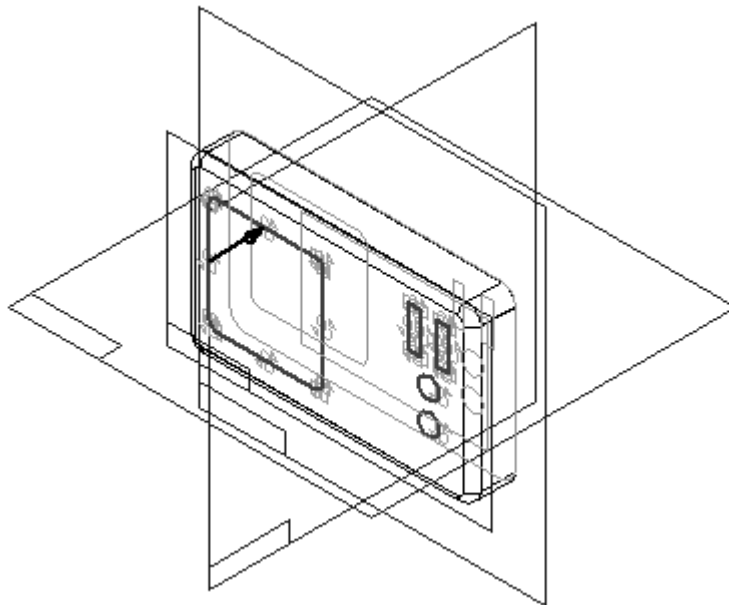


- ▶ Set the select field to Wireframe Chain.

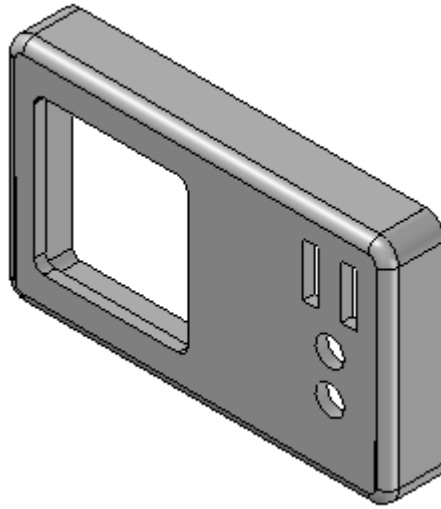
- ▶ Select the five chain sets shown and click the Accept button. Click Close Sketch.



- ▶ Select the Through Next option for the extent. Set the direction as shown and click Finish.



The front part is now complete.



- ▶ Click the Home tab. Click Close and Return to return to the assembly file *radio.asm*.



- ▶ Click the Assembly PathFinder tab and notice that the part just created is now a part of the assembly.



*Create the back half of the radio housing*

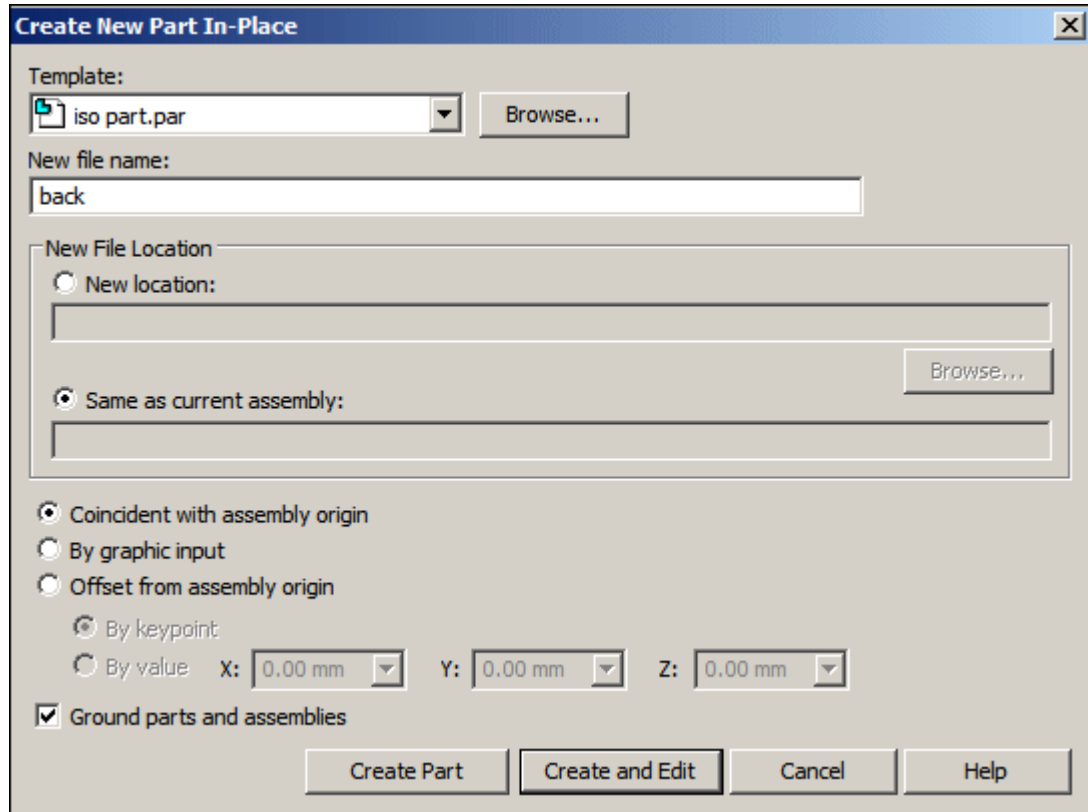
Create the back-half of the radio housing.

- ▶ Click Home® Assemble® Create Part In-Place.





- ▶ Type the new part file name as back. The new file location will be the same location as the current assembly. Click Create and Edit. Solid Edge opens the new part file and displays the sketch contained in the assembly file *radio.asm*.

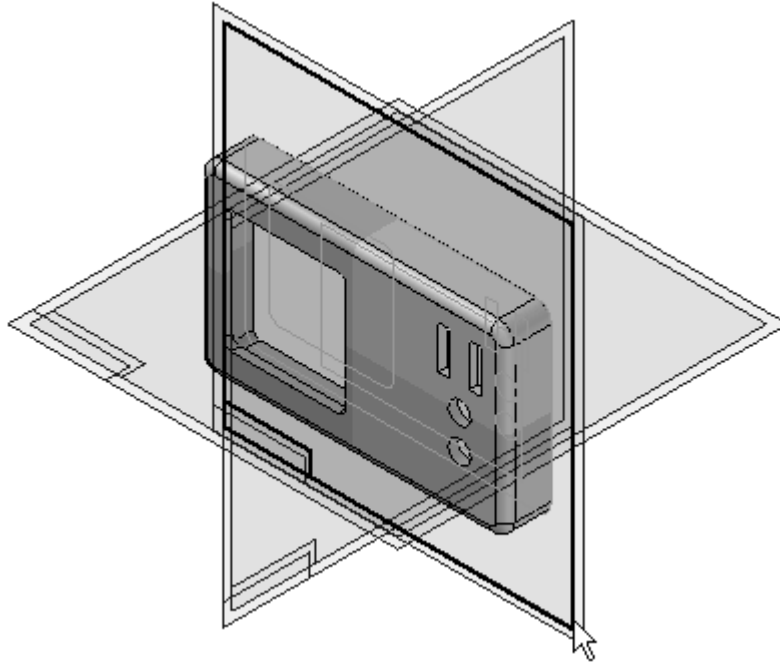


*Construct a protrusion using the assembly sketch*

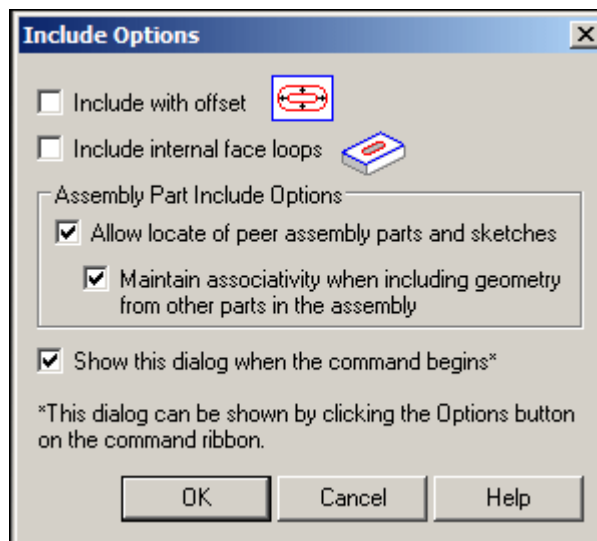
Construct a protrusion using the assembly sketch.

- ▶ If you are in the Synchronous environment, right-click in PathFinder and then click Transition to Ordered.
- ▶ In PathFinder, display the base reference planes.
- ▶ Click the Extrude command.

- ▶ Select the reference plane shown.

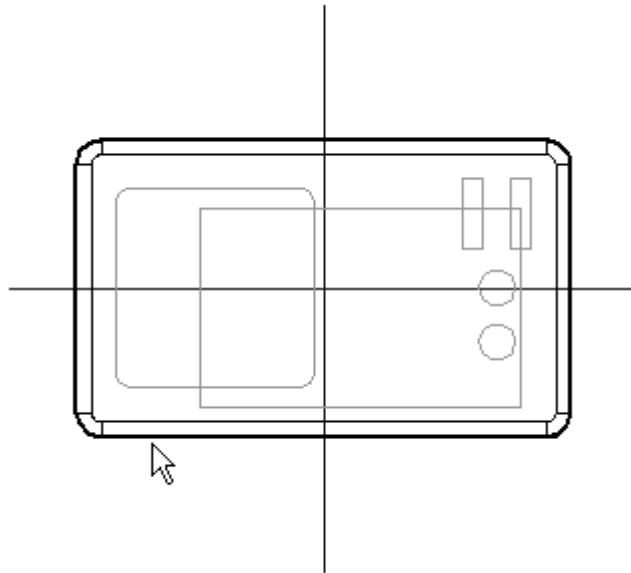


- ▶ Click the Include command. In the Include Options dialog box, make the selections as shown. Any elements you select to include will link to the assembly sketch for associative edits. Click OK.

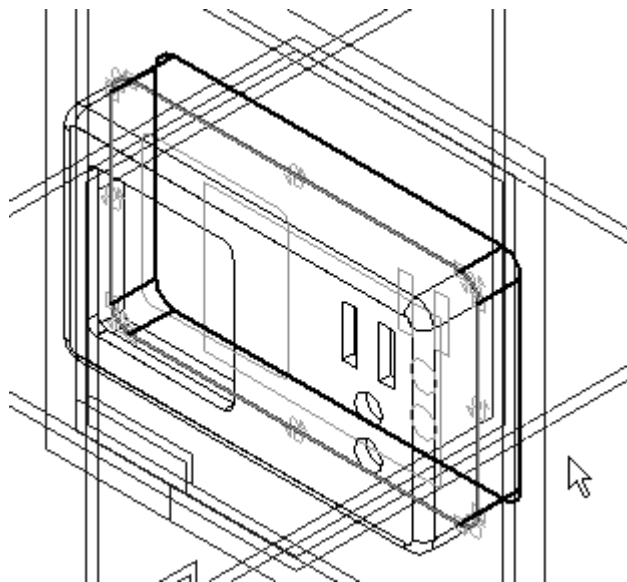


- ▶ Set the select field to Wireframe Chain.

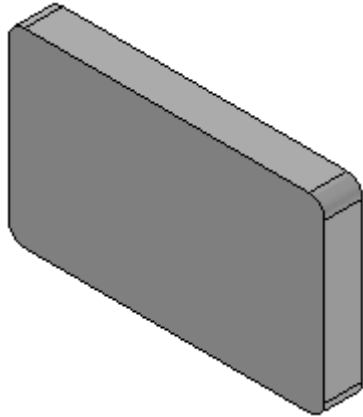
- ▶ Select the outside wireframe chain as shown. Click the Accept button and then click Close Sketch.



- ▶ Use the Finite Extent and type 12 for the distance. Set the direction as shown and click Finish.



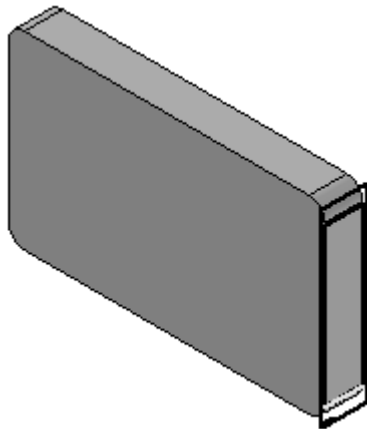
- ▶ Click View® Show® Hide Previous Level.



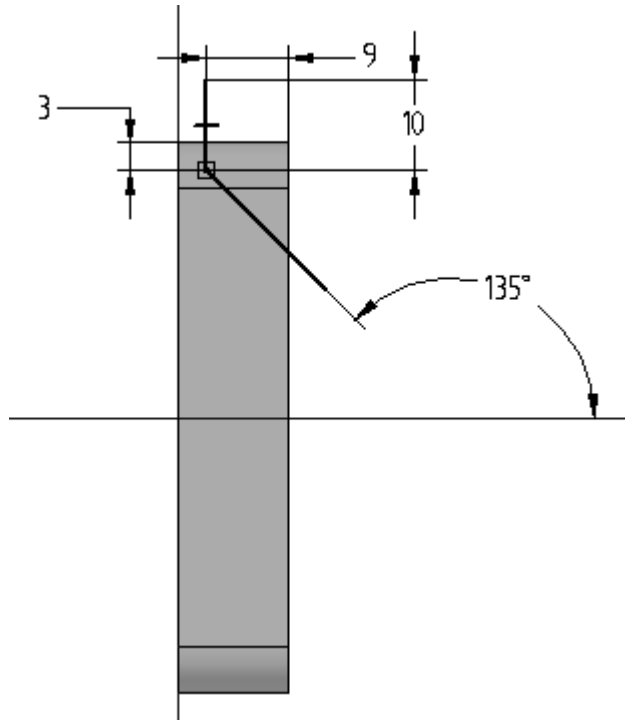
*Add a cutout to the part*

Add a cutout to the part.

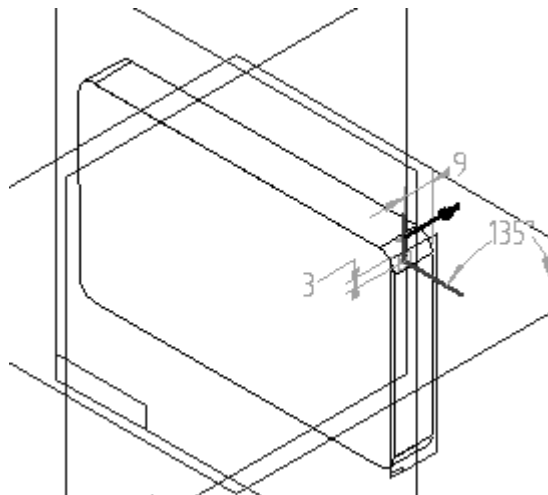
- ▶ Click the Cut command.
- ▶ Select the face shown for the profile plane.



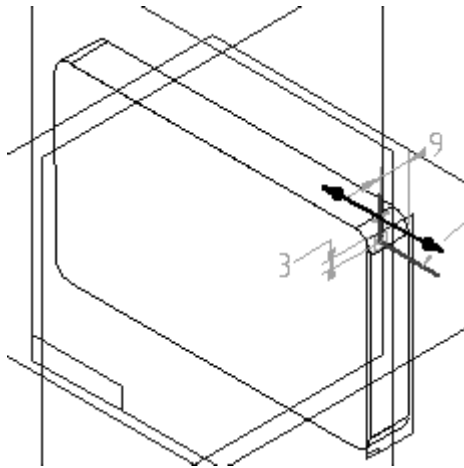
- ▶ Draw and dimension the profile as shown. Click Close Sketch.



- ▶ Position the arrow as shown and click for direction of material removal.



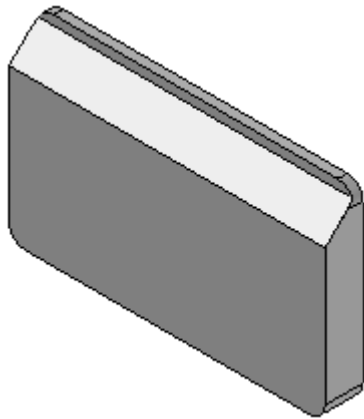
- ▶ Select the Through All Extent option and set the direction arrow to both directions as shown and click.



- ▶ Click Finish.

**Note**

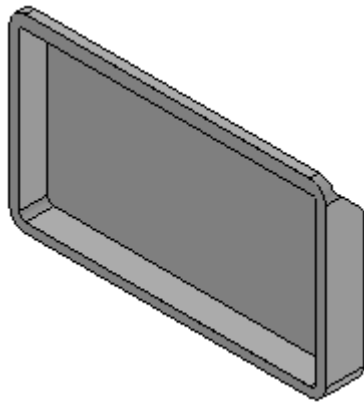
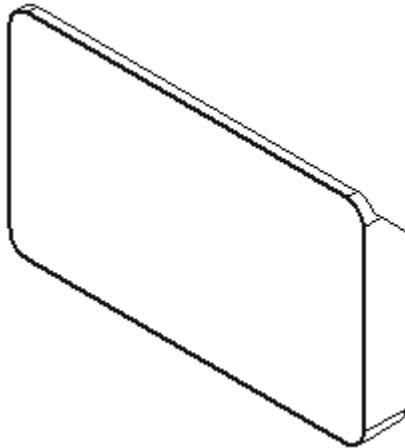
The view was rotated 180° for the image below.

*Thin wall the part*

Apply thickness to the part using the thin wall command.

- ▶ Click the Thin Wall command.
- ▶ On the command bar, type 3 for the common thickness.

- ▶ Select the face shown as the open face. Click Preview and then Finish.



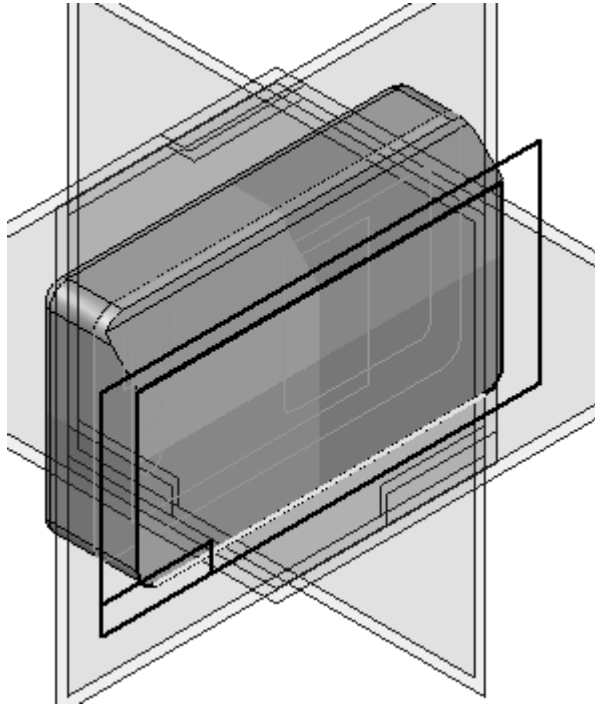
*Add another cutout*

Add another cutout using elements from the assembly sketch.

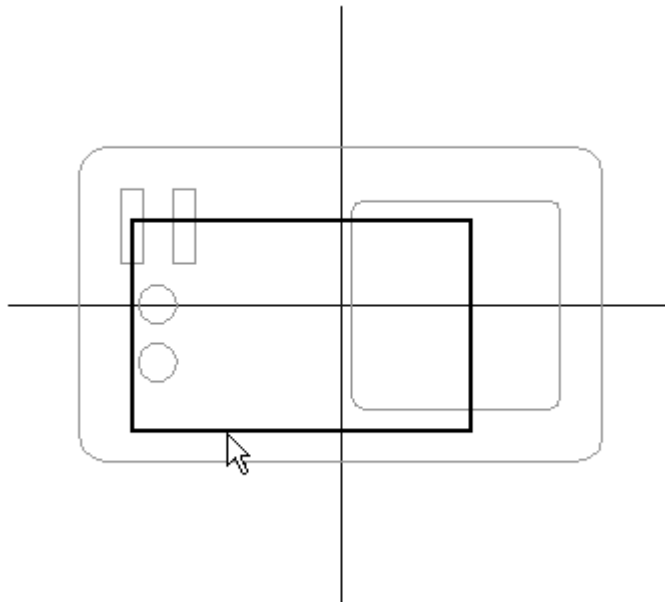
- ▶ Click the Cut command.
- ▶ Click View® Show® Hide Previous Level.



- ▶ Select the face shown for the profile plane.

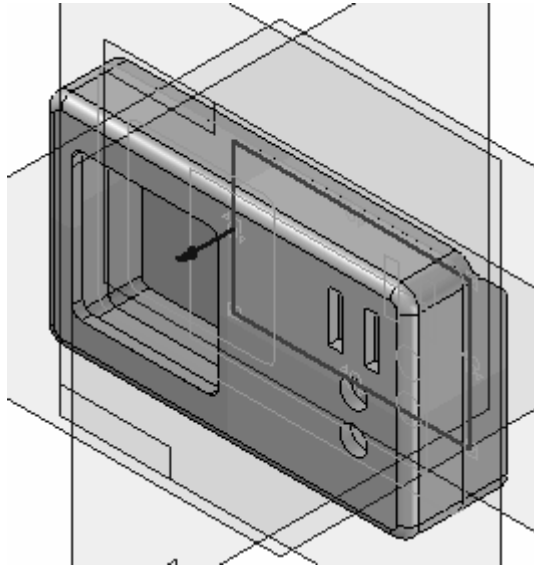


- ▶ Click the Include command. Click OK in the dialog box.
- ▶ On the command bar, set the Select field to Wireframe Chain.
- ▶ Select the wireframe chain as shown and click the Accept button. Click Close Sketch.

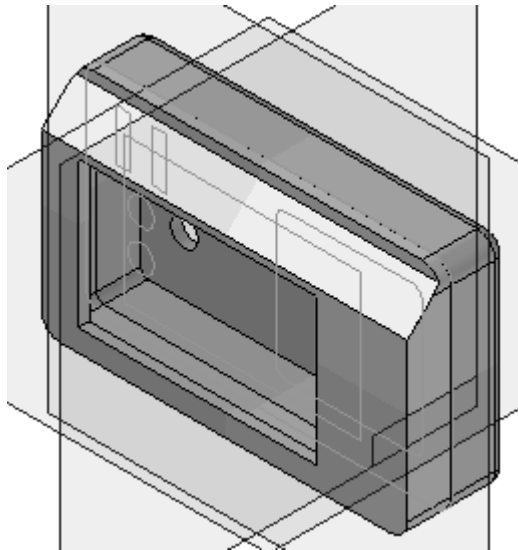




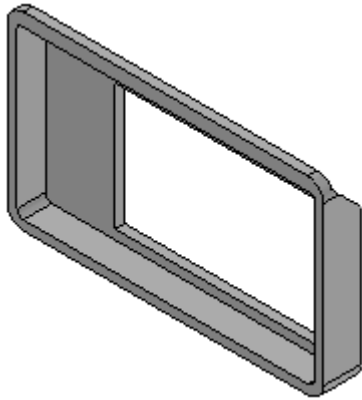
- ▶ Click the Through Next Extent option. Set the extent direction as shown and click.



- ▶ Click Finish.



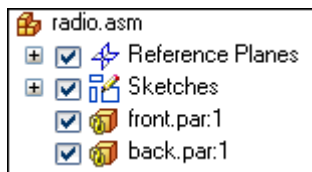
The back part is now complete.



- ▶ Click Close and Return to return to the assembly file *radio.asm*.



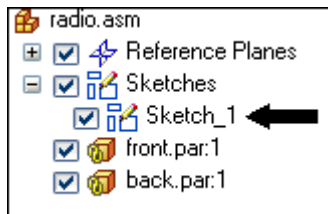
- ▶ Click the Assembly PathFinder tab and notice that the part just created is now a part of the assembly.



*Edit the assembly sketch and observe changes*

Now that two parts are created that are driven by the assembly sketch, edit the assembly sketch to observe how the two parts update to the sketch changes.

- ▶ Click the Select tool.
- ▶ In Assembly PathFinder, click the assembly sketch named *Sketch\_1*.



- ▶ Click the Edit Definition button.



- ▶ Click the Draw Profile step in the command bar.

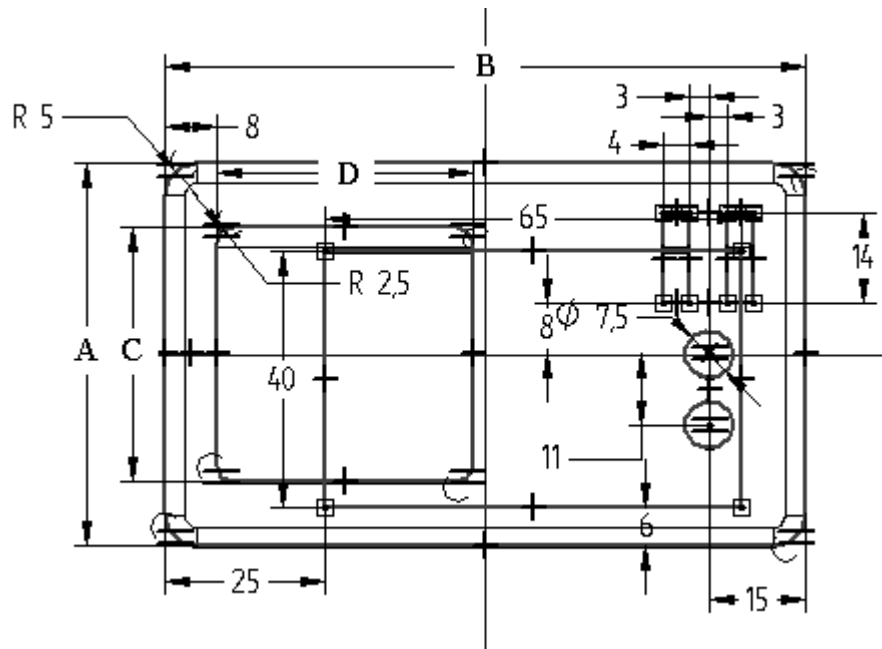
- ▶ Edit the dimensions as shown.

A=80

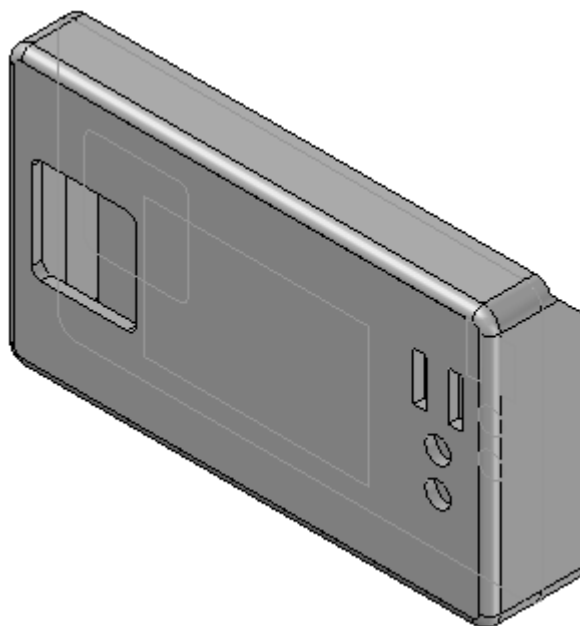
B=140

C=30

D=30



- ▶ Click Close Sketch. Click Finish, and notice how the assembly parts update to the changes made to the assembly sketch dimensions.



*Repeat the previous step*

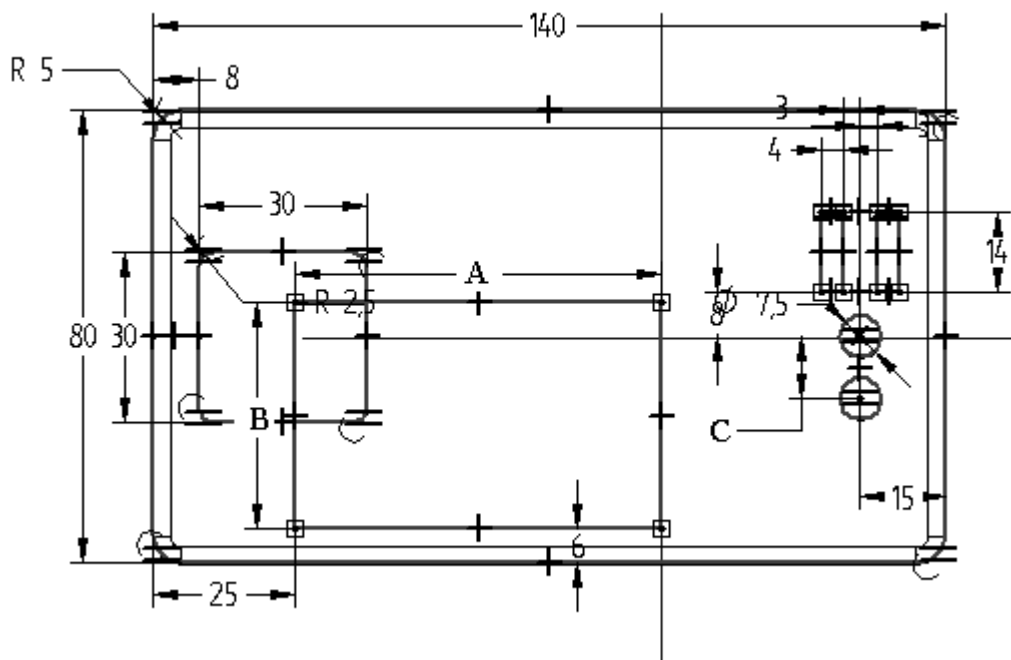
Repeat the previous step to make the following changes to the assembly sketch.

- ▶ Edit the dimensions as shown.

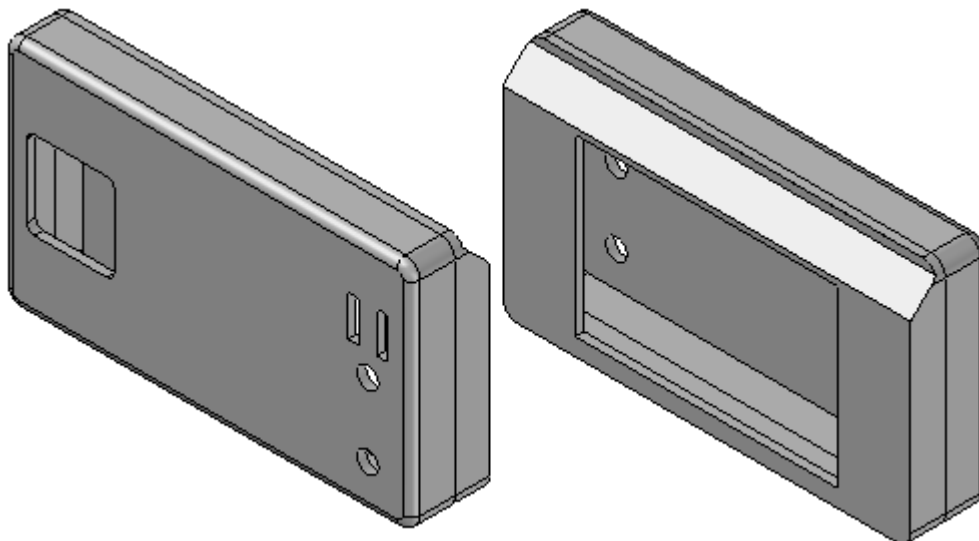
A=90

B=60

C=25



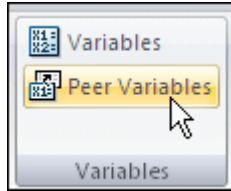
- ▶ Click Close. Click Finish to update the assembly parts.



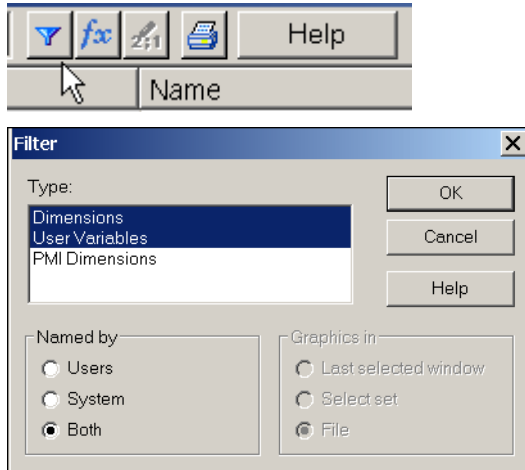
*Link the wall thickness to the two parts*

The final step in the activity is to link the wall thickness variables of the two parts. This will ensure that if a change is made to the wall thickness that both parts will update simultaneously. Use peer variables to accomplish this.

- Click Tools® Variables® Peer Variables.



- Click *front.par* in the assembly window. A variable table for the selected part displays. To display the dimensions and variables, click the Filter button. Click dimensions and user variables in the type: field and select the radio button for both as shown. Click OK.



front.par:Variable Table							
Type	Name	Value	Rule	Formula	Range	Exp...	Exposed ...
Dim	ExtrudedProtrusion_1_FiniteDepth	15.00 mm				<input type="checkbox"/>	
Dim	Round_1_Radii1	3.00 mm				<input type="checkbox"/>	
Dim	Thinwall_1_Thickness	3.00 mm				<input type="checkbox"/>	
Var	PhysicalProperties_Density	0.00 kg/mm^3	Limit		[0.0...	<input checked="" type="checkbox"/>	Density
Var	PhysicalProperties_Accuracy	0.99	Limit		(0.0...	<input checked="" type="checkbox"/>	Accuracy

- Click the left-most button on the Thinwall\_1\_Thickness variable and then right-click (the variable name may differ slightly). Select Copy Link as shown.

Type	Name	Value
Dim	ExtrudedProtrusion_1_FiniteDepth	15.00 mm
Dim	Round_1_Radii1	3.00 mm
Dim	Thinwall_1_Thickness	3.00 mm
Var	PhysicalProperties_Density	0.00 kg/mm^3
Var	PhysicalProperties_Accuracy	0.99

Cut
Copy
Paste
Columns...
Font...
Search and Replace...
Sort
Edit Links
Copy Link
Paste Link
Filter
Variable Rule Editor

- Click *back.par* in the assembly window. A variable table for the selected part displays.

Type	Name	Value	Rule	Formula
Dim	ExtrudedProtrusion_1_FiniteDepth	12.00 mm		
Dim	V678	9.00 mm		
Dim	V682	3.00 mm		
Dim	V683	10.00 mm		
Dim	V684	135.00 deg		
Dim	Thinwall_1_Thickness	3.00 mm		
Var	PhysicalProperties_Density	0.00 kg/mm^3	Limit	
Var	PhysicalProperties_Accuracy	0.99	Limit	

- Click the left-most button on Thinwall\_3\_Thickness variable and then right-click (the variable name may differ slightly). Select Paste Link.

Type	Name	Value	Rule	Formula	Range
Dim	ExtrudedProtrusion_1_FiniteDepth	12.00 mm			
Dim	V678	9.00 mm			
Dim	V682	3.00 mm			
Dim	V683	10.00 mm			
Dim	V684	135.00 deg			
Dim	Thinwall_1_Thickness	3.00 mm			
Var	PhysicalProperties_Density	0.00 kg/mm^3	Limit		[0.0...
Var	PhysicalProperties_Accuracy	0.99	Limit		(0.0...

Search and Replace...
Sort
Edit Links
Copy Link
Paste Link
Filter
Variable Rule Editor

- Notice the link placed in the formula field for the wall thickness variable for *back.par*. If the wall thickness is edited in *front.par*, *back.par* updates automatically to same wall thickness value.

Dim	Thinwall_3_Thickness	3.00 mm	@'C:\Assembly\front.par!\26a'
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- ▶ Click Close to dismiss the variable table.



- ▶ This completes this activity.

*Activity summary*

In this activity, you learned how to use assembly sketches to create and control the size and shape of parts created in a top down design. Inter-part links between the assembly geometry and the part features were used to control the part features from the assembly sketch. Variables from the assembly can be linked to variables in part files to link dimensions and control geometric behavior as the dimensions change.

## Lesson review

Answer the following questions:

1. What is an assembly layout sketch?
2. What command is used to create a new part or sheet metal document from within the context of an assembly?
3. When will changing the size and shape of geometry in a layout sketch change the size and shape of part or sheet metal components within an assembly?



## **Answers**

1. What is an assembly layout sketch?

An assembly layout sketch is a sketch created in an assembly document defining the positioning of more than one component within the assembly. Creating assembly components from layout sketches utilizes the top down approach of assembly modeling.

2. What command is used to create a new part or sheet metal document from within the context of an assembly?

The create in place command within the assembly environment creates a new part or sheet metal document within the context of an assembly.

3. When will changing the size and shape of geometry in a layout sketch change the size and shape of part or sheet metal components within an assembly?

When creating a part in place using assembly layout sketches, extrusions and sketches in the ordered environment can be linked to the assembly sketches where the assembly sketches drive the geometry.

## **Lesson summary**

In this lesson, you learned how to use assembly sketches to create and control the size and shape of parts created in a top down design. Inter-part links between the assembly geometry and the part features were used to control the part features from the assembly sketch. Variables from the assembly can be linked to variables in part files to link dimensions and control geometric behavior as the dimensions change.

## **Lesson review**

1. Can the two methods named above be used in combination? Explain.
2. What method of Top down design allows you to design an assembly structure prior to having geometrically accurate parts available? How are these parts positioned? Can existing parts be used in harmony with yet to be designed parts in this structure? What are these components called?
3. What is the key consideration in determining which assembly construction method you should use?
4. What benefit is gained from modeling a part in context of an assembly?
5. What command in Solid Edge assembly is used to create a new part or sheet metal model file? Where is the button for this command located?
6. What command in Solid Edge assembly is used to create associative relationships between parts?
7. Describe the fundamental steps necessary to perform an inter-part copy.
8. If the Include command in part and assembly sketches process is used from the inter-part tab on the options dialog box, what role does the assembly sketch play?
9. If the Cutout button is not active on the assembly feature commands, what must you do to make it active?
10. Where can you view all the inter-part relationships in an assembly?
11. Can an inter-part link be broken? If yes, how?



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Lesson

# 9 *Revising assemblies*

## **Revising assemblies**

## **Objectives**

After completing this lesson, you should be able to:

- Use the Edit and Open In Solid Edge Part commands to make modifications to parts in an assembly.
- Use the Edit Links dialog box to review and modify documents linked to an assembly.
- Use the Update Links command to manage linked documents in an assembly.
- Use View and Markup: Revision Manager to create revisions for an assembly.
- Use the Properties command to review and enter document properties.
- Set the document status and understand the effect of status on the accessibility of documents to different users.
- Generate bill of material reports, and preview parts and assemblies without having to run Solid Edge.

## **Introduction**

Solid Edge supports assembly management, including the process of revising assemblies. View and Markup: Revision Manager lets you copy an assembly, replace parts in it, and create new revisions of the assembly and its parts. Document management tools in Solid Edge enable efficient management of Solid Edge documents and related information. Use these tools to save descriptive information about a document, control access to files, and track the progress of a file through the work cycle. Searching for and retrieving documents is easy with user-defined search criteria. Documents can be routed to a group of users for review, approval, or rejection.

Windows Explorer can access many document management tasks. A shortcut menu displays Solid Edge-related commands when you right-click a Solid Edge file in Windows Explorer.



## **Document properties**

A document's properties are an important part of managing documents. Using document properties, you can store document information with the document itself, instead of in a separate database. This makes it easy to find documents and information about them. Using the File Properties command available from Application menu® Properties, you can view, edit, and store properties for a document.

### **Propseed.txt File**

Propseed.txt is an ASCII text file that is used to populate values on the Properties dialog box. You can use a text editor, such as Notepad, to add values to the file.

By default, the file is located in the Solid Edge Program folder. For example, if you loaded Solid Edge to drive C, the path would be *C:\Program Files\Solid Edge ST4\Program\propseed.txt*.

You can instruct Solid Edge to look for *propseed.txt* in a different folder, including a folder on another machine in the network. To do this, on the Application menu, click Solid Edge Options. On the File Locations tab of the Solid Edge Options dialog box, select Property Seed File and click Modify. On the Browse dialog box, specify the drive and folder containing the *propseed.txt* file. After specifying the location, click Update.

## General and summary properties

Some properties are updated automatically by the software; they can include the title, the author, and keywords that identify important information. These properties also can include document statistics, such as document size and the date that a document was created and last modified.

## Units of measure

The unit of measure settings for a document are stored as a property. When you set the units of measure on the Units tab on the File Properties dialog box, the units of measure for various types of measurements in a document are displayed in the units you specified. For example, when you set the length unit of measure to inches, then the output values for the measurement commands on the Inspect menu that measure elements lengths are displayed in inches.

You can set units of measure for values such as length, area, or angles. You can change the unit of measure at any time in a document.

The precision readout options set the number of significant figures to display. It sets the accuracy of the unit readout value. The precision setting does not alter the numbers that you type, only the display of the numbers in the box. Values ending in 5 are rounded up. For example, if the precision readout is .123 and you draw a line that is 2.1056 inches long, then the line value length is rounded. The length value appears as 2.106 inches long. If you are using mm as your drawing sheet units, you can have the values display as 3.5 mm or 3.50 mm.

### Note

The units of measure settings for a document do not affect the units used for dimensions you place with the dimensioning commands.

You specify the dimensional value units you want with the Style option on the Dimension command bar. You can modify the delivered dimension styles and create new dimension styles using the Style command on the Format menu. To make it easy to reuse the unit settings you want, you should define and use document templates. You can also set the units used for an individual dimension using the Properties command on the shortcut menu when you select a dimension.

## Document status

You can track or change the status of a document (for example, Available, In Work, Released, Baselined) in the Status page of the File Properties dialog box.

### Note

The Status page is not available in Insight XT which uses custom properties to display status information.

You can determine the status of a document within its life cycle or browse parts of the document tree. A document tree consists of a top-level document and subdocuments. The status of a document can be updated using the Update Status Info command, available from the shortcut menu in PathFinder while no item is highlighted.

## Changing Document status

The software tracks the status of a document with a status property. When you create a document, the default status property is set to Available. If you want to work with a document for an extended time, you can use Property Manager to change the status to In Work. The software then stores your user name with the document. If you want to review information in a document, you can change the status to In Review.

When a document's status is set to In Work, the document cannot be opened for edit by another user. However, a read-only copy of the document is available for review. Anyone attempting to access the document can choose to be notified when the document becomes available. When you have completed work on a document, you can make it available to other users by using the Property Manager to change the document status to Available.

### Note

Document status available to document in Insight XT are: Working, In Review, In Approval, Released, and Obsolete.

## Preview

You can save a bitmap picture of a document's graphic content and save it as a property. The bitmap picture can be used in the Open dialog box and in Windows Windows Explorer to display a preview of the document.

You can also create a custom large icon to easily view the contents of the document. The custom large icon is displayed in the Solid Edge Find Files window when you select the Large Icon View option in Solid Edge Find Files.

## Custom properties

You can define custom properties to store information regarding your documents. The custom property information is stored within the document and viewable on the Custom tab of the Properties dialog box. For example, Density and Accuracy are custom properties.

In Insight XT, custom properties store status information. Custom properties created as a result of saving or opening a Insight XT managed document are:

PDM\_Status\_ID

Contains the value associated with the status.

PDM\_Status

Displays the current status of the document (Working, In Review). This property is owned by Insight XT (/master=iman).

PDM \_Status\_User

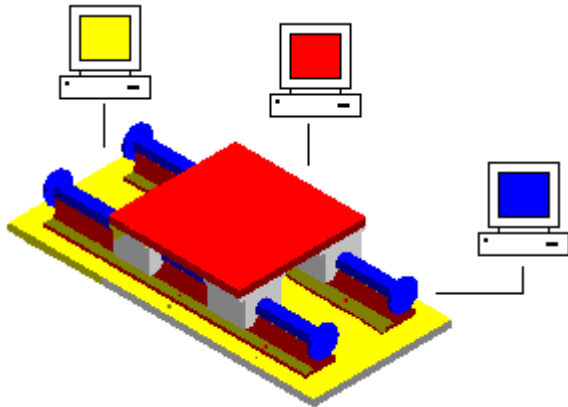
Displays the user of the document. This property is owned by Insight XT (/master=iman).

PDM \_Status\_Date

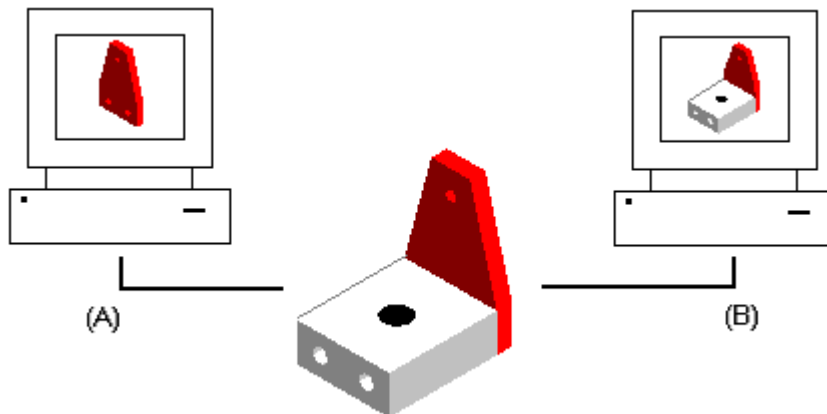
Displays the date the document is accessed.

## Concurrent design

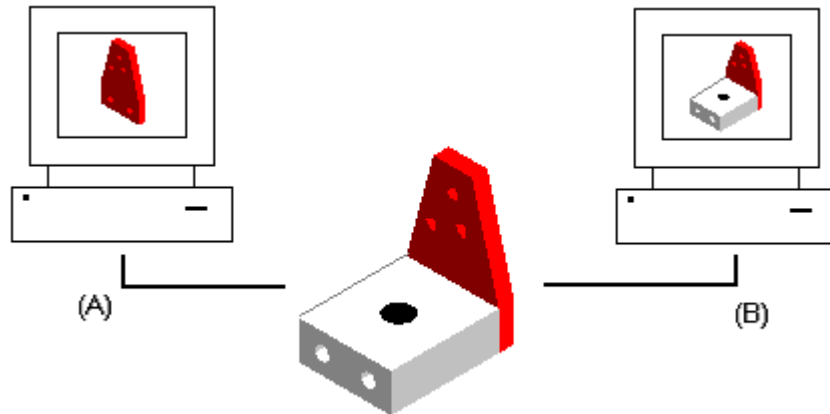
Concurrent design allows multiple users to access an assembly document simultaneously. To assist in controlling concurrent access, you can define the status of parts and assemblies with the Properties dialog box. You can provide read-only access to the top-level assembly and write-access to the parts and sub-assemblies. This allows multiple users to actually work in the context of the same assembly at the same time, but on different parts.



Another advantage of concurrent design is the ability to update the display to see the changes made to the assembly by other users. In the following example, user A can open a part in an assembly. At the same time, user B can display a read-only version of the entire assembly.



As user A places two additional holes in the part, user B can update the display of the assembly to view any changes made to the assembly.



### Baselining documents

You can use the Property Manager command to baseline a document, freezing it and preventing further modification. Documents linked to a baselined document are not frozen. You must either baseline or release any linked documents that you want to freeze from modifications.

The baselined document can then be used as a point of reference if changes are made downstream. Baselining a document also forces later modifications to be made in a new version of the document, instead of the current version.

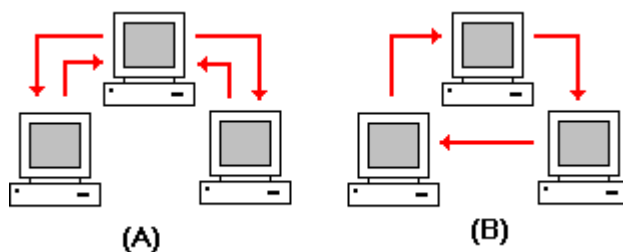
When you baseline a base document, the individual documents are not automatically set to baselined. You have to manually set the documents that you want to baseline. You can also set the status of the individual documents to released.

### Releasing documents

Base documents contain all the links for individual documents. Individual documents can contain parts, floorplans, drawings, and so forth. You can use the Property Manager command to release a base document and its individual documents at the same time, or you can release individual documents.

### Routing documents

Routing slips can be used to send documents to one or more users for review. A routing slip allows you to list the users that should receive mail containing the attached document. Documents can be routed to (A) all users at one time or (B) users one after another.



When you route a document to users one after another, and one user completes a review, the document is routed to the next user in the list. When you receive routed documents, you can double-click the attached document in the mail message to activate the appropriate application for viewing the document.

The routing slip can contain a sign-off request for approval or rejection of the contents. The software records all sign-off results. When all the users in the list have approved the document, you can use the Properties command to change the document status to released to freeze the document and prevent further modification.

**Note**

Routing is disabled for managed documents.

**Managing document properties**

You can use the Property Manager command to modify existing properties or create new properties for a document in Solid Edge or Insight Connect. The command displays the Property Manager Select dialog box that allows you to select folders and files that contain properties that you want to edit. The dialog box reacts differently depending on whether or not you have an active document.

If you have an active document in Solid Edge, the Property Manager Select dialog box is not displayed. The Property Manager dialog box is displayed to allow you to edit the properties of the active document. You can use the No Levels and BOM View options on the Property Manager dialog box to display the document in a list view with no levels or a BOM view with levels. By default, the view is a list view.

If you have an active document in Insight Connect, the Property Manager Select dialog box is displayed with both the BOM View For and User View options enabled. This allows you to edit the properties based on the BOM structure of the active document or based on a selected list of folders and documents.

If you have no documents open in Solid Edge or Insight Connect, the Property Manager Select dialog box is displayed with User View option disabled and the BOM View For option enabled. You can click the folders and files that contain properties you want to edit and click the Add button to add them to the Edit Properties box. You can select the OK button to display the Property Manager dialog box and edit the properties for selected documents.

## **Editing document properties**

Once you make selections on the Property Manager Select dialog box and click the OK button, the Property Manager dialog box is displayed to allow you to edit property values. Any properties that you cannot edit are disabled and appear in gray.

To edit a value, click on the appropriate property cell and type in the new value. When you edit a property, the document containing the property is checked out to prevent others from making changes. After you edit a property value, the property cell is underlined to indicate that it has been changed. The cell remains underlined until you click the Save button to save the changes or click the Restore button to set the value back to the previous value. You can use the Copy, Cut, and Paste buttons to edit information between cells.

### **Note**

The Save button is disabled when you use the Property Manager command within Solid Edge. To save your changes, use the Save command on the File menu to maintain assembly integrity.

You can use the Show Properties dialog box to specify the properties you want to display on the Property Manager dialog box. To display the dialog box, in the list of displayed documents and folders in the Property Manager dialog box, click the right mouse button on any row containing information, and then click Show Properties on the shortcut menu.

You can add properties to the list that is displayed on the Property Manager dialog box. To add a property to the displayed list, on the Show Properties dialog box, select the property in the Available Properties box and click the Add button to add the property to the Show These Properties in This Order Box. You can use the Select Available Properties From pulldown menu to control the list of available properties.

You can remove properties from the list that is displayed on the Property Manager dialog box. To remove a property from the displayed list, on the Show Properties dialog box, select the property in the Show These Properties in This Order box, and click the Remove button.

You can use the Move Up and Move Down buttons to change the display order for the properties. Use the Move Up button to move the property further to the left in the list and the Move Down button to move the property further to the right.

### **Note**

You can select a column heading in the Property Manager dialog box, then use drag and drop to change the order of the columns.

To create new document properties, click the New button to display the New Property dialog box. After you add the information for the new property and click the OK button, the property appears in the Show These Properties in This Order box. When you click the OK button on the Show Properties dialog box, the new property is displayed in the list.

You can use the Files of Type option to control the types of documents that appear in the list of documents. You can edit the properties for non-Solid Edge documents. However, for non-Solid Edge documents, you can only edit profile properties for managed documents.

**Note**

The Files of Type option is disabled in the BOM view.

You can edit the properties for both managed and non-managed documents. However, you cannot edit both types at the same time. You determine the type of documents you want to edit when you select the first folder or document on the Property Manager Select dialog box. You must be able to check out a managed document to edit the document properties. If you edit a property for a managed document that is not checked out, the document is checked out when you make the change, but the cache is not updated with linked documents. You can change the document profile for a managed document if you have the Profile property displayed on the Property Manager dialog box.

If a property for a managed document belongs to the workspace, but does not belong to a profile, then the property is assigned a String data type. For example, suppose a property, Color, belongs to a workspace named //servername/workspace. The workspace contains a profile, SampleProfile, that does not contain the property Color. Therefore none of the documents associated with SampleProfile contain the property. If you use the Property Manager command and select a managed document that is associated with SampleProfile, the data type of the selected property for the document is String or Text. If you edit the property, the Color property is added to the document with a data type of Text.

You can copy and paste SharePoint property values between parts. You should understand that the copy and paste procedure uses the *what you see is what you get* method. For example, suppose you have a SharePoint property that is defined to display three decimal places. Part A in Assembly 1 has a value of 5.12493 for a numeric property. If you open the assembly in Property Manager, the property for Part A displays 5.125. If you copy and paste this numeric value into Part B of Assembly 1, the defined property value is the displayed value of 5.125 instead of the actual value of 5.12493.



## **Property display**

The display of property values in the list depends on whether or not the property belongs to the profile. If the property belongs to the profile, the list displays the property value for managed document. If the property does not belong to the profile, the property is either a standard or custom property for the document and the list displays the property value from the unmanaged document.

For example, suppose you use the Property Manager command to change the value of a profile property named Width from 20 to 25. Then, you save the document, but do not close it in Solid Edge. If you re-run the Property Manager command, the value displayed in the list is still 20. Since you have not checked in the document that contains the changes to the Width property, the value for the property in the library is still 20.

## **Property synchronization during upload and download**

Each time you upload documents to a workspace or use the Create Unmanaged Copy command to create an unmanaged copy of a document, Solid Edge synchronizes the profile and document properties. Solid Edge creates custom properties in the document for the profile properties that don't exist in the document and then populates the custom property with the profile property value.

Only properties for documents that support standard property storage are synchronized. If the document does not support standard property storage, it is skipped and no error is reported.

When uploading a file, if the profile and document contain different values for the same property, the document property becomes the default value.

When using the Create Unmanaged Copy command, if a custom property already exists, Solid Edge checks to make sure the values for the profile property and custom property match. If the values are different, profile property becomes the default value.

When downloading a file or using the Create Unmanaged Copy command, if there is a custom property that matches a Solid Edge system property, such as document number, Solid Edge copies the value of the custom property to the system property, and then removes the custom property. This only applies to Solid Edge documents. If there is a Solid Edge system property defined as a custom property in a non-native document, the property is not removed from the non-native document.

## **SharePoint Version 2003 properties**

The Property Manager command, Edit Profile command, Create Unmanaged Copy command, Add Library command, report generation, and advanced search results support the lookup table, currency, and calculated properties delivered with SharePoint 2003.

## Lookup properties

When working with lookup properties, administrators can specify a SharePoint list and a specific property used to generate the lookup list. Insight uses the list of values to create a list from which you can choose.

If you modify a lookup property with the Solid Edge Custom Property tab and enter a value that is not valid according to the lookup, Property Manager or Edit Profile displays the last valid value for the property that is stored by SharePoint. For example, you can define a lookup property named supplier with three values: SupplierA, SupplierB, and SupplierC. Suppose you create a new file and check the file in, selecting SupplierA for the supplier on the Edit Profile dialog box. You then re-open the part and, on the Custom tab of the Properties dialog box, change the value for supplier to SupplierD. When you attempt to close and save the file, Edit Profile displays SupplierA as the supplier, since SupplierD is not a valid value.

## Currency properties

Solid Edge supports all of the currencies and countries supported by SharePoint 2003. Currency properties are stored as numeric custom properties on the Custom Properties tab of the Properties dialog box.

## Calculated properties

Calculated properties are added as custom properties during property synchronization with the appropriate data type, such as numeric, string, or date, based on the SharePoint data type used to display the calculation. If you manually edit a custom property that is synchronized with a calculated property, your edits are overwritten with the value from the document library.

### Note

Calculated property values are not updated until the document is checked back into the workspace, even if the properties changed during a Solid Edge session.

## Formatting SharePoint properties

Solid Edge allows you to format information for the minimum/maximum value, number of decimal places, and show as percentage properties.

The minimum and maximum values for numeric properties are displayed as part of the tool tip in error messages. For example, the text in the error message for a value that exceeded the maximum value displays *The value <value entered> is above the maximum value of <property maximum value> for <property name>. Edit the property and enter a valid value.*

You can set the number of places for decimal properties to display between 0 and 5 decimal places. Property values of 0 - 4 are rounded down to the nearest decimal place and values of 5 - 9 are rounded up to the nearest decimal value. For example, if decimal places is set to 3 for .1234, the value is displayed as .123.

The value you enter for a Show as Percentage property is multiplied by 100 when it is displayed. When you enter a value, it is divided by 100 when is stored in the document.

## Standard and Custom properties

The properties used to define the document name formula are available to unmanaged and managed documents.

### Note

Properties shown in *italics* are custom properties.

<b>Part</b>	<b>Assembly or Draft</b>	<b>Sheet Metal</b>
Filename	Filename	Filename
Material	Material	Material
Occurrence	Occurrence	Occurrence
Title	Title	Title
Subject	Subject	Subject
Author	Author	Author
Manager	Manager	Manager
Company	Company	Company
Category	Category	Category
Keywords	Keywords	Keywords
Document Number	Document Number	Document Number
Revision Number	Revision Number	Revision Number
Project Name	Project Name	Project Name
<i>Density</i>		<i>Density</i>
<i>Accuracy</i>		<i>Accuracy</i>
		<i>Material Thickness</i>
		<i>Bend Radius</i>
		<i>Relief Width</i>
		<i>Relief Length</i>
		<i>Neutral Factor</i>
		<i>Minimum Arc Length</i>
		<i>Deviational Tolerance</i>

Many of these properties are accessible through the Summary and Project tabs on the File Properties dialog box.

### Note

Solid Edge Embedded Client displays properties in addition to what is listed here due to the properties available in Teamcenter.

**Activity: Revision Manager (release and revise)****Overview**

In this activity, you learn how to change the status of a part and draft document to released and how to make a revision to a released part and draft and move files to a new location. You also learn how Revision Manager can be used to copy and rename whole or parts of an assembly and preserve the links to the parts.

**Activity: Revision Manager (release and revise)****Overview**

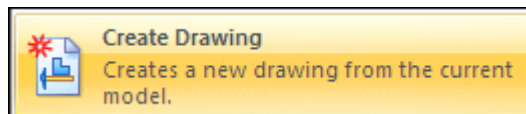
In this activity, you learn how to change the status of a part and draft document to released. You also learn how to make a revision to a released part and draft and move files to a new location.

*Open the part*

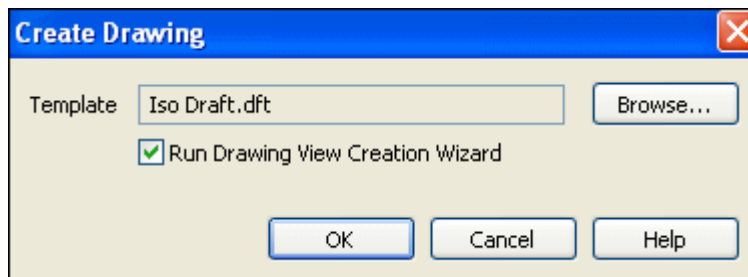
- Open *part\_a1.par*.

*Create a drawing*

- Click the Application button. Choose New® Create Drawing.

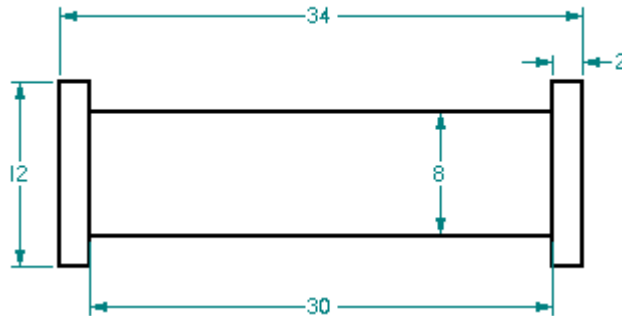


- Click OK in the Create Drawing dialog box.



- On the Drawing View Creation Wizard, click Next.
- Under Named Views, select front and then click Next.
- Click Finish.
- Place the view in the center of the drawing sheet and then click Fit.

- ▶ Use the Smart Dimension command to place the dimensions. On the command bar, set the dimension style to ISO and the text scale to be 5.0.



- ▶ Close and save the draft file as *part\_a1.dft*. The design is approved.
- ▶ Close and save the part file as *part\_a1.par*.

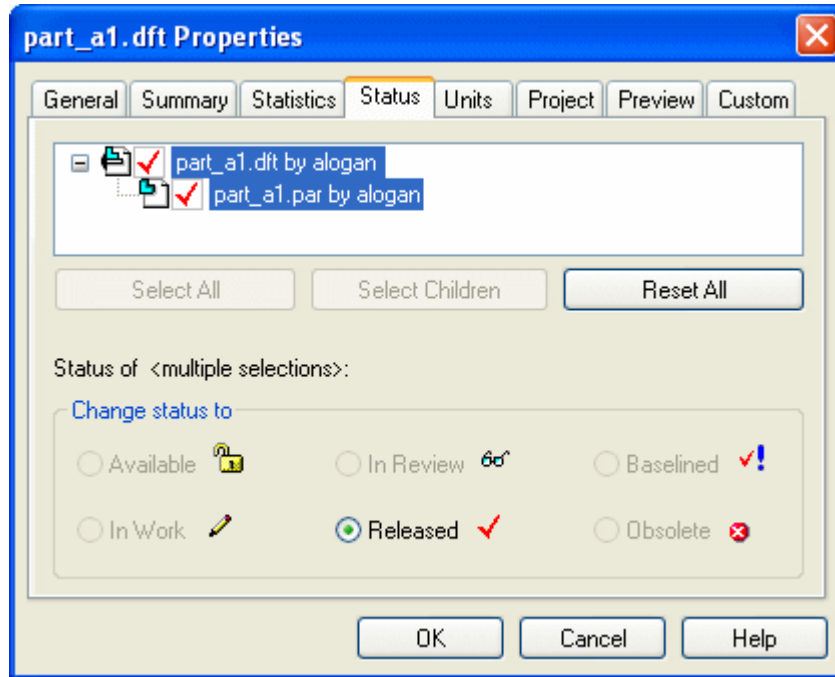
*Change the status of the draft and part files to Released*

- ▶ In Windows Explorer, browse for *part\_a1.dft* in the folder containing the where the activity files are located. Right-click the file, and then click Properties.
- ▶ In the Properties dialog box, click the Status tab.
- ▶ Click Select All.

### Note

This applies the status change to both the draft and part files.

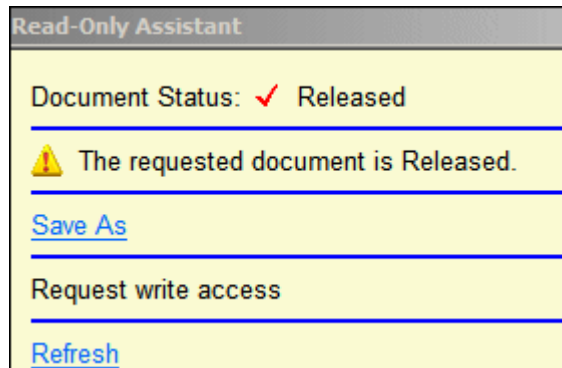
- ▶ Click Released, and then click OK.



- ▶ Both *part\_a1.dft* and *part\_a1.par* are read only because the status has been set to released.

#### Note

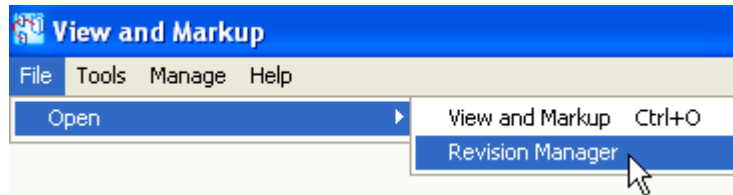
If you attempt to open the released file *part\_a1.par*, you see the following message box.



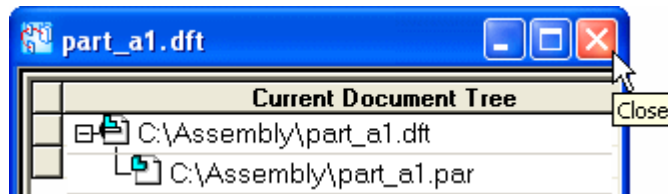
*Create a revision of the released part and draft file*

- ▶ Start View And Markup. Click Start® Programs® Solid Edge ST4® View and Markup.

- ▶ To start Revision Manager, click File® Open® Revision Manager.

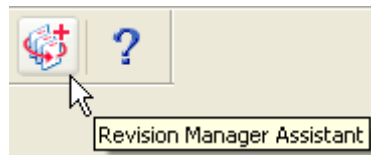


- ▶ In the Open a File dialog box, click *part\_a1.dft* and then click Open.
- ▶ Close the window.

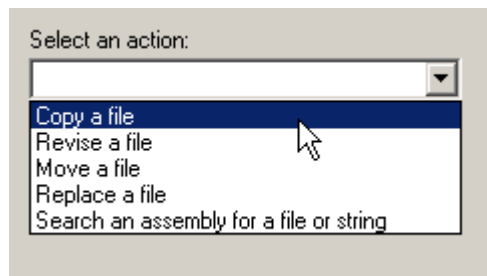


#### Use Revision Manager Assistant

- ▶ Click the Revision Manager Assistant.



- ▶ In the Select an action list, click Copy a file, and then click Next.

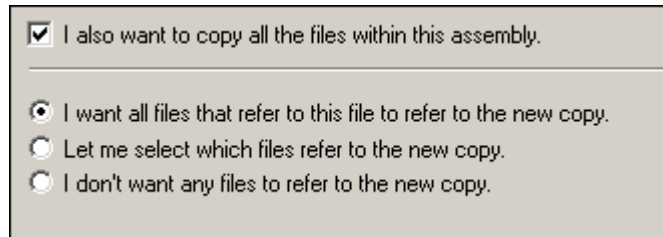


#### Note

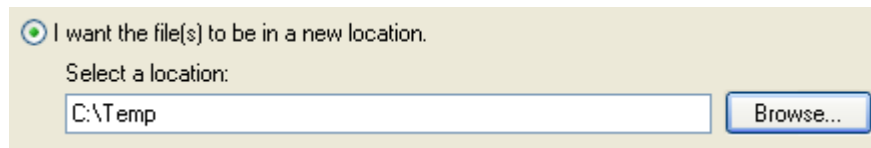
The Revise a file option only refers to managed documents. For unmanaged documents, choose the action to Copy a file.



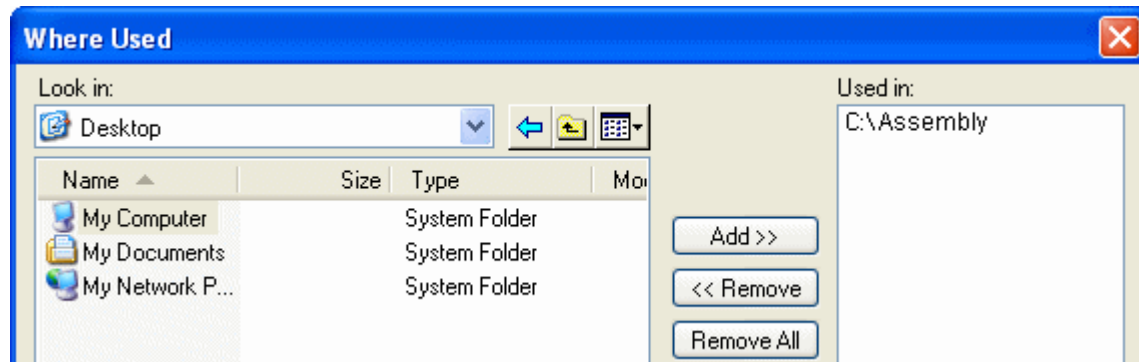
- ▶ When prompted to Select a file, click Browse to locate *part\_a1.dft*. Browse to the working folder for this document. Make sure that the other options are checked as shown, and then click Next.



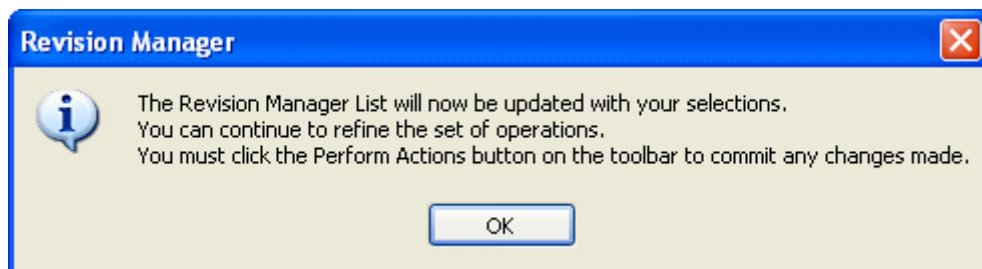
- ▶ Browse to C:/Temp as the new location for the copied files, and then click Next.



- ▶ In the Where Used dialog box, click Next.

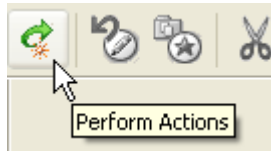


- ▶ Click Finish, and then click OK In the Revision Manager information box.



- ▶ In the New Filename column, rename the files. Rename the file *part\_a1.dft* to *parta1-1.dft*, and the new filename for *part\_a1.par* is *parta1-1.par*.

- ▶ On the Main toolbar, click Perform Actions to create the copy.



part_a1.dft						
	Current Document Tree	Size	Action	New Filename	Doc Number	Revision
	C:\Assembly\part_a1.dft	177KB	Copy	C:\Temp\part_a1-1.dft		1
	C:\Assembly\part_a1.par	206KB	Copy	C:\Temp\part_a1-1.par		1

part_a1-1.dft				
	Current Document Tree	Size	Action	New Filename
	C:\Temp\part_a1-1.dft	229KB	Unchanged	
	C:\Temp\part_a1-1.par	208KB	Unchanged	

### Note

These new files can now be revised.

- ▶ On the ribbon, click File@ Close.  
This completes the activity of learning the process for creating file revisions.

### Summary

#### Activity Summary

In this activity, you learned how to change the status of a part and draft document to released and how to make a revision to a released part and draft and move files to a new location.

## Revision Manager

Document relationship management is a complex information management problem facing businesses and industries. As more information is being shared and reused between documents, tracking revisions of documents and the links they contain becomes increasingly difficult.

Making a new revision of an existing document while maintaining the original version is a part of every document's life cycle. When you select a document for revision, Revision Manager displays a hierarchy of the related documents that may or may not need to be revised as well. Revision Manager lets you do such tasks as:

- Copy and rename documents selected for revision to locations you specify.
- Update the revision and document numbers you specify.
- Update or maintain references in the document hierarchy.
- Show the size of documents.
- Copy a row or rows of data to the clipboard for printing.
- Save previous query paths.
- Preview a document.

## Revising managed documents

When you add a document to a managed library on a server, it becomes a managed document. The Set Action To Revise command, on the Manage menu in Revision Manager, revises managed documents. When you select the command, the existing path and an incremented version of the existing document name are placed in the New Filename column on the Contents window. You must have a managed library folder as part of the new document path. If you set the Get Latest Version When Opening Document option on the Manage page of the Options dialog box, the latest version of the document is downloaded to the local cache before it is revised.

You can revise a single document or multiple documents. When updating multiple documents, all of the selected documents must be managed. Since the original document does not change when you revise it, it is not checked out of the library.

If you use the Set Action to Revise command to revise a document in the context of an assembly, the link in the assembly updates to point to the new revision. However, you need to run the Where Used command to find where the document is used and manually update any other documents you want linked to the new revision.

If you use the Set Action to Revise command to revise a 3D document, a Where Used search is performed for associated draft documents and you can choose to revise them as well. Draft documents with a status of Obsolete are ignored in the Where Used search.

The Cache Assistant dialog box contains two commands that work together with the Set Action to Revise command to provide information for revised documents. The Revised From command, on the shortcut menu in Cache Assistant, displays the name and folder path of the document that was used as a template to create the selected document. The Show Revisions command, on the shortcut menu in Cache

Assistant, displays the name and folder path of any documents that are revisions of the selected document.

### **Revision Manager Assistant**

The Revision Manager Assistant helps you perform a variety of basic document revision tasks. The assistant presents a list of common operations to you. You can select from the list of operations and create a streamlined set of subsequent steps to be completed. To use the Revision Manager Assistant command, on the Main toolbar, click the Revision Manager Assistant button.

### **Disabling Revision Manager**

If you work strictly with viewing and marking up Solid Edge data, you may not be concerned with the management of the data. Therefore, your needs may be limited to those commands need to view and mark up the data. You can use Solid Edge Administrator to disable Revision Manager and limit user access to Revision Manager commands and other Insight utilities. To disable the option, start Solid Edge Administrator and set the Allow Override field to Yes. Set the Value field to Yes. Once disabled, an error message is displayed if you select Revision Manager from the Start menu. If you disable Revision Manager and start Insight Connect, the default environment will be View and Markup.

## Redefining links

When you move documents from one location to another, sometimes the links to these documents break. The Redefine Links command, located on the Tools menu, allows you to redefine the links on a selected set of files. The command does not actually copy or move files, but instead allows you to redefine links in assemblies after the files have already been moved.

The Redefine Links command presents a series of dialog boxes that allows you to define information needed to redefine the links.

The first dialog box allows you to select the folders or documents containing the links that you want to define. You can choose from several filters to specify the location and type of files you want to process. You can specify any type of Solid Edge file or you can choose to process files listed in an ASCII text file. When you have determined the location of the files, you can process a selected list of files, all files within a selected folder, or all files within a selected folder and any subfolders. Once you have selected the files you want to process, click the OK button to proceed to the next dialog box.

The second dialog box allows you to specify the link path redefinition. First, you specify the current link path and then specify the path to which you want to redefine the link. The dialog box provides an example of these link path definitions work. Once you have specified the link path redefinition, click the OK button to proceed to the next dialog box.

### Note

Redefine Links only searches and replaces the first occurrence of the Current Link Path. It will not search the BOM tree and replace other occurrences that you may not be aware of.

The third dialog box confirms the information you entered in the previous dialog box. It lists the number of files you have selected and the path redefinition information. The dialog box also contains a very important option, Do not Save Files When Links Point to Non-existent Files. If you do not select this option and attempt to process a link that points to a non-existent file, the system will open the parent assembly and save the redefined link path string in the parent assembly. An error is written to the log file stating that the non-existent files could not be found. If you select the option, the system does not open and save the redefined link path string in the parent assembly, but does write an error message to the log file stating that the non-existent files could not be found. Once you have confirmed this information, click the OK button to proceed to the final dialog box.

The final dialog box tracks the processing of the selected files. It specifies the number of the number of documents processed and the number of links repaired. You can click the Stop button to stop the processing. You can also click the View button to view the ReDefLinksLog.txt file, which contains information about the links you redefined. Once you have completed redefining the links, you can click the Finish button.

### **Redefined Links Log Files**

Two log files are created when you redefine links. These files are located in the same directory that contains the files being processed.

ReDefLinksLog.txt is an ASCII text file that contains information about the files being processed. It contains general information, such time, date, login name, and nodename. It also contains a list of all results, both successful replacements and problems encountered.

ReDefLinksErr.txt contains the same information as ReDefLinksLog.txt except for the successful replacements. This is very useful when you are processing a large number of files, most of which are successful. You can check to the errors log to quickly see the files that need correcting. To view these files, on Redefine Links Step 3 of 4 dialog box, select the log file you want to view and click the View button.

## **Activity: Revision Manager (revise and release an assembly draft)**

### **Overview**

In this activity, you learn how to release a draft of an assembly. You will revise the draft, assembly and a single part to a new location. You will edit a part, update the assembly and draft. When complete, you will re-release the draft.

**Activity: Revision Manager (revise and release an assembly draft)****Overview**

In this activity you release a draft of an assembly. You will then revise the draft, assembly and a single part to a new location. You will then edit a part, update the assembly and draft. When complete, you will re-release the draft.

*Open the draft file*

Open *module.dft*.

*Release the draft file*

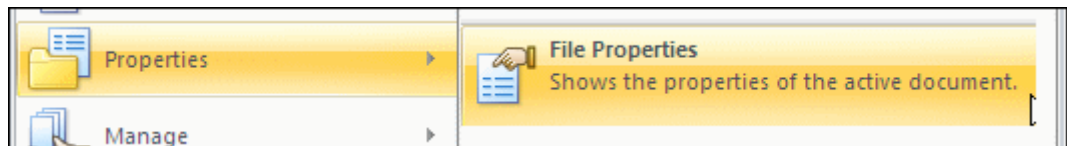
Release the draft file *module.dft*.

**Note**

A draft cannot be released unless its children (assembly, parts) have been released.

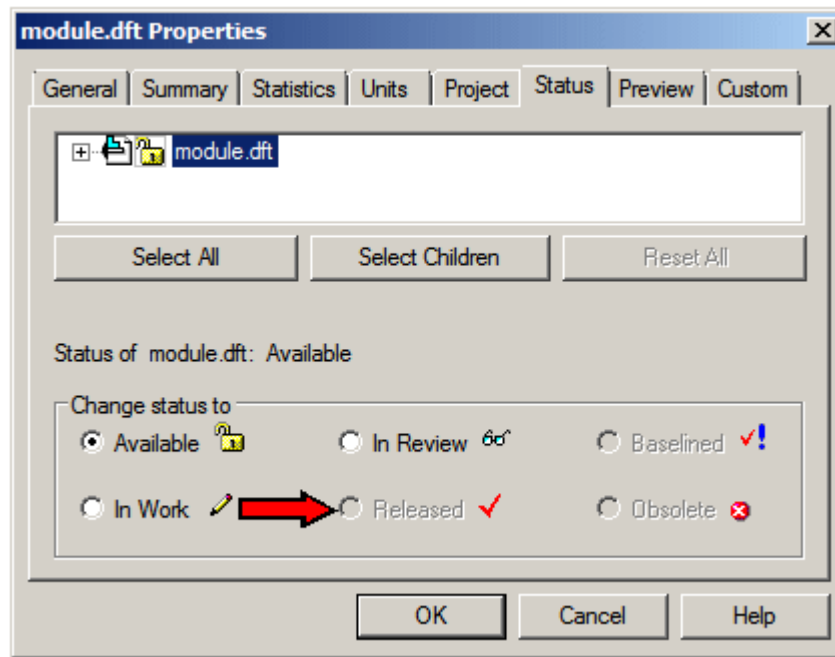


- ▶ Click the Application button.
- ▶ Choose Properties® file Properties.



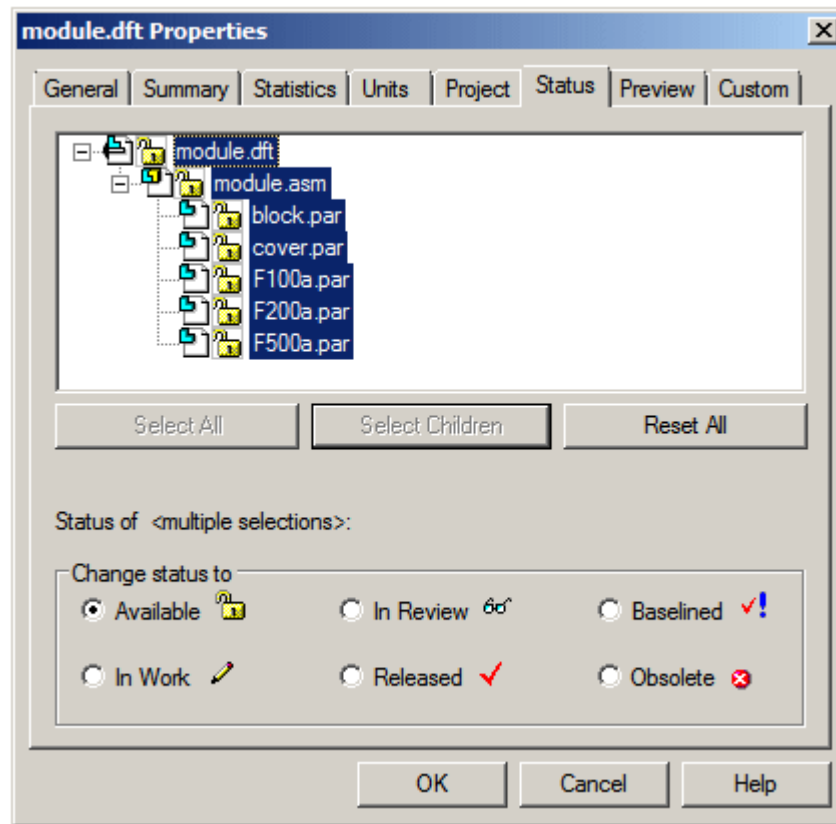


- ▶ In the Properties dialog box, click the Status tab.  
Notice that the Released option is unavailable.



- ▶ Click the + to expand the display of the files associated with *module.dft*.

- ▶ Click Select All, and then click Select Children and click the Released button. Click OK.



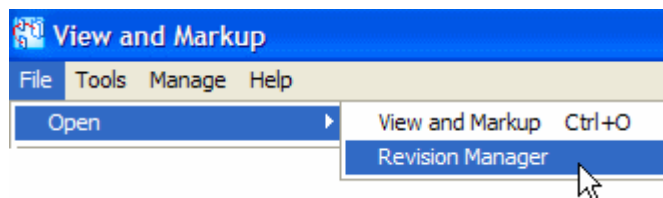
- ▶ Close and save *module.dft*. At this point, the draft, assembly and part files are released.
- ▶ Exit Solid Edge.

### Create a revision

A design change is needed to one part in the assembly. Create a revision of the draft, assembly and the part that requires a design change.

- ▶ Start View and Markup. Click Start® Programs® Solid Edge ST4® View and Markup.
- ▶ Start Revision Manager.

Click File® Open® Revision Manager.



- ▶ Open *module.asm*.

- ▶ Start the Revision Manager Wizard by clicking the Revision Manager Assistant



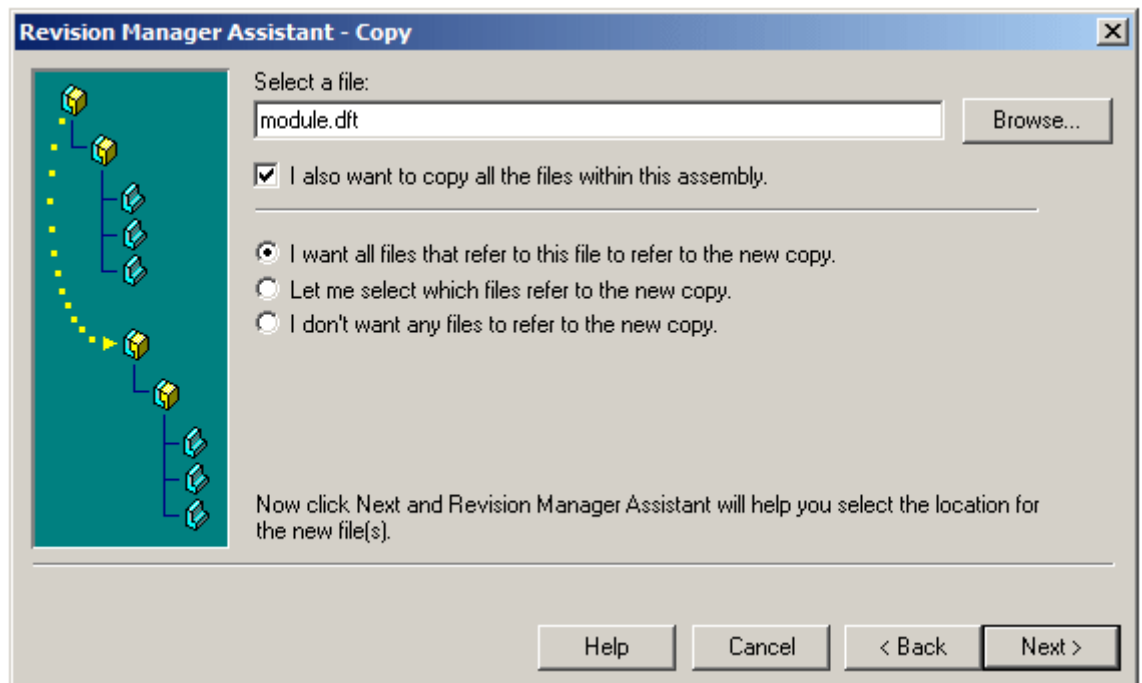
on the main menu.

- ▶ Select the action Copy a file, and then click Next.

### Note

The Revise a file option only refers to managed documents. For unmanaged documents, choose the action to Copy a file.

- ▶ Browse for *module.dft* in the folder where your activity files reside. Set the options as shown and click Next.



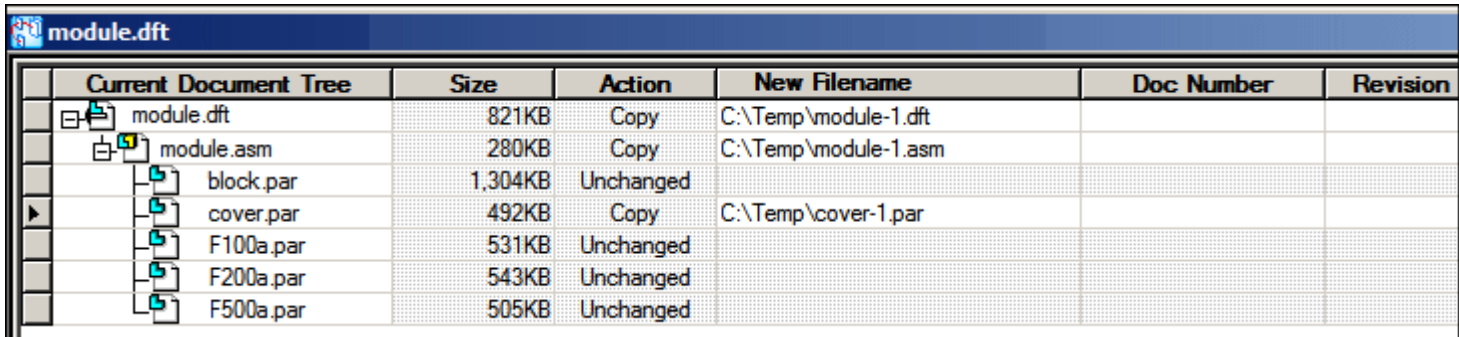
- ▶ For the new location, browse to C:\Temp and click Next.
- ▶ Click Next, and then click Finish in the Where Used dialog box.
- ▶ Click OK in the Revision Manager information box.

### Clear action

Clear files that do not need a revision. Clear action on parts *block.par*, *F100a.par*, *F200a.par* and *F500a.par*.

## Lesson 9 *Revising assemblies*

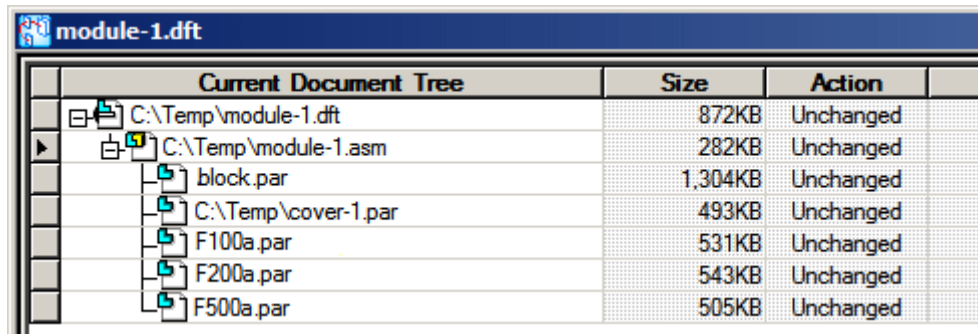
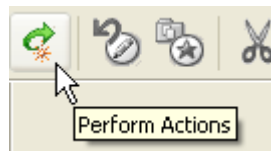
- ▶ Right-click the filename and click Clear Action.
- ▶ In the New Filename field, rename the files to the names shown.



The screenshot shows the Revision Manager dialog box for the assembly 'module.dft'. It displays a table with columns for 'Current Document Tree', 'Size', 'Action', 'New Filename', 'Doc Number', and 'Revision'. The table lists the following items:

Current Document Tree	Size	Action	New Filename	Doc Number	Revision
module.dft	821KB	Copy	C:\Temp\module-1.dft		
module.asm	280KB	Copy	C:\Temp\module-1.asm		
block.par	1,304KB	Unchanged			
cover.par	492KB	Copy	C:\Temp\cover-1.par		
F100a.par	531KB	Unchanged			
F200a.par	543KB	Unchanged			
F500a.par	505KB	Unchanged			

- ▶ Click Perform Actions to apply the revisions.



The screenshot shows the Revision Manager dialog box for the assembly 'module-1.dft'. It displays a table with columns for 'Current Document Tree', 'Size', and 'Action'. The table lists the following items:

Current Document Tree	Size	Action
C:\Temp\module-1.dft	872KB	Unchanged
C:\Temp\module-1.asm	282KB	Unchanged
block.par	1,304KB	Unchanged
C:\Temp\cover-1.par	493KB	Unchanged
F100a.par	531KB	Unchanged
F200a.par	543KB	Unchanged
F500a.par	505KB	Unchanged

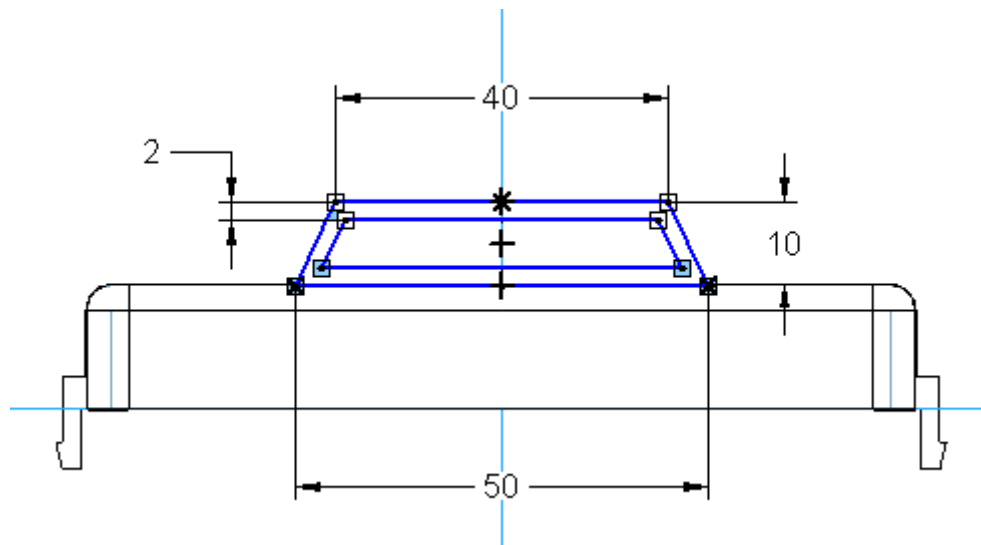
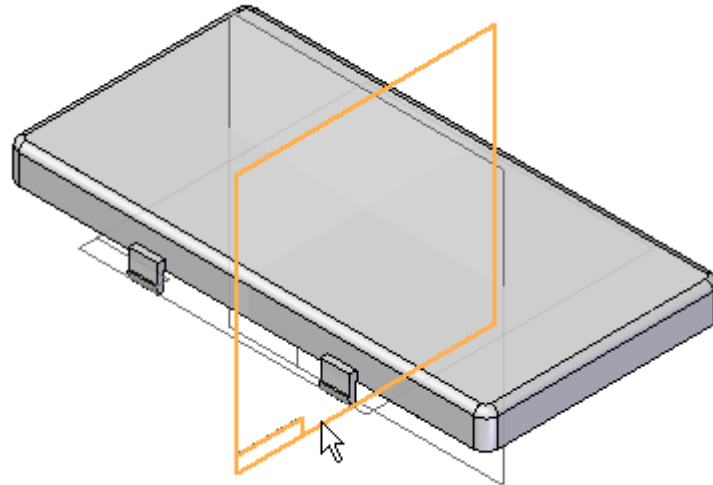
- ▶ Exit Revision Manager.

*Make a design change to the revision part*

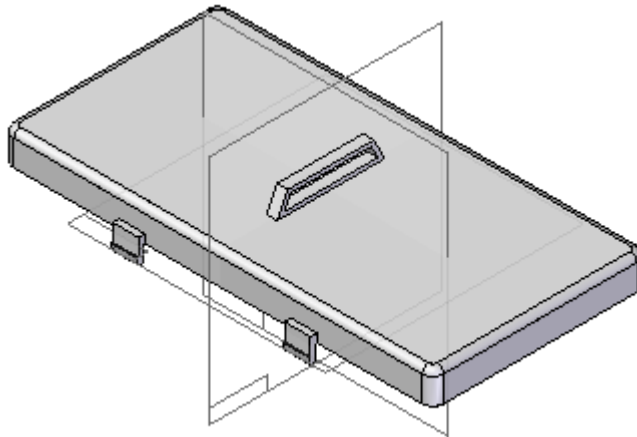
Make a design change to the revision part *cover-1.par*. Add a handle to the cover.

- ▶ Open *C:\Temp\cover-1.par*.

- ▶ Create a sketch on the Right (yz) reference plane and centered on the top of the cover.



- ▶ Create a protrusion from the sketch. Use a 5 mm symmetric extent.

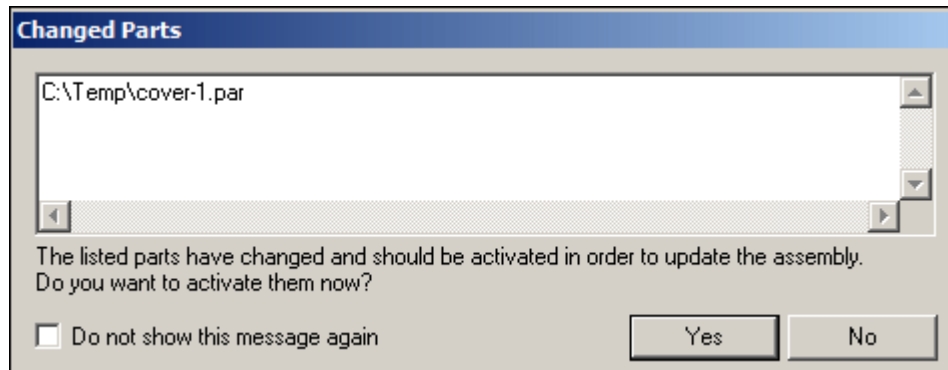


- ▶ Close and save the file.

#### *Update the assembly and drawing*

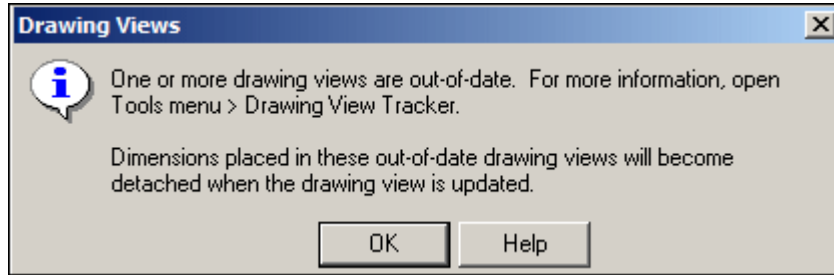
Update the assembly and draft file due to the design change.

- ▶ Open *C:\Temp\module-1.asm*.
- ▶ If you see the following dialog box, click Yes.

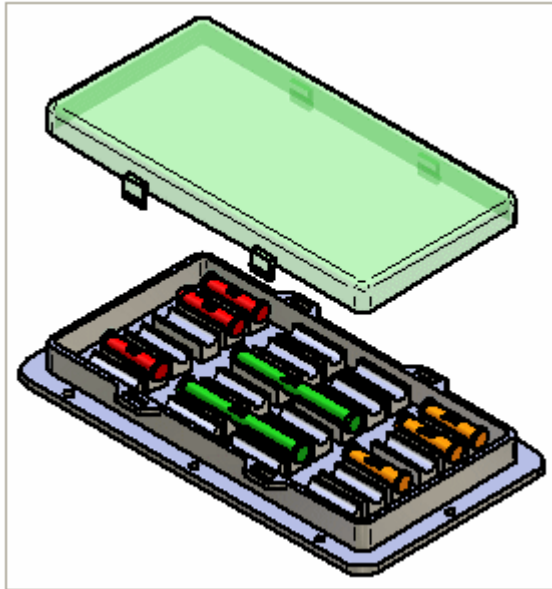


- ▶ Close and save the file.
- ▶ Open *C:\Temp\module-1.dft*.

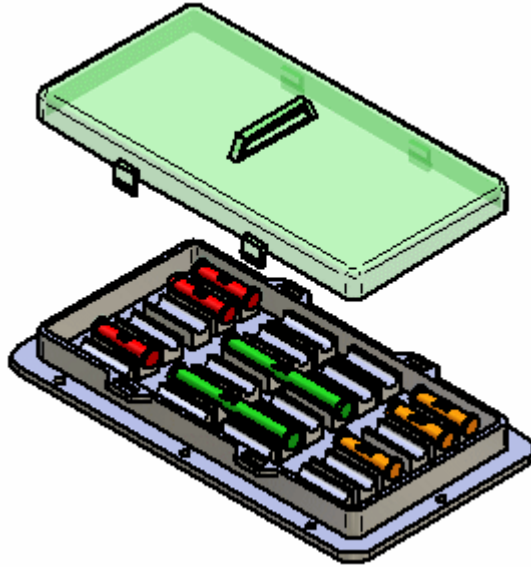
- ▶ Click OK in the Drawing Views information box.



The box around the drawing lets you know that the view is out-of-date.



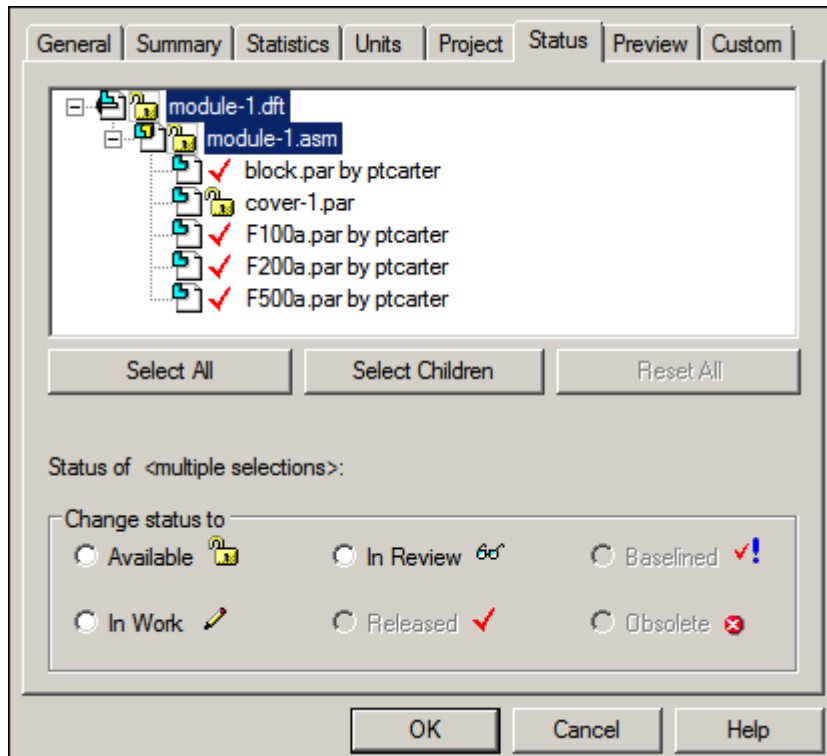
- ▶ Click the Update Views command 



- ▶ Save the draft file.

*Re-release the draft file*

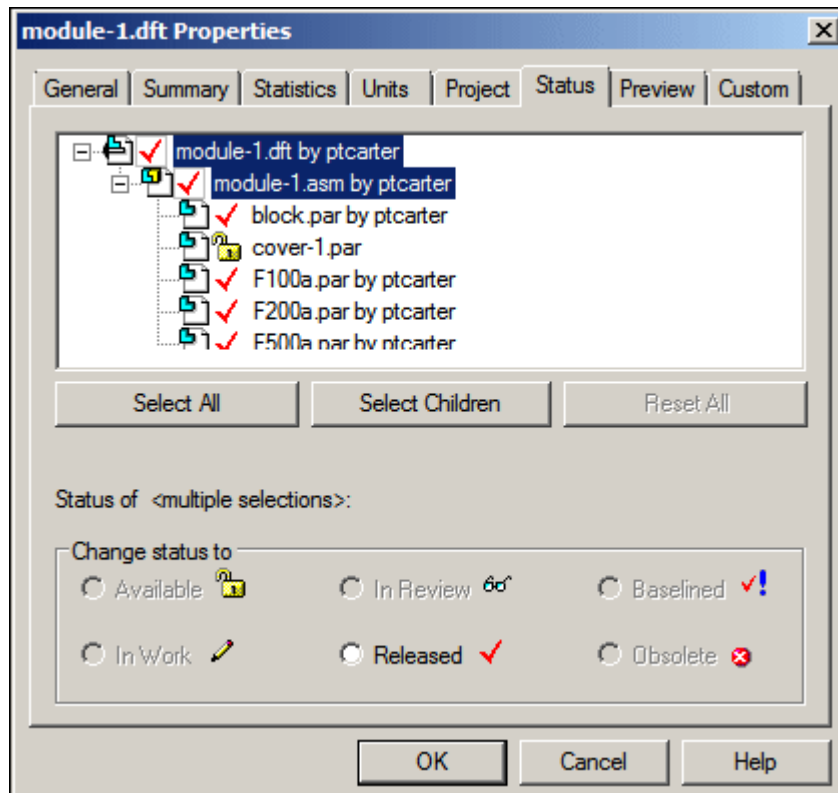
The revisions have been made. Re-release the draft file.



- ▶ Click the Application button, and then choose Properties® File Properties.



- ▶ Click *module-1.asm* and set the status to Released.
- ▶ Click *module-1.dft* and set the status to Released.



- ▶ As you did in the previous steps, open *cover-1.par* and set the status to Released.
- ▶ Exit and save the file. This completes the activity on the process for revising a draft of an assembly.

### Activity Summary

### Activity Summary

In this activity, you learned how to release a draft of an assembly. You revised the draft, assembly and a single part to a new location. You edited a part, updated the assembly and draft. When complete, you re-released the draft.

**Activity: Editing a part in context of the assembly****Overview**

In this activity you edit a part in the context of an assembly and make a new revision of a part in the assembly.

## Activity: Editing a part in context of the assembly

### Overview

In this activity you will edit a part in the context of an assembly. You create a new revision of a part in the assembly.

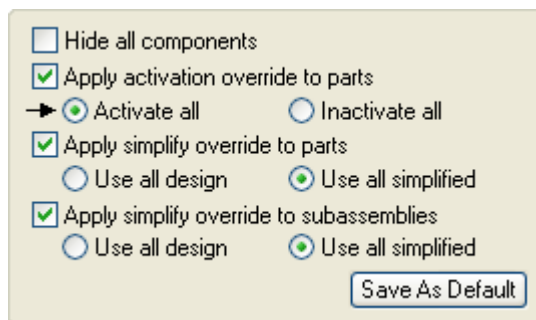
### Objectives

In this activity you will:

- Use the Edit command to modify a part in an assembly.
- Set document properties.

*Open the assembly with all parts active*

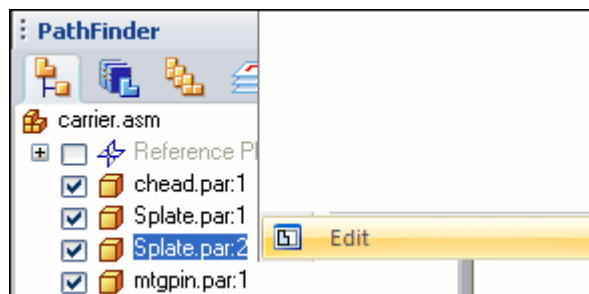
Open *carrier.asm*. In the Open File dialog box, select the Activate all option before you click Open.



*Edit a part in the assembly*


Begin the activity by editing a part in the assembly.

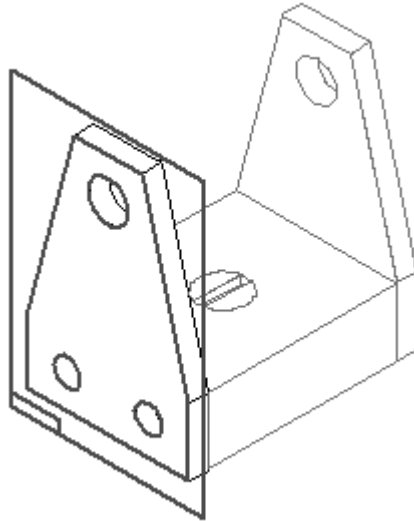
- On the Assembly PathFinder tab, right-click the left side plate *Splate.par:2*, and on the shortcut menu, click Edit. The part *Splate.par* is displayed in the Part environment. Use the Hide Previous Level command on the View tab to turn on and off the display of the other parts in the assembly as needed.




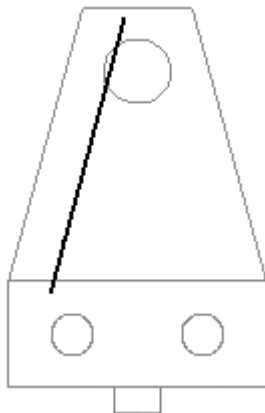
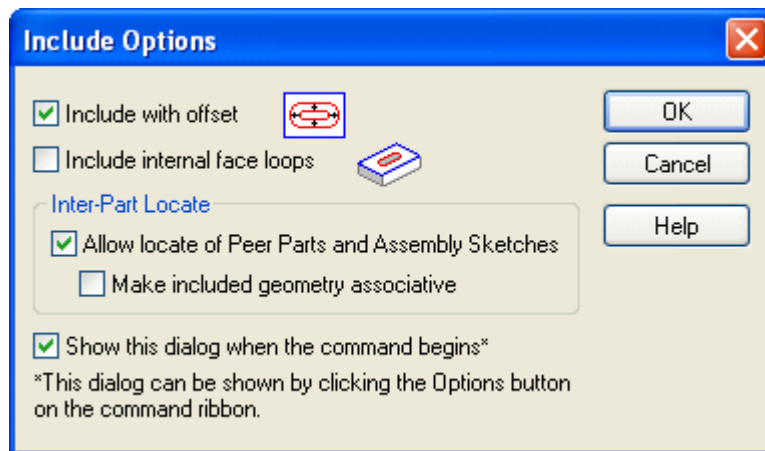
*Edit the part*

*Splate.par* is now open in the part environment. Edit the part.

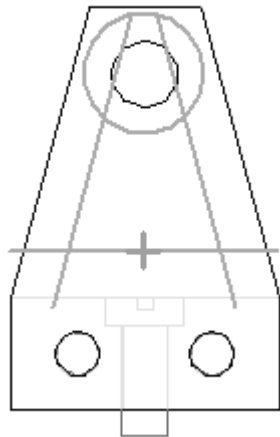
- ▶ Click the Cut command , and select the front face of the plate as shown.




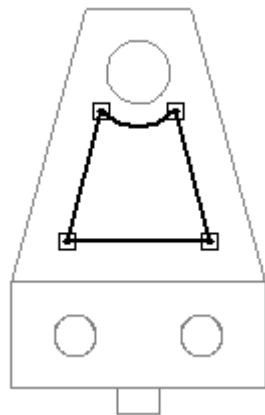
- ▶ Click the Include button . Set the Include Options as shown, and then click OK. Offset four edges from the side plate one edge at a time.




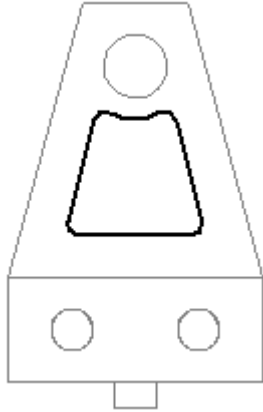
- ▶ Select the first edge to offset, and then on the command bar, click the Accept button.
- ▶ In the distance box, type 20 mm and press the Enter key. Click toward the inside of the part to specify the side to offset.
- ▶ Repeat the steps for the remaining two lines and the circle as shown. Offset all the lines a distance of 20 mm, and offset the circle a distance of 12 mm.




- ▶ Click the Trim command , and then trim the geometry to create the profile as shown.

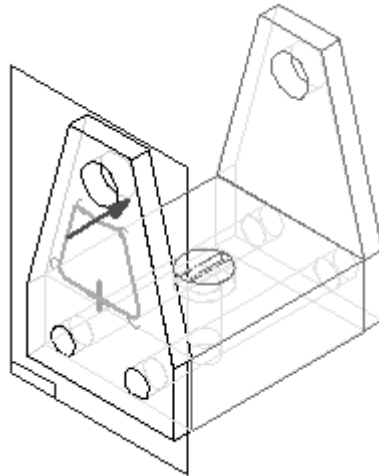


- ▶ Use the Fillet command  to add four 5 mm fillets, one at each corner, as shown.




- ▶ Click Close Sketch .

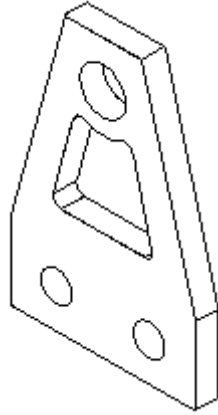
- ▶ For the extent of the cutout, click the Through Next button , and then click when the extent arrow points toward the part.



- ▶ Click Finish.

*Hide the other parts*

- ▶ On the View tab, click the Hide Previous Level command  to hide the other parts in the assembly.



*Save the file with a new name*

Save the file with a new name.

- ▶ Click the Application button. Click Save As@ Save Copy As.
- ▶ In the Save As dialog box, save this part as *splate02.par* in the folder that the activities are in. Overwrite any existing files, if necessary.

*Return to the assembly file*

Return to the assembly file.

- ▶ Click the Application button. Click Close. Do not save changes to the assembly.

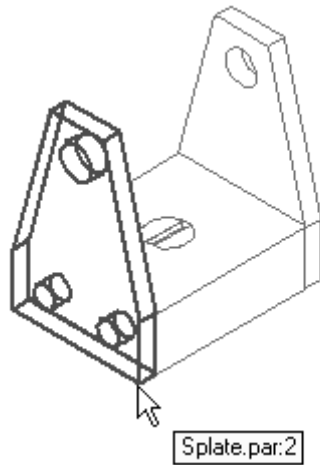
*Replace parts in the assembly*

Replace parts in the assembly with a new part created in the previous steps.

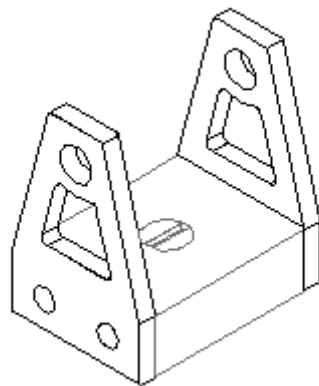
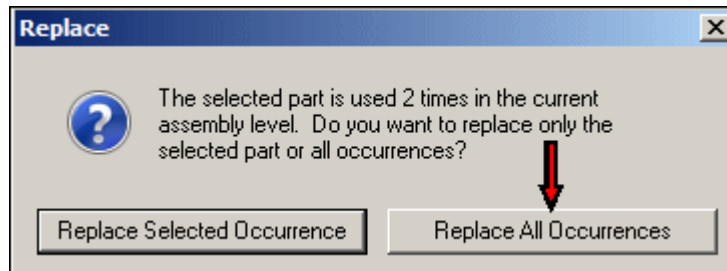
- ▶ Open *carrier.asm* located in the in the folder that the activities are in, with all the parts active.

- ▶ Click the Replace Part command .

- ▶ Select *Splate.par:2* as shown as the part to be replaced.



- ▶ Select *Splate02.par* in the Replacement Part dialog box and click Open.
- ▶ Since there are two *Splate.par* occurrences in the assembly, you are asked to replace a selected occurrence or all occurrences. Click Replace All Occurrences.



### Save the assembly

Save the assembly as *carrier2.asm* in the folder where your activities are located and then close the file. This completes this activity.



*Activity Summary*

**Activity Summary**

In this activity you edited a part in the context of an assembly and made a new revision of a part in the assembly.

**Lesson review**

Answer the following questions:

1. Describe the main difference between the Edit and the Open In Solid Edge Part commands that are used to modify parts while working in an assembly.
2. Where is Revision Manager located?
3. What purpose does the Revision Manager Assistant serve?
4. What happens when a document is opened with a status of In-Work?
5. How can properties be automatically loaded into newly created documents?

## Answers

1. Describe the main difference between the Edit and the Open In Solid Edge Part commands that are used to modify parts while working in an assembly.

The edit command opens the part or sheet metal document in the context of the assembly and allows geometric links to other components in the assembly. Open in Solid Edge part opens the part document stand alone without any access to the assembly where the part resides.

2. Where is Revision Manager located?

Revision Manager can be found in the View and Markup application in the Solid Edge Start menu.

3. What purpose does the revision manager assistant serve?

The Revision manager assistant populates the parameters needed to revise an assembly by interactively asking a series of questions about the intent of the changes desired for the assembly.

4. What happens when a document is opened with a status of In-Work?

The document is opened as Read Only. It can be modified after the document status is returned to available.

5. How can properties be automatically loaded into newly created documents?

Propseed.txt is an ASCII text file that is used to populate values in the Properties dialog box. You can use a text editor, such as Notepad, to add values to the file.

By default, the file is located in the Solid Edge Program folder. For example, if you loaded Solid Edge to drive C, the path would be C:\Program Files\Solid Edge ST4\Program\propseed.txt.

You can instruct Solid Edge to look for *propseed.txt* in a different folder, including a folder on another machine in the network. To do this, on the Application menu, click Solid Edge Options. On the File Locations tab of the Solid Edge Options dialog box, select Property Seed File and click Modify. In the Browse dialog box, specify the drive and folder containing the *propseed.txt* file. After specifying the location, click Update.

## **Lesson summary**

In the Assembly, you can use the Edit command to in-place activate a part to make design modifications to the part while viewing the remainder of the assembly. A part does not have to be activated in the assembly to edit it. When you select the Edit command, the part is automatically activated.

Solid Edge supports assembly management, including the process of revising assemblies. Revision Manager lets you copy an assembly, replace parts in it, and create new revisions of an assembly and its parts.

The Application button® Manage® Edit Links command is used to manage linked documents such as an assembly. All Solid Edge parts placed in an assembly are linked objects.

Document properties store information about a Solid Edge document. All OLE compatible applications, such as Microsoft Word and Excel, use document properties to manage document information. To access document properties, use the Properties command.

The Search button located in the Open File dialog box is used to start an advanced search for files. The file name, file type, and date last modified boxes allow you to search for documents using known information.

The document management tools within Solid Edge let you generate reports on files without actually opening those files in Solid Edge.

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**Lesson**

# *10 Assembly reports*

## **Assembly reports**

## Assembly reports

### Types of assembly reports

Solid Edge creates reports that provide information about the components that make up an assembly. You can create:

- **Assembly reports**—For assembly and structural frame models, assembly reports list the parts and subassemblies in the model. You can generate these types of reports:
  - **Bill of Materials**—This report supports item number levels that match the assembly structure.
  - **Exploded Bill of Materials**
  - **Summary of Atomic Reports**
  - **Parts List**—This report is a flat list that shows the quantity of each element.
- **Pipe reports**—For pipe and tubing models, lists the pipes and fittings in an assembly.
- **Harness reports**—For wire harness models, lists the harness components and connections in an assembly. You can create the following types of wire reports:
  - **Top Level**
  - **Expanded**

You can create assembly reports for managed documents and unmanaged documents.

- For unmanaged documents, the report information is collected from the local file properties of the referenced part or assembly documents.
- For managed documents, the property information is collected from the list of properties defined in the SharePoint managed document profile, not the local file properties.

### Producing assembly bill of material reports

To produce an assembly bill of materials, you can run the Reports command from the Solid Edge Assembly environment, from [Revision Manager](#), or from Windows Explorer. In Explorer, the Reports command is available on the shortcut menu of an assembly document (.asm).

#### Note

The Reports command is not available in Windows Explorer for assemblies that are in a managed library. To create a report for a managed assembly, you must open the assembly in Solid Edge or Revision Manager. Also, the Reports command is not available if you have standalone Insight Connect loaded. To create a report in standalone Insight Connect, you must open the assembly in View and Markup or Revision Manager.

### Producing pipe reports

For pipe parts, you can use the Pipe Report command in XpresRoute to create a report for the pipes and fittings in an assembly. This displays the Piping Report dialog box for you to specify report type and content:

- Report all the pipes in the assembly.
- Report only the currently selected pipes.
- Include pipe fittings.
- Create a total length or cut length report.

The Pipe Report command is not available if there are no pipes in the file.

### Producing harness reports

For wire harness parts, you can run the Harness Reports command from Harness Design to create a report for the components and connections in an assembly.

When you run the Harness Reports command in Harness Design, you can specify which harness elements to include in the report:

- All the harness components (or connections) in the assembly
- Only the currently selected components (or connections)
- Only the currently displayed components (or connections)

#### Note

- To create a report on selected parts, select the parts before selecting the Reports command.
- You can use the Show and Hide commands to control which harness elements are displayed.

### Weldment assembly reports

You can use the Reports dialog box to specify whether a weldment assembly is treated as a single component, similar to a part; or as an assembly, where all its component parts are included in the report.

When you set the Expand Weldment Assemblies option, the component parts in the assembly are included in the report. When you clear this option, the weldment assembly is treated as a single part, and the component parts are excluded from the report.

### Alternate assembly reports

If the assembly has been converted to an alternate assembly, in Solid Edge the report is created for the active member. In Windows Explorer and [Revision Manager](#) you can use the FOA Member Names dialog box to specify which member you want to create a report for.



## **Virtual components**

If the assembly for which you are creating a report contains virtual components, use the Reports command on the Tools menu in the Assembly environment to generate the report. Teamcenter-managed virtual components are included in the report.

### **Caution**

If you run the Reports command from Windows Explorer on an assembly that has virtual components, the virtual components will not be contained in the report. In a case where the assembly contains only virtual components, a message may be displayed that states that no parts are in the file.

## **Showing assembly item numbers in reports**

You can show assembly item numbers in assembly and structural frame reports, and in pipe reports. Assembly item numbers are saved with the assembly and are available for downstream use, for example, in balloons placed on a model and in ballooned parts lists placed on an assembly drawing. This ensures consistent part numbering is propagated across all applications of the model.

- To create item numbers in the assembly, select the Maintain item numbers check box in the Item Numbers page (Solid Edge Options dialog box).
- To use these item numbers in assembly reports, select the Use assembly generated item numbers check box, which is available in the Report dialog box.

Alternatively, you can leave this option unchecked and have the assembly Report command generate item numbers on the fly.

## **Formatting a report**

You can format the report using the Format option on the Reports dialog box. Using the Format Report dialog box, you can set the following:

- Font
- Column headings (based on available properties)
- Sorting method

Each property you include in the report will be a new report column. Standard properties you can choose from include Quantity, Document Number, Revision, and Author. You can also include custom file properties that you have recorded in the part and subassembly documents.

After you have formatted the report, you can preview the report output by selecting the OK button on the Format Report dialog box.

## Report Column Source

Many of the values shown in a report are stored in File Properties for individual parts and subassemblies that make up the assembly. Other values, such as mass, are calculated or added by choosing a material from the material table.

You can control some of the report values from the File Properties dialog box. The following shows the tab in the File Properties dialog box where these column headings reside:

- Document Number (Project Tab)
- Revision Number (Project Tab)
- Title (Summary Tab)
- Author (Summary Tab)
- Subject (Summary Tab)
- Keywords (Summary Tab)
- Comments (Summary Tab)
- Category (Summary Tab)
- Manager (Summary Tab)
- Company (Summary Tab)
- Project Name (Project Tab)
- Custom (Custom Tab)

More information on the File Properties dialog box is found in the Property Text source list help topic.

## Outputting the report

After previewing the report, you have the following options:

- Save the current report.
- Print the current report.
- Copy the current report to the Clipboard.
- Create another report.

## Saving a report

The Save As option allows you to define a storage location, document name, and output type. You can output the report as a text file (*.txt*) or rich text file (*.rtf*).

## **Adding properties to managed documents**

If you add new custom properties to managed Solid Edge documents, you must also add these properties to the managed document profile on the SharePoint server to make them available in an assembly report. Managed document profiles are stored in the Document Profiles folder on the SharePoint server. To properly synchronize the Solid Edge file properties and the SharePoint managed document profile properties, the property names must match exactly.

### Item numbers in assemblies

Item numbers for parts and subassemblies can be created and maintained automatically in an assembly model. Item numbers are based on the assembly structure, that is, the order in which parts, subassemblies, and assemblies are displayed in Assembly PathFinder.

Once created, you can use assembly item numbers in assembly reports, including pipe and tubing reports produced in XpresRoute, and structural frame design reports. The item numbers are saved with the assembly and are available for downstream use, for example, in PMI balloons placed on the model and in ballooned parts lists on an assembly drawing. This ensures consistent part numbering is propagated across all applications of the model.

### Creating item numbers in assemblies

You can create item numbers in an assembly model by selecting the Maintain item numbers check box in the Item Numbers page (Solid Edge Options dialog box). As new items are added to the assembly, new item numbers are created. If items are removed from the assembly, their item numbers are not available for reuse.

You can remove item numbers from the assembly by deselecting the Maintain item numbers check box.

Item number creation can be enabled and disabled at any time.

- Item numbers can be generated as the assembly is built by enabling item number creation before items are added to the assembly.
- In some cases, it may be better to enable item numbers after most of the assembly structure has been created.

### Editing item numbers

You can edit item numbers in the model by selecting the top level assembly occurrence in PathFinder and choosing the Occurrence Properties command. The Item Number column in the Occurrences Properties dialog box shows the assembly item numbers. You can:

- Change existing item numbers.
- Add missing item numbers using the Next Available Number command on the shortcut menu.
- Use the Reset Item Numbers command on the shortcut menu to cancel your edits and restore the item numbers generated by the assembly structure.

### Using assembly item numbers in reports


You can use the assembly-generated item numbers in assembly reports when you select the Use assembly generated items number check box in the assembly Report dialog box. The item numbers are shown when you select the Format button to open the Format Report dialog box or in the Report Output dialog box.


See the help topic, Create a report.

## Using assembly item numbers in PMI balloons

You can display assembly item numbers in PMI balloons placed on the model.

When you select the PMI tab® Annotation group® Balloon command, you can use these options on the Balloon command bar to retrieve the item numbers from the assembly and display them in the balloon:

- Item Number button 

- Item Count button 

## Using assembly item numbers on a drawing

On an assembly drawing, you can:

- Review the assembly generated item numbers on the Item Number page (Parts List Properties dialog box). The numbers shown in the first column correspond to the assembly structure.
- Show the assembly generated item numbers in a ballooned parts list by selecting the Use assembly generated items number check box on the Options page (Parts List Properties dialog box).

## Item numbering schema

Use the Item Numbers page (Solid Edge Options dialog box) to select an item numbering schema for the parts and subassemblies in the assembly model. Refer to the following table for a comparison of item numbering options.

### Example

The gray shaded cells below indicate the assembly structure, where A1 and A2 are subassemblies, and P1, P2, P3, P4, and P5 are parts.

The unshaded cells show how flat list item numbering is applied to the assembly structure.

The last column shows the level based item number option, which is available only for Exploded - Top down item numbers.

	Level 1	Level 2	Level 3	Top level only	Atomic - Top down	Exploded - Top down	Use level based numbers
<b>Top assembly</b>	A1			1		1	1
		P1			1	2	1.1
		P2			2	3	1.2
		A2				4	1.3
			P3		3	5	1.3.1
			P4		4	6	1.3.2
		P6			5	7	1.4
		P7			6	8	1.5
	P2			2	2	3	2
	P4			3	4	6	3
	A2			4		4	4
		P3			3	5	4.1
		P4			4	6	4.2
	P5			5	7	9	5

## Formatting item numbers

You can format the item numbers shown in the assembly model by selecting the following options on the Item Numbers page (Solid Edge Options dialog box):

- Use level based numbers
- Start number
- Increment by

## Synchronizing Item Numbers with Teamcenter Find Numbers

In the Teamcenter-managed environment, the integration of Solid Edge Item Numbers with Teamcenter Find Numbers is governed by the Teamcenter preference *SEEC\_Synchronize\_ItemNumbers*. This site preference, delivered by SEEC Administrator, provides a single point for enabling and configuring Item Number options. When you enable *SEEC\_Synchronize\_ItemNumbers* the Solid Edge Options, Item Numbers page is available for viewing only. Definitions are based on the values specified by the preference.

By enabling *SEEC\_Synchronize\_ItemNumbers*, you can construct a view in Structure Manager that is consistent with a draft's parts list. Synchronization is bi-directional. Existing structures that contain non-modeled items (virtual components) and associated occurrences cannot be reset. The item numbers are locked. However, when an Item Number is assigned either through reset or when placing a new occurrence, the numbers assigned do not conflict with numbers locked by virtual components.

There are a few important governing rules to be aware of:

- Item Numbers are supported at the top level only.
- Any change to the Item Number options results in a reset.
- Item Numbers assigned to non-modeled Item Revisions are considered locked or fixed. These BOM lines are not included in Solid Edge Save.
- Solid Edge will not assign an Item Number that is currently assigned to non-modeled Item Revisions.
- The reset command will not impact an Item Number that is currently assigned to a non-modeled revision.
- Item Numbers must be unique by document reference. If they are not unique, Solid Edge keeps the lowest Item Number and assigns it to all occurrences of that reference.

The *Teamcenter Preferences* section of the *Solid Edge Administrator's Guide* provides details regarding preference options.

## Exploded parts lists

You can use the Parts List command and options in the Parts List Properties dialog box to define and place an exploded parts list on an assembly drawing.

Use the General page (or the Parts List command bar) to:

- Select the default Solid Edge Exploded parts list style from the Saved Settings list.
- Define a custom exploded parts list style.

Use the Options page to:

- Show the item numbers that were created in the assembly.

Using item numbers from the model ensures the parts list item numbers do not change unless the model does. Otherwise, item numbers are generated on-the-fly by the Parts List command.

### Note

You can use the assembly model item numbers when the Maintain item numbers check box is selected in the assembly document on the Item Numbers page, Solid Edge Options dialog box. To learn more, see Help topic [Item numbers in assemblies](#).

Use the List Control page to:

- Display subassemblies and subassembly parts in an exploded parts list.
- Choose the item numbering format:
  - Level based item numbers, which indicate the hierarchy of an exploded parts list.
  - Flat list item numbers.
- Show the top level assembly in a row by itself.

Use the Columns page to:

- Select and format the Item Number data column.
- Select additional data columns—Mass (Item), Mass (Quantity), Miter Cut 1, and Miter Cut 2—when generating parts lists for assembly models containing frames, pipes, or tubes.
- Indent the item numbers or the content of any column.

Use the Sorting page to:

- Sort the item numbers in the same order that they are shown in Assembly PathFinder.

### Note

Unless the assembly has been saved in V17 or later, the parts list order may not match the Assembly PathFinder order.



## **Activity: Assembly reports**

### **Overview**

The objective of this activity is to demonstrate some of the options when generating an assembly report.

### **Activity**

In this activity you will use different options to create assembly reports.

**Activity: Generating a report in an assembly**

**Overview**

This activity demonstrates how to create and format a report in assembly.

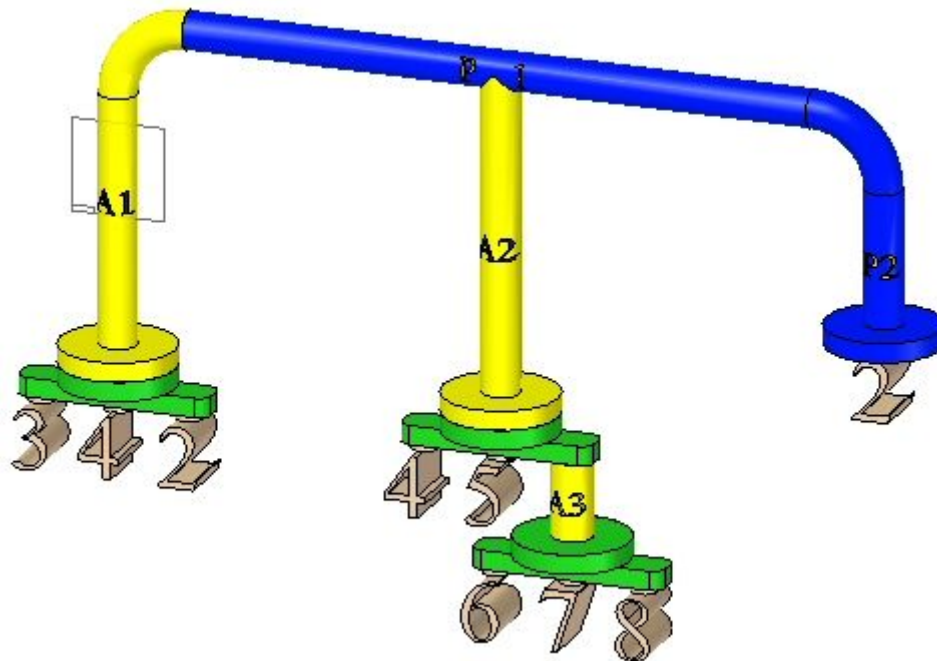
**Objectives**

You will create different reports based on different options.

*Open an existing assembly with all parts active*

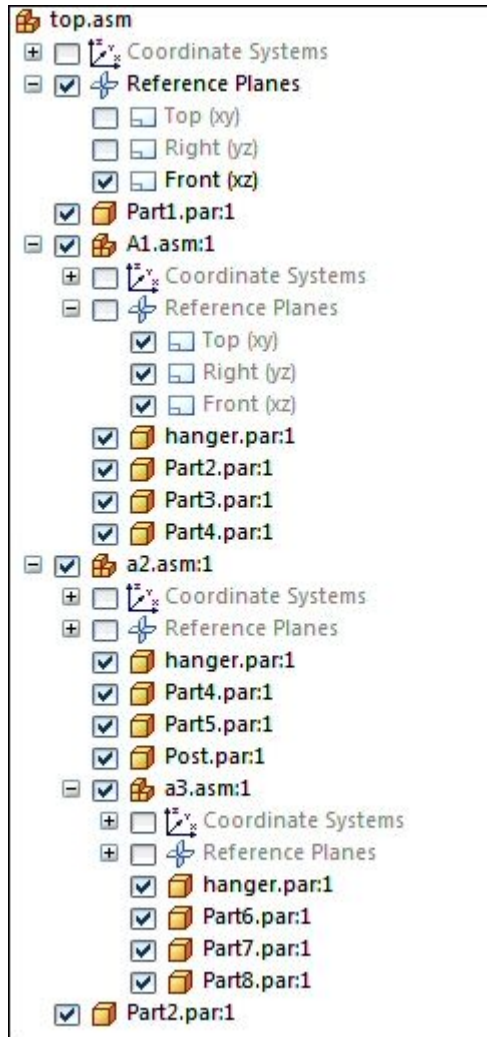
The assembly you open has assemblies and subassemblies with parts at all levels.


- From the Solid Edge start screen, click Open Existing Document. Browse for *top.asm* in the folder where the activity files are located.



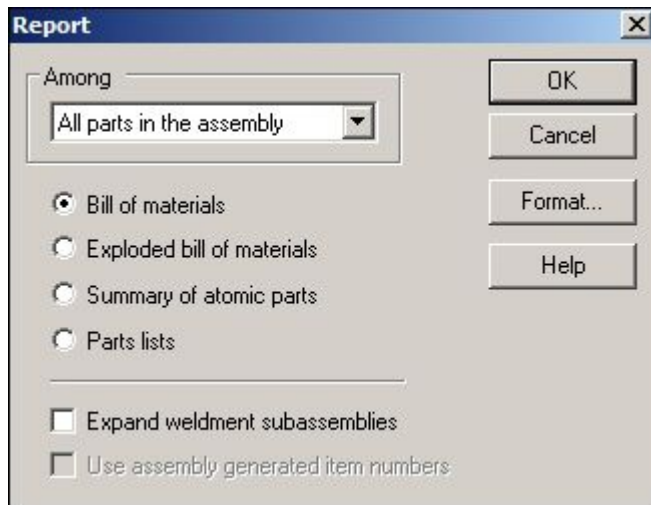
Generate a bill of materials report

- ▶ Expand PathFinder to show all the parts and subassemblies.

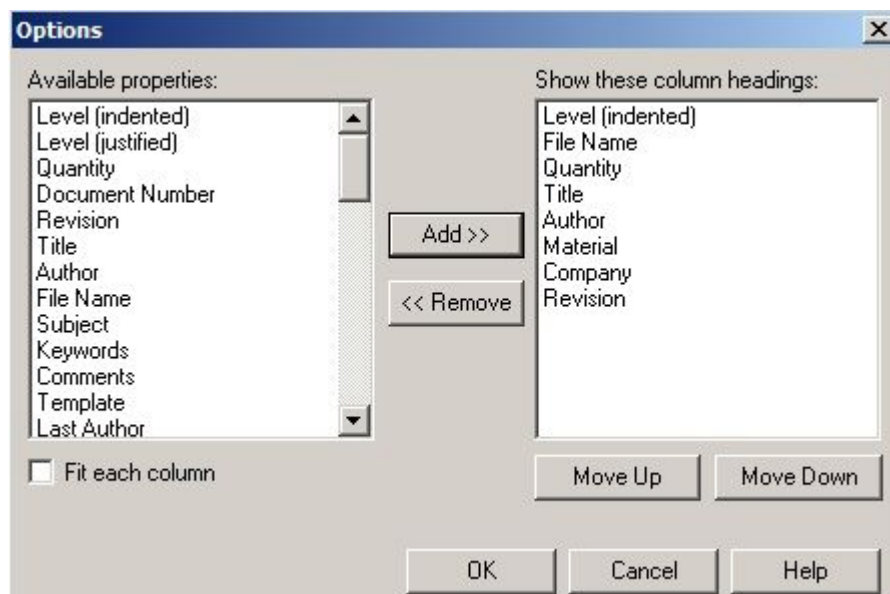


- ▶ Choose the Tools tab® Assistants group® Reports command .

- ▶ Select the options shown, and then click Format.



- ▶ In the Bill of Materials Format dialog box, click Options.
- ▶ Use the Add and Remove buttons to set the options as shown and click OK.

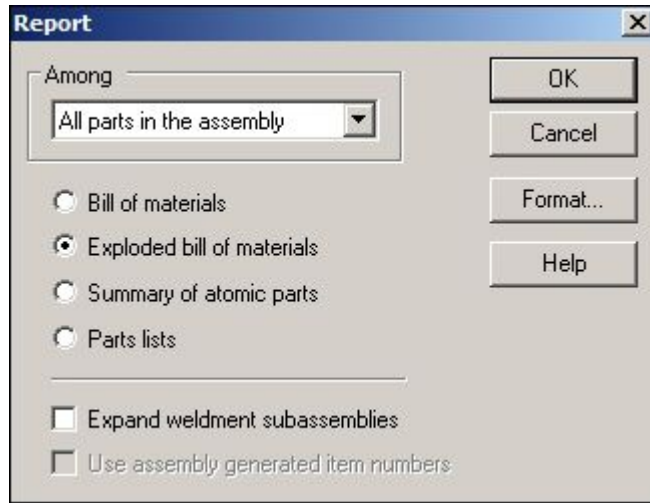


- ▶ The report is generated. Observe the results. Save the report as rich text format as *bom.rtf* in the folder where the activity files are located.

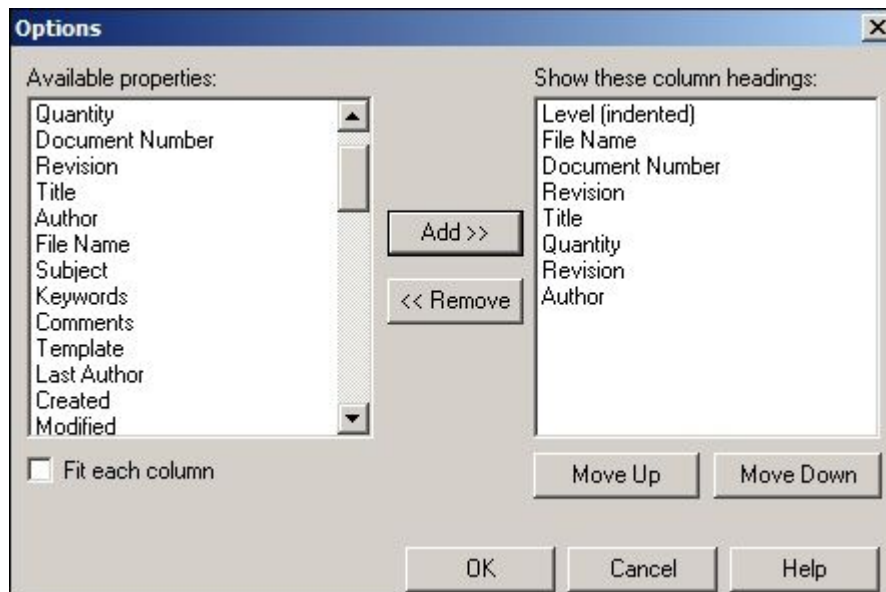
*Generate an exploded bill of materials report*

- ▶ Click New Report to generate an exploded bill of materials report.

- ▶ Select the options shown, and then click Format.



- ▶ In the Bill of Materials Format dialog box, click Options.
- ▶ Use the Add and Remove buttons set the options as shown and click OK.

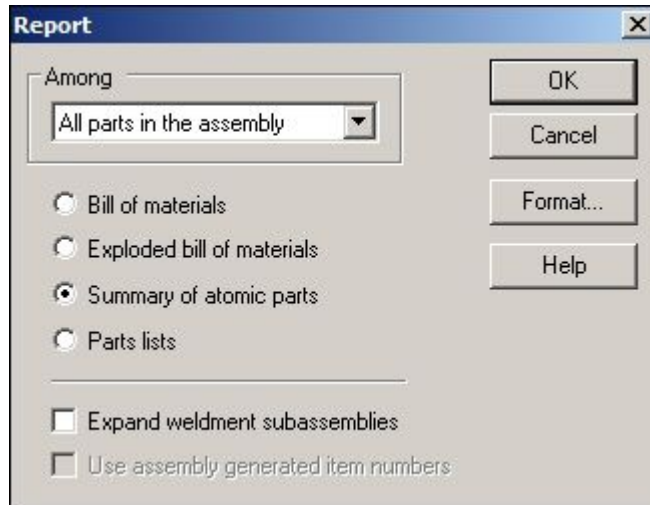


- ▶ The report is generated. Observe the results. Save the report as rich text format as *exploded\_bom.rtf* in the folder where the activity files are located.

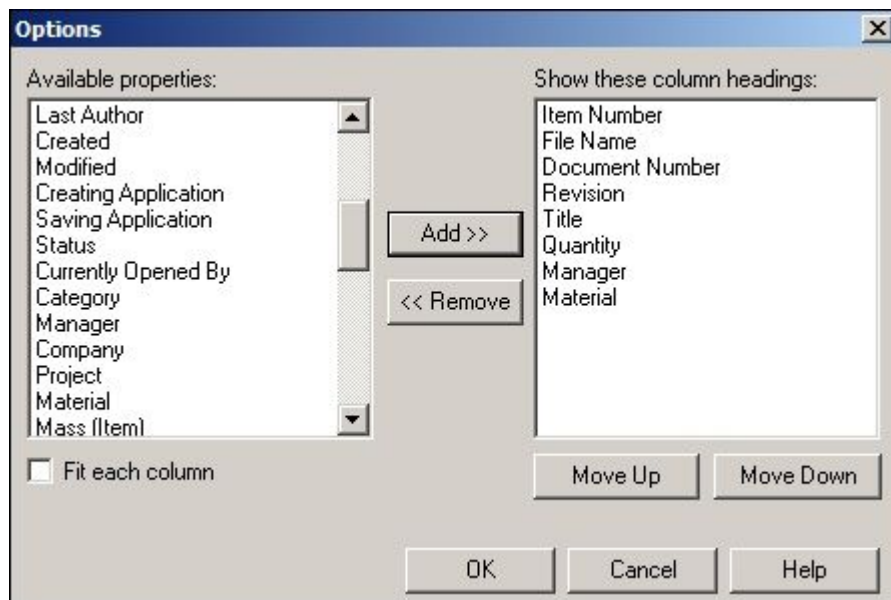
*Generate a summary of atomic parts report*

- ▶ Click New Report to generate a summary of atomic parts report.

- ▶ Select the options shown, and then click Format.



- ▶ In the Bill of Materials Format dialog box, click Options.
- ▶ Use the Add and Remove buttons to set the options as shown and click OK.

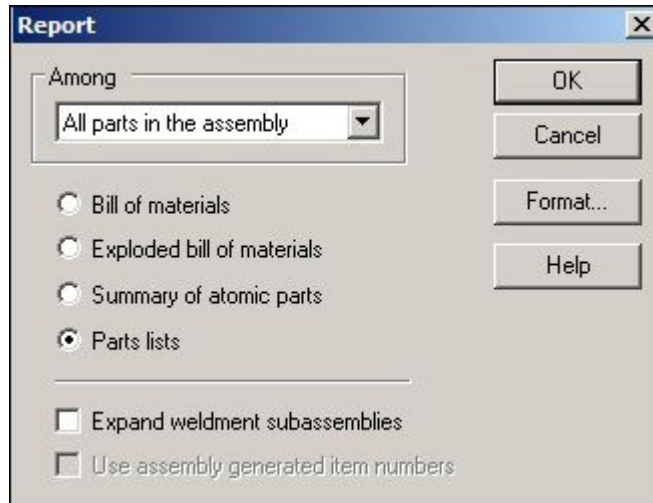


- ▶ The report is generated. Observe the results. Save the report as rich text format as *atomic.rtf* in the folder where the activity files are located.

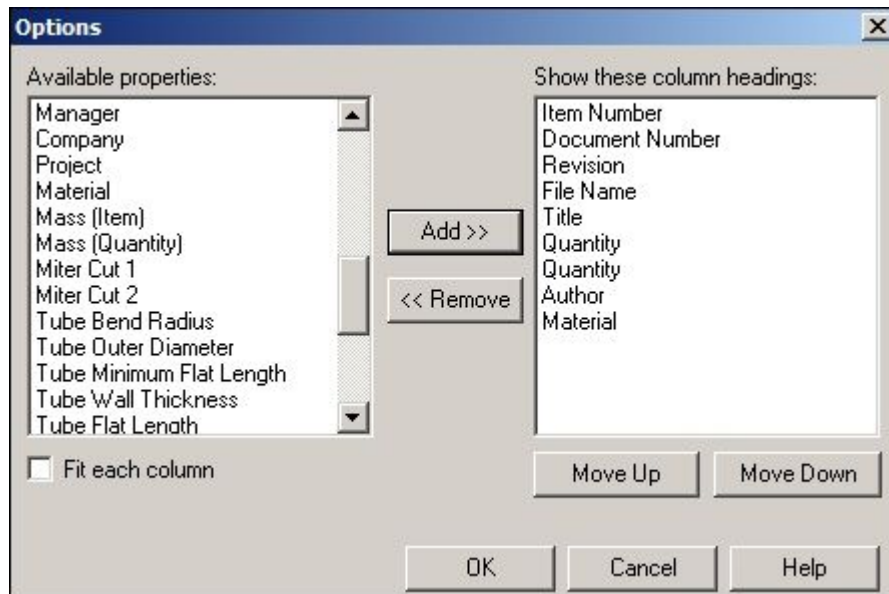
### Generate a parts list report

- ▶ Click New Report to generate a parts list report.

- ▶ Select the options shown, and then click Format.



- ▶ In the Bill of Materials Format dialog box, click Options.
- ▶ Use the Add and Remove buttons to set the options as shown and click OK.



- ▶ The report is generated. Observe the results. Save the report as rich text format as *parts\_list.rtf* in the folder where the activity files are located.
- ▶ Click Close

### Assigning top level item numbers in an assembly

Item numbers can be assigned at the assembly level and used downstream in reports and on a drawing sheet.



- ▶ Click the Application button.
- ▶ Click Solid Edge Options, and then click the item numbers tab.
- ▶ Set the options as shown.



**Note**


Item numbers are determined by the Assembly PathFinder order. Item numbers can be controlled by editing the occurrence properties in PathFinder.


- ▶ Click Apply, and then click OK to dismiss the Solid Edge Options dialog box.
- ▶ To view the occurrence properties for the top level of the assembly, right-click *top.asm* in PathFinder, and then choose Occurrence Properties.

Placement Name	Item Num...
[-] top.asm	
[-] Part1.par:1	1
[-] A1.asm:1	2
[-] hanger.par:1	
[-] Part2.par:1	4
[-] Part4.par:1	
[-] Part3.par:1	
[-] a2.asm:1	3
[-] hanger.par:1	
[-] Part4.par:1	
[-] Post.par:1	
[-] Part5.par:1	
[-] a3.asm:1	
[-] hanger.par:1	
[-] Part6.par:1	
[-] Part7.par:1	
[-] Part8.par:1	
[-] Part2.par:1	4

- ▶ Click OK.



- ▶ Choose the PMI tab® Annotation group® Balloon command 

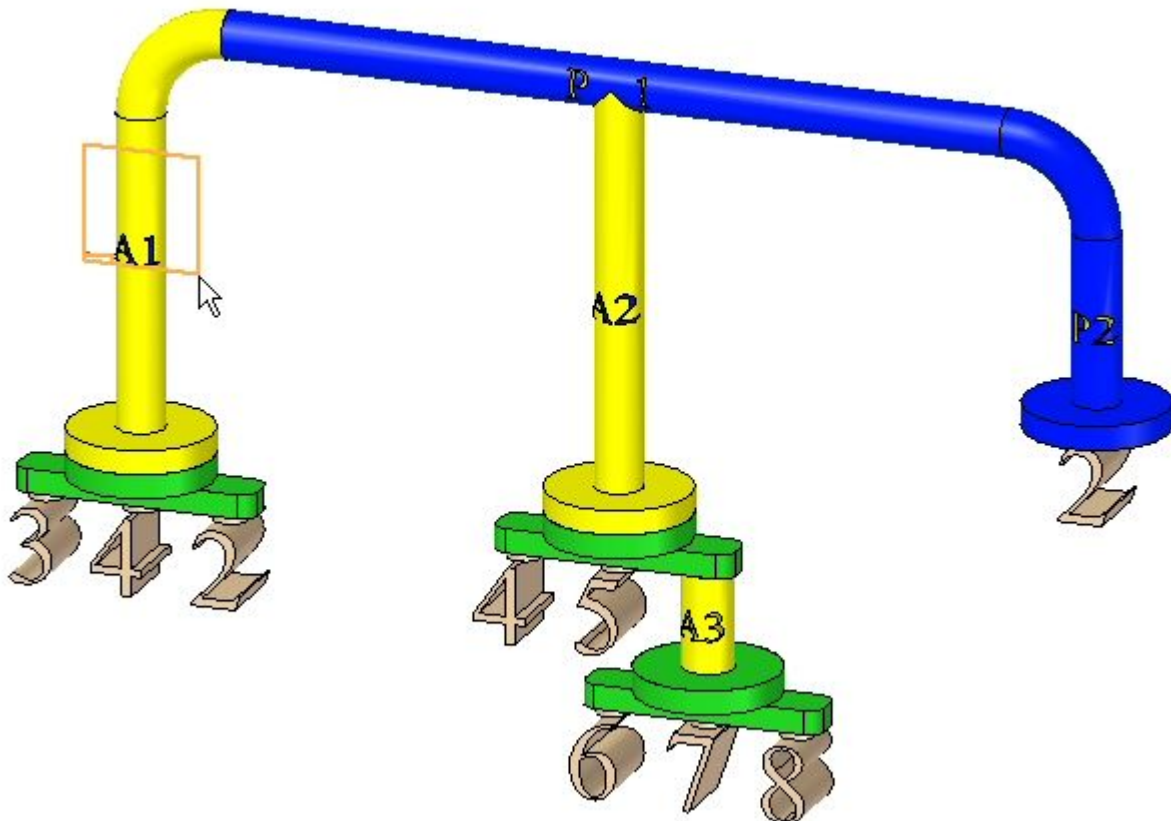
- ▶ Set the balloon shape to Circle 

- ▶ Ensure that the Item Number and Item Count buttons are selected as shown.



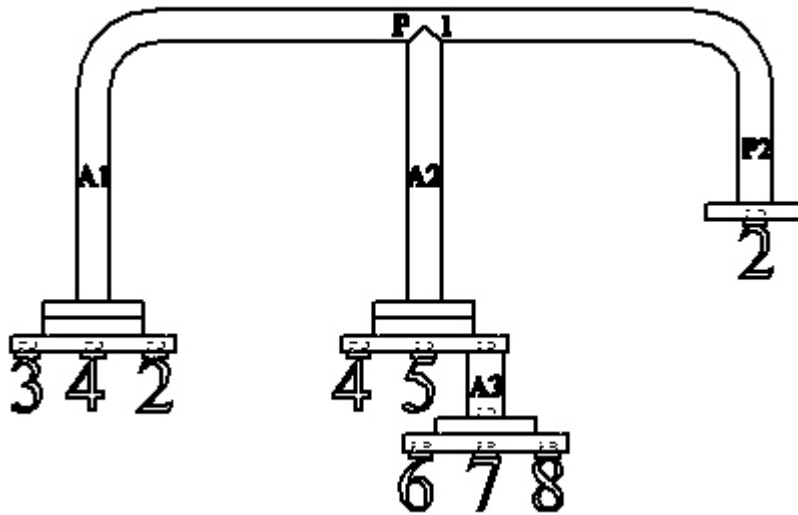
- ▶ Click Lock Dimension Plane 


- ▶ Select the Front reference plane as shown.

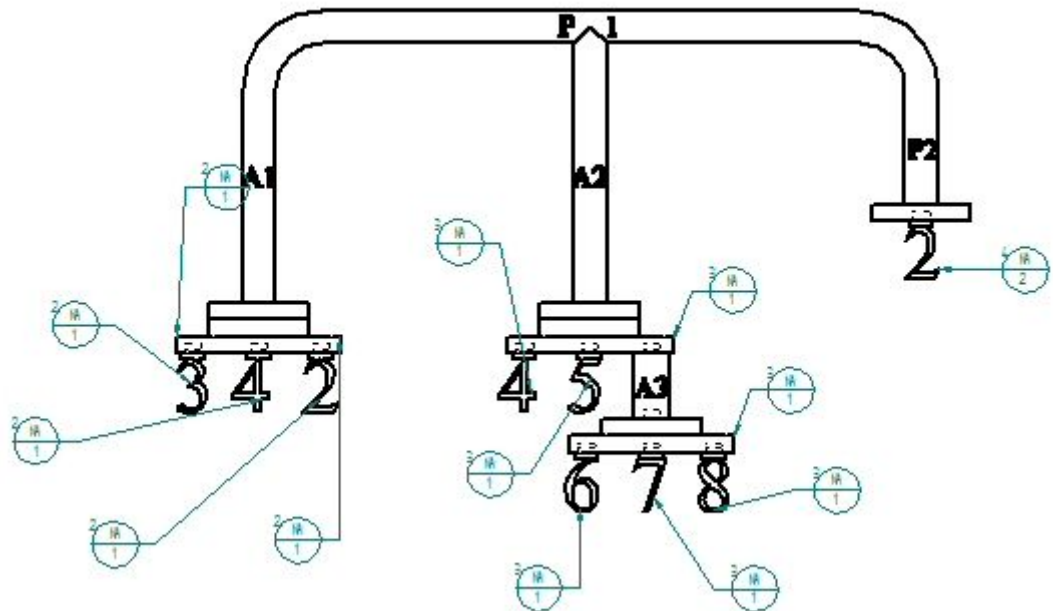




- Place the view on the drawing sheet as shown.



- Choose Home tab@ Dimension group@ Retrieve Dimensions command .
- Select the view. The balloons created in the assembly are placed on the drawing sheet with the corresponding item numbers.



- Save the draft document as *top\_level.dft*.
- Close the draft file. The assembly document is displayed..

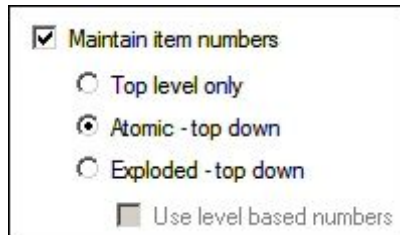
- ▶ Close the assembly.

*Assigning atomic top down item numbers in an assembly*


- ▶ Open the assembly *top\_atomic.asm* with all the parts active.




- ▶ Click the Application button.
- ▶ Click Solid Edge Options, and then click the Item Numbers tab.
- ▶ Set the options as shown.



- ▶ Click Apply, and then click OK to dismiss the Solid Edge Options dialog box.

- ▶ Choose the PMI tab@ Annotation group@ Balloon command .

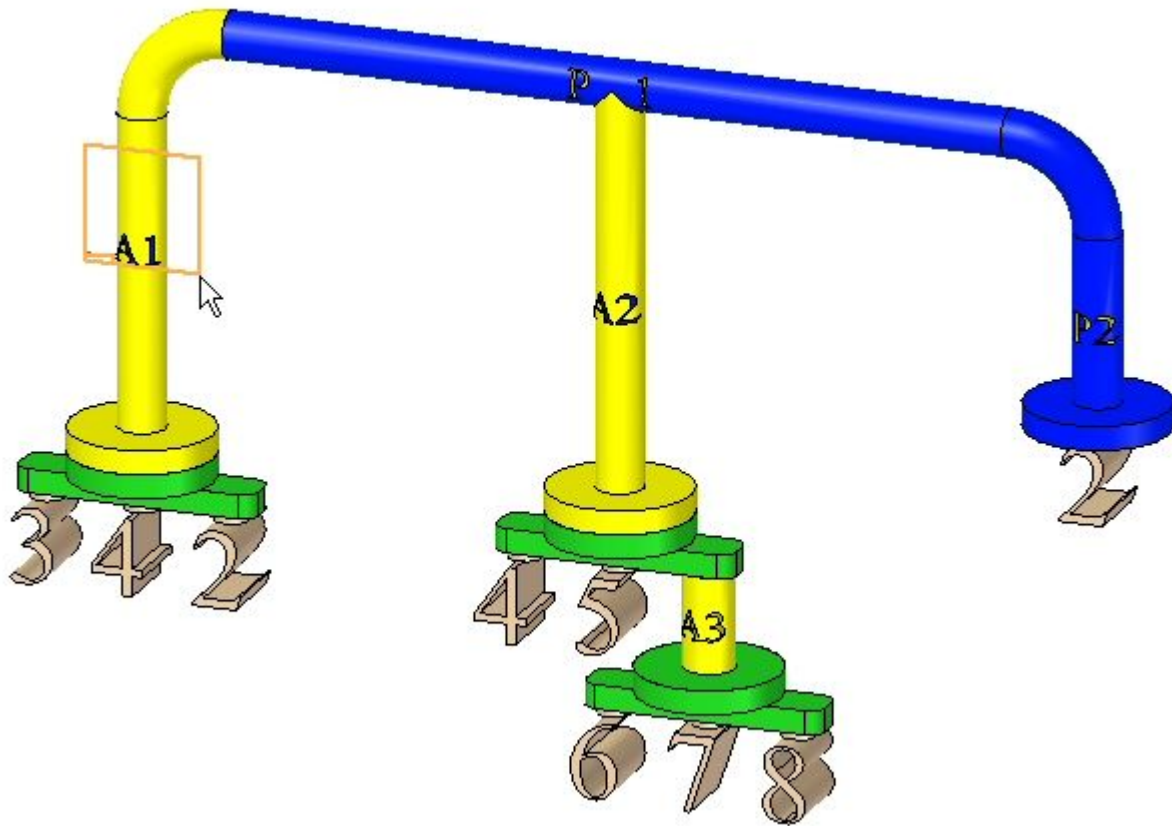
- ▶ Set the balloon shape to Circle .

- ▶ Ensure that the Item Number and Item Count buttons are selected as shown.

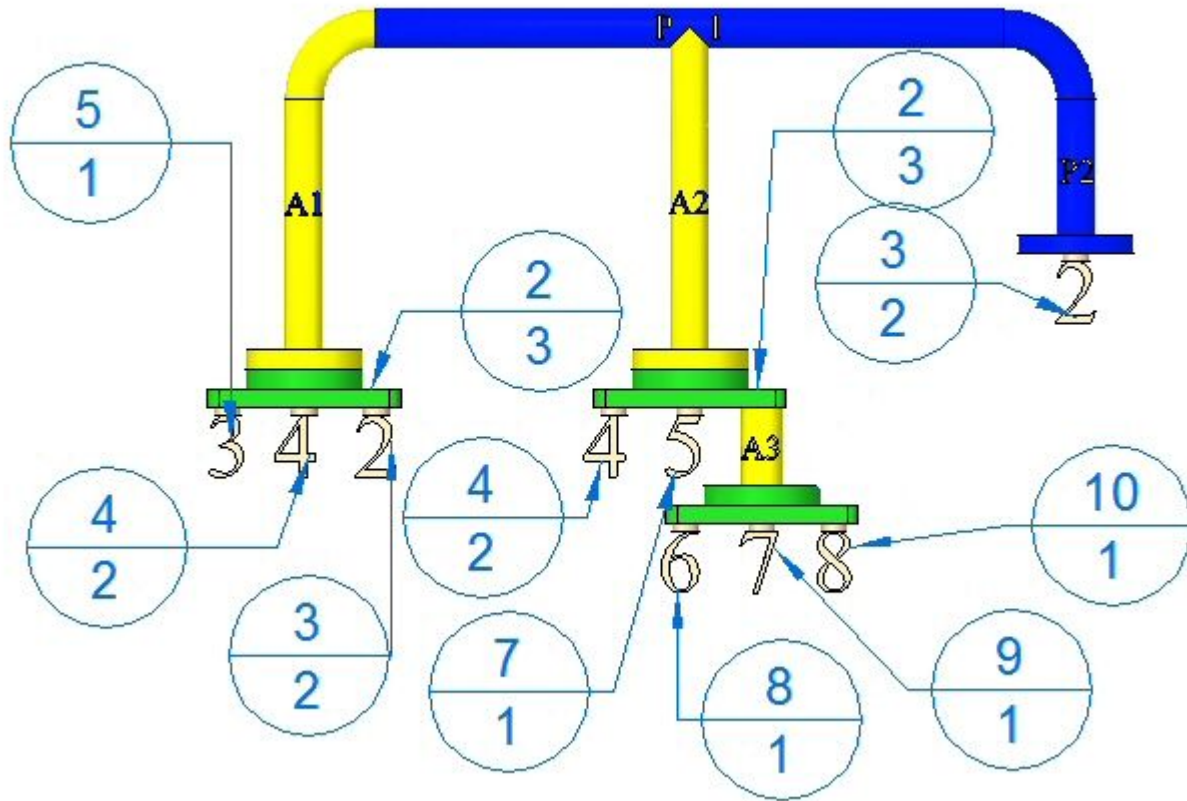



- ▶ Click Lock Dimension Plane .

- ▶ Select the Front reference plane as shown.



- ▶ Annotate the parts as shown. Placement is approximate.




- ▶ Click Save  to save the assembly.
- ▶ An optional step is to create a drawing sheet as you did for the top level item numbers, and place the front view and the balloons on the drawing sheet.
- ▶ Close the assembly.

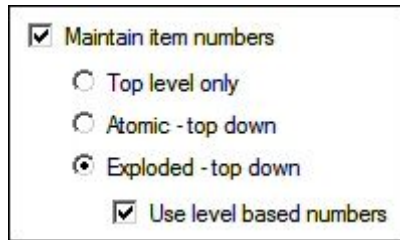
*Assigning atomic top down item numbers in an assembly*

- ▶ Open the assembly *top\_explode.asm* with all the parts active.




- ▶ Click the Solid Edge application button .
- ▶ Click Solid Edge Options, and then click the Item Numbers tab.

- ▶ Set the options as shown.



- ▶ Click Apply, and then click OK to dismiss the Solid Edge Options dialog box.

- ▶ Choose the PMI tab® Annotation group® Balloon command .

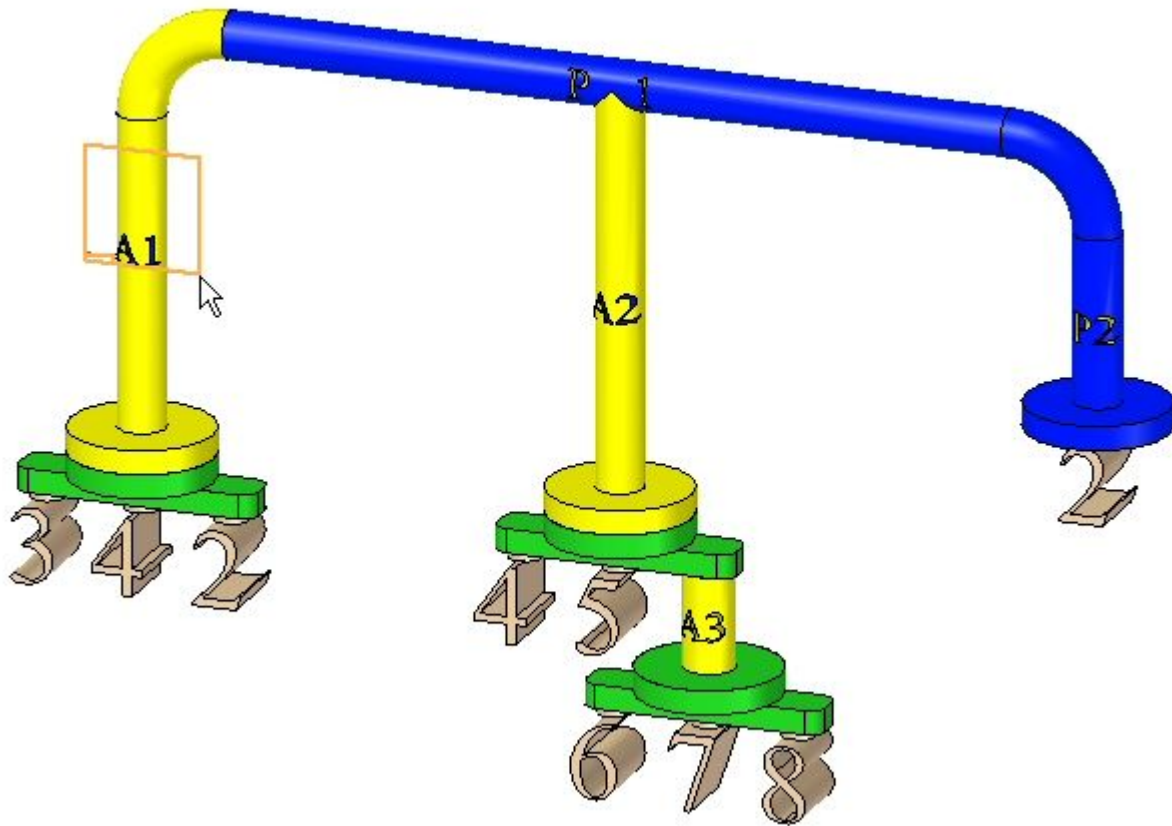
- ▶ Set the balloon shape to Circle .

- ▶ Ensure that the Item Number and Item Count buttons are selected as shown.



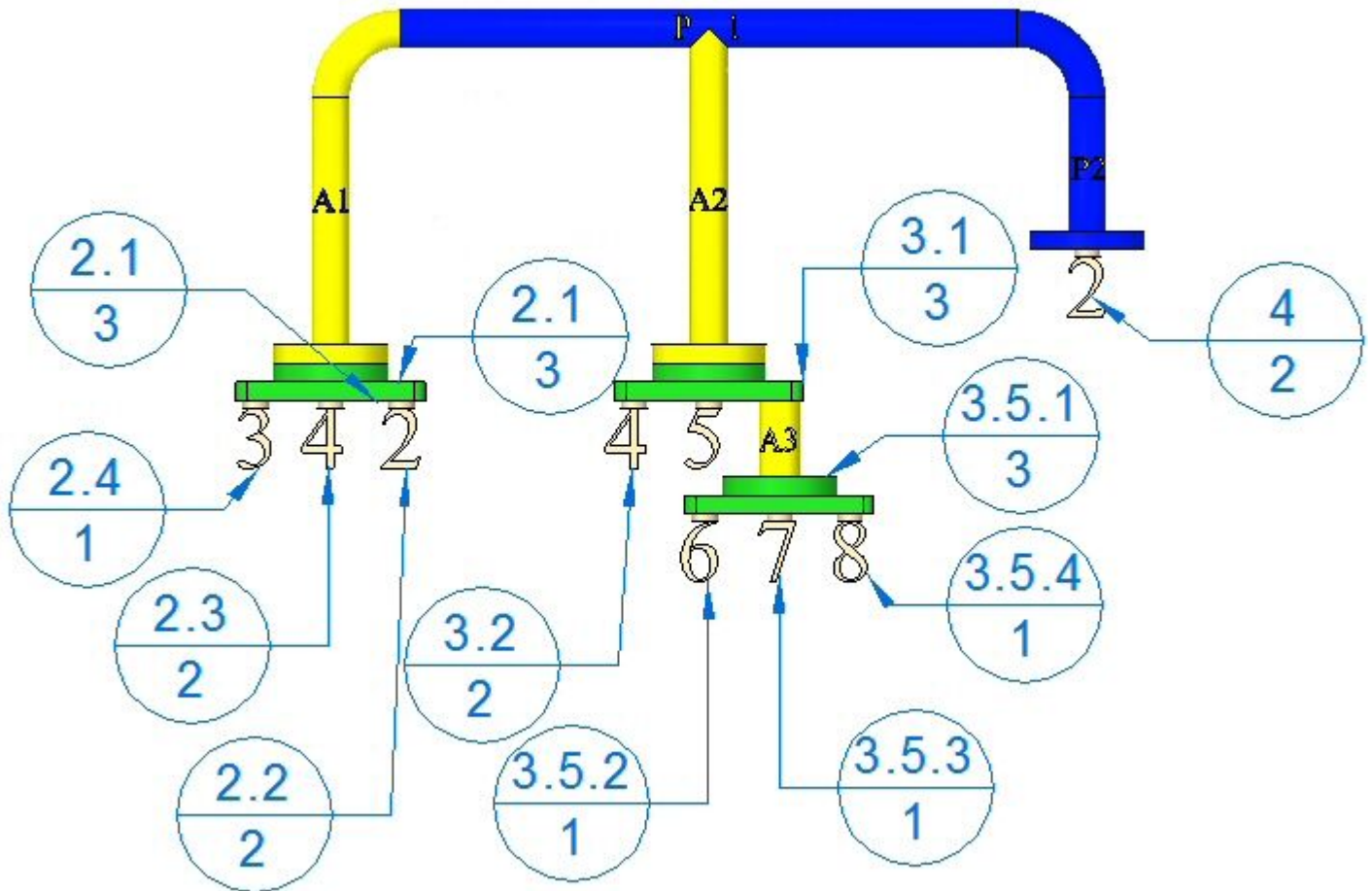
- ▶ Click Lock Dimension Plane .


- ▶ Select the front reference plane as shown.





- Annotate the parts as shown. Placement is approximate.



- Click Save  to save the assembly.
- An optional step is to create a drawing sheet as you did for the top level item numbers, and place the front view and the balloons on the drawing sheet.
- Save and close the assembly. This completes the activity.

### Summary

In this activity you learned some of the options available for generating assembly reports.

### **Lesson review**

Answer the following questions:

1. When item numbers are maintained, what is the difference between top level only, Atomic — top down, and exploded — top down?
2. What are level based item numbers?
3. If item number PMI balloons are placed in an assembly document, are you able to see these in a draft document, and if so, how?
4. Can the user define item numbers and if so how?

## Answers

- When item numbers are maintained, what is the difference between top level only, Atomic — top down, and exploded — top down?

The different item number schemes are defined as follows:

- Top level only item numbers are assigned to only the items directly placed in an assembly. These can be assemblies or parts placed into an assembly, but not the parts within the assemblies. These items are what pathfinder displays when every member is collapsed.
- Atomic — top down item numbers are assigned to only the parts that make up an assembly regardless of whether they are in the top level, or whether they are in a subassembly. Assemblies do not get item numbers, only parts.
- Exploded— top down item numbers are assigned to both parts and assemblies by the order they appear in pathfinder.

- What are level based item numbers?

Level based item numbers will have a number for each part or assembly. Parts and parts in subassemblies will be designated by which top level member they belong to, separated by a period. You can quickly tell which parts belong to subassemblies. See the table below for an example of all item numbering schemes.

	Level 1	Level 2	Level 3	Top level only	Atomic -Top down	Exploded - Top down	Use level based numbers
<b>Top assembly</b>							
A1				1		1	1
		P1			1	2	1.1
		P2			2	3	1.2
		A2				4	1.3
			P3		3	5	1.3.1
			P4		4	6	1.3.2
		P6			5	7	1.4
		P7			6	8	1.5
P2				2	2	3	2
P4				3	4	6	3
A2				4		4	4
		P3			3	5	4.1
		P4			4	6	4.2
P5				5	7	9	5

- If item number PMI balloons are placed in an assembly document, are you able to see these in a draft document, and if so, how?

The view containing the PMI balloons in assembly is placed onto a drawing sheet. The retrieve dimension command will display the balloons on the drawing sheet.

4. Can the user define item numbers and if so how?

User defined item numbers can be defined by right clicking the top level assembly in pathfinder, and then clicking occurrence properties. You can:

- Change existing item numbers.
- Add missing item numbers using the Next Available Number command on the shortcut menu.
- Use the Reset Item Numbers command on the shortcut menu to cancel your edits and restore the item numbers generated by the assembly structure.

## **Lesson summary**

In this lesson you learned some of the options available for generating assembly reports.



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**Lesson**

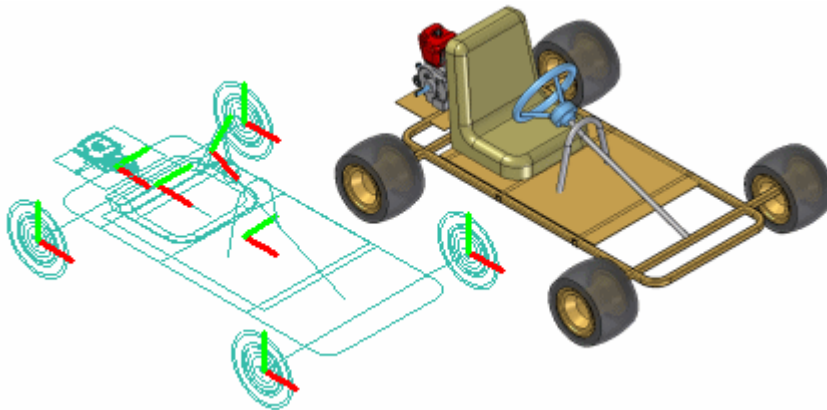
# *11 Virtual components in assemblies*

## Creating and publishing Virtual Components

When you start a new design project, you may want to define the overall product structure for the project before creating new Solid Edge documents, or before positioning 3D geometry for existing Solid Edge documents in the top-level assembly. In effect, you use a top-down design approach to define the assembly structure using virtual components as place holders until real components are defined.

You can use the Virtual Component functionality in Solid Edge or in Teamcenter to define the assembly structure for a new design project.

When you have finished defining the assembly structure, you can publish the assembly. Publishing the assembly creates the new Solid Edge documents required, copies assembly sketch geometry to the new documents, and adds the 3D geometry for existing Solid Edge documents to the assembly.



### Virtual components in Solid Edge

The basic workflow for creating and publishing a virtual assembly in the unmanaged Solid Edge environment is:

- Define the virtual components you need.
- Add existing documents to the virtual assembly structure.
- Assign 2D geometry to individual virtual components.
- Position virtual components.
- Publish virtual components.

To make it easier to understand the concepts and capabilities of the virtual component functionality in Solid Edge, a Virtual Component tutorial is available.

### Virtual components in a Teamcenter-managed Solid Edge environment

While you can use the virtual component functionality in Solid Edge to develop unmanaged assembly structures, most often in a managed environment, your product structure is created and modified in another Teamcenter PDM client such as Teamcenter Structure Manager. Then the empty items that are created in the PDM client are opened for further development along with the remaining structure in either Solid Edge or Structure Editor.



**Note**

As a best practice, you should choose between using Solid Edge Virtual Components and a workflow where Teamcenter empty items are created in a PDM client. Using both in the same structure is not recommended.

Once you have finished defining the assembly structure, you can *publish* the virtual components, converting non-modeled objects into physical documents containing Solid Edge 3D datasets. Publishing the components assigns the Solid Edge template (part, assembly, sheet metal or weldment), and Teamcenter attributes to the real document.

The basic workflow for working with Teamcenter-managed virtual assembly is:

- Create the virtual components you need using a Teamcenter PDM client.
- Open the structure in Solid Edge where you can view the structure in PathFinder.
- Add existing documents to the virtual assembly structure.

**Note**

If you are working in Solid Edge in the Teamcenter mode, you cannot place unmanaged parts into a Teamcenter-managed virtual component structure.

- Publish virtual components.
- Upload the real document into the Teamcenter database.

**Defining the assembly structure**

The Virtual Component Structure Editor command defines the assembly structure for a new design project in an unmanaged environment. When you click the Virtual Component Structure Editor command, the Virtual Component Structure Editor dialog box is displayed.

The Virtual Component Structure Editor defines the name and document type for new virtual components, and allows you to drag any existing Solid Edge documents into the virtual assembly structure.

**Defining new virtual components**

You use the right pane on the Virtual Component Structure Editor to define virtual components and organize the assembly structure. The Component Type options on the right pane of the Virtual Component Structure Editor define the document type for new virtual components:

- Assembly components
- Part components
- Sheet metal components

The Name option assigns a name to the virtual component you are creating. You can select a default name from the list, or type the name you want in the Name box.

You can customize the Names list by editing *VCNames.txt* in the Solid Edge Program folder. Separate sections in *VCNames.txt* define unique virtual component names for part, sheet metal, and assembly components.

After you define the component type and name, you can click the Add Virtual Component button on the Virtual Component Structure Editor dialog box to add the virtual component to the tree structure list. You can also add a virtual component to the tree structure list by pressing Enter after typing the virtual component name.

**Note**

When you create a virtual component, it has no physical position and no associated graphics. Later, you can position the virtual component and/or assign graphics to it.

You can use the Promote Component and Demote Component buttons to further define the assembly structure. For example, if you want to move a component from a virtual subassembly into the next higher level assembly, you can select the component and click the Promote Component button.

You can also drag virtual components within the Virtual Component Structure Editor dialog box to rearrange the assembly structure. For example, you can drag a virtual part from one virtual subassembly to another virtual subassembly.

### Adding existing documents to the virtual structure

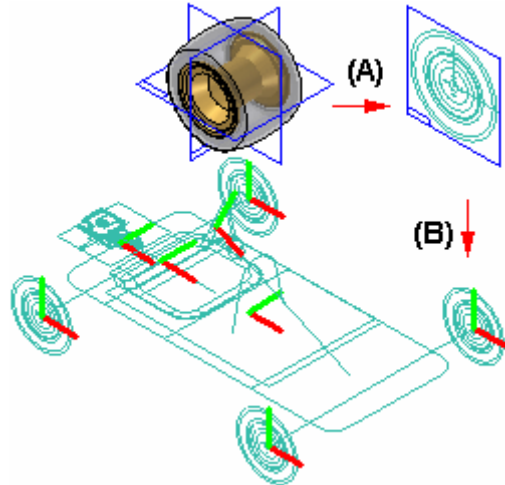
You can also place existing Solid Edge documents into the assembly structure using the Virtual Component Structure Editor dialog box.

The left pane on the Virtual Component Structure Editor dialog box browses to and selects existing documents on your computer or another computer on your network. If you are working in Teamcenter mode, you can select existing documents from your Teamcenter database.

You can place an existing document into the virtual assembly structure in one of two ways: as a pre-defined component or as a real component.

#### Pre-defined components

Pre-defined components are virtual components that are based on an existing document. You cannot place 3D geometry using a pre-defined component, but you can create 2D geometry in the parent document (A), then position the 2D geometry in the assembly sketch (B). You can reuse existing documents in new design projects without the overhead of 3D geometry.



When you clear the Add As Real Component option on the Virtual Component Structure Editor, then drag an existing component into the right pane of the structure editor, the component is placed as a pre-defined component. If you want to add the document to a virtual subassembly, position the cursor over the virtual subassembly when dropping the pre-defined component.

Pre-defined components are typically purchased or released parts that are not subject to significant changes.

#### Adding real components

When you set the Add As Real Component option on the Virtual Component Structure Editor before you drag an existing document into the virtual assembly structure, the 3D geometry associated with the component is added to the assembly. When this option is set you can only place the component in the top-level assembly.

The component is placed in the assembly hidden, with no relationships applied, and is oriented in the assembly by aligning the base reference planes of the component with the base reference planes in the assembly.

You can position the real component within the assembly sketch using assembly relationships such as mate and align, or 2D relationships and dimensions such as Connect and Distance Between.

### Copying virtual and pre-defined components

You can specify that the same part or subassembly occurs more than once using the Virtual Component Structure Editor dialog box. To specify that a component occurs more than once, you can select an existing virtual component or pre-defined component, then click the Copy Definition command on the shortcut menu.

This copies the component name and component type to the Name box and Component Type list. You can then press Enter or click Add Virtual Component to add the component to the tree structure list. You can also add the same component multiple times by pressing Enter repeatedly.

### Saving the tree structure

When you click the OK button on the Virtual Component Structure Editor dialog box, the virtual assembly structure you defined is added to the PathFinder tab. You can make changes to the virtual assembly structure later using the Virtual Component Structure Editor dialog box.

You can dismiss the Virtual Component Structure Editor dialog box without saving changes by clicking the Cancel button.

### Assigning sketch geometry to Virtual Components

You can assign 2D sketch geometry from the assembly layout to a virtual component. You can create the 2D geometry before or after you define the assembly structure.

Assigning geometry to a virtual component defines its size and position in the assembly sketch. You can only assign geometry to one occurrence of a particular virtual component.

When you assign 2D geometry to a virtual component, it becomes the master, or source component. You can edit the sketch geometry associated with the source component directly. If there are additional occurrences of a particular virtual component, they become slave or instance components.

The sketch geometry for an instance component is an associative copy of the sketch geometry of the source component. When you update the sketch geometry for a source component, the sketch geometry for the instance components updates automatically. You cannot modify the sketch graphics for an instance component directly.

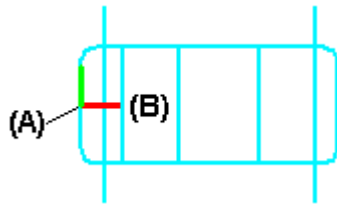
You assign geometry to a virtual component using the Edit Definition command on the Assembly PathFinder shortcut menu. This command is available only when an assembly sketch window is open.

When you click the Edit Definition command, a command bar is displayed so you can select the geometry and define an origin. You can select the 2D sketch geometry using the cursor or by dragging a fence. When you finish selecting geometry, click Accept or right-click to proceed to the Orientation Step.

The Orientation Step defines the origin location for the virtual component and the reference plane that the 2D sketch graphics are placed on when you publish the virtual component. You can specify that the graphics are placed on one of the three base reference planes.

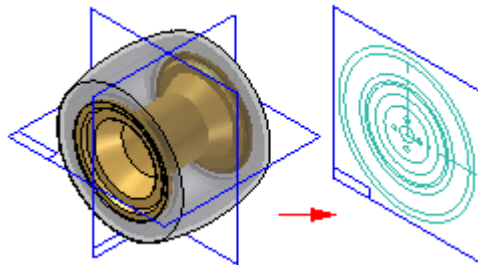
When defining the origin you can specify that the origin is based on a keypoint or a free space point. With either method you must also define the x-axis direction. When you click the Finish button, a symbol is displayed that represents the origin (A) and x-axis direction (B) for the virtual component geometry.

You can constrain to the origin point on a virtual component to position it within the assembly.



### **Creating sketch geometry for pre-defined Virtual Components**

You create the sketch geometry for a pre-defined virtual component by opening the parent document, then using the Component Sketch command to create a component sketch.



There are two types of graphics you can create in a component sketch: component image graphics, and wireframe graphics. Component image graphics are created using the Component Image command. This command creates a 2D representation of the visible edges on a part.

Component image graphics are not individually selectable in the assembly sketch, but provide a reference envelope.

You supplement the component image graphics with wireframe graphics you create using the Include command or drawing wireframe graphics manually. The wireframe graphics can be selected and used for constraining the component sketch in the assembly sketch.

You typically would create both types of graphics for a component sketch. In some cases, you may only create component image graphics, then use the virtual component symbol origin to constrain the pre-defined component in the assembly sketch.

Using too many wireframe graphic elements can impact performance when placing the component sketch in the assembly sketch.

## Updating sketch geometry for pre-defined Virtual Components

As discussed earlier, pre-defined components are typically purchased or released parts. If the parent 3D geometry for a pre-defined component sketch changes, you must delete the component image graphics and create new graphics using the Component Image command.

You can then open the assembly sketch and select the pre-defined component within PathFinder. Click the Update Component command on the shortcut menu to update the pre-defined component graphics in the assembly.

## Replacing pre-defined components

You can use the Replace command on the shortcut menu when a sketch is active to replace a pre-defined component with another Solid Edge document you specify. You can select the pre-defined component in Assembly PathFinder or the sketch window.

## Displaying Virtual Component sketch geometry

Before you assign assembly sketch graphics to a virtual component, you use the Show and Hide commands on the Layers tab shortcut menu to display and hide the assembly sketch graphics. An effective layer management scheme can make working with complex assembly sketches more productive.

After you assign assembly sketch graphics to a virtual component, you can use the Show and Hide commands on the PathFinder shortcut menu to display and hide the virtual component sketch graphics. For example, you can select the virtual component in PathFinder, or in the graphics window, then click the Show and Hide commands on the shortcut menu.

You control the display of dimensions you apply to the sketch geometry by showing and hiding the layers on which the dimensions were created, both before and after you assign graphics to a virtual component.

Because geometric relationship handles do not reside on a layer, you always control their display using the Relationship Handles command on the Tools menu when in the assembly sketch.

## Positioning Virtual Components

You use the Position Virtual Component command on the PathFinder shortcut menu to position virtual components. This command is only available when you are editing an assembly sketch. You can position an empty virtual component (a virtual component that has no geometry assigned), an instance virtual component, or a pre-defined component.

You can position a virtual component by dragging it from the PathFinder tab and dropping it into the sketch window.

As discussed earlier, source virtual components are positioned as part of the process of assigning geometry.

### Positioning empty Virtual Components

If you have not assigned geometry to any occurrences of a particular virtual component, you can select the virtual component in PathFinder, then click the Position Virtual Component command on the shortcut menu. The virtual

component symbol is attached to the cursor so you can position the virtual component on the assembly sketch.



The symbol represents the approximate position for the empty virtual component in the assembly. Later, you can select the empty virtual component entry in PathFinder and use the Edit Definition command to assign geometry to one occurrence of the empty virtual component. This instance of the virtual component becomes the source virtual component.

The other occurrences of the virtual component update with the graphics you assigned to the source component, and they become instance components.

You can then add relationships and dimensions to the sketch geometry to precisely locate the virtual component in the assembly sketch.

### **Positioning instance Virtual Components**

After you assign sketch geometry to a virtual component, it becomes the source component for additional occurrences of that virtual component. You can select another occurrence of the virtual component in PathFinder and use the Position Virtual Component command on the shortcut menu to position the virtual component on the active assembly sketch.

A copy of the sketch graphics that were assigned to the source component is attached to the cursor. As discussed earlier, the sketch geometry for an instance component is an associative copy of the sketch geometry of the source component. You can apply relationships and dimensions to the sketch graphics or to the virtual component symbol to precisely position the instance component on the assembly sketch.

### Determining the status of a Virtual Component

The symbols in PathFinder and the Virtual Component Structure Editor dialog box reflect the current status of the virtual components in the assembly.

Teamcenter-managed non-modeled nodes can either be assembly or leaf nodes. Assembly nodes contain references or children, while leaf nodes do not contain references or children. The symbols in PathFinder, Assembly Reports, and the Property Manager dialog box reflect the state of the virtual components in the assembly. The following table explains the symbols used:

Teamcenter				Solid Edge			
ASM	PAR	PSM	Leaf	ASM	PAR	PSM	
							Source part
							Instance virtual part that has been positioned
							Position-less virtual part whose source virtual component has graphics assigned
							Positioned Empty virtual part (no graphics assigned)
							Position-less pre-defined component
							Positioned pre-defined component

**Legend**

Teamcenter				Solid Edge				
ASM	PAR	PSM	Leaf	ASM	PAR	PSM		
<hr/>								
							Source part	
							Instance virtual part that has been positioned	
							Position-less virtual part whose source virtual component has graphics assigned	
							Positioned empty virtual part (no graphics assigned)	
							Position-less pre-defined component	
							Positioned pre-defined component	

**Note**

The Teamcenter Status and Checked Out By information displayed in PathFinder is not available for an object until it is published.



## **Publishing Virtual Components**

When you are ready to create the document set for the new design project, you can click the Publish Virtual Components command. You can then use the Publish Virtual Components dialog box to specify the target folder and template you want to use to create the document set.

In an unmanaged Solid Edge environment, when you click Publish, all virtual components are published at one time. The unmanaged Solid Edge documents are created using the names, folder path, and templates you specified.

If you have associated 2D assembly sketch graphics with source virtual components, the sketch graphics, including dimensions and relationships, are copied to the proper document as sketches. The sketch graphics are positioned in the new document using the Publish On option you specified on the Edit Definition command bar when you assigned the sketch geometry to the source component.

There is no associative link between the sketch graphics in the original assembly and the sketch graphics copied to the new documents.

The sketch graphics associated with instance virtual components are deleted from the source assembly.

You cannot publish a virtual component to the same folder as a real component of the same name. For example, if you add both a pre-defined component named *bolt.par* and a virtual component named *bolt* to the virtual component structure, they are not allowed to reside in the same folder when you publish the virtual components.

Conflicts such as these are indicated in the Publish Virtual Components dialog box using red text and an exclamation mark (!). In this example, you can resolve the conflict by renaming the virtual component or by specifying a different folder for the virtual component.

If you do not rename the virtual component or specify a different folder, the existing document will be used and the new document for the virtual component will not be created. The existing part is positioned as defined by the virtual component sketch, but the sketch geometry is not added to the existing document.

### **Publishing pre-defined components using the simplified representation**

If a simplified representation of a part or subassembly exists, you can specify that the simplified representation is used for pre-defined components. When you set the Publish Pre-defined Components Using Simplified Representations option on the Publish Virtual Components dialog box, the simplified representation is used.

This can improve processing time and reduce document size. When you publish a simplified representation of a subassembly, the subassembly is published as a single unit and the assembly structure is not loaded into memory. The assembly structure is also not displayed within PathFinder.

You can use commands on the PathFinder shortcut menu to specify whether the simplified or as designed version of the parts and subassemblies are displayed. For more information on working with simplified parts and assemblies, see the [Simplifying Parts](#) and [Simplifying Assemblies](#) Help topics.

## **Publishing documents in the Teamcenter-managed environment**

When you are ready to create the document set for the new design project, you have three options. You can perform an ad hoc publish on a single object by selecting

the virtual component in PathFinder and dragging it into the Solid Edge graphic window. You are then prompted to choose the template for the object to give it a Solid Edge dataset, and then the New Document dialog box displays so you can check in the real document.

Another option is to use the Publish Virtual Components command. You can then use the Publish Virtual Components dialog box to select components for either a partial or full publish, and to determine the template you want to use to create the document.

During a partial publish, you select a component of an assembly for publishing by selecting the associated check box. Then you are given the opportunity to assign a template. The New Document common property dialog box assigns Teamcenter attributes, and the physical document is created with a Solid Edge dataset.

A full publish involves selecting all components of a structure. The entire virtual structure is published at the same time. You use the New Document common property dialog box to assign Teamcenter attributes, and the documents are created with Solid Edge datasets.

In all three cases, if a virtual component is denoted as an assembly, only Solid Edge assembly templates are displayed during template selection. If the virtual component is a leaf node (one that has no references or children), then you can choose from the 3D Solid Edge templates during template selection.

### **Virtual Components and assembly reports**

If the assembly for which you are creating a report contains virtual components, you should always use the Reports command in the Assembly environment.

When you run the Reports command from Windows Explorer on an assembly that has virtual components, the virtual components will not be contained in the report. If the assembly contains only virtual components, a message may be displayed that states that no parts are in the file.

When you run the Reports command from Windows Explorer on an assembly that has virtual components, the Assembly Report shows Teamcenter virtual components and uses the same symbols for identification that you see in PathFinder.

### **Assigning properties to Virtual Components**

When an active document contains a virtual component, you can use the Property Manager command to modify existing properties or create new properties for the virtual component in Solid Edge.

#### **Note**

Virtual components created in Teamcenter are shown, but are read-only and cannot be edited.

When you select the Property Manager command, the Property Manager dialog box is displayed for editing property values. Any properties that you cannot edit are disabled and appear in gray.

To edit a value, click the appropriate property cell and type in the new value. When you edit a property, if the document containing the property is a managed document, it is checked out to prevent others from making changes. After you edit a property value, the property cell is underlined to indicate that it has been changed. The cell

remains underlined until you click the Save button to save the changes or click the Restore button to set the value back to the previous value. You can use the Copy, Cut, and Paste buttons to edit information between cells. When you click OK, the property changes are written back to the document in memory. The changes are not written until you save the document.

When you publish the virtual components, the properties you assigned to the virtual components are added to the new documents.

For more information on editing document properties, see [Document Properties](#).

## **Activity: Virtual component editor**

Activity guides you through the process of laying out an assembly using the virtual component structure editor and then publishing the assembly to create the basic design which is further refined.

## **Activity: Virtual component editor**

### **Overview**

This activity shows top down assembly design using the virtual component structure editor.

### **Objectives**

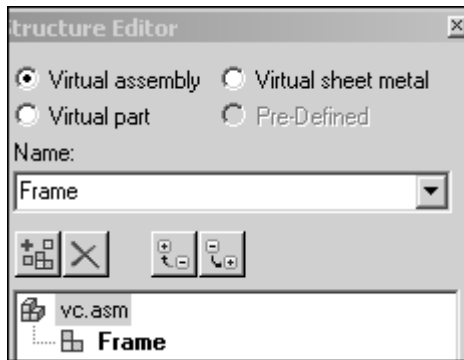
The virtual component editor allows a designer to use Solid Edge assembly sketches as a component layout for a new assembly. The orientation and positioning of future parts and subassemblies can be defined in the new assembly as well as existing parts and subassemblies. In this activity, you will use a Solid Edge assembly sketch to define geometric positions of parts that will be created in a top down method. When the virtual components are published, the files needed to complete the assembly will exist and the sketches will be used to create the new geometry.

**Define the virtual assembly structure**

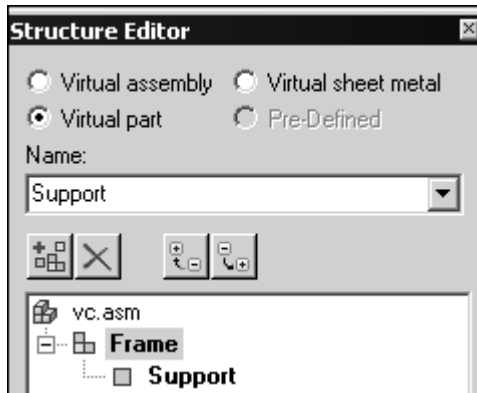
- ▶ Open *vc.asm* located in the folder where you put the activity documents.
- ▶ Click the Home tab, in the Assemble group, click Component Structure Editor.



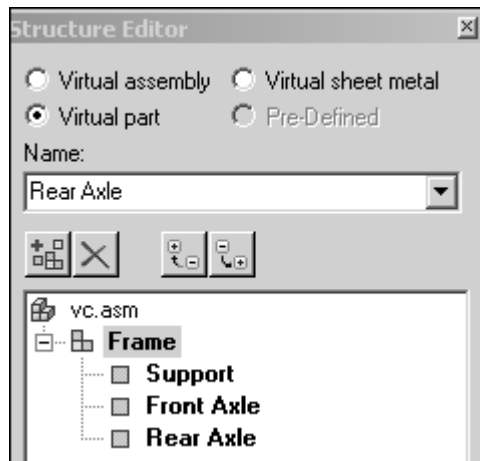
- ▶ Click the Virtual assembly option to set the type. Type Frame in the Name box and press the Enter key.



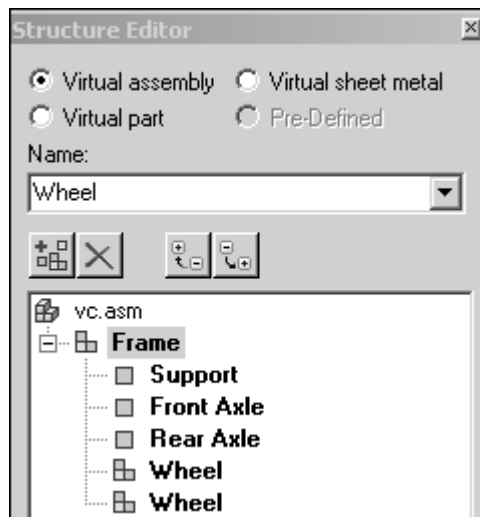
- ▶ Click the Frame assembly, and then click the Virtual part option. Enter Support in the Name box.



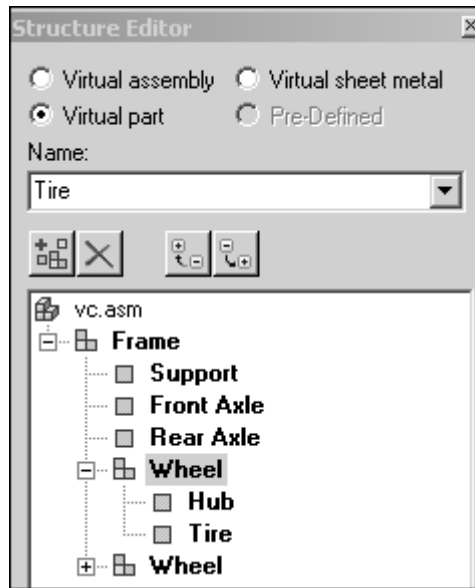
- ▶ Click the Frame assembly, and add the virtual parts Front Axle and Rear Axle.



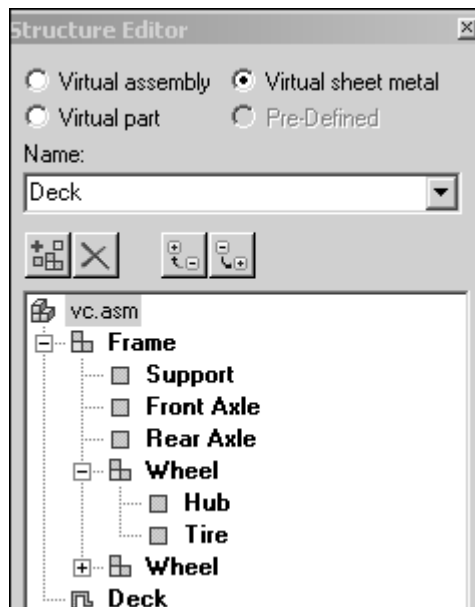
- ▶ Click the Virtual assembly button and enter Wheel in the Name box. Enter Wheel again to create a second assembly of the same name.



- ▶ Select the Wheel assembly and click the Virtual part option. Enter Hub, and then enter Tire in the Name box.



- ▶ Select *vc.asm*. Click the Virtual sheet metal option and add the sheet metal part Deck, then click OK to exit the Virtual Component Structure Editor.





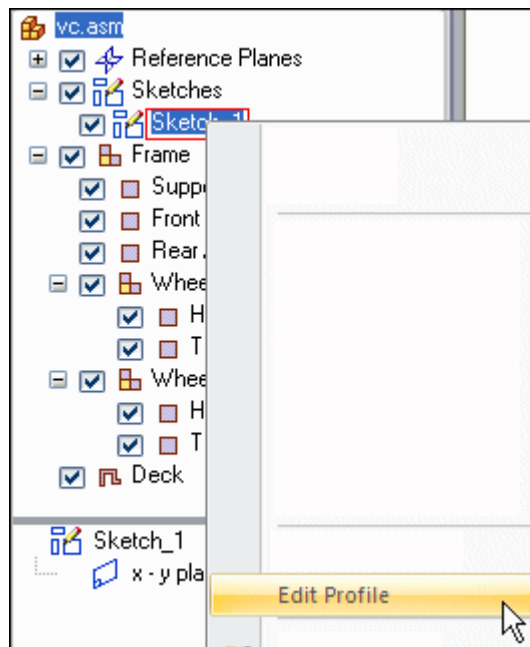
## Assign geometry to parts

Assign geometry from an existing sketch in *vc.asm* to parts in the virtual component structure.

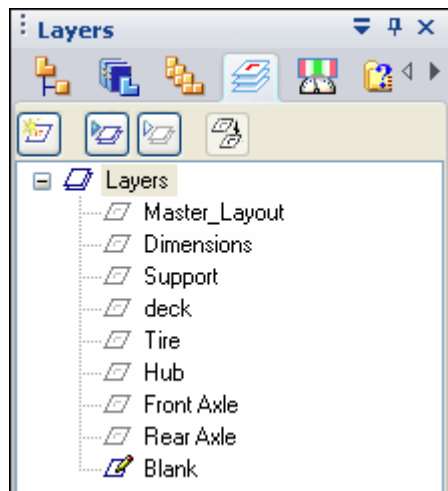
### Note

Layers are used on the sketch so that geometry can easily be chosen and assigned to the proper virtual component. Once assigned, a graphic sketch element cannot be selected, preventing geometry being assigned to multiple virtual components.

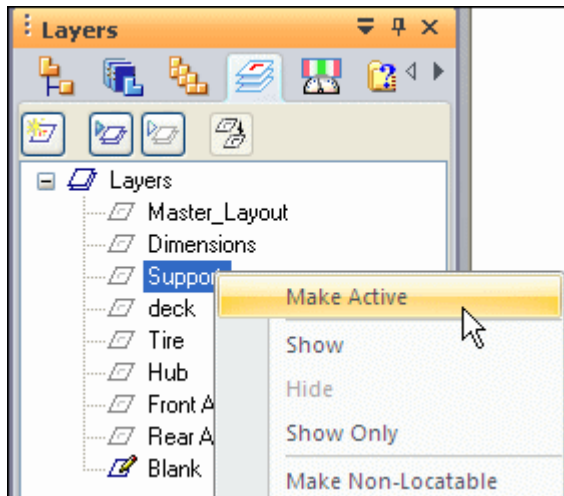
- ▶ On the Assembly PathFinder, expand the assembly structure as shown by clicking the + symbol. Right-click *Sketch\_1* and select Edit Profile.



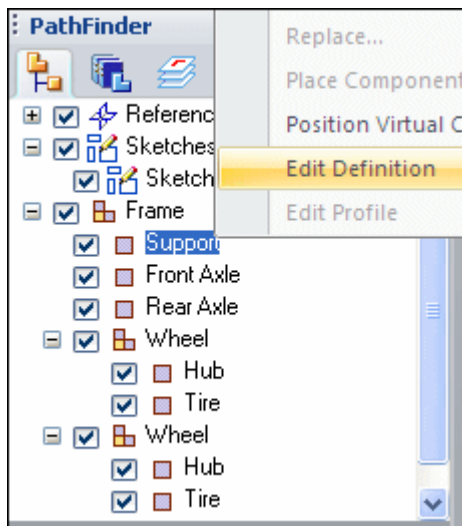
- ▶ Click the Layers tab on PathFinder.



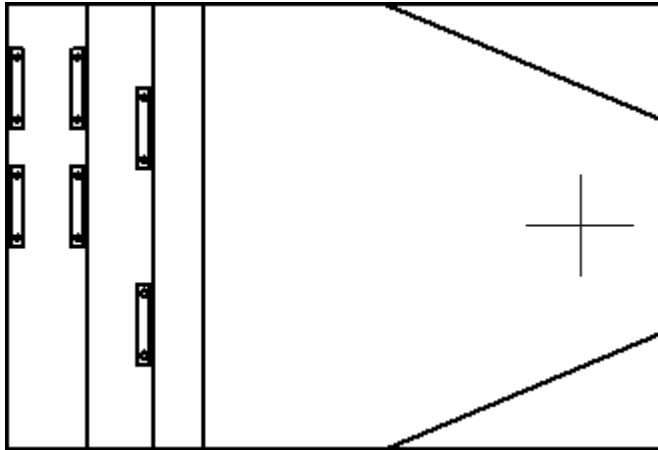
- ▶ Right-click the Support layer and click Make Active.



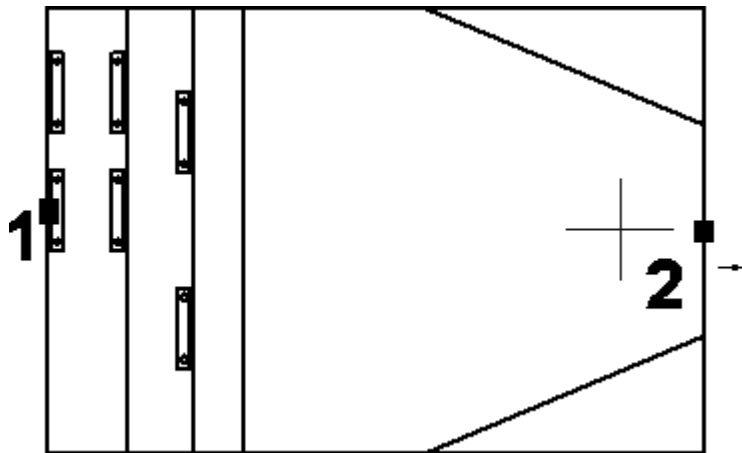
- ▶ On the Assembly PathFinder, right-click the virtual part Support and click Edit Definition.



- ▶ Using the left mouse button, drag a box around the sketch to select everything on the layer. Click the Accept button.



- ▶ After clicking Accept in the previous step, the origin of the orientation axis is ready to be defined. To select the origin of the axis, click the midpoint of the left most vertical line (point 1) as shown. To orient the x axis, click the midpoint of the right most vertical line (point 2) as shown.

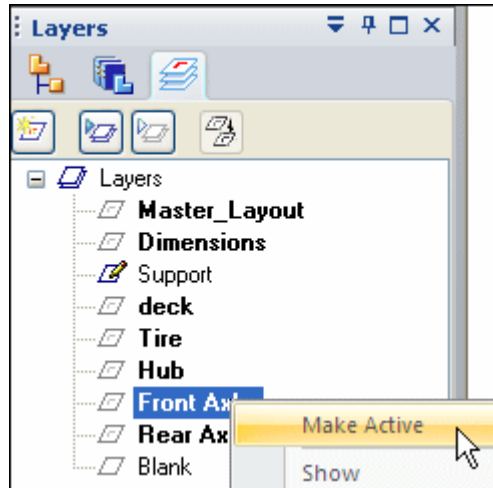


- ▶ Click Finish.

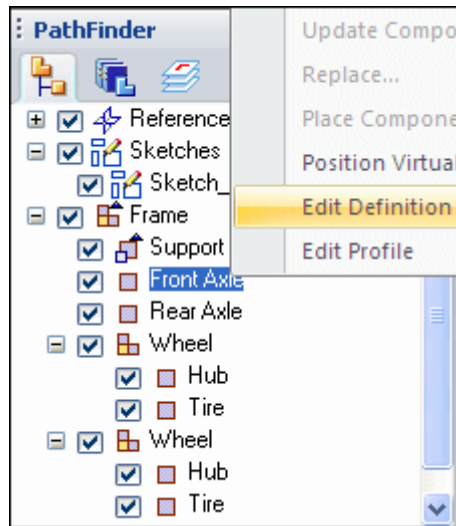
- ▶ In PathFinder select the Layers tab and right-click the layer Front Axle and make it active as shown.

**Note**

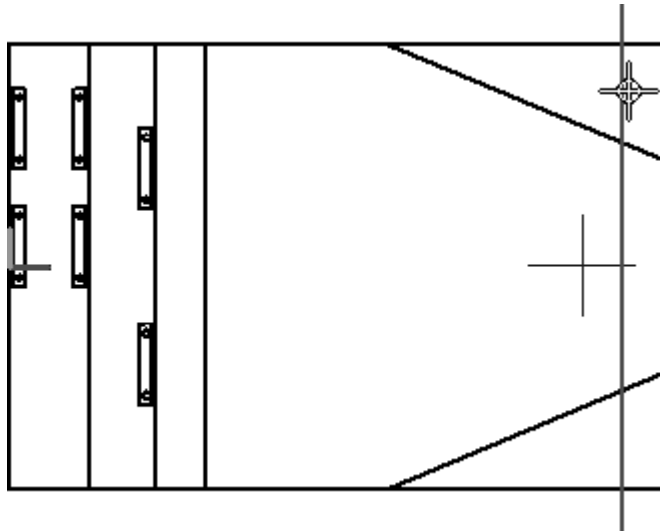
Displayed geometry that has been assigned to a virtual component is not selectable when editing the definition of another virtual component.



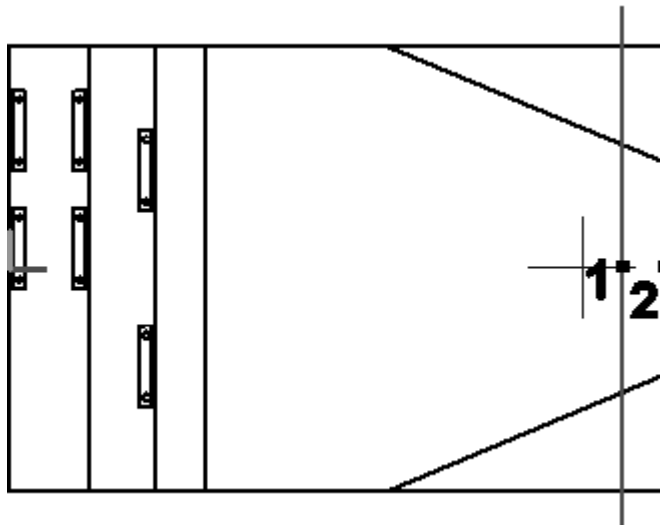
- ▶ On the Assembly PathFinder, right-click the virtual component, Front Axle, and click Edit Definition.



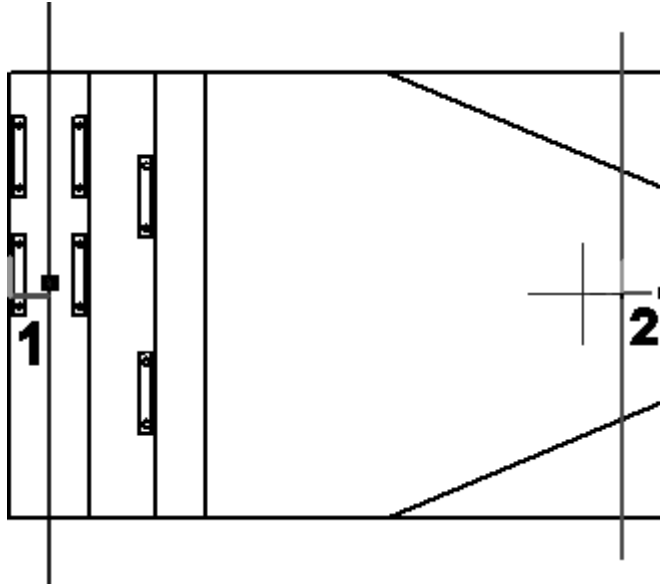
- ▶ Select the Front axle as shown and click Accept.



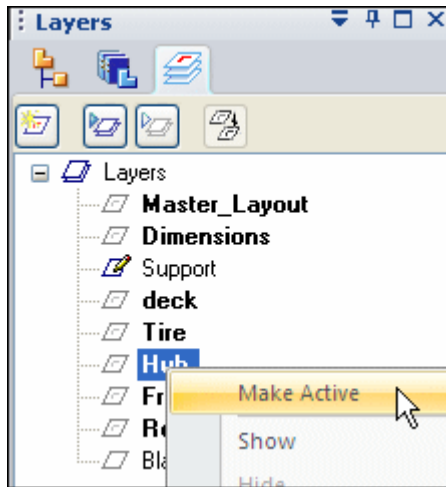
- ▶ Position the front axle by selecting the origin point of the axis to be the midpoint of the line as shown (point 1) and to orient the axis, select the midpoint of the right-most vertical line as shown (point 2), then click Finish.



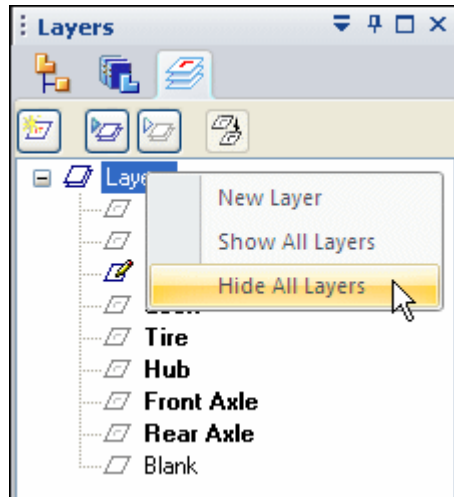
- ▶ In PathFinder, click the Layers tab and make Rear Axle the active layer. Click the Assembly PathFinder tab, then right-click the virtual component Rear Axle and then choose Edit Definition. Select the rear axle and accept. Orient the axis using the midpoints of the lines shown below, and click Finish.



- ▶ Click the Layers tab. Right-click the layer Hub and make it active.



- ▶ Right-click Layers and click Hide All Layers.



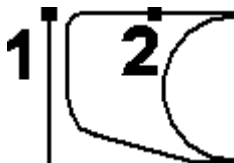
**Note**

Geometry that has been previously added to virtual components cannot be hidden by sketch layers and is not available to be added to additional virtual components.

- ▶ On the Layers tab, show the layer Rear Axle.
- ▶ On the Assembly PathFinder tab, right-click the virtual component Hub and click Edit Definition. Use a fence to select all the geometry on the layer Hub and click Accept.



- ▶ Orient the axis using the points shown below, and click Finish.

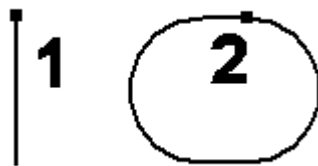


- ▶ On the Layers tab, make the layer Tire active and hide the layer Hub.

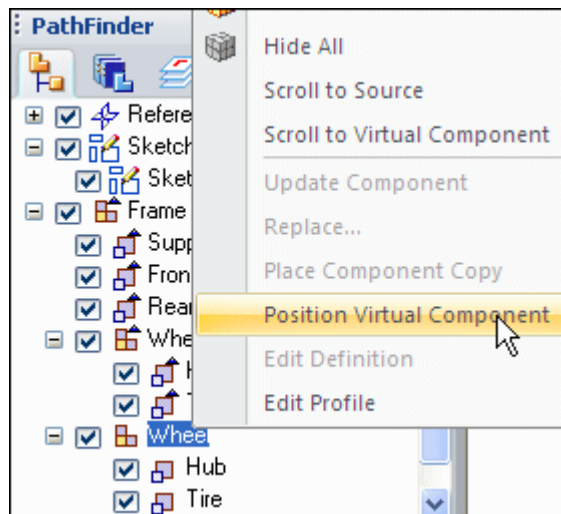
- ▶ On the Assembly PathFinder tab, right-click the virtual component Tire and click Edit Definition. Use a fence to select all the geometry on the layer Tire and click Accept.



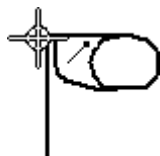
- ▶ Orient the axis using the points shown below, and click Finish.



- ▶ On the Layers tab, show the layer Front Axle.
- ▶ On the Assembly PathFinder tab, right-click the second occurrence of the assembly Wheel and click Position Virtual Component.



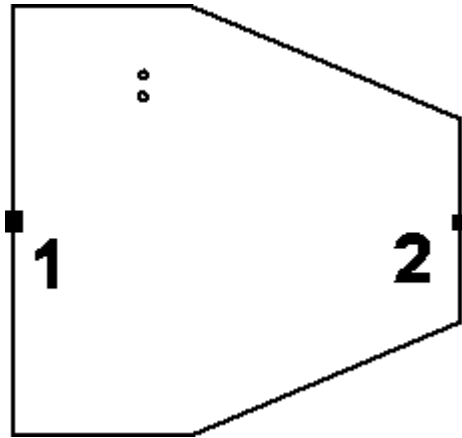
- ▶ Click the top of the vertical line that represents the front axle.



- ▶ On the Layers tab, make the layer Deck the active layer and hide all the other layers.



- ▶ On the Assembly PathFinder tab, right-click the virtual component Deck, and click Edit Definition. Use a fence to select all the geometry on the layer Deck, and accept. Orient the axis using the two points shown, and then click Finish.

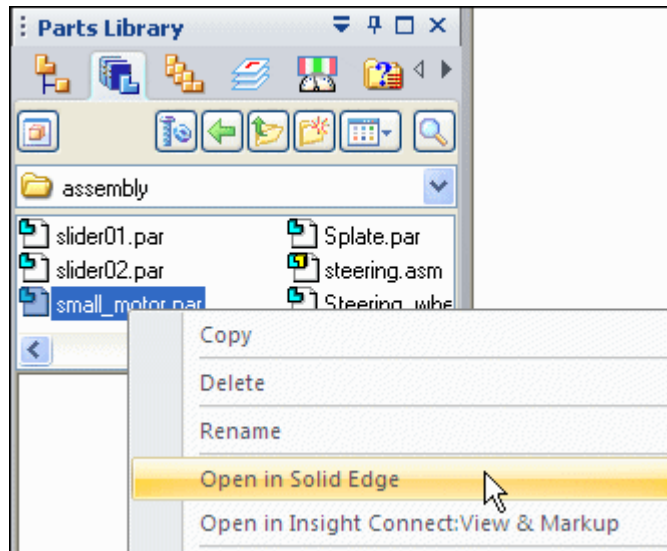


- ▶ On the Layers tab, show all the layers.
- ▶ Click Close Sketch to exit the sketch, then click Finish.

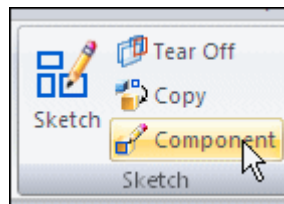
### Create component sketches

Create component sketches in three existing parts to position the parts in the virtual assembly.

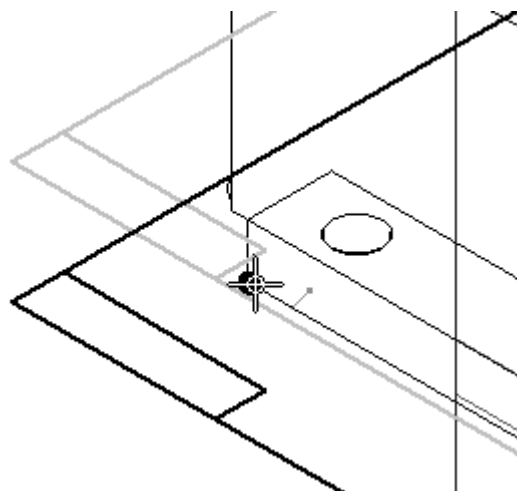
- ▶ In the Parts Library tab of PathFinder, right-click *small\_motor.par* and click Open in Solid Edge. Browse to the location where the activity files are located.



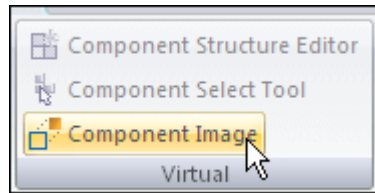
- ▶ On the Home tab, click the command Component Sketch.



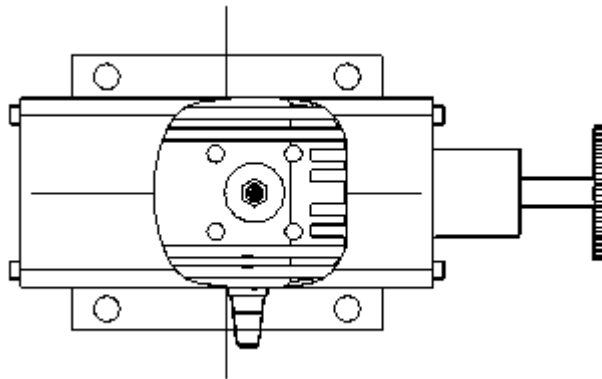
- ▶ Create the sketch on a plane parallel to the top reference plane. Use the keypoint shown to position the parallel plane.



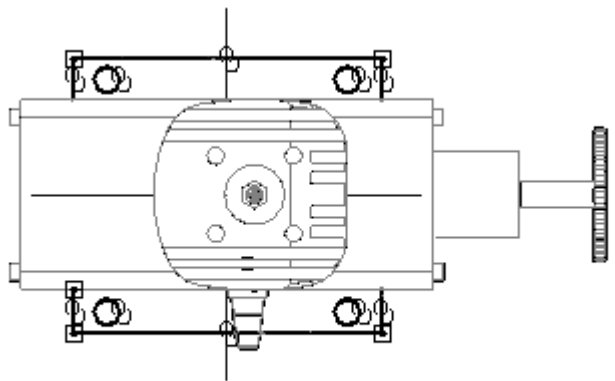
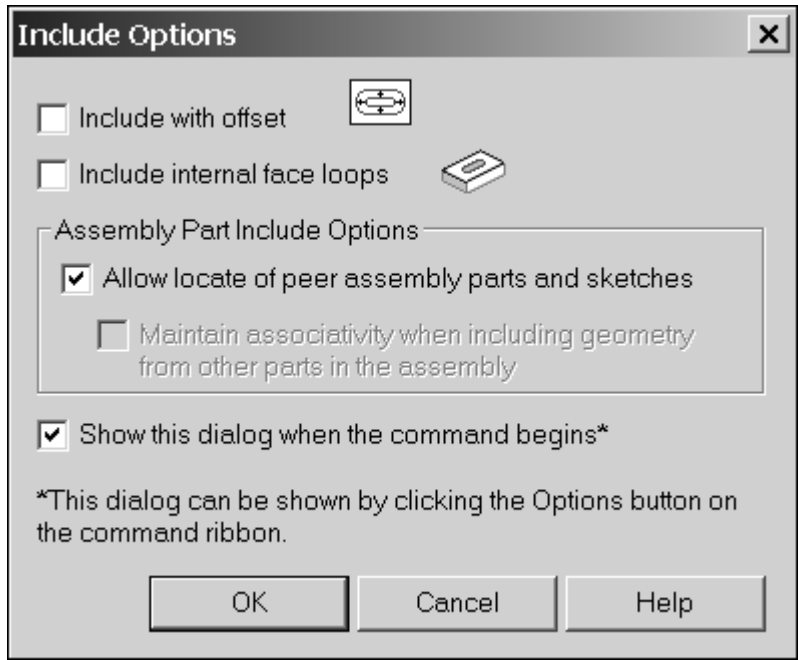
- ▶ Click the Tools tab. In the Virtual group, click the Component Image command.



- ▶ Click Accept to add the highlighted geometry to the sketch.

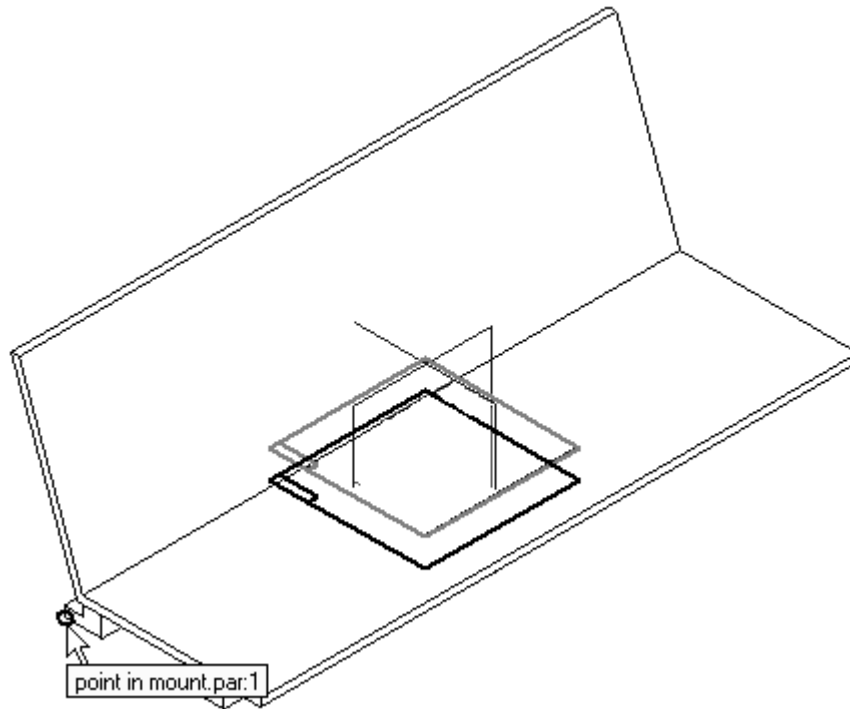


- On the Home tab, in the Draw group, click the Include command and include the edges shown. The Include command is needed to add precise geometry to the sketch in order to accurately position the part.

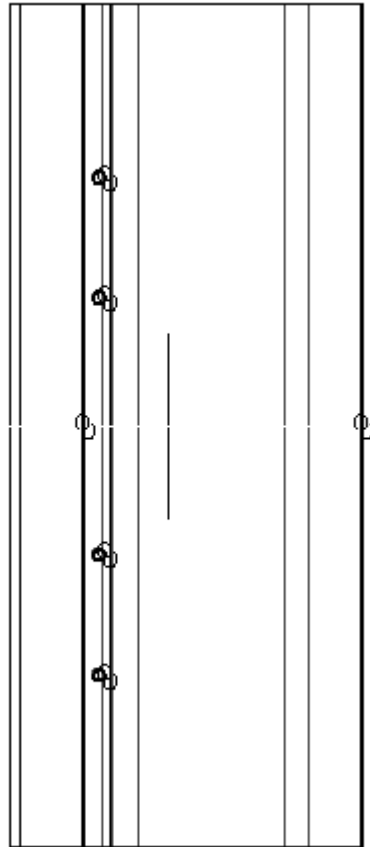


- Click Close Sketch, then click Finish. Save and close the file.
- In the Parts Library tab of PathFinder, right-click *wide\_seat.par* and click Open in Solid Edge.

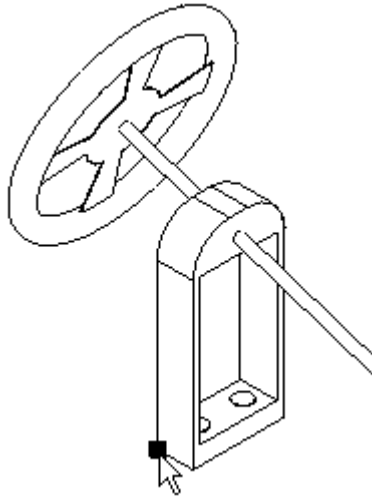
- Create a component sketch on a plane parallel to the top reference plane. Use a keypoint on the bottom face of the mount as shown to position the sketch's reference plane.



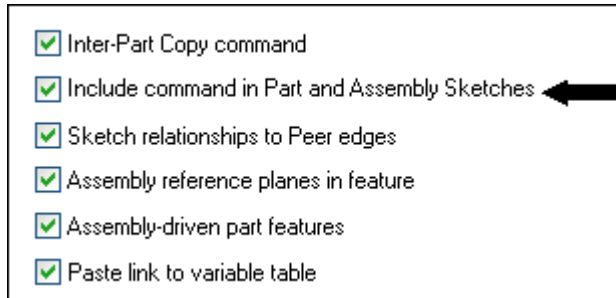
- ▶ Click the Component Image command. Accept the highlighted geometry, and then select the Include command and include the cutouts and edges shown. Click Close Sketch, then Finish to complete the component sketch. Save and close the file.



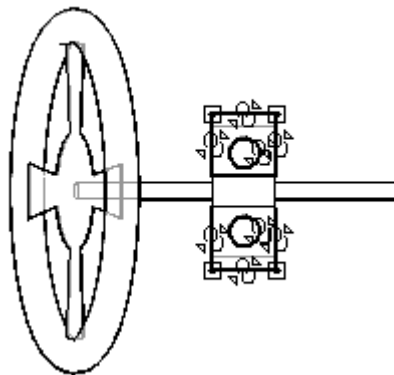
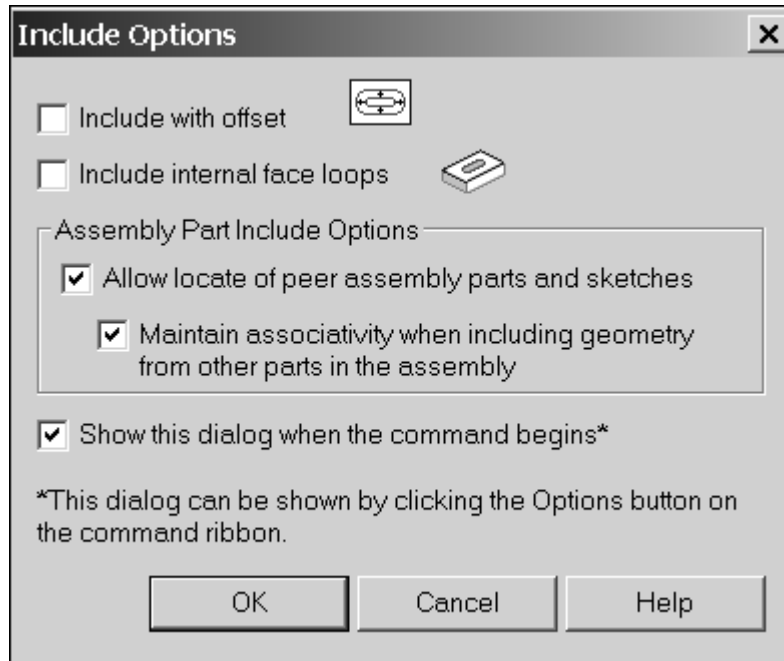
- ▶ In the Parts Library, right click *steering.asm* and click Open in Solid Edge. Activate all the parts in the assembly. Create a component sketch on a plane parallel with the top reference plane. Use the keypoint shown to position the reference plane.



- ▶ Click the Application button. Click Solid Edge Options and then click the Inter-Part tab. Make sure the Include command in part and assembly sketches box is checked.



- ▶ Drag a box to Include all the geometry shown in the sketch. Set the include options as shown.

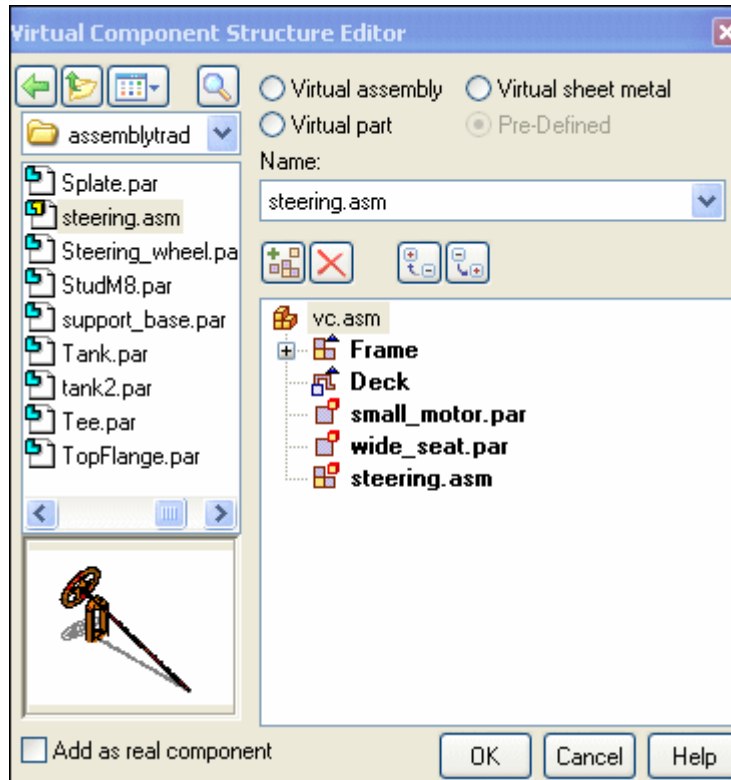


- ▶ Click Close Sketch to exit the sketch, then click Finish.
- ▶ Save and close the file. This returns you to the assembly.

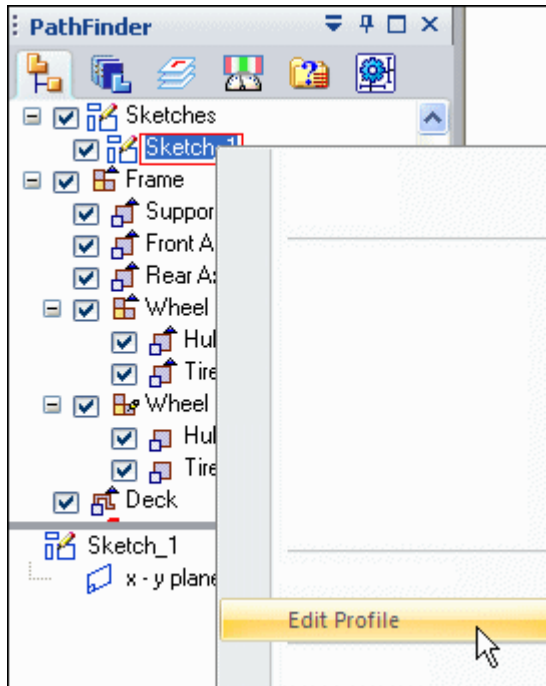


### Position the component sketches in the existing parts as part of the virtual component structure

- ▶ Open the Component Structure Editor and drag the three Solid Edge files where you created component sketches into the assembly, then click OK.



- ▶ The parts with the component sketches will now be positioned in the virtual assembly. On the Assembly PathFinder, right-click *Sketch\_1* and then click Edit Profile.

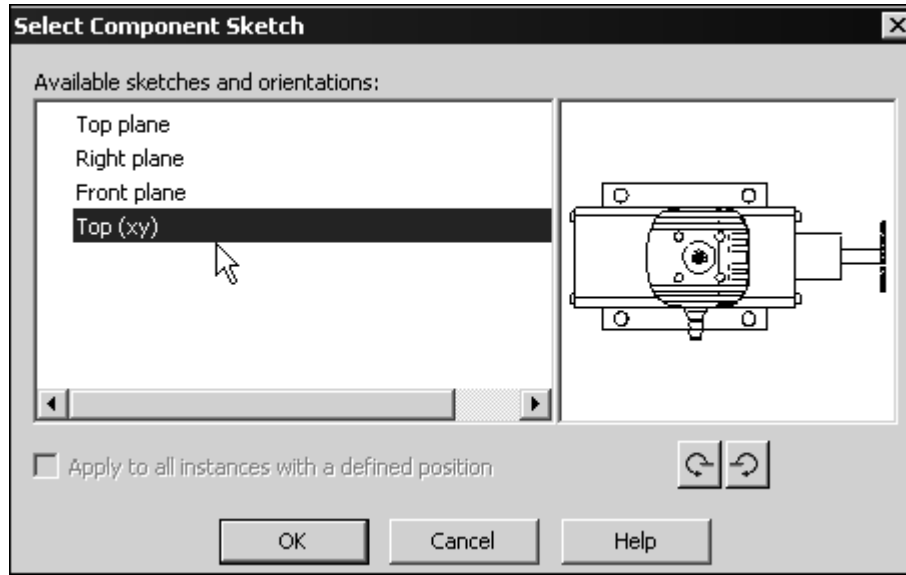


- ▶ On the Layers tab, hide the layers named *Master\_Layout* and *Dimensions*, if they are not already hidden.
- ▶ In Assembly PathFinder, right-click the virtual component *small\_motor.par* and click Position Virtual Component.

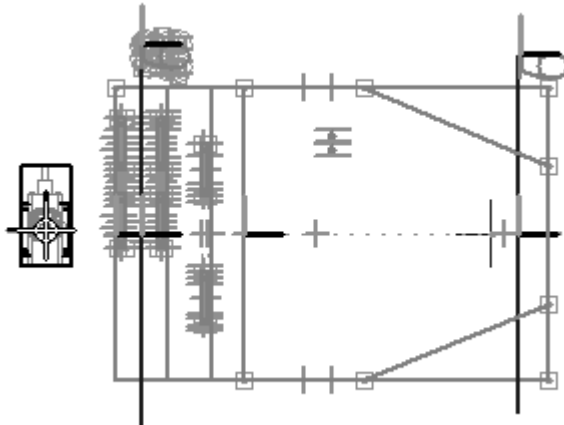
- ▶ Select the plane shown. The component sketch will show in the preview area. Click OK.

**Note**

You will know you have selected the correct plane when the preview shows your component sketch.



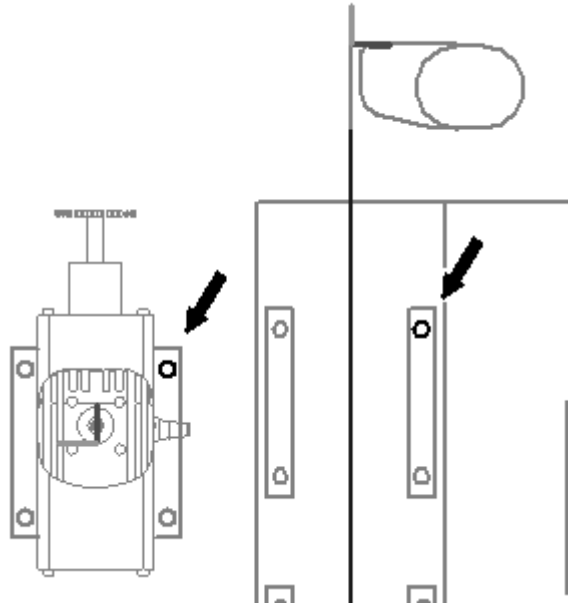
- ▶ Observe the prompts. On the keyboard, press A to rotate the motor into the position shown. Click to the left of the sketch and place the component approximately where shown.



- ▶ Click the Home tab. In the Relate group, turn off Relationship Handles.

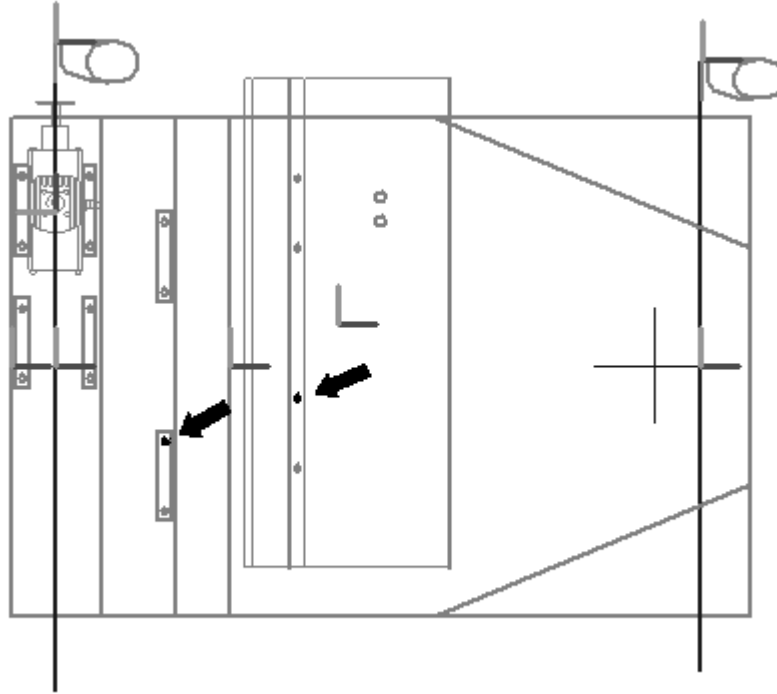


- ▶ In the Relate group, click the Connect command, and connect the center of the circles shown below.

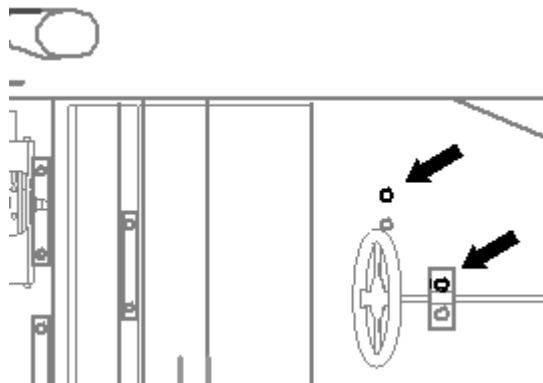


- ▶ Click the select tool. In Assembly PathFinder, right-click the virtual component *wide\_seat.par* and click Position Virtual Component.

- ▶ Click the Select tool. Use the same procedure to position the seat as you used to position the component motor. Use the Connect command to position the seat using the circles shown.



- ▶ Click the Select tool.
- ▶ In Assembly PathFinder, right-click the virtual component *steering.asm* and click Position Virtual Component.
- ▶ Use the same procedure used to position the seat to position the steering wheel. Use the Connect command to position the steering wheel using the circles shown.



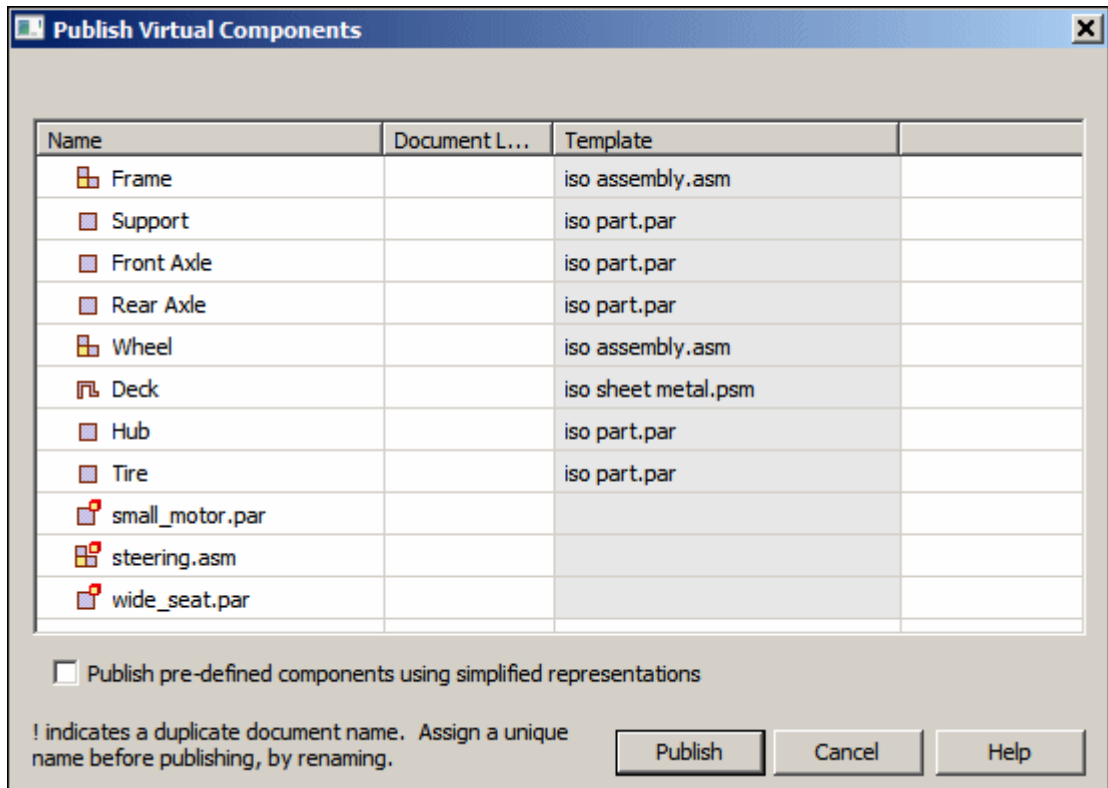
- ▶ Click Close Sketch to exit the sketch. Then click Finish. Save the assembly.

**Publish the virtual assembly**

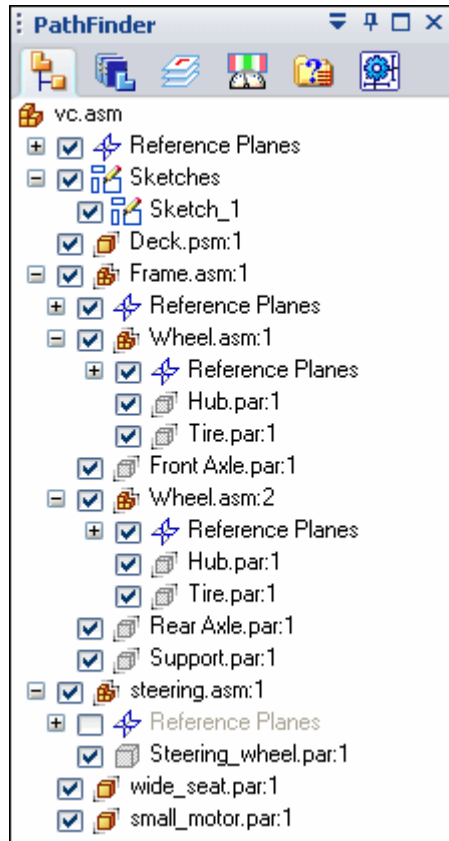
- ▶ On the Home tab, in the Assemble group, click the command Publish Virtual Components. This will create real components from the virtual components.



- ▶ Set the path to correspond to the directory structure you have been using for this activity and click Publish.

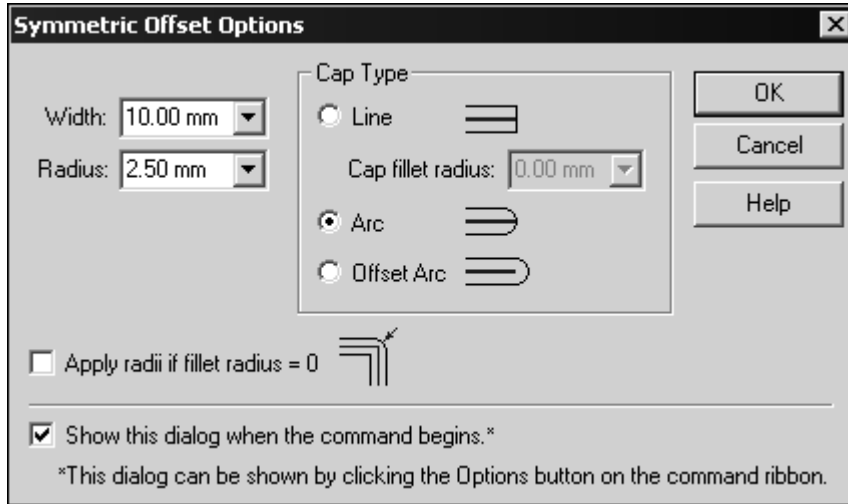


- ▶ Notice that now there are real parts Assembly PathFinder. Activate all the parts.

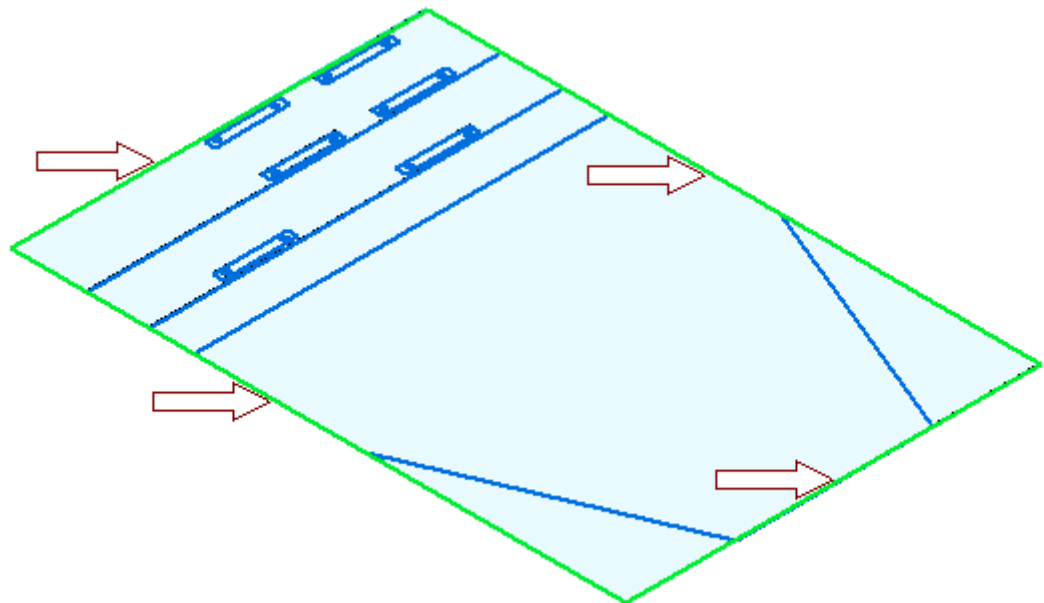


**Create the frame from the sketch geometry**

- ▶ In Assembly PathFinder, right-click *support.par* and then click Open in Solid Edge part.
- ▶ Choose the Home tab® Draw group® Symmetric Offset command, and enter the values shown.

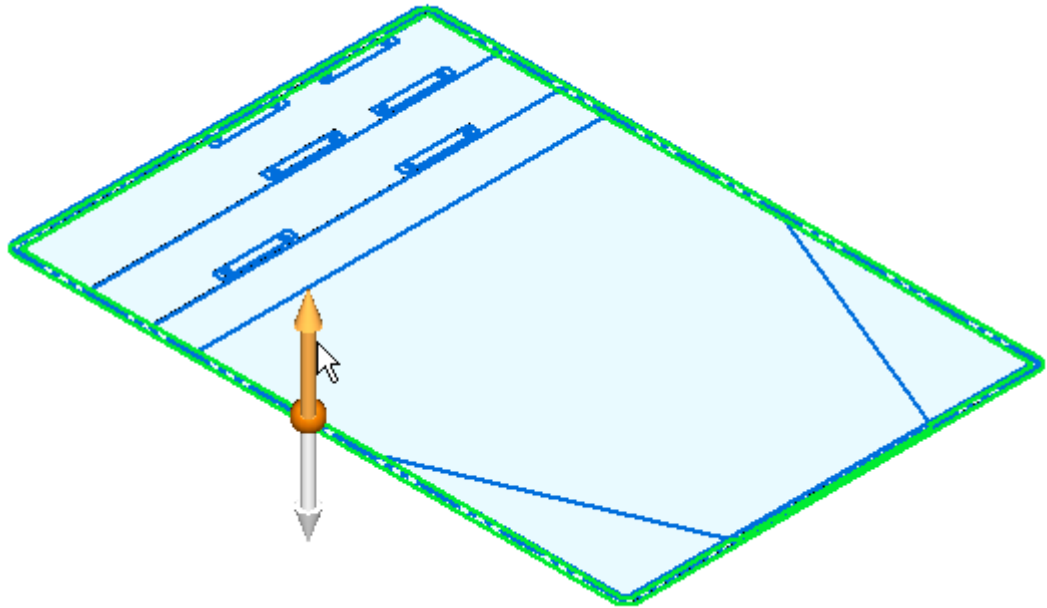


- ▶ Set the type to chain.
- ▶ Select the rectangular perimeter of the sketch as shown below.

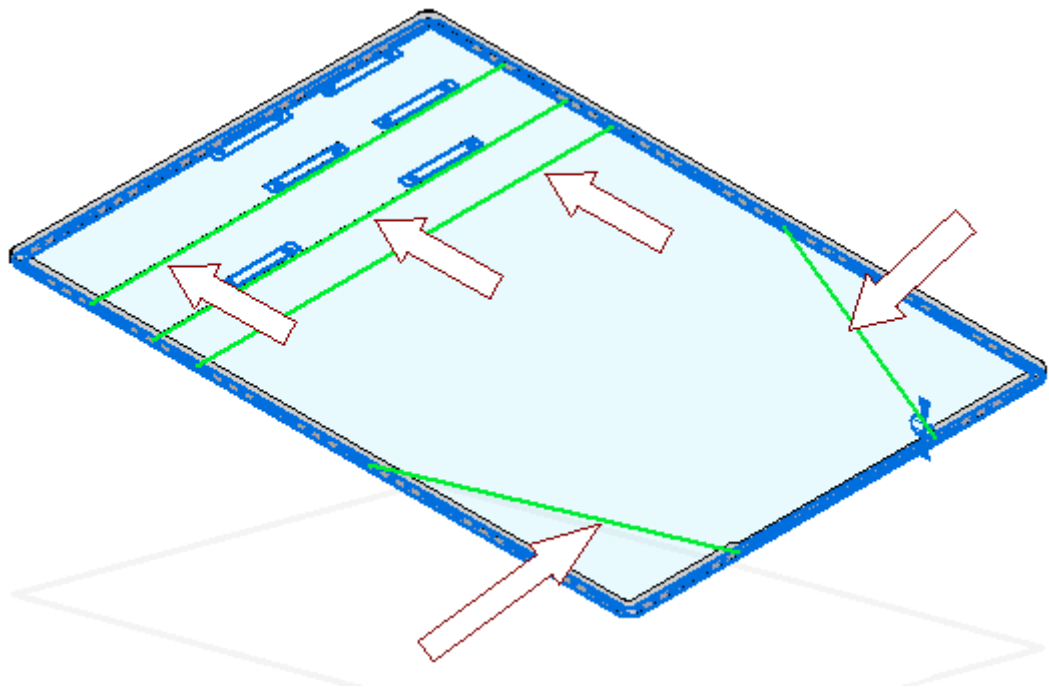




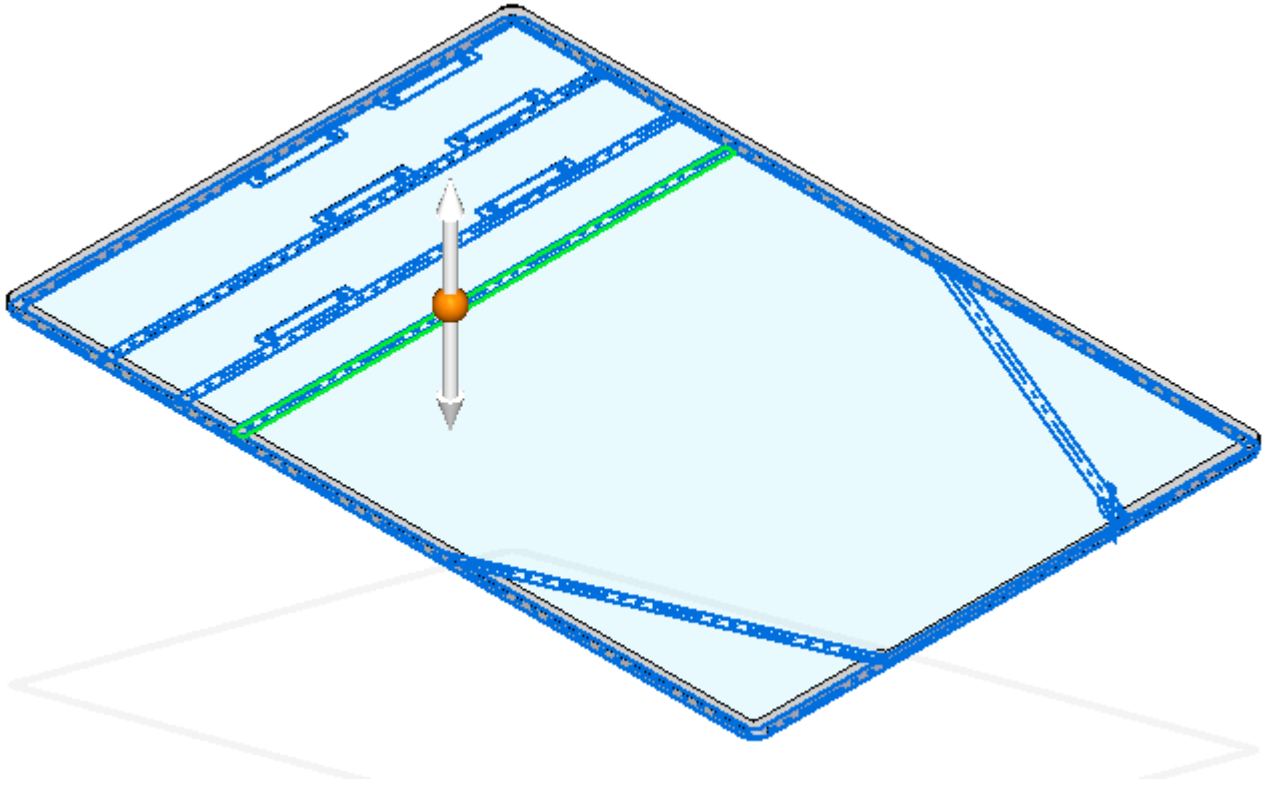
- ▶ Accept the selection.
- ▶ Click the Select tool and use QuickPick to select the region shown. Extrude the selection 8 mm in the +Z direction.



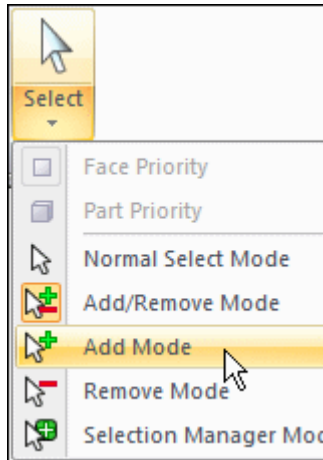
- ▶ Click the Symmetric Offset command. Using the same values as before, select the 5 lines shown below. Accept the selection.



- ▶ Click the Select tool. Select the region shown below.



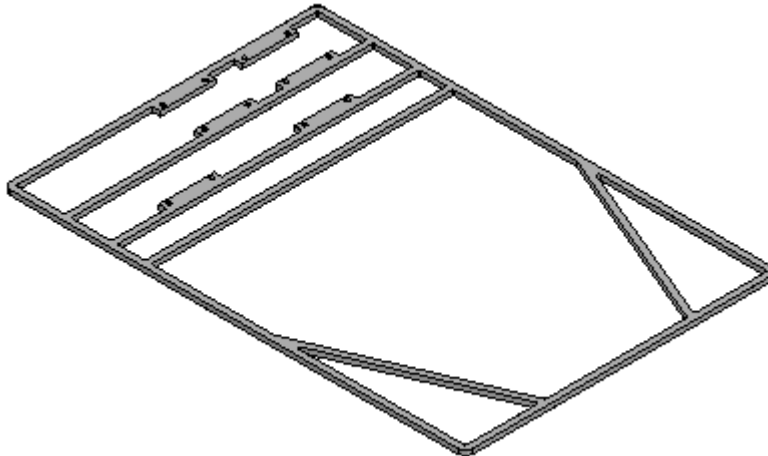
- ▶ Set the Selection mode to add mode as shown below.



- Continue selecting the additional regions required to extrude the remainder of the frame and extrude them 8 mm in the +Z direction.

**Note**

Use QuickPick to select the regions easily.



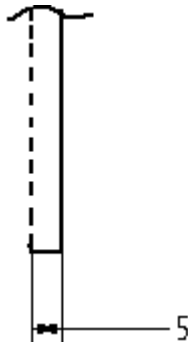
- Save and close the file to return to the assembly.

**Model the front axle**

- ▶ On Assembly PathFinder, select *frame.asm* and then choose Home tab® Relate group® Ground relationship. This will ground the top level of the frame such that additional relationships can be added to the remaining parts without moving the frame.



- ▶ Assembly PathFinder, right-click the part *Front Axle.par* and then click Edit.
- ▶ Use the Coincident Plane command to create a plane parallel to, and 5 mm below, the x-y plane. Use the Include command or the Project to Sketch command to project the line of the front axle onto the plane.
- ▶ Draw a rectangle with a short side that is 5 mm wide and as long as the front axle sketch element. Lock the sketch onto the plane you created.
- ▶ Use the Revolve command to create an axle with radius of 5 mm by selecting the rectangle you created. Use the included line in the front axle sketch to represent the center-line of the 3D axle. Sweep the revolution 360° to create the axle.

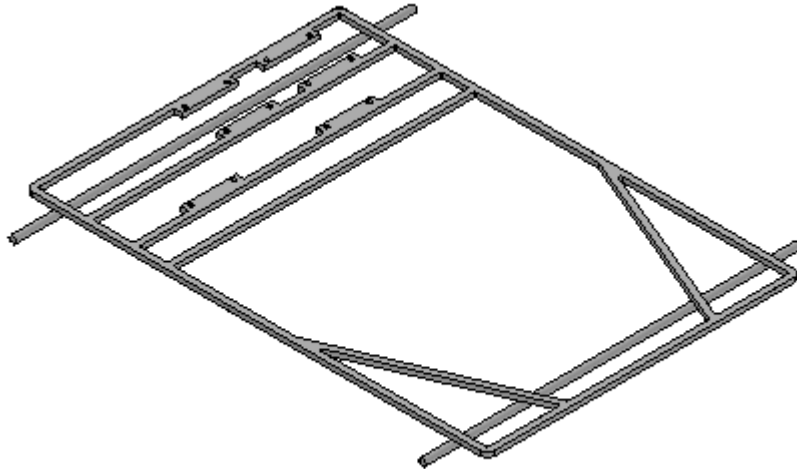


- ▶ After creating the revolved protrusion, click Close and Return.

**Model the rear axle**

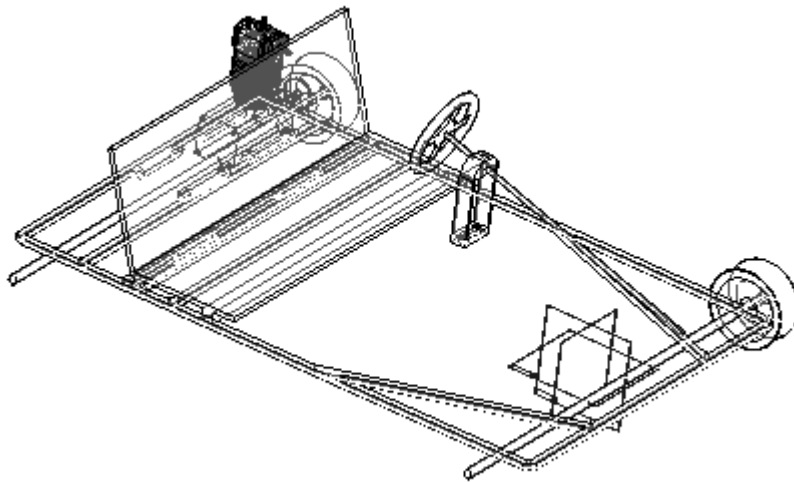
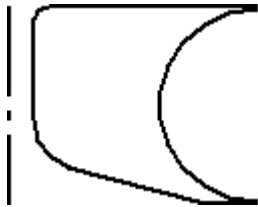
- On the Assembly PathFinder, right-click the part *Rear Axle.par* and then click Edit. Create a revolved protrusion on a parallel plane 5 mm below the x-y plane, as you did with the Front Axle in previous steps. The radius of the protrusion is 5 mm.

Click Close and Return to return to the assembly.



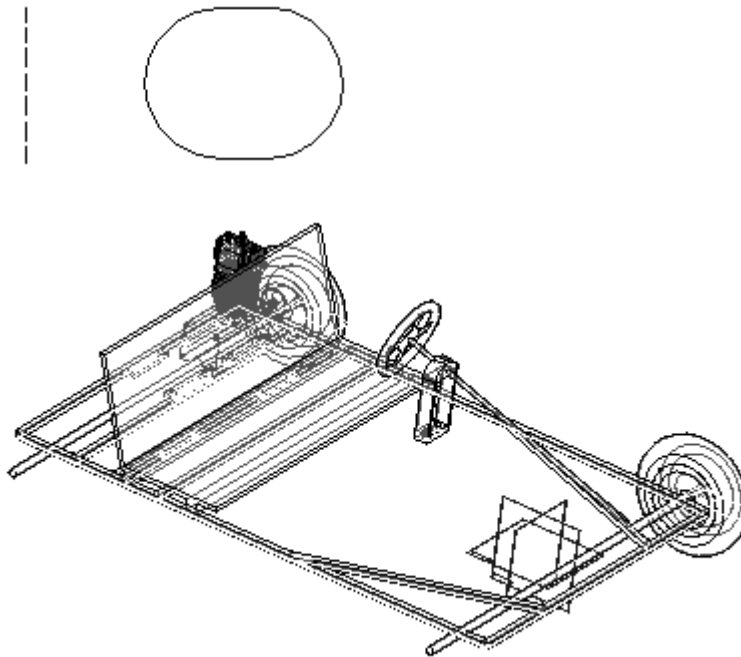
**Model the wheel hub**

- ▶ On the Assembly PathFinder, right-click the part *Hub.par* and click Edit.
- ▶ Create a 360° revolved protrusion on a parallel plane 5 mm below the x-y plane. Define the axis of revolution using the dashed line as shown. Use the include command with the wireframe chain option. Click Finish to complete the revolved protrusion, then click Save and close to return to the assembly.



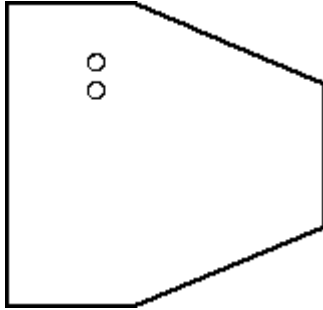
**Model the tire**

- On the Assembly PathFinder, right-click the part *Tire.par* and click Edit.
- Create a 360° revolved protrusion on a parallel plane 5 mm below the x-y plane. Define the axis of revolution using the dashed line as shown. Click finish to complete the revolved protrusion, then click close and return.



**Model the sheet metal deck**

- ▶ On the Assembly PathFinder, right-click the sheet metal part *Deck.psm* and click edit.
- ▶ Select the region shown and create the deck. The circles will be the cutout in the deck. The extent of the tab will be below the profile plane and the sheet metal thickness will be 3.00 mm.



- ▶ Click close and return.

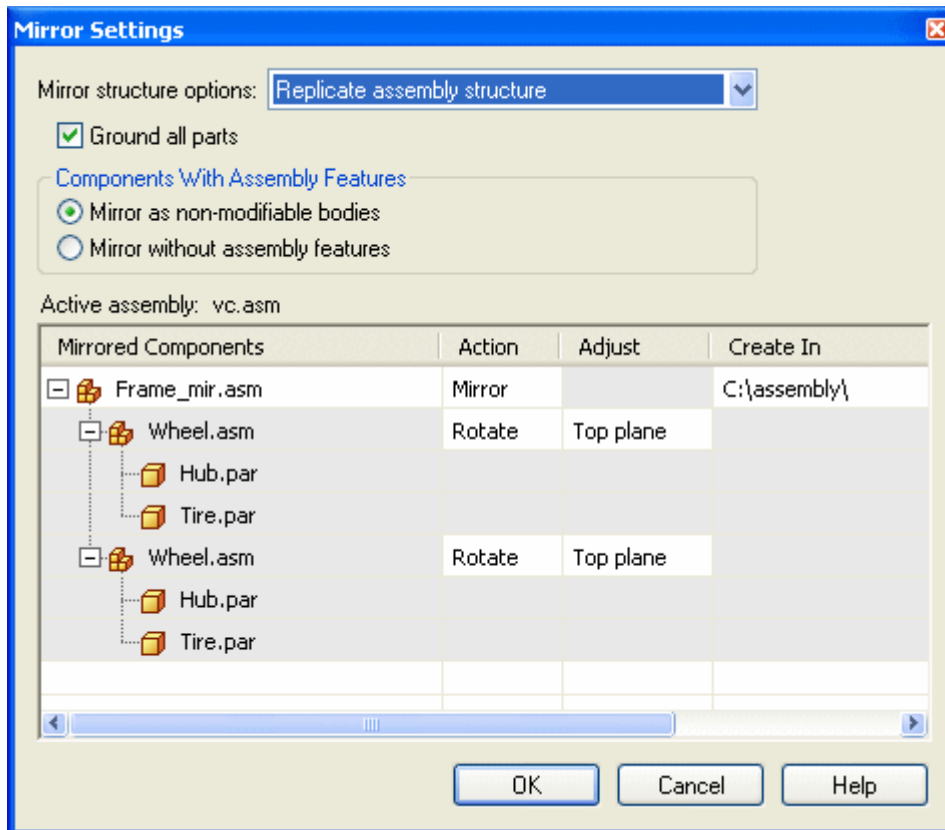


### Mirror the wheels

- ▶ On the Assembly PathFinder show the reference planes.
- ▶ On the Home tab in the Pattern group, click the mirror command. Select the x-z plane to mirror about.



- ▶ In the Assembly PathFinder, select the two assemblies named *Wheel.asm*, and on the command bar, click accept.
- ▶ Set the options shown and then click ok, then click finish.



- ▶ Assembly relationships can be edited to further position and constrain the parts. Save and close this file. This concludes this activity.

**Activity summary**

In this activity you learned how to create a virtual assembly structure using virtual components. Sketch geometry in the assembly file was assigned to virtual part files and once published, the sketch geometry was moved into those files and used to create parts. Existing parts were placed in the virtual structure and positioned using component sketches. This is a top down approach to laying out and modeling complex assemblies.

## **Lesson review**

Answer the following questions:

1. How is assembly sketch geometry assigned to a virtual component?
2. What is the purpose of a component sketch?
3. What is created when virtual components are published?

**Answers**

1. How is assembly sketch geometry assigned to a virtual component?

The geometry can be assigned by editing the profile of a sketch, and right clicking a virtual component and editing the definition. Geometry from the sketch is assigned and oriented in the position it will be when the component is published.

2. What is the purpose of a component sketch?

A component sketch creates sketch geometry in an existing part or assembly needed to position the component in a virtual assembly such that when the virtual assembly is published, the part will be in the correct position relative to the other components in the assembly.

3. What is created when virtual components are published?

Part, sheet metal and assembly files are created from designated templates and if sketch geometry has been assigned to these components, sketches will exist in the newly created documents. The pathfinder structure is the same as it was in the component editor. Existing parts that have been positioned with component sketches are placed in the assembly.

## **Lesson summary**

In this lesson you learned how to create a virtual assembly structure using virtual components. Sketch geometry in the assembly file was assigned to virtual part files and once published, the sketch geometry was moved into those files and used to create parts. Existing parts were placed in the virtual structure and positioned using component sketches. This is a top down approach to laying out and modeling complex assemblies.



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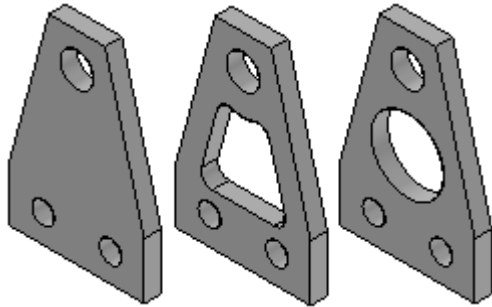
**Lesson**

# *12 Alternate assemblies*

## Define Alternate Component Command

Defines a list of documents which are related to the selected part or subassembly. Alternate components are typically similar in nature, and can be based on documents that were created independently. Defining a list of alternate components makes it easier to find and select valid components when replacing parts in an assembly.

A simple example of an application for an alternate component group is where there are several versions of a part that have different finishing details. The parts can be independently created or members of a family of parts.

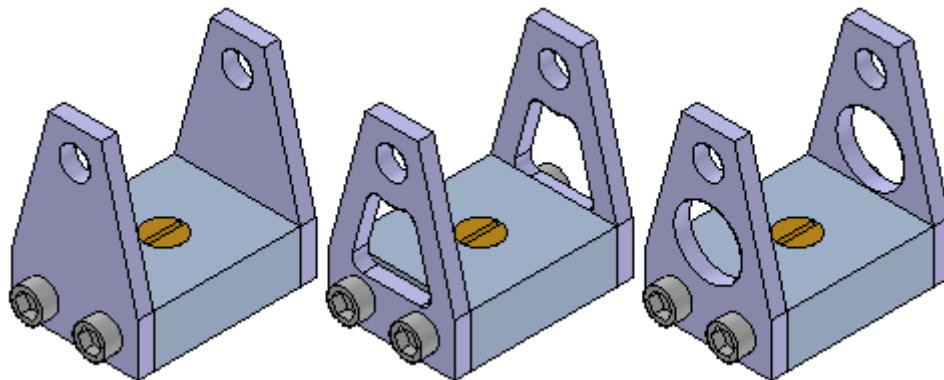


In an assembly where one of the parts is used, you can use the Define Alternate Components command to specify that all three parts are members of the same alternate component group.

### Note

Note: A special symbol is used in Assembly PathFinder to indicate that an assembly component is part of a alternate component group.

You can then use the Replace command to replace the part with one of the alternate component members. When you select a part to replace that is a member of a alternate component group, the Alternate Member dialog box is displayed so you can select another alternate component member as the replacement part.



## Defining an Alternate Component Group

You can select any part or subassembly in the active assembly to define an alternate component group. When working with a nested assembly, you cannot select individual components that are in a subassembly.

Although an alternate component group would typically contain similar parts, you can define an alternate component group that contains different document types



than the original component. For example, you can define an alternate component group that consists of a Part document, a Sheet Metal document, and an Assembly document.

When you select a component in an assembly, then click the Define Alternate Components command, the Define Alternate Components dialog box is displayed so you can browse or search for the documents you want define as alternates for the selected component.

Defining documents as alternate components is a two step process. First, you browse or search for documents that you want to add to the Alternate Candidates list in the left pane of the Define Alternate Components dialog box. Then you can select documents in the Alternate Candidates list and click the Add button to add them to the Alternates List in the right pane of the dialog box.

If the component you select to define the alternate components group is a member of a family of parts or a family of assemblies, all related members from the same master document are added to the Alternate Candidates list automatically.

The assembly document in which you define the alternate components group stores the group member data.

## **Using Alternate Component Groups**

As described earlier, a alternate components group makes it easier to find and select valid components when replacing parts in an assembly. When you select a component to replace in an assembly, and the component is a member of a alternate components group, the Alternate Member dialog box is displayed so you can select another group member as the replacement part.

### **Note**

When you replace an assembly component with an alternate component that is a different document type than the original component, assembly relationships can fail to recompute properly.

If you want to replace the component with a component that is not an alternate components member, you can click the Browse button to display the Replacement Part dialog box to select another component.

If you create alternate component groups that are used frequently in many assemblies, you should consider creating an assembly that contains only the master alternate component and define the alternate components group in this document. This assembly document then becomes the master document for the alternate components group definition.

To place one of the group members in an assembly later, you can place the assembly document instead. If you also set the Disperse This Assembly During Place Part option on the Assembly tab of the Options dialog box for this assembly, when you place the assembly into another assembly, it will be placed as a part that contains the alternate components group data.

This workflow can be useful when you know a part will be replaced later by another part. For example, you may create a prototype version of a part that you know will later be replaced by a production version of the part that will be created independently in a separate document.

### **Alternate Components and Alternate Assemblies**

You can use an alternate component to replace a component in an alternate assembly that is a family of assemblies, but not an alternate position assembly. For example, you can use the Replace command to replace a part in a family of assembly member with an alternate component, rather than use the Exclude Occurrence functionality on the Alternate Assembly tab. Although this approach can be quicker than using the Exclude Occurrence functionality, when you use Revision Manager to copy or move the assembly data set to a new location, any members of the alternate components group that are not used in a family of assembly member will not be included in the data set.

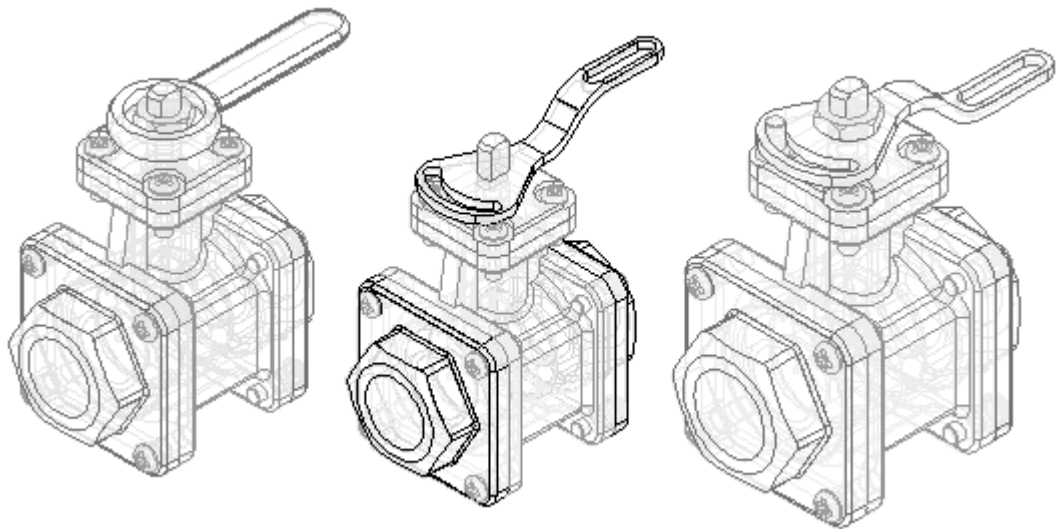
For more information on working with alternate assemblies, see the *Alternate Assemblies Help* topic.

### **Comparing Alternate Components and Family of Parts**

Alternate Component groups are similar to a family of parts, but provide additional flexibility because you can define an alternate components group using independently created documents and different document types.

## **Activity: Creating an alternate assembly**

This activity guides you through the process creating alternate versions of an assembly.



## **Activity: Creating an alternate assembly**

### **Objectives**

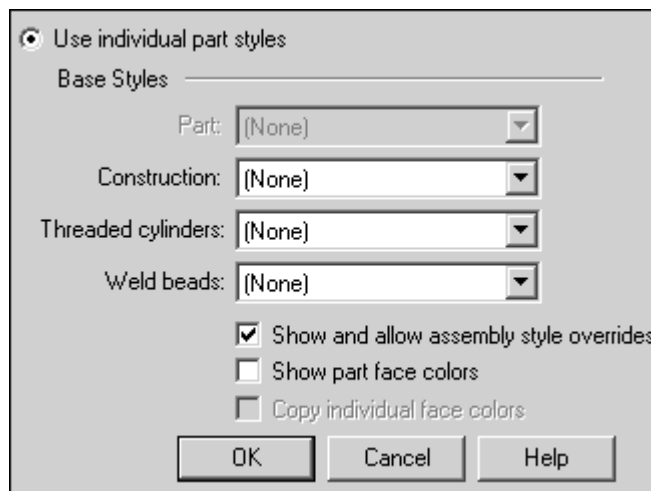
When you complete this activity, you will be able to:

- Convert a normal assembly into an alternate assembly.
- Add members to an alternate assembly.
- Define unique variable values for individual members.
- Replace occurrences for a member.
- Exclude occurrences for a member.
- Exclude and reapply assembly relationships for a member.

### Create an alternate assembly for a valve

Create an alternate assembly for a valve. The valve in the existing assembly is open. Create an alternate assembly with the valve closed.

- ▶ Open *Alternate.asm* located in the folder where you loaded the parts files needed for this activity. Open the assembly with all the parts active.
- ▶ On the View tab, click Color Manager, and ensure the following options are set.
- ▶ Set the use individual part styles option.
- ▶ Set the show and allow assembly style overrides option.
- ▶ Clear the Show part face colors option.



- ▶ Click OK.

#### Note

You create a family of assemblies by converting a normal assembly document into an alternate assemblies document. You can only convert the assembly document once, and you must specify whether you want to convert it into a family of assemblies or an alternate position assembly. In this activity, you convert the assembly into a family of assemblies.

- ▶ In PathFinder, click the Alternate Assemblies tab.



- ▶ On the Alternate Assemblies tab, click the New button.



- ▶ In the Alternate Assemblies dialog box, do the following:
- ▶ Ensure that the Create a Family of Assemblies option is selected.

- ▶ In the Name current member box, type Default Type 1 Open and press the Tab key.
- ▶ In the Name new member\* box, type Type 1 Closed.
- ▶ Click OK.



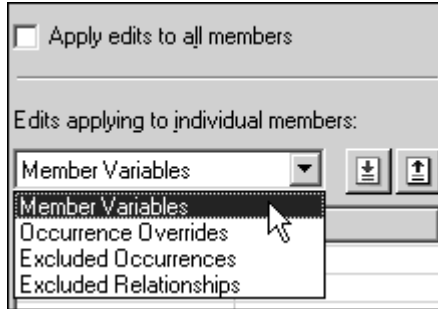
**Note**

On the Alternate Assemblies tab, notice that the Type 1 Closed member is the active member. When you create a new family of assemblies, the name you type in the Name new member box becomes the active member.

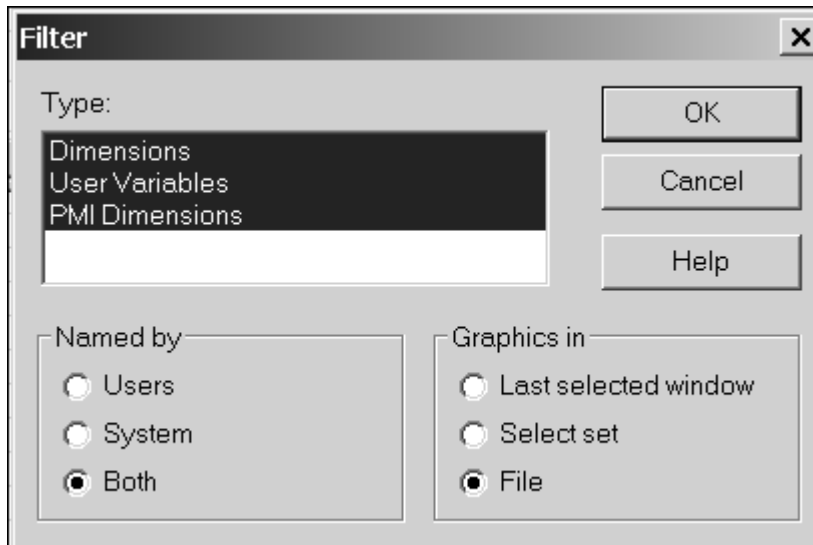
### Modify the Type 1 Closed family member

Modify the Type 1 Closed family member by overriding the value of the angular relationship that controls the position of the valve subassembly and handle. First, add the variable to the member variables list.

- On the Alternate Assemblies tab, select the Member Variables option.



- On the Tools tab, click Variables to display the Variable Table. Click the Filter button and set the values as shown, and then click OK.



- Click the ValveAngle variable, as shown.

Type	Name	Value	Rule	Formu
Dim	V174	0.00 mm		
Dim	V206	0.00 mm		
Dim	V208	5.00 mm		
Dim	V210	0.00 mm		
Dim	V212	0.00 mm		
Dim	V315	0.00 mm		
Dim	ValveAngle	0.00 deg		
Dim	V364	0.00 mm		

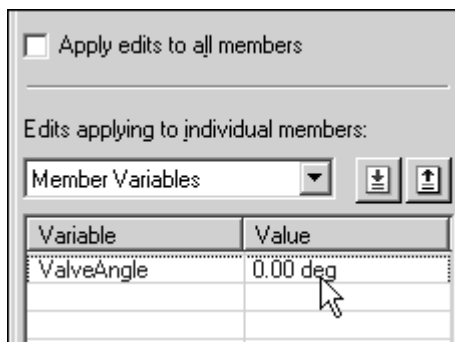
- On the Alternate Assemblies tab, click the Add Variables button.



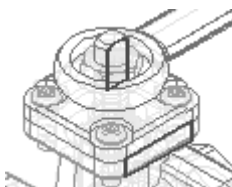
**Note**

The variable is added to the Member Variables list. When you add variables to the Member Variables list, you can no longer edit them within the Variable Table. You must edit them in the member variables list on the Alternate Assemblies tab.

- Close the Variable Table.
- In the Member Variables list, position the cursor over the variable as shown, but do not click yet.

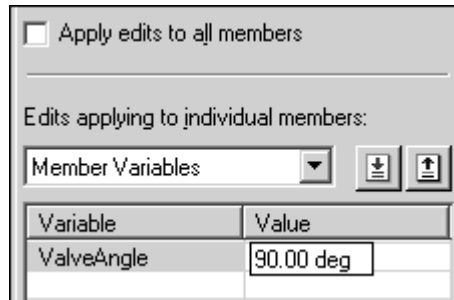


- In the graphics window, notice the faces used to define the angle relationship highlight as shown in the following illustration.

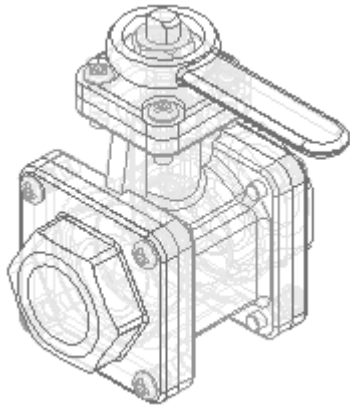




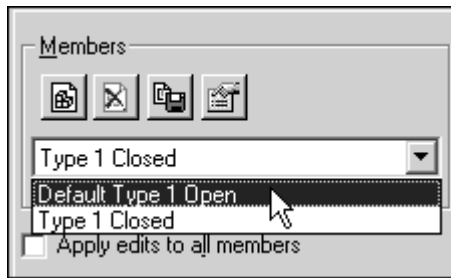
- ▶ Position the cursor over the variable value as shown, and then double click the left mouse button.
- ▶ Type 90, and then press the Enter key.



- ▶ In the graphics window, notice that the handle rotated 90° clockwise. In addition to the handle, the valve subassembly also rotated.



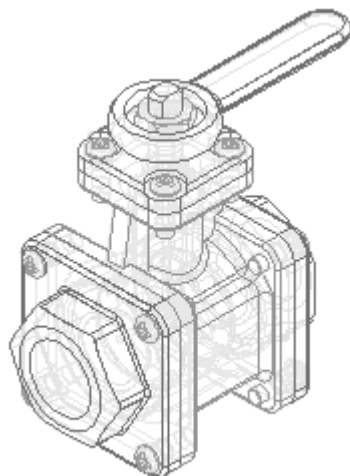
- ▶ On the Alternate Assemblies tab, select the Default Type 1 Open member as shown.



**Note**

For the Default Type 1 Open member, the ValveAngle variable was also added to the Member Variables list, but the valve handle is still in the open position. The variable value you changed only affected the Type 1 Closed family member.

When you add a variable to the Member Variables list for one member, the variable is added to the Member Variables list for all members that contain the variable, but each member can have a different value assigned to the variable.



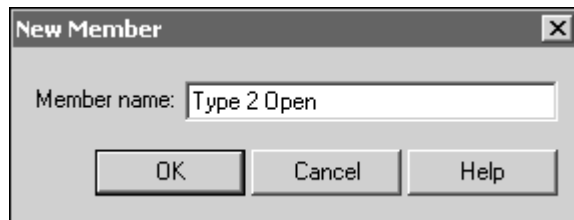
### Create the member Type 2 Open

Create another family member, as shown. For this family member, delete the original handle and place a new handle. Use the replace part command to replace the *TopFlange01.asm* subassembly with a different subassembly.

- ▶ On the Alternate Assemblies tab, click the New button.

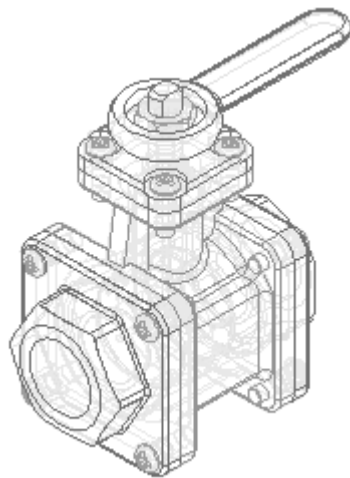


- ▶ In the New Member dialog box, type Type 2 Open, and then click OK.



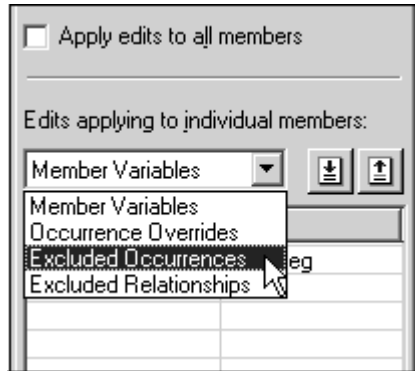
#### Note

The new family member has the same characteristics as Default Type 1 Open. When you create a new family member, it has the same characteristics as the family member that is active when you create the new member. In this case, the active member was Default Type 1 Open.



- ▶ On the Alternate Assemblies tab, do the following as shown.
- ▶ Select the Excluded Occurrences option.

- ▶ Ensure that the Apply edits to all members check box is cleared.



**Note**

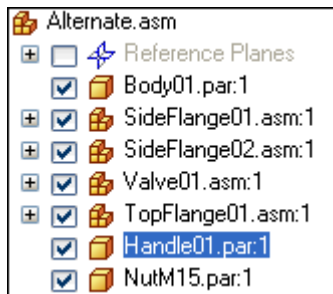
When you exclude occurrences, you can exclude them for all family members or for only the active member. In this case, you want to exclude the handle for only the active member.

When you exclude an occurrence for all family members, the occurrence is physically deleted from the assembly file, and it is not added to the excluded occurrences list.

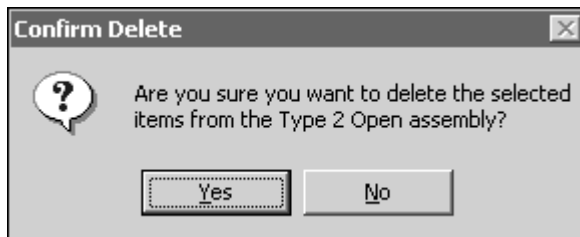
- ▶ Click the Assembly PathFinder tab.



- ▶ In Assembly PathFinder, select *Handle01.par* and delete it.

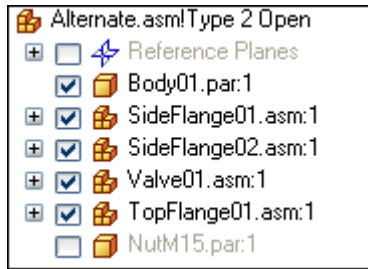


- ▶ A dialog box displays to confirm that you want to exclude the part for the active family member.



- ▶ In the Confirm Delete dialog box, click Yes.

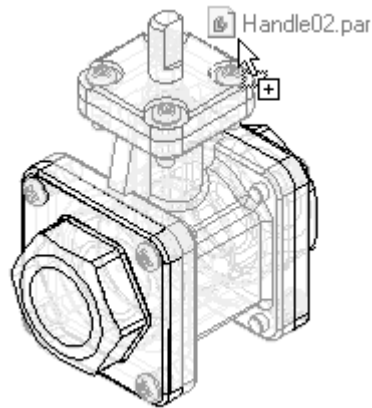
- ▶ To make it easier to position the new handle, hide the nut part.



- ▶ In PathFinder, click the Parts Library tab.

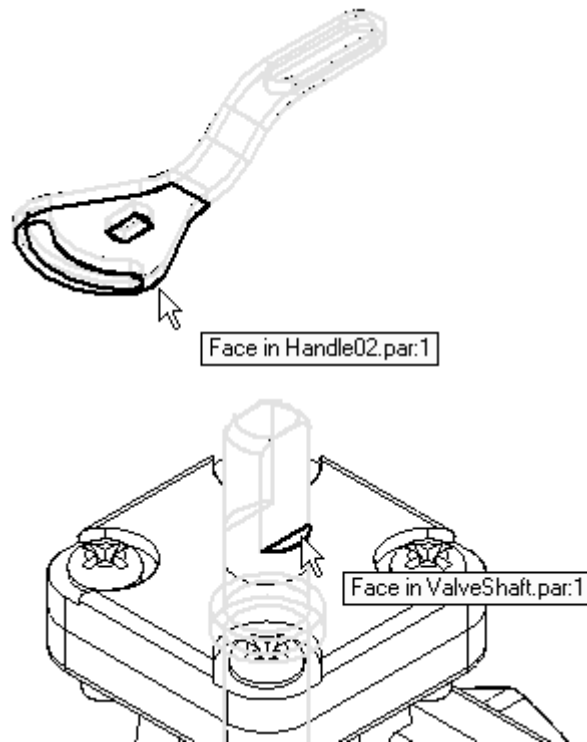


- ▶ On the Parts Library tab, click the arrow on the right side of the Look In list. Browse to the folder where the activity files are located.
- ▶ Drag *Handle02.par* into the assembly window.

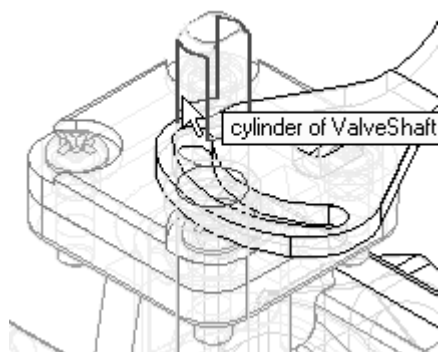
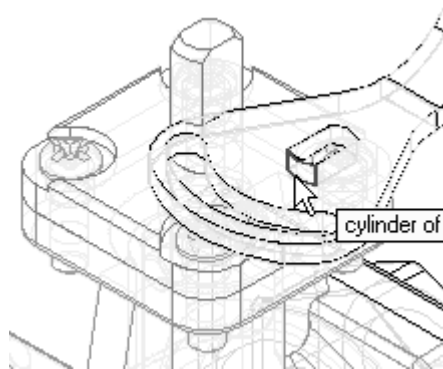


- ▶ Position the handle using FlashFit as directed in the following steps.

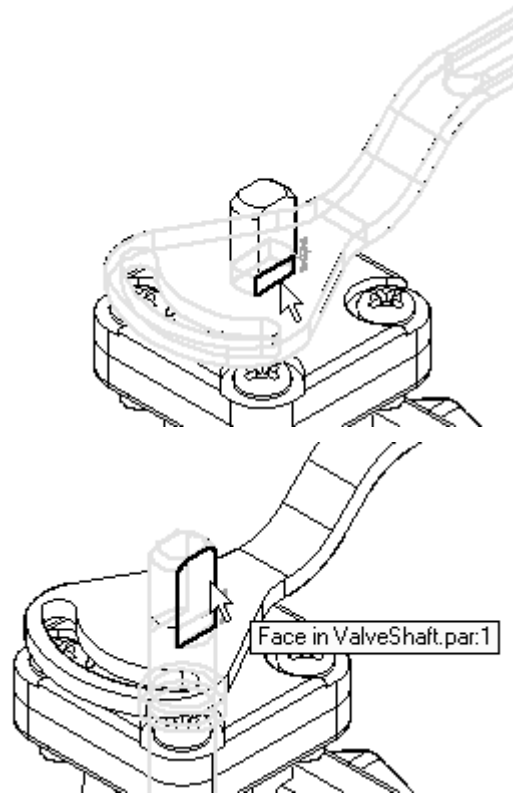
- ▶ Mate the two planar faces shown.



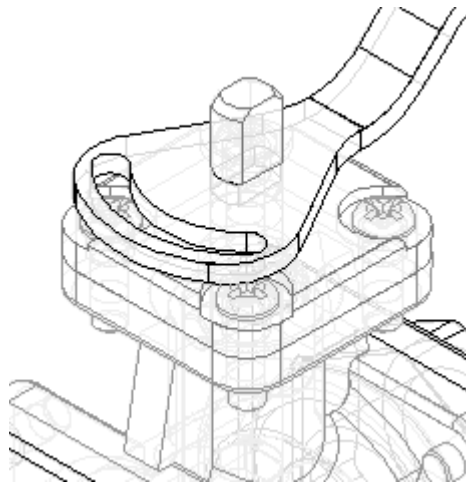
- ▶ Axial align the curved faces shown.



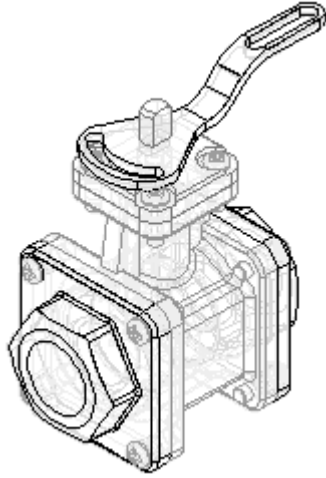
- Mate the two planar faces shown.



The handle is fully positioned in the assembly.



- Save the assembly.





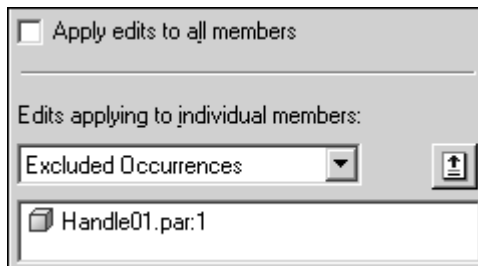
**Observe results**

Make some observations of the alternate assemblies created so far.

- ▶ In PathFinder, click the Alternate Assemblies tab.



- ▶ In the Excluded Occurrences list for the Type 2 Open family member, notice that *Handle01.par* has been added to the list.



- ▶ On the Alternate Assemblies tab, activate the member named Default Type 1 Open.



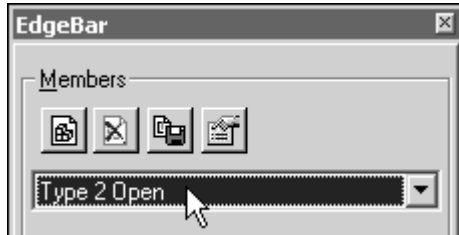
**Note**

For this family member, *Handle02.par* is in the excluded occurrences list. Also notice that the assembly display updated when you activated this family member.

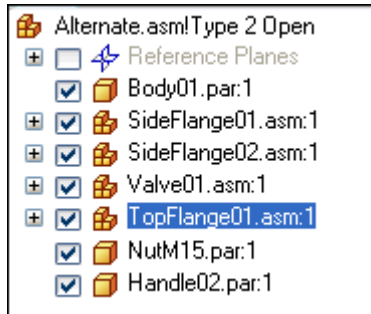
**Finish defining the Type 2 Open member**

Finish defining the characteristics for member Type 2 Open by replacing a subassembly.

- ▶ On the Alternate Assemblies tab, activate the member named Type 2 Open.



- ▶ Click the Assembly PathFinder tab.
- ▶ In Assembly PathFinder, select the entry *TopFlange01.asm*.



- ▶ Click the Home tab. In the Modify group, click the Replace Part command.

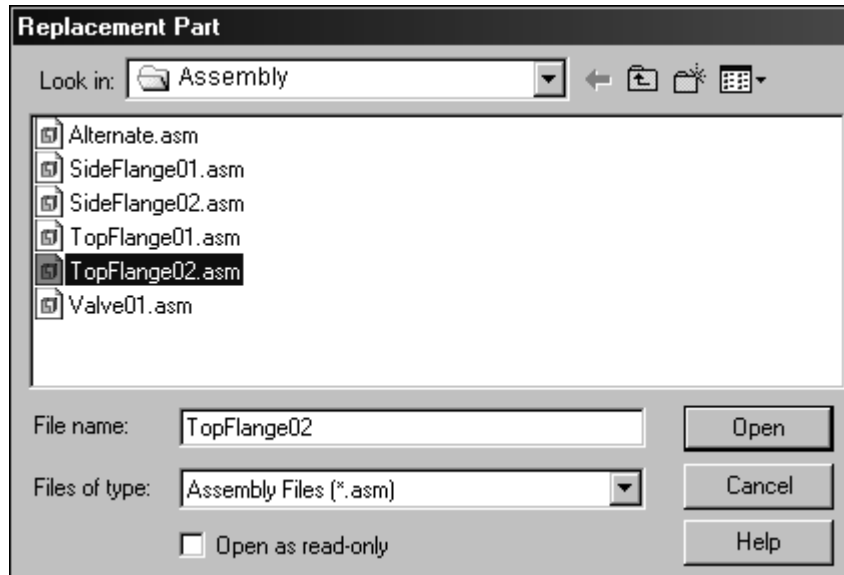


**Note**

You can use the replace Part command to replace a part or subassembly within the assembly. When replacing a subassembly, you must select it within Assembly PathFinder.

- ▶ In the Replacement Part dialog box, do the following:
- ▶ Set the Look In location to the folder where the activity files are located.
- ▶ Set the Files of type option to Assembly Files (\*.asm).
- ▶ Select the assembly file *TopFlange02.asm*.

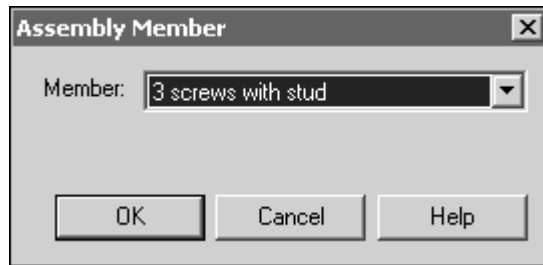
- ▶ Click the Open button.



**Note**

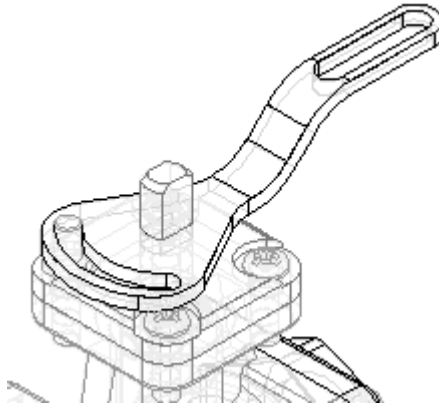
Since the replacement subassembly is also a family of assemblies, you must specify which family member you want to use as the replacement.

- ▶ In the Assembly Member dialog box, select the member named 3 screws with stud, and then click OK.



**Note**

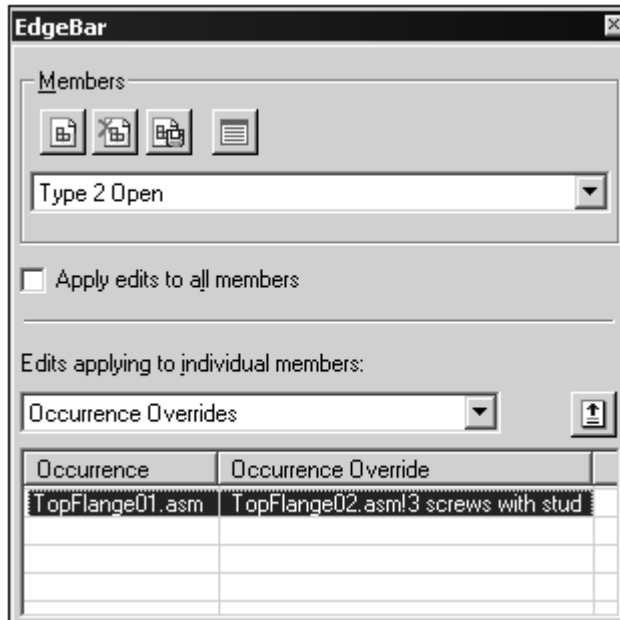
In the assembly window, notice the replacement subassembly does not have four Phillips-head screws. In place of one of the screws is a stud that serves to limit the rotation of the new handle. Also notice in Assembly PathFinder, the member name is appended to the filename.



- ▶ In PathFinder, click the alternate assemblies tab.



- ▶ On the Alternate Assemblies tab, select the Occurrence Overrides option.



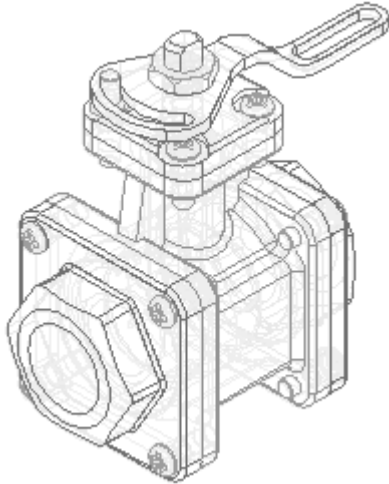
**Note**

Notice for this member that *TopFlange01.asm* was overridden by *TopFlange02.asm!3 screws with stud*.

When you replace a part or subassembly, the original occurrence is added to the left column in the occurrence overrides list, and the replacement occurrence is added to the right column in the occurrence overrides list.

### Create a new family member

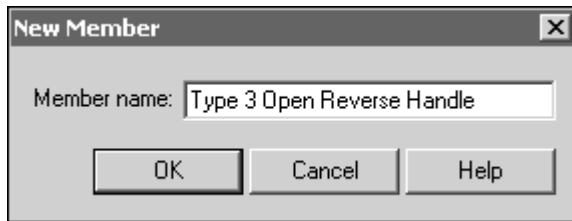
Create another family member as shown. For this family member, change the position of *Handle02.par*. This family member illustrates how you can exclude relationships for a part. Excluding relationships for a part allows you to add new relationships that change the position of the part.



- ▶ On the Alternate Assemblies tab, click the New button.



- ▶ In the New Member dialog box, type Type 3 Open Reverse Handle, and then click OK.



#### Note

The new member you created is active, and it has the characteristics of the Type 2 Open member.

- ▶ On the Alternate Assemblies tab, do the following as shown.
- ▶ Select the Excluded Relationships option.

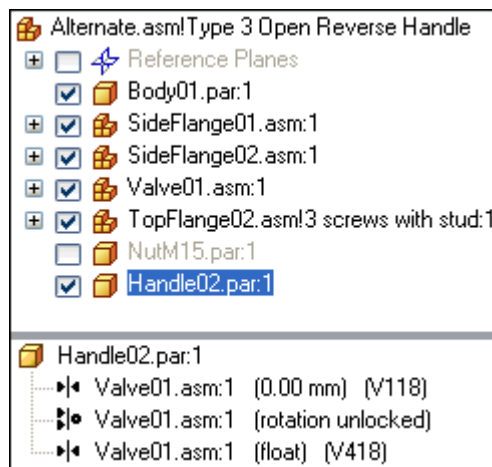
- ▶ Ensure that the Apply edits to all members check box is cleared.

**Note**

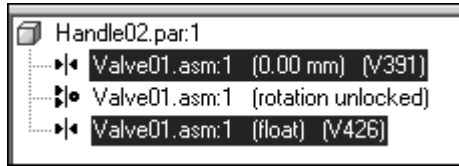
When you exclude relationships, you can exclude them for all family members or for only the active member. In this case, you want to exclude the relationships for only the active member.



- ▶ Click the Assembly PathFinder tab.
- ▶ In the top pane of Assembly PathFinder, select *Handle02.par* as shown.

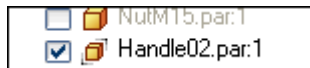


- ▶ In the bottom pane of Assembly PathFinder, delete the two relationships shown.



**Note**

Notice that the symbol adjacent to *Handle02.par* changes to indicate it is no longer fully positioned.

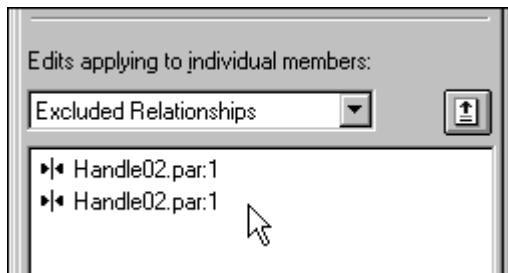


- ▶ In PathFinder, click the Alternate Assemblies tab.



**Note**

Notice that the relationships you excluded were added to the Excluded Relationships list for the family member named Type 3 Open Reverse Handle.

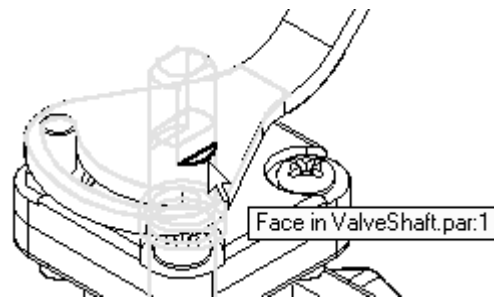
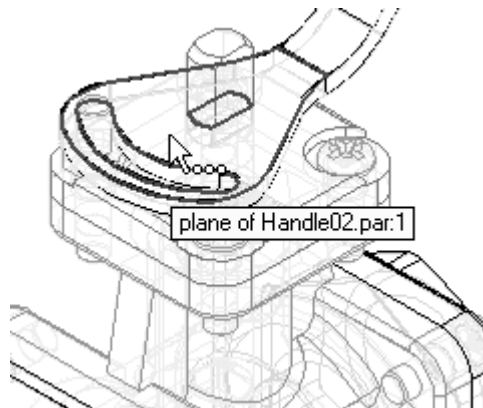


- ▶ Click the Assembly PathFinder tab.
- ▶ In the top pane of Assembly PathFinder, select *Handle02.par*.
- ▶ On the command bar, click the Edit Definition button.

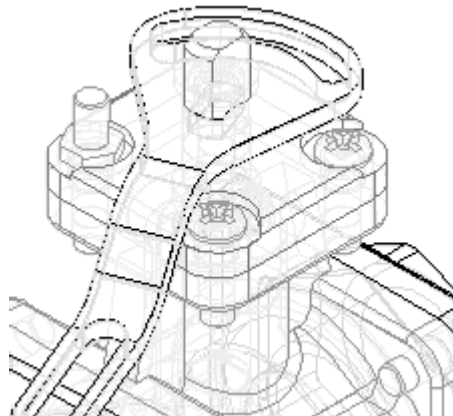




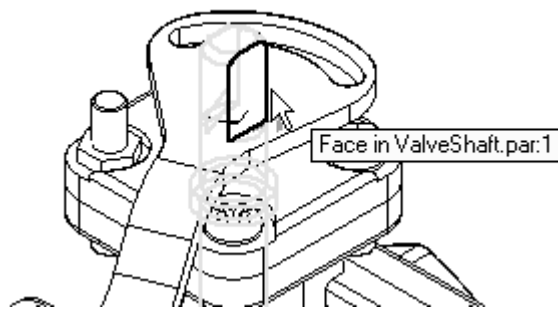
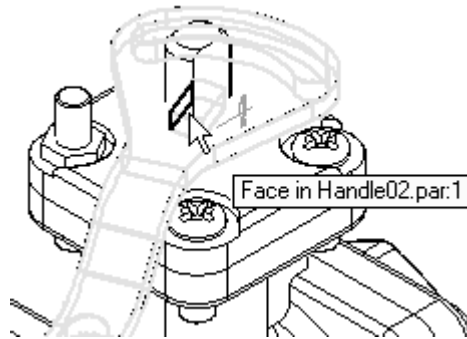
- ▶ Establish a Mate relationship between the faces shown.



- ▶ The handle is mated to the shaft as shown.



- ▶ Continue using FlashFit to mate the faces shown.



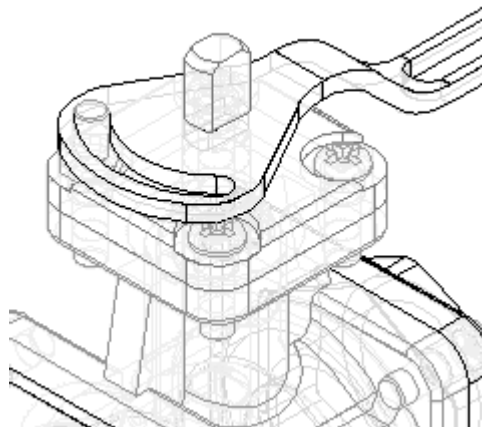
- ▶ Click the Offset Type button, and then click Floating Offset.



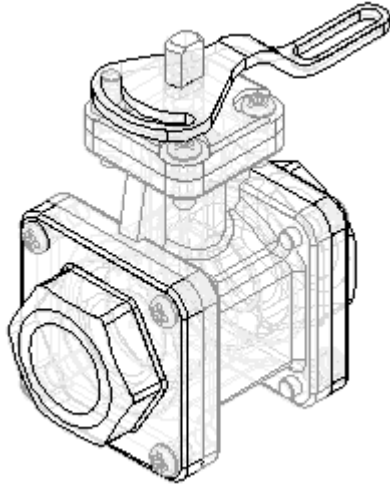
**Note**

This setting allows the faces to take on whatever offset value is appropriate to satisfy the other relationships that position the two parts.

The handle is fully positioned in the assembly.



- Fit the assembly and then save the assembly.



- In Assembly PathFinder, show *NutM15.par*.

**Observe results**

Make some observations of the alternate assemblies created so far.

- ▶ In PathFinder, click the Alternate Assemblies tab.

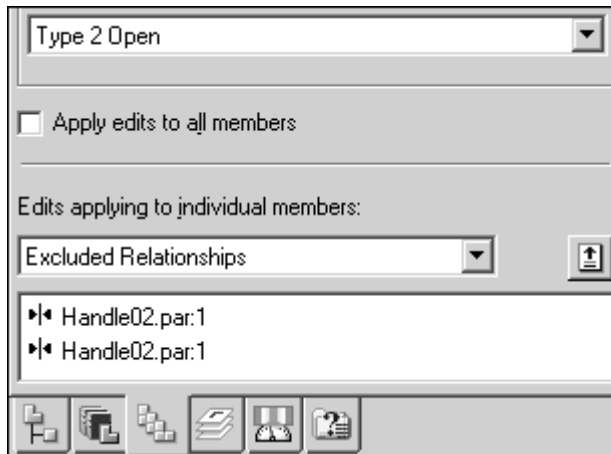


- ▶ On the Alternate Assemblies tab, activate the member named Type 2 Open.

**Note**

There are two relationships on the excluded relationships list for this family member. These relationships were added to the list automatically when you repositioned the handle for the member named Type 3 Open Reverse Handle. When the Apply edits to all members option is cleared, the relationships you add to position a part for one member are automatically excluded from the other members that contain the same part.

Also notice that for this member, *NutM15.par* is still hidden. When you change the display of a part for one family member, the display of the same part in other family members is not changed.



**Create family member named Type 3 Open Reverse Handle.**

Create a new assembly document for the family member named Type 3 Open Reverse Handle.

- ▶ On the Alternate Assemblies tab, activate the member named Type 3 Open Reverse Handle.
- ▶ On the Alternate Assemblies tab, click the save member as button.

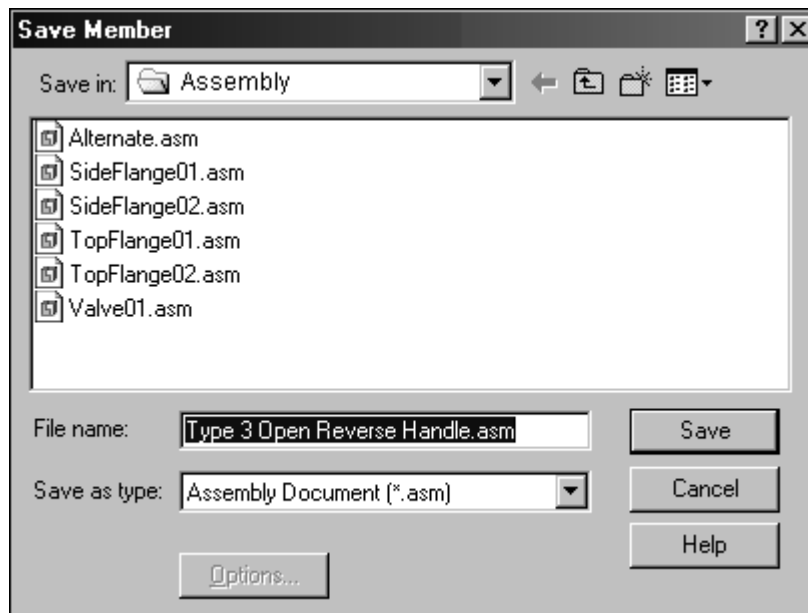


- ▶ In the Save Member dialog box, save the active member (using the default name) to the folder where the activity files are located.

If a dialog box displays and asks if you want to save the current assembly first, click the Yes button.

**Note**

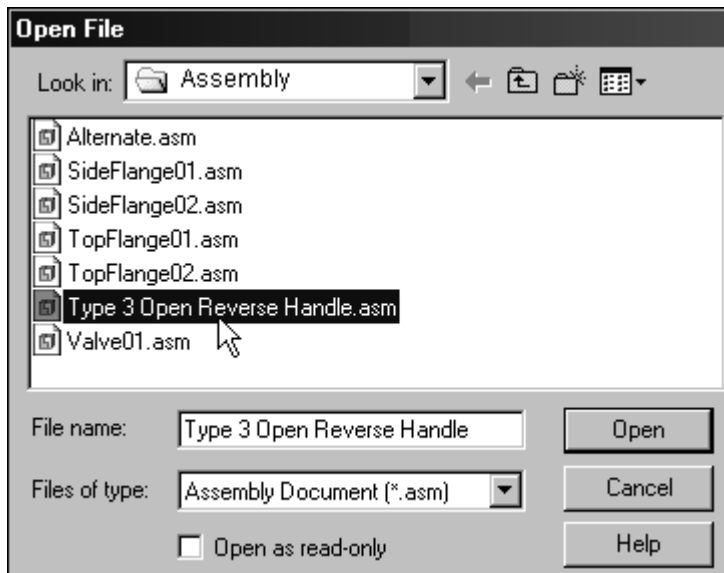
Remember the name and location, because you will be opening this document in the next step.



- ▶ Click the Application button. Click the Open button.



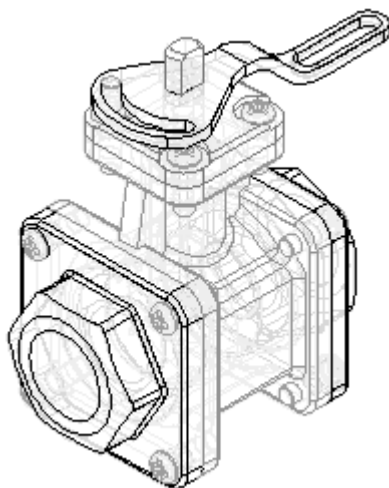
- ▶ In the Open File dialog box, browse to the location where you saved the new file, and then open it with all the parts active.



**Note**

For the new assembly, no family members are listed on the Alternate Assembly tab.

When you save a family member as a separate assembly, it is saved as a normal assembly document. Saving family members as separate assembly documents is useful because some downstream applications do not recognize assembly documents that have been converted to alternate assemblies. For example, Simply Motion only allows you to open normal assembly documents.

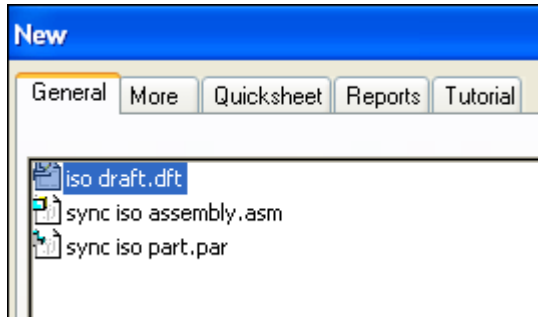


- ▶ Save the assembly.
- ▶ Close and save both assembly documents.

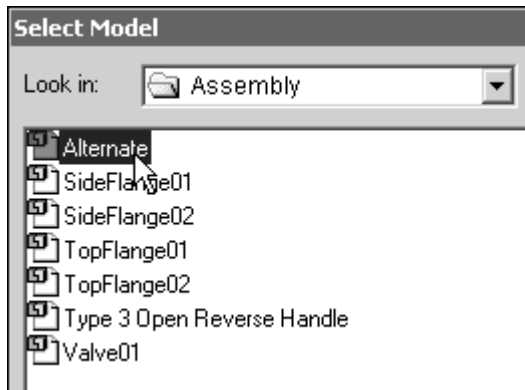
### Draft family of assembly members

Create a new draft document and place a drawing view of one of the family of assembly members.

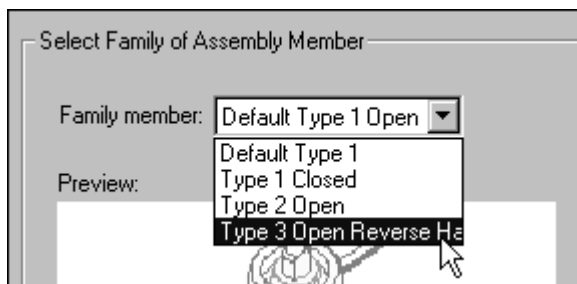
- ▶ Click the Application button. Click the New button.
- ▶ In the New dialog box, click *iso draft.dft*, and then click OK.



- ▶ Choose the Home tab® Drawing Views group® View Wizard command
- ▶ In the Select Model dialog box, select *Alternate.asm*, and then click Open.

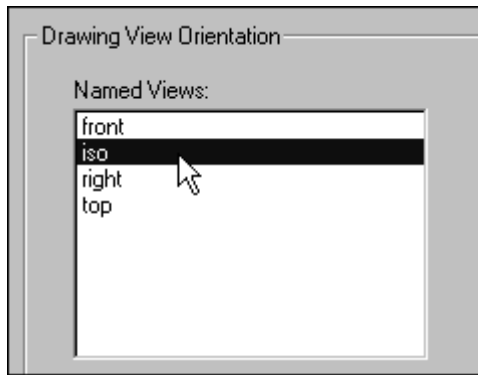


- ▶ The Family of Assembly Member page of the Drawing View Wizard displays, so you can specify which family member you want. From the Family member list, select the Type 3 Open Reverse Handle option, and then click the Next button.

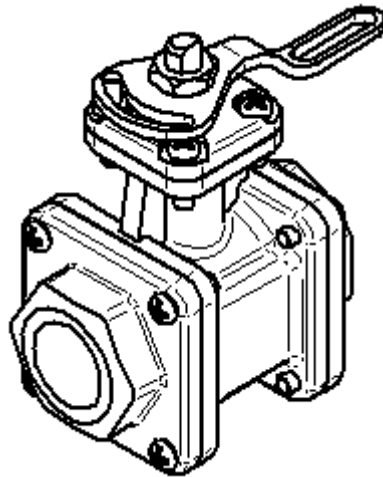


- ▶ On the Assembly Drawing View Options page, click the Next button.

- ▶ On the Drawing View Orientation page, select the iso option, and then click Finish.



- ▶ Position the drawing view on the drawing sheet, and then click to place the drawing view.



- ▶ Save and close the draft document. This completes the activity.



**Activity summary**

In this activity you learned how to create an alternate assembly. In the alternate assembly, you learned how to add new members, define unique variables for individual members, exclude member occurrences and reapply assembly relationships for a member.

## **Lesson review**

Answer the following questions:

1. In the Alternate Assemblies tab in PathFinder, what two types of assemblies can you create?
2. In a family of assemblies, which substitutions, modifications or exclusions are allowed?
3. Can an assembly family member be saved as a unique assembly?
4. How do you place an assembly family member on a drawing sheet?

## Answers

1. In the Alternate Assemblies tab in PathFinder, what two types of assemblies can you create?

In the Alternate Assemblies tab in PathFinder, you can create a family of assemblies or an alternate assembly.

2. In a family of assemblies, which substitutions, modifications or exclusions are allowed?

The list of substitutions modifications and exclusions is:

- Offset values for assembly relationships
- Occurrence overrides
- Excluded occurrences
- Excluded relationships
- Excluded assembly features
- Excluded frames
- Excluded piping routes
- Excluded pattern occurrences
- Excluded fasteners

3. Can an assembly family member be saved as a unique assembly?

An assembly family member can be saved as a unique assembly.

4. How do you place an assembly family member on a drawing sheet?

When the drawing view wizard is run on the alternate assembly, the list of family members is provided for selection of the correct member

## **Lesson summary**

In this lesson you learned how to create an alternate assembly. In the alternate assembly, you learned how to add new members, define unique variables for individual members, exclude member occurrences and reapply assembly relationships for a member.

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**Lesson**

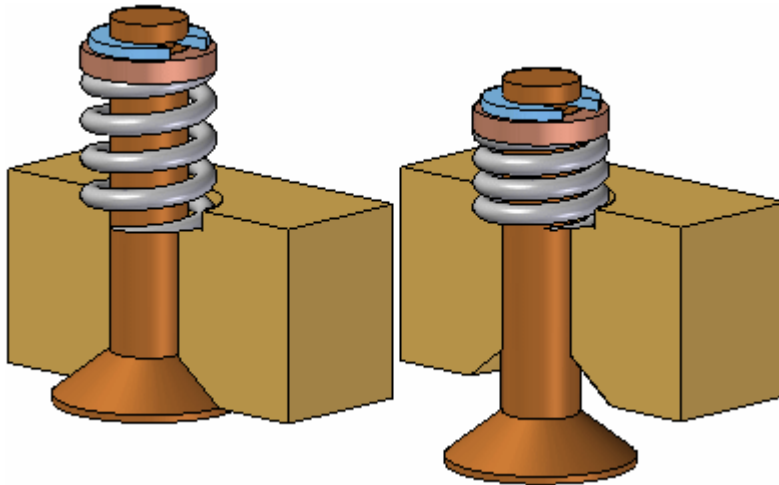
# *13 Adjustable parts and assemblies*

## **Creating an adjustable part**

Adjustable parts will change relationship values to fit when placed in an assembly.

## Adjustable parts in assemblies

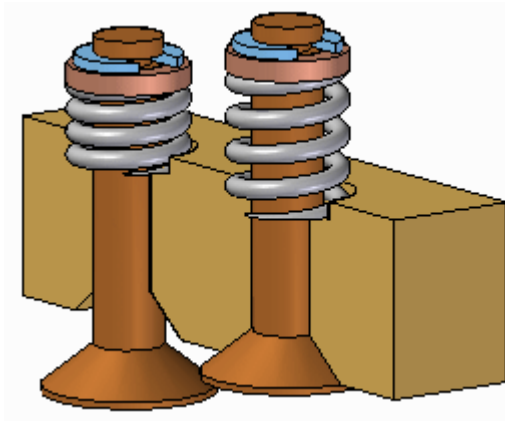
In some designs, there are parts that must react to changing conditions in the assembly. For example, a spring that is compressed or uncompressed based on the position of other parts in the assembly.



The Adjustable Parts functionality in Solid Edge allows you to define parameters in a part model that will adjust with respect to corresponding parameters within the assembly. This allows you to control the size and shape of the part based on parameters you define in the assembly.

When you specify that a part is adjustable, the design body in the part model does not change when the assembly parameters change. An associative copy of the design body in the assembly changes. The associative copy of the design body is placed in the assembly automatically and is managed by Solid Edge when you specify that a part is adjustable within the context of the assembly.

This allows you to place several occurrences of an adjustable part into an assembly, and each occurrence of the adjustable part will conform to the current parameter values for that occurrence of the part. For example, one occurrence of a spring can be shown compressed while another occurrence of the spring can be shown uncompressed.



**Note**

Only the design body for an adjustable part is associatively copied to the assembly. If the adjustable part contains construction bodies, they are not associatively copied to the assembly.

**Making a part adjustable**

To make a part adjustable within the context of an assembly, you first must define the parameters you want to adjust in the part document. You can then define corresponding parameters in the Assembly environment.

You can use driving dimensions and variables that control a feature, reference plane, or construction element as the parameters to define an adjustable part.

When you specify that a part is adjustable, you cannot in-place activate the part using the Edit command. You can use the Open command to open the part.

**Defining the part parameters**

The Adjustable Part command on the Tools tab in the Part or Sheet Metal environments displays the Adjustable Part dialog box so you can define or edit the adjustable parameters.

When the Adjustable Part dialog box is displayed, you can select features to display their dimensions, or you can click the Variable Table button on the Adjustable Part dialog box to display the variable table.

For example, to make the length of the spring shown adjustable, you can add the variable which controls spring length: SprLngPrt, to the adjustable parameters list by selecting the variable in the Variable Table.



<input type="checkbox"/>	Dim	<input type="checkbox"/>	AxLng	45.00 mm	SprLngPrt + SpDia
<input type="checkbox"/>	Dim	<input type="checkbox"/>	SpDia	5.00 mm	
<input type="checkbox"/>	Dim	<input type="checkbox"/>	RefPI4Dis	5.00 mm	SpDia
<input type="checkbox"/>	Dim	<input type="checkbox"/>	RefPI5Dis	45.00 mm	AxLng - SpDia
<input type="checkbox"/>	Dim	<input type="checkbox"/>	SprRad	15.00 mm	
<input checked="" type="checkbox"/>	Var	<input type="checkbox"/>	SprLngPrt	40.00 mm	
<input type="checkbox"/>		<input type="checkbox"/>			

When you add a variable or dimension to the Adjustable Part dialog box, the parameter name is added to the Variable Name column (A). You can also add text to the Notes column (B) to make it easier to remember later what aspect of the part the adjustable parameter controls.

	(A)	(B)
1	SprLngPrt	Spring Length



## Placing adjustable parts in an assembly

When adding an adjustable part to an assembly, you should place and position the parts which interact with the adjustable part first. This allows you to use the surrounding parts to define the assembly parameters required to complete the process. You can specify whether an adjustable part is adjustable or rigid in the assembly when placing the part or after you position the part in the assembly.

When you drag and drop an adjustable part into an assembly, a dialog box is displayed that allows you to specify whether the part is rigid or adjustable.

When you set the Place Rigid option, the part placement process proceeds as it would for a typical part. You can then define assembly relationships to position the part in the assembly. An adjustable part placed as rigid in an assembly behaves the same as any other part in an assembly.

When you set the Place Adjustable option, the part placement process is temporarily suspended so you can define the adjustable parameters in the assembly using the Adjustable Part dialog box.

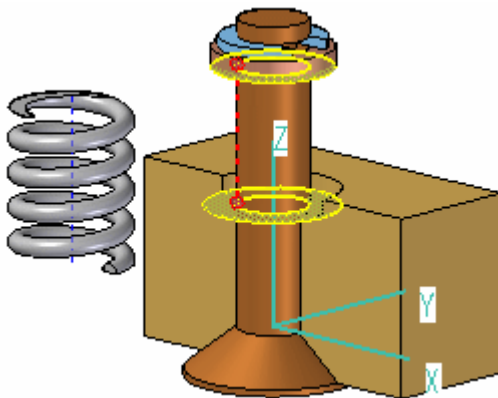
### Note

When positioning an adjustable part, the option to use a separate Place Part window is not available. The part is placed in the assembly window so you can define the adjustable parameters and the assembly relationships in one window.

## Defining the assembly parameters

In addition to the options for selecting driving dimensions and variables, the Adjustable Part dialog box in the Assembly environment contains options that allow you to define a measurement variable. This allows you to use geometry on other parts in the assembly to define variables which will control the size and shape of the adjustable part in the assembly.

The measurement variable options activate one of the Measurement commands that are also available on Inspect® Measure. For example, you can use the Measure Minimum Distance option to specify that the minimum distance between the two faces shown controls the height parameter of the part.



After you select the elements in the assembly that define the distance you want to measure, an assembly variable is created automatically and added to the Assembly Variable cell in the Adjustable Part dialog box for the adjustable part you are placing or editing.

There are three columns in the Adjustable Part dialog box in the Assembly environment: Part Variable (A) Notes (B), and Assembly Variable (C). In this example, the part variable SprLngPrt is controlled by the measurement variable SprLngAsm in the assembly.

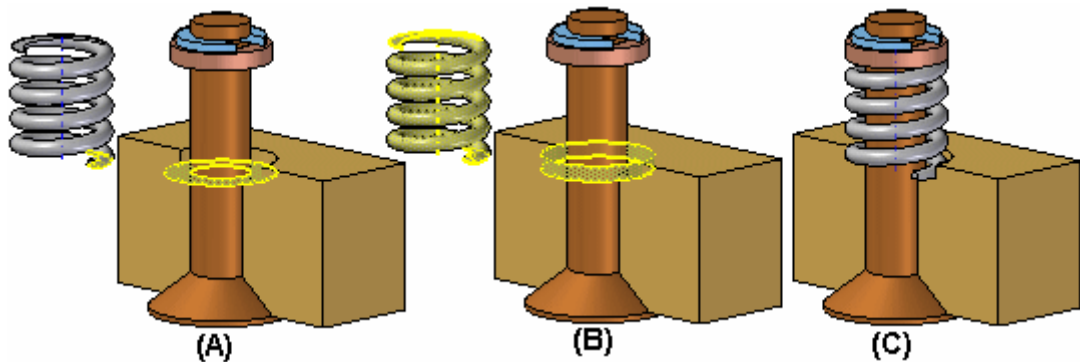
(A)	(B)	(C)
1 SprLngPrt	Spring Length	SprLngAsm

In addition to defining measurement variables, you can also use assembly relationship variables for an adjustable part. For example, you can use the offset value for a mate or planar align relationship as an assembly variable by selecting the variable value for the relationship in the Variable Table.

The place like a spring option will use the variable created by measuring a distance to adjust the length of the corresponding variable in the part or sheet metal document. The position of the parts attached to the adjustable part determines the length of the variable defining the distance.

The adjust to fit and allow assembly relationships option will use the variable created by the measurement to change the length of the adjustable part, and reposition parts within the assembly that are not constrained. The length of the variable defining the adjustable part length is used to position the unconstrained parts connected to the adjustable part.

After you have defined all the parameters in the assembly to control the adjustable part, click the OK button on the Adjustable Part dialog box to resume the part placement process. In this example, a mate relationship (A) and an axial align relationship (B) fully position the part in the assembly (C).



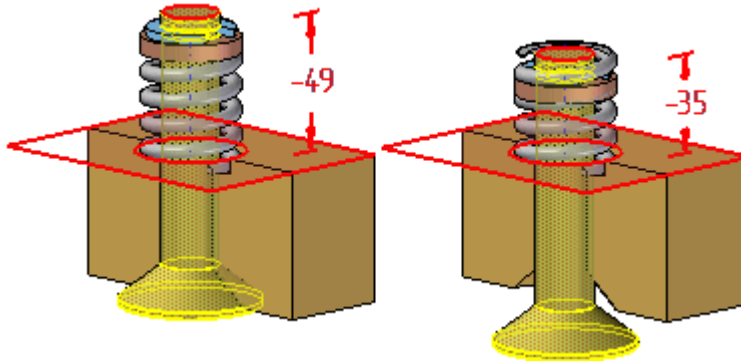
You can also specify that a part is adjustable after it has been positioned in the assembly. First, you must define the adjustable parameters for the part in the Part or Sheet Metal environment. Then, in the assembly, you can use the Adjustable Part command on the shortcut menu when a part is selected to specify that the part is adjustable and then define the adjustable parameters.

**Note**

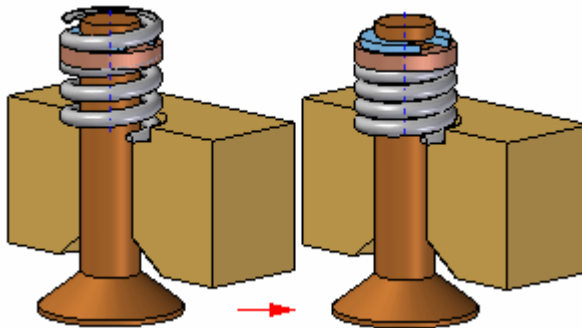
When you specify that a part is adjustable, you cannot in-place activate the part using the Edit command. You can use the Open command to open the part.

### Updating adjustable parts

When you edit the assembly such that the adjustable part must change, the size and shape of the adjustable part updates automatically when the Automatic Update option is set. For example, in this assembly, if you edit the offset value for the planar align relationship between the valve and body parts, the valve opens.



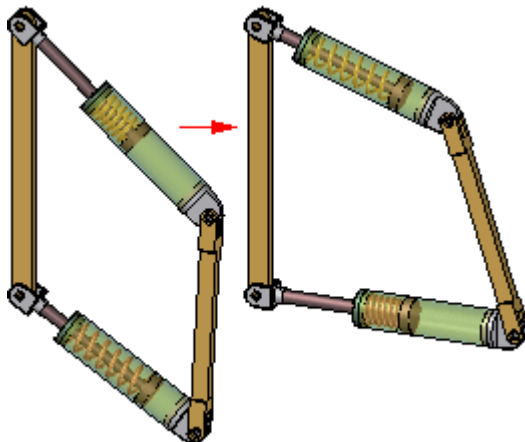
This causes the size and shape of the adjustable part to update automatically.



### Adjustable parts in adjustable subassemblies

You can place a subassembly that contains an adjustable part into an assembly, then make the subassembly adjustable. For example, you may need to place two instances of a cylinder subassembly, with each subassembly in different positions.

Each cylinder assembly contains a spring that is an adjustable part, which allows the spring to change length as the cylinder subassemblies change positions.



When you make a subassembly adjustable that contains adjustable parts, the adjustable part variables are promoted to the current assembly.

For more information on creating and using adjustable assemblies, see the [Adjustable assemblies](#) Help topic.

### **Using reference geometry to constrain adjustable parts**

You can use part reference planes or construction geometry to define positioning relationships for an adjustable part in an assembly, but in some cases this can prevent the adjustable part from reacting properly to assembly changes.

If this occurs, you can edit the positioning relationship to use geometry on the design body on the adjustable part instead.

### **Adjustable parts and Parts Lists**

When you place the same adjustable part several times in an assembly in different states of adjustment, all occurrences have a single part number. If you use several family of parts members to simulate adjustable parts in different states of adjustment, you can have more than one part number, because different family of parts members have unique part numbers. Typically, a single part number is the preferable result.

### **Adjustable parts and alternate assemblies**

You can use adjustable parts in a family of assemblies. You can edit the assembly variable used to control an adjustable part on a per member basis by clearing the Apply Edits to All Members option.

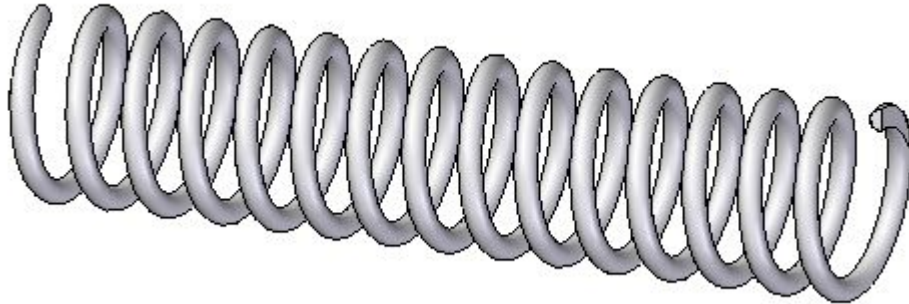
## **Activity: Creating an adjustable part**

### **Overview**

The objective of this activity is to show how to create an adjustable part to be used in an assembly.

### **Activity**

In this activity you will create a spring that adjusts its length when placed in an assembly.



**Activity: Creating an adjustable part**

**Overview**

This activity demonstrates to create an adjustable part and place it into an assembly.

**Objectives**

You will create a spring and use it as an adjustable part and a rigid part in an assembly.

*Creating the variable defining the adjustable distance*

Sketches created in a part document will be used to define the adjustable variable.

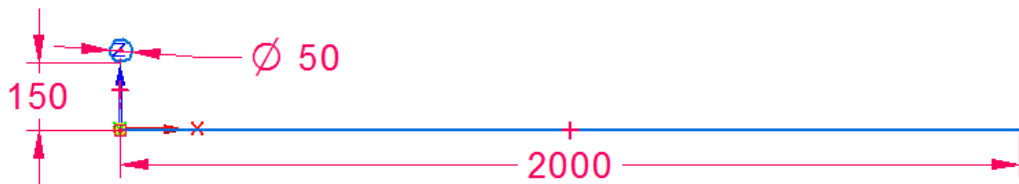
- ▶ From the Solid Edge start screen, click Open Existing Document. Browse for *spring.par* in the folder where the activity files are located.
- ▶ Select Sketch1 in pathfinder and then select Edit Profile to edit the sketch.



**Note**

This sketch will be used to create a helix defining the spring. To make the length adjustable, a dimension controlling the length will be defined.

- ▶ Dimension the horizontal line in the sketch. The length is 2000 mm.



- ▶ Click Tools, then click Variables to show the variable table.

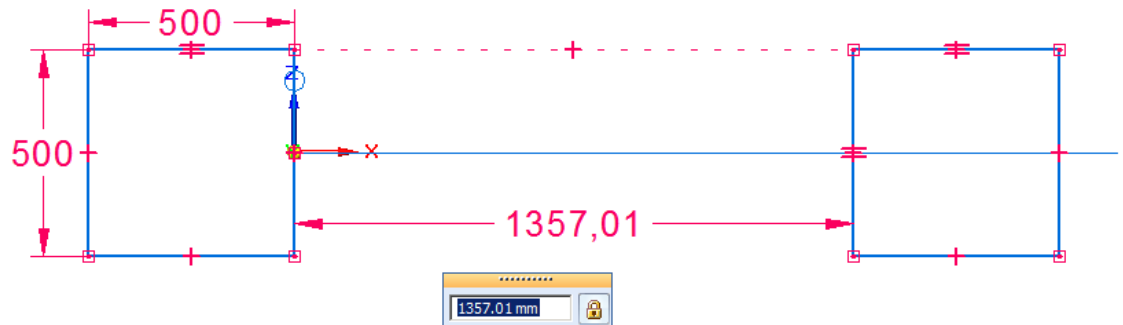
- Find the variable with the length equal to 2000 and change the name of the variable name to `spring_length`.

Type	Name	Value	Rule	Formula
Var	PhysicalProp...	0.000 kg/m <sup>3</sup>	Limit	
Var	PhysicalProp...	0.990	Limit	
Dim	V677	500.00 mm		
Dim	<b>spring_length</b>	2000.00 mm		
Dim	V676	500.00 mm		
Dim	V408	150.00 mm		
Dim	V403	50.00 mm		

- Dismiss the variable table.
- Click the home tab, then click Close Sketch.
- Click finish.
- Select Sketch2 in pathfinder and then select Edit Profile to edit the sketch.



- Place a horizontal dimension between the two rectangles.



### Note

The rectangles will be used to create a cutout and shave off the ends of the spring creating a planar face on each end. The spacing will be controlled by a formula in the variable table equating the spacing between the rectangles to the `spring_length` variable previously defined.

- Click Tools, then click variables to show the variable table.

## Lesson 13 *Adjustable parts and assemblies*

- Find the variable with the length equal to the horizontal dimension just created. In the formula field, set the value equal to the variable `spring_length`.

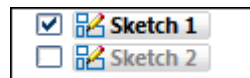
Var	PhysicalProp...	0.000 kg/m <sup>3</sup>	Limit	
Var	PhysicalProp...	0.990	Limit	
Dim	V677	500.00 mm		
Dim	V730	1357.01 mm		<code>=spring_length</code>
Dim	spring_length	2000.00 mm		
Dim	V676	500.00 mm		
Dim	V408	150.00 mm		
Dim	V403	50.00 mm		

- Dismiss the variable table.
- Click the home tab, then click Close Sketch.
- Click finish.

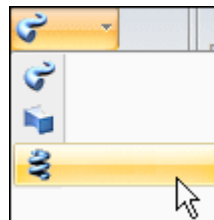
*Create the spring from the sketches*

Create the helix from the sketch1.

- Hide Sketch2.



- On the home tab, in the solids group, click add helix.



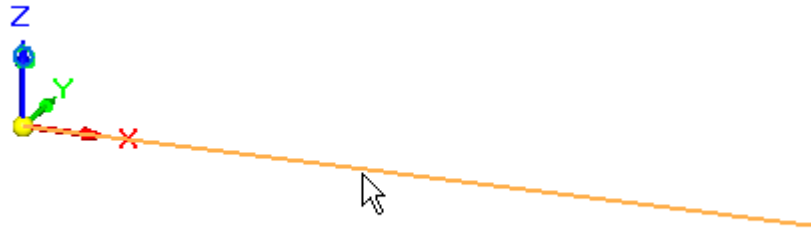
- Set the create from option to: Select from Sketch.



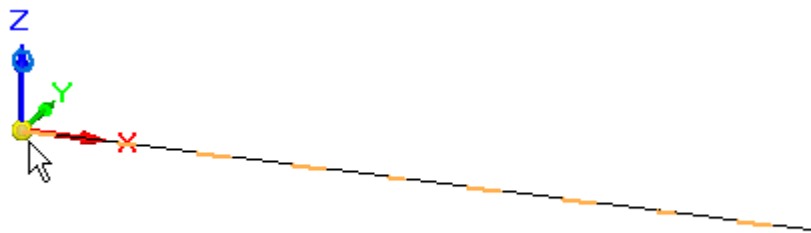
- Select the circle as the sketch chain and then click accept.



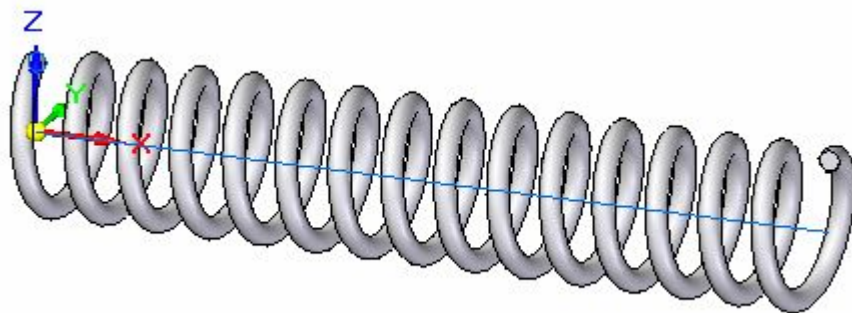
- ▶ Select the horizontal line as the axis.



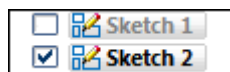
- ▶ Select the left side of the line as the origin of the axis.



- ▶ Set the method to Axis length and turns and the number of turns to 15.
- ▶ Click Next.
- ▶ Click Preview.
- ▶ Click Finish.



- ▶ Hide Sketch1 and show Sketch2.



**Lesson 13**     *Adjustable parts and assemblies*

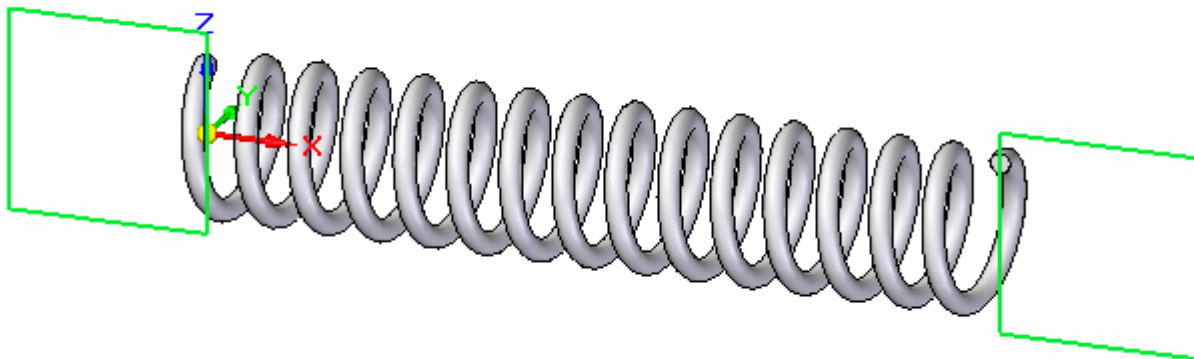
- ▶ On the Home tab, in the Solids group, click the Cut command.



- ▶ Set the create from option to: Select from Sketch.



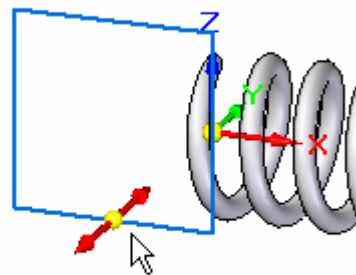
- ▶ Select each of the rectangles in the sketch, then click the accept button.



- ▶ Select the through all option for the extent of the cut.



- ▶ Select both directions to define the extent of the cut.



- ▶ Click Finish to complete the cut.

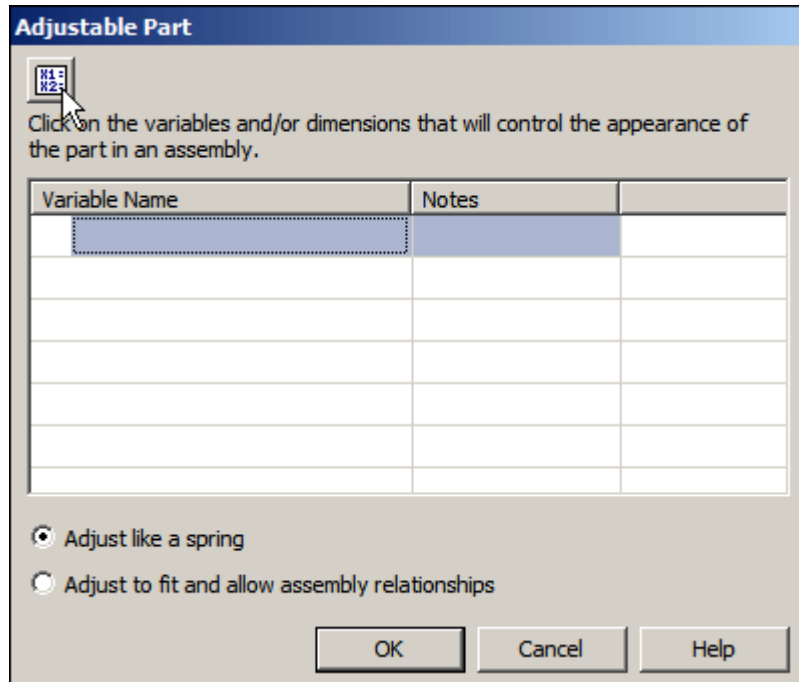
*Define the adjustable variable*

The variable defining the axis length of the spring will be defined as the adjustable variable.

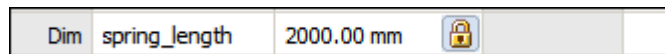
- ▶ Click the Tools Tab. In the Assistants group, click Adjustable part.



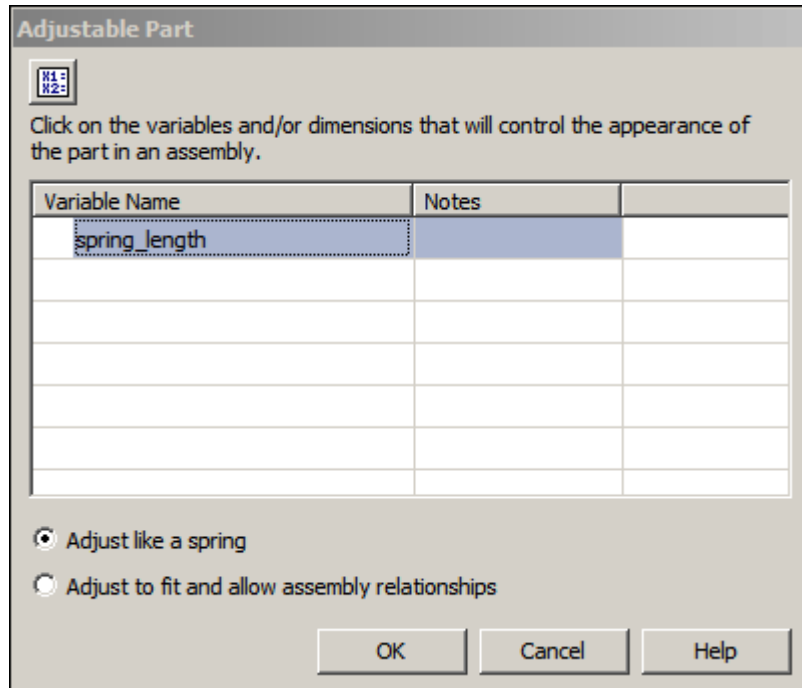
- ▶ Click the variable table button.



- ▶ Select spring\_length as the adjustable variable.



- ▶ Dismiss the variable table. Spring\_length will be defined as the adjustable variable.

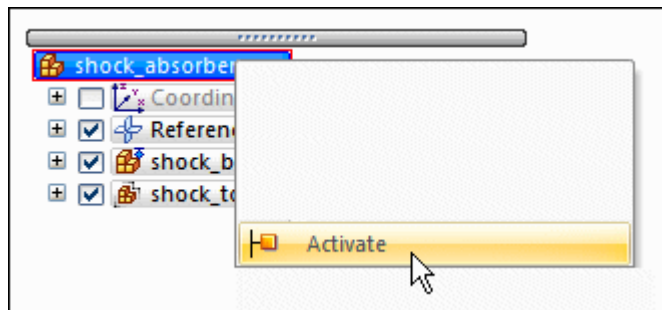


- ▶ Click OK.
- ▶ The spring is complete. Save and close the document.

*Place and define the adjustable part as a spring*

The spring will be placed and positioned in the assembly as an adjustable part.

- ▶ Open the assembly *shock\_absorber.asm*.
- ▶ In pathfinder, right click *shock\_absorber.asm* and then click Activate to activate all the parts.



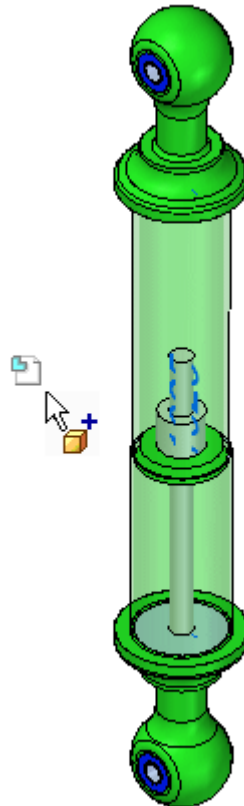
The subassembly, *shock\_top.asm*, examine the relationships used to position the subassembly relative to *shock\_bottom.asm*.

- In pathfinder, click *shock\_top.asm*. In the lower pane notice that there is an axial align relationship and a floating planar align.

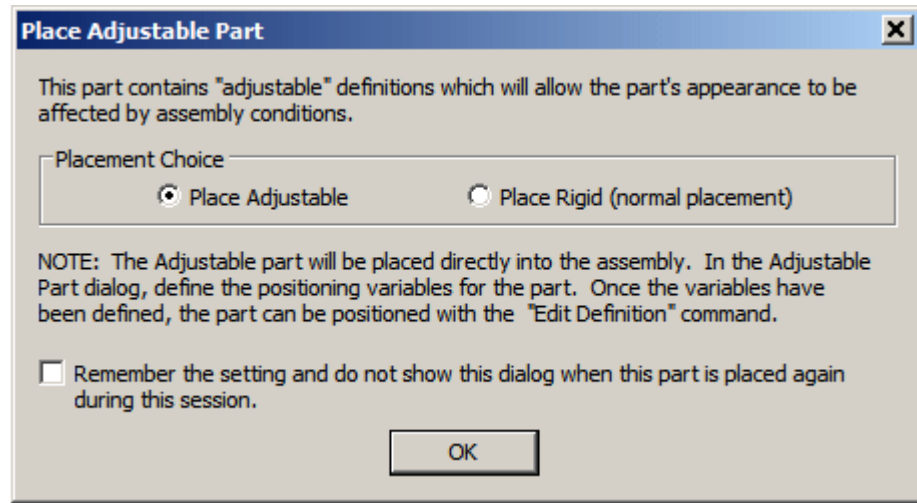
**Note**

These relationships keep the cylindrical parts aligned and keep the holes containing the bushing and sleeve parallel. There is still freedom to move along the axis of the cylinders.

- From the parts library, drag *spring.par* into the assembly.



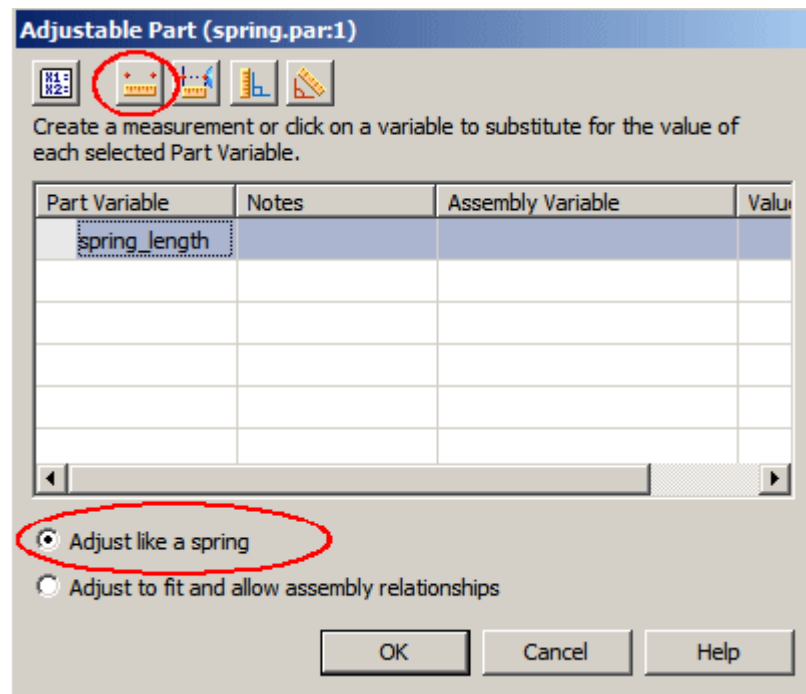
- ▶ Set the placement choice as Place Adjustable, then click OK.



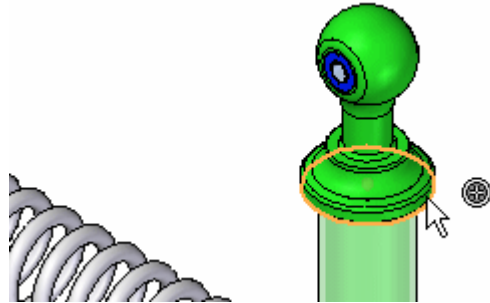
**Note**

The adjustable variable in the spring will be controlled by the measured distance between two faces defined in the next steps.

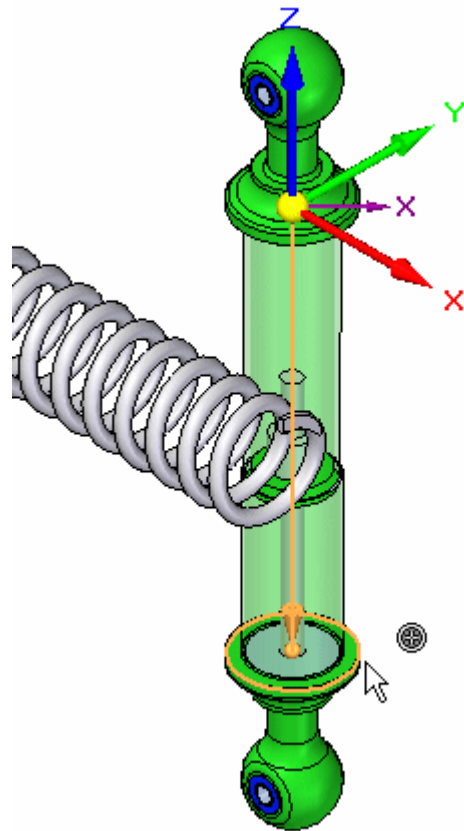
- ▶ Click Adjust like a spring and then click the measure button.



- ▶ Select the circular face shown for the point to measure from. The measurement tool will lock to the radial point of the circle when selected.



- ▶ Select the circular face shown for the point to measure to. The measurement tool will lock to the radial point of the circle when selected.



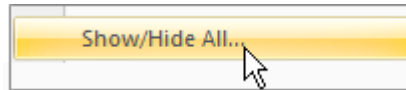
**Note**

The adjustable distance has been set and the spring length adjusts to the distance defined.

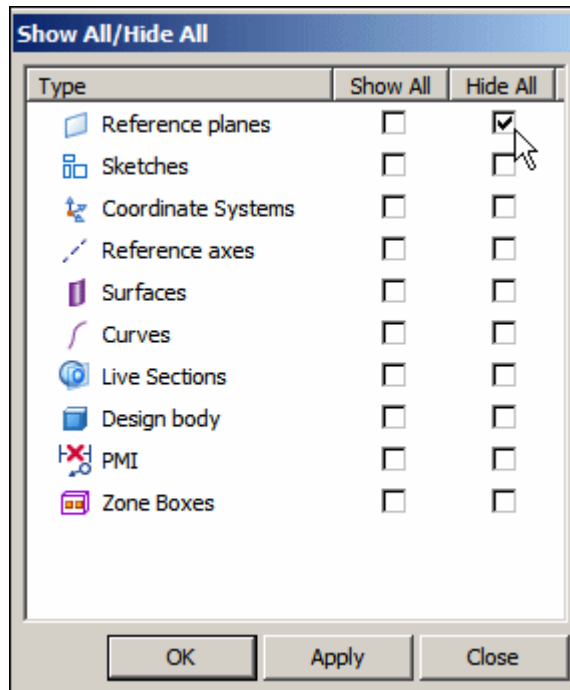
- ▶ Click OK to dismiss the dialog box.

The reference planes will be used to position the spring. The next steps will turn on the planes needed to position the spring.

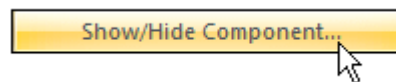
- ▶ Click the Select tool. Right mouse click in the assembly window. Click Show/Hide All.



- ▶ Hide all the reference planes.

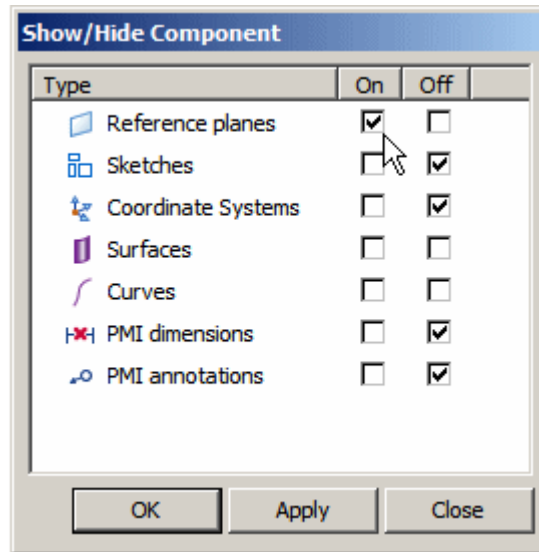


- ▶ Click OK to dismiss the dialog box.
- ▶ In pathfinder, right click *shock\_bottom.asm* and then click Show/Hide Component.





- ▶ Turn on the Reference planes for *shock\_bottom.asm*. Then click OK to dismiss the dialog box.



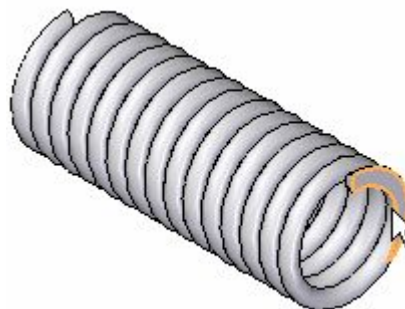
- ▶ On the Home tab in the Select group, click Clear Selection.



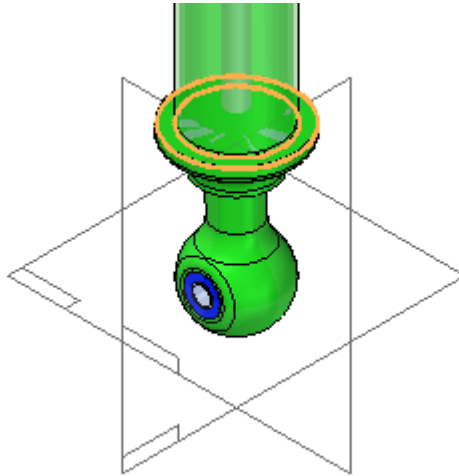
- ▶ In the Assemble group, click Assemble.



- ▶ Using the mate relationship, select the face shown.



- ▶ For the target face, click the face shown.



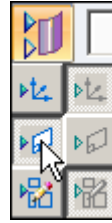
- ▶ Using the mate relationship, select the face shown.



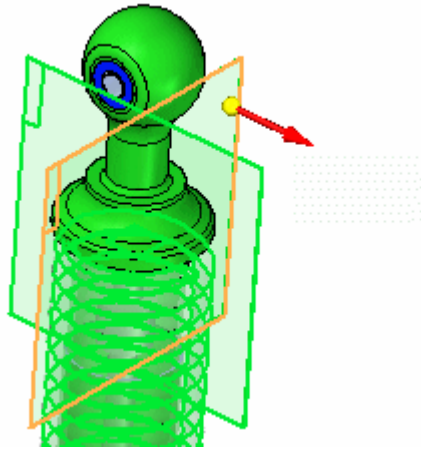
- ▶ For the target face, click the face shown.



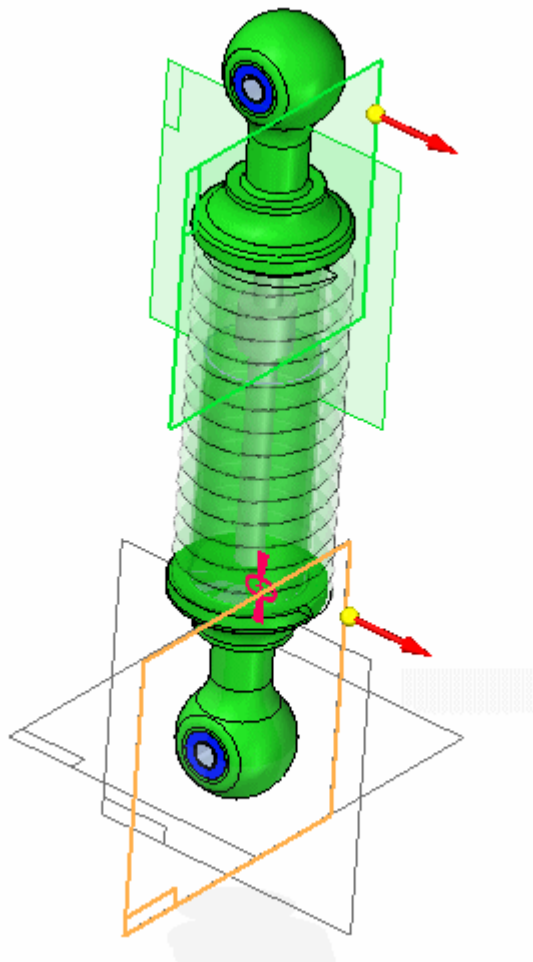
- ▶ Click the construction display to turn on the reference planes for *spring.par*.



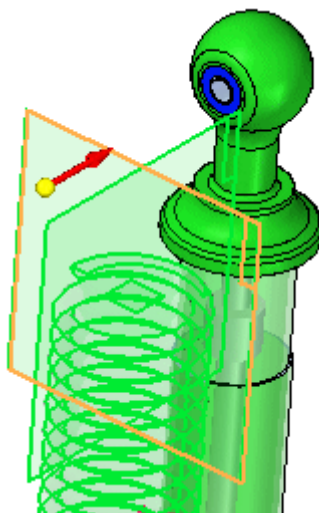
- ▶ Using the mate relationship, select the reference plane shown.



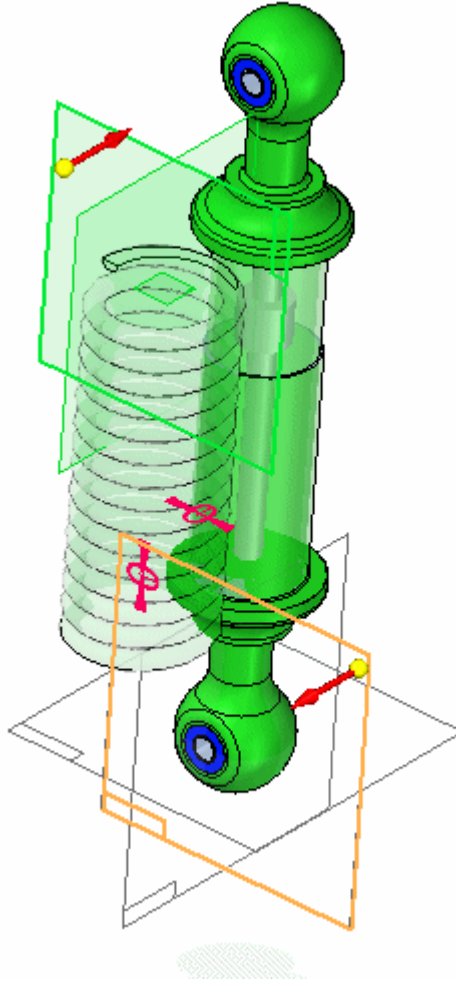
- ▶ Select the reference plane shown as the target.



- ▶ Using the mate relationship, select the reference plane shown.



- Select the reference plane shown as the target.



- The Spring is placed.

- Turn off the display of the reference planes..



*shock\_top.asm* is still free to move along the axis of the cylinders. You will move this part and the spring will adjust size based on the position of this subassembly.

- Click the home tab. In the modify group, click the drag command.



- Drag *shock\_top.asm* to increase the separation distance between the subassemblies.



- The spring will adjust to the spacing between the faces.

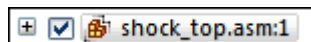


- Use the drag command to change the spacing and observe how the spring reacts.

*Define the adjustable part as a rigid part*

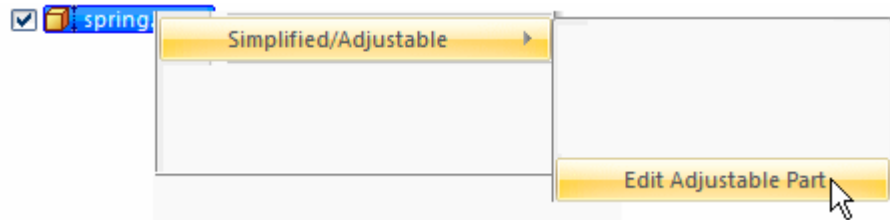
Previously, the spring was set to adjust as a spring. The length of the spring was determined by the spacing between two faces on different parts. The adjustable part also be used to determine the spacing between the faces which removes the freedom to move and makes the assembly rigid. This will be demonstrated in the next steps.

- Notice the icon in pathfinder for the subassembly *shock\_top.asm* shows it as under constrained.

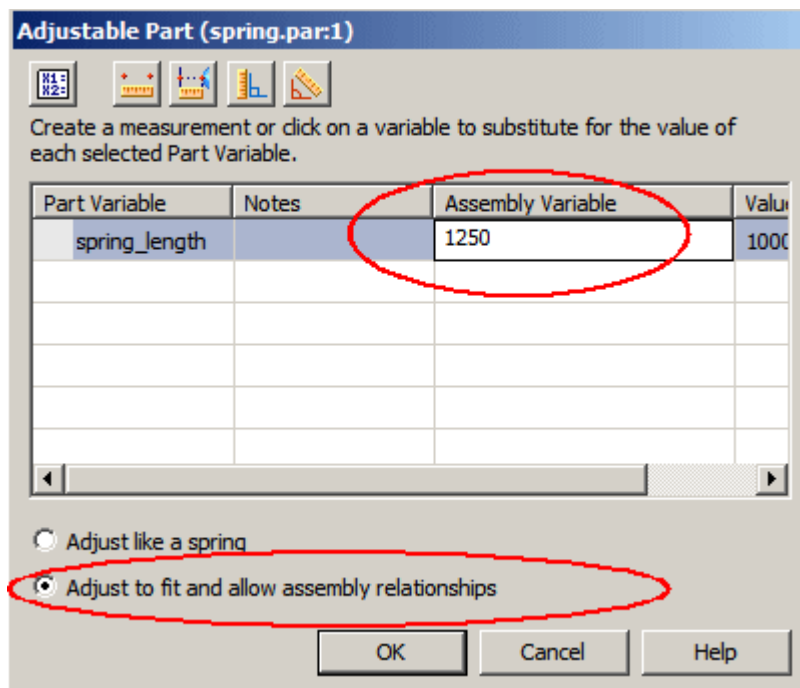




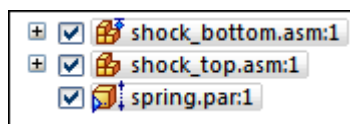
- Click the select tool. In pathfinder, right click *spring.par*. Click Simplified/Adjustable. Click Edit Adjustable Part.



- Change the behavior to Adjust to fit and allow assembly relationships, and enter 1250 as the Assembly Variable value as shown.



- Click OK. Notice that *shock\_top.asm* is now constrained and the variable defining spring length determines the offset value.



- Save and close the document. This completes the activity.

### Summary

In this activity you learned how to create and adjustable part and place it in an assembly as a spring, or to adjust the fit to allow assembly relationships.

### **Lesson review**

Answer the following questions:

1. Is the following statement true or false? When an adjustable part is placed in an assembly and adjusts to fit, the part document containing the part also adjusts to a specific size, as well as every occurrence of that part in the assembly and other assemblies that it may reside.
2. Fill in the blank in the following statement. When defining a part as adjustable, the adjustable value is defined by a \_\_\_\_\_.
3. Is the following statement true or false? After defining a part as adjustable, it is impossible to place the part as a rigid part.
4. What is the difference between the following placement options of an adjustable part?
  - Adjust like a spring
  - Adjust to fit and allow assembly relationships

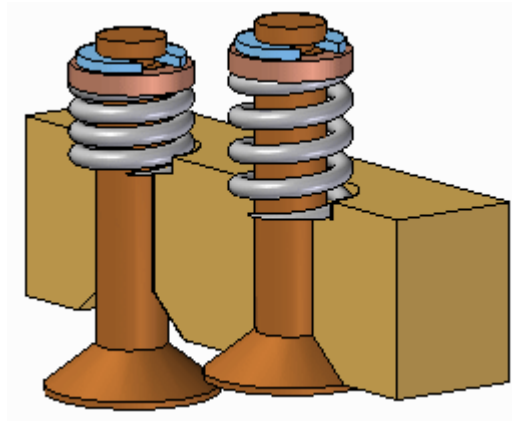
## Answers

1. Is the following statement true or false? When an adjustable part is placed in an assembly and adjusts to fit, the part document containing the part also adjusts as well as every occurrence of that part in the assembly and other assemblies that it may reside in.

The answer is false.

When you specify that a part is adjustable, the design body in the part model does not change when the assembly parameters change. An associative copy of the design body in the assembly changes. The associative copy of the design body is placed in the assembly automatically and is managed by Solid Edge when you specify that a part is adjustable within the context of the assembly.

This allows you to place several occurrences of an adjustable part into an assembly, and each occurrence of the adjustable part will conform to the current parameter values for that occurrence of the part. For example, one occurrence of a spring can be shown compressed while another occurrence of the spring can be shown uncompressed.



2. Fill in the blank in the following statement. When defining a part as adjustable, the adjustable value is defined by a **variable**. A dimension value has a variable associated with it which shows up in the variable table.
3. Is the following statement true or false? After defining a part as adjustable, it is impossible to place the part as a rigid part.

The answer is false.

The user has the option to place the part as either adjustable or rigid.

4. What is the difference between the following placement options of an adjustable part?
  - Adjust like a spring
  - Adjust to fit and allow assembly relationships

When using the adjust like a spring option, the variable length is controlled by the spacing between the components used to position the adjustable part. When using the adjust fit and allow assembly relationships option, the spacing between

the other components is controlled by the size of the adjustable dimension in the adjustable part.

## **Lesson summary**

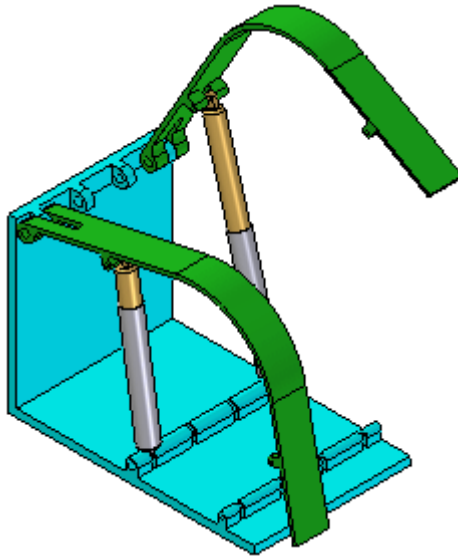
In this lesson you learned how to create and adjustable part and place it in an assembly as a spring, or to adjust the fit to allow assembly relationships.

## **Creating an adjustable assembly**

Adjustable assemblies will change relationship values to fit when placed in a higher level assembly.

## **Adjustable and rigid assemblies**

When working with assemblies, it is sometimes necessary to allow movement within a subassembly while working in a higher-level assembly. In other instances, it can be necessary to show identical subassemblies in different positions. For example, you can have two identical hydraulic cylinder subassemblies in an assembly, but need to show the hydraulic cylinders in different positions.



The Adjustable Assembly functionality allows you to address both of these issues.

### **Comparing rigid and adjustable subassemblies**

Specifying that a subassembly is adjustable allows you to place positioning relationships between parts in the subassembly while in the higher-level assembly. This is not possible with a rigid subassembly.

When you specify that a subassembly is adjustable, you are prevented from in-place activating the subassembly. For example, when you try to in-place activate the subassembly using the Edit command, a dialog box is displayed that informs you that the subassembly is adjustable and to use the Open command to open the subassembly.

### **Displaying identical subassemblies in different positions**

There are several approaches to solving this problem:

You can create uniquely-named subassemblies for each of the otherwise identical subassemblies. This allows you to assign unique offset values to the affected relationships, but creates extra files and complicates data management.

You can create a single-level assembly where the subassembly components are placed as discrete parts, instead of as a subassembly. This also allows you to assign unique offset values to the affected relationships, but makes it more difficult to reuse the hydraulic cylinder components later in another assembly. Another disadvantage of this method is that the parts are listed individually, rather than as a subassembly.

Alternately, you can use the Adjustable Assembly functionality within Solid Edge. This approach eliminates the need to create multiple copies of the hydraulic cylinder subassembly data set or to create single-level assemblies.

### **Preparing the subassembly**

To use the Adjustable Assembly functionality, the subassembly should be left under-constrained in the range of motion in which you want to adjust. This allows you to apply the relationship(s) that you want to adjust in the higher level assembly, not in the subassembly.

### **Placing the subassembly into the higher-level assembly**

You place the subassembly into the higher-level assembly in the same manner as you would any subassembly. There are several methods available to specify that you want the subassembly to be considered an adjustable assembly.

To specify that the subassembly is considered adjustable while you are placing the subassembly, set the Place As Adjustable option on the Options dialog box on the Assemble command bar.

To specify that the subassembly is considered adjustable after you have completed positioning the subassembly, select the subassembly in PathFinder, then click the Adjustable Assembly command on the shortcut menu.

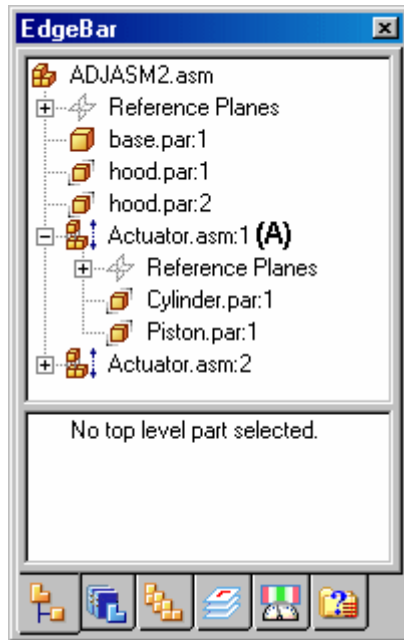
#### **Note**

Only subassemblies that contain parts that are not fully positioned can be marked as adjustable.

You can also specify that a subassembly is adjustable by setting the Place as Adjustable when this Assembly is Placed into Another Assembly option on the Assembly tab on the Options dialog box.

Regardless of the method used, a special symbol is used in PathFinder (A) to indicate the subassembly is adjustable.





## Working with adjustable assemblies

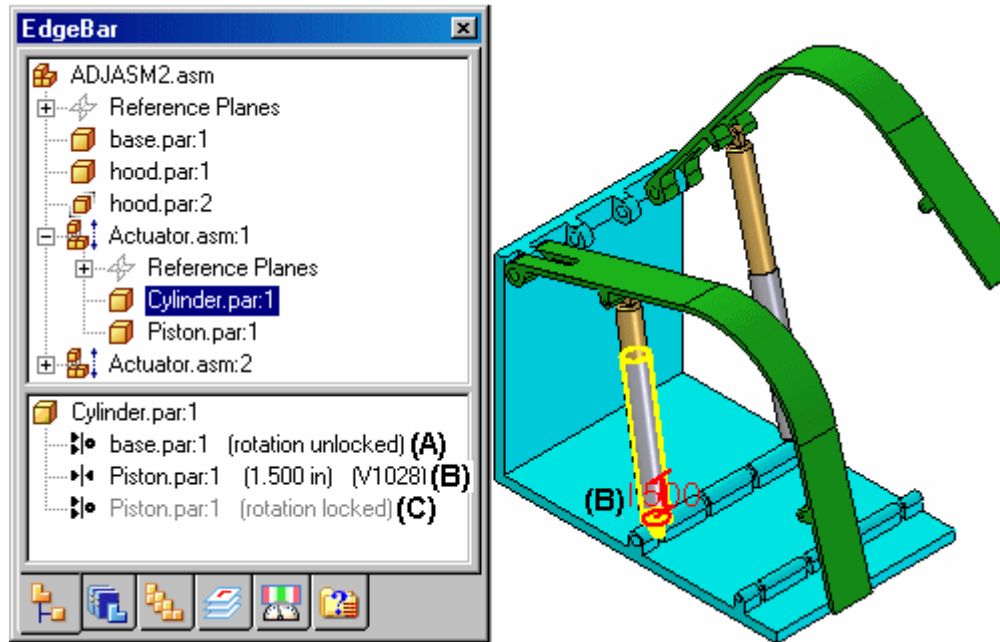
When a subassembly is set to adjustable, all assembly relationships existing within the subassembly are solved at the level of the active assembly. In other words, the relationships in the subassembly are promoted to the higher-level assembly for solve purposes.

The relationships used to position the parts within the subassembly can be viewed in the bottom pane of PathFinder when you select a part in the subassembly. These relationships are read-only and the text label is gray to indicate that the relationship cannot be edited. Displaying the read-only relationships makes it easier to evaluate the existing relationships and apply the remaining relationships.

For example, when you select cylinder.par:1 in the adjustable assembly named Actuator.asm:1, three relationships are displayed. The axial align relationship to base.par (A) was placed in the current assembly. It was used to position the subassembly in the current assembly, and is editable.

The Mate relationship to piston.par:1 (B) was placed in the current assembly after the subassembly was made adjustable. Its purpose is to adjust the length of the hydraulic cylinder subassembly and the relationship is editable.

Notice that no visual distinction is made between relationships (A) and (B), although one of the relationships was used to position the subassembly in the current assembly (A), and the other was used to position the two parts in the subassembly.



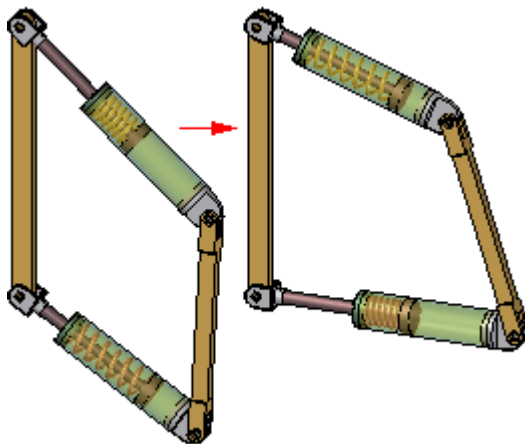
The remaining axial align relationship to piston.par:1 (C) was placed in the subassembly, is read-only and not editable within the current assembly. Notice that the text label is gray, indicating that the relationship is read-only.

If you specify that a subassembly is flexible, add positioning relationships, and then specify that the subassembly is rigid, conflicting relationships can occur. You can delete or suppress relationships to correct this situation.

### Adjustable assemblies and adjustable parts

You can create assemblies that contain adjustable parts within an adjustable subassembly. For example, you may need to place two instances of a cylinder subassembly, with each subassembly in different positions.

Each cylinder assembly contains a spring that is an adjustable part, which allows the spring to change length as the cylinder subassemblies change positions.



When you make a subassembly adjustable that contains adjustable parts, the adjustable part variables are promoted to the current assembly.

If you define an assembly variable in the variable table for a subassembly that controls a part variable, the subassembly variable is promoted to the current assembly. The promoted variable is a linked variable.

For more information on creating and using adjustable parts in assemblies, see the [Adjustable parts in assemblies](#) Help topic.

### **Adjustable assemblies and the Drag Part command**

If you specify that a subassembly is adjustable, and the combination of relationships at the active level and the promoted relationships allow movement, you can use the Drag Part command to reposition the parts. Adjustable assemblies work with all modes of the Drag Part command.

Because an adjustable subassembly is typically used to drive movement in an assembly, you may need to provide for that movement by suppressing or deleting relationships in the related parts and subassemblies.

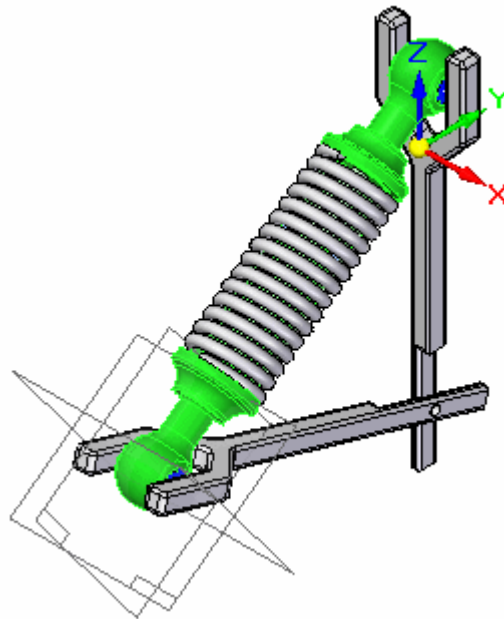
## **Activity: Creating an adjustable assembly**

### **Overview**

The objective of this activity is to show how to create an adjustable assembly to be used in a higher level assembly.

### **Activity**

In this activity you will create and place an adjustable assembly.



**Activity: Creating an adjustable assembly**

**Overview**

This activity demonstrates to create an adjustable assembly.

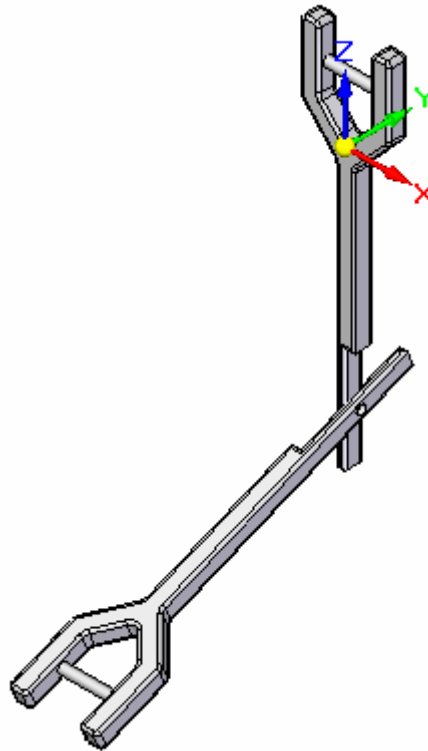
**Objectives**

You will place an assembly and define it as adjustable. The assembly will contain an adjustable part.

*Place an assembly containing an adjustable part into a higher level assembly*

The assembly you will place will later be defined as adjustable.

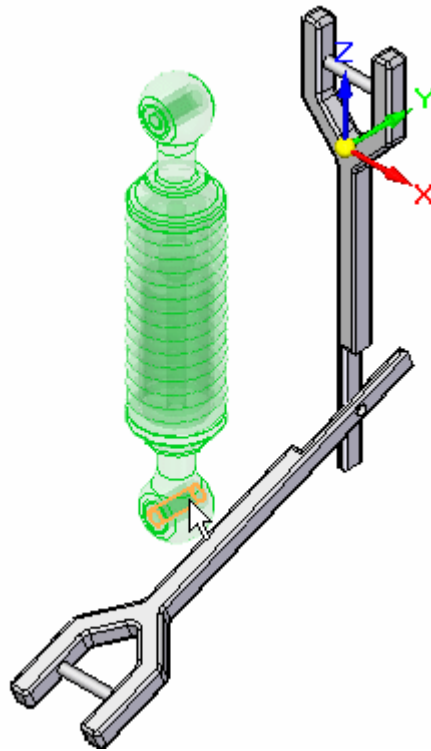
- Open the assembly *arms.asm*. Activate all the parts in the assembly.



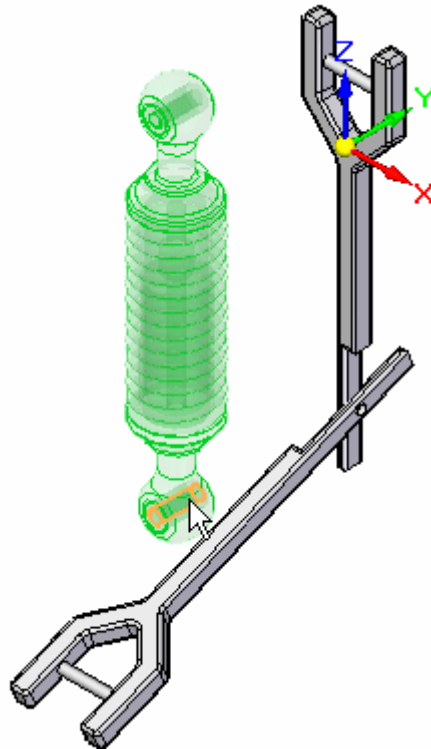
- From the parts library, drag *shock\_absorber1.asm* into the assembly window.
- Click the activate button on the assemble command toolbar.



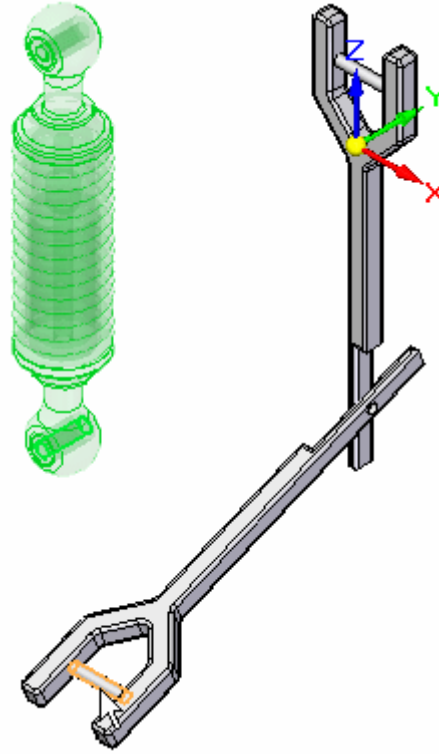
- Using quickpick, activate the part *sleeve.par*.



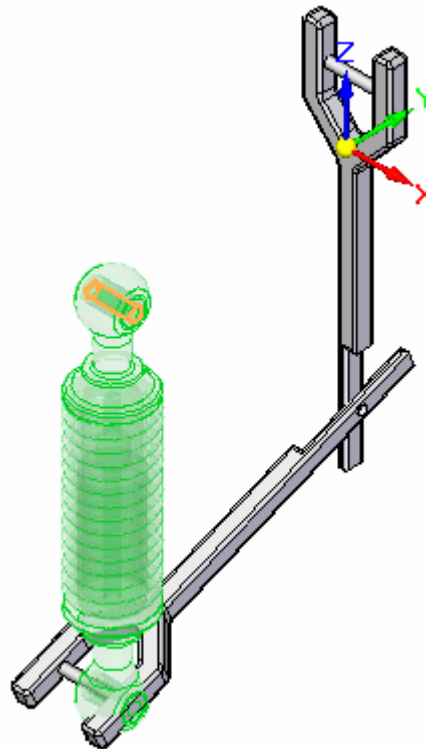
- Using Flashfit, select the cylinder in *sleeve.par*.



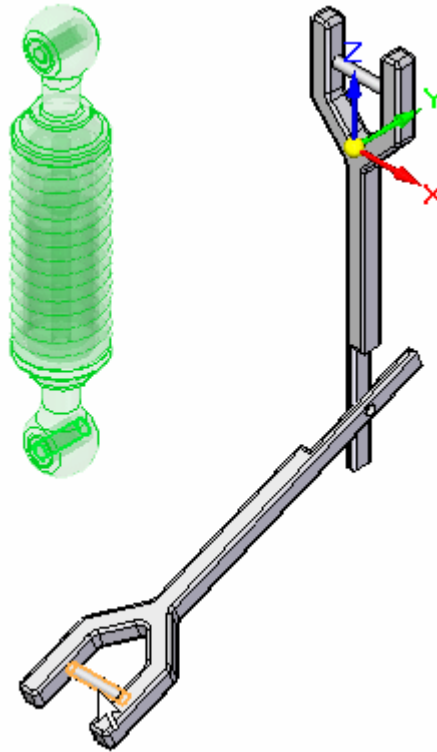
- ▶ Select the cylindrical shaft in *arm.par* as shown.



- ▶ For the next relationship select the cylinder in *sleeve.par* as shown.



- ▶ Select the cylindrical shaft in *arm.par* as shown.

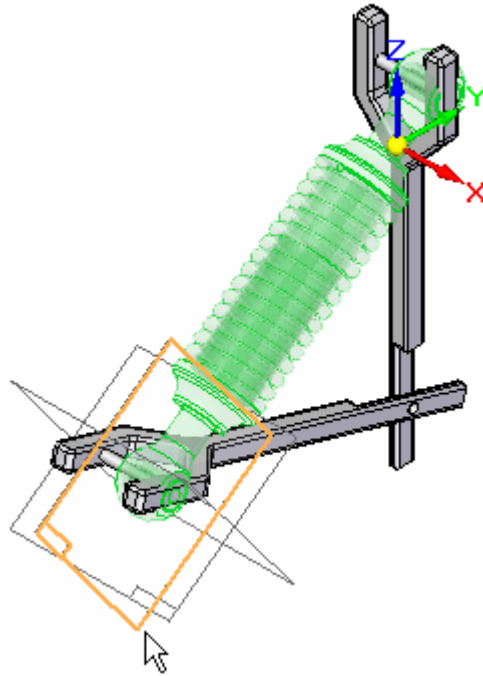


- ▶ Click the construction display to turn on the reference planes for *shock\_absorber1.asm*.

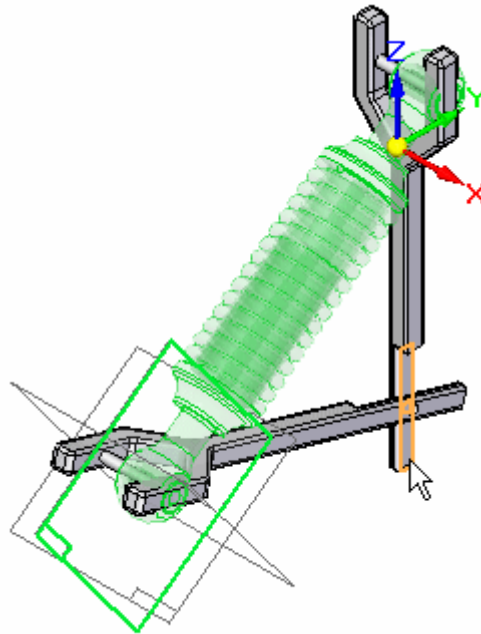




- Select the reference plane shown.



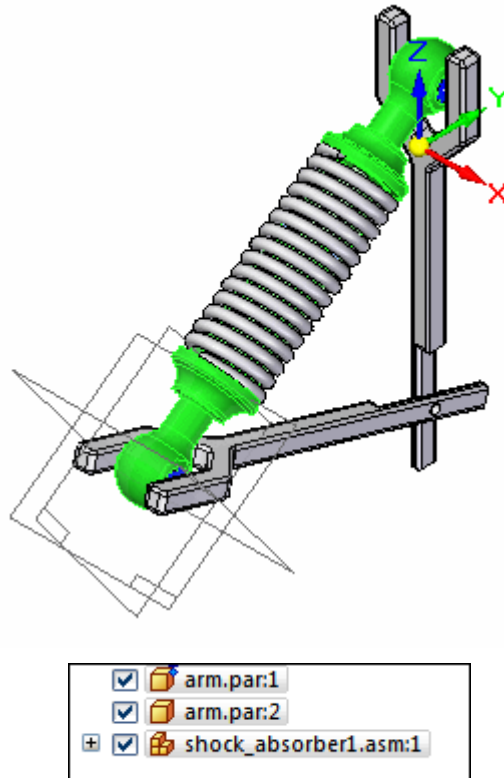
- Select the face shown in *arm.par*.



**Note**

The subassembly is placed and is fully constrained.

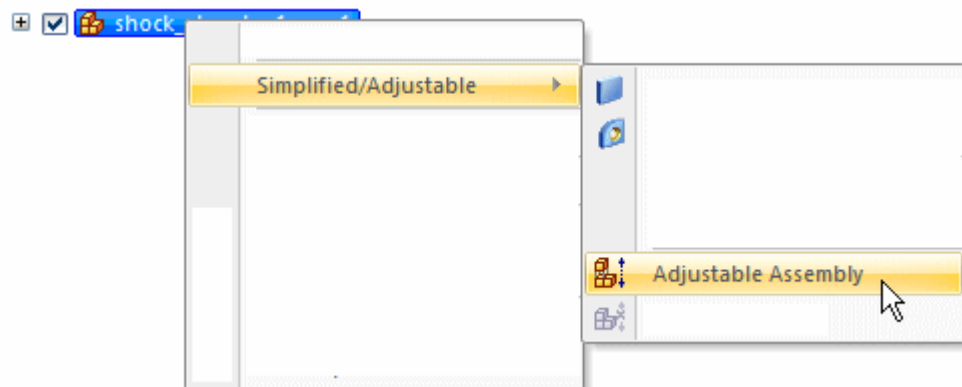
- ▶ Observe in pathfinder that all the parts of the assembly are fully positioned.



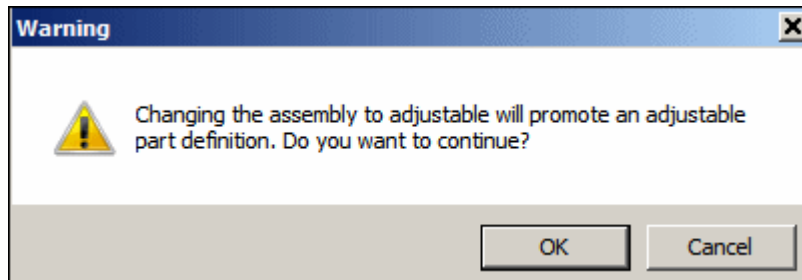
*Make the assembly adjustable.*

The assembly you will place will be defined as adjustable.

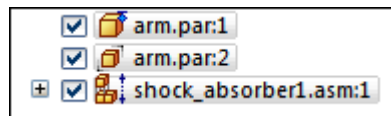
- ▶ Click the Select tool. In pathfinder, right click the subassembly *shock\_absorber1.asm*. Click Simplified/Adjustable then click Adjustable Assembly.



- ▶ Click OK to accept the warning message shown.



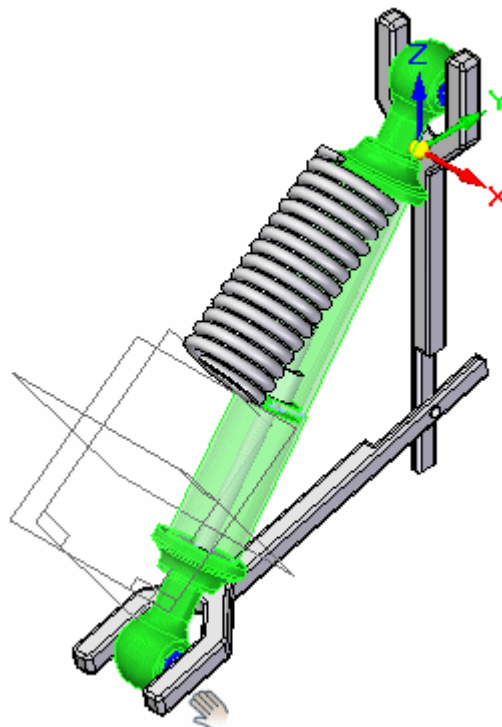
- ▶ Observe in pathfinder that all the parts of the assembly are not fully positioned. Because the assembly is adjustable, the arm has freedom to move.



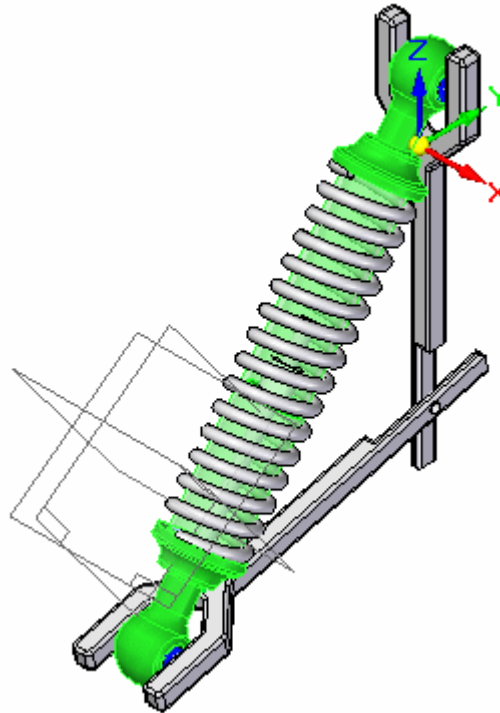
- ▶ Click the home tab. In the modify group, click the drag command.



- ▶ Drag *arm.par* as shown into different positions. Observe how the spacing between the cylinders adjusts and that the spring adjusts to the spacing defined by the new location of the arm.



- ▶ Drag the arm to several different positions and observe how the assembly adjusts.



**Note**

The assembly *shock\_absorber1.asm* has a mate relationship defined with a range offset. This limits the range of travel for the shock absorber.

- ▶ Save and close the assembly. This completes this activity.

**Note**

Motors defined in the top level of an assembly will move under constrained parts. If a subassembly contains a motor, the motor will not move the unconstrained parts unless the subassembly is made adjustable.

*Summary*

In this activity you placed an assembly with an adjustable part and defined the assembly as adjustable.

## **Lesson review**

Answer the following questions:

1. What are the characteristics of an adjustable assembly?
2. How do you prepare an assembly to be adjustable?
3. If a motor exists in a subassembly and you would like to have that motor control the position of under constrained parts, how could you do it?
4. When a subassembly contains an under constrained part and the subassembly is made adjustable, constraints to the under constrained part can be made in the higher level assembly. Where can you view and edit the relationships used to position the part?

## **Answers**

1. What are the characteristics of an adjustable assembly?

Specifying that a subassembly is adjustable allows you to place positioning relationships between parts in the subassembly while in the higher-level assembly. This is not possible with a rigid subassembly.

2. How do you prepare an assembly to be adjustable?

To use the Adjustable Assembly functionality, the subassembly should be left under-constrained in the range of motion in which you want to adjust. This allows you to apply the relationship(s) that you want to adjust in the higher level assembly, not in the subassembly.

3. If a motor exists in a subassembly and you would like to have that motor control the position of under constrained parts, how could you do it?

Make the subassembly with the motor adjustable.

4. When a subassembly contains an under constrained part and the subassembly is made adjustable, constraints to the under constrained part can be made in the higher level assembly. Where can you view and edit the relationships used to position the part?

You can view and edit the relationships of an under constrained part in an adjustable subassembly in the lower pane of pathfinder.

## **Lesson summary**

In this lesson you placed an assembly with an adjustable part and defined the assembly as adjustable.





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**Lesson**

# *14 Exploding assemblies*

## **Course overview**

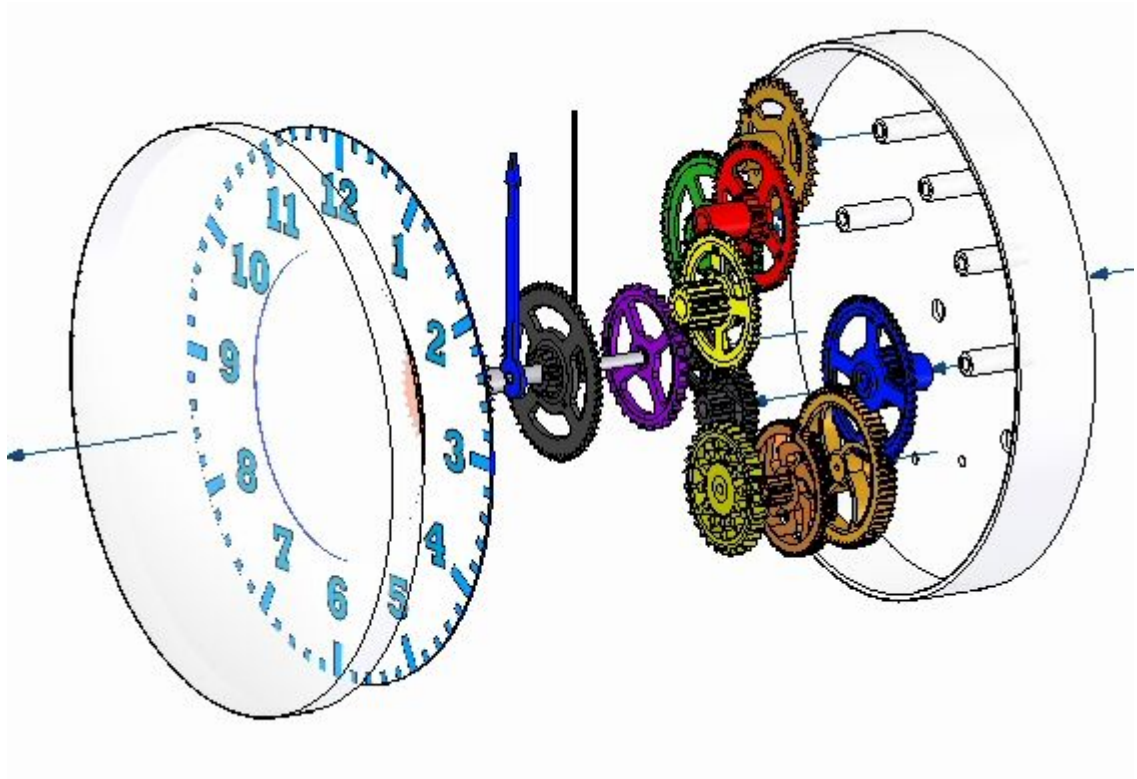
### **Course overview**

The Explode-Render-Animate application within the Solid Edge assembly environment is a tool for creating different types of presentations of Solid Edge assemblies. Exploding an assembly allows you to control the movement, sequence, and grouping of parts and subassemblies. Rendering a view allows you to define textures, lighting, shadows, backgrounds, and other properties to create presentation style images. Motors apply movement to under constrained parts in an assembly that can be animated. Using Animation, you can combine previously created exploding sequences and custom camera movement to create animation. Each frame of the animation can be rendered to create presentation quality animations.

Once you complete the activities in this course, you will be able to control the sequence and direction of explode events.

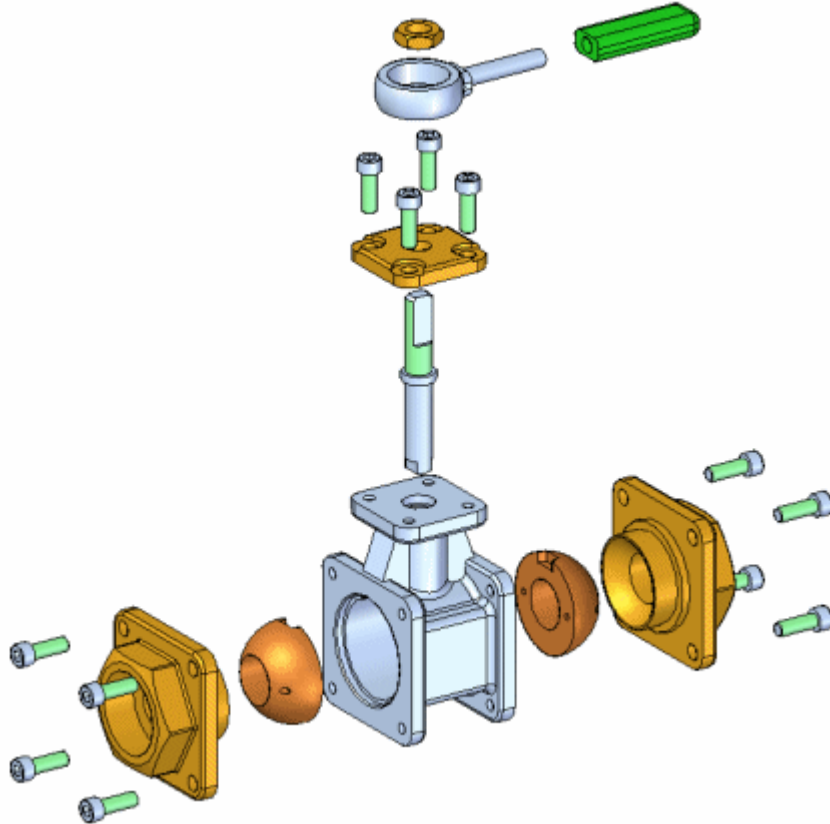
## Exploding assemblies

Solid Edge enables you to easily create exploded views of your assemblies. You can use the exploded views you define in the Assembly environment to create exploded assembly drawings in the Draft environment. You can also create presentation-quality renderings and animations of exploded assemblies.



## Creating Exploded Views of Assemblies

Solid Edge enables you to easily create exploded views of your assemblies. You can use the exploded views you define in the Assembly environment to create exploded assembly drawings in the Draft environment. You can also create presentation-quality renderings and animations of exploded assemblies.



To access the commands for creating assembly explosions, click **Explode-Render-Animate** on the **Tools** tab in the **Assembly** environment. A set of commands specifically tailored for working with explosions, renderings, and animations is displayed. You can use the menu commands and the **Explode PathFinder tab** shortcut menu to create, view, and edit exploded views of an assembly.

The operations you perform while creating an exploded view are captured as events and displayed in the **Explode PathFinder** tab on **PathFinder**. You can edit these events later.

When you have finished defining an exploded view, you can save the exploded view display configuration to a name you define.

## **Exploding Assemblies Automatically**

Many assemblies can be quickly exploded using the Automatic Explode command. You can use this command to explode all the parts in an assembly; or to explode only the parts in subassemblies you select.

When you explode selected subassemblies, you can select the subassemblies in the graphic window or the PathFinder.

The Automatic Explode command explodes assemblies based on the relationships applied between parts. In assemblies where the parts are positioned using mate or axial align relationships, the Automatic Explode command will quickly give you excellent results.

The Automatic Explode command cannot explode parts that are grounded. For example, when you create a new part within the context of the assembly using the Create In-Place option, the part is positioned using a ground relationship. You can use the Explode command to manually explode a grounded part, or you can delete the ground relationship, then position the part using assembly relationships, such as mate and align.

## **Exploding Assemblies Manually**

The Explode command gives you more control over assembly explosions than the Automatic Explode command. You should use Explode for assemblies where many of the parts were positioned without using mate or axial align relationships or when you want to explode the parts in a different direction than the Automatic Explode command uses.

The Explode command allows you to define an explode direction for one or more selected parts. You can select the parts in the graphic window or PathFinder.

When exploding parts manually, you first define the parts you want to explode, then select a base part and a face on the base part to define the explode direction.

You can also use the Explode command to edit explosions created with the Automatic Explode command.

## **Exploding Subassemblies**

If you want all the parts in a subassembly to remain a single unit (no offset distance is applied between the parts), there are two approaches you can use. If you want all the subassemblies in the assembly to remain a single unit, you can set the Bind All Subassemblies option available with the Automatic Explode command.

If you only want some of the subassemblies in the assembly to remain a single unit, you can use the Bind Subassembly command on the Home tab to specify that a selected subassembly remains a single unit. A symbol is added to PathFinder to indicate the subassembly is bound.

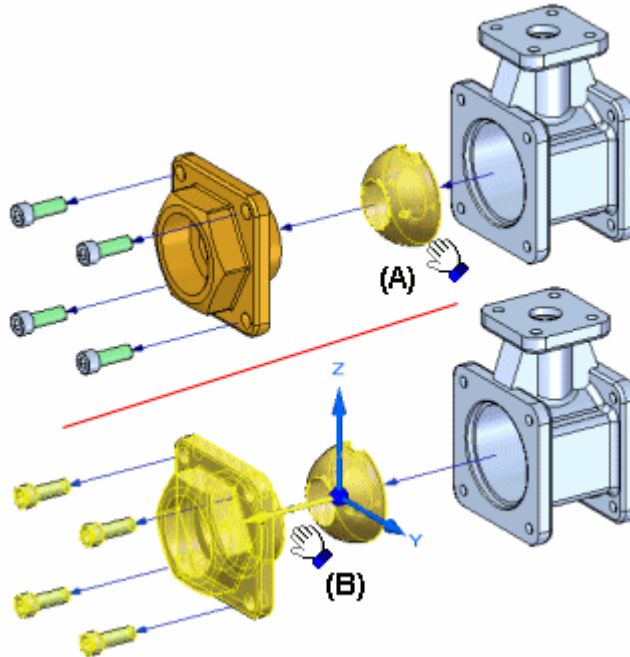
If you want to explode the subassembly later, you can use the Unbind Subassembly command to unbind the subassembly.

## **Modifying an Exploded Assembly**

You can use the other commands to modify the position and display of parts in your explosions.

### Moving and Offsetting Parts

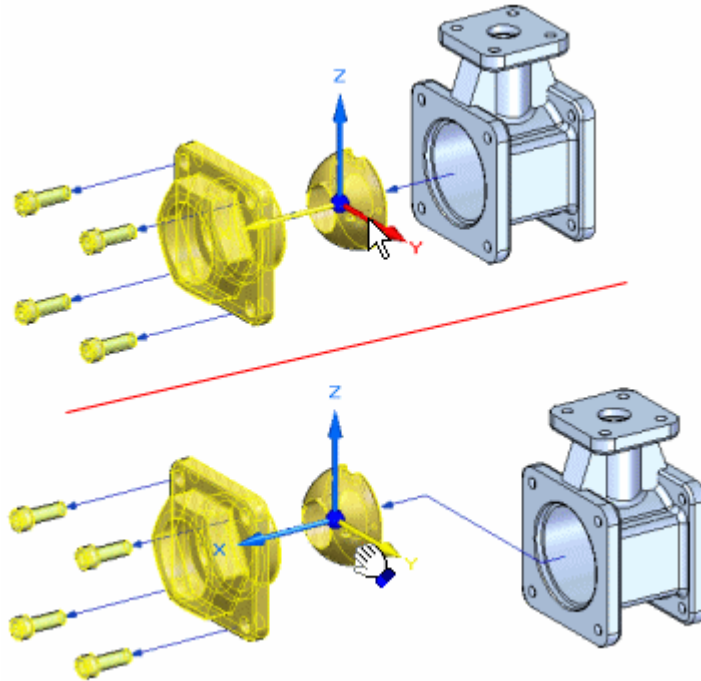
The Move Exploded Part command allows you to move or rotate one or more parts along the original explode vector, or along another vector you define. You can use the buttons on the command bar to move only the parts you select (A), or the selected part and its dependent parts (B).



#### Note

The Move Exploded Part command cannot reorder parts in an exploded view by moving a part past any adjacent parts.

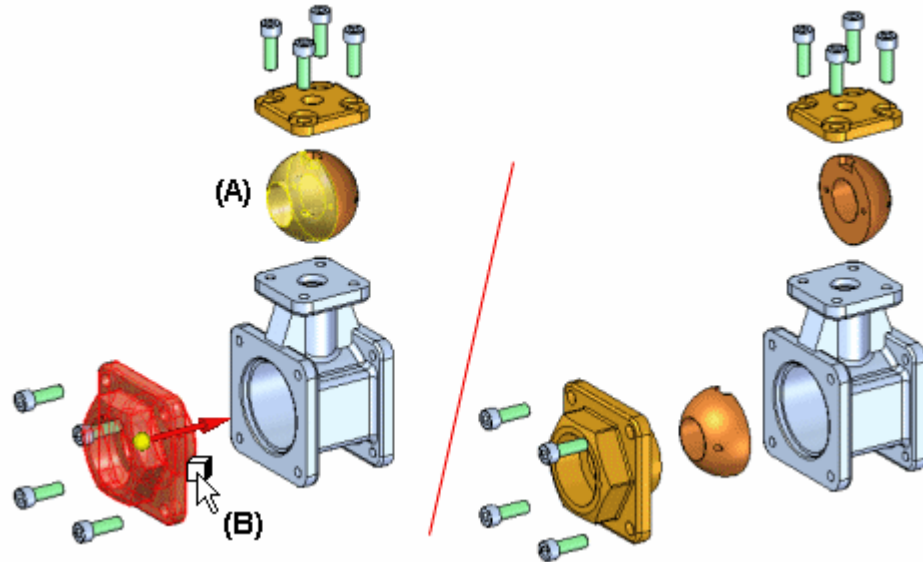
When you select a part to move, an orientation triad is displayed with the original explode vector axis highlighted. To offset the parts in a new direction, select one of the other axes, or use the options on the command bar to reorient the triad to define the vector you want.



When you offset the part in a new direction, a joggle is added to the part and a new explode event is added to the Explode PathFinder tab.

### Repositioning Parts

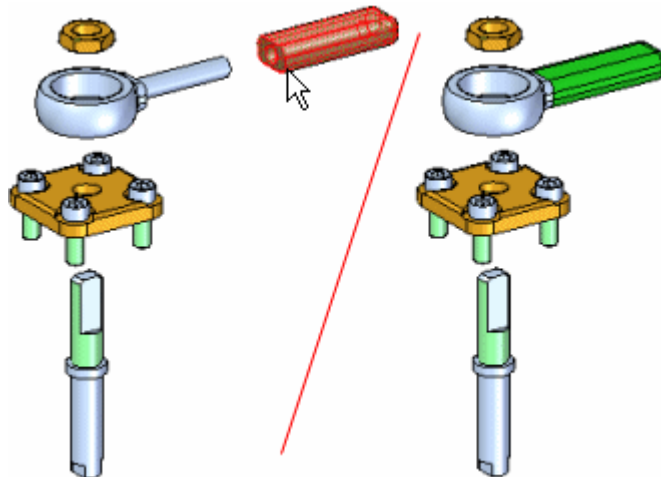
The Reposition command allows you to change the order of a part in an exploded view. To reposition a part, select the part you want to reposition (A), then highlight a reference part (B) in the explosion. An arrow is displayed to indicate where the part will be repositioned. If that is not the correct position, highlight a different reference part. You can reposition a part by placing it into a new position in its original explode vector or into the explode vector of another group of parts.



When you reposition a part, the spacing of adjacent parts is adjusted. You can also reposition all the parts in a bound subassembly.

### Collapsing Parts

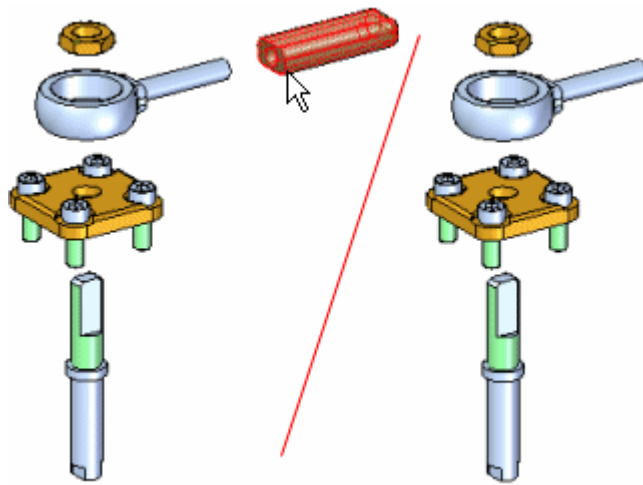
The Collapse command allows you to quickly return a part to its original assembly position relative to its parent part, but still show it in the exploded view.





### **Removing Parts**

The Remove command allows you to hide a part in the exploded view. When you remove a part, it is returned to its original, unexploded assembly position. You can re-display it using PathFinder.



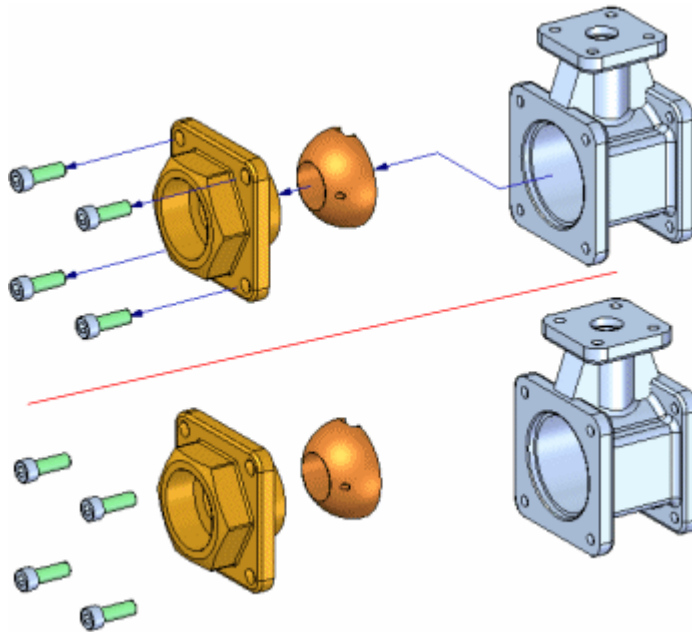
### Flow Lines

Flow lines are used between the parts in an exploded view to represent how the parts are related to one another. You control the display of flow lines and flow line terminators in an exploded view using the Flow Lines and Flow Line Terminators commands on the Home tab.

There are two types of flow lines, event flow lines and annotation flow lines. Event flow lines and annotation flow lines. Event flow lines are created using the explode commands and show the path the assembly components follow during an explode event in an animation.

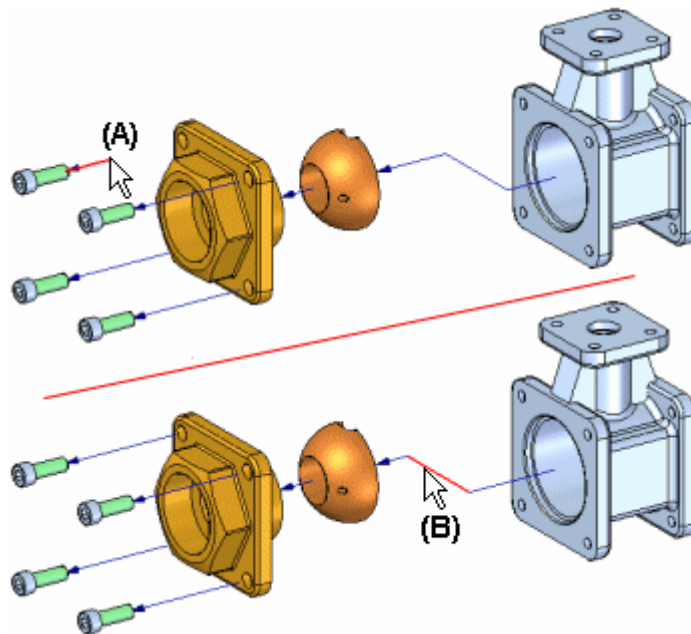
Annotation flow lines are used to create exploded views for draft documents. The Event flow lines can be dropped, which converts them into annotation flow lines, and creates an individual entry in the explode pathfinder for each flow line. Annotation flow lines can be added or modified to create the desired exploded view.

Once you have completed placing annotation flow lines on an exploded view, a display configuration must be saved in order to place the exploded view on a drawing sheet.



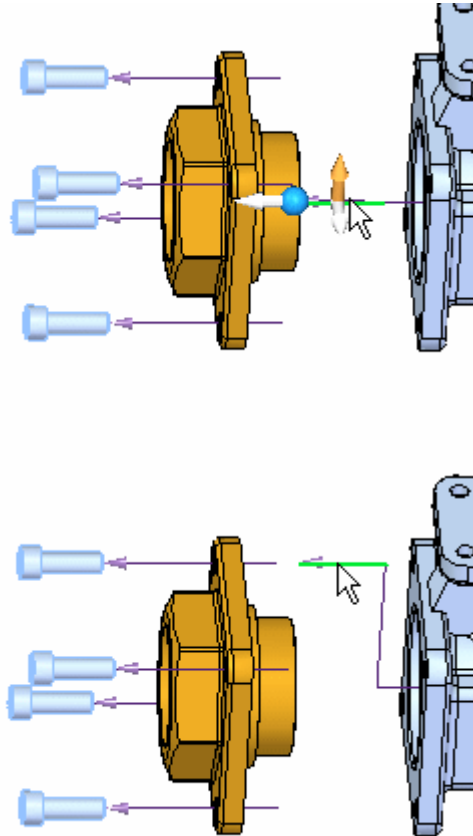
### **Editing Event Flow Lines**

You can edit the length of an end segment of a flow line (A), or the position of a joggle segment (B) of a flow line using the Modify command, or by using the drag component command.



### Editing Annotation Flow Lines

Annotation flow lines can also be edited with the Modify command. Annotation flow lines can be modified by dragging the handles and using key points on destination geometry to determine the length of the flow line.



### Displaying and Hiding Individual Flow Lines

You can also display or hide the flow line between two parts using the explode pathfinder to show or hide the flow lines. The flow line command will turn on or off the display of flow lines. The flow line terminator command will turn on or off the display of the arrowheads at the end of each flow line.

#### Note

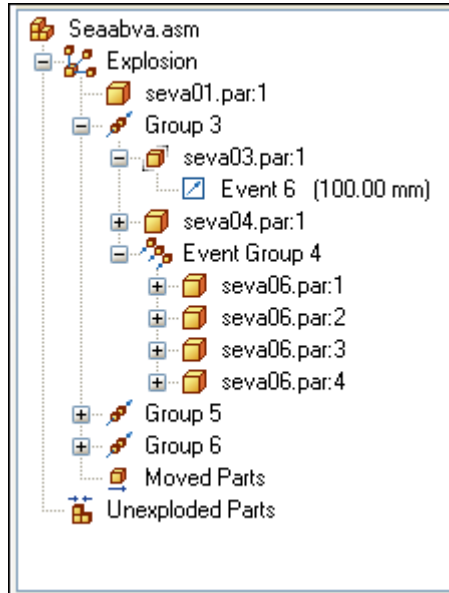
You can only select individual flow lines in the graphic window with the Edit Flow Lines command or in the Explode Pathfinder tab with the Select Tool.

### Deleting Flow Line Segments

For event flow lines, you cannot delete the first or last segments of a flow line, but you can delete the joggle segment of a flow line using the Explode PathFinder tab. For annotation flow lines, you can delete segments or complete flow lines using the modify command.

## Explode PathFinder Tab

The **Explode PathFinder tab** on PathFinder displays the structure for the current exploded view configuration in a hierarchical list. As discussed previously, the operations you perform while creating an exploded view are captured as events and displayed in the Explode PathFinder tab.



You can use the Explode PathFinder tab to review and modify explode operations. For example, you can move an explode Group to another position in the explode structure, add and remove parts from Event Groups, edit the linear or rotational offset values of an explode event and so forth.

## Editing Explosion Offset Distance and Angle

You can use the Select Tool to edit the offset distance or rotation angle for one or more explode events. You can select a single event in the Explode PathFinder tab, or multiple events by selecting one or more parts in the graphic window.

### Editing Single Events

When you select an explode event entry in the Explode PathFinder tab, the current offset distance or rotational angle is displayed on the command bar. You can type a new value to change the distance or angle.

### Editing Multiple Events for a Single Part

When a single part has multiple linear offset events, such as for a part with a joggled flow line, you can change all the linear distance values to a common value in one operation. Select the part in the graphic window, then type a value in the Distance box on the command bar. The Distance box on the command bar will be blank the first time you do this. Only linear distance events are recognized.

### Editing Multiple Events for Multiple Parts

You can also define a common linear offset value for multiple parts in one operation. Drag a box around the parts with the Select Tool, then type a value in the Distance box. Again, only linear distance events are recognized.

## **Saving Exploded Display Configurations**

You can use the Display Configurations command on the Home tab to save the display configuration of an exploded view so you can recall it later. When you save an exploded view configuration, the set of explode operations used to create the exploded view is captured and saved. When you activate an exploded view configuration, the Explode PathFinder tab updates to list the captured operations for the current configuration.

You can also use exploded display configurations when creating drawings and technical documents of exploded assemblies, and when creating animations of exploded assemblies using the Animation Editor command on the Home tab.

## **Creating Multiple Exploded Views**

If you need to create several drawings or animations of the same assembly, but with different parts displayed or the parts displayed in different positions, you can save additional exploded display configurations. After you have saved an exploded display configuration, you can use the Unexplode command on the Edit menu to reassemble the parts so you can start a new exploded view.

## **Using Assembly Display Configurations**

You can use display configurations of regular assembly windows to control the display status of parts in an exploded view. For example, when you apply an assembly display configuration which has a hidden subassembly, the subassembly is also hidden in the exploded view. The exploded positions of the parts are not changed.

## **Animating Exploded Views**

You can use the Animation Editor command to create an animation of an exploded view. The Animation Editor tool has options that allow you to specify the explode configuration, initial state, speed, and animation order. For more information, see the Creating Assembly Animations Help topic.

## **Creating Drawings of Exploded Views**

When creating a drawing of an assembly in the Draft environment, you can specify an exploded view configuration name on the Drawing View Wizard to create a drawing view of an assembly explosion. You can also use the Create Drawing command on the Application menu under the New command in the main Assembly environment to create a drawing of an assembly explosion.

### **Note**

The Create Drawing command is not available when working in the Explode-Render-Animate application.

## **Flow Lines in Exploded Assembly Drawings**

When you create drawings of assembly explosions, the flow lines are automatically displayed in the drawing views. You can specify whether flow lines are displayed in a drawing view using the Annotation tab on the Drawing View Properties dialog box. You can also modify the display of the flow lines using the Draw in View command on the shortcut menu.

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## **Overview**

- Exploding, rendering, and animating assemblies
- Creating assembly animations
- Drawing Production Overview
- [PathFinder in assemblies](#)

## **Commands**

- View Wizard command
- [Animation Editor command](#)
- [Automatic Explode command](#)
- [Bind Subassembly command](#)
- [Collapse command](#)
- [Display Configurations command](#)
- [Modify command](#)
- [Explode command](#)
- [Move Part command](#)
- [Remove command \(Explode application\)](#)
- [Reposition command](#)
- [Unbind Subassembly command](#)
- [Unexplode command](#)
- [Flow Lines command](#)

## **Procedures**

- Collapse a part in an exploded assembly
- Explode an assembly automatically
- Explode individual parts in an assembly
- Remove a part from an exploded assembly
- Reposition a Part in an Exploded Assembly
- Unexplode an Assembly

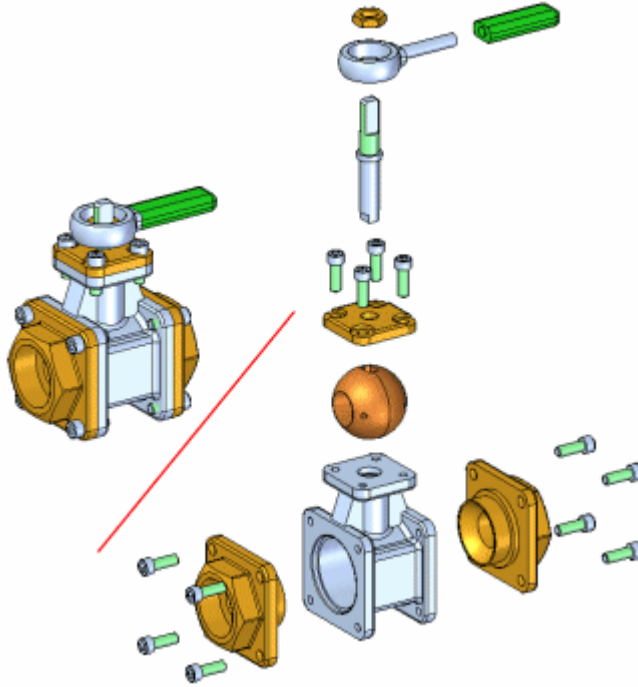
**User Interface**

- [Explode Pathfinder tab](#)



## Automatic Explode command

Explodes the active assembly by applying a spread distance between parts.



The Automatic Explode command explodes assemblies based on the relationships applied between parts. In assemblies where the components are positioned using mate or axial align relationships, the Automatic Explode command quickly gives you excellent results.

### Note

You cannot use this command to explode grounded parts or pipe components.

### Steps

The basic steps for automatically exploding an assembly are:

- Specifying the components to explode.
- Defining the explode settings.

#### Specifying the components

You can use the Select option on the command bar to specify whether all the parts and subassemblies in the assembly are exploded or only the subassemblies you select are exploded. When you explode selected subassemblies, you can select the subassemblies in the graphics window or the Assembly PathFinder tab.

#### Defining the explode settings

You can use the options on the command bar and the Automatic Explode Options dialog box to specify how the components are exploded. For example, you can specify whether the spread distance between parts is calculated automatically, or you can specify the spread distance yourself.

When exploding an assembly that contains subassemblies, you can specify how the parts in the subassemblies are exploded using the Automatic Explode Options dialog box.

### Calculating spread distance

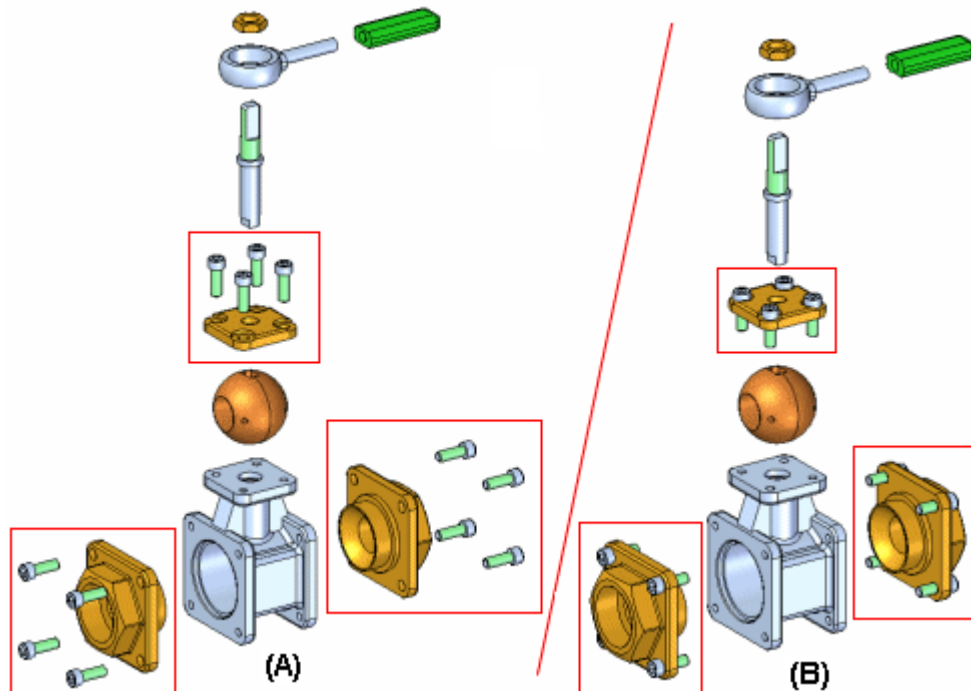
The Automatic Spread Distance button on the command bar allows you to specify whether the spread distance between parts is calculated automatically by the Automatic Explode command, or that you want to specify the spread distance yourself.

- To have the spread distance calculated automatically, set the Automatic Spread Distance option.
- To specify the spread distance yourself, clear the Automatic Spread Distance option, then type the value you want in the Distance box.

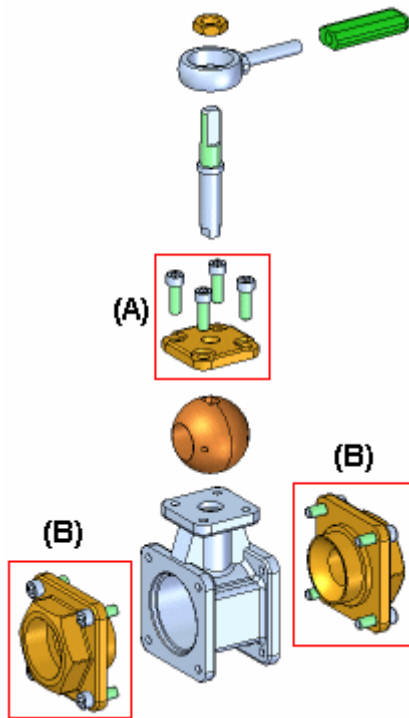
When defining the spread distance yourself, you can type the value you want, then click the Explode button to see the result. To try another spread distance, type a new value, then click the Explode button again.

### Binding subassemblies

When using the Automatic Explode command on an assembly that contains subassemblies, you can specify whether the parts in subassemblies are exploded (A) or the parts in subassemblies are grouped together as a single unit (B). To keep the parts in the subassemblies grouped together, set the Bind All Subassemblies option on the Automatic Explode Options dialog box.



If you want to explode the parts in some subassemblies (A), but not others (B), you can use the Bind Subassembly command to select the subassemblies you want to remain a single unit when exploded.



First, select the subassemblies in Assembly PathFinder, then click the Bind Subassembly command. A symbol is added adjacent to the subassembly entry in Assembly PathFinder to indicate that the subassembly is bound.

You can then clear the Bind All Subassemblies option on the Automatic Explode Options dialog box, and only the bound subassemblies you selected remain a single unit when you complete the command.

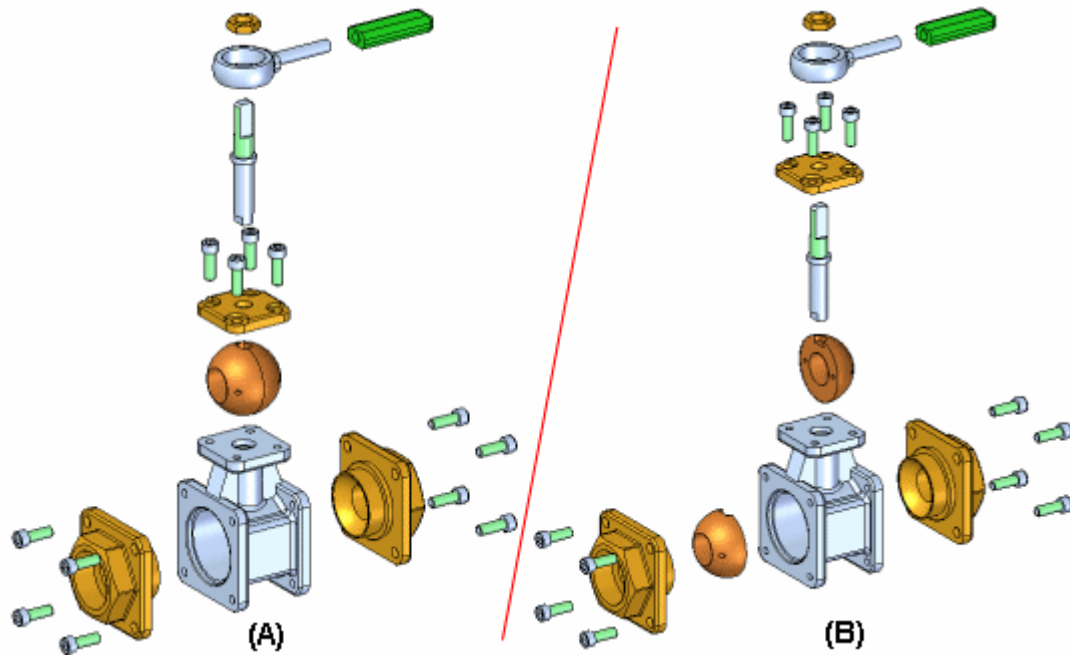
If you want to explode the subassembly later, you can use the Unbind Subassembly command to unbind the subassembly.

### Explode technique

The Explode Technique option on the Automatic Explode Options dialog box allows you to specify whether the subassembly is considered or ignored when creating the explosion.

The By Subassembly Level option specifies that each subassembly is considered as a unique explosion. This keeps the parts in a subassembly adjacent to one another as they are exploded (A).

The By Individual Part option specifies that the subassembly structure is ignored when the parts are exploded. The parts are exploded based on their proximity to one another. This can result in parts in separate subassemblies being intermingled with one another. (B) This option duplicates the behavior used prior to version 19 of Solid Edge.



### Explode PathFinder tab

The Explode PathFinder tab on PathFinder lists the explode operations in the order you perform them. You can use the Explode PathFinder tab to review and modify explode operations. When you save explode configurations, each explode configuration captures a separate set of explode operations. When you activate an explode configuration, the Explode PathFinder tab updates to list the operations captured for that configuration.

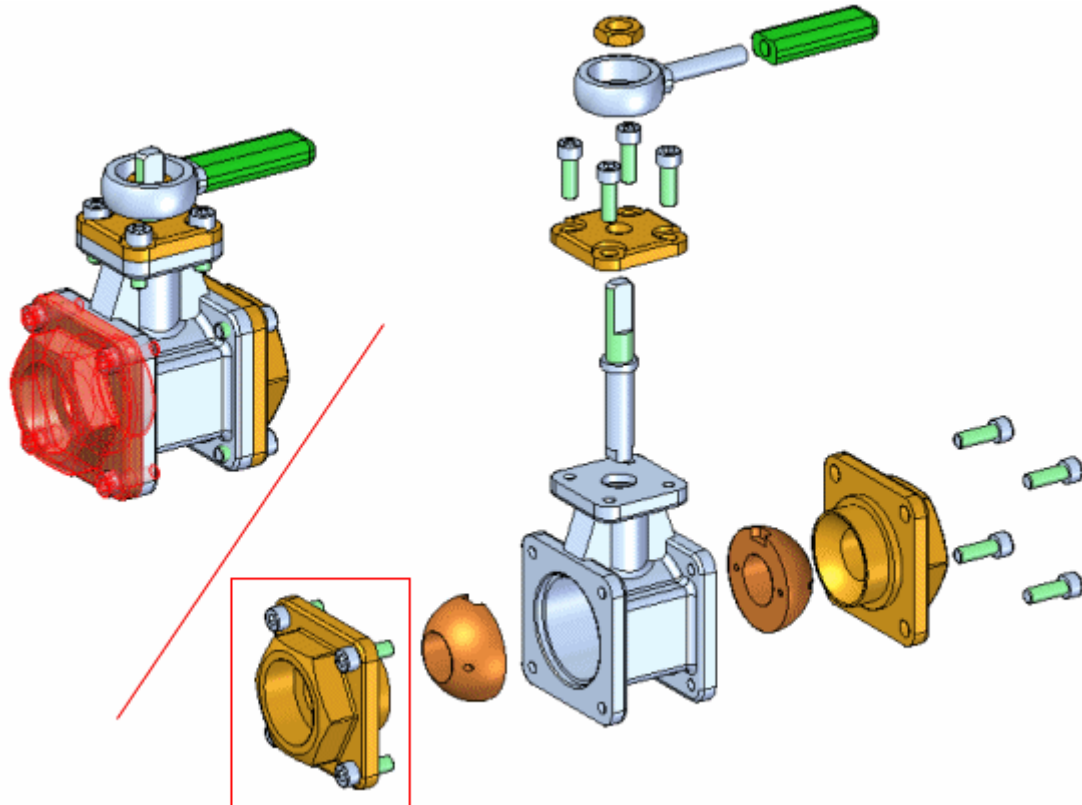
### Grounded parts

The Automatic Explode command cannot explode parts that are grounded. For example, when you create a new part within the context of the assembly using the Create In-Place option, the part is positioned using a ground relationship. You can use the Explode command to manually explode a grounded part, or you can delete the ground relationship, then position the part using assembly relationships, such as mate and align.

 **Bind Subassembly command**

Groups the parts in a subassembly so they will explode as a single unit when using the Automatic Explode or Explode commands. To bind a subassembly, you must first select it using PathFinder.

A symbol is added adjacent to the subassembly entry in PathFinder to indicate that the subassembly is bound. You can use the Unbind Subassembly command to unbind a subassembly.



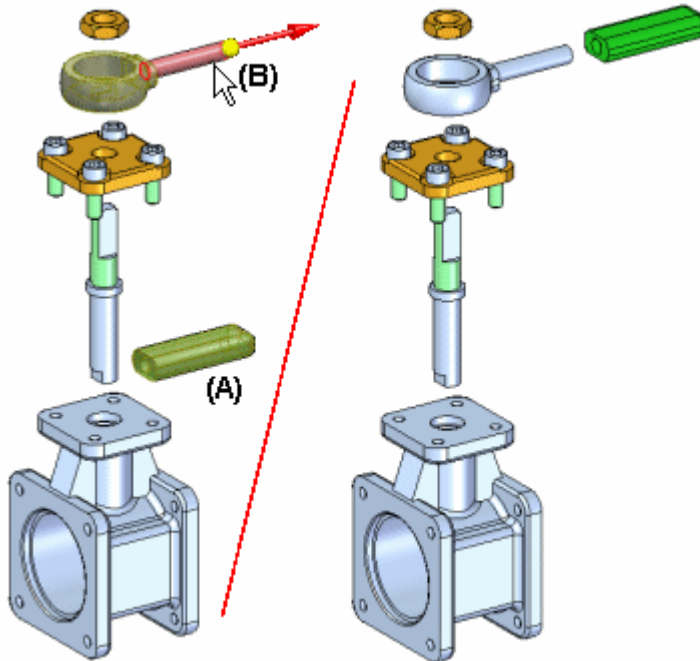


### **Unbind Subassembly command**

Ungroups a subassembly that was grouped using the Bind Subassembly command.

## Explode command

Explodes one or more parts in a specified direction. The parts (A) selected for explosion are offset along an explode vector you define by selecting a face or reference plane on a base or reference part (B).



You can manually explode a single part, multiple parts, and bound subassemblies. You define the offset distance using the Distance box on the command bar.

### Note

Parts that were exploded with the Automatic Explode command can be re-exploded along a different explode vector using this command.

## Steps

The basic steps for manually exploding parts are:

- Select the parts to explode.
- Select the base part.
- Select a face or reference plane on the base part.
- Specify the explode direction.

## Exploding Multiple Parts

When you select multiple parts to explode in one operation, you can use the Manual Explode Options dialog box to specify whether the parts are spread out evenly, or that the parts are moved as a single unit.

### Spreading Components Evenly

When you set the Spread Components Evenly option, you can use the Manual Explode Options dialog box to define the explode order you want. You can select one or more parts in the Explode Order list, then use the Move Up and Move Down buttons on the dialog box to define the explode order you want. When you select a part in the list, it highlights in the graphic window. This allows you to reorder the parts as you explode them.

### **Moving as a Single Unit**

When you set the Move Components as a Single Unit option, the current relative position of the set of parts is maintained, and they are relocated along the explode vector you defined.

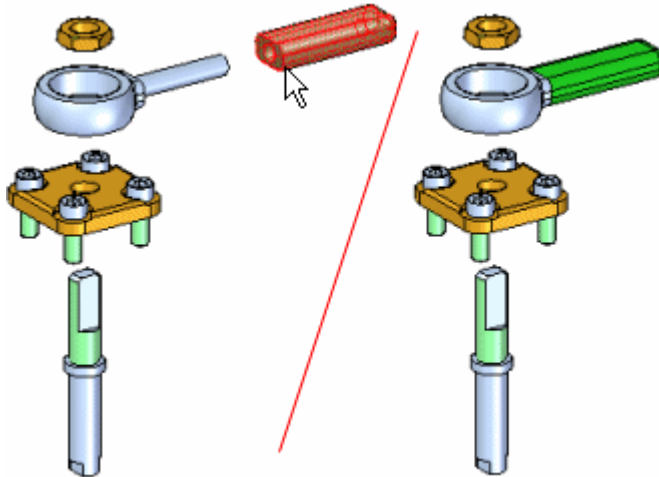
### **Exploding Bound Subassemblies**

When you select a bound subassembly to explode, it is exploded as a single unit. In other words, the components within the subassembly are not spread out.



 **Collapse command**

Returns an exploded part to its original assembly position relative to its parent part.

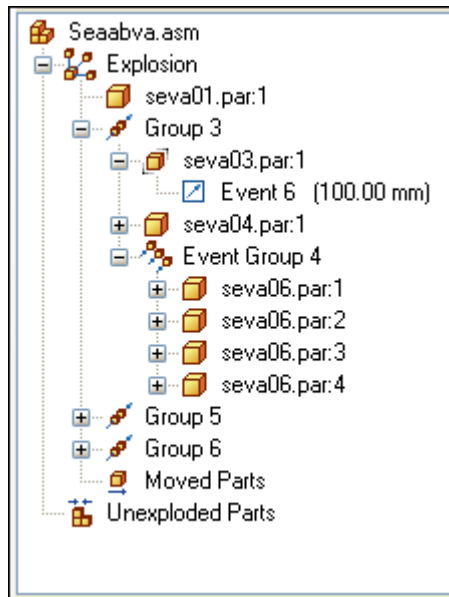


You can collapse several parts in one operation by holding the Shift key and selecting the parts you want to collapse. If you select a part that is a component in a bound subassembly, the entire subassembly is collapsed.

When you collapse a part, the flow line for the part is deleted.

## Explode PathFinder tab

Provides alternate ways of viewing and editing an exploded assembly. The Explode PathFinder tab displays the structure for the current exploded view configuration in a hierarchical list. The Explode PathFinder tab helps you work with the components that make up an exploded view.



This allows you to view the structure and perform edit operations on the current exploded view configuration. Some of the operations you can perform include:

- You can select an explode event, then edit the offset or rotational value using the command bar.
- You can use the commands on the shortcut menu to show and hide parts, collapse parts, show and hide flow lines, and so forth.
- You can add and remove parts from explode Groups and Event Groups. This is useful when working with animations.

The following table explains the symbols used in Explode PathFinder:

### Legend

	Part
	Assembly
	Group
	Event Group
	Linear Event
	Rotational Event
	Moved Parts
	Unexploded Parts

## Explode groups and event groups

The way the parts and subassemblies are arranged in the Explode PathFinder tab is based on the operations you performed to create the exploded view. These operations, or events, are collected into one or more explode operation collections in Explode PathFinder.

There are two types of explode operation collections: Groups and Event Groups. The groups and event groups are created automatically as part of the explosion creation process, but you can edit them later.

### Groups

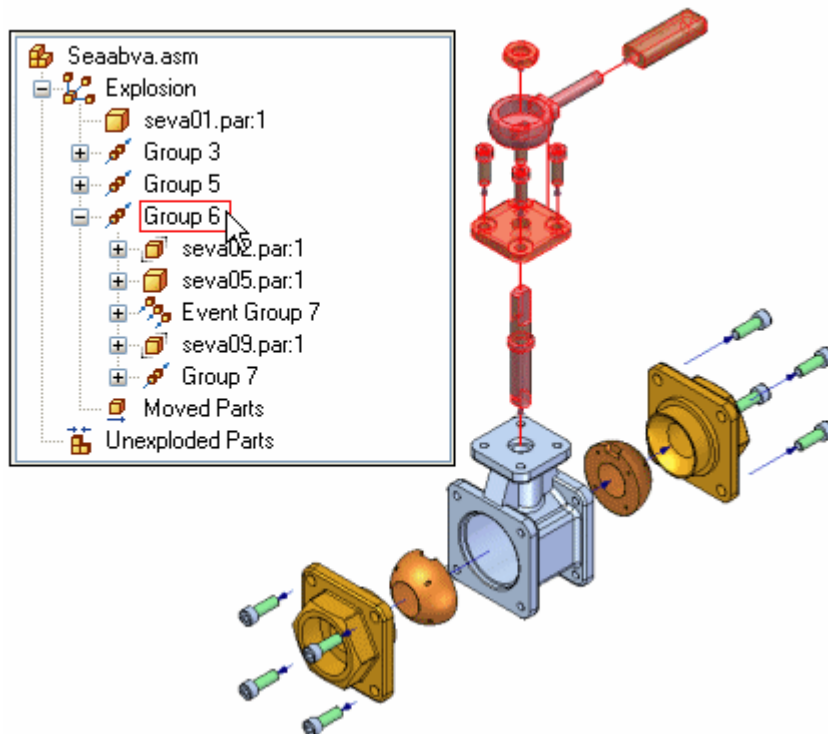
A Group collects all the parts and subassemblies that participate in a common explode vector. It is possible for a group collector to have additional groups nested within it. For example, if one part in the explode vector has parts branching off into a different explode vector, the branched parts will be in a group nested within the main group. The components within a group animate sequentially.

### Event Groups

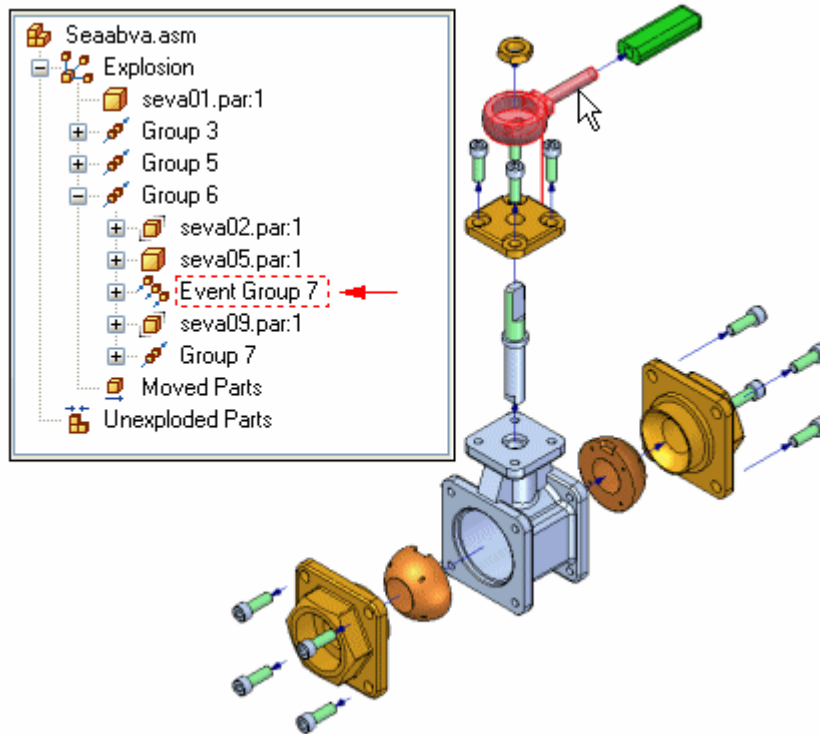
An Event Group collects all the components that will move simultaneously in an animation. For example, a pattern of fasteners would typically be in an Event Group. You can add parts to and remove parts from an event group.

## Explode PathFinder and the graphics window

Similar to the PathFinder tab in an assembly, when you select an entry in Explode PathFinder, the associated parts highlight in the graphics window. For example, if you select a Group entry in Explode PathFinder, the parts associated with that entry highlight in the graphic window.



If you select a part in the graphics window, a box is displayed in Explode PathFinder to indicate where in the exploded view tree structure the selected part is located.



### Explode configurations and explode events

When you save an exploded view configuration, each explode configuration captures a separate set of explode events. When you activate an exploded view configuration, the Explode PathFinder tab updates to list the events captured for the current configuration.

### Adding and removing parts from event groups

You can add and remove parts from an Event Group to control which parts move simultaneously during an assembly animation. The Add To Event Group and Remove From Event Group shortcut menu commands in the Explode PathFinder tab allow you to make these types of changes. For example, you may want all the fasteners in an assembly to animate at one time, even though the results of the Automatic Explode command placed them in different explode event groups.

#### Note

The Add To Event Group and Remove From Event Group commands are not available when the Animation Editor is displayed.

For more information on assembly animations, see the Creating Assembly Animations Help topic.

## **Reordering groups**

You can change the order of an explode Group, but not an Event Group using the Explode PathFinder tab. This allows you to change the sequential order in which the parts move in an assembly animation.

Use the Select Tool to drag and drop an explode Group to a different location. Explode PathFinder displays a symbol to show where you can reposition the Group in the explode structure. The symbol changes if you drag the Group to an invalid location in the explode structure.



## Display Configurations command

Saves, applies, or deletes a display configuration of an assembly. A display configuration captures the display status of the parts, assemblies, assembly sketches, weld beads, and reference planes in an assembly.

Using display configurations can be useful when working with large assemblies.

To apply a saved configuration, you can also use the Configuration drop list on the Home tab® Configurations group.

You can apply display configurations that were defined in the current assembly, or display configurations that were defined in a subassembly. To apply a display configuration defined in a subassembly, you must select the subassembly first.

For more information, see the [Using display configurations](#) and [Working with large assemblies efficiently](#) Help topics.

## **Display Configuration Options dialog box**

### **Fit View After Apply**

Fits the view after the display configuration is applied.

List exploded configuration in modeling mode when applied.

Options for displaying exploded configurations outside of Explode Render Animate environment:

### **Prompt For Action**

Gives you the choice to apply display settings only, or enter the ERA environment.

### **Apply Display Settings Only**

Applies the display settings which are not unique to the ERA environment without entering ERA.

### **Switch to ERA and Explode**

Switches to the ERA environment and applies the saved display configuration settings.

### **Use Single Default Configuration**

Creates a Solid Edge default configuration rather than a default configuration associated with a user name.

### **Apply Activation Override**

Allows you to specify how you want to apply the display configuration. When this option is set, you can specify whether the parts in the assembly are activated or inactivated when you apply the display configuration. When this option is cleared, the active or inactive status of the parts is determined by their status when the configuration was saved.

#### **Activate All Parts**

Specifies that all parts in the assembly become active when you apply the configuration.

#### **Inactivate All Parts**

Specifies that all parts in the assembly become inactive when you apply the configuration.

#### **Maintain Current State**

Specifies that the current activation state (inactive or active) of the parts in the assembly is not changed when you apply a display configuration. Setting this option improves performance when working with large assemblies.

When you apply a display configuration from the Select Tool command bar, the current dialog box setting is honored. If you want to change the current activation or override setting when applying a display configuration, you must apply the configuration using the Display Configurations dialog box.

### **Apply Simplify Override**

Allows you to specify how you want to apply the display configuration. When this option is set, you can specify whether the parts in the assembly are simplified or designed when you apply the display configuration. When this option is cleared,

the simplified or designed status of the parts is determined by their status when the configuration was saved.

**Use All Simplified Parts (When Available)**

Specifies that the simplified version of parts is used when you apply the configuration.

**Use All Design Parts**

Specifies that the designed version of parts is used when you apply the configuration.

**Maintain Current State**

Specifies that the current simplified state (simplified or as designed) of the parts in the assembly is not changed when you apply a display configuration. Setting this option improves performance when working with large assemblies.

When you apply a display configuration from the Select Tool command bar, the current dialog box setting is honored. If you want to change the current activation or override setting when applying a display configuration, you must apply the configuration using the Display Configurations dialog box.



 **Move Part command**

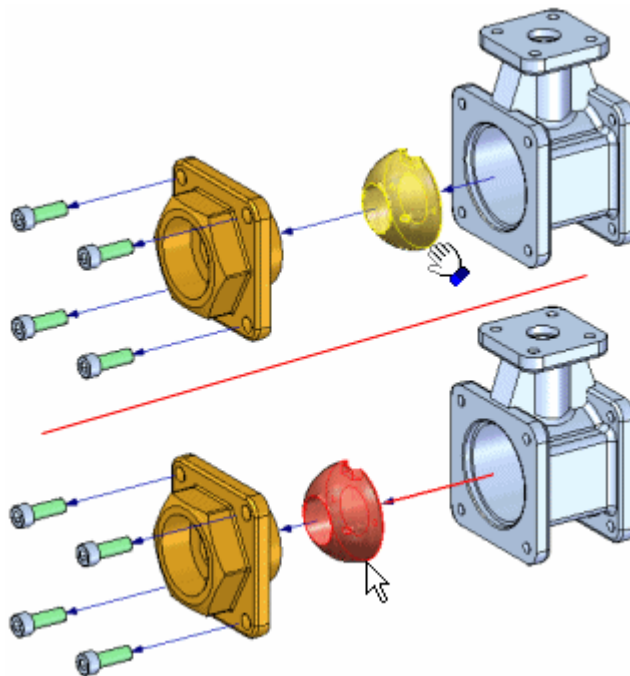
Moves or rotates parts in an exploded view in an assembly. You can use this command to do the following:

- Move one or more parts along the original explode vector or a new vector you define.
- Rotate one or more parts along the original explode vector or a new vector you define.
- Move one or more parts within a plane you define.

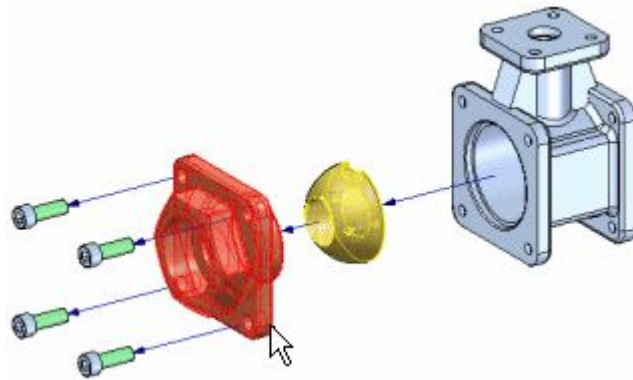
You can move or rotate a single part, a set of parts, or a part and all its dependent parts. The same basic steps apply whether you are moving, rotating, or moving within a plane.

**Moving parts along the original explode vector**

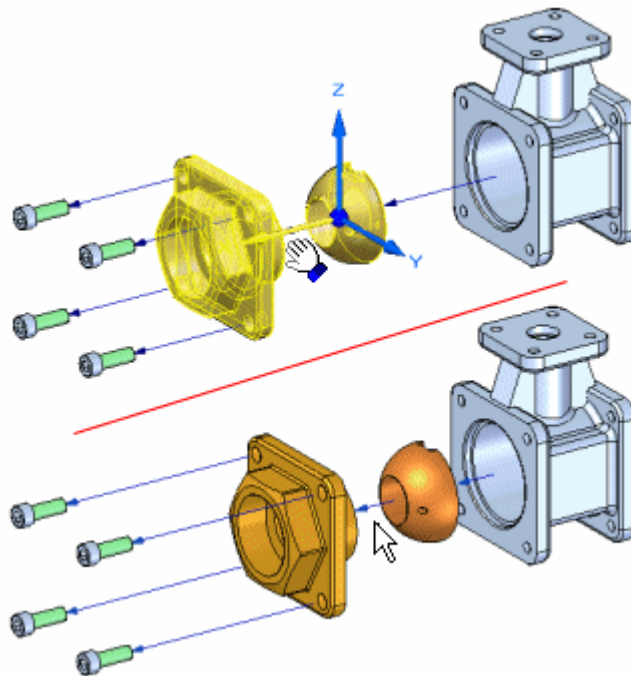
- To move a single part: select the part, then drag it to the new location.



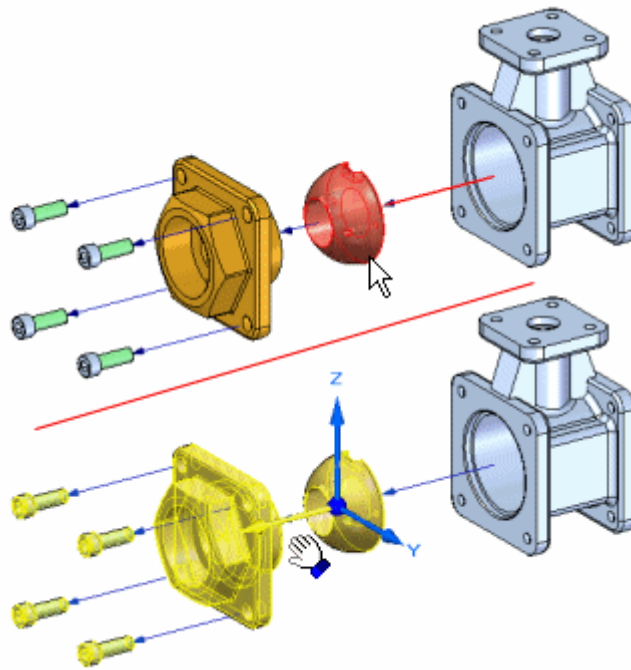
- To move a set of parts: select the parts, then click the Accept (check mark) button on the command bar.



When you click the Accept button, an orientation triad is displayed, with the X-axis oriented to the explode vector. By default, the original explode vector axis is selected. Drag the cursor to move the set of parts to the new location.

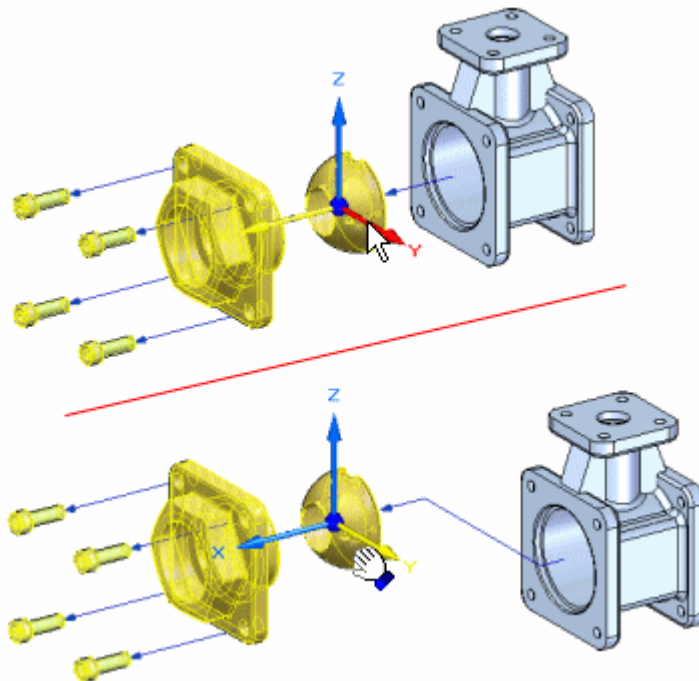


- To move a part and all its dependent parts: set the Move Dependent Parts option on the command bar, then select the part you want to move. The part and all its dependent parts highlight. When you click the Accept button, an orientation triad is displayed. By default, the original explode vector axis is selected. Drag the cursor to move the set of parts to the new location.



### Moving parts along a different explode vector

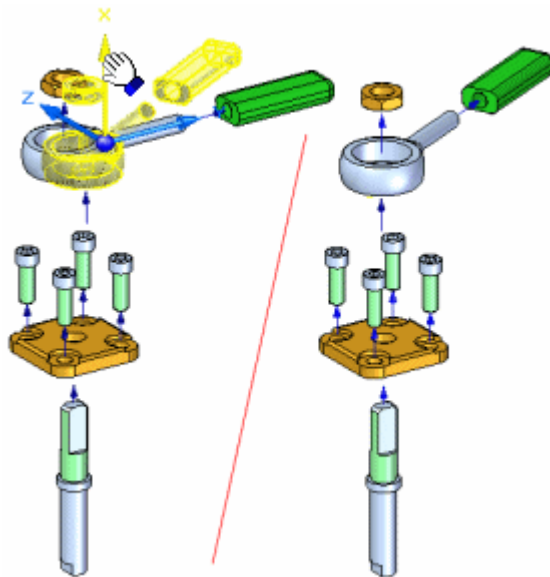
To move one or more parts along a different explode vector, first define the select set of parts. For example, you can use the command bar to specify that you want to move a part and all its dependent parts. After you click the Accept button, position the cursor over the triad axis you want to move along, then drag the cursor. A joggle is added to the flow line automatically. When you offset the part in a new direction, a new explode event is added to the [Explode PathFinder tab](#).



When parts have been moved outside of their original explode vector, you can also move them back into their original explode vector. Select the parts you want to move, select the proper axis, then drag the parts back to the original explode vector. When you get close to the original explode vector, the parts will automatically lock into the original vector. The joggle segment will be automatically removed.

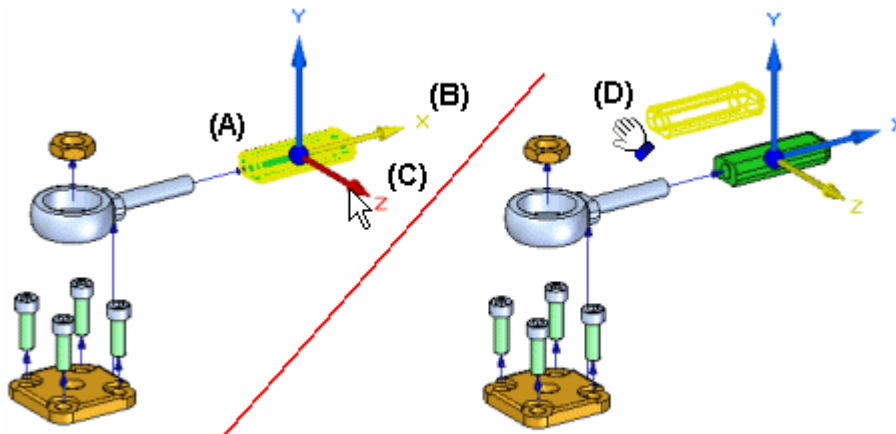
### Rotating parts

To rotate one more parts, first set the Rotate option on the command bar. Define the select set of parts, then position the cursor over the axis you want to rotate about, and then drag the cursor to the new location.

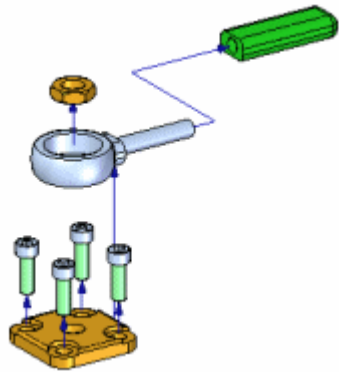


### Moving parts within a plane

To move one or more parts within a plane, first set the Move Planar option on the command bar. Define the parts you want to move (A), then define the plane you want to move the parts within. The movement plane is defined by the X axis (B) and another axis you select. For example, you can move a part within the plane defined by the X axis and Z axis (C). Then drag the cursor to the new location (D).



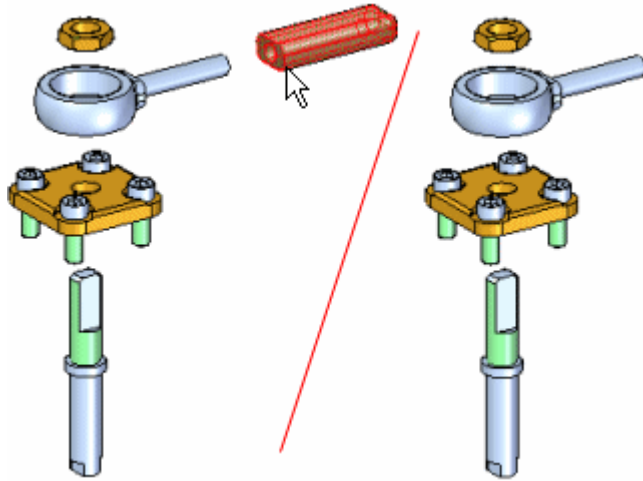
With the Move Planar option, parts are typically moved outside of the original explode vector axis. If so, a joggle is added to the flow line.



For additional information on working with exploded views, see the [Creating Exploded Views of Assemblies](#) and [Explode PathFinder tab](#) Help topics.

 **Remove command (Explode application)**

Hides the selected part in the exploded view and returns the part to its unexploded assembly position.



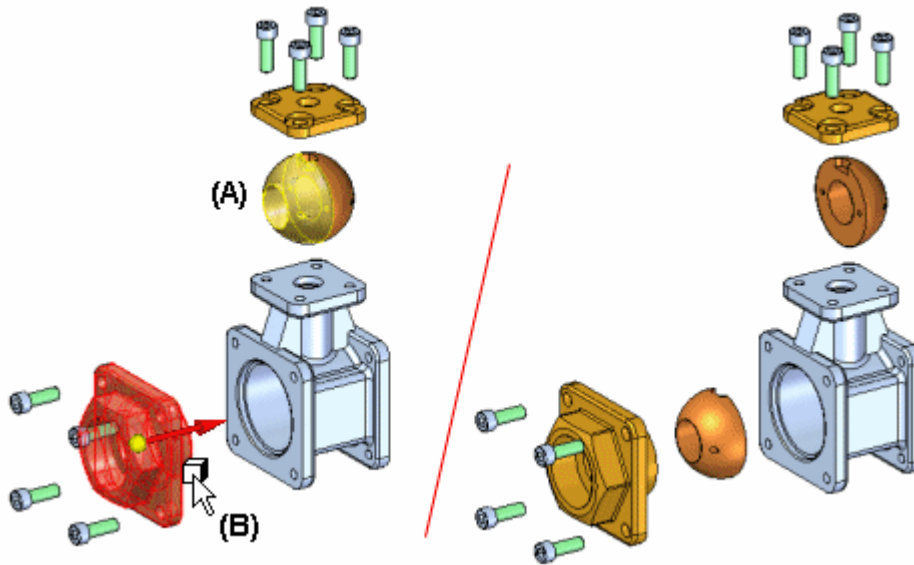
You can remove several parts in one operation by holding the Shift key and selecting the parts you want to remove. If you select a part that is a component in a bound subassembly, the entire subassembly is removed.

You can use the PathFinder tab to re-display the parts later.

## **Reposition command**

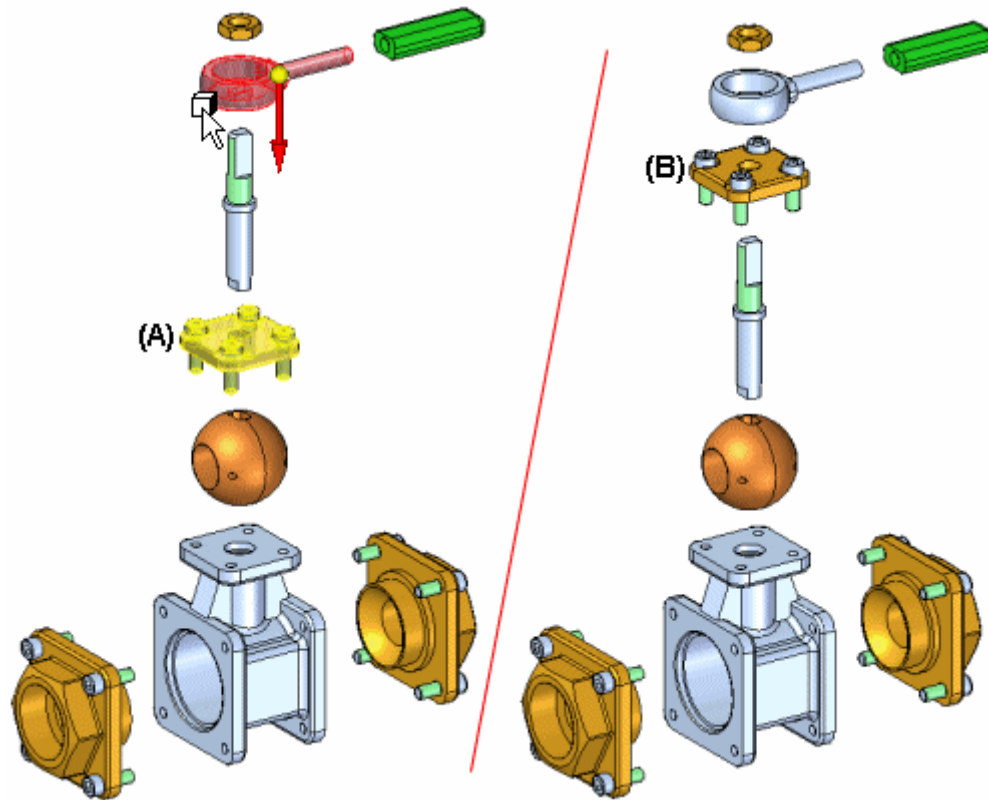
Repositions a part with respect to another reference part in an exploded view. This can be useful when you want to change the position of a part after using the Automatic Explode command.

To reposition a part, select the part you want to reposition (A), then position the cursor over the reference part (B). The reference part highlights, and an arrow is displayed on the reference part to indicate which side of the reference part the selected part will be repositioned to.



If the arrow does not point in the direction you want, highlight a different reference part. If the reference part is the last part in an explode vector sequence, a dynamic arrow is displayed to indicate that you can choose the side of reference part where you want to reposition the selected part.

If the part you select is within a bound subassembly (A), all the parts in the subassembly will be repositioned (B).



You use this command to change the order in which the parts are exploded. To relocate a part without changing its order, you can use the Move Exploded Part command.



## **Explosion Properties dialog box**

### **Configuration**

Specifies which explode configuration you want to use.

#### **Use Explode From Configuration**

Lists the existing exploded view configurations you can use in an animation. You can specify one exploded view configuration for an animation.

### **Speed**

Specifies the speed you want to use.

#### **Common Velocity for All Parts**

Specifies that the same velocity is used for all parts. You can type a value.

#### **Explosion Duration**

Specifies that you want to type a value in seconds for each explode event.

### **Animation Order**

Specifies how you want the animation to start.

#### **Innermost First**

Specifies that you want to start the animation by moving the innermost parts first.

#### **Outermost First**

Specifies that you want to start the animation by moving the outermost parts first.

### **Initial State**

Specifies whether you want to start the animation with the parts collapsed or expanded.

#### **Collapsed**

Specifies that the animation starts with the parts collapsed.

#### **Expanded**

Specifies that the animation starts with the parts expanded.

## **Activity: Explode**

### **Activity objectives**

In this activity, you will use the Explode-Render-Animate application to explode an assembly. In this activity, you will accomplish the following:

- Use the Manual Explode command to order and sequence the events of an explosion.
- Define the distances and directions of exploding parts along a time line.
- Group parts and subassemblies and control how they behave during and explosion and when they explode.
- Creating an animation time line to be used in an animation sequence.
- Use the Automatic Explode command to begin an exploding sequence.

**Activity: Exploding an assembly****Step 1**

In this activity you open an assembly and enter the Explode-Render-Animate application. You use the automatic explode command to create an exploding time line used to animate the explosion. Once the initial explosions are created, you use the manual explode command to group parts and subassemblies, and to sequence the explosion and define the behavior of the parts as they explode.

You enter the Explode-Render-Animate application and explode the assembly using the Auto Explode command. You explode the assembly with the bind subassemblies option set. After the command is finished, you examine the results, and then use the Unexplode command to reset the assembly.

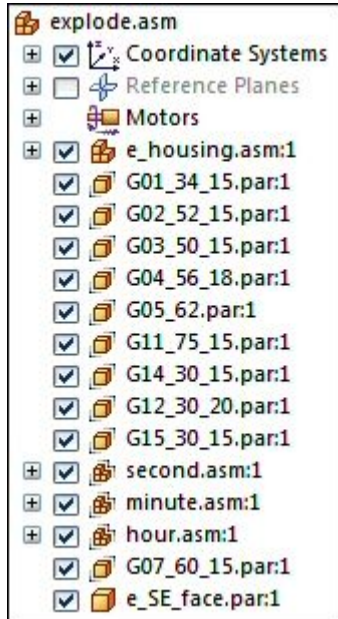
**Note**

When using the Auto Explode command, the results are dependent on several factors. Relationships used in building the assembly determine how the Auto Explode command behaves. Parts positioned with an axial align will explode away from the adjacent part in the direction of the axis. The same parts that can be positioned with an axial align, can also be positioned with mate, and planar align, however how these parts explode using the auto explode command may not be as desired.

The behavior of subassemblies using the Auto Explode command can be defined. Subassembly parts can be bound together as a group causing it behave as if it is a single part, or the subassembly can be exploded into its constituent parts.

For the Auto Explode command to give desirable and predictable results, you must consider which relationships to use during the process of positioning parts in the assembly.

- ▶ Open the assembly *explode.asm* with all the parts active.
- ▶ In PathFinder, observe the relationships and grouping of parts in subassemblies that were used to create the assembly. These relationships and subassemblies will be used in the auto explode command.



- ▶ On the ribbon, choose Tools tab® Environ group® ERA



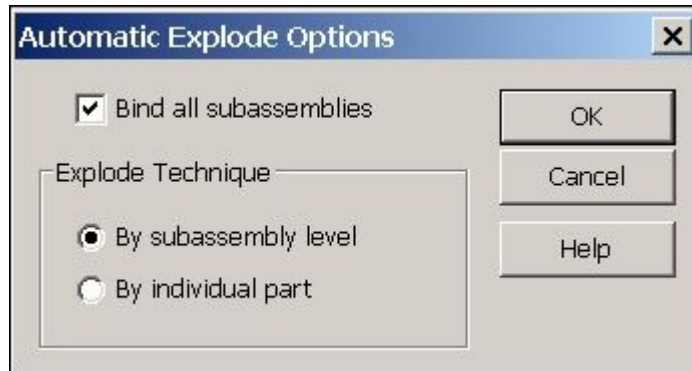
- ▶ On the ribbon, choose Home tab® Explode group® Auto Explode
- ▶ On the Auto Explode command bar, select Top level assembly, and then click the Accept button.



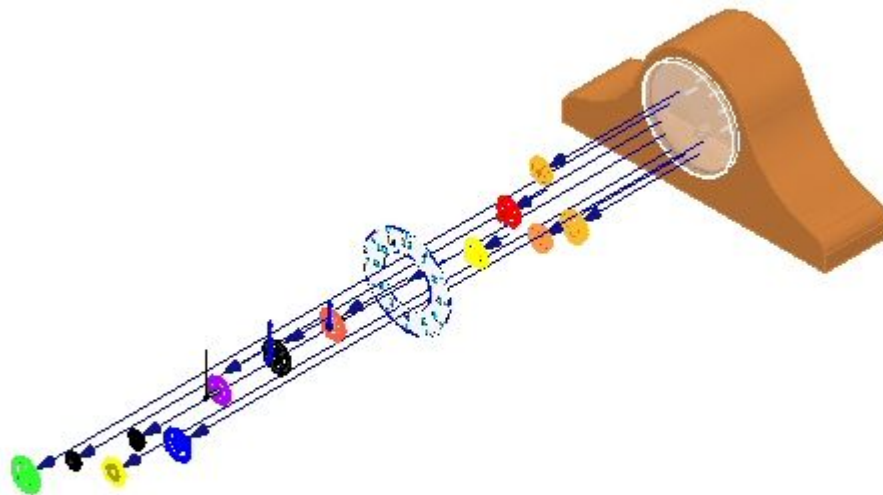
- ▶ Click Automatic Explode Options.



- ▶ In the Automatic Explode Options dialog box, select Bind all subassemblies. Set the Explode Technique to By subassembly level and click OK.

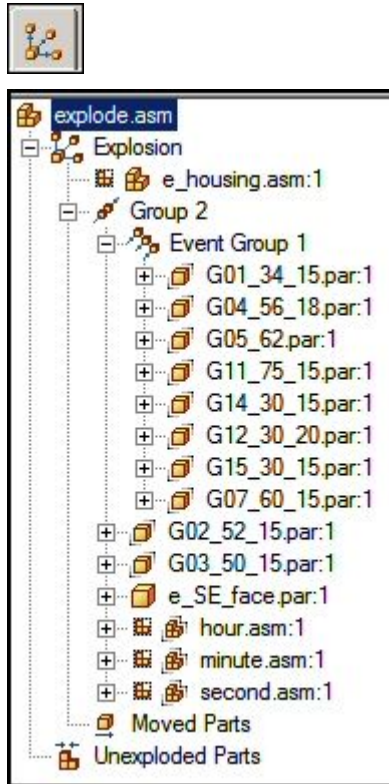


- ▶ On the command bar, click Explode, click Finish and then click Cancel. The results are shown.





Examine the results. The parts that were in the top level of the assembly exploded, and the subassemblies stayed intact.

- Click the Explode PathFinder tab and observe the grouping. The display shows that the subassemblies are bound and behave as if they were a single part.

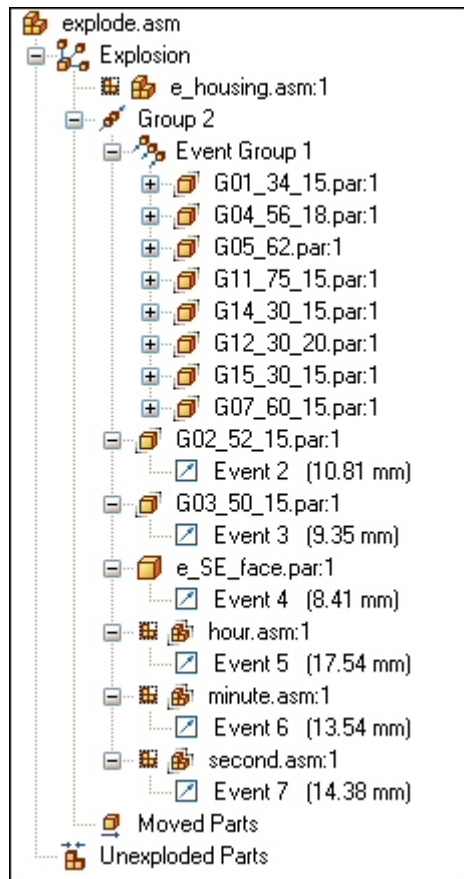


### Note

If PathFinder does not contain the tab you are looking for, such as Explode PathFinder, Parts Library, or Alternate Assemblies, you can display it by doing either of the following:

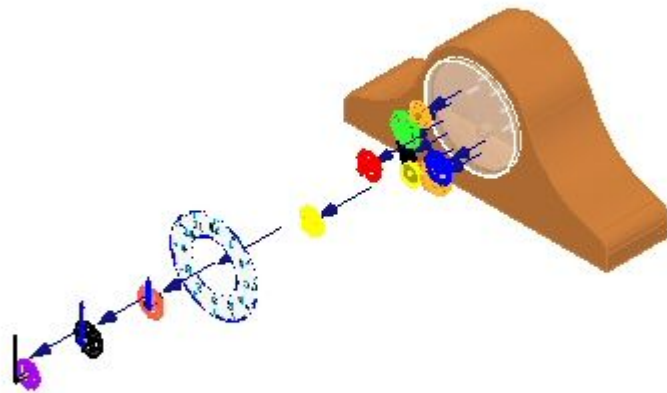
- Choose View tab@ Show group@ Panes , and then select the tab name from the menu.
- In any of the other open docking windows, such as the Layers tab or the Sensors tab, click the Display Docking Window Menu button , and then select the tab name.

- ▶ Expand some of the parts and observe the offset values. The Explode PathFinder allows you to modify events and parameters that define the explosion.







- ▶ Select Group 2 in the Explode PathFinder, and in the command bar enter 15 mm for the explode Distance, and then press Enter. This will set a uniform explosion distance.



**Note**

If you exit the Explode-Render-Animate application, or use the Unexplode command to collapse the explosion without saving a display configuration, any information about the explosion is lost.

- ▶ Choose the Home tab® Configurations group® Display Configurations command .
- ▶ Click New to create a new configuration, type exp01, and then click OK. Click Close.
- ▶ Click the Unexplode command  to restore the assembly to the unexploded state. When asked to delete the current explosion, click Yes.

*Step 2*

Explode the assembly with the bind subassemblies option turned off.

- ▶ Click the Auto Explode command.



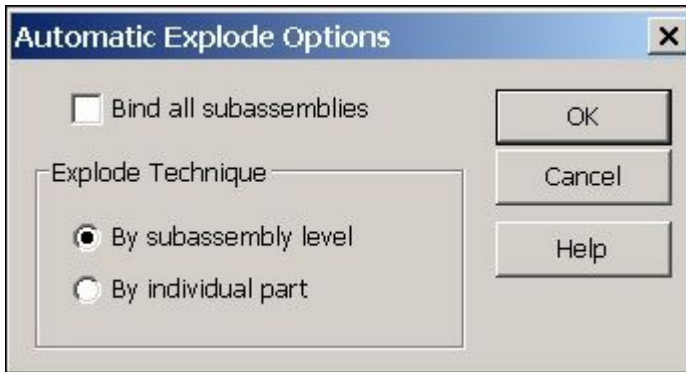
- ▶ Select Top-level assembly and click the Accept button.



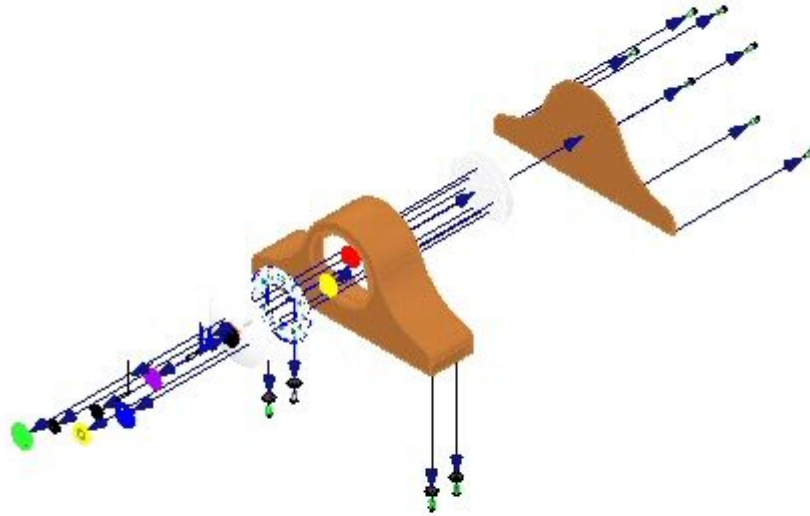
- ▶ Click the Automatic Explode Options button.





- ▶ Deselect Bind all subassemblies and click OK.



- ▶ On the command bar, click **Explode**, click **Finish** and then click **Cancel**. The results are shown.



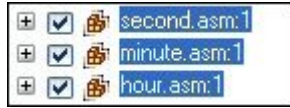
Examine the results. All the parts exploded as if they were in the top level assembly.


- ▶ Click the **Display Configurations** command .
- ▶ Click **New** to create a new configuration, and enter `exp02`, and then click **OK**. Click **Close**.
- ▶ Click the **Unexplode** command  to restore the assembly to the unexploded state. When asked to delete the current explosion, click **Yes**.

Step 3

Bind the clock hand subassemblies so that they do not explode, and repeat the Auto Explode command.

- ▶ In PathFinder, select the subassemblies defining the clock hands.




- ▶ Click the Home tab® Modify group® Bind command .

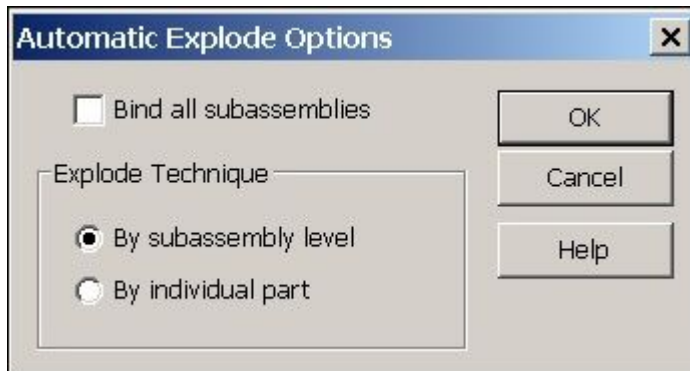
- ▶ Notice the display in PathFinder has changed to indicate the subassemblies are bound.



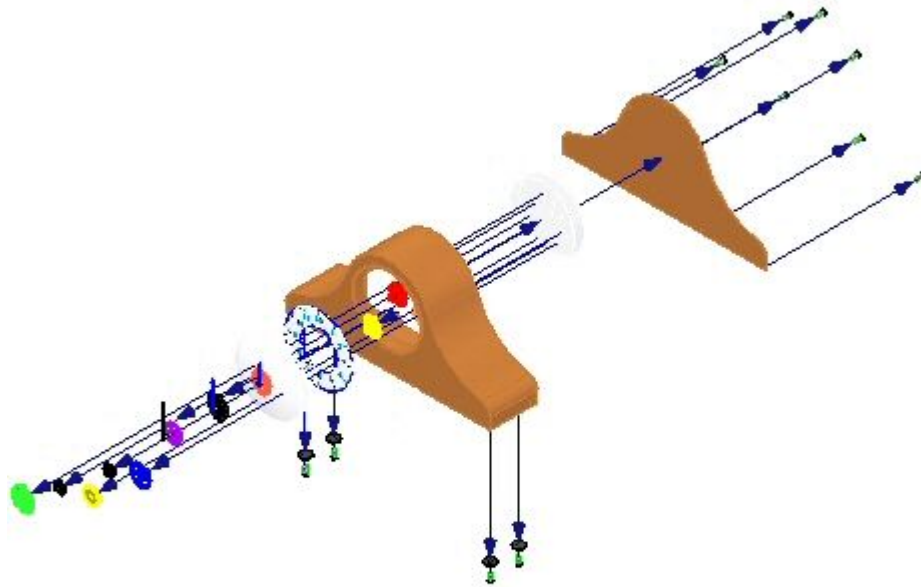
**Note**

If you need to unbind a subassembly, select the subassembly and click the Unbind command .

- ▶ Repeat the Auto Explode command with Bind all subassemblies turned off.



- Click Explode, click Finish and then click Cancel. The results are shown.

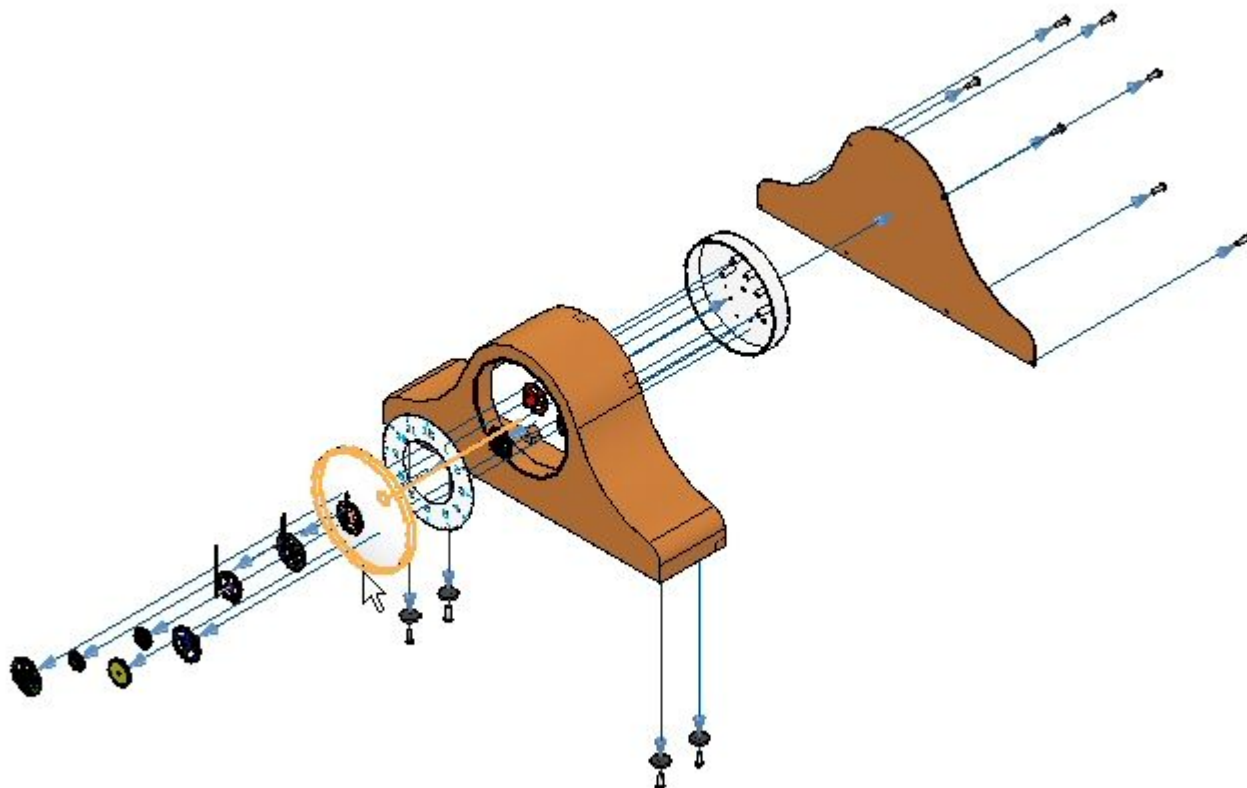


Examine the results. All the parts exploded as if they were in the top level assembly except the clock hands.

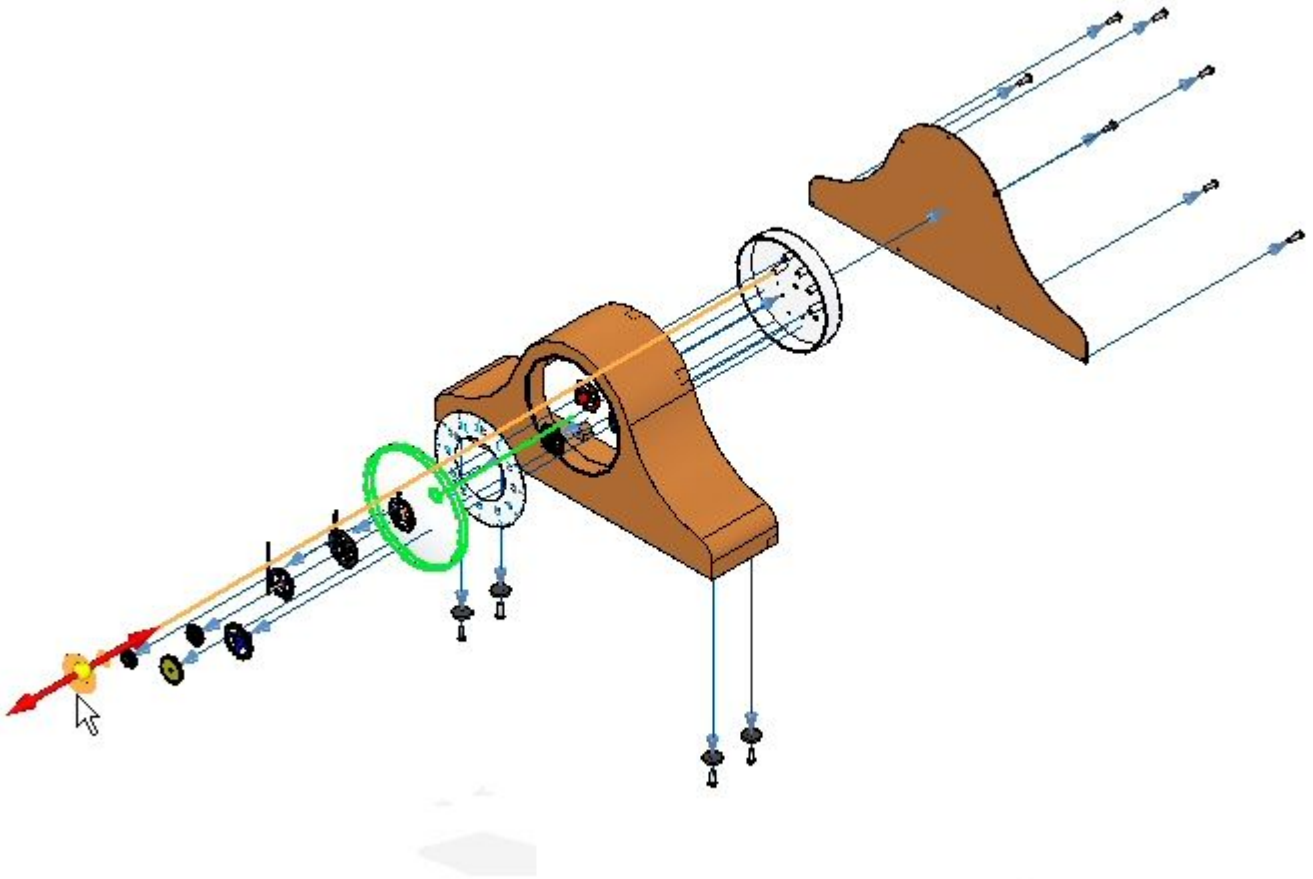
- ▶ You will reposition the parts *e\_glass.par* and *e\_SE\_face.par* in the explosion.

Choose the Home tab® Modify group® Reposition command .

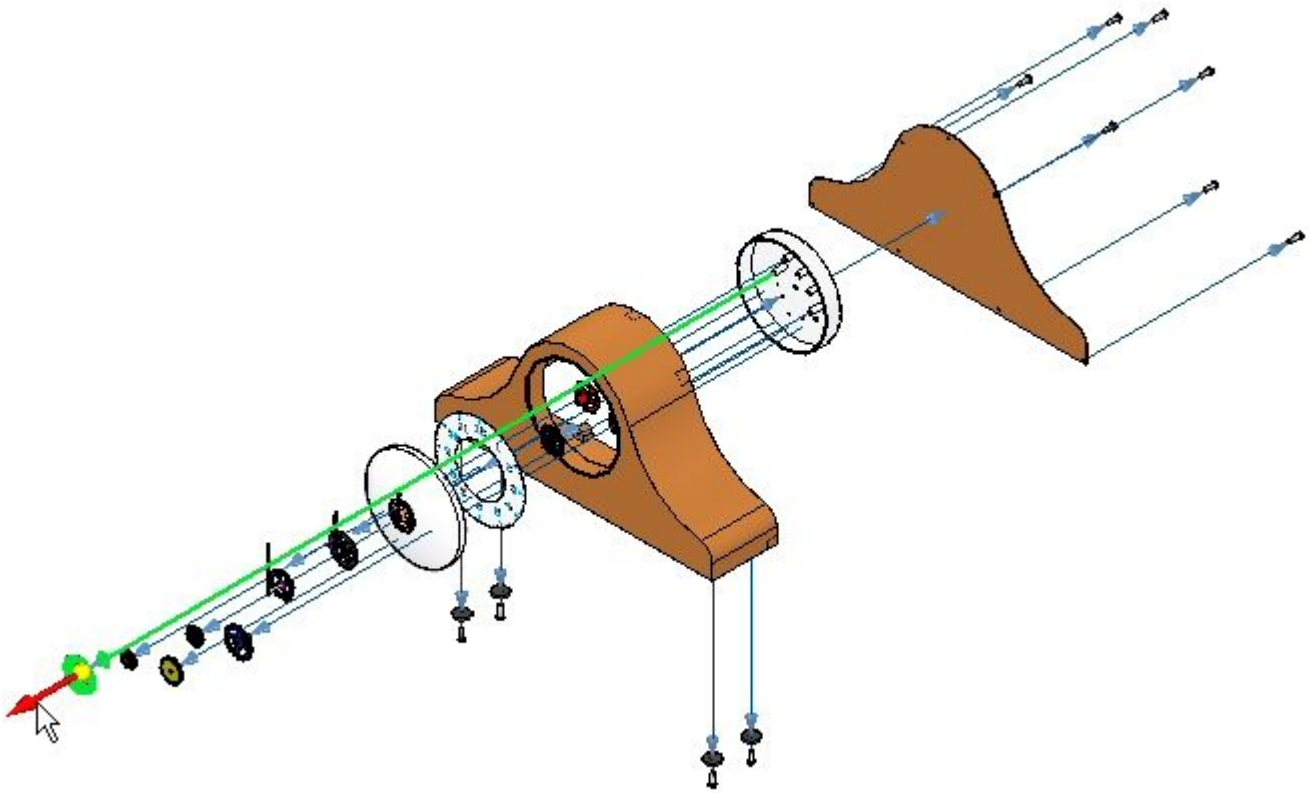
- ▶ Select *e\_glass* as the part to reposition.



- Select *G07\_60\_15.par* as the part to place the part next to.



- Select the side away from the clock housing to place the part.

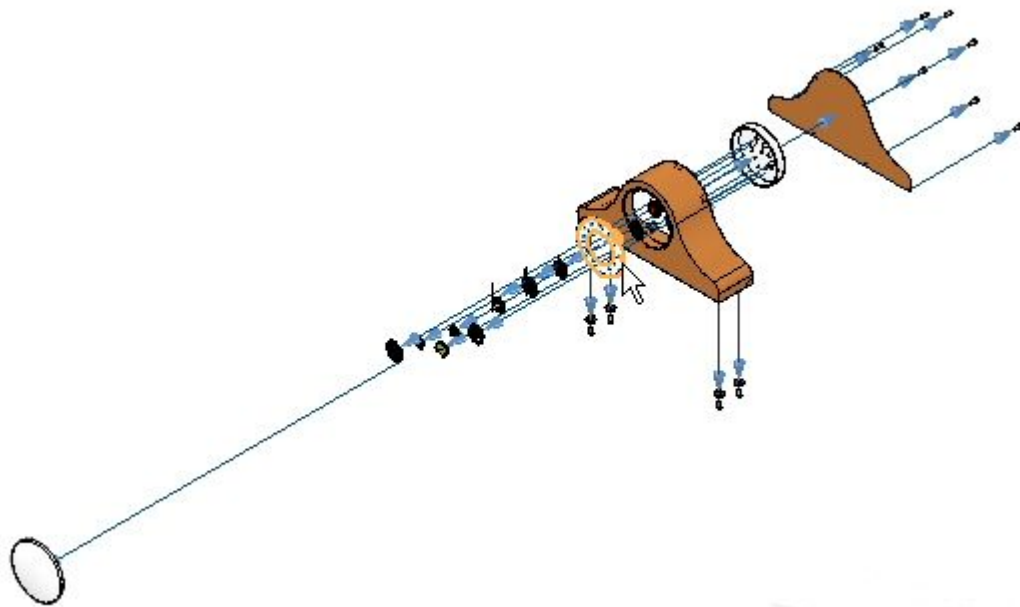
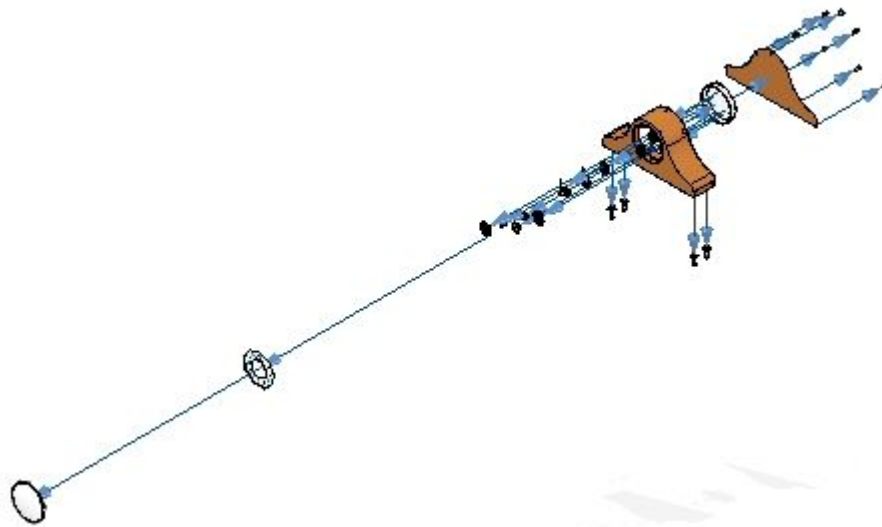


**Note**

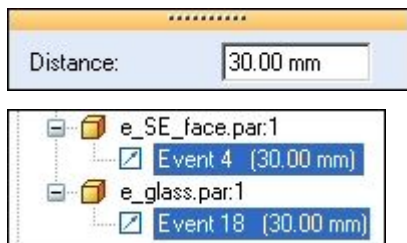
The flow line retains its length and may be longer than desired.





- Reposition the part *e\_SE\_face.par* by repeating the steps above. The part to place the part next to will be *e\_glass.par*. The direction is towards the clock body. This positions the face between the glass and the rest of the clock.



- Click the Select command and in the Explode PathFinder, find *e\_glass.par* and *e\_SE\_face.par* and set the offset distance to 30 mm.



- Click the Display Configurations command .
- Click New, enter exp03, and then click Save. Click Close.
- Click the Unexplode command  to restore the assembly to the unexploded state. When asked to delete the current explosion, click Yes.

Step 4

Use the Auto Explode command with the Subassembly option set. This is the first step in creating the final explosion. After the automatic explosion, use the manual Explode command to further control the events in the explosion.

- Click the Auto Explode command.

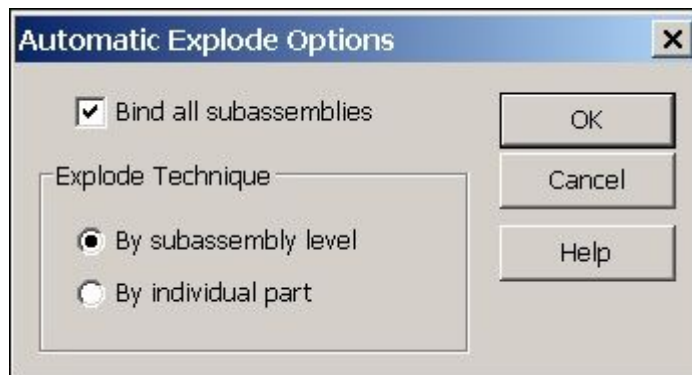


- On the Auto Explode command bar, select Subassembly. Select *e\_housing.asm*, and then click the Accept button.



- Click the Automatic Explode Options button .

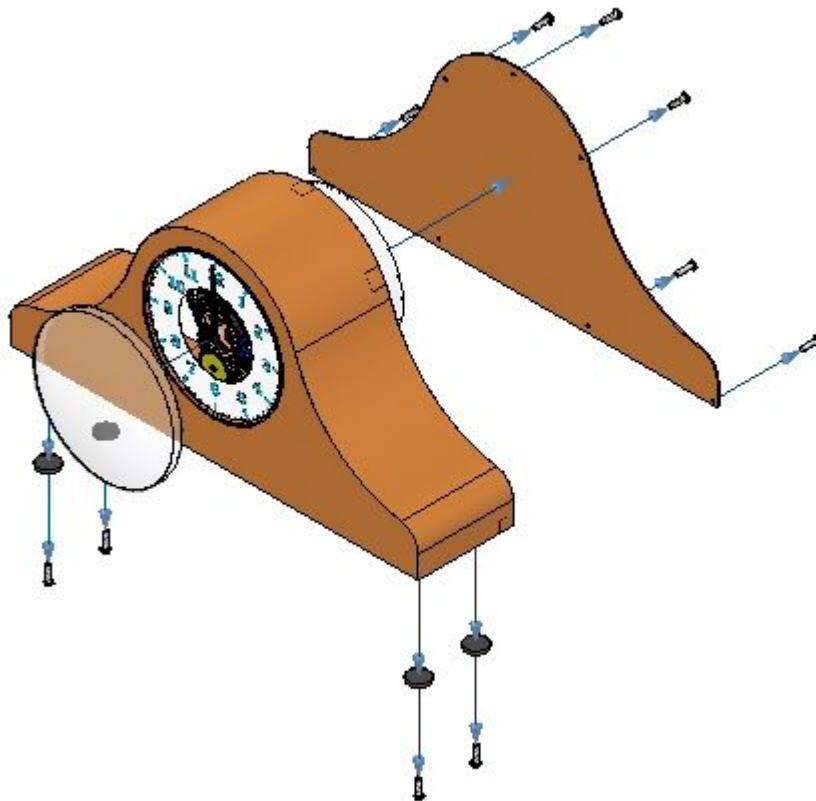
- ▶ Select Bind all subassemblies. Set the Explode Technique to By subassembly level and click OK.



- ▶ Click the Automatic Spread Distance button and enter a value of 15 mm.




- ▶ Click Explode, click Finish and then click Cancel. The results are shown.

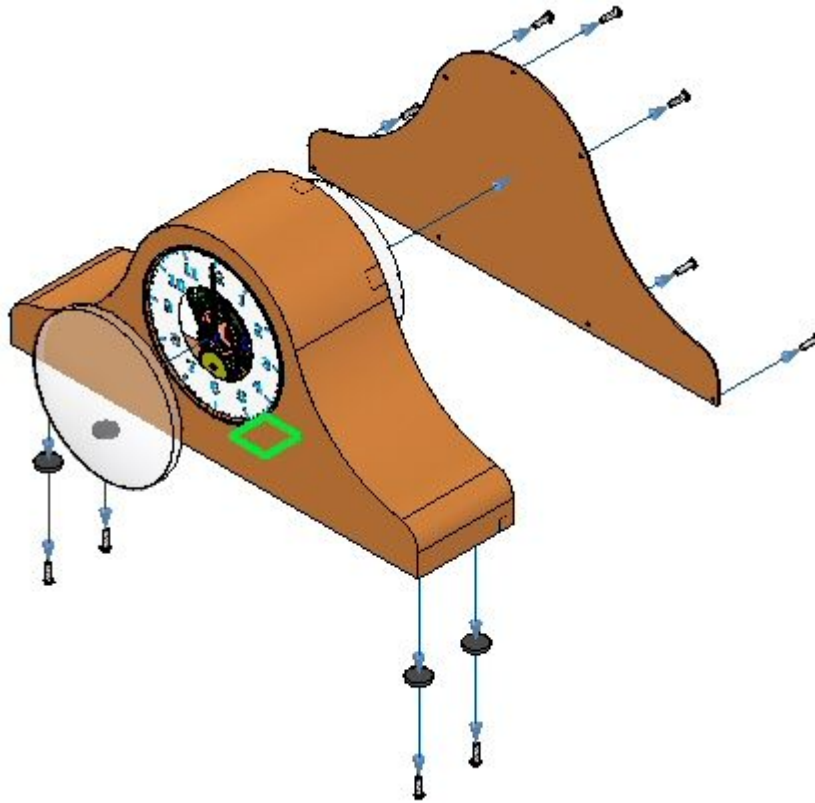


Examine the results. Only the subassembly chosen exploded.

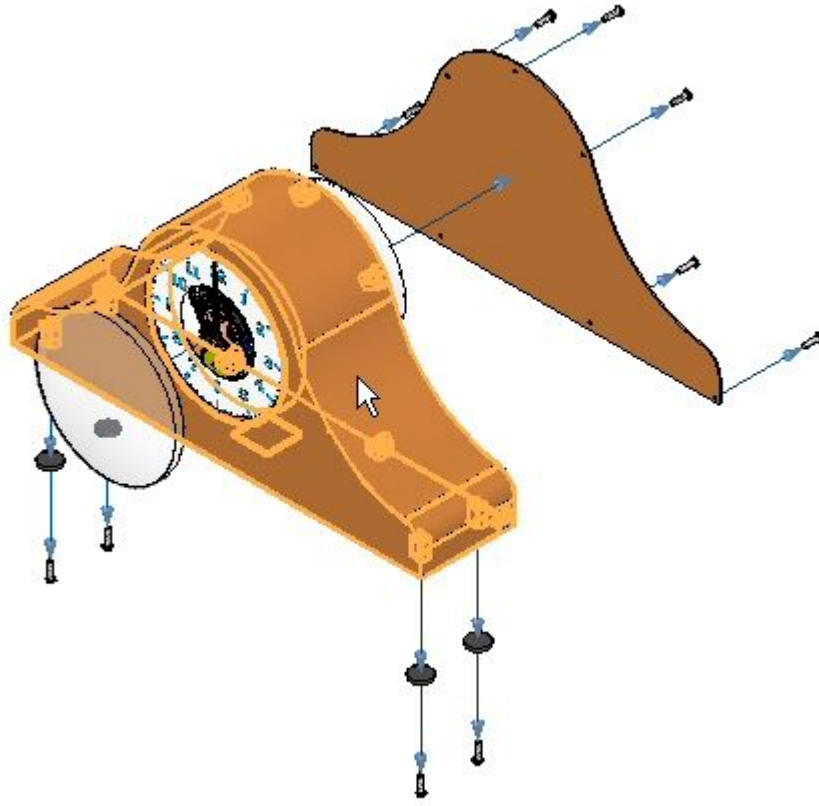
Step 5

Now use the manual Explode command to refine the explosion events.

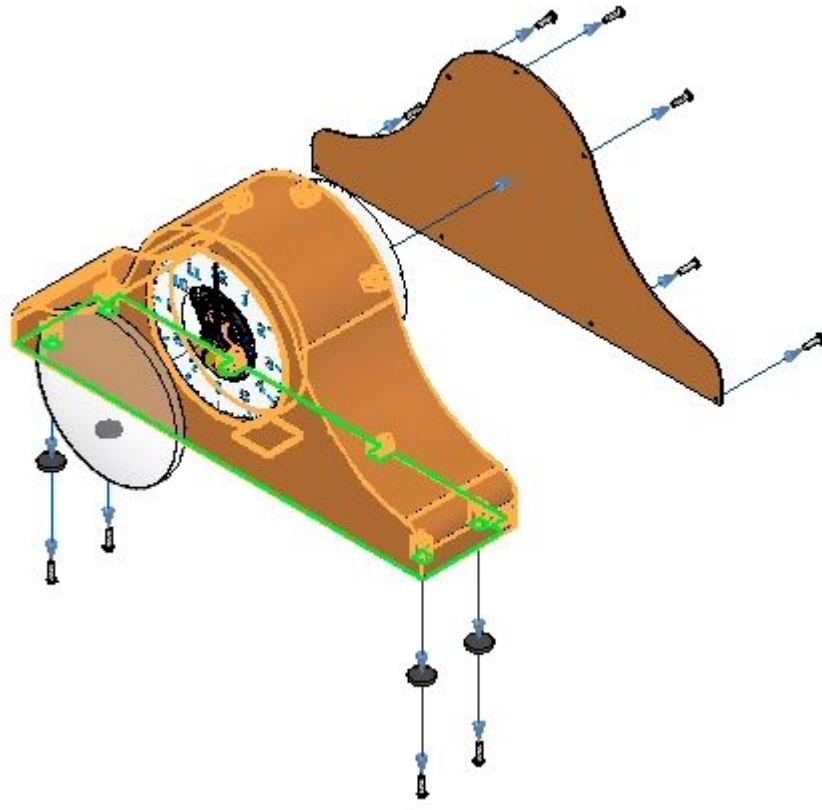
- ▶ Choose Home tab@ Explode group@ Explode .
- ▶ In PathFinder, in the subassembly *e\_housing.asm*, select *e\_feltpad.par*, and then click Accept. This part is on the bottom of the housing and is positioned using a ground relationship. You will explode it in the same direction as the foot pads.



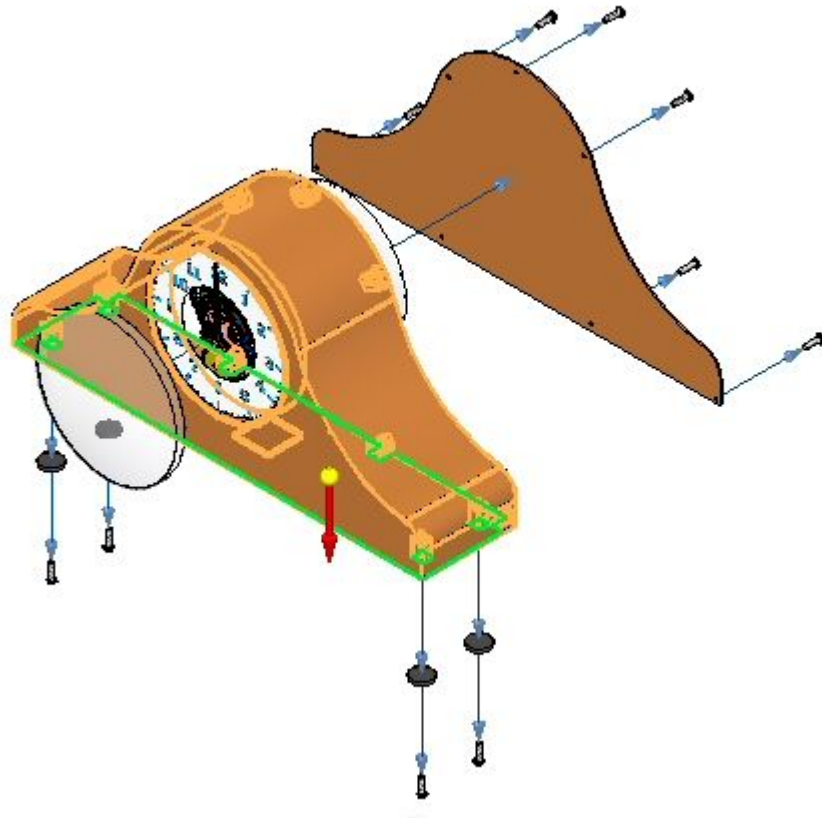
- Select the *e\_case.par* as the stationary part.



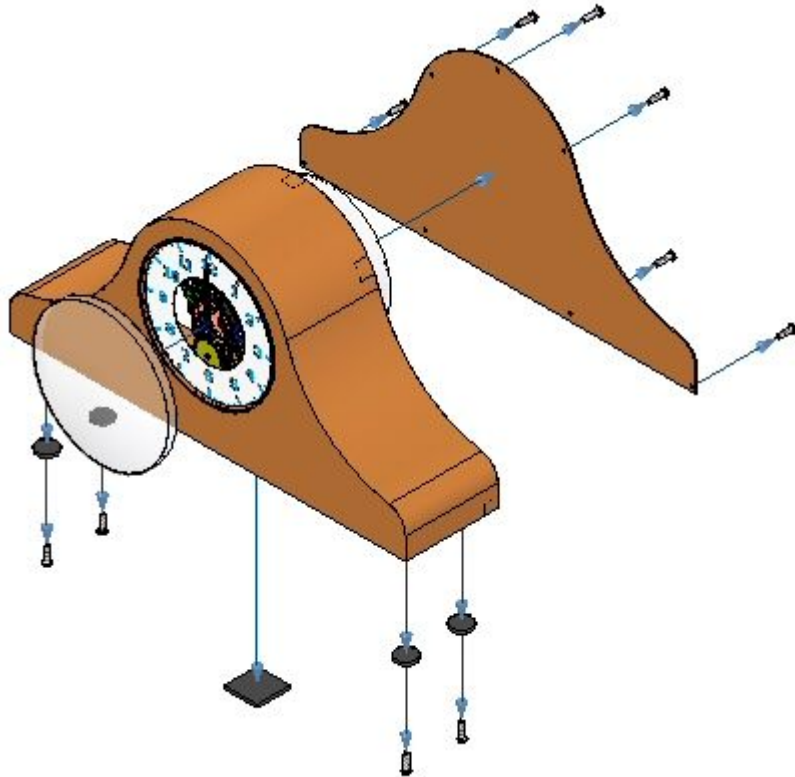
- Select the bottom face of *e\_case.par* as the stationary part face to explode from.



- Select down as the direction to explode.



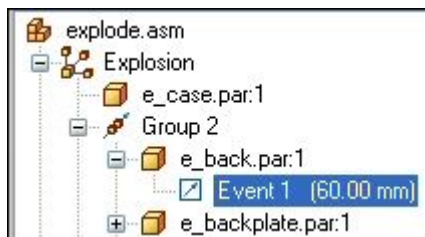
- ▶ Set the offset distance to 35 mm and then click Explode. Click Finish, then click Cancel.



Step 6

Using the manual Explode command, move the gears as a group. Gears will be placed between the clock housing and the circular back plane that is used to position the gears in the clock. You first need to correct the spread distance between the housing and the back to make room for the gears.

- ▶ In the Explode PathFinder, select the Event 1 in *e\_back.par* and change the distance to 60.00 mm.



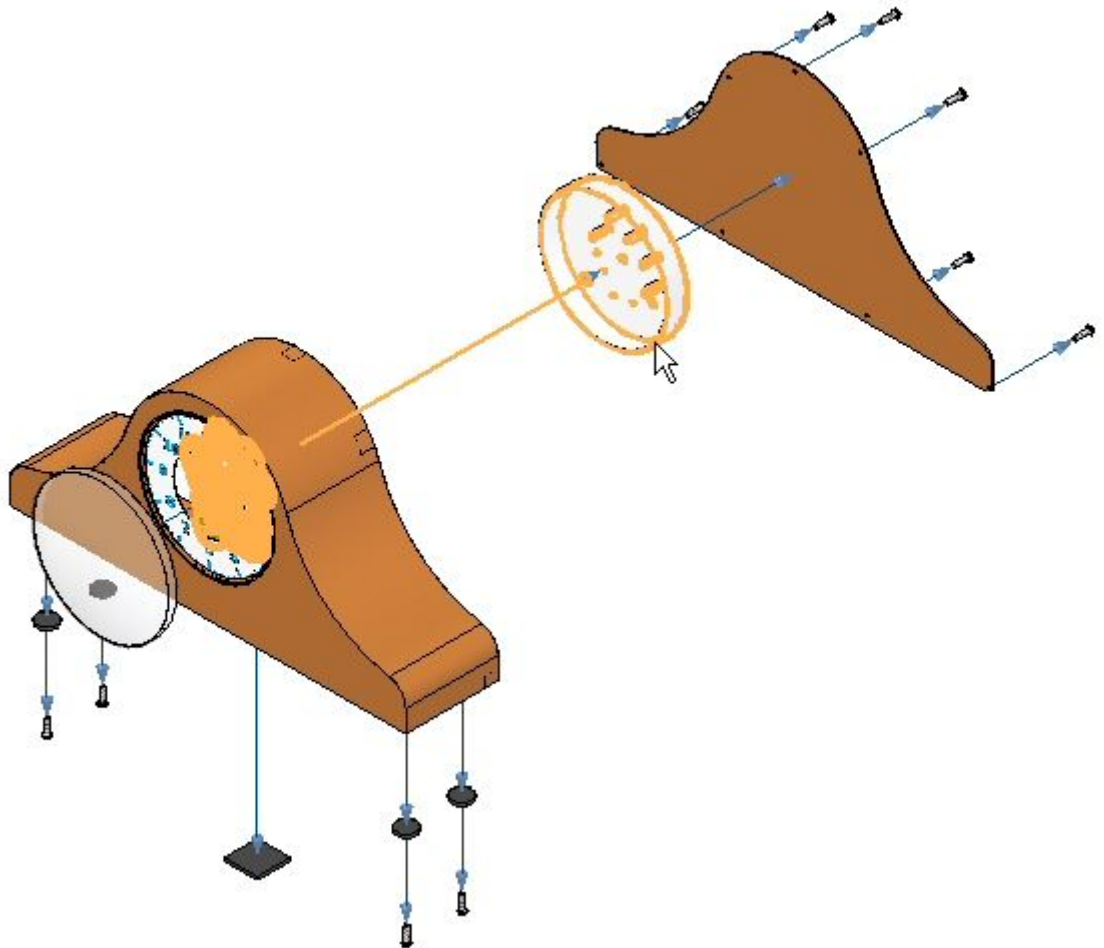
- ▶ Click the Explode command .



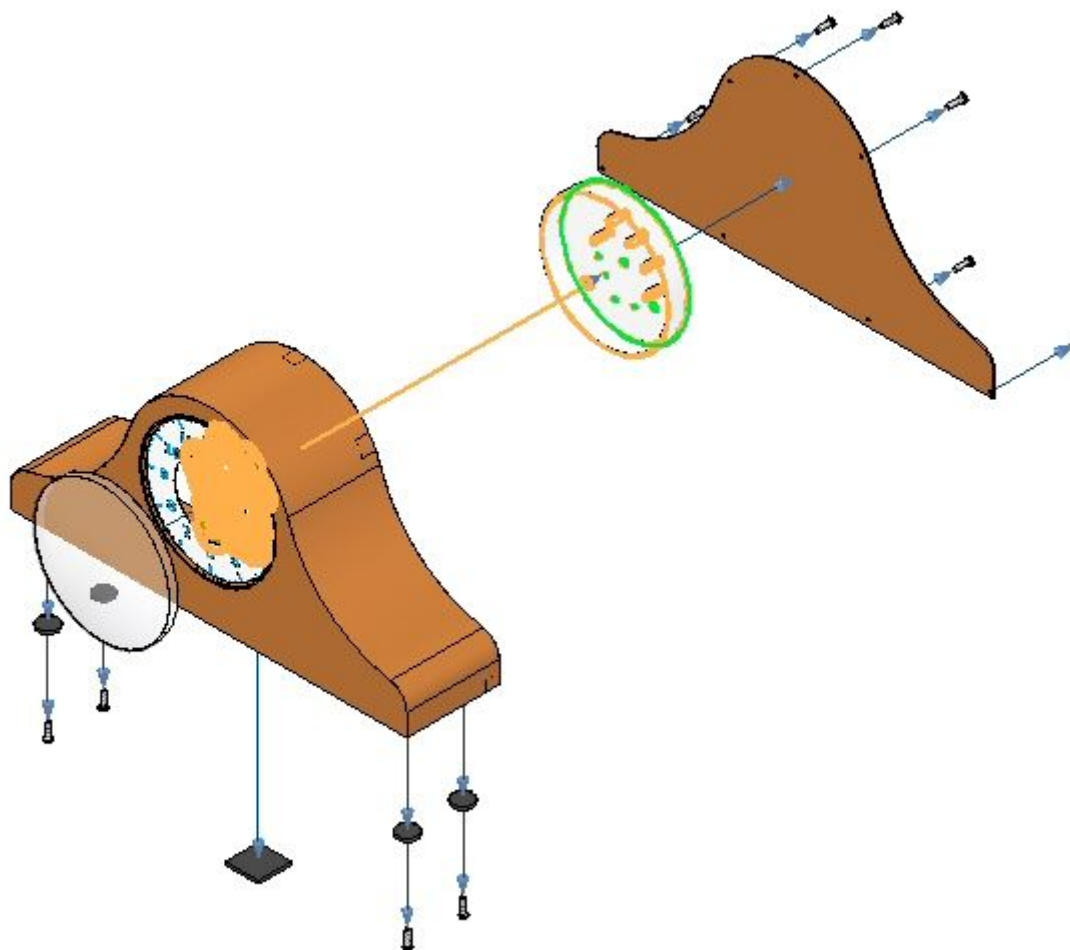
- ▶ In PathFinder, select all the gears and the subassemblies that define the clock hands, and then click Accept.



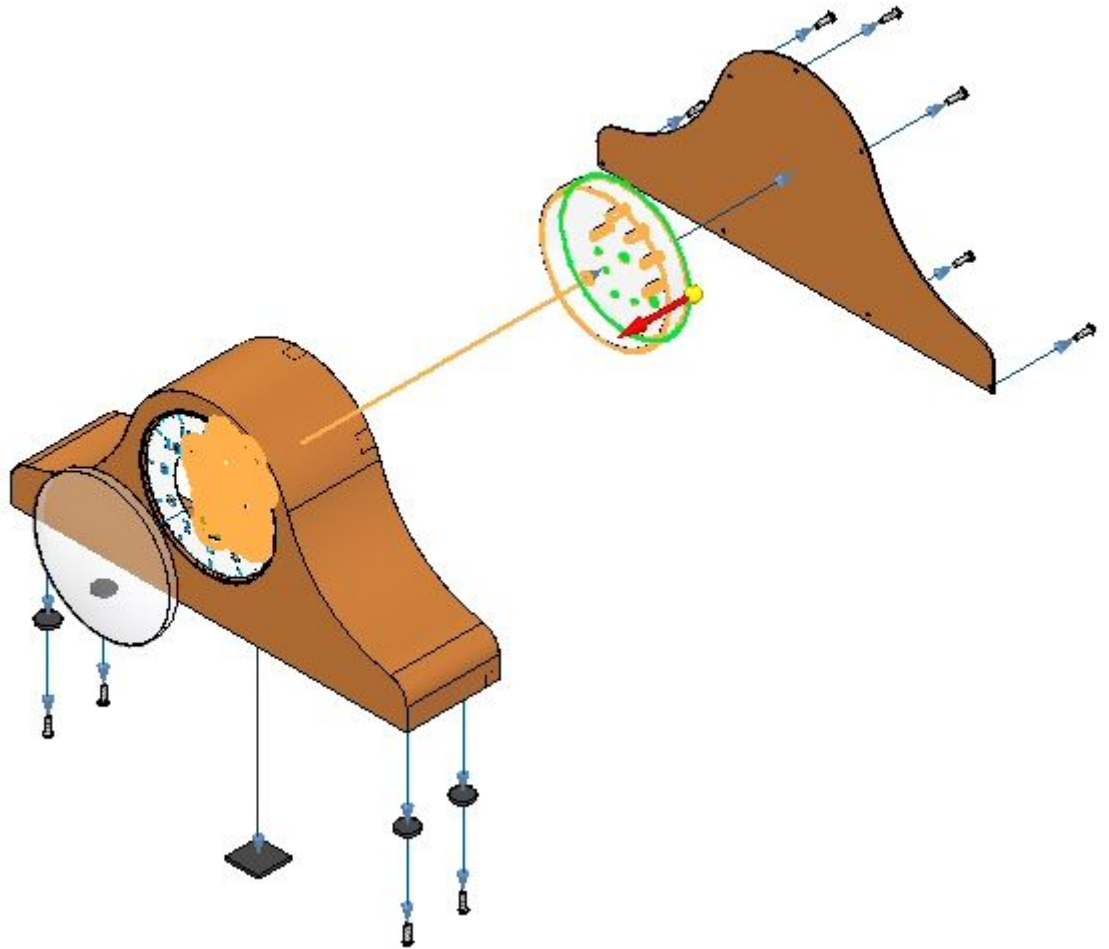
- ▶ Select *e\_back.par* as the part to remain stationary.



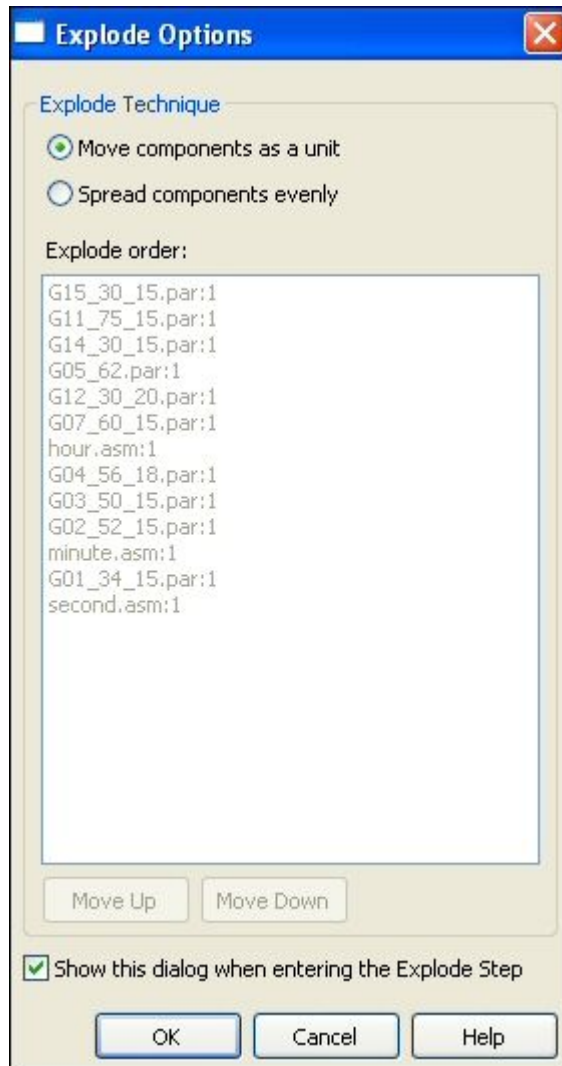
- ▶ Select the circular face shown as the stationary face from which to explode.



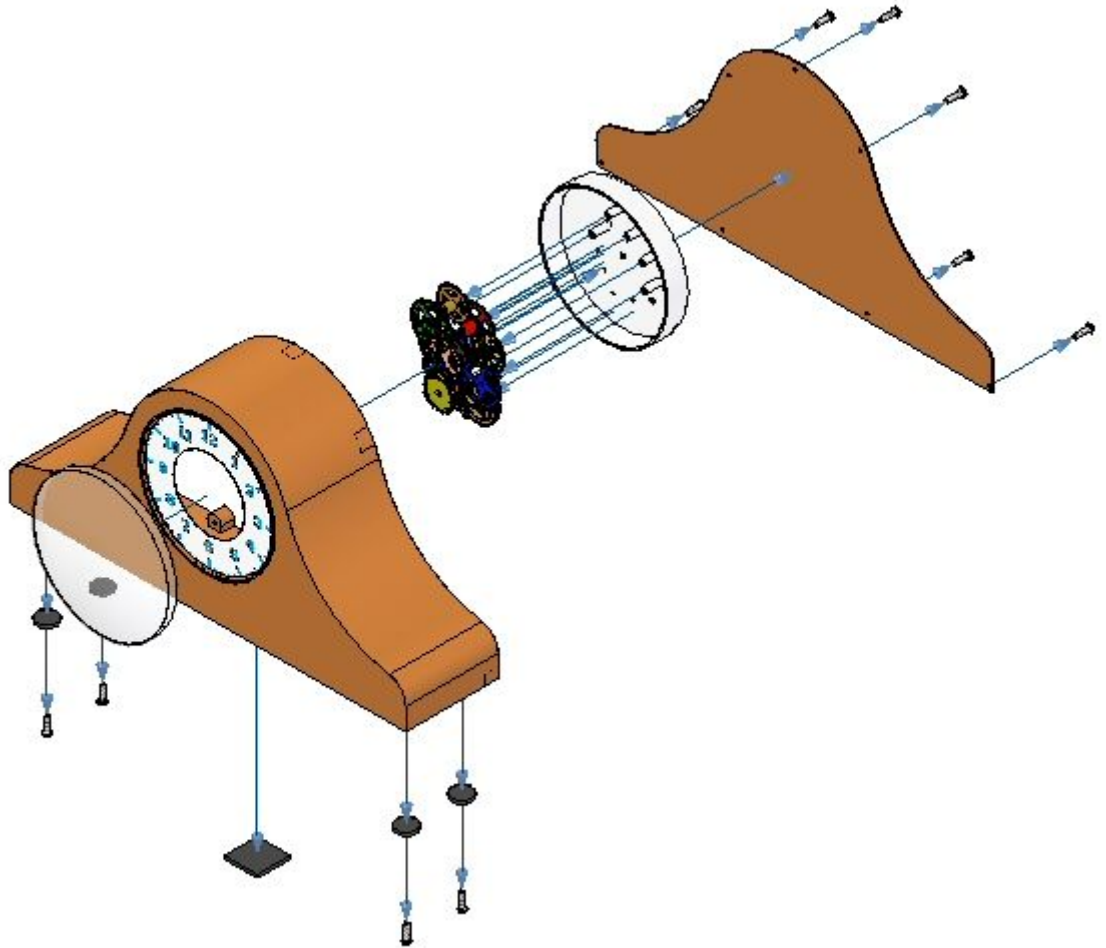
- Select the direction shown as the explosion direction.




- ▶ Set the parameters shown, then click OK.



- ▶ Enter a distance of 25 mm and then click Explode, click Finish, then click Cancel.




- ▶ Click the Display Configurations command .
- ▶ Click New, and enter exp04. Then click OK. Click Close.

**Note**

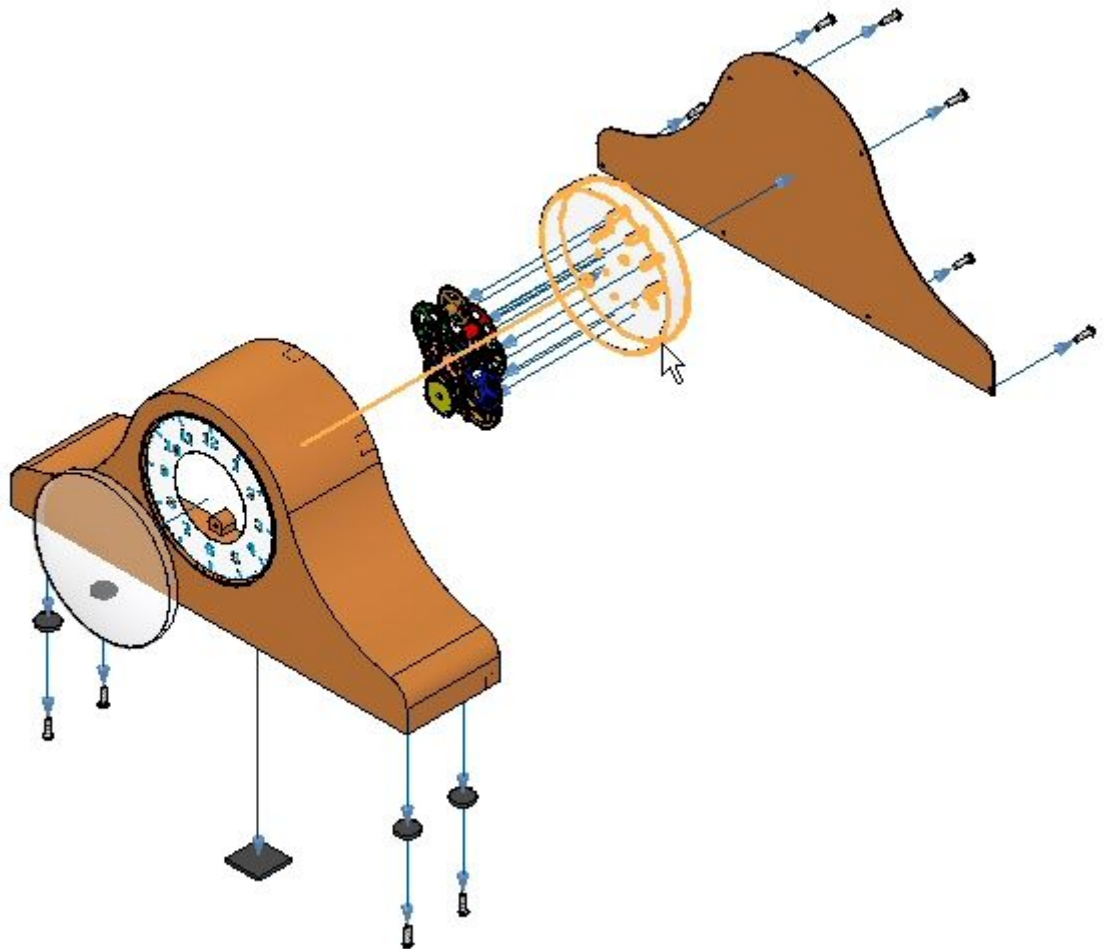
You will overwrite this configuration at a later time. It is good practice to incrementally save the exploded views in case you need to revert back to the point at which you saved.

*Step 7*

Use the Move Exploded Part command to reposition a portion of the explosion.

- ▶ Choose Home tab@ Modify group@ Drag Component .

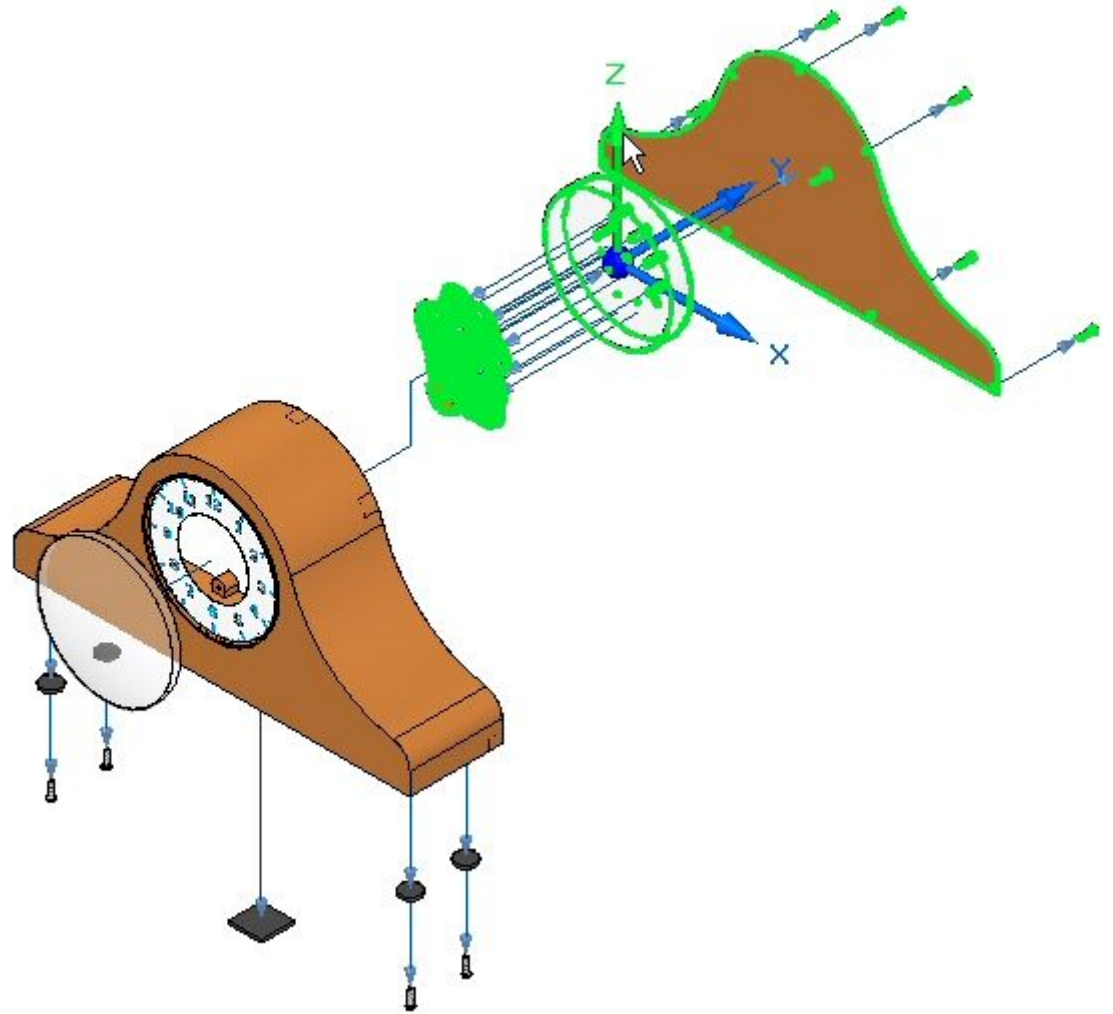
- ▶ From the Explode PathFinder, select *e\_back.par*, and then click Accept on the Drag Component command bar.



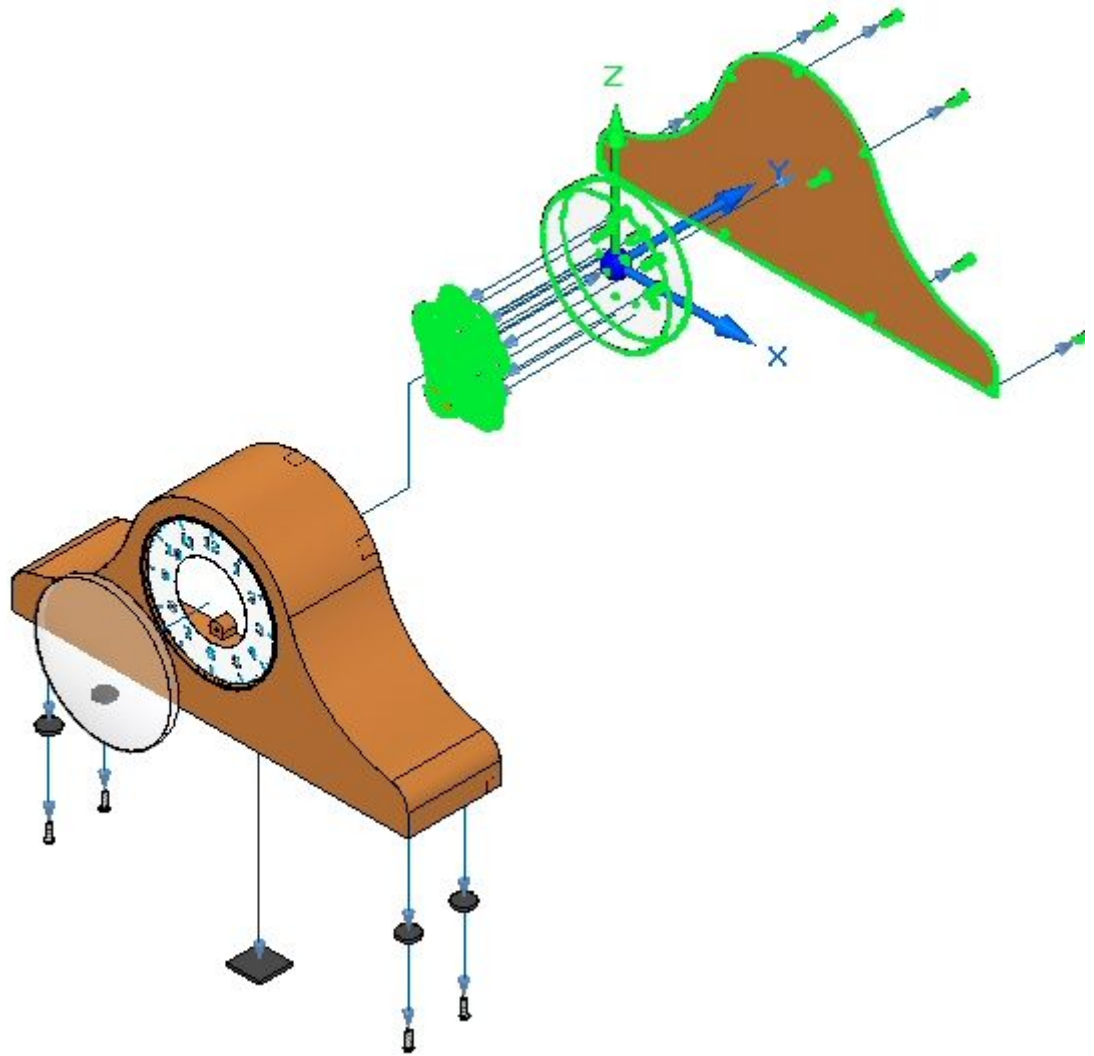
- ▶ On the command bar, select Move



- Drag the Z axis vertically.

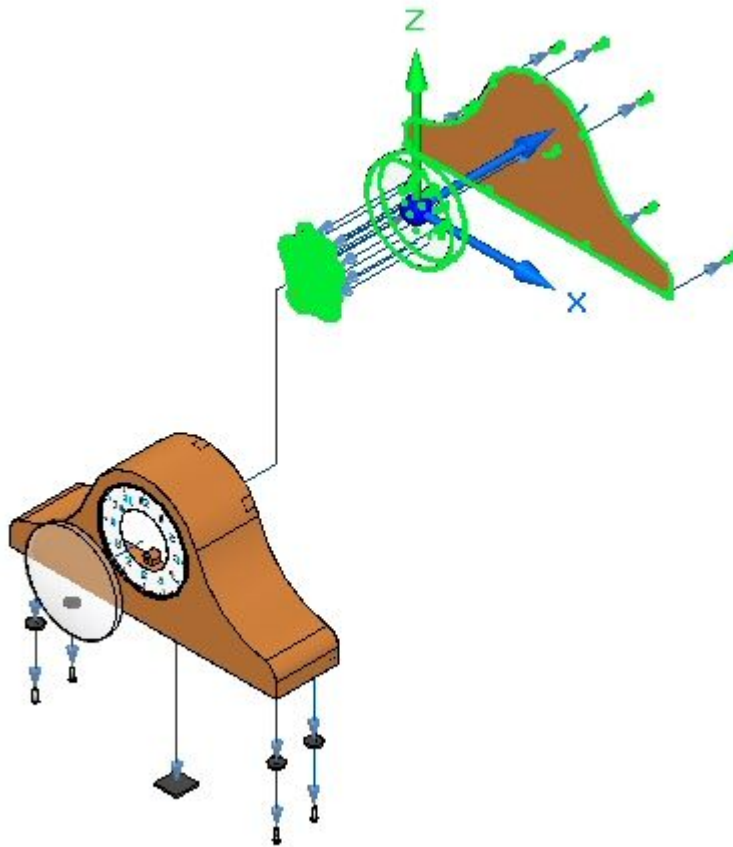


- Position the parts as shown.





- ▶ The result is shown.



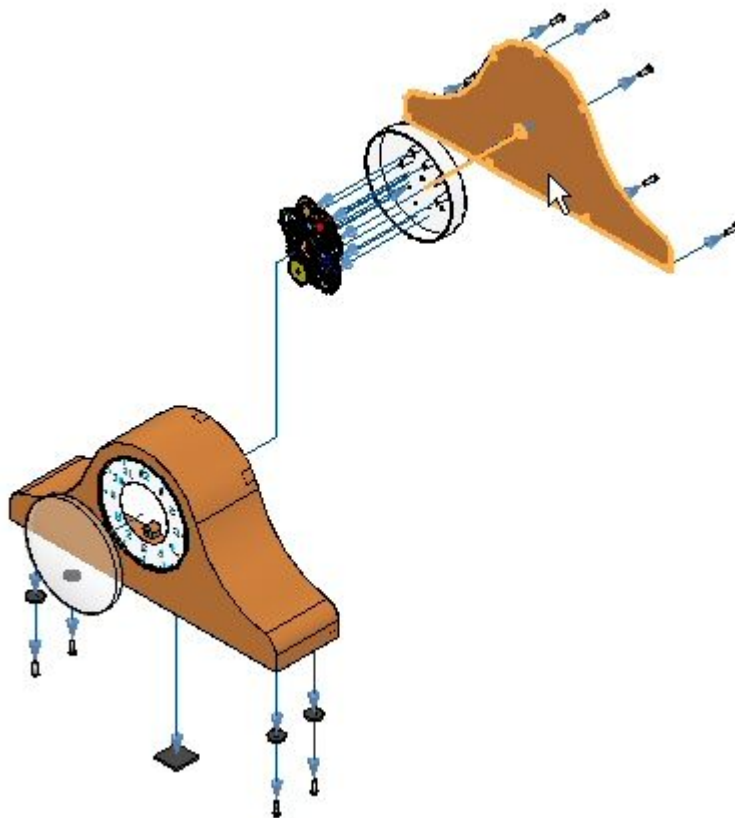
- ▶ Choose the Select command, and in the Explode PathFinder, select the event you just created. The vertical flow line will highlight. Set the distance to 50 mm, and then click OK.



- ▶ Click the Drag Component command



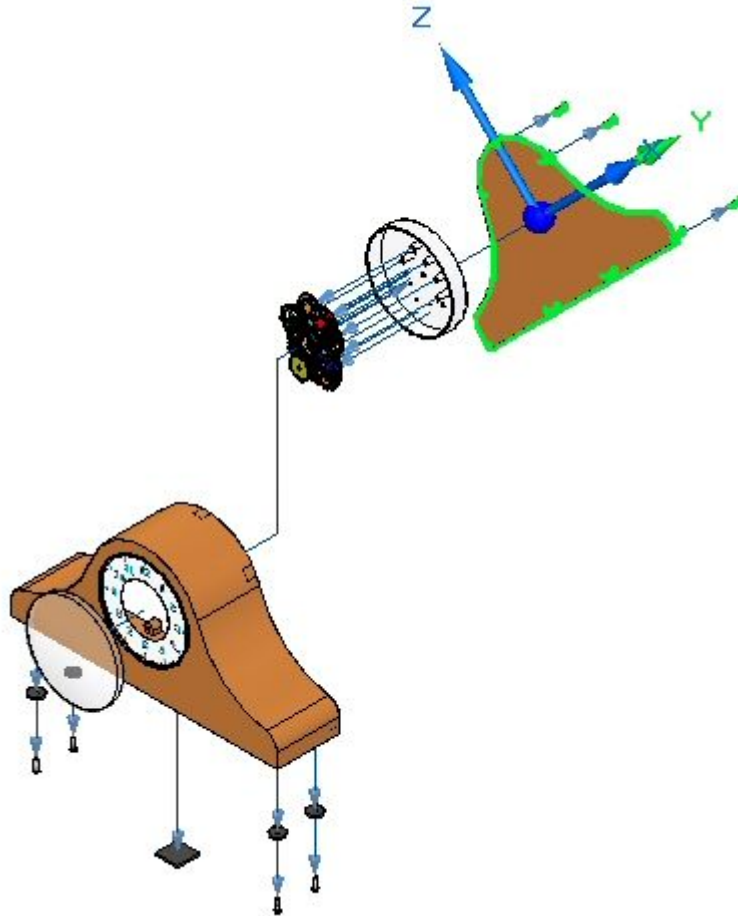
- ▶ Select *e\_backplate.par*, and then click Accept.



- ▶ On the command bar, select Rotate



- ▶ Enter 45° as the angle to rotate about the Y axis. The results are shown.



- ▶ In the Configuration group on the ribbon, click Save Display Configuration.



**Note**

The Save Display Configuration command saves the changes to the configuration name that is currently displayed on the ribbon. This is a quick way to save a configuration.

*Step 8*

Animate the explosion.

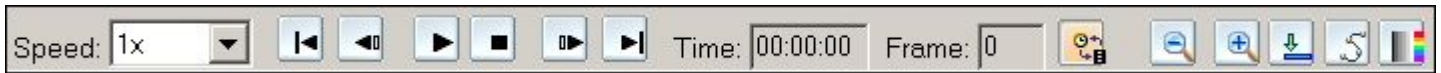
**Note**

Creating an animation of an exploded view is the only portion of the animate command covered in this activity.

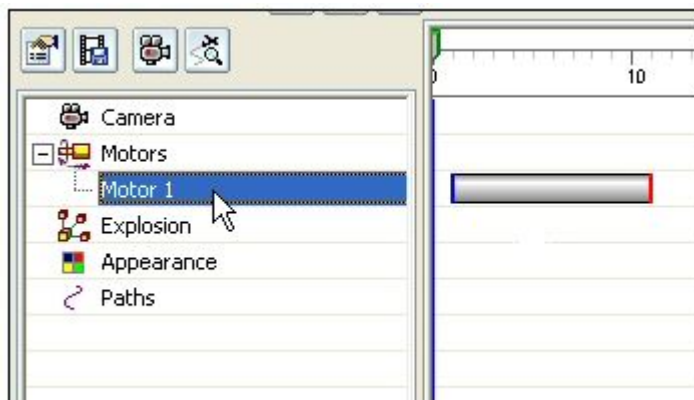
- ▶ Choose Home tab® Animate group® Animation Editor .

- ▶ Examine the Animation Editor.

The right pane is the time line for each of the animation events. A motor was previously defined in this assembly. Controls for playing the animation are displayed.



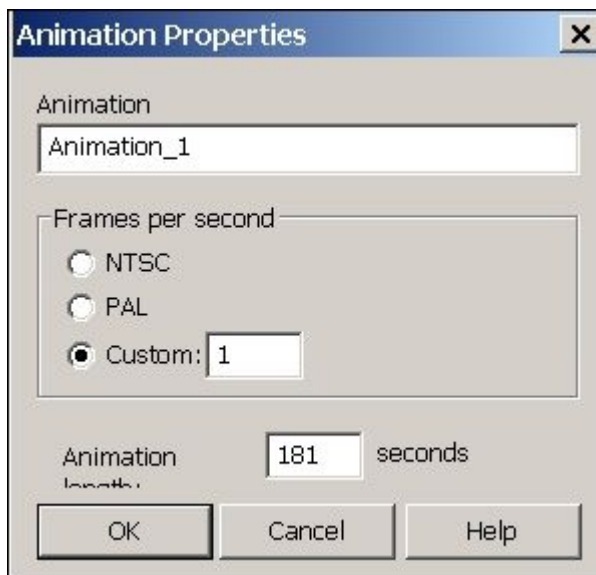
The left pane displays the animation events, and the right pane displays the event duration bars. These can be used to define and sequence the events of the animation.



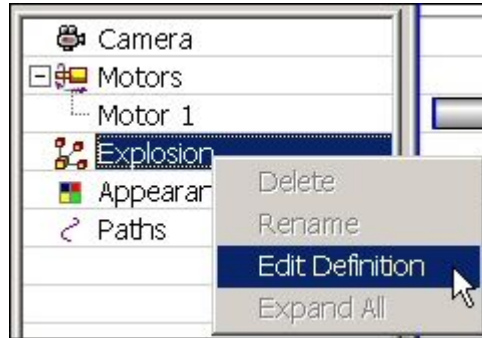
- ▶ Click Animation Properties.



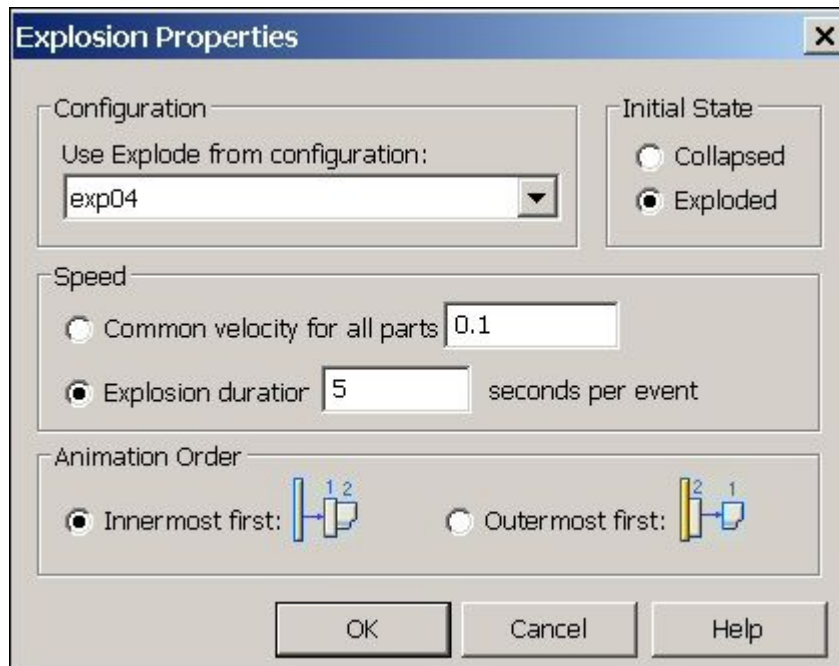
- ▶ Set the values as shown, and then click OK.



- ▶ Right-click the Explosion event, and then click Edit Definition.



- ▶ Set the parameters as shown.
  - Initial State: Exploded.
  - Speed: Explosion duration 5 seconds per event.
  - Animation Order: Innermost first.



Click OK.

- ▶ The explosion events are populated in the left pane.

#### Note

During an animation, you can zoom and pan. It is good practice to arrange the display window before the animation to fit the animation in the view. To do this, choose the View tab@ Window group@ Arrange, and then choose Horizontal.

- ▶ Click Play on the animation controls and observe the explosion.



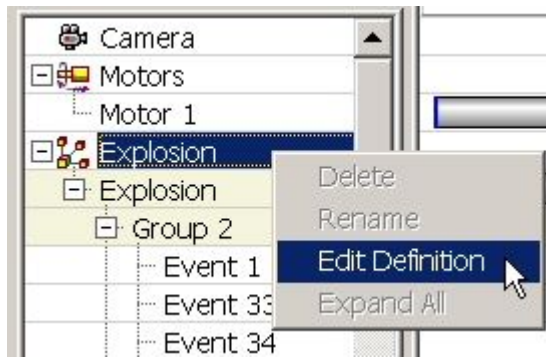
- ▶ When the explosion is complete, click Stop on the animation controls.



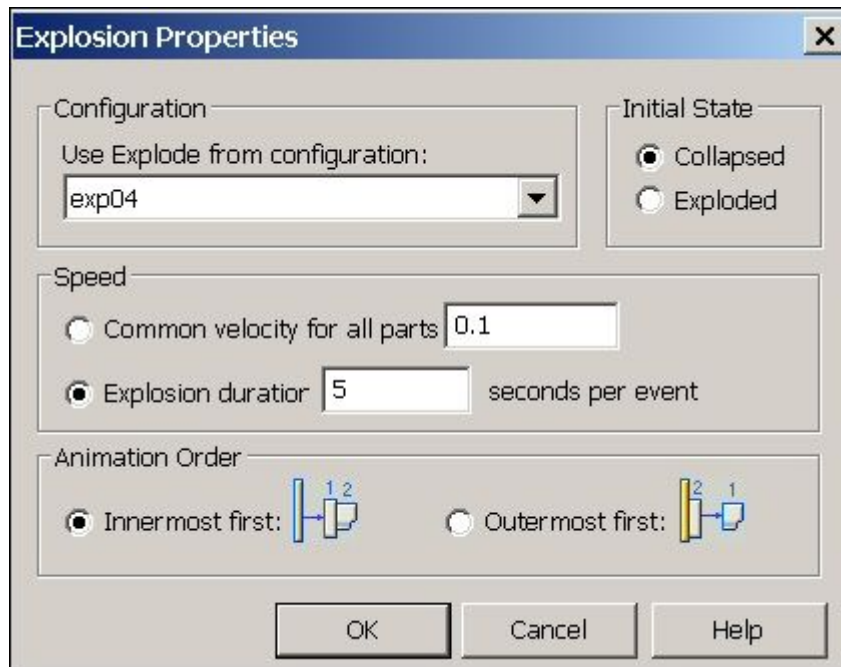
- ▶ Click Go to Start on the animation controls.



- ▶ Now change the sequence of the explosion. Right-click the explosion event, and then click Edit Definition.



- Set the parameters as shown.
  - Initial state: Collapsed.
  - Animation order: Innermost first.



Click OK.

- Click Play on the animation controls and observe the explosion.



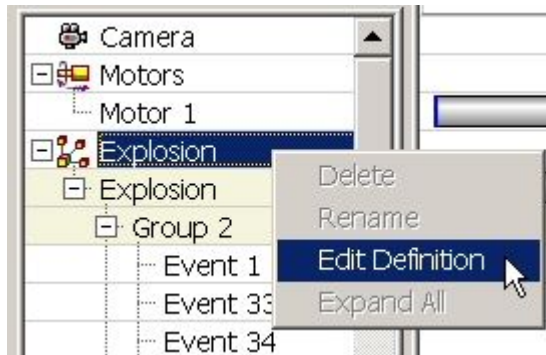
- When the explosion is complete, click Stop on the animation controls.



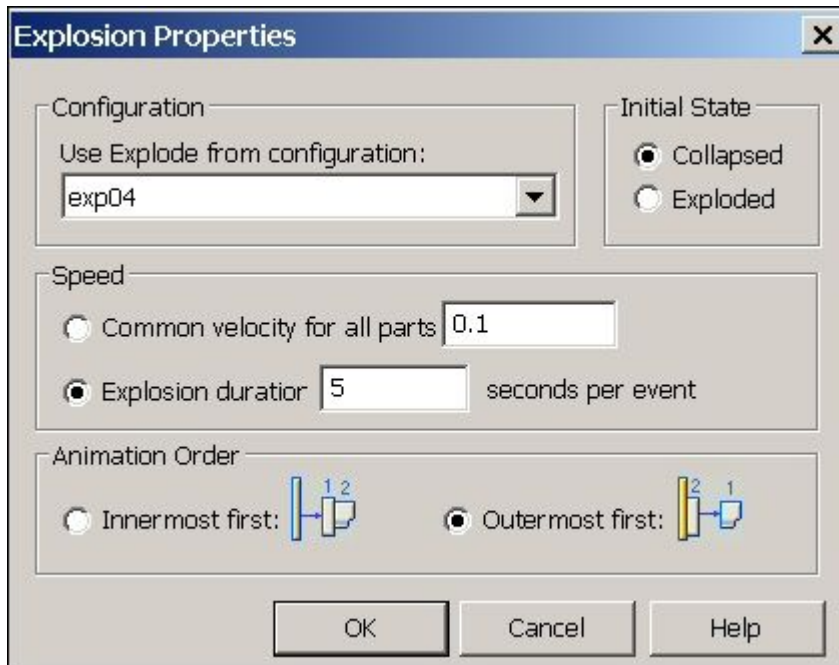
- Click Go to Start on the animation controls.



- ▶ Now change the sequence of the explosion. Right-click the explosion event, and then click Edit Definition.



- ▶ Set the parameters as shown.
  - Initial state: Collapsed.
  - Animation order: Outermost first.



Click OK.

- ▶ Click Play on the animation controls and observe the explosion.



- ▶ When the explosion is complete, click Stop on the animation controls.





- ▶ Click Go to Start on the animation controls.



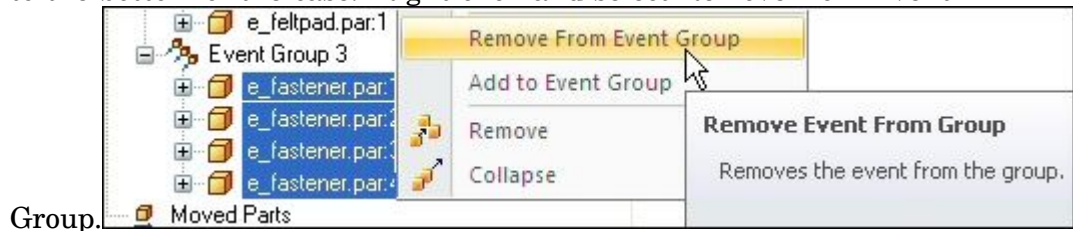
- ▶ To exit the Animation Editor, click the Animation Editor command again. Click Yes to save the changes to the current animation.



*Step 9*

To get all the fasteners to explode at the same time, move all the fasteners into the same event group for animation purposes.

- ▶ In the Explode PathFinder, select the fasteners connecting the footpads to the bottom of the case. Right-click and select Remove from Event



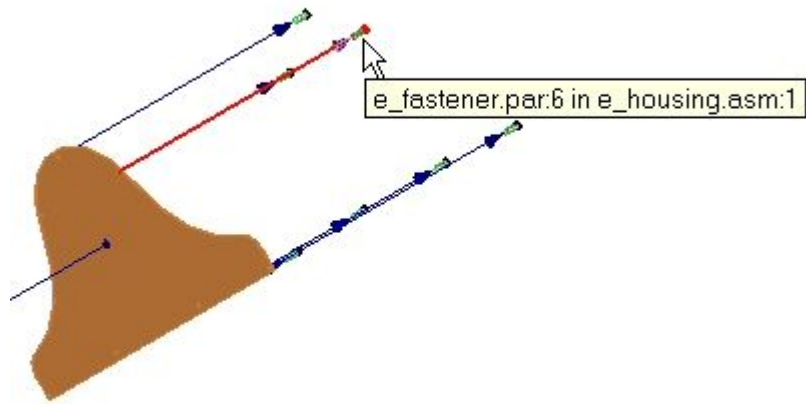
**Note**

The event group that contained these fasteners was dissolved because it no longer contains any events.

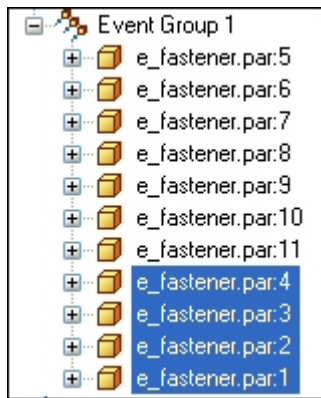
- ▶ Right-click the fasteners and select Add to Event Group.



- ▶ Either from Explode PathFinder or in the graphics window, select a fastener in the group you are adding these fasteners to.



- ▶ The fasteners now all belong to the same event group.



- ▶ Click Save Display Configuration.



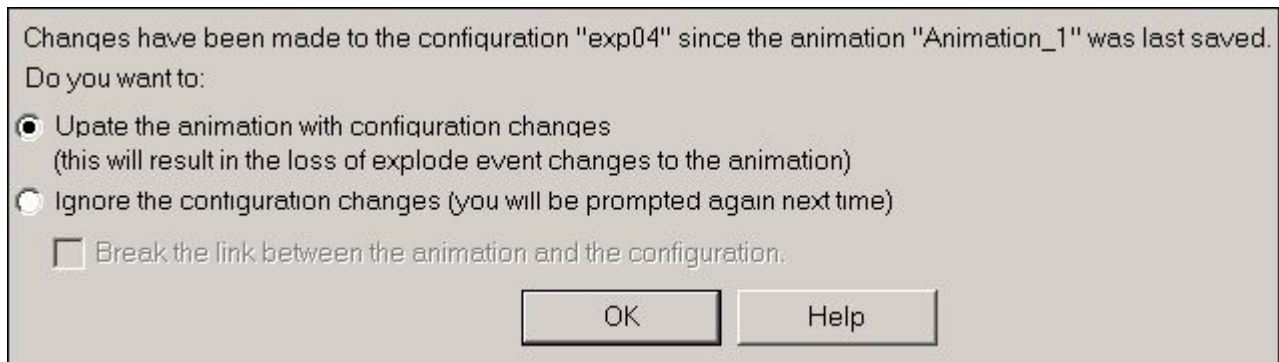
*Step 10*

Replay the animation and observe the behavior of the fasteners now that they are all in the same event group.

- ▶ Click the Animation Editor command.



- ▶ Update the animation with the configuration changes.



- ▶ Click Play on the animation controls and observe the explosion.



- ▶ When the explosion is complete, click Stop on the animation controls.

**Note**

Notice the fasteners exploded simultaneously.

- ▶ Click Go to End on the animation controls.

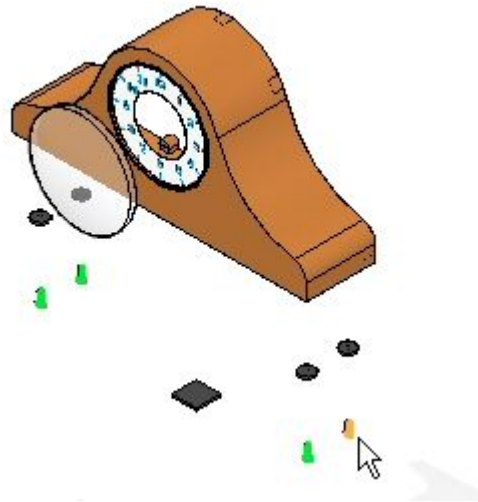
*Step 11*

Add a motion path to the explosion to control movement of the bottom fasteners.

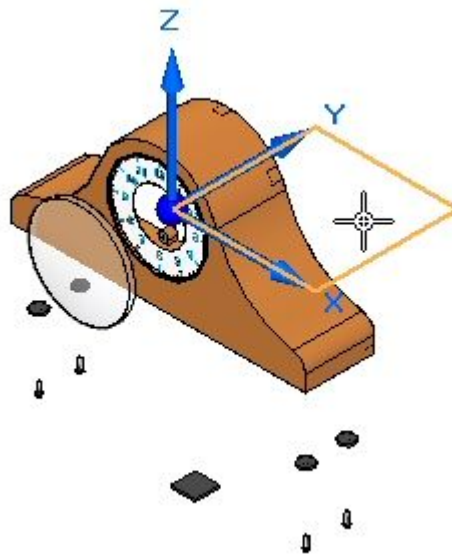
- ▶ Click the Motion Path command on the animation controls.



- ▶ Select the bottom fasteners as the components that follow the motion path, and then click Accept.



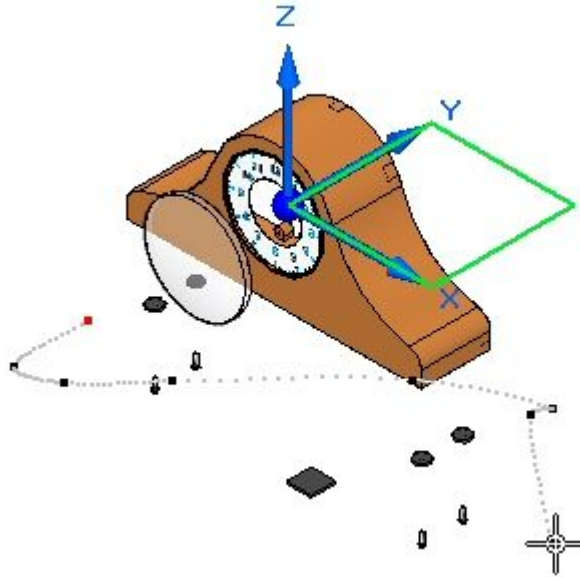
- ▶ Press X on the keyboard as many times as it takes to lock the XY plane as shown.



- ▶ On the command bar, enter 35 as the frame count.



- ▶ Enter the curve approximately as shown and Accept. Click Finish.



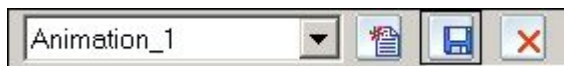
#### Note

This curve is a free-form 3D space curve locked into the XY plane. Your results may vary slightly.

- ▶ Play the animation. Notice the fasteners follow the motion path at the beginning of the animation.
- ▶ In the animation time line, drag the motion path event bar to the right as far as it can go.



- ▶ Right-click the event bar and check the Properties. These can be modified if necessary.
- ▶ Run the animation from the beginning. Notice the fasteners follow the motion path at the end of the animation rather than the beginning.
- ▶ Stop the animation and reset to the beginning. Save the changes by clicking the Save Animation command.



- ▶ To exit the Animation Editor, click the Animation Editor command.



- This completes the activity. Click Close ERA to exit the Explode-Render-Animate application. Save the assembly.



*Activity summary*

In this activity, you used the Explode-Render-Animate application to explode an assembly. You accomplished the following:

- Used the Auto Explode command to begin an exploding sequence.
- Defined the distances and directions of exploding parts along a time line.
- Used the manual Explode command to order and sequence the events of an explosion.
- Grouped parts and subassemblies and controlled how they behave during and explosion and when they explode.
- Created an animation time line to be used in an animation sequence.

**Lesson review**

Answer the following questions:

1. Where are exploded views stored?
2. Name two methods of binding a subassembly during an explosion?
3. How can explode events be moved to other explode groups?
4. Why would you use manual explode rather than auto explode?
5. What type of flow lines are created with the auto explode command?

**Answers**

1. Where are exploded views stored?

Exploded views are stored as a display configuration. Display configurations are stored in .cfg files usually having the same name as the assembly document.

2. Name two methods of binding a subassembly during an explosion?

Subassemblies can be bound either using the explode options, or by forcing an assembly to stay bound by using the bind assembly command. Pathfinder graphically shows which assemblies are bound.

3. How can explode events be moved to other explode groups?

After selecting the explode events from explode pathfinder, right click and remove the explode events from the current explode group. Use the same procedure to add the events to a different group.

4. Why would you use manual explode rather than auto explode?

Manual explode positions assembly components based on which assembly relationships used to place the components. Sometimes the results do not convey the intended results. By using the manual explode command, you can control the placement of exploded assembly components. This can be done after the auto explode command has finished.

5. What type of flow lines are created with the auto explode command?

Event flow lines are created with the auto explode command. Event flow lines are used to create animations of exploded parts and will show up in draft documents. They are stored in a display configuration.



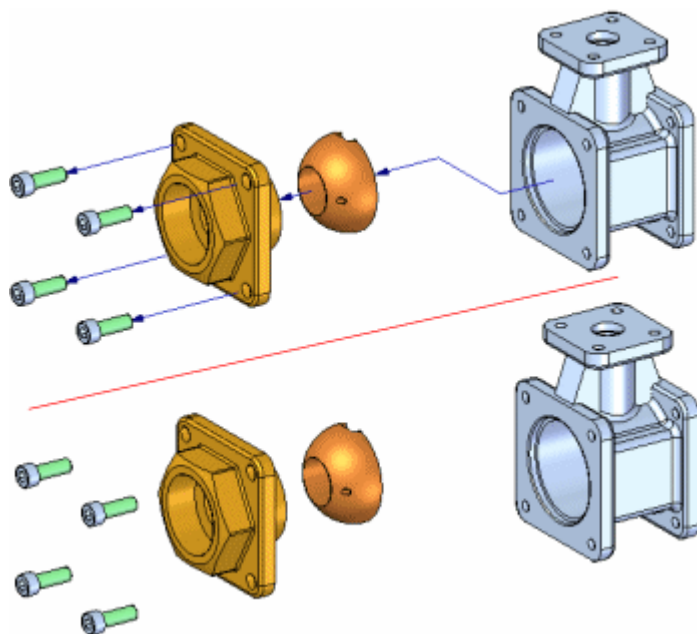
### **Lesson summary**

In this lesson, you used the Explode-Render-Animate application to explode an assembly. You accomplished the following:

- Used the Auto Explode command to begin an exploding sequence.
- Defined the distances and directions of exploding parts along a time line.
- Used the manual Explode command to order and sequence the events of an explosion.
- Grouped parts and subassemblies and controlled how they behave during and explosion and when they explode.
- Created an animation time line to be used in an animation sequence.

### Flow Lines command

Displays or hides all the flow lines between exploded parts.





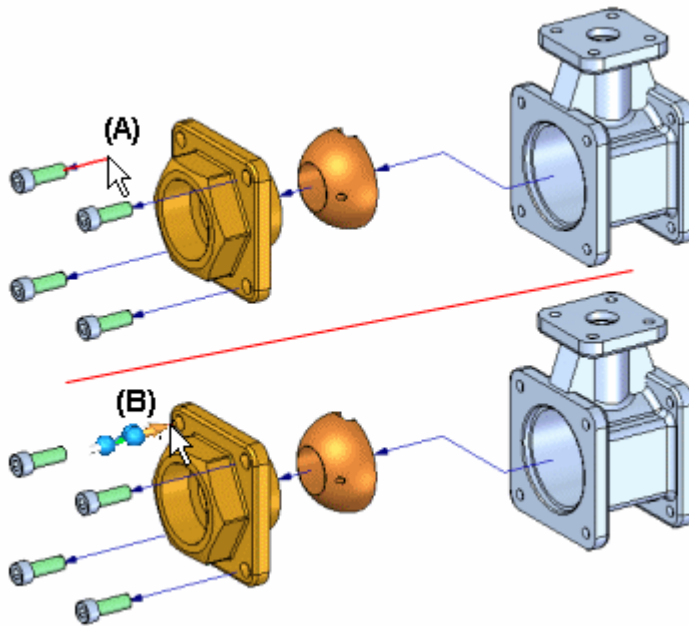
## Modify command

Edits a flow line between two exploded parts. You can make the following types of changes to a flow line:

- You can change the length of a flow line by editing the end point position of either end of the flow line.
- You can change the location of a joggle segment on a flow line.
- You change the location of the entire flow line.

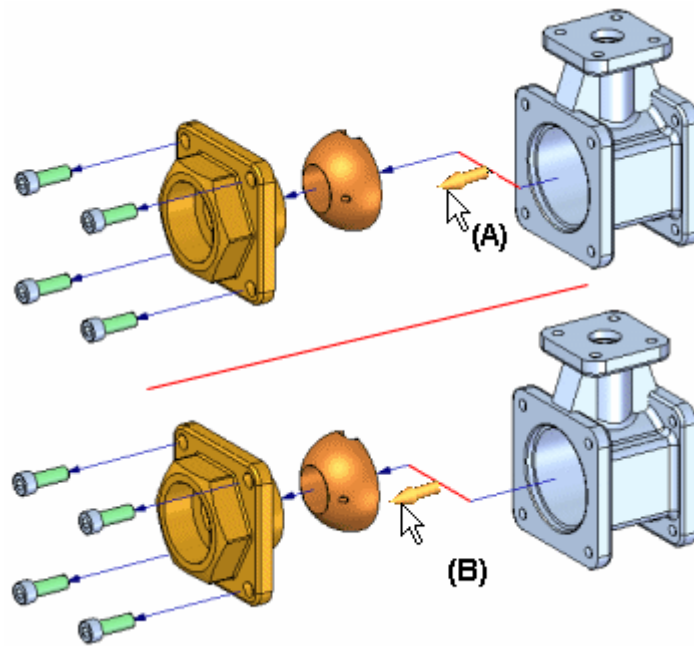
### Changing Flow Line Length

You change the length of the flow line by selecting it at the end you want to edit (A), then drag the cursor to the new position you want (B).



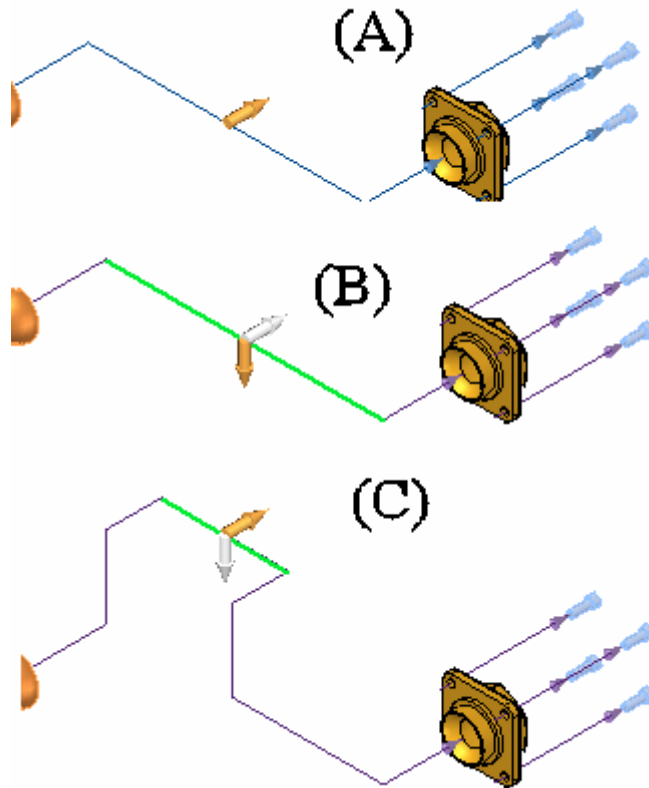
### Changing Joggle Segment Position

If the Move Exploded Part command is used to move the part outside of the original explode vector, a joggle is added to the flow line. You can use the Edit Flow Lines command to drag the joggle segment (A) to a new position (B).



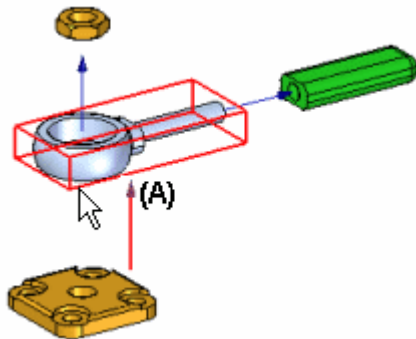
### Flow Line Handles

An event flow line mid-segment, or joggle, can be modified by dragging the handle (A). An annotation flow line can be modified by dragging either of the handles shown (B), and can be split (C) if needed.

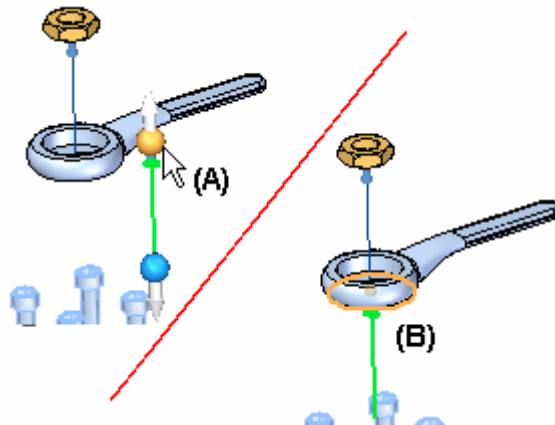


### Changing Flow Line Location

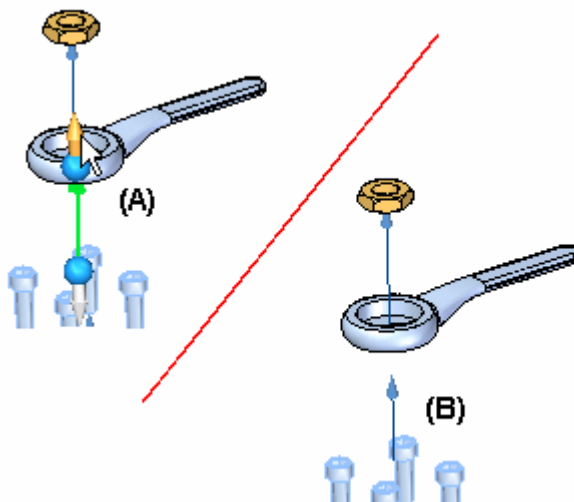
The location and length of a flow line is determined automatically by using the range box of the parent and child parts. The flow line terminator end originates at the center of the range box on the parent part. (A) The range box is the theoretical 3D envelope that the solid body is contained within. For some parts, you may want to change the flow line location.



To relocate an entire flow line, click near one end of the flow line (A), then click an edge or face that you want to connect that end of the flow line to (B).



The flow line position is updated (A). You may also want to change the flow line length so you can better view the flow line terminator (B) in the current view orientation.



### Other Flow Line Operations

You can also perform the following actions on flow lines:

- To display or hide flow line terminators, set or clear the Flow Line Terminators command on the View menu.
- To display or hide all flow lines, set or clear the Flow Lines command on the View menu.
- To display or hide an individual flow line, select the part in the graphic window or PathFinder, then click the Show Flow Lines or Hide Flow Lines commands on the shortcut menu.
- To delete a joggle segment on a flow line, select the proper event in the Explode PathFinder tab, then click the Delete command on the shortcut menu. This has the same effect as using the Move Exploded Part command to move the part back within an earlier explode vector.
- When you collapse a part with the Collapse command, the flow line is deleted.

The Show Flow Lines and Hide Flow Lines commands are also available on the shortcut menu when you select an explode event in the Explode PathFinder tab.

**Note**

You can only select flow lines in the graphics window with the Edit Flow Lines command, or in the Explode Pathfinder tab with the Select Tool.

## **Activity: Manipulating flow lines in an exploded assembly**

### **Activity objectives**

In this activity, you will accomplish the following:

- Modify event flow lines.
- Create and modify annotation flow lines.
- Delete annotation flow lines.
- Split annotation flow lines and then modify them.
- Place an exploded view on a drawing sheet.



## Activity: Manipulating flow lines in an exploded assembly

### Step 1

In this activity you open an assembly that has an exploded view that was previously created. The event flow lines will be modified, then converted to annotation flow lines with the drop command. During this activity, assembly components will be repositioned along with the flow lines associated with each component.

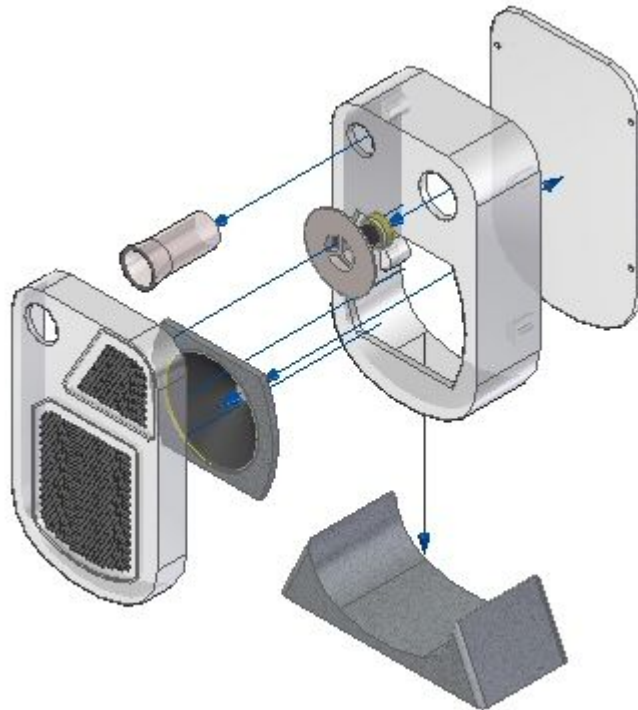
- ▶ Open the assembly *computer\_speaker.asm* with all the parts active.
- ▶ On the tools tab in the environs group, click ERA.



- ▶ On the home tab in the configurations group, click the display configurations command.



- ▶ Select the configuration named exploded. Click apply and then click close.



### Note

The flow lines in this display configuration are event flow lines.

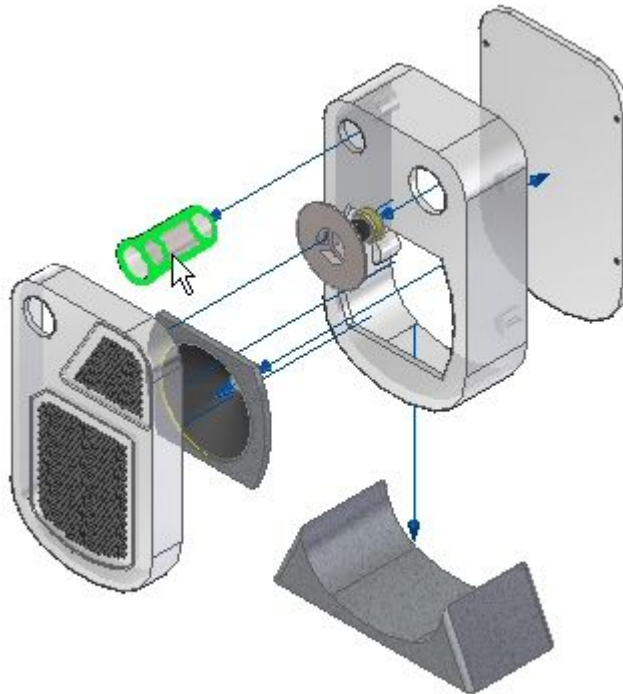
Step 2

The drag component command will be used modify an event flow line.

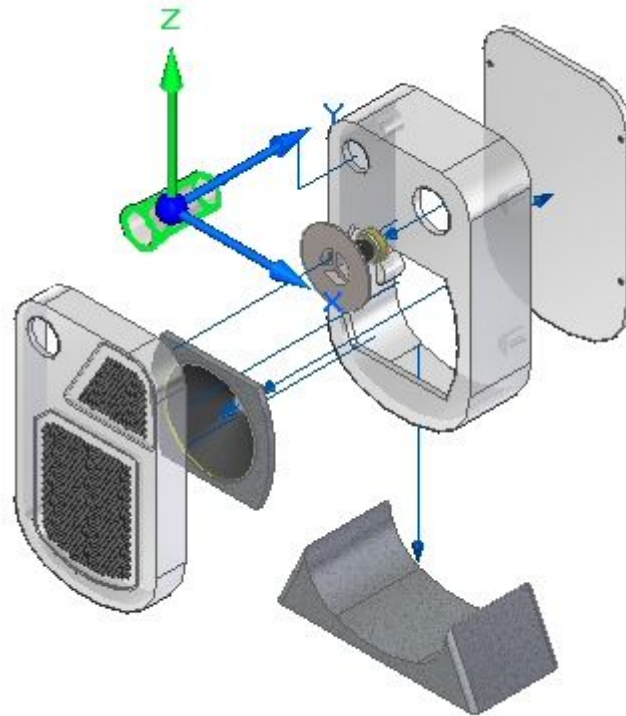
- ▶ Click the drag component command.



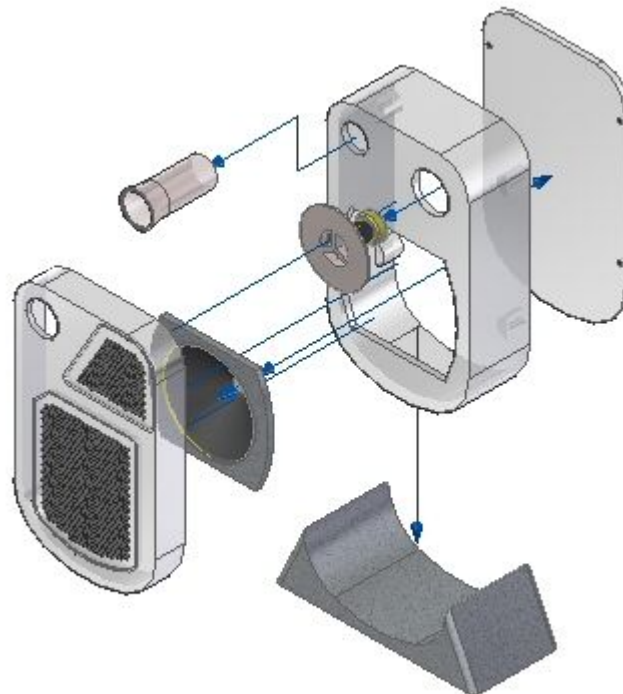
- ▶ Select the assembly component *woofer\_tube1.par* as shown and accept.



- ▶ Select the Z axis and enter the distance value of 50 mm.



- ▶ Click the select tool to exit the drag command. The flow line has been modified as shown.



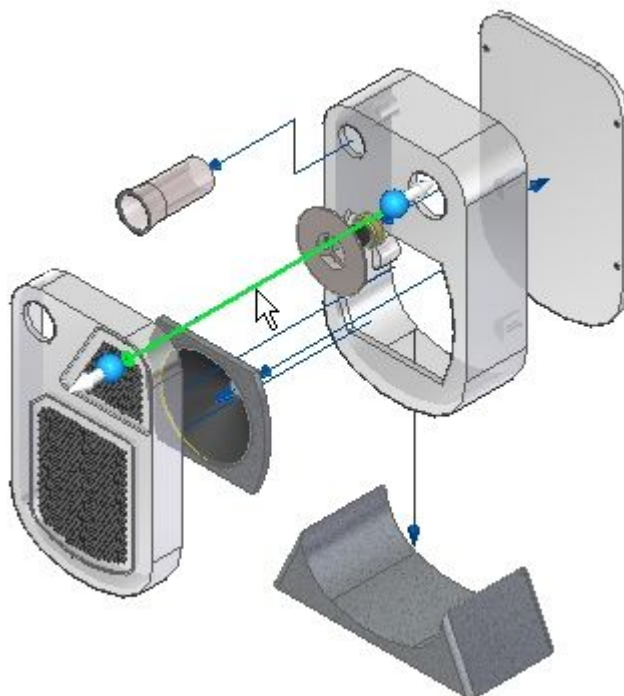
*Step 3*

An event flow line will be repositioned.

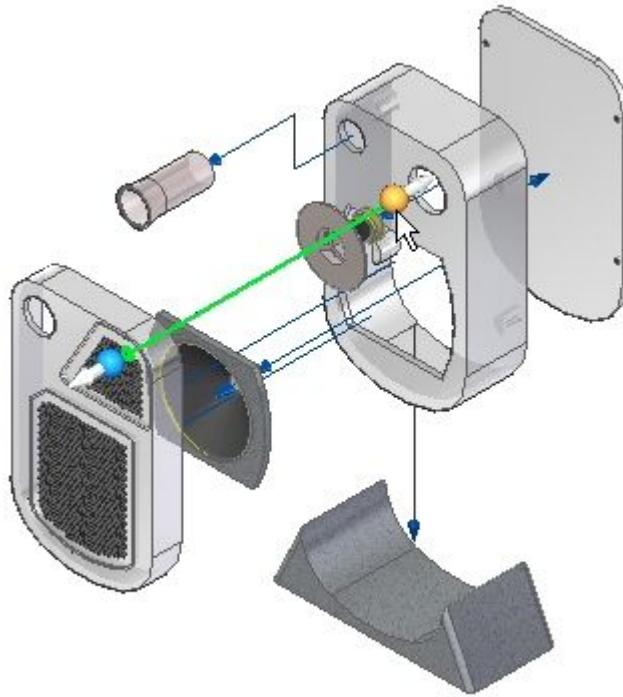
- ▶ Click the modify command.



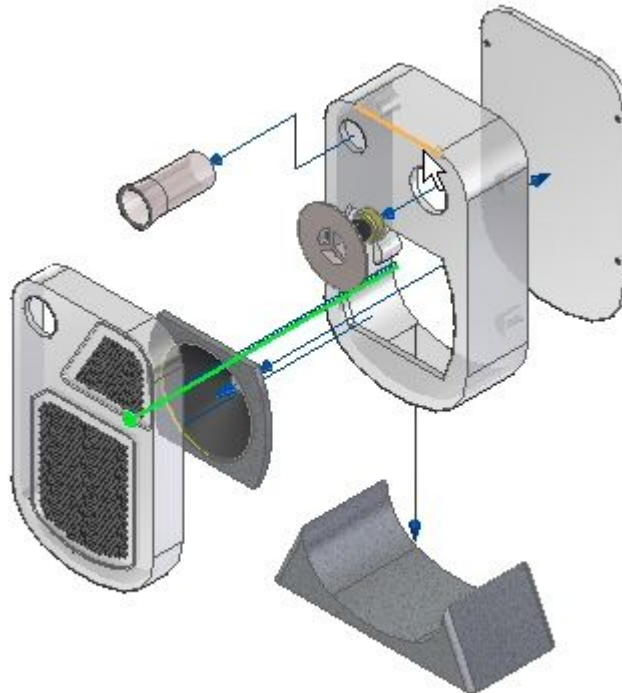
- ▶ Select the flow line shown.



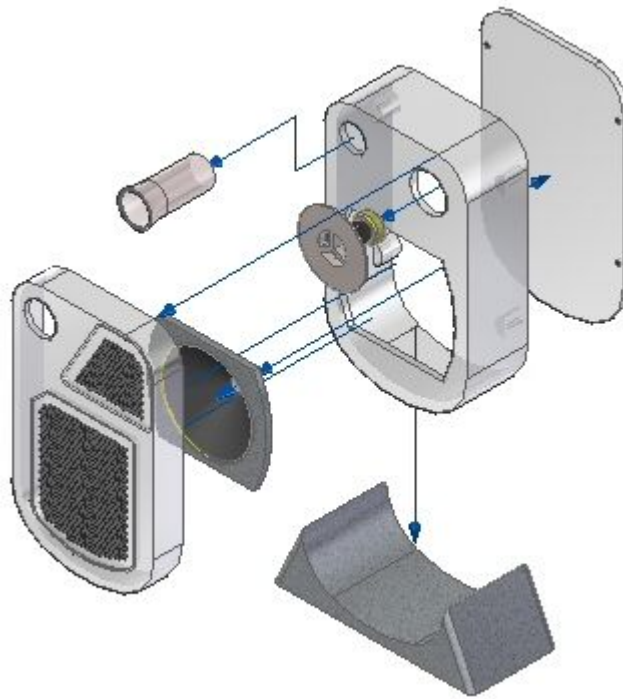
- ▶ Click the handle shown.



- ▶ Select the end point of the line segment shown.



- ▶ The flow line is moved as shown.



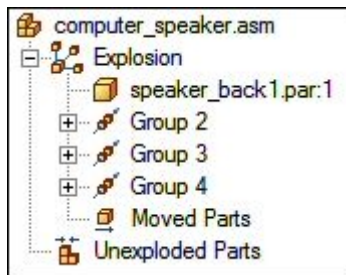
- ▶ Save the display configuration.



Step 4

Event flow lines will be converted to annotation flow lines.

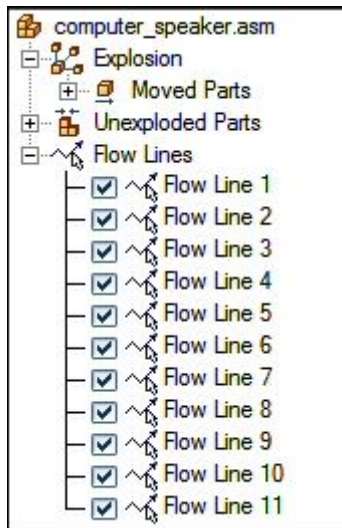
- ▶ Observe the explode pathfinder. Event flow lines are not displayed in explode pathfinder.



- ▶ Click the drop command.



- ▶ Observe the explode pathfinder. The event flows lines have been converted to annotation flow lines.



**Note**

The event flow lines have not been lost. The event flow lines are stored and saved in the display configuration that was saved prior to dropping the flow lines. A display configuration with the annotation flow lines should be saved with a different display configuration name.

- ▶ Click the display configurations command.



- ▶ Click New. Enter annotation then click OK. Click Close

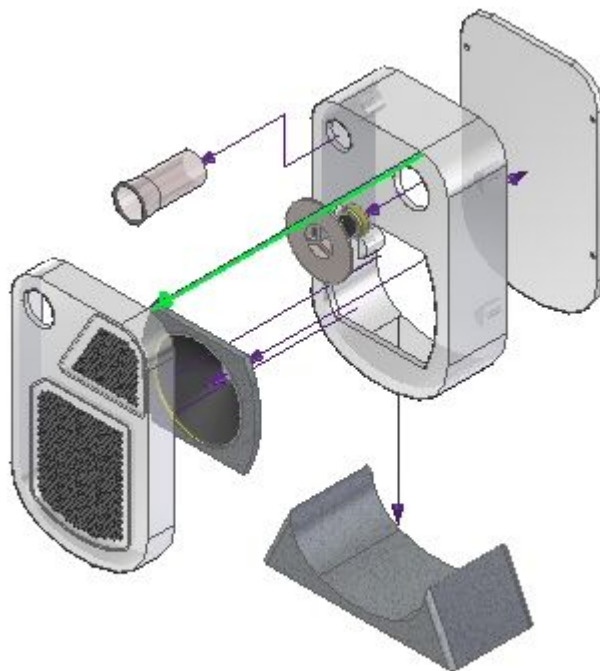
**Note**

The display configuration containing the annotation flow lines has been saved.

*Step 5*

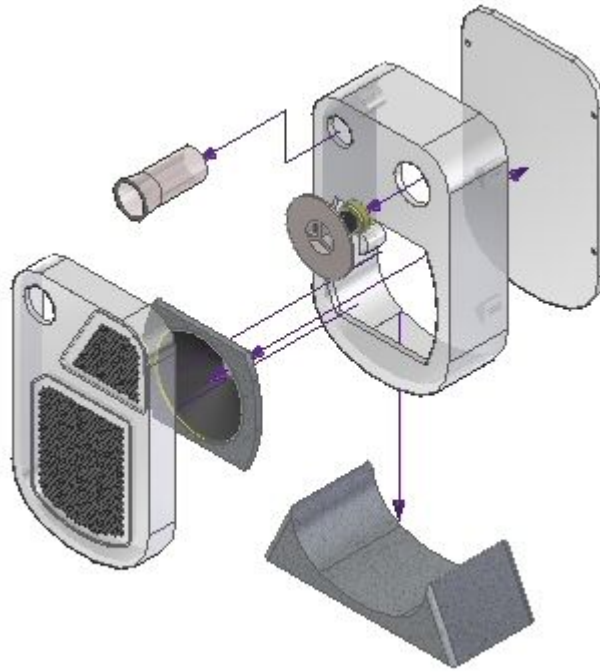
An annotation flow line will be deleted and replaced by a new annotation flow line.

- In the explode pathfinder, select flow line 3.





- ▶ Delete flow line 3.



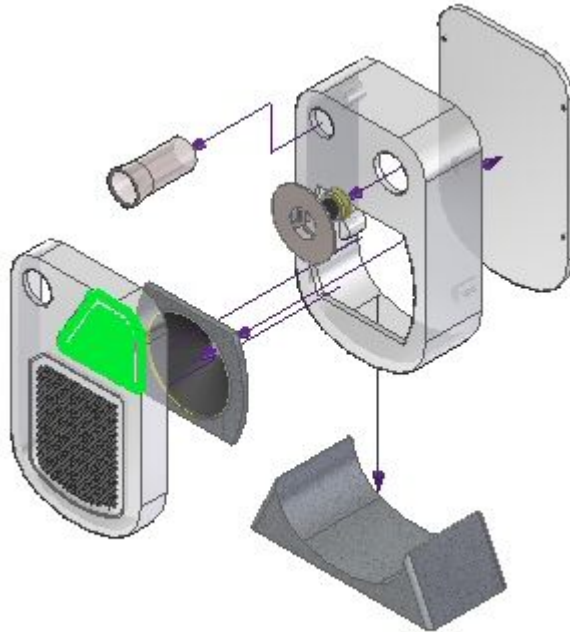
**Note**

The deleted flow line was connected to the part *grate2\_1.par*.

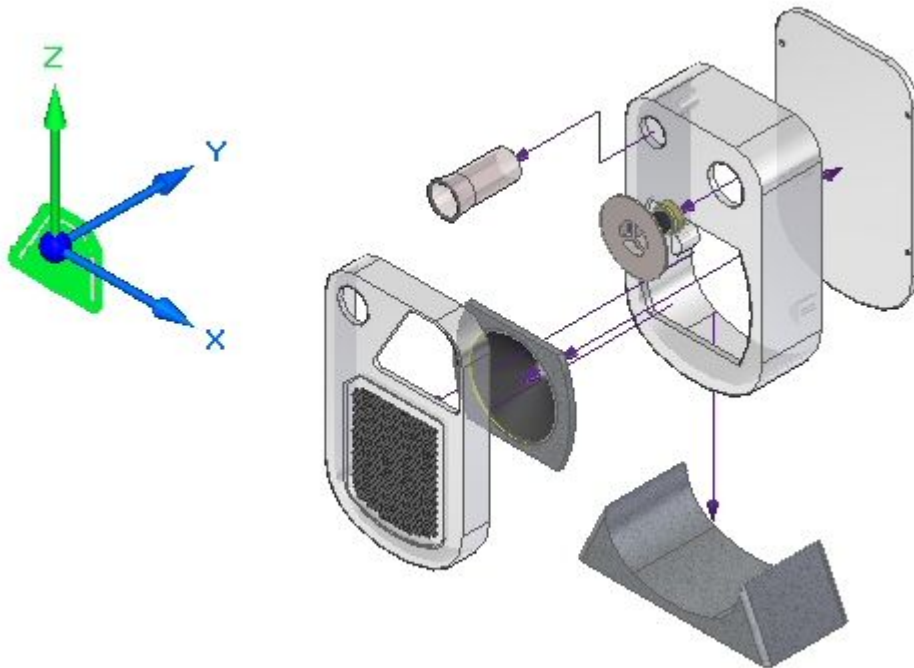
- ▶ Click the drag component command.



- ▶ Select *grate2\_1.par* and accept.



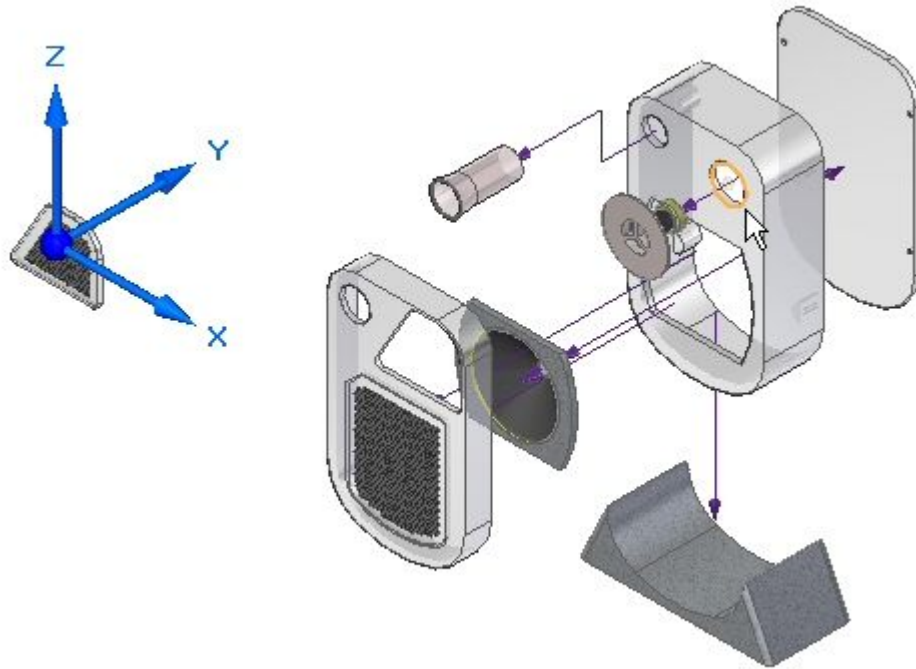
- ▶ Move the *grate2\_1.par* -300 mm in the Y direction, -100 in the X direction, and 200 mm in the Z direction.



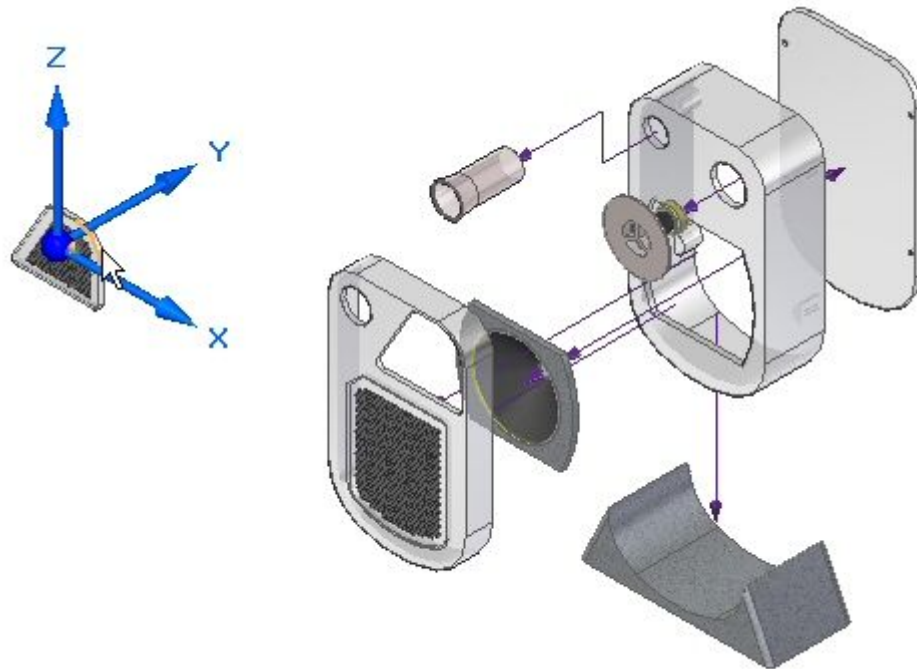
- ▶ A new annotation flow line will be created. Click the draw command.



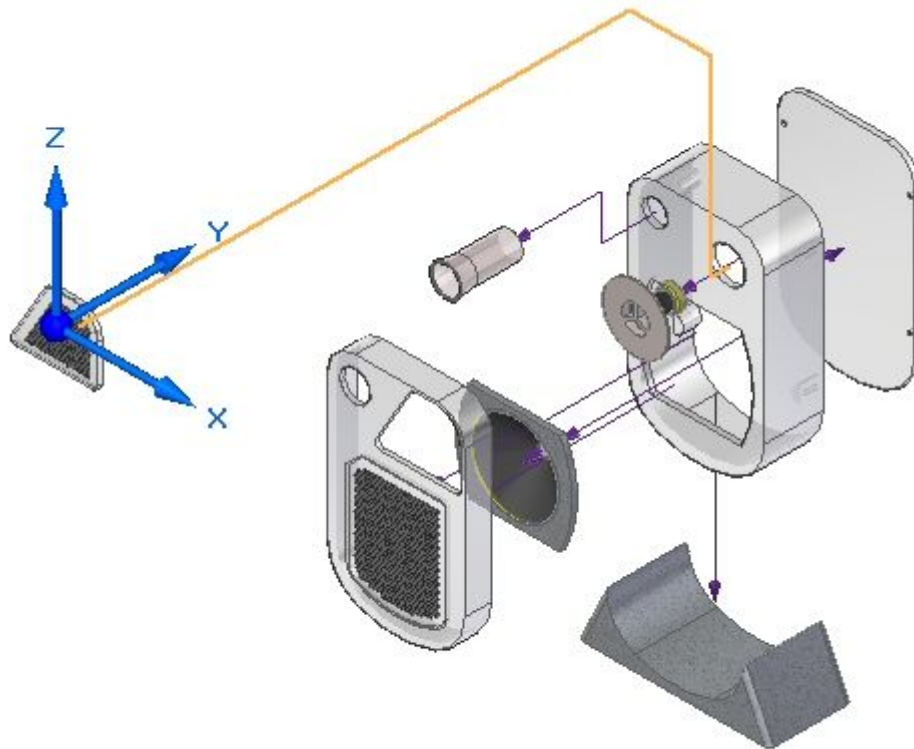
- ▶ For the point of the flow line, click the circle shown.



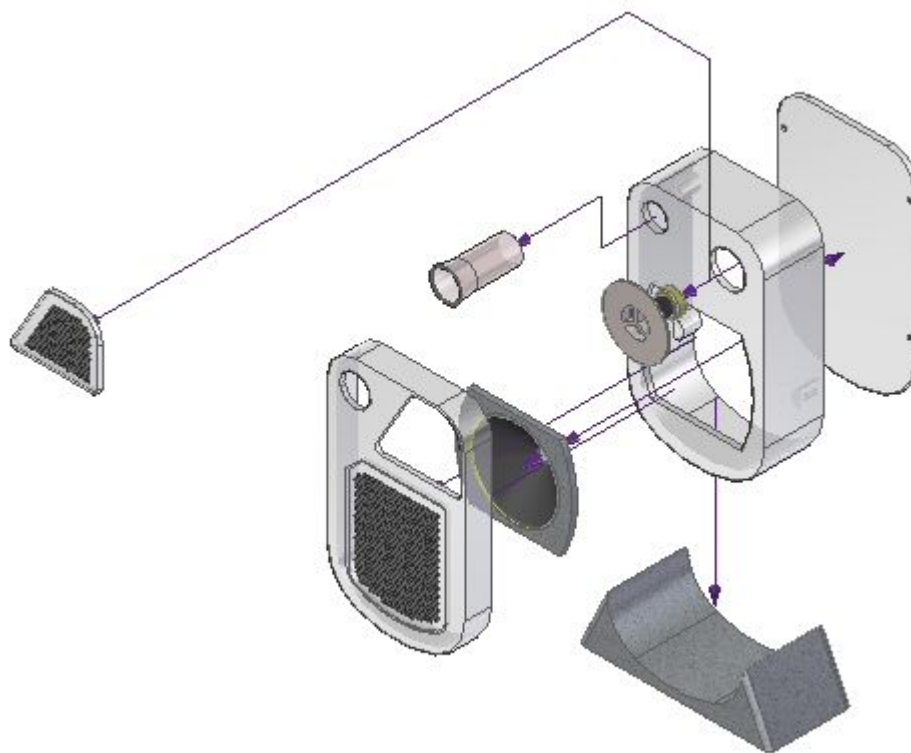
- For the end of the flow line, click the circle shown.



- ▶ Click the draw previous and draw next buttons to get the result shown. Click Finish and then click Cancel.



An annotation flow line is created.



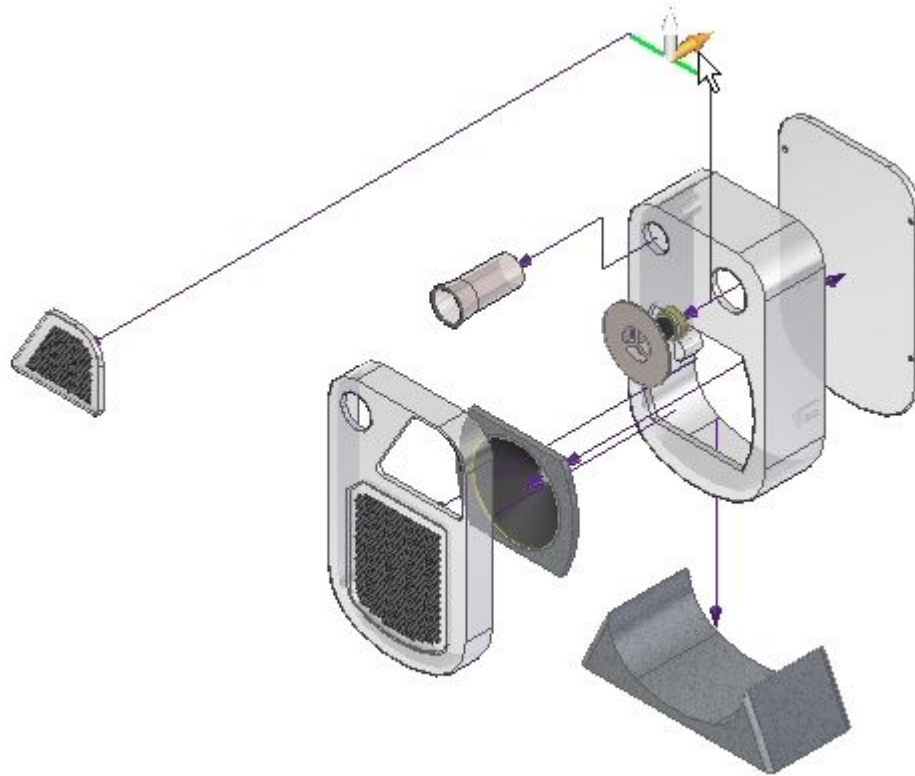
*Step 6*

An annotation flow line will be repositioned.

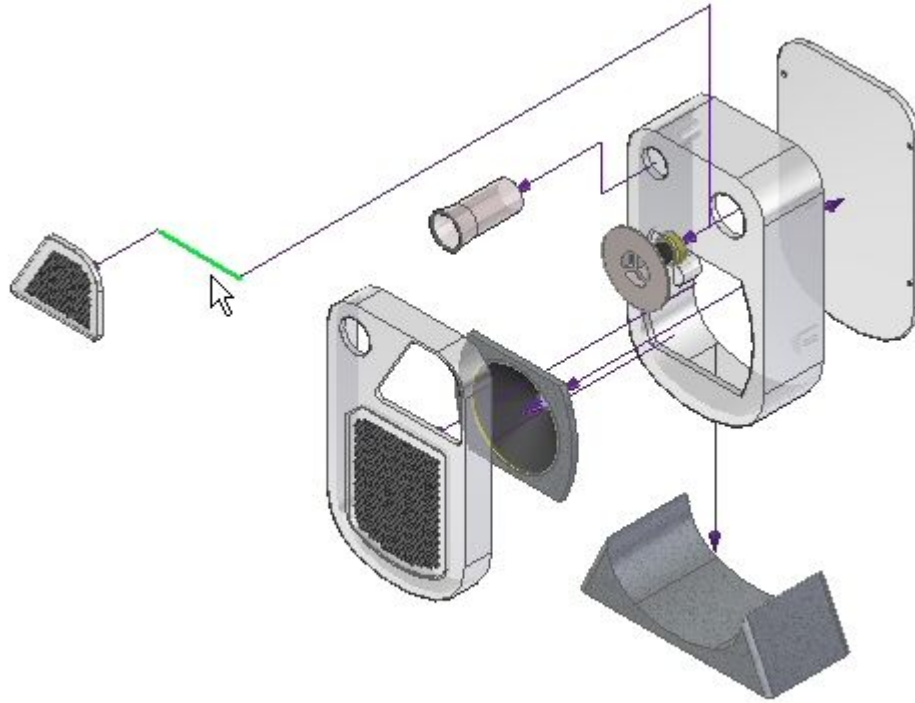
- ▶ Click the modify command.



- ▶ Select the segment handle as shown.



- ▶ Drag the segment to the approximate position shown.

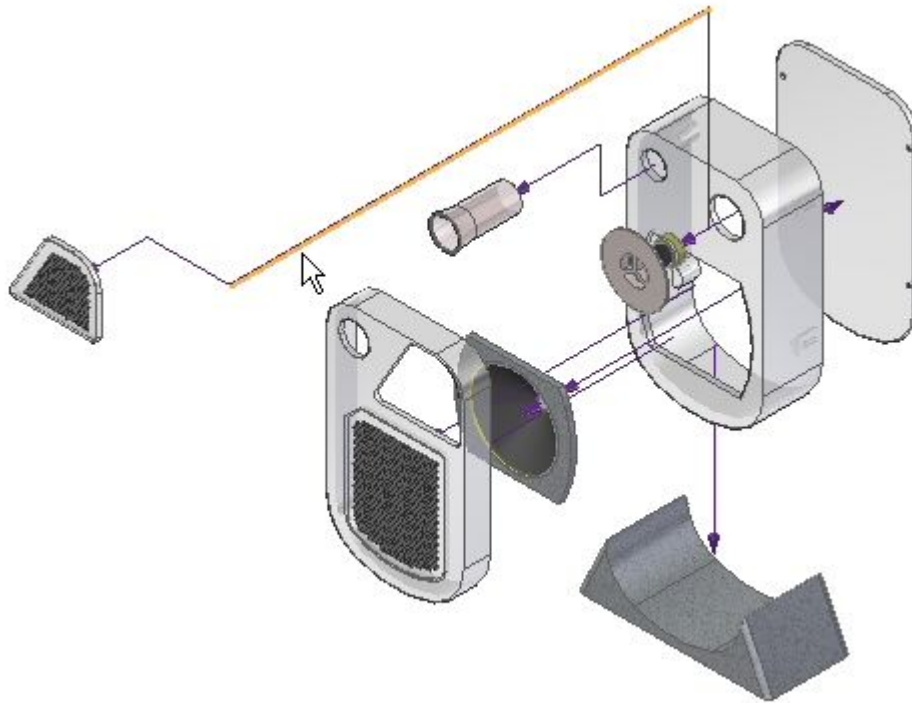


- ▶ Click the split flow line command.





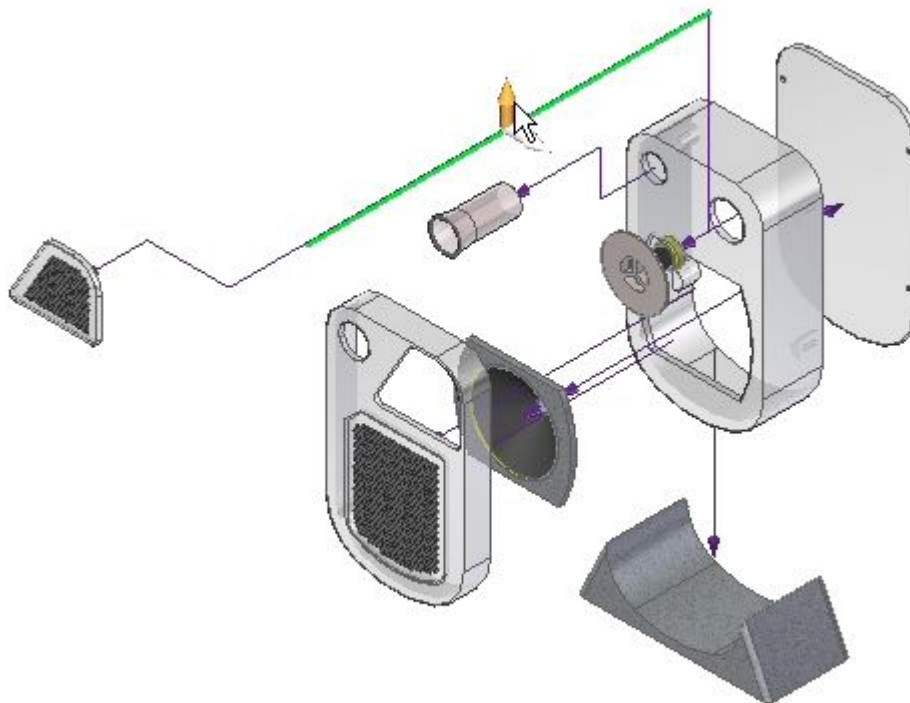
- ▶ Select the flow line shown in the approximate position shown.



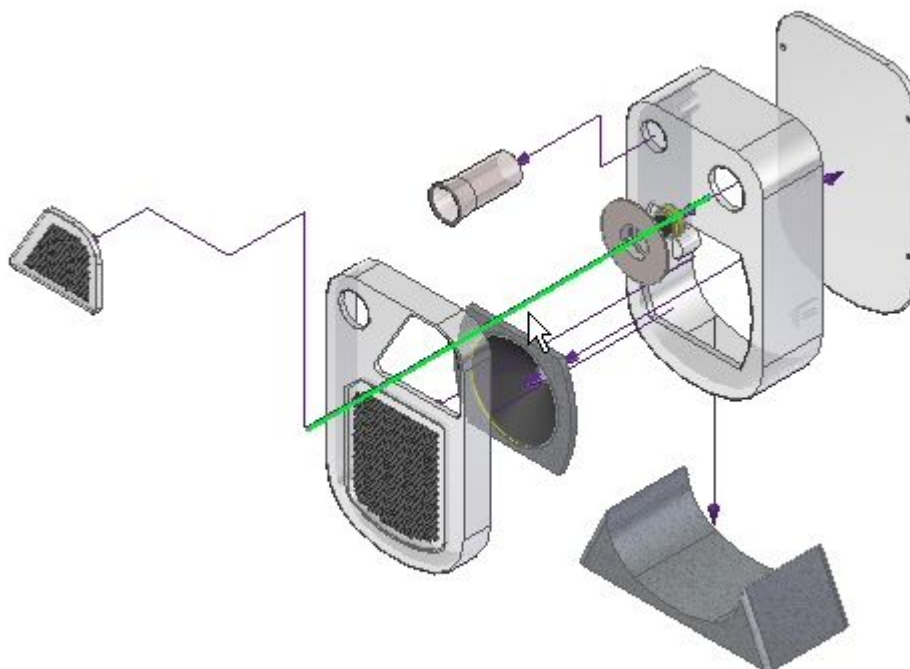
- ▶ Click the modify command.



- ▶ Select the segment handle as shown.



- ▶ Drag the segment to the approximate position shown.



- ▶ Save the display configuration.



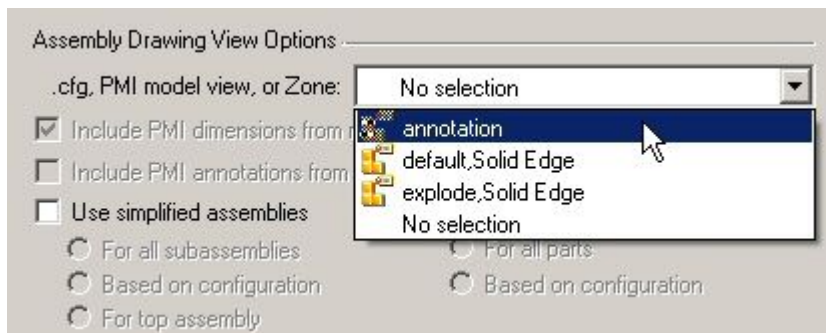
- ▶ Close ERA and save the assembly.



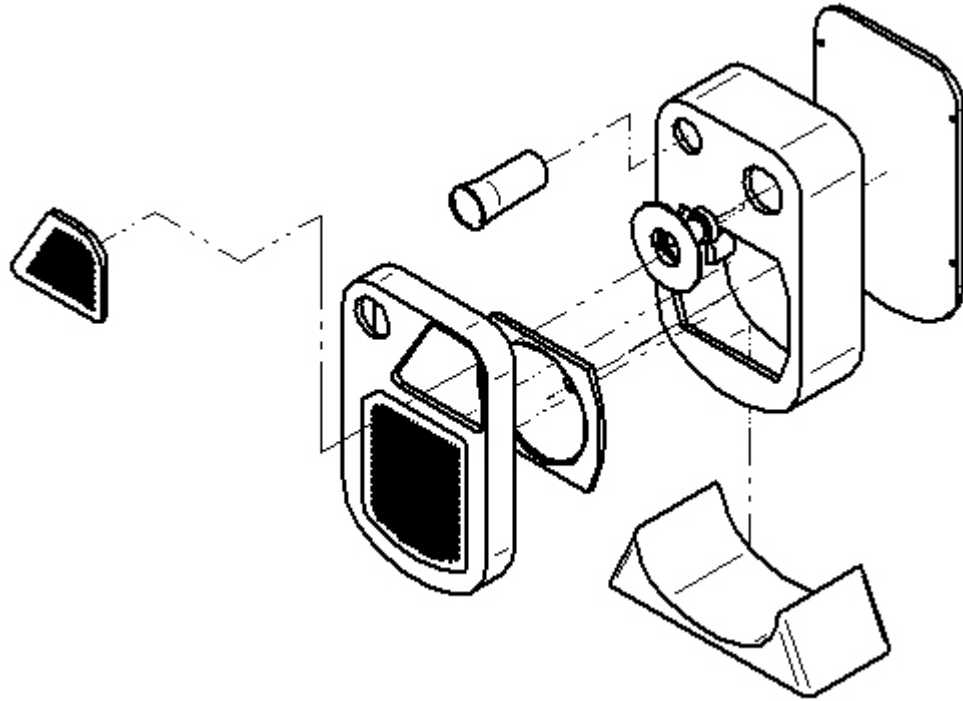
*Step 7*

Create a drawing sheet of the exploded view.

- ▶ Click the Solid Edge Application button. Then click New>Create Drawing. Click OK when prompted for the default template.
- ▶ Select the display configuration named annotation in the drawing view wizard.



- Place the exploded view on the drawing sheet.



- Save and close all files. This completes this activity.

*Activity summary*

In this activity, you used the Explode-Render-Animate application to accomplish the following:

- Modify event flow lines.
- Create and modify annotation flow lines.
- Delete annotation flow lines.
- Split annotation flow lines and then modify them.
- Place an exploded view on a drawing sheet.

**Lesson review**

Answer the following questions:

1. What is the difference between an event flow line and an annotation flow line?
2. Can annotation flow lines be stored in a display configuration?
3. Can annotation flow be split and segments modified?

**Answers**

1. What is the difference between an event flow line and an annotation flow line?  
Event flow lines control animation explode events, and are created with the explode commands. Dropping the flow lines changes them into annotation flow lines which can be used in the draft environment.
2. Can annotation flow lines be stored in a display configuration?  
Annotation flow lines must be stored in a display configuration to later be placed on a drawing sheet.
3. Can annotation flow be split and segments modified?  
Annotation flow lines can be split into segments and each segment dragged to a new position.

### **Lesson summary**

In this lesson you learned that event flow lines, which control animation events can be changed into annotation flow lines for better assembly annotation of exploded views. As an annotation flow line, they are easily modified to produce high quality exploded views of an assembly.





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**Lesson**

# *15 Rendering assemblies*

## Course Overview

### Course overview

The Explode-Render-Animate application within the Solid Edge assembly environment is a tool for creating different types of presentations of Solid Edge assemblies. Exploding an assembly allows you to control the movement, sequence and grouping of parts and subassemblies. Rendering a view allows you to define textures, lighting, shadows, backgrounds and other properties to create presentation style images. Motors apply movement to under constrained parts in an assembly which can be animated. Using animation, you can combine previously created exploding sequences and custom camera movement to create animation. Each frame of the animation can be rendered to create presentation quality animations.

Once you complete the activities in this course, you will be able to:

- Assign material properties and textures to parts and subassemblies.
- Set viewing properties to change lighting, backgrounds, shadows, reflections, refractions and perspectives.
- Edit material properties to create different results in the final rendering.

## Rendering

Limited rendering is available in Solid Edge assembly using the settings in the View command. Advanced rendering in assembly is covered here and is contained in the Explode-Render-Animate environment.



## Working with advanced rendering



The advanced rendering capabilities in Solid Edge are an extension of the Explode-Render-Animate application. This functionality can enhance the quality of images used in documents such as customer presentations or marketing and sales literature.

With the advanced rendering capability, you have a single view style called presentation view style that contains all of your rendering settings.

A library of predefined entities such as materials, backgrounds, render modes, and light studios is available. You can customize the library with user-defined settings, but you cannot edit, delete, or modify the contents of the original library. This ensures protection for the original data stored in the library. Any custom settings you add to the library are stored in a separate file. You can have multiple libraries.

The advanced rendering capability uses the Solid Edge display for view manipulations. It supports part visibility and will only render what is on the screen. The Sharpen command is available when working with advanced rendering to allow you to make your graphics crisp and clear. Sharpening the display of the model will increase the quality of the rendering.

### Supported advanced rendering entities

Advanced rendering provides rendering support for these entities:

- [Backgrounds](#)
- [Foregrounds](#)
- [Materials](#)

- Environments
- [Lighting Studios](#)
- [Render Mode](#)
- [Scenery](#)
- [Schemes](#)

### **Activating advanced rendering**

Advanced rendering is activated automatically when you click the Render Scene and Render Area buttons when you have a Solid Edge Classic license. When you click these commands, two new tabs are added to PathFinder automatically.

- The Session Entities tab displays the name of the active assembly, along with a tree structure that shows you the entities that are applied to the assembly. You can right-click an entity to display a shortcut menu to:
  - Edit the properties of an entity as the basis for creating a new entity.
  - Detach (remove) a material that you have applied to the model.
  - Cut, copy, and paste an entity.
  - Rename an entity you created.
- The Predefined Archives tab displays a list of folders that contain predefined entities, such as backgrounds, foregrounds, and materials. You can right-click an entity to display a shortcut menu to:
  - Create a new archive folder or advanced rendering entity.
  - Apply an entity to the model.
  - Cut, copy, and paste an entity.
  - Rename an entity you created.
  - A toolbar on the Predefined Archives page contains commands used to create, save, open, close, and import customized entities in user-defined archive folders.

Any material, color, background, light, or scene that you want to customize needs to be applied to the assembly first, and then modified within the Session Entities tab to look the way you want it to. When you have adjusted the parameters of an entity so that it looks good on the model, you then copy the entity from the Session Entities page to a user-defined entity archive on the Predefined Archives page on PathFinder.

### **Editing advanced rendering entities**

You can use the <Entity> Editor dialog box to edit the advanced rendering entity settings. To display the dialog box, click the Sessions Entity tab on PathFinder. Right-click the entity you want to edit. For example, if you want to change the

material settings for the rendering, right click Materials. On the shortcut menu, click Edit Definition.

The options on the dialog box change based on the shader you select. A *shader* is designed to imitate a real-world look for things such as material, light, and rendering mode. Every component in advanced rendering is based on a shader, and each shader has multiple options that allow you to control the look and feel of the renderings.

### **Applying advanced rendering entities**

There are two ways to apply an advanced rendering entity.

- You can select the entity from the library in the Predefined Archives tab and drag it onto the model in the graphics window.
- You can first select a model part in the graphics window, then right-click an entity in the Predefined Archives tab and choose Apply To Selected on its shortcut menu.

After applying the entity, you can use the Render Scene or Render Area command to see what the change looks like in the model.

### **Saving an advanced rendered image**

You can use the Save As command to save a rendered image.

### **Guidelines for saving advanced rendered images**

When saving images, you should follow these guidelines.

- To calculate the memory requirements for the image, multiply the image size (in pixels) \* 4 (true color). For example, for an image 500 x 500 pixels, the memory requirements are 10 MB (500 x 500 x 4).
- To calculate the file size for the image, multiply the image size \* 3 (true color).
- On the Image Options form, set the Alternate View Style to Presentation View Style.

### **Backgrounds**

Backgrounds are a simple way of enhancing the screen area behind a model. They appear in reflective surfaces so they can impact a model's appearance and add context to the model's image.



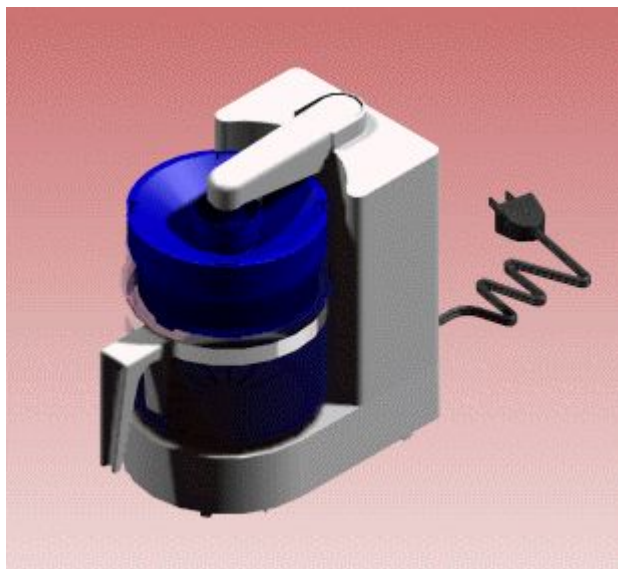
**Note**

Backgrounds in advanced rendering inherit the background in Assembly.

Advanced rendering supports the following background shaders.

- Graduated

A smooth linear transition from one color to another. In other words, there is a gradual change in background colors as you go from the top to the bottom.

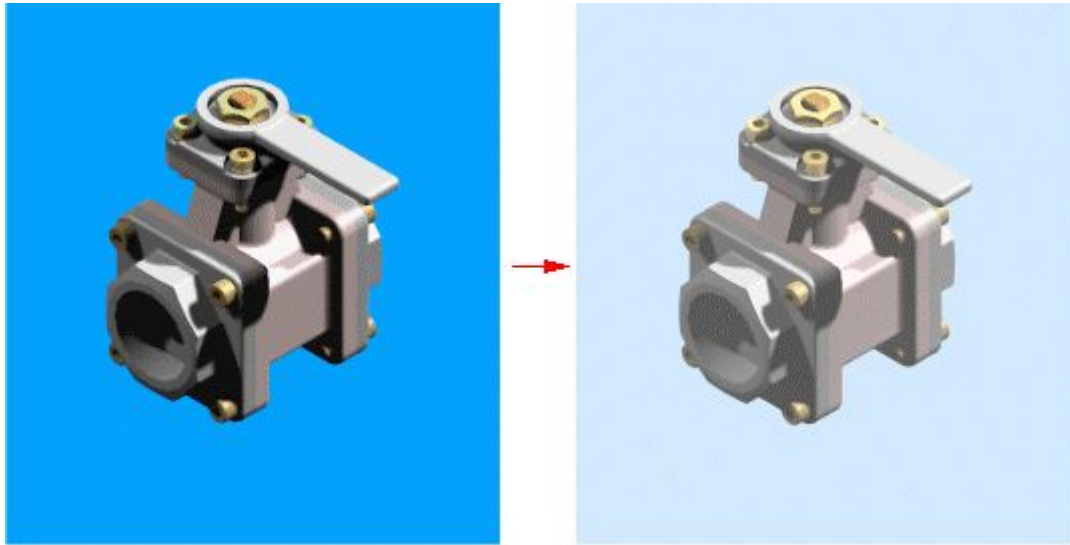


- Image
- Plain
- None

To learn how to stretch a background image so that it fills the window, see the Help topic, *Set a Background Image in Explode-Render-Animate*.

### **Foregrounds**

Foregrounds allow you to add additional view effects that help simulate various atmospheric effects such as fog. You can add foregrounds from the library or you can edit the view settings directly.



**Note**

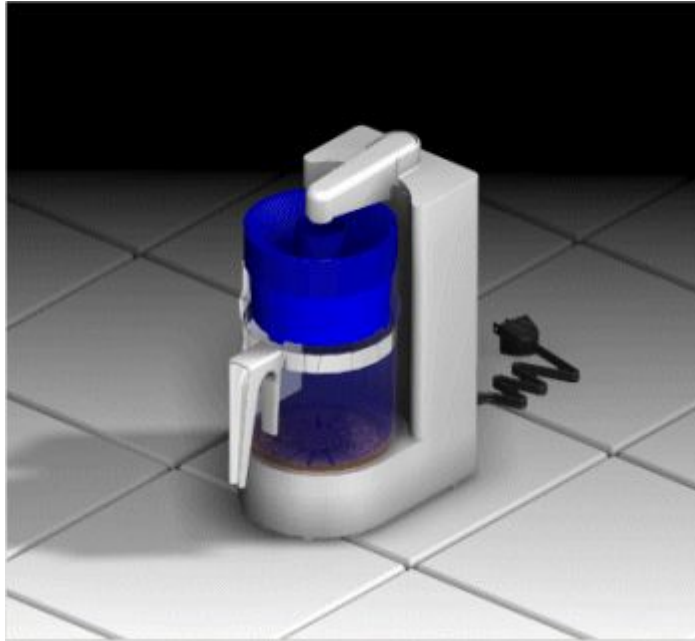
Foregrounds in advanced rendering do not inherit any assembly settings.

Advanced rendering supports the following foreground shaders.



- Depth Cue

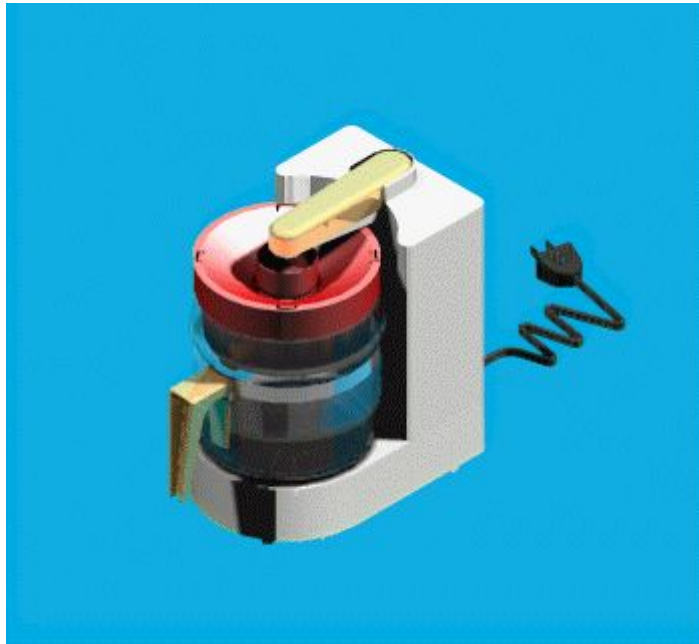
Depth Cue fades an image to a particular color to portray a sense of depth in the image.



- Fog
- None

### **Materials**

Advanced rendering includes a wide range of materials to simulate such things as wood, plastic, marble, and metal. It also supports advanced surface effects for texture maps and bump maps.



The material definition is inherited from the style definition in the assembly. For example, a part with a green style in the Assembly environment will initially be green material in advanced rendering. The materials applied to a model in advanced rendering are available only when using the Render Scene and Render Area commands in the Explode-Render-Animate application. For example, if you apply a red material to a green part, the part will still be green when you are not using these commands.

You can apply a material to a single or multiple occurrences of parts. For example, suppose you have several parts with a material definition of blue. If you change the material definition to dark blue, all parts with blue material are affected.

Advanced rendering supports several properties and allows you to set the following settings:

- Color
- Reflectance
- Transparency
- Displacement
- Texture space

When working in Explode-Render-Animate, you can modify the existing materials to create different colors and properties. Once that is done, you can create a new archive (.lwa file) for the modified entities, so you can apply them to different projects whenever you want.

### **Color support**

You can define colors as plain colors, patterns such as wood, granite, and marble, or a textured map based on a bitmap image. The color settings support shaders such as plain, woods, marble, and wrapped images.

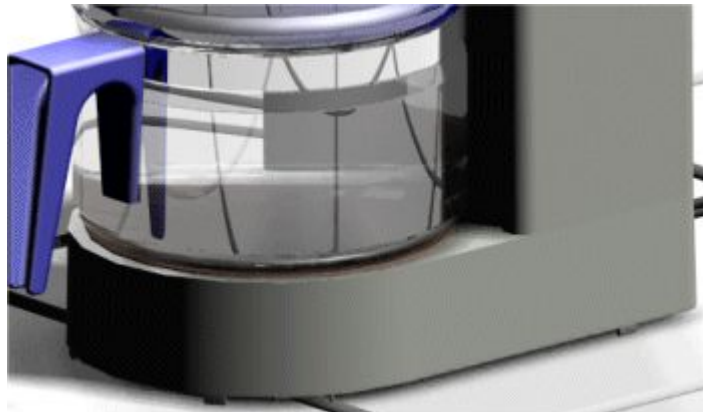
## Reflectance support

Reflectance properties affect the way light interacts with the materials. You can use these properties to apply such effects as:

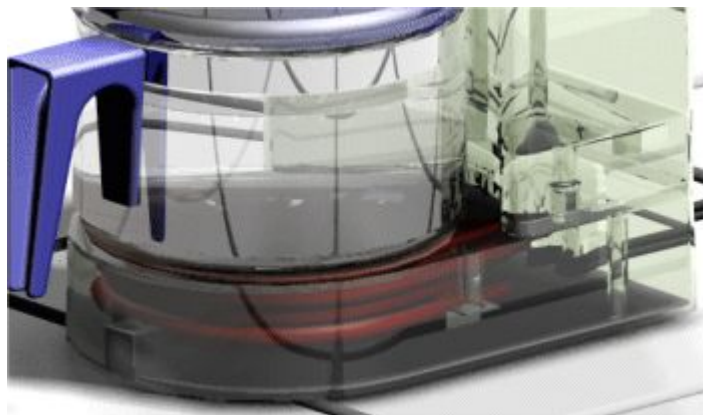
- Mirror reflectance



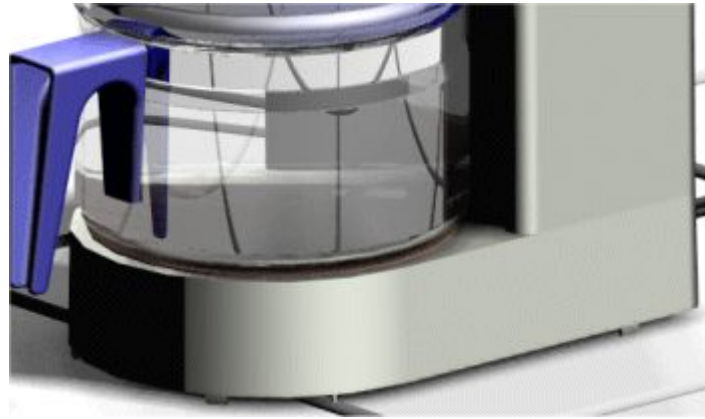
- Plastic reflectance



- Glass reflectance



- Metal reflectance

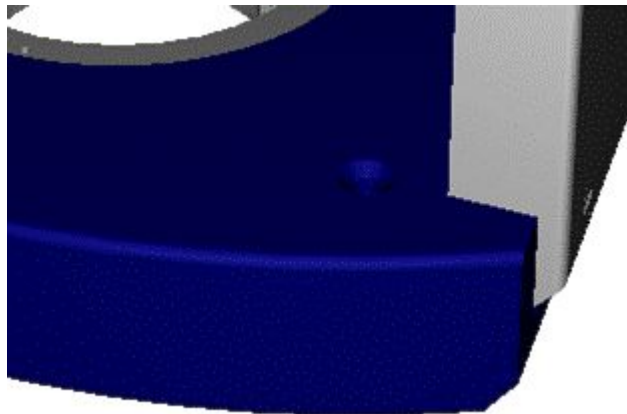


The reflectance settings support shaders such as chrome 2D, conductor, glass, matte, metal, and mirror. Some of the most common options for these shaders include:

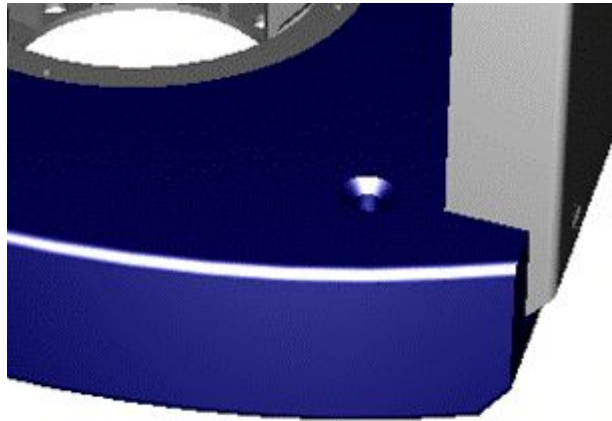
- Specular factor

The specular factor is the amount of highlight or light reflected from a triangle where its normal matches the light vector.

With a low specular factor, the appearance of the blue plastic seems to be dull,



compared to a high specular factor that produces a shinier, almost mirror-like effect.



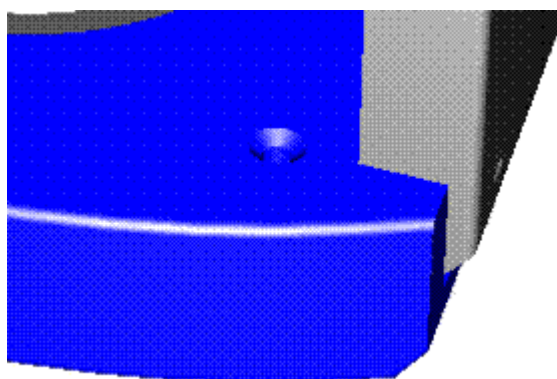
– Diffuse factor

The diffuse factor is the amount of light reflected from triangles where the light is approximately 45 degrees from the light ray.

A low diffuse factor produces a dark image,



compared to a high diffuse factor.

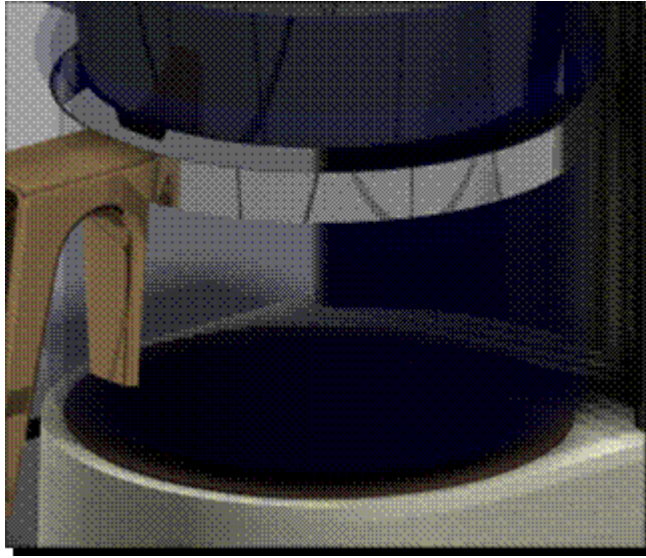


– Mirror factor

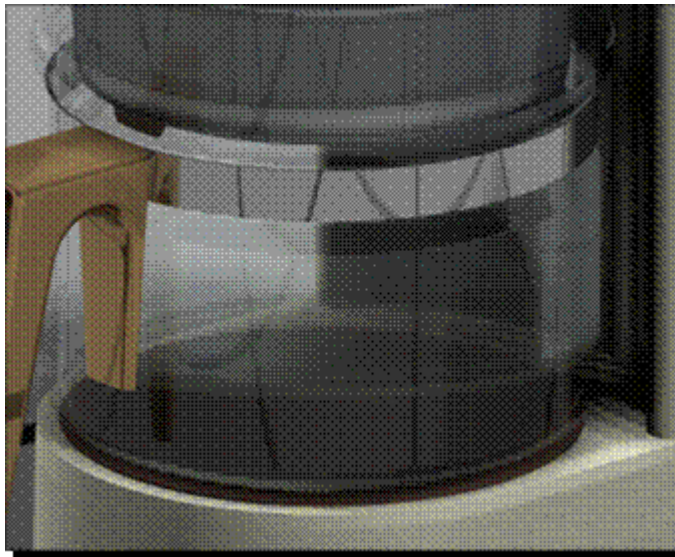
### **Transparency support**

Transparency is the amount of coverage by a color filter to simulate glass or some plastic materials. The values range from 0 for transparent to 1 for opaque. The transparency settings support shaders such as none, glow, plain, and wrapped image.

You can apply simple transparency to a model,



or combine it with properties such as reflectance, you can produce a more accurate looking image.



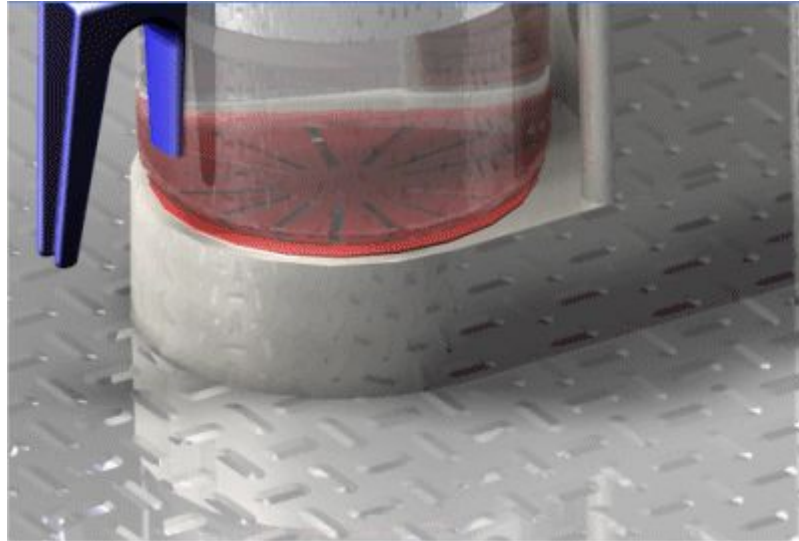
### **Displacement support**

Surface displacement defines surface deformations by applying such effects as simple surface roughness, patterns such as leather and dimples, or tread plates. The displacement settings support such shaders as none, casting, leather, rough, wrapped displacements.

Wrapped displacement shaders simulate materials with an imprinted pattern. Some of the common wrapped displacement shaders include:

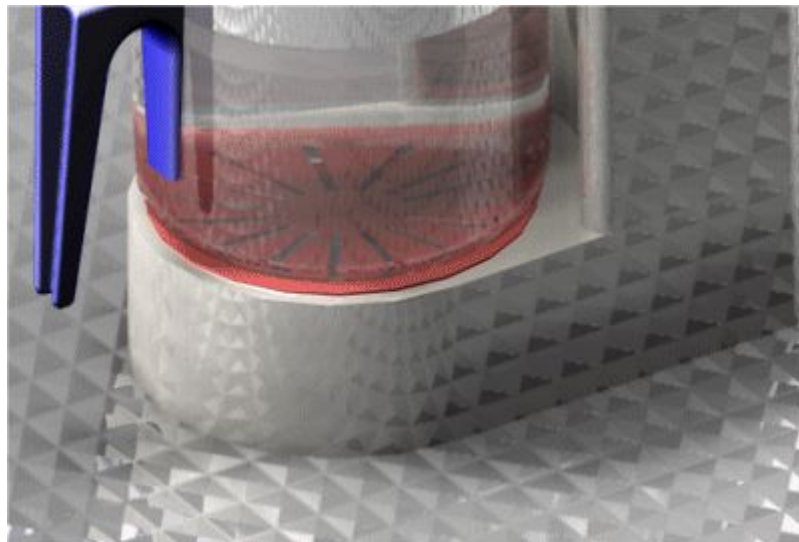
- Wrapped tread plate

Wrapped tread plates imitates slip resistant shaped tread pattern. Also known as diamond plate, this material is commonly used for outdoor steps.



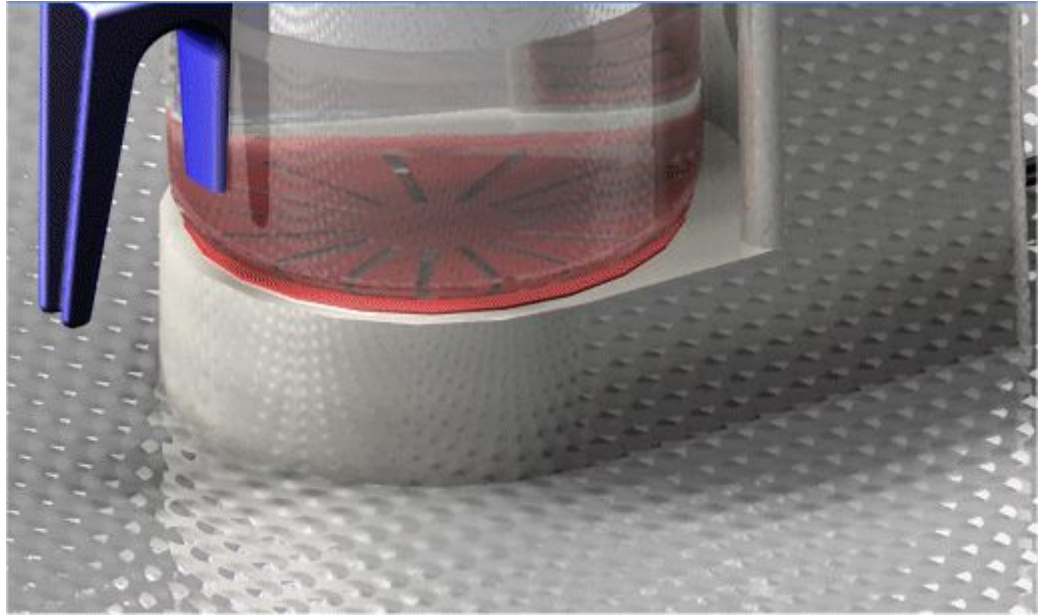
- Wrapped knurl

Wrapped knurl imitates handgrips and is a common finish for barbell handgrips.



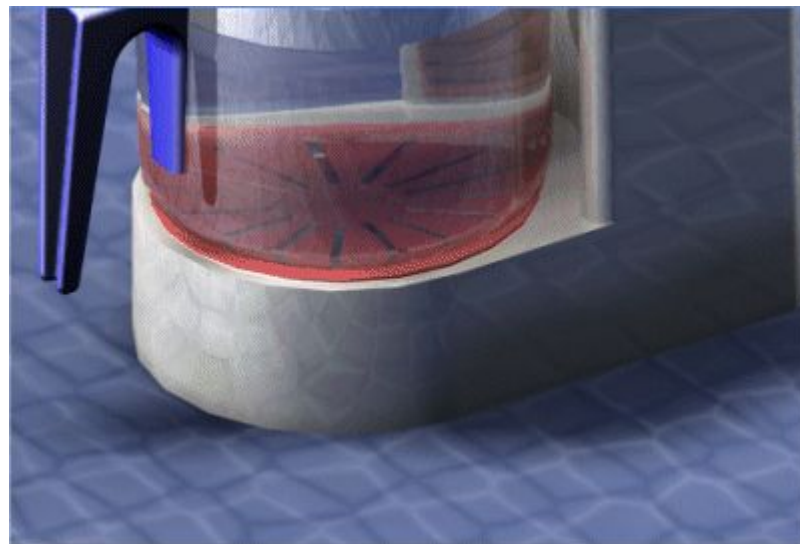
- Wrapped dimple

Wrapped dimple simulates a slip resistant bubble-shaped pattern. Also known as diamond plate, this material is commonly used for outdoor steps.



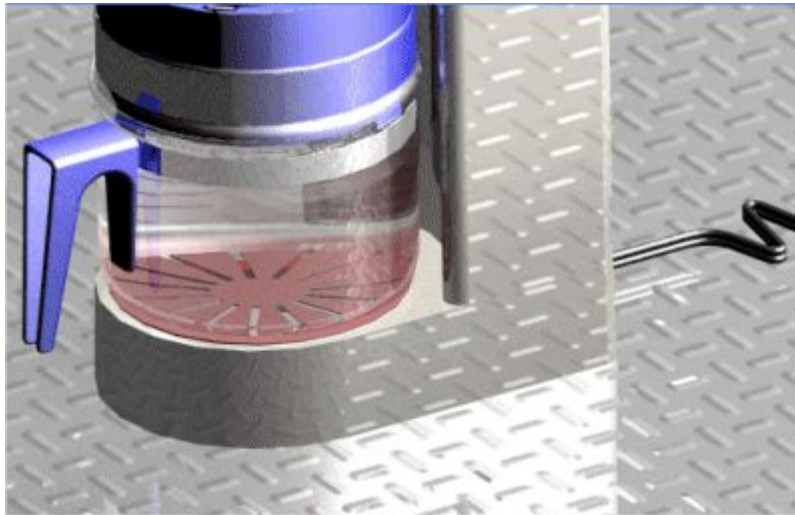
- Wrapped leather

Wrapped leather imitates leather and is commonly used for camera bodies, chairs, and desk ground.

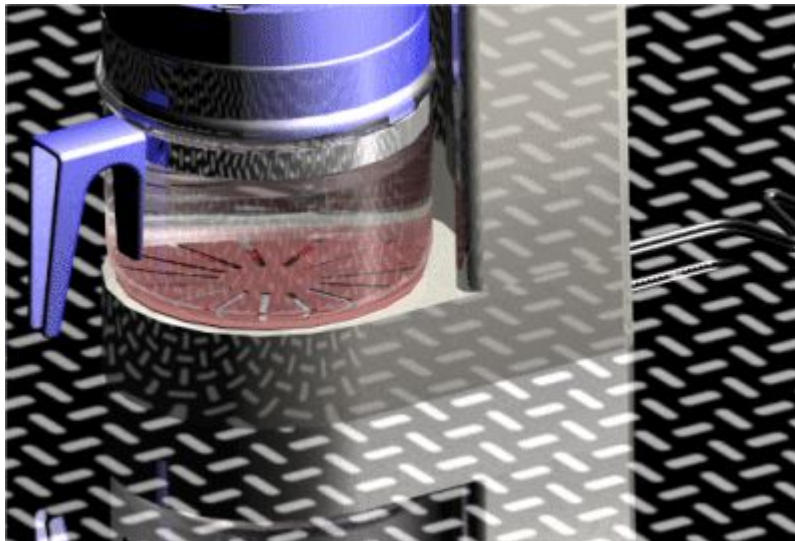


You can use the displacement settings to define a bump map based on bitmap images. The direction of the light source determines the surface texture of the rendering. Lighter areas are rendered as raised portions of the surface and darker areas are rendered as depressions in the surface. A gradient render is done between the light and dark areas.





You can use texture mapping to define how a texture is wrapped onto the surface.



## Light studios

A light studio is the top-level container for all lights used in a model. It provides a fast and simple way to change the entire light scheme. You can change such things as:

- Shadows
- Intensity
- Shadow type

Shadows can be soft or hard. Soft shadows are gradually diffused to create a more realistic lighting effect. Hard shadows can be colored and are useful for such effects as stained glass.

- Shadow resolution

Each light in the light studio has its own set of attributes, such as light type and color. These attributes are defined by a light shader.

You can quickly turn light sources on and off by setting or clearing the check box displayed adjacent to each light in the Light Studio branch of the Session Entity tree on the EdgeBar tool.

Lighting is very important to rendering because it gives a sense of depth to the scene and highlights the shiny surfaces. Lighting supports the following shaders:

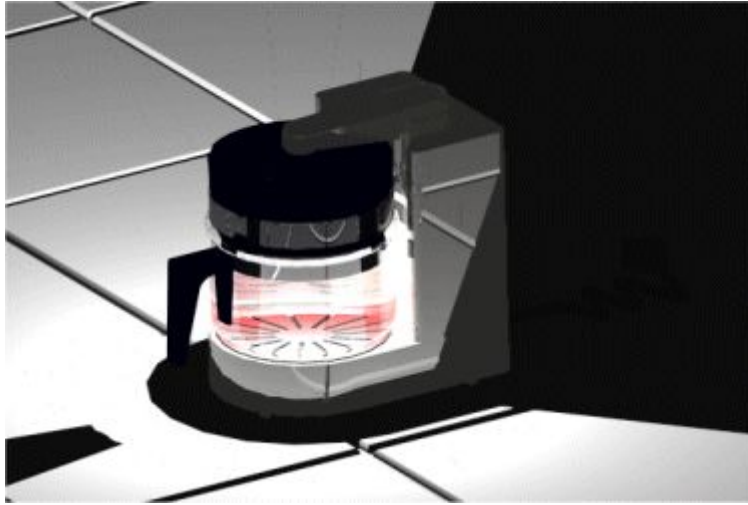
- Ambient light
- Point light
- Spot light
- Distant light
- Sun
- Sky
- More

These shaders contain options that affect the light. Some of the most common options include color, intensity, and location.

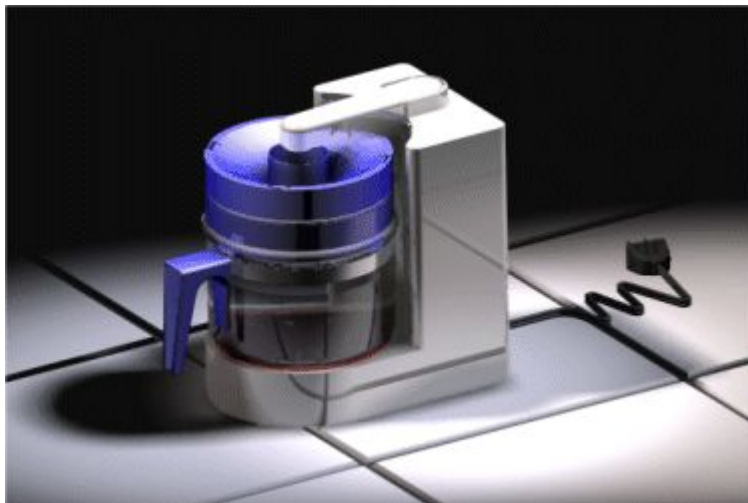
Ambient light illuminates all surfaces regardless of orientation. This is useful for illuminating the scene that is not illuminated by other light sources.



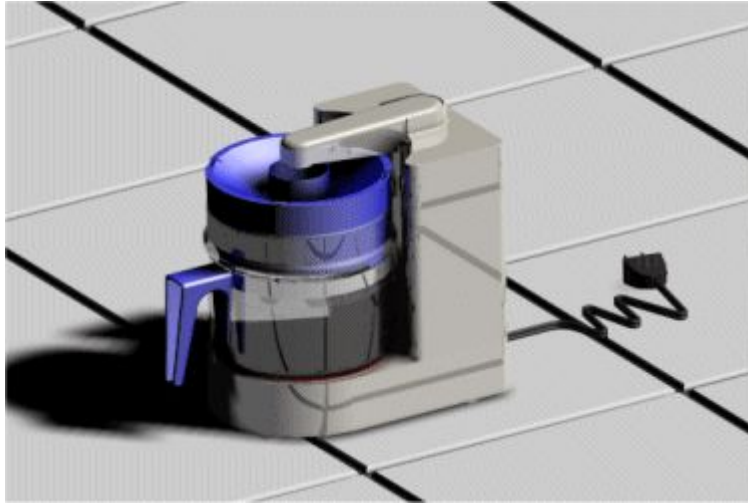
Point light emits light equally in all directions from a point specified by an X, Y, and Z definition. This type of light is useful when lighting enclosed spaces or simulating the effect of a light bulb.



Spot light emits light from a single point and is contained by a cone. The starting and ending positions of a light are specified by X, Y, and Z definitions. This type of light is useful for focusing attention on the model or part of the model.



Distant light emits light parallel to a specific point as if is from a very distant source. Solid Edge uses distant lights, which are used for general lighting.



To learn how to use the lighting controls, see the Help topic, *Edit Light Entity Properties for Advanced Rendering*.

### **Render mode**

Advanced rendering provides rendering settings that allow you to control such things as reflections and the amount of light that bounces between objects. Advanced rendering does not inherit any rendering settings from the Assembly environment. Any changes you make to the rendering settings in advanced rendering do not change in Assembly.

Advanced rendering supports two classes of rendering:

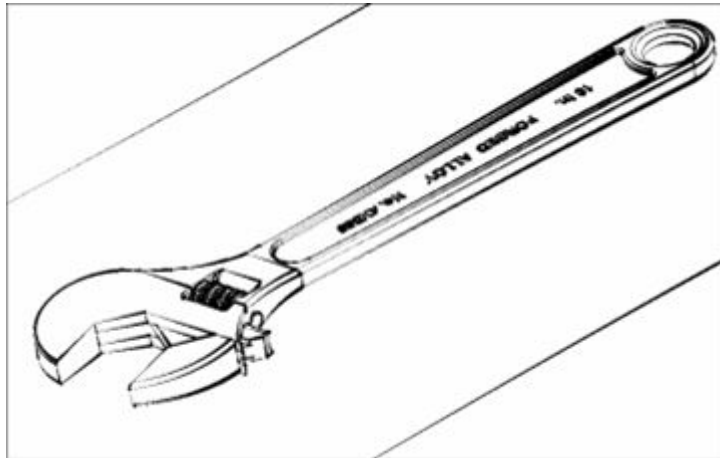
- Photorealistic
- Artistic

Each render mode uses a shader to produce special effects and each shader includes a variety of options. The available options differ depending on the shader you select.

Photorealistic rendering supports ray tracing to show reflections and refractions. Photorealistic rendering supports such shader options as anti-aliasing, transparency, and reflections.

- Ray tracing—plots a path of imaginary rays from the observer’s eye through each pixel on the screen, back to the 3D environment. These rays are tracked as they bounce from one object to another to the light source.
- Anti-aliasing—a method of displaying elements on a low resolution device to make the object appear smoother. You can control the level of anti-aliasing. The more anti-aliasing you apply, the smoother the display will be, but it will also take longer to process.

You can choose from a variety of artistic rendering modes to make a 3D model look as if it was hand-drawn, painted, a mosaic, among others.



## Scenery

Scenery allows you to enhance your renderings by adding such affects as tiled floors or water.



Advanced rendering supports the following types of scenes.

- Circular bases
- Panoramas
- Rooms
- Square bases
- None

## **Schemes**

Schemes allow you to enhance your renderings by adding background images and lighting settings listed below:

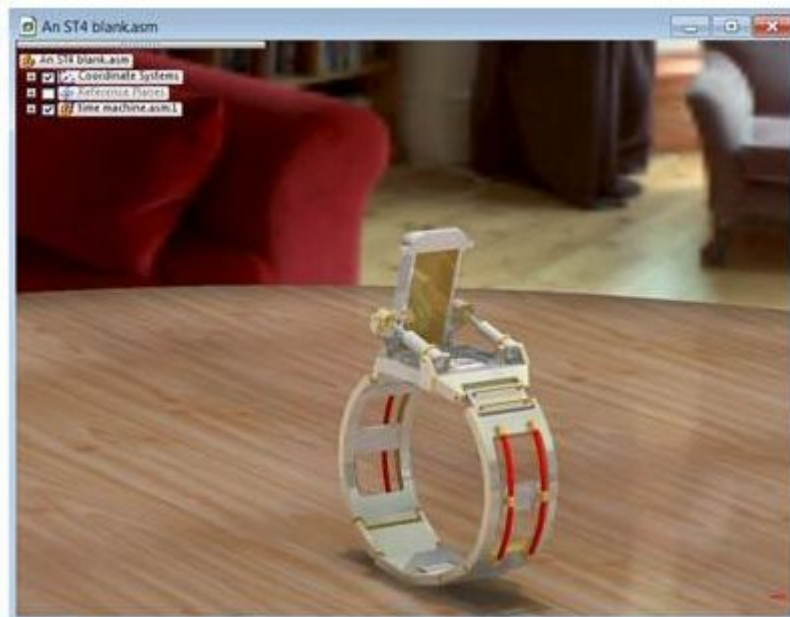
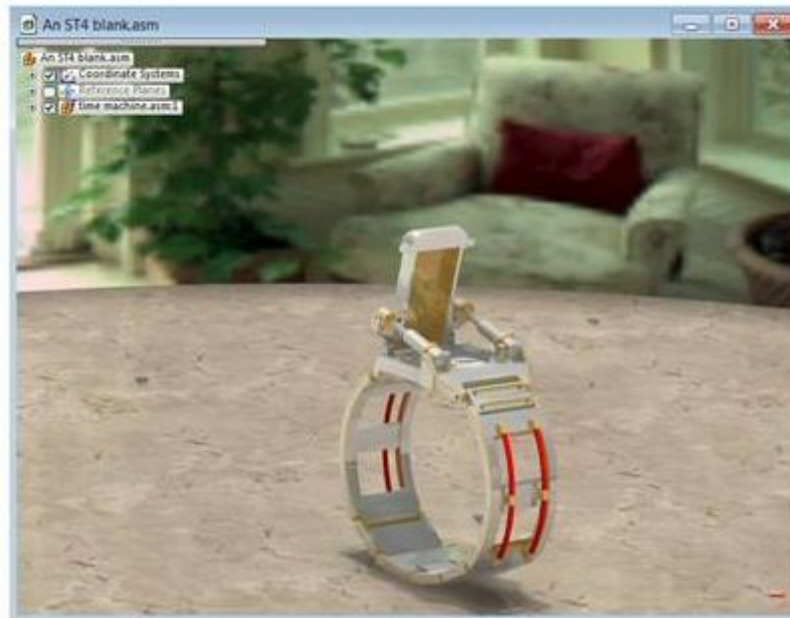
- 2D Schemes (HDR) with 2d backgrounds and lighting.
- 3D Schemes (HDR) with 360 backgrounds and lighting. The 3D schemes rotate relative to the view.

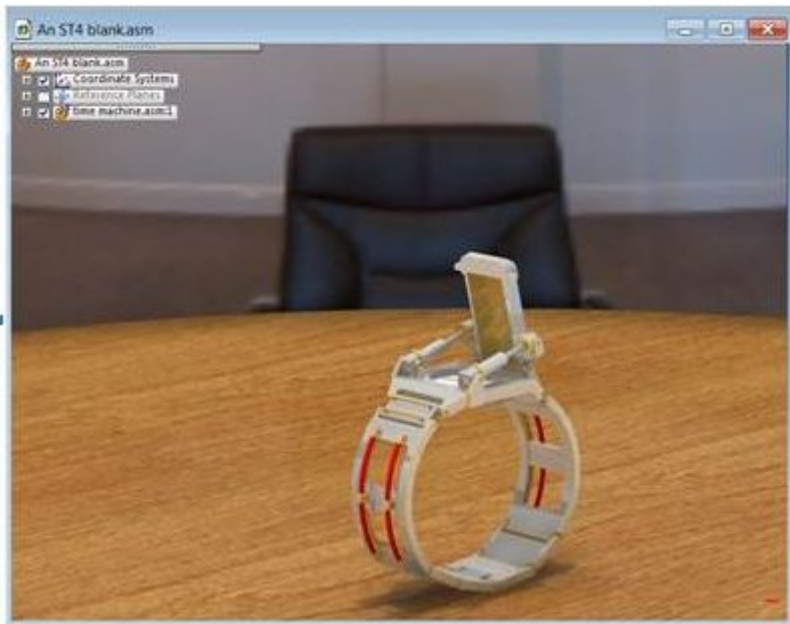
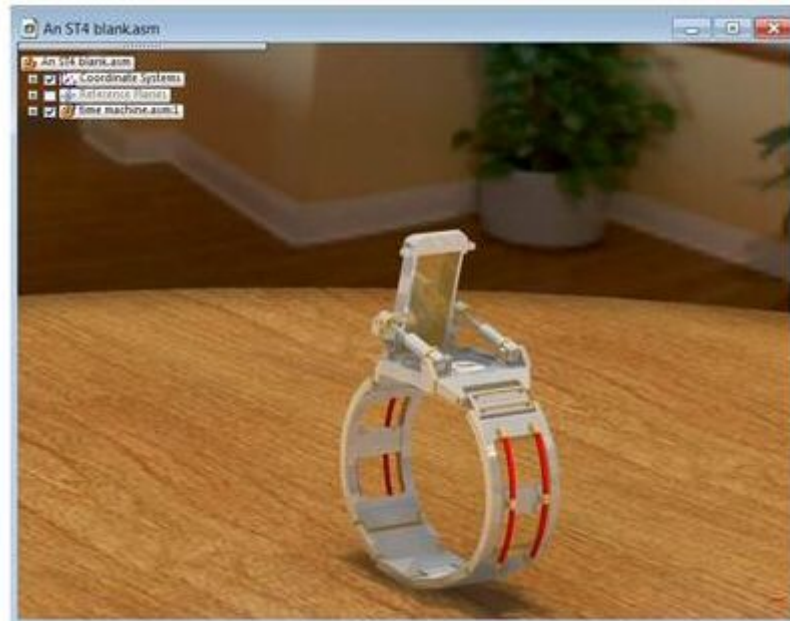
### **Note**

These 3D scenes are not displayed during real time rotations. The scene will need to be rendered again to see the scene.

- Four levels of quality are provided. These are:
  - High Quality HDR
  - Medium Quality HDR
  - Draft HDR
  - Ambient Occlusion (AO)
- Schemes:
  - Interior Schemes
  - Exterior Schemes
  - Studios
  - 2D Abstract Backgrounds and Reflections
  - 2D Exterior Back plates
  - 2D Interior Back plates

Examples:





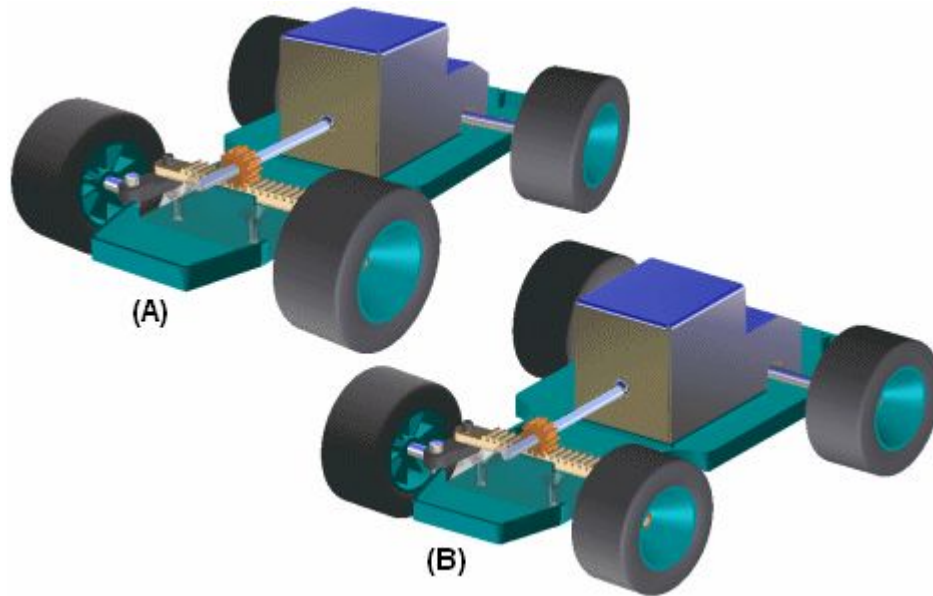
**Note**

Digital background images can be substituted by editing the definition of a scheme in the session entities tab of pathfinder.



 **Perspective command**

Applies perspective to (A) or removes perspective from (B) the view for the active window.



You can quickly add or remove a perspective override using this command, but cannot change the perspective angle with this command.



### **Render Area command**

Renders a fenced area.

Depending on the type of Solid Edge license you have, you may have advanced rendering capabilities available to you in the Explode-Render-Animate application in the Assembly environment. For example, if you have a Solid Edge Classic license, advanced rendering is available. If you have a Foundation license, advanced rendering is not available.



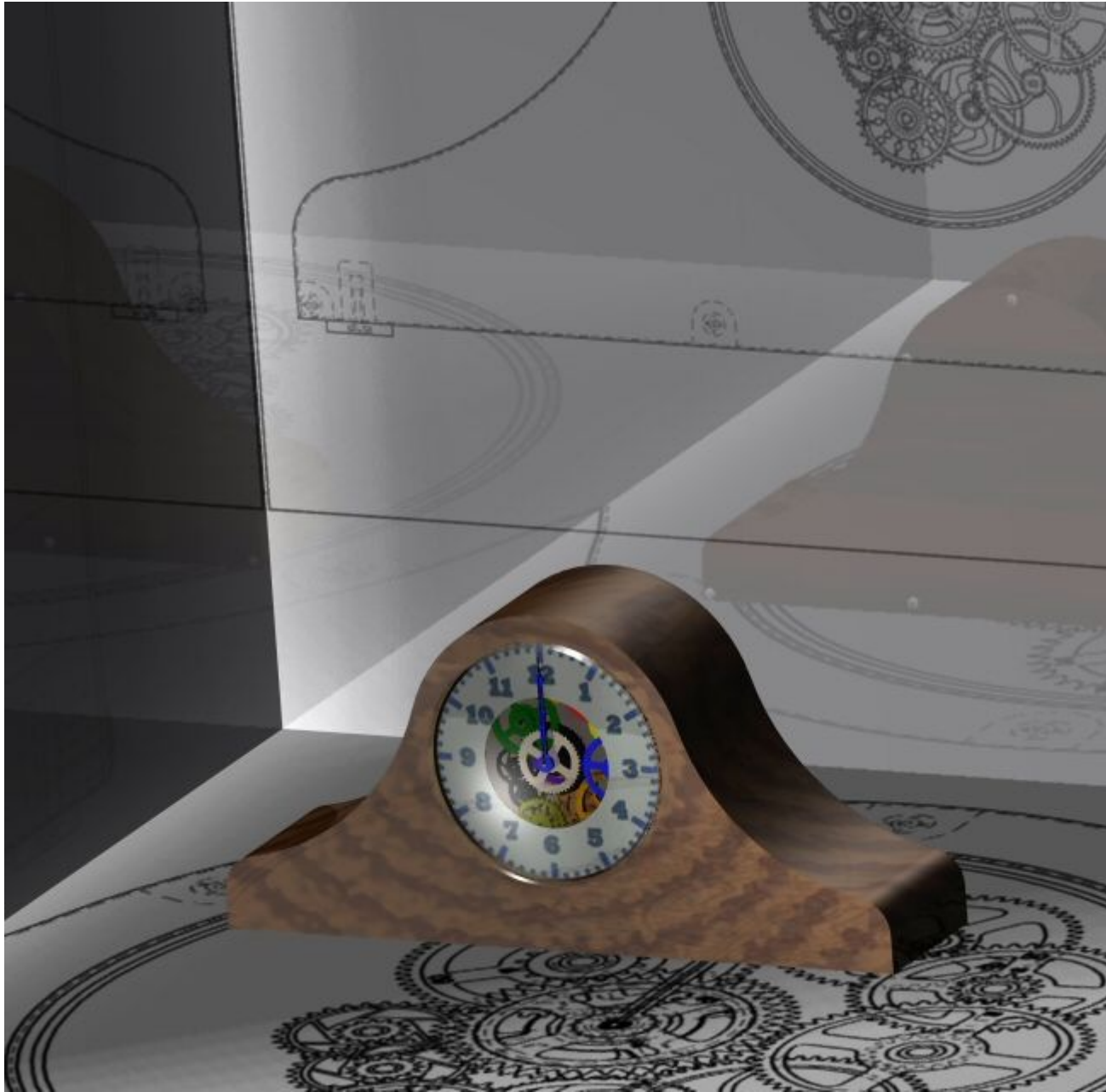
## **Render Scene command**

Renders the active window.

Depending on the type of Solid Edge license you have, you may have advanced rendering capabilities available to you in the Explode-Render-Animate application in the Assembly environment. For example, if you have a Solid Edge Classic license, advanced rendering is available. If you have a Foundation licence, advanced rendering is not available.

## Activity: Rendering

### Activity objectives



You will be working in the context of an assembly named *render.asm*. In this activity you will:

- Set parameters for changing backgrounds, foregrounds, light sources, and perspectives to control the display of a Solid Edge assembly.
- Enter the assembly application Explode-Render-Animate and assign materials and rendering properties to a Solid Edge assembly.

- Edit parameters assigned from the predefined archive and better control the image display of rendered scene.
- Generate presentation quality images of a Solid Edge assembly using the rendering tools in the Explode-Render-Animate application.

Turn to appendix A for the Activity: Rendering an assembly.

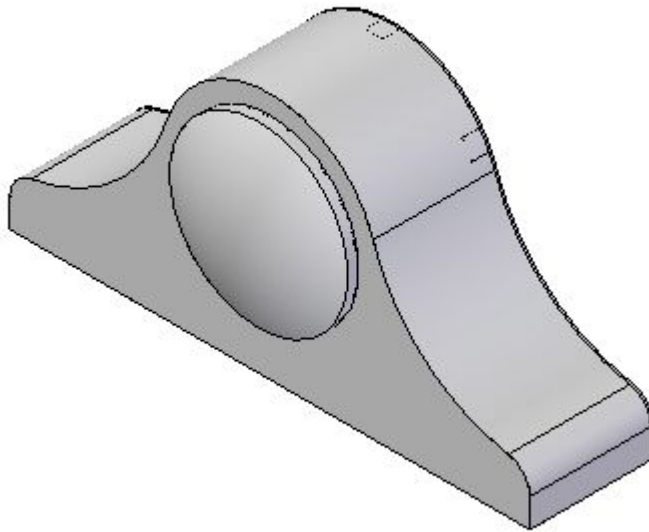
**Activity: Rendering an assembly**

**Step 1**

In the following steps, you will be working in the context of an assembly document *render.asm*. Before entering the Explode-Render-Animate application, you will assign properties to some parts and change some viewing parameters within the Solid Edge Assembly environment. These settings will carry over into the Explode-Render-Animate application and will be available there unless they are overridden by another parameter in that application.

You will make the glass face clear by setting the face style.

- Open the assembly *render.asm* and activate all the parts in the assembly.



- ▶ In PathFinder, right-click *housing.asm*, and then click Edit.
- ▶ In PathFinder, right-click *glass.par*, and then click Edit.
- ▶ On the ribbon, choose View tab® Style group® Part Painter.






- ▶ On the Part Painter command bar, set the Style to White (glass) and the Select method to Any.



- ▶ In the graphics window, select the revolved protrusion to set the face style.
- ▶ On the command bar, click the Close button.
- ▶ On the ribbon, click the Close and Return button to return to *housing.asm*.
- ▶ On the ribbon, click the Close and Return button to return to *render.asm*.

## Step 2


The numbers for the clock face have been created and stored in a TIFF format image. You will now assign this image to the clock face as a texture. To do this, you will create a new face style based on the properties of an existing face style. You will then modify the face style by assigning the TIFF image as a texture and then orienting it appropriately.

- ▶ In PathFinder, right-click *SE\_face.par*, and then click Edit.
- ▶ On the ribbon, choose View tab® Show group® Hide Previous Level  to turn off the display of the assembly (if it is visible).
- ▶ Choose View tab® Style group® View Overrides .
- ▶ Set the Render mode to Smooth with VHL Overlay.
- ▶ Turn on Textures, then click OK.
- ▶ Choose View tab® Style group® Styles .

- ▶ Set the Style type to Faces Styles.
- ▶ From the Styles list, select White (glass).
- ▶ Click New.
- ▶ In the Name field, type Clockface, but do not press Enter.

**Note**

When you press Enter, the new style Clockface is created and you return to the Faces Style menu. To make further changes to the style Clockface, you need to select Clockface, and then click Modify.

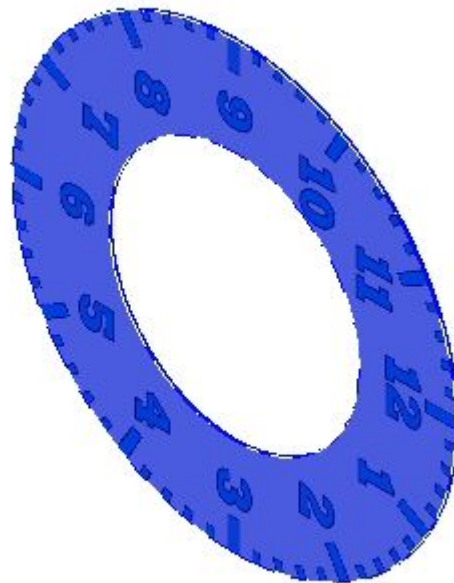
- ▶ Click the Texture tab.
- ▶ Browse for the texture file *clockface.tif*, and click Open.
- ▶ Click OK to close the New Faces Style dialog box, and then click Close to dismiss the Style dialog box.
- ▶ On the ribbon, choose View tab® Style group® Part Painter .



- ▶ Set the Style to Clockface and the Select method to Face.




- ▶ Select the front face of the clock, and then click the Close button on the command bar.

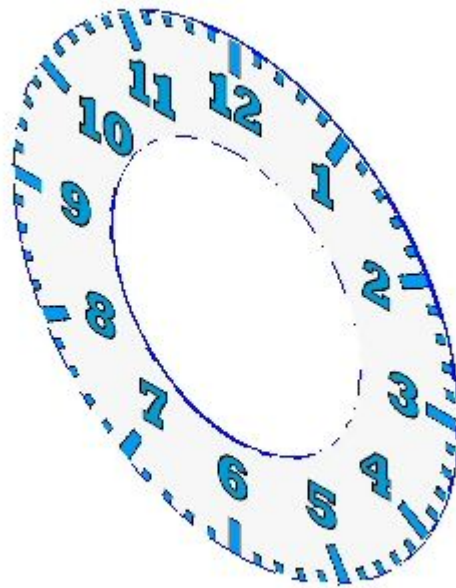


**Note**

You will edit the style Clockface to correct the problems with the texture.


- ▶ Choose View tab@ Style group@ Styles .
- ▶ Select Faces Style.
- ▶ Select Clockface, and then click Modify.
- ▶ Click the Texture tab, and modify the Rotation angle to 270, such that the orientation of the clock face is correct.

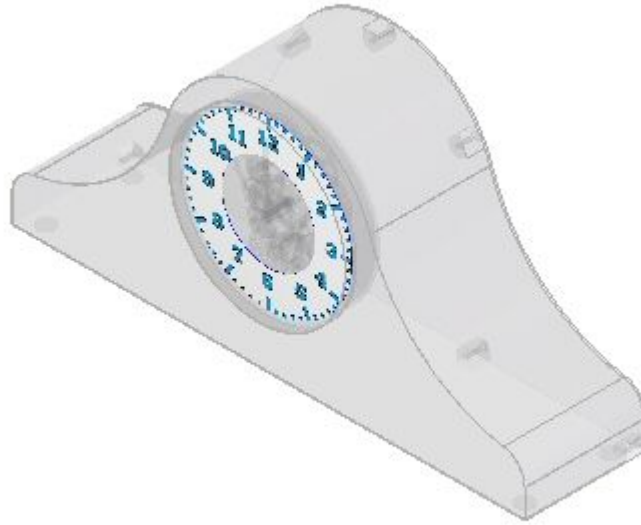
- Click the Appearance tab. Set the values as shown. Click OK, and then click Apply.
  - Shininess: 0.25
  - Reflectivity: 0.20
  - Opacity: 1.0



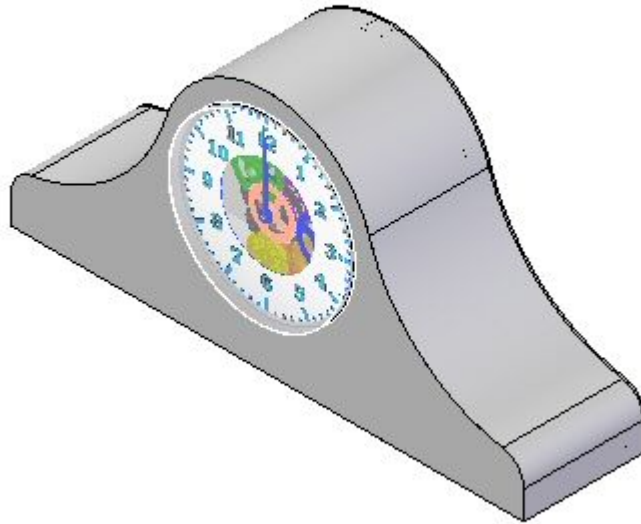
**Note**

Create the face style in the template and it will be available for future geometry created with that template.

- ▶ Choose View tab@ Show group@ Hide Previous Level  to display the assembly again.






- ▶ On the ribbon, click Close and Return to return to *render.asm*. Save the assembly.



*Step 3*

You will now enter the Explode-Render-Animate application and apply settings to create your desired rendered images.

- ▶ Choose Tools tab® Environs group® ERA .
- ▶ Choose Home tab® Render group® Render Setup .
- ▶ In the Render Setup dialog box, select all of the Automatic Render Options and Use Progressive Rendering, and then click OK.
- ▶ In PathFinder, click the Session Entities tab .

**Note**

The Session Entities tab in PathFinder contains rendering settings and parameters that have been defined up to this point in time. These parameters can be edited and the values modified. You will be adding new values from the Predefined Archives. Values in Predefined Archives cannot be changed, but once used, the entry in the Session Entities tab can be renamed and modified.

- ▶ In PathFinder, click the Predefined Archives tab .

**Note**


To use a setting in the Predefined Archives, you will drag the setting into the rendered scene. To set geometry-specific settings, such as material textures, you will drag the material onto the desired part. For view-specific settings, such as backgrounds and lighting, you will drag the setting anywhere into the rendered scene.

- ▶ Drag Backgrounds® Graduated® Green into the rendered scene.

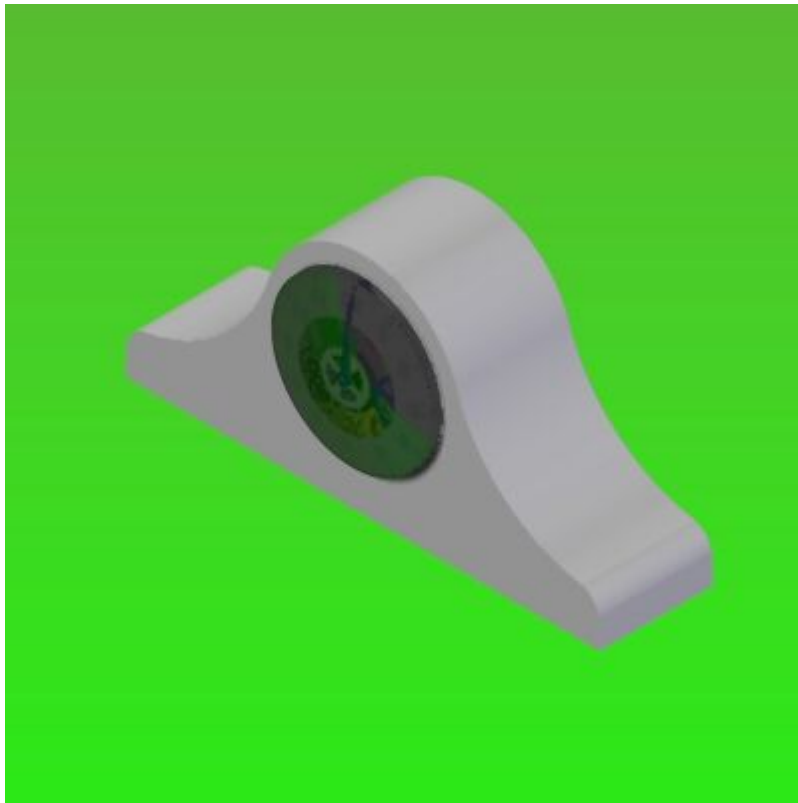
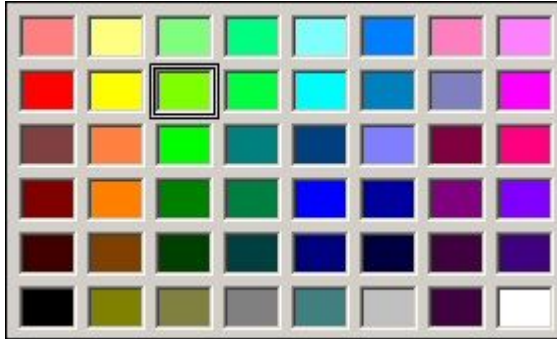
**Note**

Because you set the rendering options to automatically render upon a background change, the scene is automatically rendered. When this parameter is turned off, you need to manually render the scene by clicking the Scene command.



- ▶ To modify the parameter you just set, in PathFinder, click the Session Entities tab .

- ▶ In PathFinder, right-click Background, and then click Edit Definition.
- ▶ In the Background Editor, set the Bottom Color to be the color shown, and click OK. Click OK to exit the Background Editor.

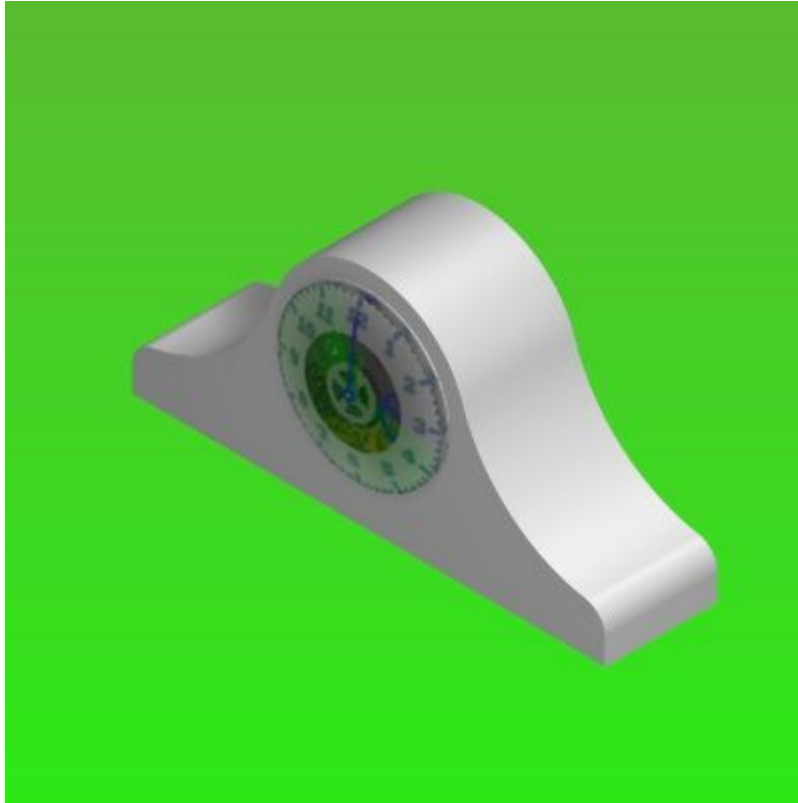


- ▶ In PathFinder, click the Predefined Archives tab.



- ▶ Drag Render Modes® Photorealistic® Photorealistic, high quality into the rendered scene.

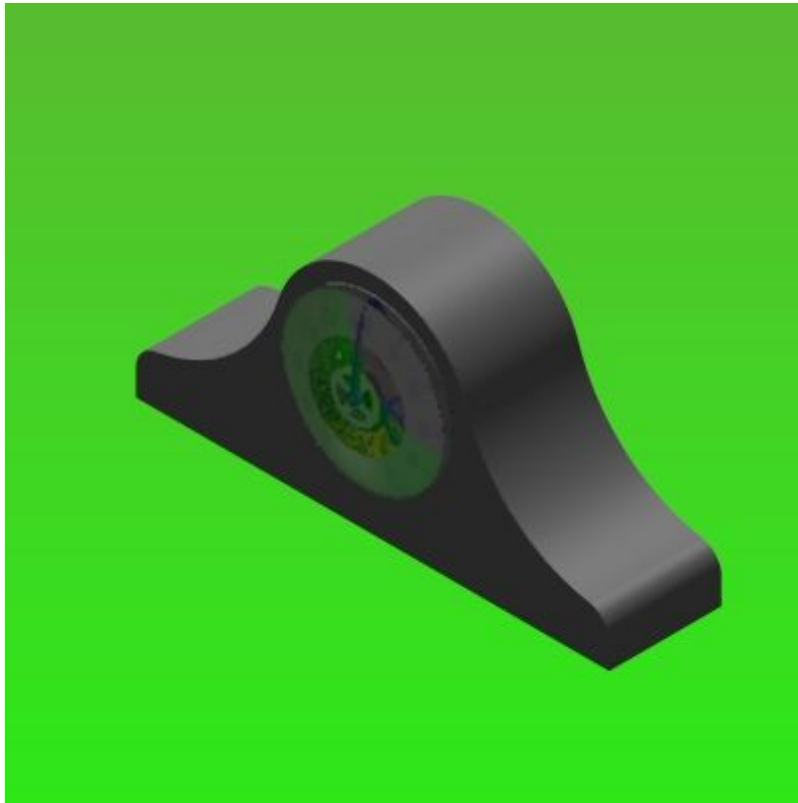
- Drag Light Studios® High-Contrast® Left spot with back into the rendered scene.



- ▶ Now modify the parameter you just set. In PathFinder, click the Session Entities tab.

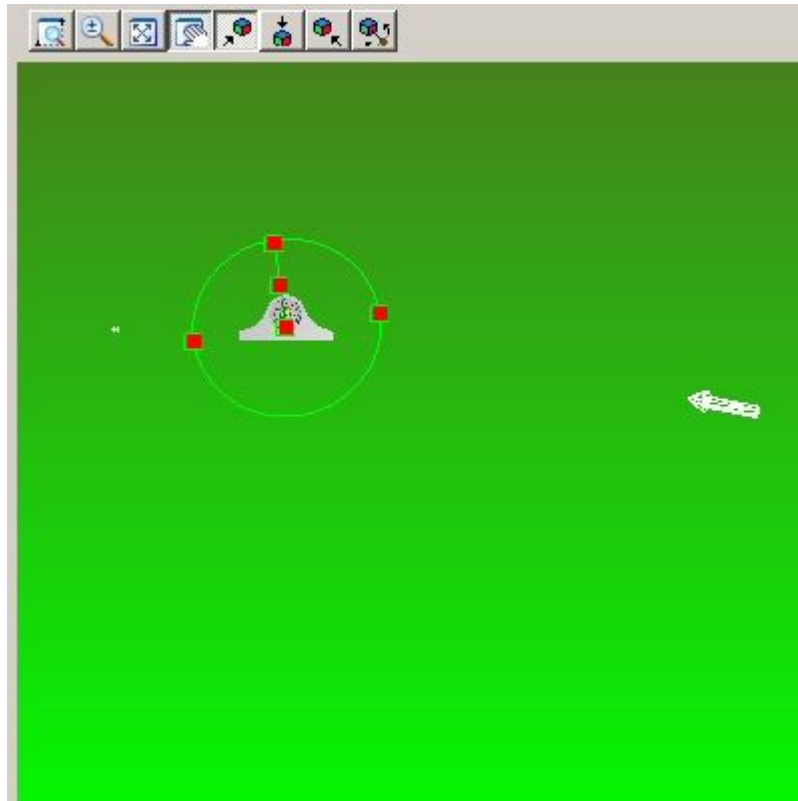


- ▶ To see the effect of the left spot on the rendered scene, turn off the Left Spot, and then click the Home tab® Render group® Scene command.

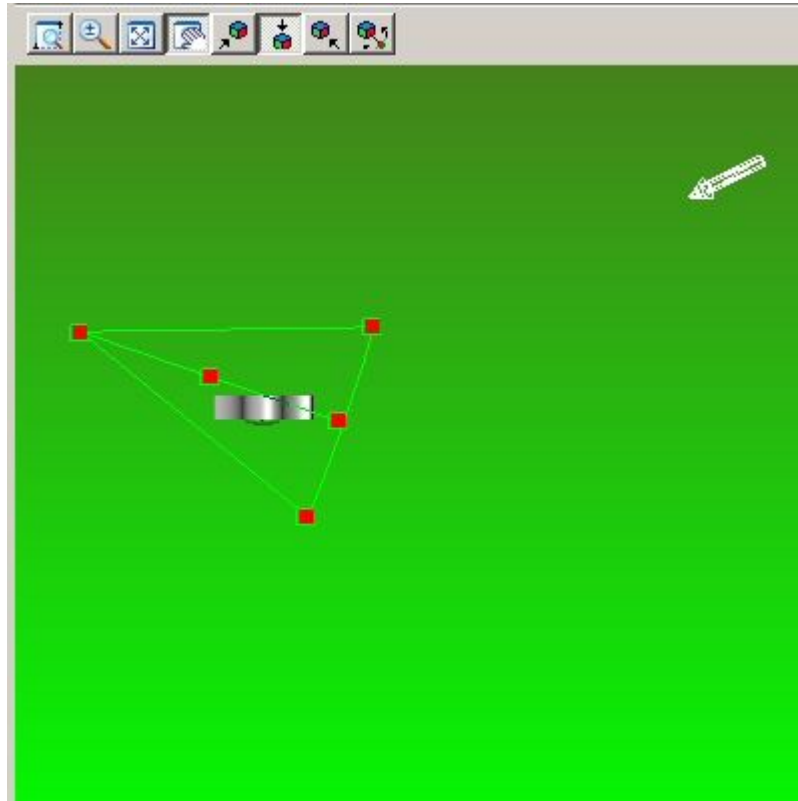




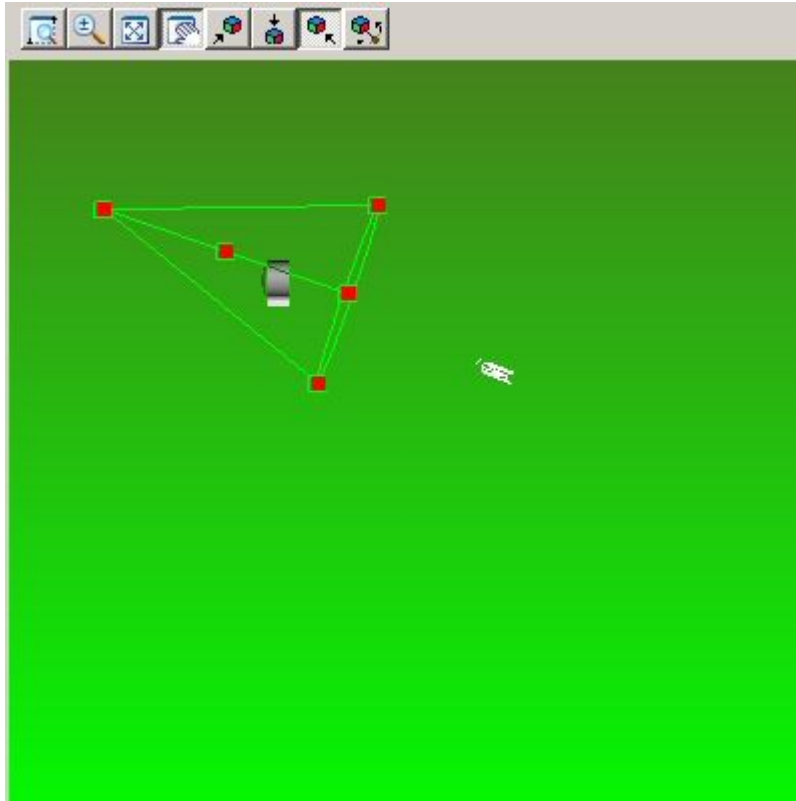
- ▶ Turn on the Left Spot again, and then click the Scene command.
- ▶ Modify the Left Spot definition. Right-click Left Spot, and then click Edit Definition.
- ▶ In the Light Editor dialog box, set the cone angle to 40°.
- ▶ Click the Placement tab. Click the Front View button, and drag the handles so that the front looks approximately like the view below.



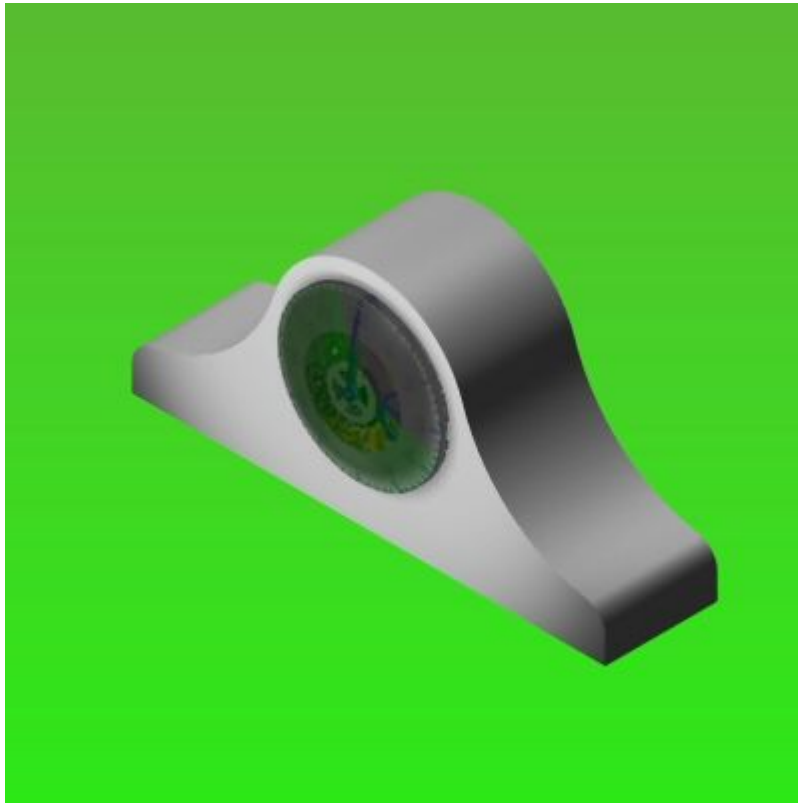
- ▶ Now select the Top View button and drag the handles of the cone to the approximate position shown.



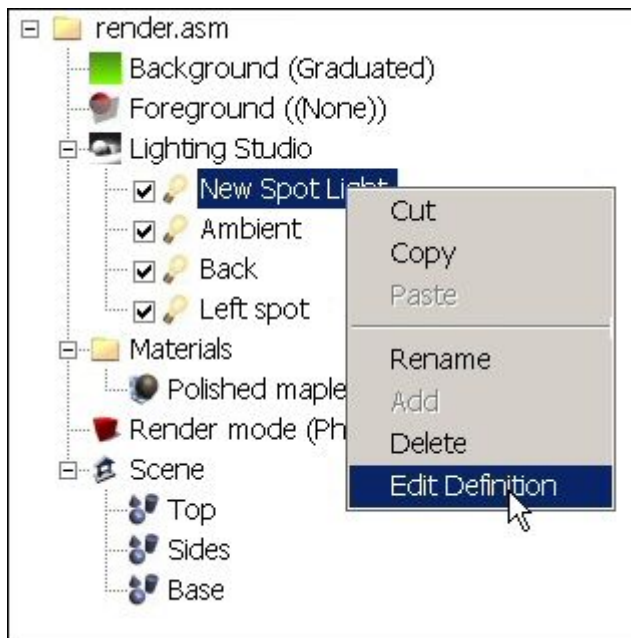
- ▶ Select the Right View button and drag the handles of the cone to the approximate position shown. Check the top, front and right views. Changes made in one view could move the viewing cone in another. It may take you several iterations to achieve the desired results.



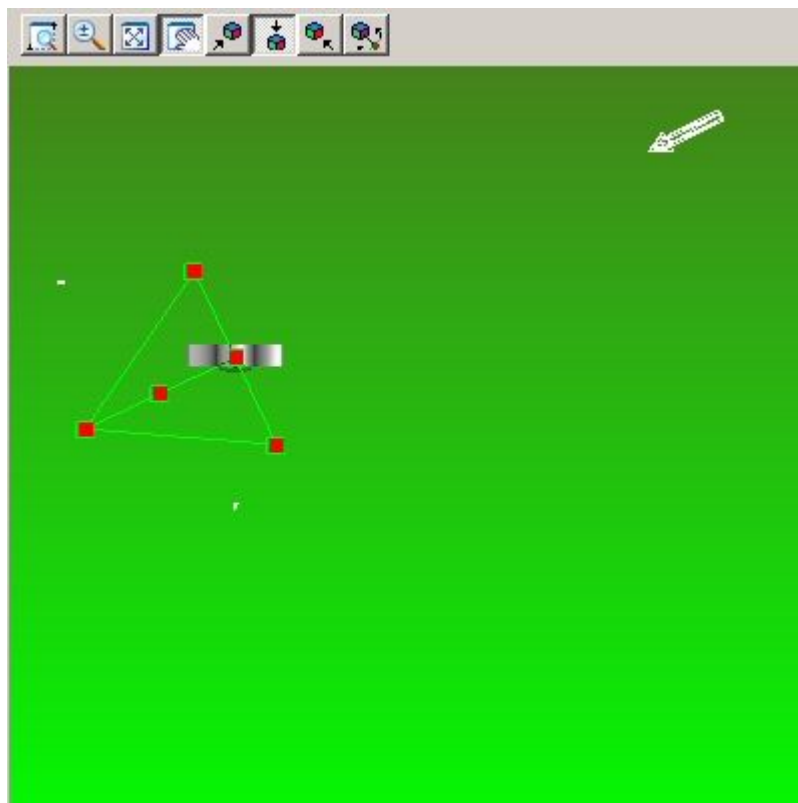
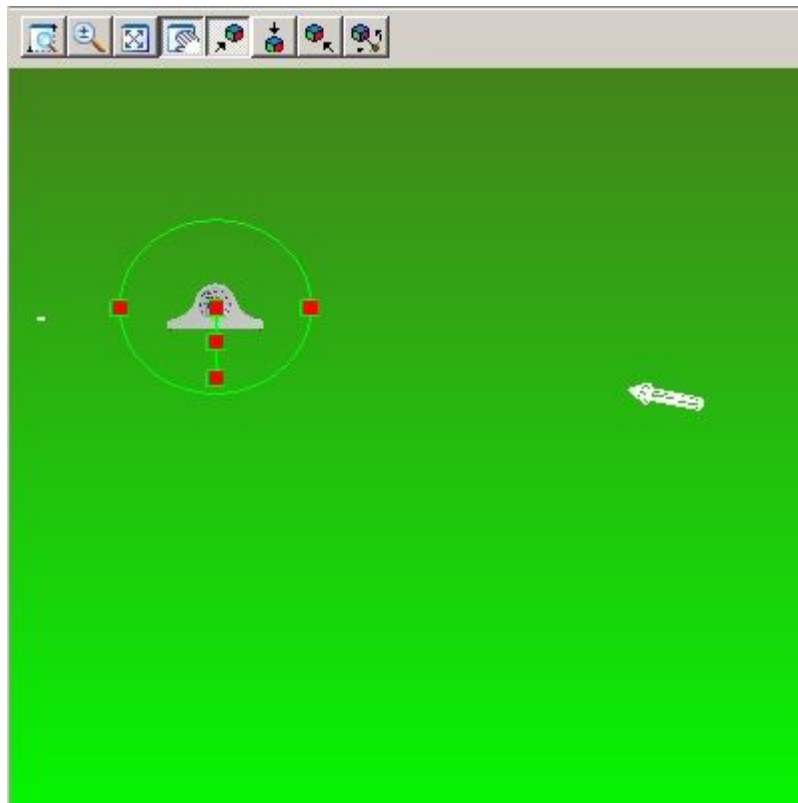
- ▶ Click OK to exit the Light Editor dialog box, and return to the rendered scene.

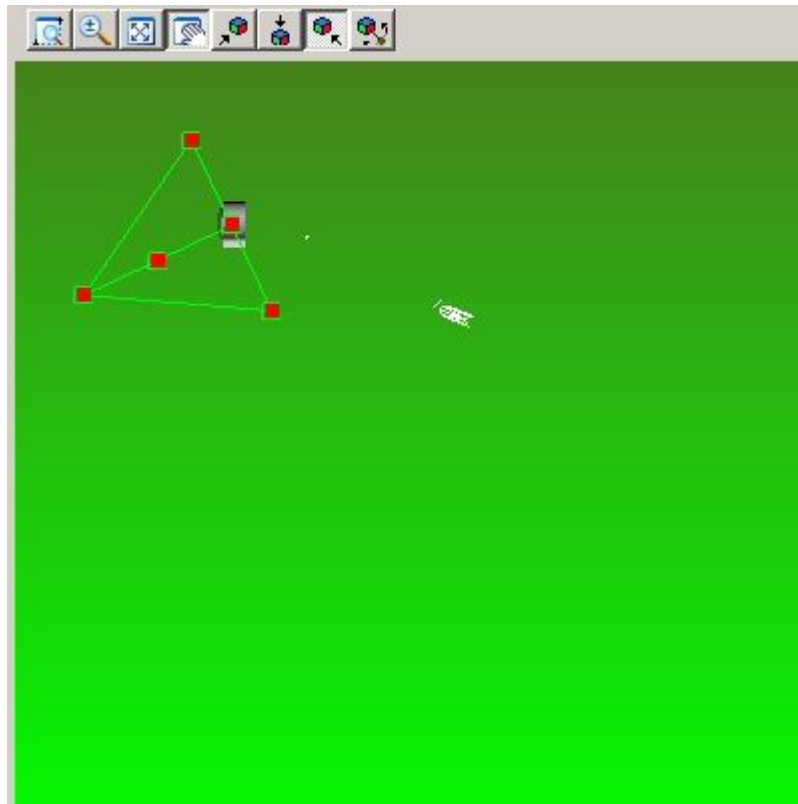


- ▶ Add a new light source. In PathFinder, right-click Lighting Studio and click Add.
- ▶ Right-click New Spot Light, and then click Edit Definition.

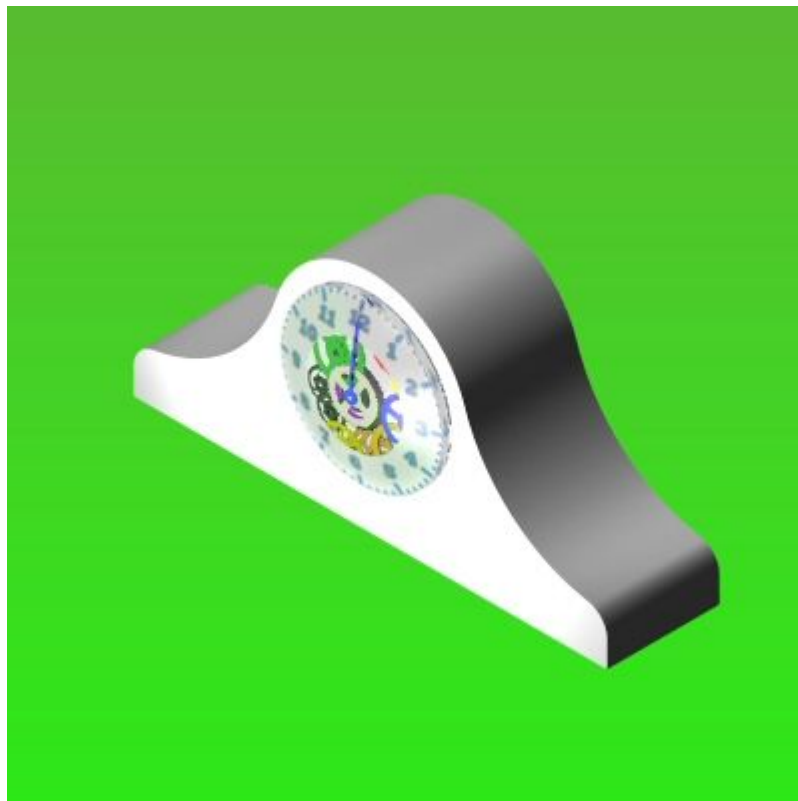


- ▶ On the Placement tab, set the viewing cone as shown for the top, front and right buttons.





- ▶ Click OK to exit the Light Editor dialog box and return to the rendered scene.



*Step 4*

You will now create presentation quality images.

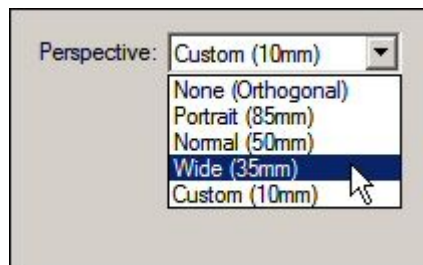
- ▶ On the view tab, in the style group, click the perspective command.



- ▶ On the view tab, in the style group, click the view overrides command.



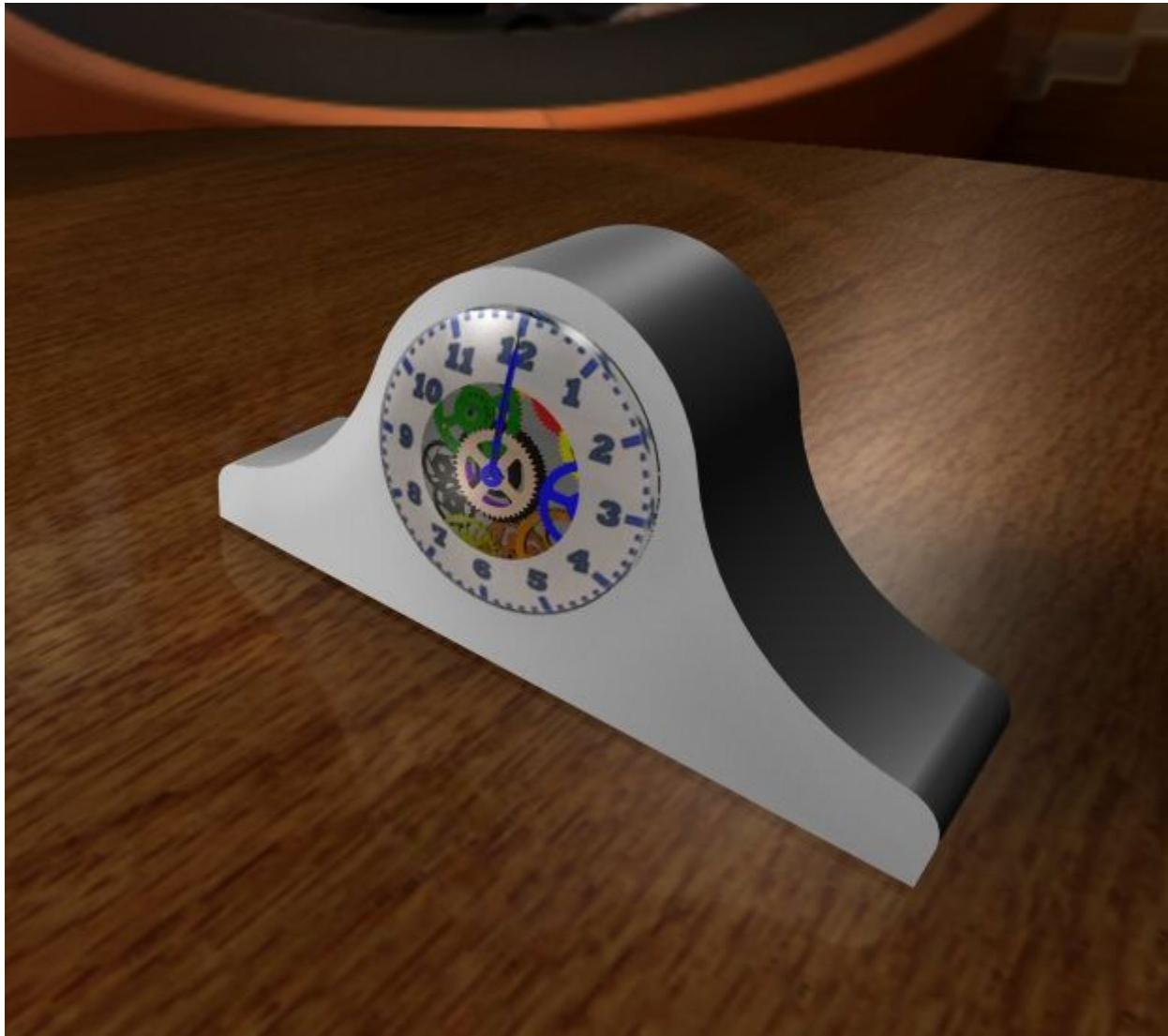
- ▶ On the view overrides dialog box, set the perspective to Wide (35mm).



- ▶ In PathFinder, click the Predefined Archives tab.



- ▶ Drag Scenery® 3D backgrounds® Interior — Lobby into the rendered scene.



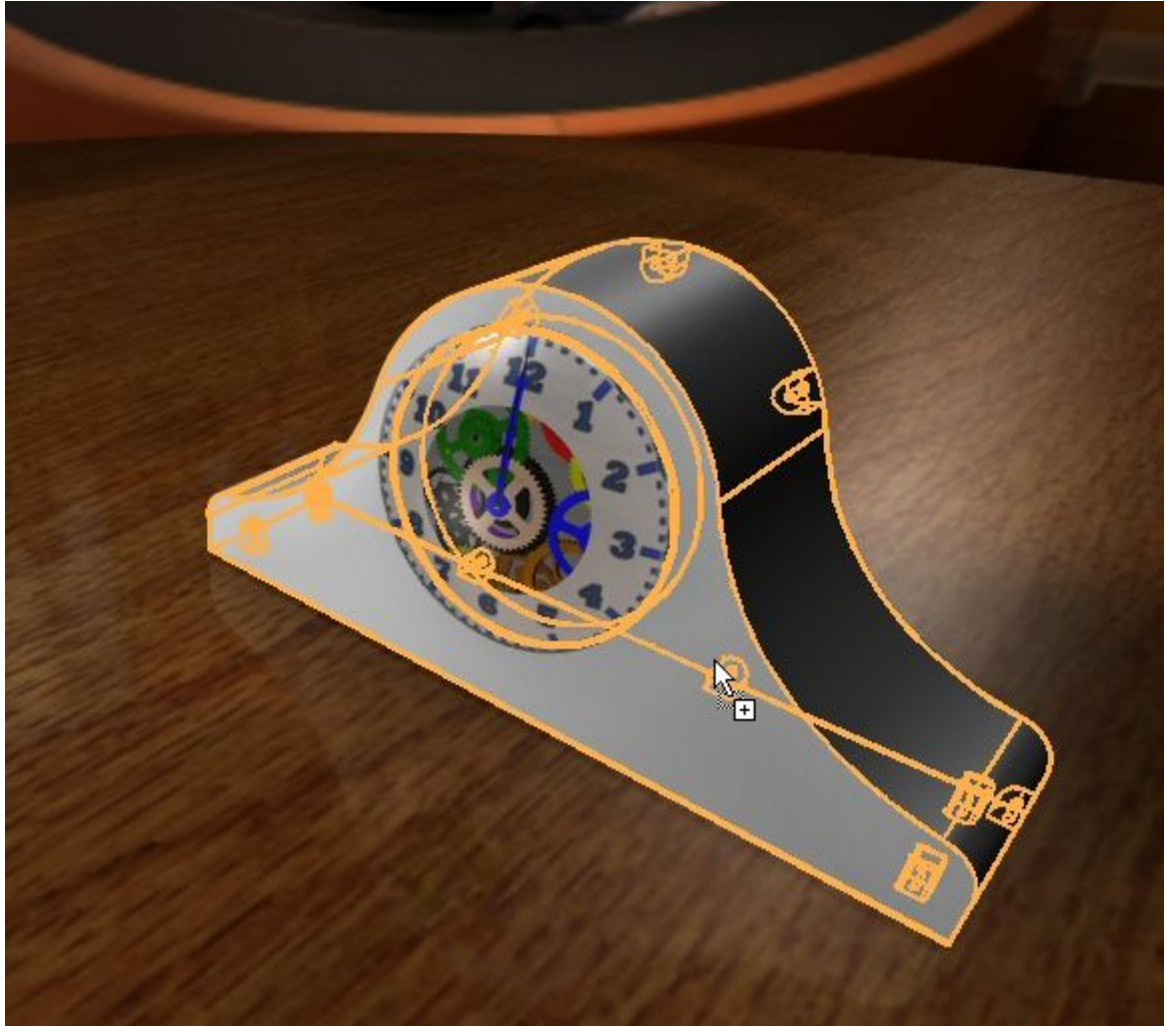
**Note**

The scene is now in the session entities. By editing the session entity, different digital images can be substituted for the current images.

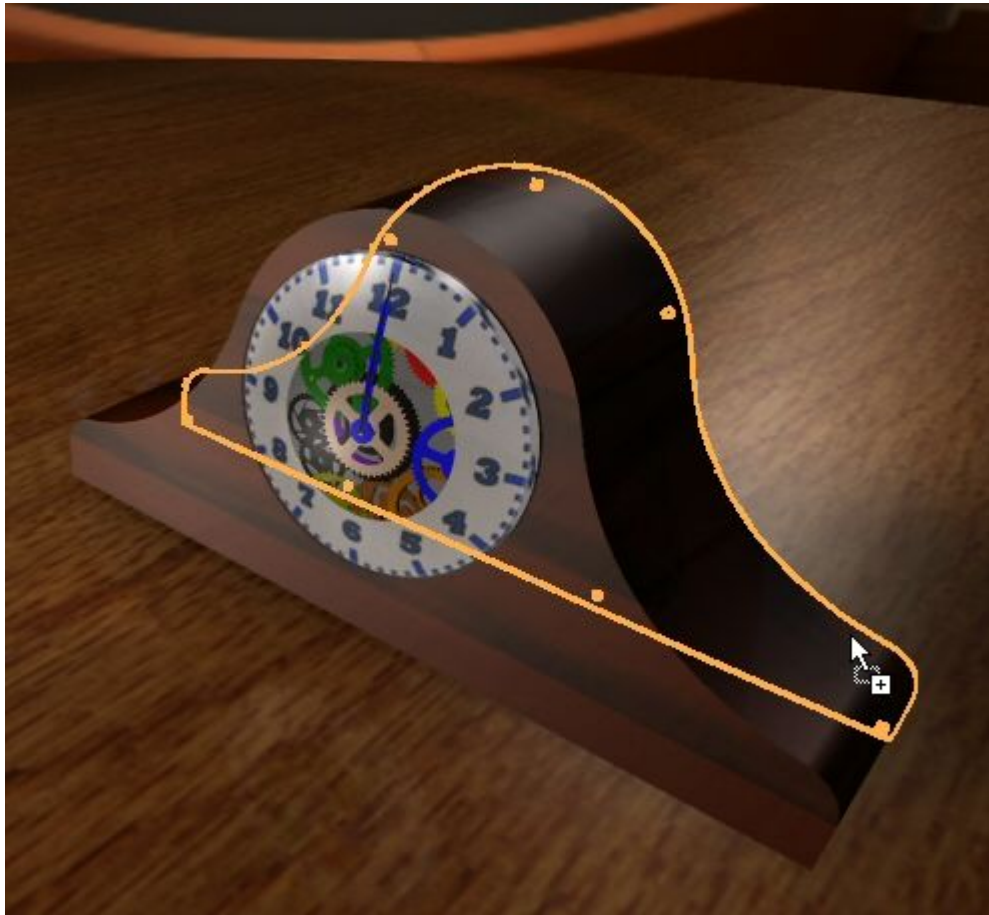
- ▶ In PathFinder, click the Predefined Archives tab .



- Drag Materials® Wood® Mahogany® Polished mahogany onto the part *case.par*.



- ▶ Drag Polished mahogany onto the part *backplate.par*.

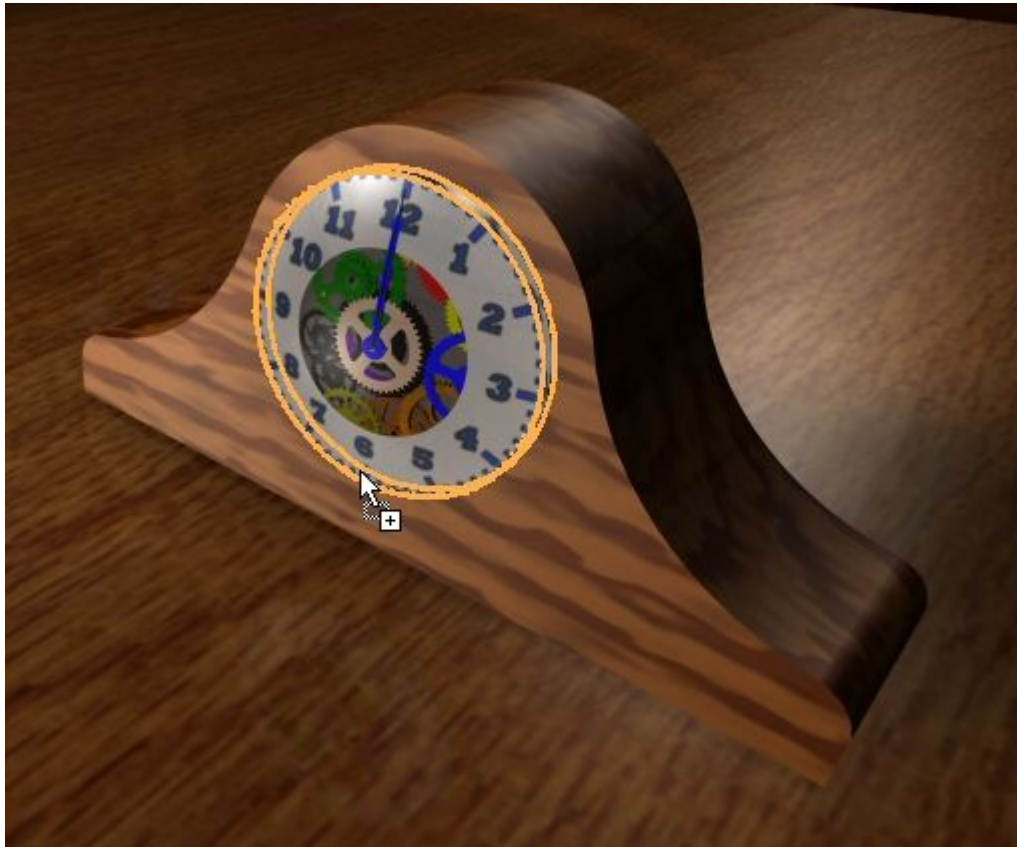


- ▶ Now assign a different wood to the same two parts. Drag Materials® Wood® Japanese Oak® Polished Japanese oak onto the part *case.par*.

- Drag Polished Japanese oak onto the part *backplate.par*.



- ▶ Drag Materials® Glass® Clear (glass) onto the part *glass.par*.



- ▶ Now modify the parameters you just set. In PathFinder, click the Session Entities tab.



- ▶ Modify the properties on Polished Japanese oak and reapply it to the parts. Right-click the material Polished Japanese oak and click Edit Definition.
- ▶ Change the trunk direction to 0, 1, 0.
- ▶ Now modify the glass parameters to allow more light to pass through. On the Sessions Entities tab, right-click the material Clear (Glass), and then click Edit Definition.

- ▶ In the Material Editor dialog box, click the Reflectance tab and set the transmission factor to 1.30. Click OK to dismiss the dialog box.



### Step 5

You will now replace the tread plate scene with textures created in the Draft Environment. The textures are JPEG format images that have been assigned to faces of parts in the same manner as the numbers you previously placed on the clock. Since you know how to do this, there is no need for you to do those steps. To see the textures, you will turn off the tread plate and show the textured parts and then render and save the final image.

- ▶ On the view tab, in the style group, click the perspective command. This turns the perspective display off.

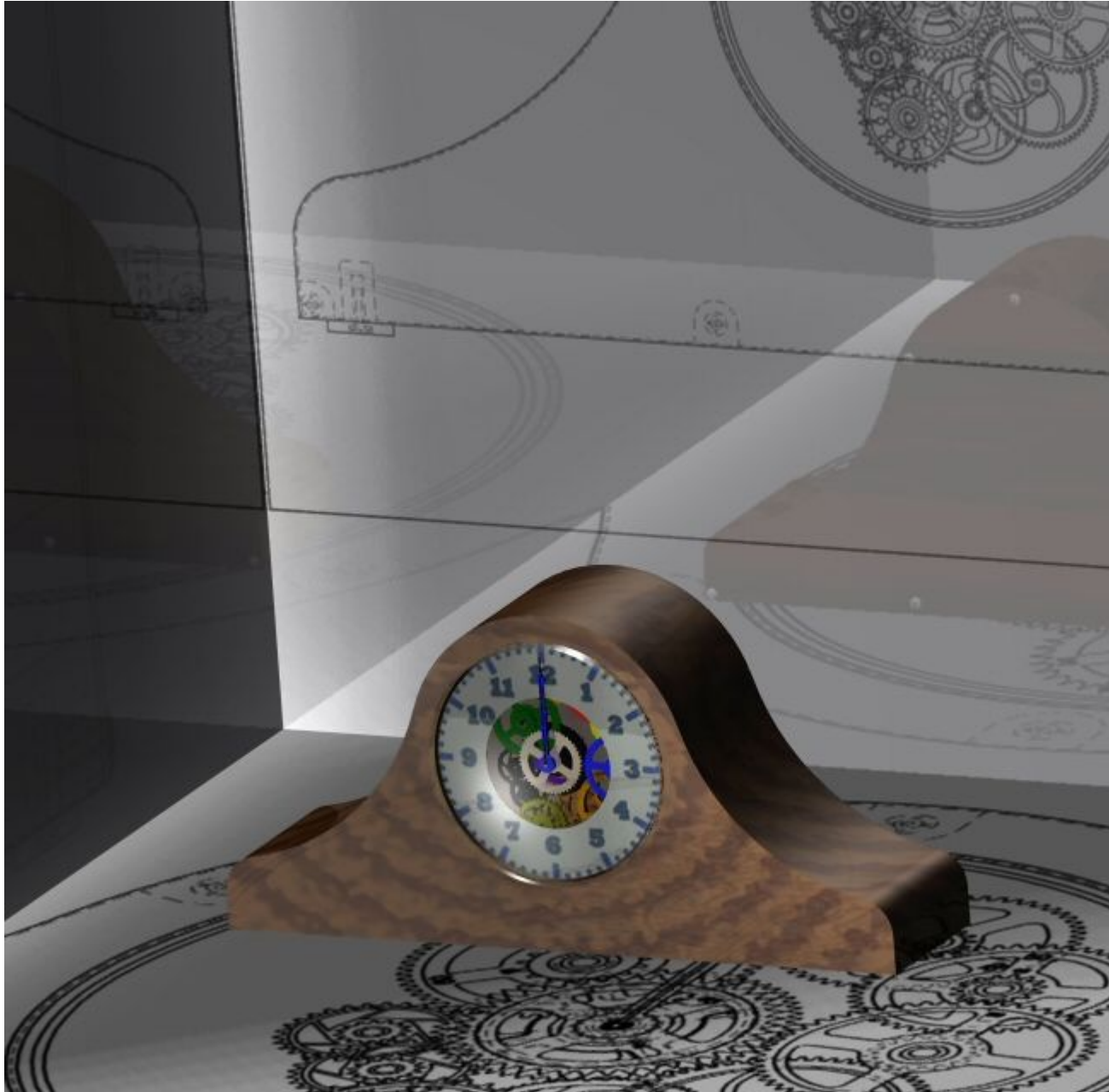


- ▶ In PathFinder, click the Predefined Archives tab .

- ▶ Drag Scenery® No Scenery into the rendered scene.

- ▶ The parts *wall1.par*, *wall2.par* and *wall3.par* are hidden. Find them in Assembly PathFinder and show them.

- ▶ Render the scene.



- ▶ Now save the rendered view as a presentation quality image. Click the Application button and then choose Save As® Save As Image. Set the file type to JPEG and then click the Options button. Set the alternate view style to Presentation View Style. Set the resolution to 300 DPI. Set the units to Pixels. Click OK and then save the file as *Japanese Oak Clock.jpg* in the folder containing the documents for this activity.

**Note**

From the Predefined Archives, you have taken rendering parameters and materials and applied them to a rendered scene. There are many combinations and modifications that can be made to enhance the rendered scene that were not covered. The workflow for using the predefined parameters and then modifying them is the same as covered in this activity. You may want to continue experimenting with the Predefined Archives by using other backgrounds, environments, foregrounds, light studios, materials, render modes and scenery not covered in this activity.

- ▶ Click the Application button, and then click Save.



- ▶ Click Close ERA to exit the Explode-Render-Animate application.



- ▶ Save and close the assembly. This completes this activity.

#### *Activity summary*

In this activity you learned how to generate presentation quality images of a Solid Edge assembly. In the activity, the following topics were covered:

- Creating and editing backgrounds for rendered scenes.
- Creating and editing light sources for rendered scenes.
- Assigning predefined material textures to parts within an assembly.
- Editing the material properties to give the desired result in the rendered scene.
- Placing and then editing predefined scenery to enhance the rendered scene.
- Saving presentation quality JPEG images once the desired rendering options have been finalized.

### **Lesson review**

Answer the following questions:

1. What makes the session entities tab appear in pathfinder?
2. How do you apply a texture to a face?
3. What is the difference between the session entities tab and the predefined archives tab?
4. How do you create a high quality rendered image?



## **Answers**

1. What makes the session entities tab appear in pathfinder?

In the ERA environment, the session entities tab appears after a scene is rendered.

2. How do you apply a texture to a face?

Within the part or sheet metal file where the face exists, apply a face style that contains a texture image. Textures must be turned on in the view format dialog box.

3. What is the difference between the session entities tab and the predefined archives tab?

Predefined archives contain settings for backgrounds, light sources, scenery and so forth. They cannot be changed, however when a setting is used, it populates the session entities tab in pathfinder. The settings can be edited as a session entity.

4. How do you create a high quality rendered image?

Settings that contribute to high quality rendered images are:

- In the format view dialog box, set the anti-alias level to high.
- In the style group, set the sharpness to the highest setting.
- From the predefined archives, drag Render Mode>Photorealistic>High quality into the scene
- When saving the image, click options and set the alternate view style to be presentation view style.

### **Note**

These parameters affect system performance.

## **Lesson summary**

In this lesson you learned how to generate presentation quality images of a Solid Edge assembly. In the activity, the following topics were covered:

- Creating and editing backgrounds for rendered scenes.
- Creating and editing light sources for rendered scenes.
- Assigning predefined material textures to parts within an assembly.
- Editing the material properties to give the desired result in the rendered scene.
- Placing and then editing predefined scenery to enhance the rendered scene.
- Saving presentation quality JPEG images once the desired rendering options have been finalized.

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Lesson

# *16 Animating assemblies*

## Course Overview

### Course overview

The Explode-Render-Animate application within the Solid Edge assembly environment is a tool for creating different types of presentations of Solid Edge assemblies. Exploding an assembly allows you to control the movement, sequence and grouping of parts and subassemblies. Rendering a view allows you to define textures, lighting, shadows, backgrounds and other properties to create presentation style images. Motors apply movement to under constrained parts in an assembly which can be animated. Using animation, you can combine previously created exploding sequences and custom camera movement to create animation. Each frame of the animation can be rendered to create presentation quality animations.

Once you complete the activities in this course, you will be able to:

- Add a motor to a unconstrained part of an assembly to create movement for an animation.
- Manipulate animation events and explode events to create an animation of an assembly.

## Defining motors



There are two types of motors that can be defined in Solid Edge: rotational and linear.

You use motor features to help you observe how a set of under-constrained parts will move relative to the part you define as a motor. This allows you to design and simulate complex mechanisms where the movement of a set of interrelated parts needs to be simulated.

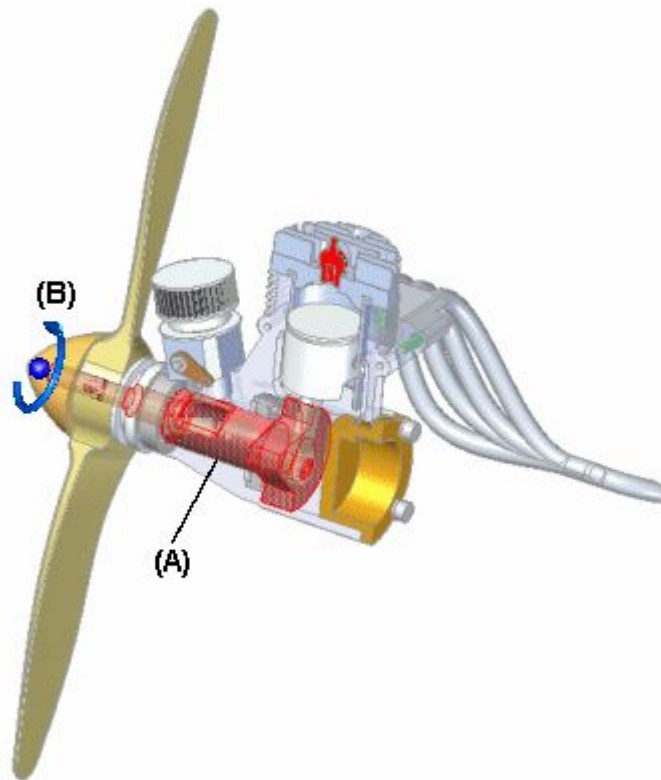


## Motor command

Defines a rotational or linear motor using an element on a selected part. You can then use the Simulate Motor command to display a kinematic simulation of the motion in an assembly.

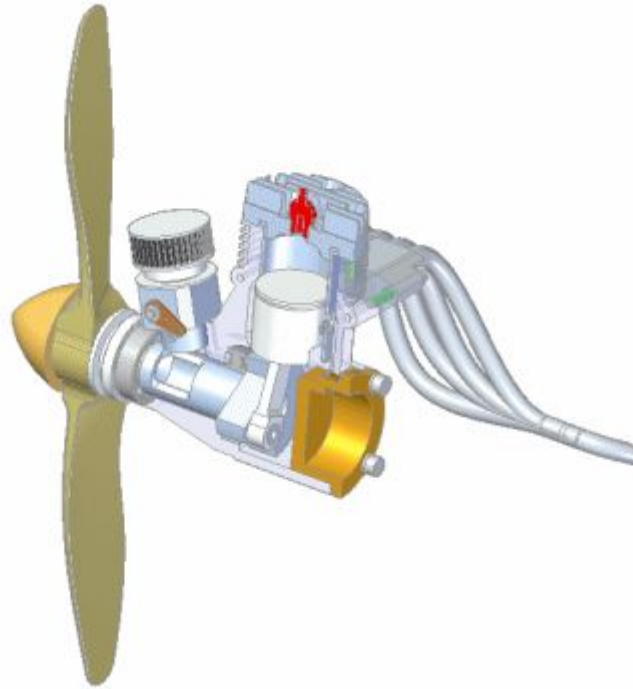
You use motor features to help you observe how a set of under-constrained parts will move relative to the part you define as a motor. This allows you to design and simulate complex mechanisms where the movement of a set of interrelated parts needs to be simulated.

This is useful when working with assemblies that contain moving parts such as gears, pulleys, crankshafts, parts that travel in grooves or slots, and hydraulic or pneumatic actuators. For example, you can specify that a crankshaft part (A) in a mechanism rotates around an axis you specify (B).



You can then use the [Motor Simulation command](#) to playback a kinematic simulation of how the under-constrained parts in the assembly move.

Press F5 to replay the animation.



You can define properties for the motor, such as the type of motor, the motor rate or speed, motor direction, and any limits you may want to place on the motor.

When you define a motor feature using the Motor command, an entry is added for the motor feature to PathFinder. You can select the motor entry in PathFinder to edit the motor feature later.

## **Types**

You can define the following types of motors:

- Rotation
- Linear

## **Steps**

The basic steps for defining a motor are:

- Specify the type of motor you want, Rotation or Linear.
- Select the part you want to act as a motor.
- Define the movement axis.
- Specify the motor rate and limits.

## **Specifying Motor Type**

The Motor Type list on the command bar allows you to define the type of motor you want. You can specify whether you want the motor type to be Rotation or Linear.

### Selecting the Part

You can only select a part that is under-constrained, or has relationships suppressed. The assembly should also be under-constrained such that the mechanism is free to move in the proper axes.

### Defining the Movement Axis

Depending on the motor type you specify, you can select faces, edges, or cylindrical axes to define the motor axis. For example, to define a Rotary motor, you can select cylindrical faces, cylindrical edges, or cylindrical axes.

### Specifying the Motor Rate and Limits

The Motor Value and Limits options on the command bar allow you to specify the speed or rate you want the motor operate at, and any limits on the travel you want to impose. For example, you may want to specify that a rotational motor rotates at 1750 revolutions per minute, and makes two complete revolutions (720 degrees).

You can set the working units you want to use for the angular and linear velocity of a motor using the Advanced Units button on the Units tab of the File Properties dialog box, on the File menu.

## Motor Definition and Simulation Guidelines

You can define as many motors as you want in an assembly. When you define multiple motors in an assembly, use the [Motor Group Properties dialog box](#), available with the Simulate Motor command and the [Animation Editor tool](#) to specify which motors you want to use, whether you want to detect collisions during the simulation, and so forth.

When working with more than one motor, use the Animation Editor tool to specify when the motors start time, duration time and stop time for each motor. This allows you to design and simulate complex mechanisms where the timing and positioning of the parts is critical to understanding the behavior of the mechanism.

### Note

Only motors in the active assembly participate in a motor simulation. If you want subassembly parts to move in response to a motor simulation, you need to make the subassembly adjustable, using the Adjustable Assembly command on the PathFinder shortcut menu.

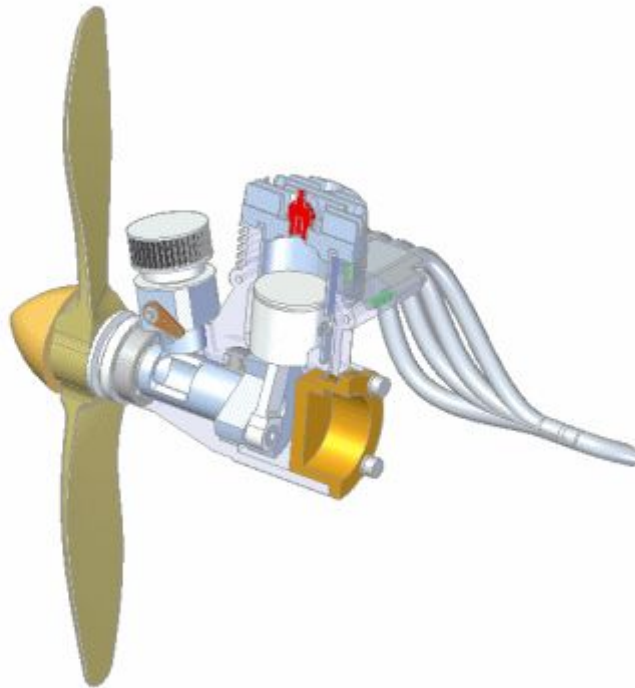




## **Simulate Motor command**

Display a kinematic simulation of motion in an assembly. You use motor features to define how a set of interrelated parts will move. This is useful when working with assemblies that contain crankshafts, gears, pulleys, and hydraulic or pneumatic actuators.

Press F5 to replay the animation.



When you click the Simulate Motor button, the Motor Group Properties dialog box is displayed, so you can specify which motors you want to use, whether you want to detect collisions during the simulation, and so forth. When you click OK, the Animation Editor tool is displayed so you can run the simulation. To run the simulation, click the Play button.

### **Note**

The Simulate Motor command contains a subset of the Animation Editor functionality. To access the full functionality of the Animation Editor tool, you must use the Animation Editor command in the Explode-Render-Animate application. To access the Explode-Render-Animate application, on the Tools tab, click Explode-Render-Animate.

**Motor Group Properties dialog box****No Analysis**

Allows you to move under-constrained parts and observe the results.

**Detect Collisions**

Allows you to detect collisions during motor animation.

**Physical Motion**

Allows you to simulate physical motion between parts. This option detects contact between unconstrained surfaces and applies temporary constraints between the contacting surfaces. This makes it possible to analyze motion in mechanisms that contain gears and other forms of sliding contact.

**Motor Duration**

Specifies how the motor duration is defined.

**Use Motor Limits as Duration if Defined**

Specifies that the motor limits define the duration.

**Default Motor Duration**

Specifies the motor duration in seconds. You can type a value.

**Available Motors**

Lists the available motors. You can use the Add and Remove buttons to add motors to and remove motors from the Motors in Animation list.

**Add**

Adds a motor to the Motors in Animation list.

**Remove**

Removes a motor from the Motors in Animation list.

**Motors in Animation**

Lists the motors that will be used in the animation.

## **Activity: Motor**

### **Activity objectives**

In this activity, you assign a motor to a part in an assembly. The motor type is rotational, and it is applied to a gear in a clock. The speed of the motor is defined so that the second hand of the clock moves at the operating speed of 1 rpm. The gear relationships are predefined, and by assigning the motor to the appropriate gear, you can show motion through motor simulation. This motor simulation is used in a later activity to create an animation of the clock.

**Activity: Creating a motor**

**Step 1**

You will open an assembly, then inspect and change the rotation units for the assembly and assign a motor to a part in the assembly. Once the motor is defined, a motor simulation will be generated to be used later in an animation.


**Note**

A motor can only be assigned to a part that is under constrained in an assembly. If the under constrained part used to define the motor exists in a subassembly, then the subassembly will have to be made into an adjustable assembly rather than a rigid assembly. By making a subassembly adjustable, the relationships used to position the parts in the subassembly are promoted into the higher level assembly and solved at that level.

You will set the Angular Velocity units to revolutions per minute.

- Open the assembly *motor.asm* with all the parts active.



- Click the Application button , and then choose Properties@ File Properties.
- Click the Units tab, and on the Units tab, click Advanced Units. Set the Angular Velocity to rpm, and then click OK to dismiss the Advanced Units dialog box. Click OK to return to the assembly.

Name	Value	Precision	
Density	kg/m <sup>3</sup>	0.123	
Volume	mm <sup>3</sup>	0.123	
Mass	kg	0.123	
Scalar		0.123	
Coef. of Thermal E...	/C	0.1234	
thermal conductivity	kW/m-C	0.123	
Specific Heat	J/kg-C	0.123	
Stress	kPa	0.123	
Linear Density	kg/m	0.123	
Force	mN	0.123	
Frequency	MHz	0.123	
Angular Velocity	rpm	0.123	
Linear Velocity		0.123	

	Cancel
--	--------

*Step 2*


Create the motor and define the motor parameters.

- In PathFinder, right-click the part *G05\_62.par* and then click Show Only. Click Fit to see the gear.



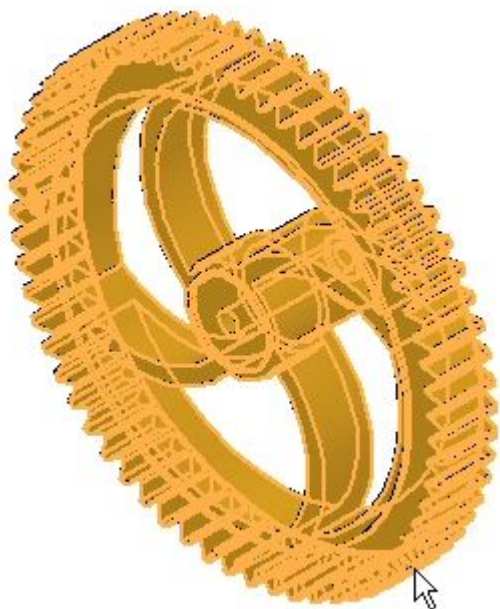
**Note**

As an option, to better understand how the predefined gear relationships in this assembly, you can review the spreadsheet named *clock\_gears.xls*, which is located in the same folder as the assembly. This spreadsheet shows the relationships and gear ratios used to create the clock mechanism.

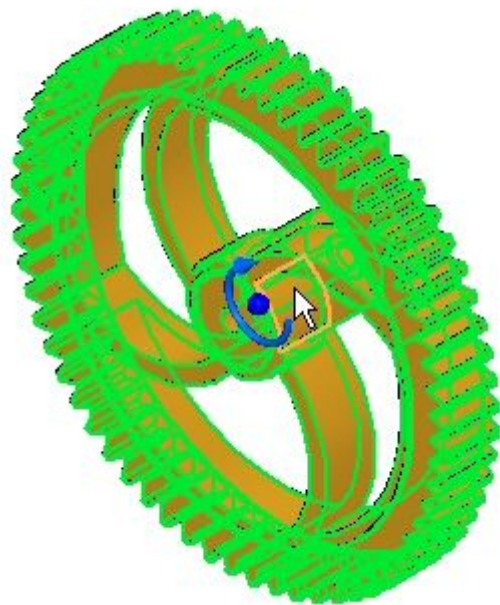
- Choose Home tab® Assemble group® Motor .
- Set the motor type to Rotation.
- Set the rotation rate in rpm by entering 1/60.



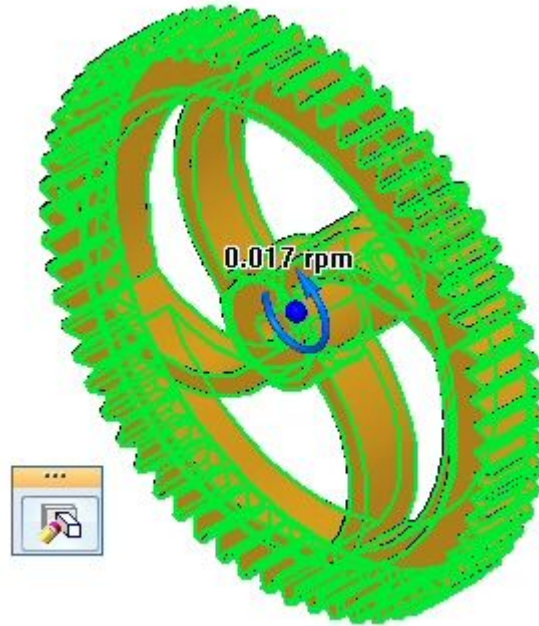
- ▶ Select the gear as the under-constrained part.



- ▶ Select the interior cylindrical face to define the rotation axis.



- The rotation is defined as counterclockwise as shown. You can change the direction by clicking the Flip direction button.



- Click Finish to complete the motor definition.

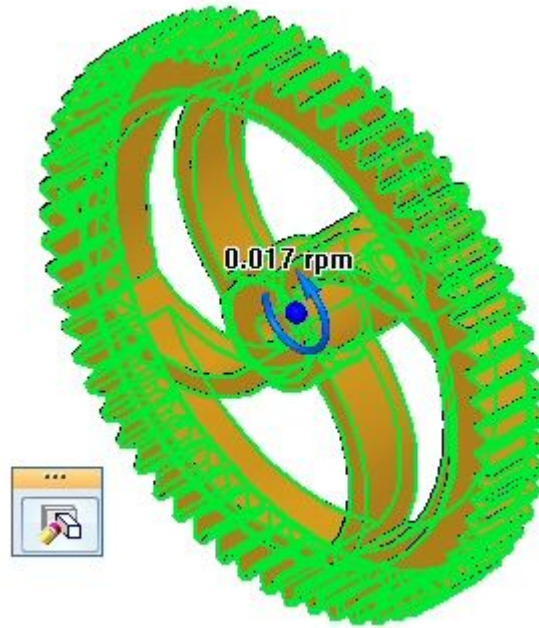
### Step 3

Once you have defined a motor, you can change a parameter by editing the motor. Even though the motor is defined correctly, this step shows how you can edit it. You will ensure that the direction of rotation for this motor is counter clockwise. If not, you can reverse the rotation.

- On the ribbon, click the Select command and select Motor 1 in PathFinder.



- ▶ The rotation is displayed. If the rotation is counter clockwise as shown, skip the next step.



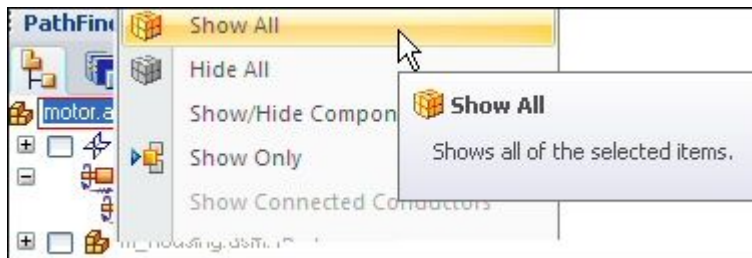
- ▶ To change the direction of rotation, click Edit Definition. Click the Flip Direction button, then click Finish.



Step 4

Display the motor. For best visibility, all the parts of the assembly will be shown, and then some will be hidden.

- ▶ Click the Select command and right-click *motor.asm* in PathFinder. Click Show All.





- Using the same procedure as in the previous step, hide *m\_housing.asm*, and then fit the assembly.

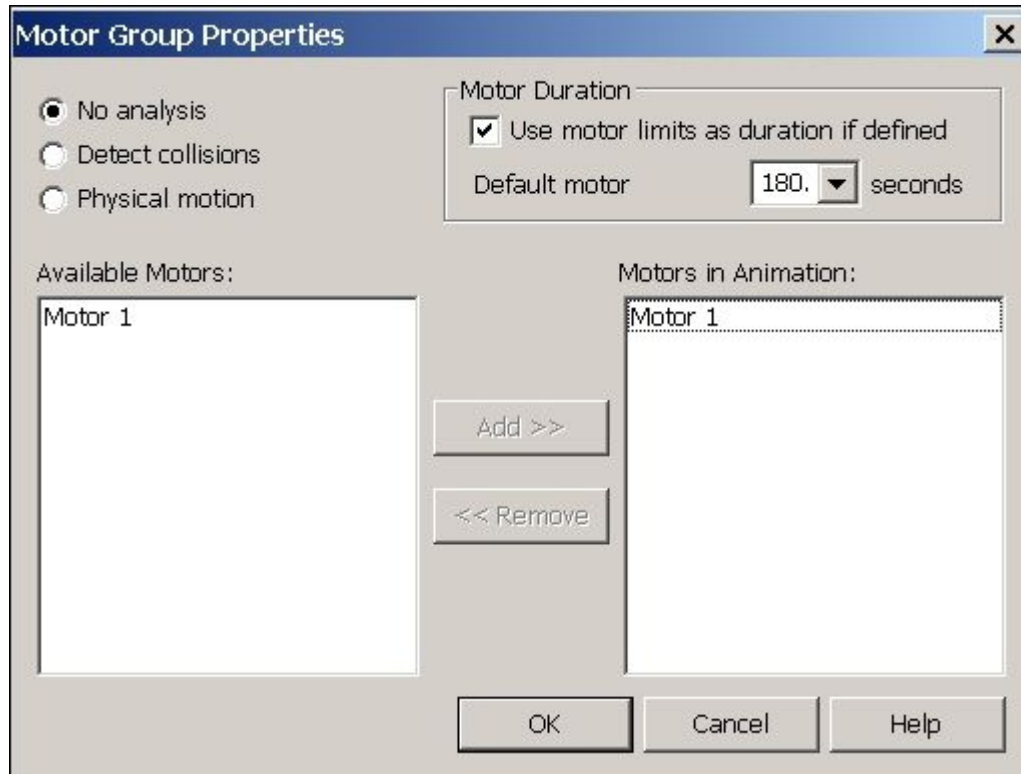


Step 5

Create a motor simulation.



- ▶ Choose Home tab@ Assemble group@ Simulate Motor
- ▶ In the Motor Group Properties dialog box, set the Motor Duration to 180 seconds (3 minutes) and the other values as shown, and then click OK.



**Note**

If you need to change any of these values at a later time, right-click Motors in the time line and then click Edit Definition. You can define multiple motors in a simulation but for this activity, you define a single motor.

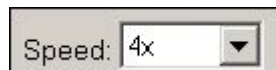
- ▶ The controls for playing the animation are shown below.



- ▶ Click Play to start the animation.



- ▶ As the animation is playing, increase the Speed to 4x.



**Note**

Changing the speed to 4x is for animation display purposes only. The motor is still spinning at the assigned rpm.

- ▶ Click the Stop button to halt the motor simulation.



- ▶ Click Go to Start, to reset the animation to the initial point.



- ▶ Set the Speed back to 1x.
- ▶ Click the Application button.



- ▶ Click Save. When prompted to save changes to the animation editor, click yes.
- ▶ In PathFinder, right-click *motor.asm* and click Show All.
- ▶ Save and close this assembly. This completes this activity.

*Activity summary*

In this activity, you learned how to create and simulate a motor. The motor animation created will be used later during the explode sequence. In the activity, the following topics were covered:

- The speed and direction of a rotational motor was defined.
- Parameters used to define the motor were changed by an editing process.
- The motor simulation was created and run.
- Animation controls and time line were introduced.
- A motor time line was created to be used in exploding and an animation.

**Lesson review**

Answer the following questions:

1. Name two types of motors.
2. How do you change the rotational units from degrees per second to revolutions per minute?
3. Can a motor that exists in a subassembly be used to apply motion to unconstrained parts in a higher level assembly? If so, explain how.
4. Once a motor is defined, how do you get it to drive unconstrained parts and see the movement?
5. Can the movement created by a motor be seen during an animation, along with explosion events?

**Answers**

1. Name two types of motors.

The two types of motors are linear and rotational.

2. How do you change the rotational units from degrees per second to revolutions per minute?

To change the rotational units from degrees per second to revolutions per minute, click the advanced units button in file properties>units.

3. Can a motor that exists in a subassembly be used to apply motion to unconstrained parts in a higher level assembly? If so, explain how.

A motor that exists in a subassembly can be used to apply motion to unconstrained parts in a higher level assembly if the subassembly is defined as an adjustable assembly.

4. Once a motor is defined, how do you get it to drive unconstrained parts and see the movement?

The simulate motor command will show the movement created by a motor.

5. Can the movement created by a motor be seen during an animation, along with explosion events?

The movement created by a motor can be seen during an animation, along with explosion events.

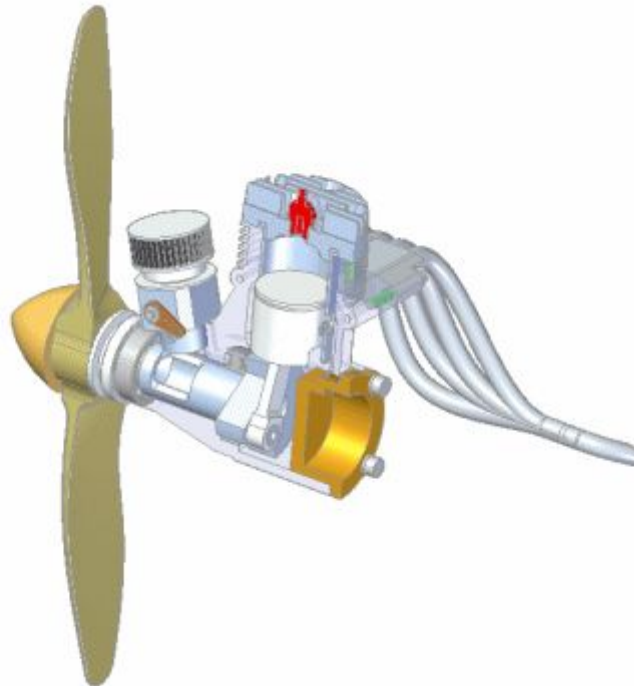
### **Lesson summary**

In this lesson, you learned how to create and simulate a motor. The motor animation created will be used later during the explode sequence. In the activity, the following topics were covered:

- The speed and direction of a rotational motor was defined.
- Parameters used to define the motor were changed by an editing process.
- The motor simulation was created and run.
- Animation controls and time line were introduced.
- A motor time line was created to be used in exploding and an animation.

## Animating an assembly

Solid Edge enables you to easily create animated presentations of your assemblies. Assembly animations can be useful for motion studies of mechanisms, visualizing how parts are assembled into a completed assembly, and for vendor or customer presentations.







## **Animation Editor command**

Displays the Animation Editor Tool so you can create, display, and edit animations of an assembly.



You can define the following types of animation events:

- Camera
- Motor
- Explosion
- Appearance
- Motion Path

You can use the controls on the Animation Editor tool to play, stop, pause, and rewind the animation in the graphic window.

You can also save an assembly animation in AVI format with the Save As Movie button on the Animation Editor tool.

### **Note**

When you are working in Solid Edge Embedded Client, AVI files are saved to unmanaged locations.

For more detailed information on creating assembly animations, see the Creating Assembly Animations Help topic.

## Animation Editor Tool

### Animation List

Lists the existing animations. You can select an animation entry from the list for playback and editing purposes.



#### New Animation

Displays the [Animation Properties dialog box](#) so you can define the properties for a new animation.



#### Save Animation

Saves the current animation.



#### Delete Animation

Deletes the current animation.



#### Animation Properties

Displays the [Animation Properties dialog box](#) so you can edit the properties for an existing animation.



#### Save as Movie

Displays the Save as Movie dialog box so you can save the current animation as an AVI file.



#### Camera Path

Displays the Camera Path Wizard so you define the camera path you want.



#### Display Camera Path

Displays the camera path as a curve in the graphic window. This can be useful in visualizing the path the camera will take during the animation.

### Animation Events List (Left Pane)

Lists the event types available in the current animation. Depending on the current animation, you can define events or select existing events for the camera, motor, explosion, appearance, and path you want to use for the current animation. You can expand, collapse, and select items in the list. Shortcut menu commands are available that allow you to define the event you want to use for the animation, delete the current event, and so forth.

### Speed

Specifies the speed you want to use for playback purposes. The speed setting does not affect the speed of an AVI recording, or the relative speed of the animation entries.

### Go to Start

Moves the current frame indicator to the start of the animation.

### Previous Frame

Moves the indicator to the previous frame.

**Play/Pause**

Plays or pauses the current animation.

**Stop**

Stops the playback of the animation.

**Next Frame**

Moves the indicator to the next frame.

**Go To End**

Moves the current frame indicator to the end of the animation.

**Time**

Displays the current time in the animation.

**Frame**

Displays the current frame in the animation.



**Toggle Scale**

Toggles the timeline scale between frames and seconds.



**Zoom Out**

Reduces the scale of the animation timeline.



**Zoom In**

Increases the scale of the animation timeline.



**Minimize**

Minimizes the Animation Editor.



**Motion Path**

Displays the [Motion Path command bar](#) so you can select components and draw a curve path to guide component movement.

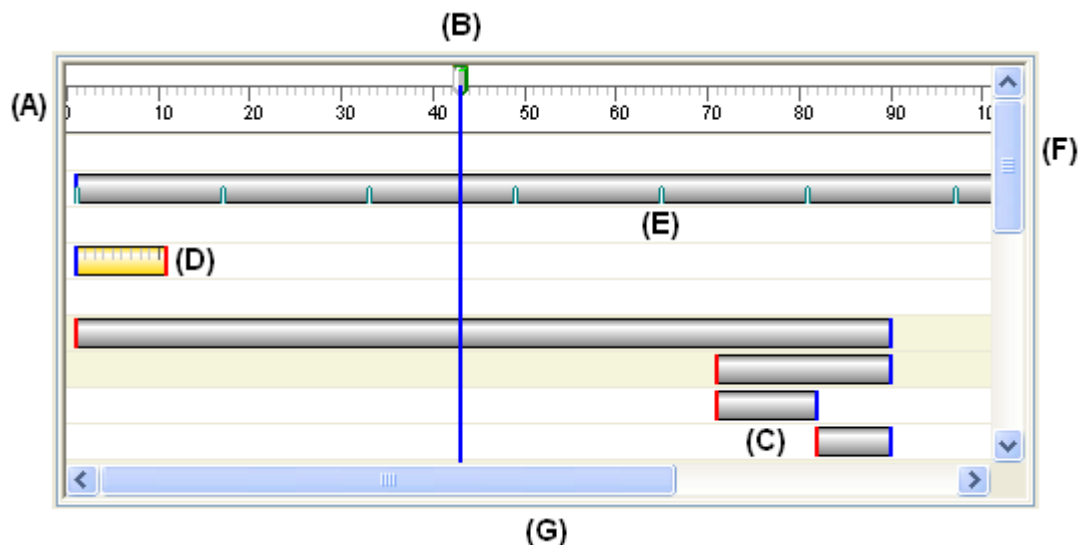


**Appearance**

Displays the Appearance command bar so you can create an appearance event. For example, you may want a part to fade in or fade out in your animation.

**Event Timeline and Duration List (Right Pane)**

Displays [event duration bars](#), which represent the start and stop times, and elapsed time for each event in the current animation timeline. You can edit event duration bars by dragging with the cursor or using shortcut menu commands. This allows you to customize the animation.



Basic user-interface elements in the right pane include:

(A) Frame Scale. You can use the Toggle Scale button to change the scale display between Frames and Seconds.

(B) Current Frame Indicator. The current frame is the frame which is displayed in the graphic window. You can use the cursor to drag the Current Frame Indicator to another location to view individual frames in the animation.

(C) Event Duration Bars. Notice that a different color is used at the start and end locations.

(D) Selected Event Duration Bar. Notice that a scale is displayed when a duration bar is selected. This can make it easier to precisely relocate a duration bar with respect to Frame Scale.

(E) Event Duration Bar Key Frame Indicator.

(F) Vertical Scroll Bar. Allows you to scroll the timeline up and down.

(G) Horizontal Scroll Bar. Allows you to scroll the timeline left and right.

### Shortcut Menu Commands

The following shortcut menu commands are available. The shortcut menu commands are context-sensitive. In other words, the commands which are available change based on what is selected.

#### Left Pane Shortcut Menu Commands

Delete

Deletes the selected event.

Rename

Renames the selected event.

### Edit Definition

Allows you to define a new event or edit an existing event. The action you can perform depends on the event type and whether you are defining a new event or editing an existing event.

#### Camera

The Edit Definition command displays the [Camera Path Wizard](#) when you click the Camera category entry, and displays the Path command bar when you select an existing camera event.

#### Motor

The Edit Definition command displays the [Motor Group Properties dialog box](#) when you select the Motor category entry.

#### Explosion

The Edit Definition command displays the [Explosion Properties dialog box](#) when you select the Explosion category entry.

#### Appearance

The Edit Definition command displays the Appearance command bar when you select an existing Appearance event.

#### Paths

The Edit Definition command displays the Path command bar when you select an existing Path event.

### Expand All

Expands all the event collections.

### Right Pane Shortcut Menu Commands

#### Cut

Cuts the selected event from the animation and places it on the Clipboard.

#### Copy

Copies the selected event from the animation and places it on the Clipboard.

#### Paste

Pastes an event onto the animation timeline.

#### Delete Duration

Deletes the selected event duration.

#### Insert Key Frame

Inserts a keyframe at the current cursor position. This option is available when the cursor is over a camera or motion path event duration bar.

#### Delete Key Frame

Deletes a keyframe at the current cursor position. This option is available when the cursor is over a camera or motion path event duration bar.

#### Insert Camera Location

Inserts a camera location at the current cursor position. This option is available when the cursor is over a camera event duration bar.

**Add Frames**

Displays the Add Frames dialog box so you can add frames or time to an animation event.

**Remove Frames**

Displays the Remove Frames dialog box so you can remove frames or time to an animation event.

**Properties**

Displays the [Duration Properties dialog box](#) so you can redefine the start time, end time, or elapsed time for an animation event.

## **event duration bar**

A user-interface element on the timeline (right) pane of the Animation Editor tool.

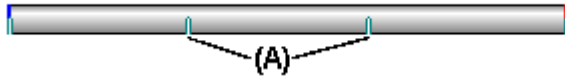
Event duration bars allow you to visualize and control the timing of events in an assembly animation.

Event duration bars represent the start time, elapsed time, and end time for an animation event. There are two basic types of event duration bars:

- Duration bars for explode, appearance, and motor events.



- Duration bars for camera and motion path events. Duration bars for these event types also support key frames (A).



You can move and modify duration bars using the cursor and shortcut menu commands.

**Camera Path Wizard command**

Runs the Camera Path Wizard, which guides you through the process of creating a camera path for an assembly animation. The Camera Path Wizard allows you define the camera path name, camera direction, and so forth. You can use camera paths as part of an assembly animation.



## **Animation Properties dialog box**

### **Animation Name**

Displays the name of the animation.

### **Frames Per Second**

Specifies how many frames per second are used to build your animation. You can specify a standard frame rate, such as NTSC or PAL, or a custom frame rate.

### **NTSC**

Specifies that the NTSC standard frame rate is used to build the animation.

### **PAL**

Specifies that the PAL standard frame rate is used to build the animation.

### **Custom**

Allows you to define a custom frame rate. Type the number of frames per second you want.

### **Animation Length**

Specifies the duration of the animation in seconds.

## **Duration Properties dialog box**

---

### **Start Frame**

Lists the current start frame. You can type a new value to change the starting time for the event duration.

### **End Frame**

Lists the current end frame. You can type a new value to change the ending time for the event duration.

### **Entry Duration**

Lists the current duration of the event entry. When you change this value, the End Frame value also updates.

## Path command bar (Animation Editor Tool)

This command bar is displayed when you are creating or editing Motion Path or Camera Path events.

### Select Parts Step

Specifies the parts you want to follow the motion path. Select the parts you want in the graphic window. This step is only available when creating or editing a motion path.

### Draw Path Step

Draws a path to guide component or camera motion. You can draw a curve that defines the path in the graphic window.

### Finish/Cancel

This button changes function as you move through the motion path definition process. The Finish button applies the motion path properties you defined. The Cancel button discards any input and exits the command.

### Select Parts Step Options

#### Deselect (x)

Clears the selection.

#### Accept (check mark)

Accepts the selection.

### Draw Path Step Options

#### Activate Part

Activates the selected part.

### Keypoints

Sets the type of keypoint you can select to define the motion path curve. The available keypoint options may be different than displayed below.



Allows you to select any keypoint.



Allows you to select an end point.



Allows you to select a midpoint.



Allows you to select the center point of a circle or arc.



Allows you to select a tangency point on an analytic curved face such as a cylinder, sphere, torus, or cone.



Allows you to select a silhouette point.



Allows you to select an edit point on a curve.

### Open Path

Sets the path curve type to open.

**Closed**

Sets the path curve type to closed. The start and end points of a closed path curve are coincident. If the start and end points you selected are not coincident, and you set the Closed option, a keypoint that is coincident with the start point is automatically added to the curve.

When you set this option, the Start and End options are automatically set to Periodic. Periodic curves are closed, connected, and tangent at the first and last points on the curve.

**Frame Count**

Specifies the total frame count for the path.

**Straight Path**

Creates the path with straight segments.

**Blend Path**

Creates the path with curved or blended segments.

**Hold**

Holds the component position until the next key frame.

**Deselect (x)**

Clears the selection.

**Accept (check mark)**

Accepts the selection.

## **Activity: Animating an assembly**

### **Activity objectives**

In this activity, you will use the animation portion of the Explode-Render-Animate application to produce a presentation quality animation. The animation will be created and then different effects will be used to edit the animation and give the desired results. The animation time line you create will consist of a motor driving under-constrained parts, explosion events, camera events, and appearance events. The end result will be an *.avi* movie.

**Activity: Animating an assembly**

**Step 1**

In this activity you will open an assembly that contains a motor and an exploded configuration. You will use the Animation Editor to manipulate the events that occur during the animation. You will create an animation that consists of camera movement, changes in part appearance, part motion paths, exploded views and motion from motors.

You will define a camera path for a predefined animation.

**Note**

The description of the animation controls is also covered in the activity exploding an assembly.

- ▶ Open the assembly *animate.asm* with all the parts active.



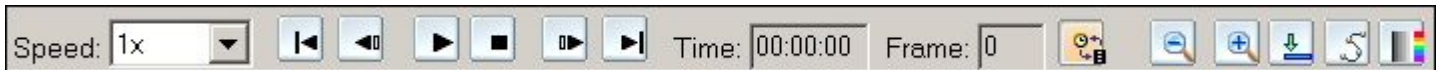
- ▶ Choose Tools tab@ Environs group@ ERA



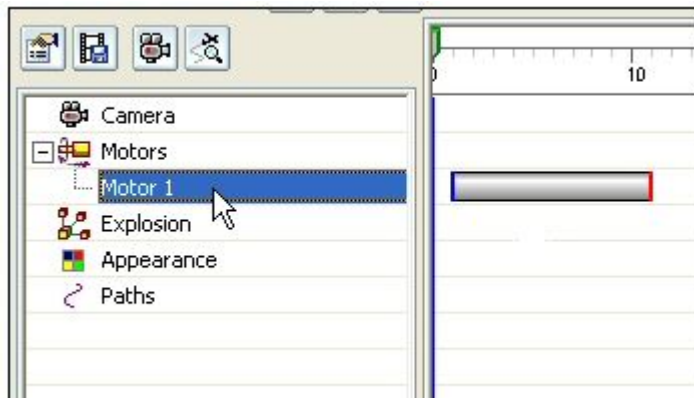
- ▶ Choose Home tab@ Animation group@ Animation Editor

- ▶ Examine the Animation Editor.

The right pane is the time line for each of the animation events. A motor was defined previously in this assembly. Controls for playing the animation are displayed.



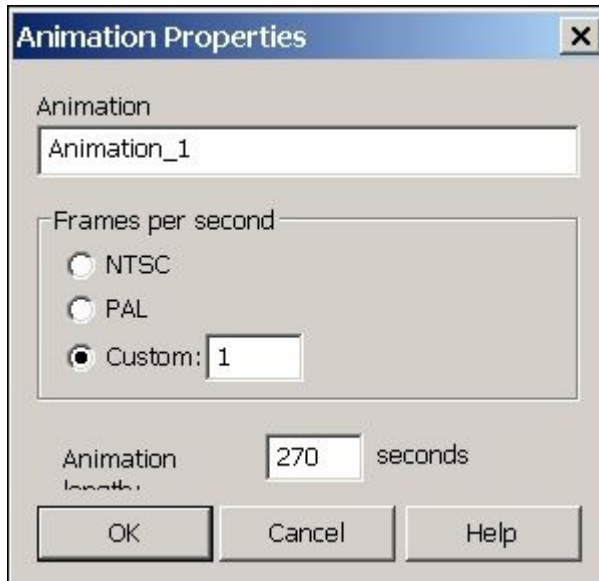
The left pane displays the animation events, and the right pane displays the event duration bars. These can be used to define and sequence the events of the animation.



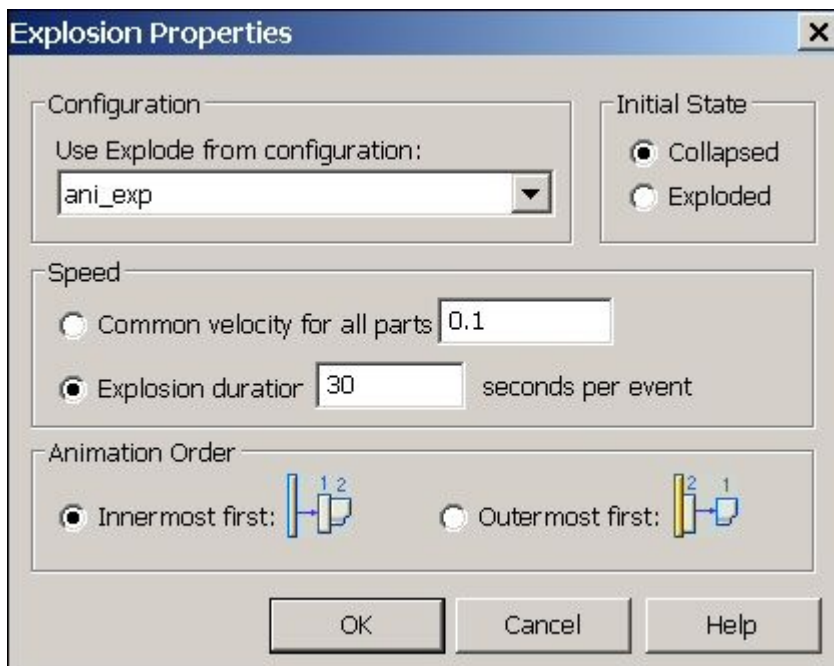
- ▶ Click Animation Properties.



- ▶ Set the values as shown, and then click OK.



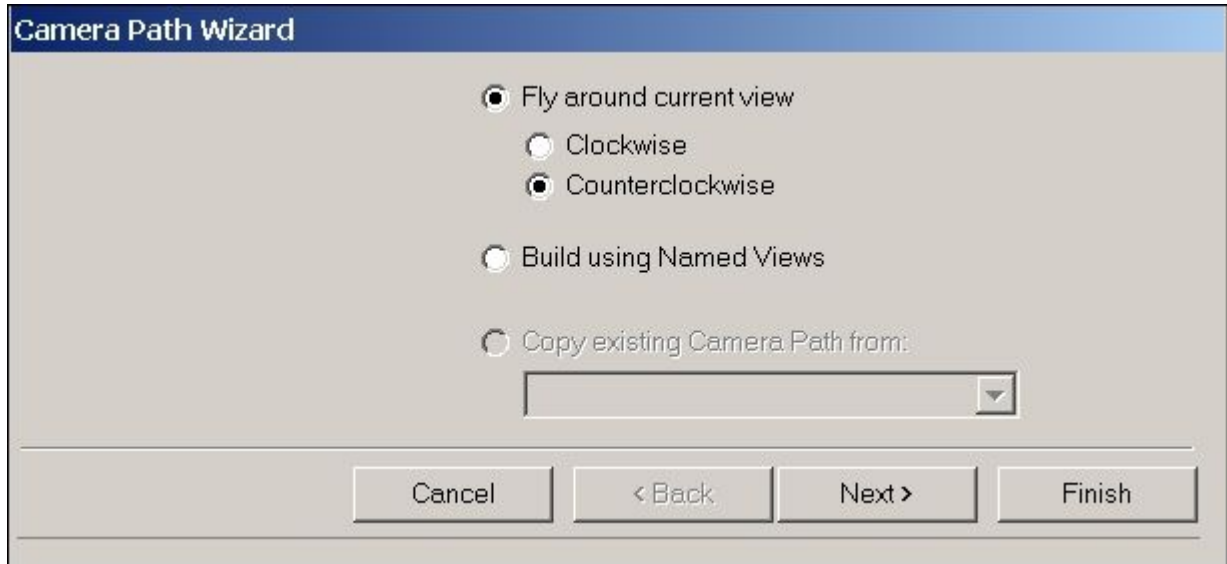
- ▶ Right-click the Explosion event and then click Edit Definition. Examine the parameters previously defined for this explosion. Click OK when finished.



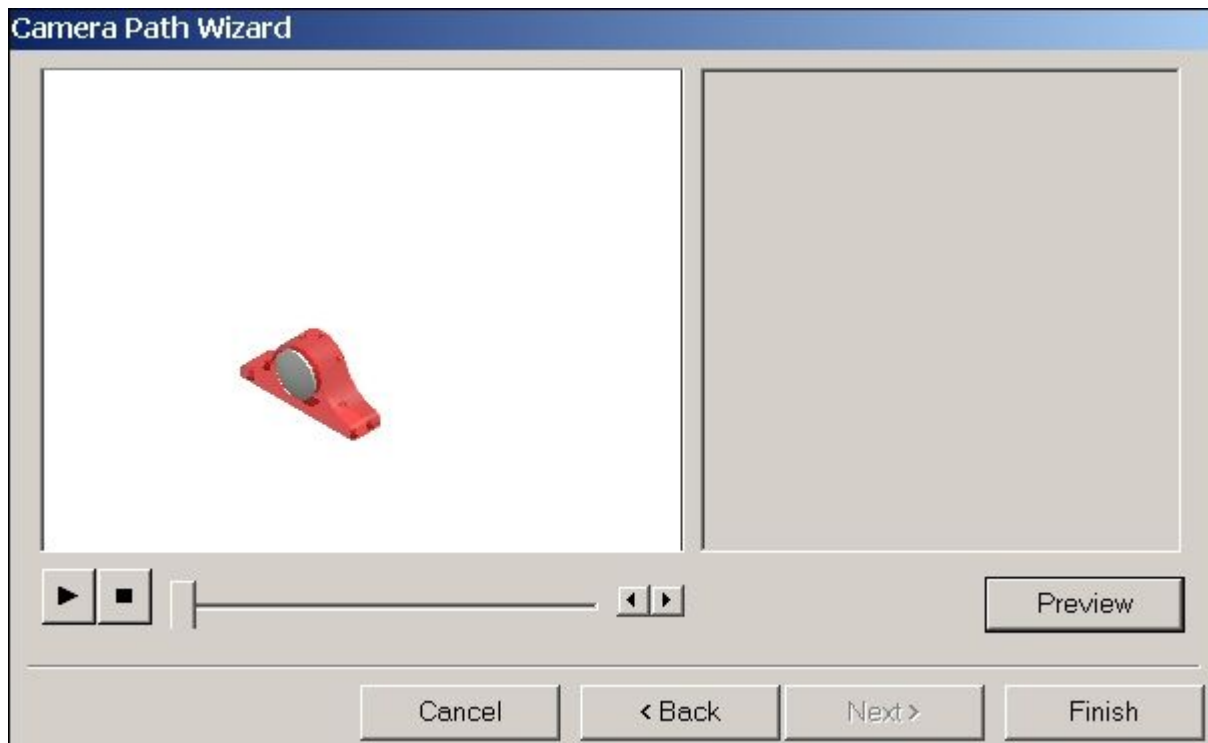
- ▶ Click the Camera Path command to open the Camera Path Wizard.



- ▶ Set the values as shown, and then click Next.



- ▶ Click Preview. Observe the animation preview, and then click Finish. The camera path is created.

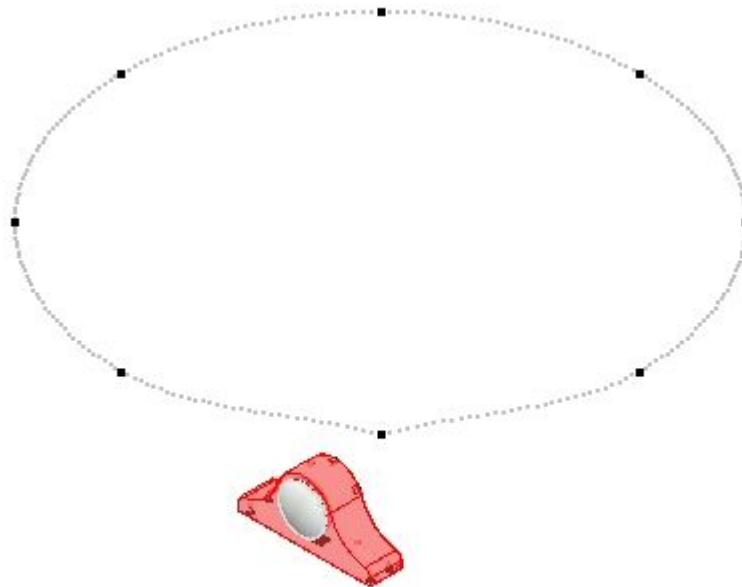




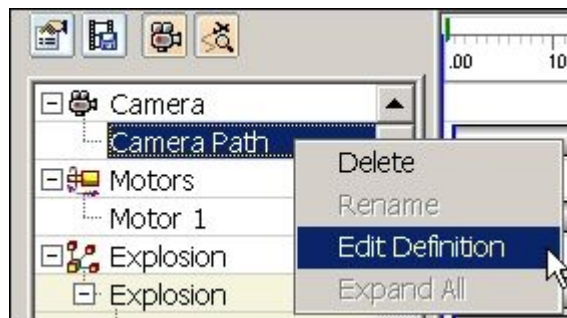
Step 2

You will view the camera path and then edit the path.

- ▶ Click the Show Camera Path command. The path is displayed.

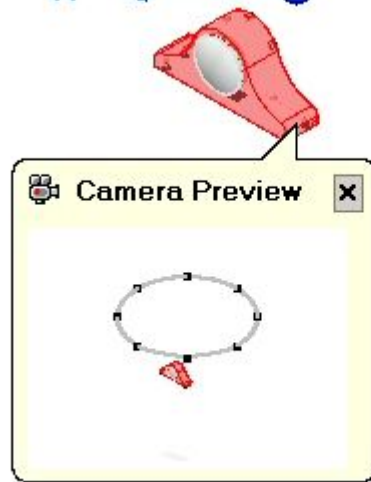
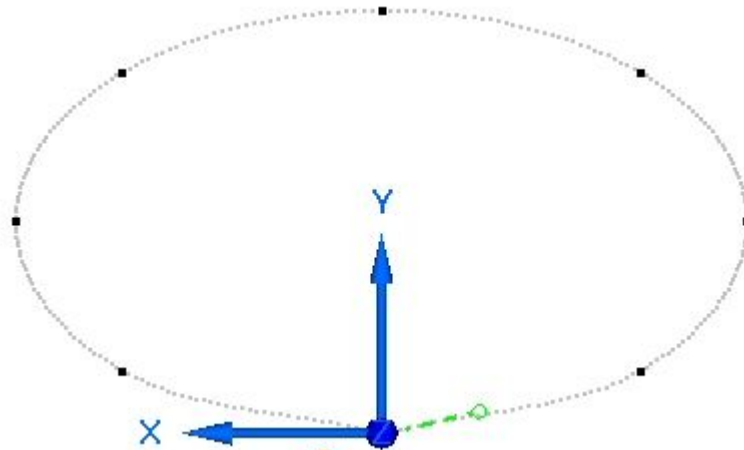


- ▶ Right-click the Camera Path in the event time line, and then click Edit Definition.



- ▶ On the command bar, click the Draw Path Step group button and observe the controls.

Name:



- ▶ The camera path can be either open or closed. For this path click Closed.



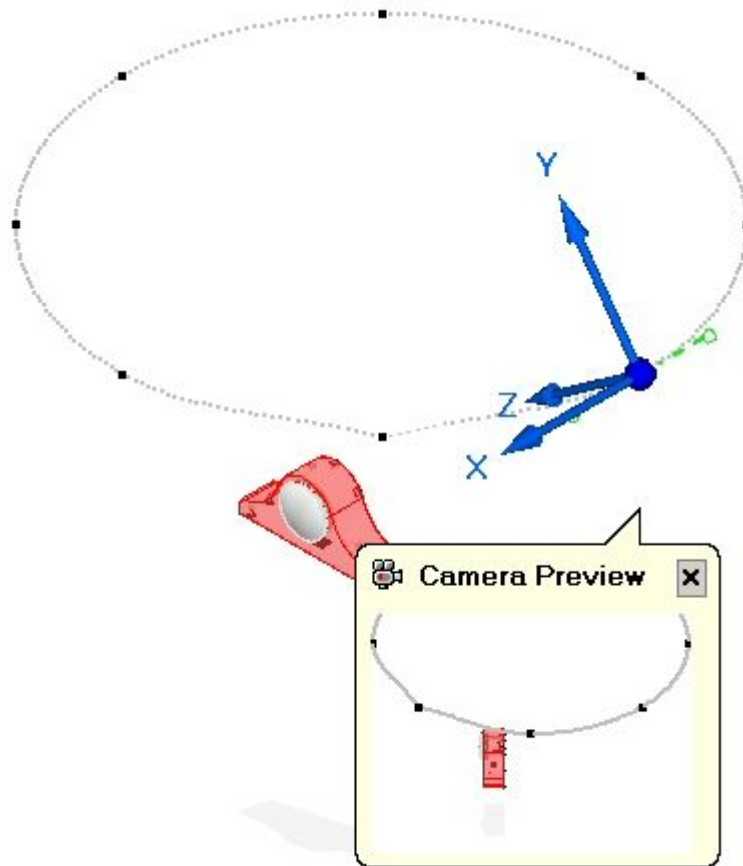
- ▶ The key points of the camera path are graphically displayed as points on the curve. The X-Y-Z axis displays at the location of the key point which is currently being edited and a preview of what the camera sees at that frame is shown. The frame count showing the duration of the camera movement is displayed.

Frame count:

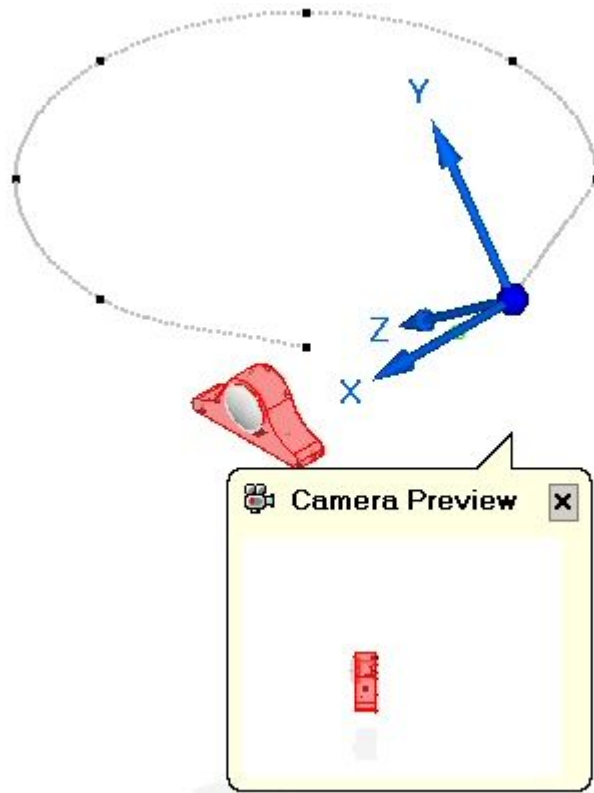
- ▶ Camera movement and direction can be edited at each key point. The blue navigation arrows move to the next or previous point for making changes to those points. Click the Next Point button, which is the right blue arrow.



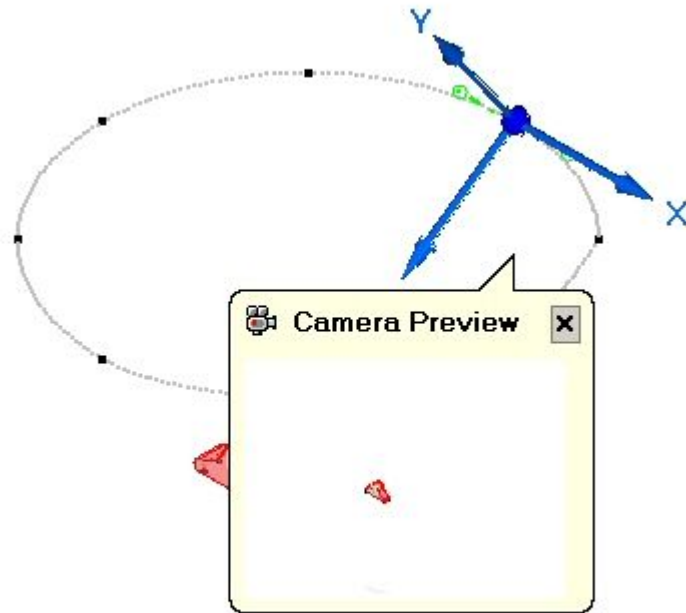
Notice the camera preview and X-Y-Z axis move to the next point.



- ▶ Click the Straight Path button. This changes the curvature of the previous point from a curve to a straight path.



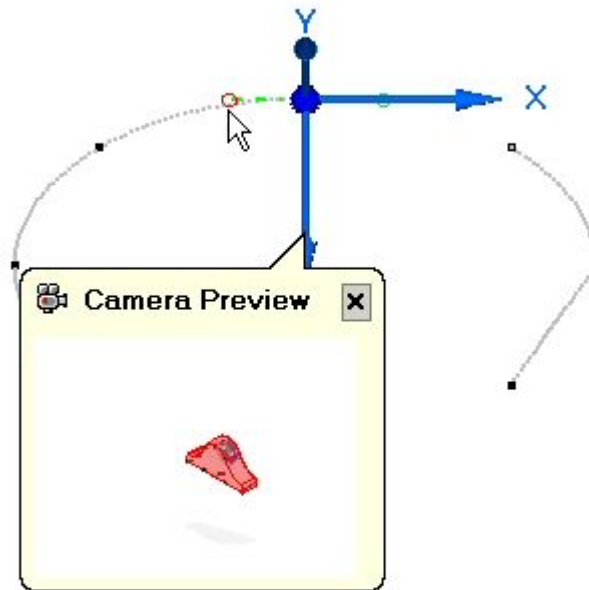
- ▶ Click the Next Point button twice to move the camera.



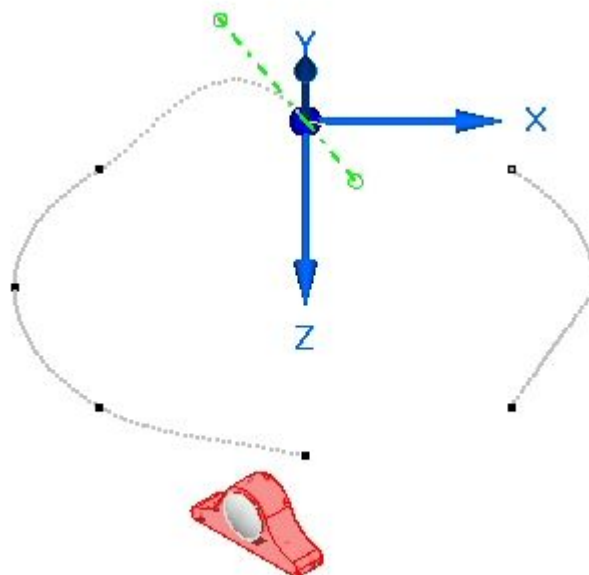
- ▶ Click Hold. This freezes the animation until the next key frame is reached in the time line.



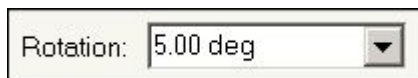
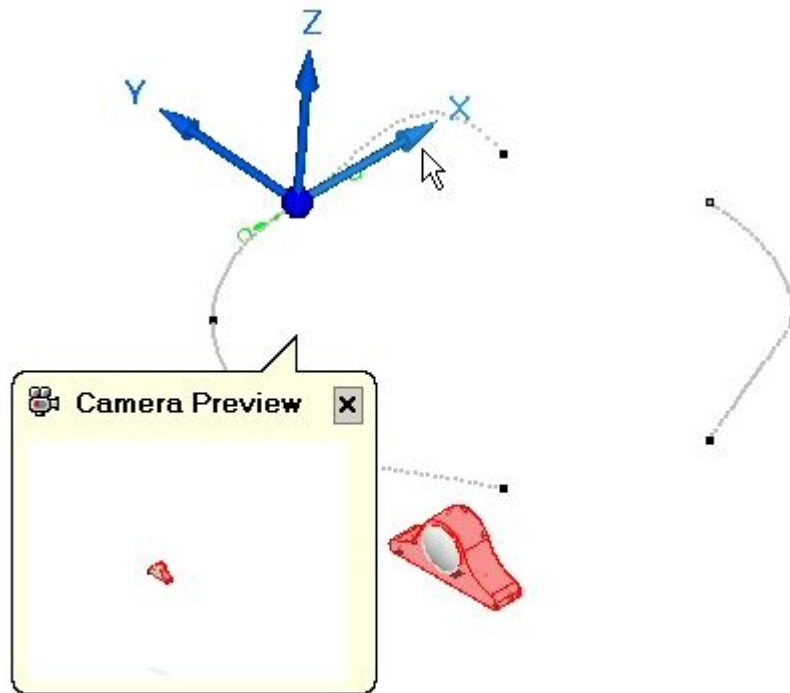
- ▶ Click the Next Point button once. Drag the tangency control handle of the key point.



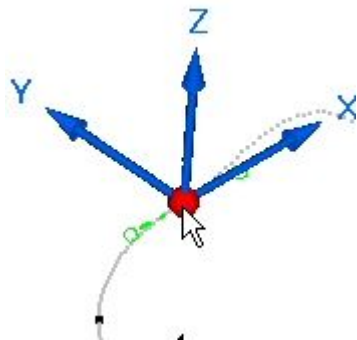
- ▶ Drag the handle so that the curve is approximately positioned as shown.



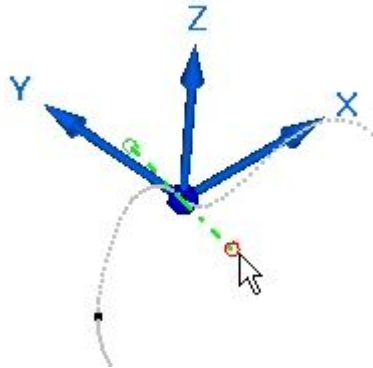
- ▶ Click Next Point. Select the X axis of the triad and enter a rotation angle of 5°.



- ▶ Select the origin of the triad. The curvature tangency handles are available at the key point.



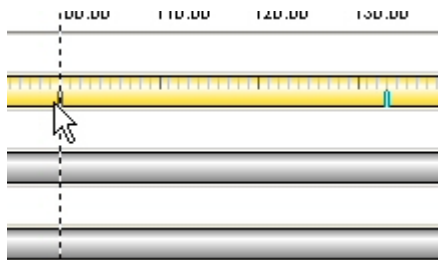
- ▶ Drag one of the handles so that the curve is approximately modified as shown.



- ▶ Click Finish. Play the animation and observe how the edits to the camera path affected the animation. After the animation plays, reset the animation by stopping the animation and then clicking the Go to Start button.



- ▶ On the camera path event bar, notice the key points are displayed as green graduation marks. Click and drag any of the key points to a different position. This changes the time at which a key point occurs, and increases the speed of the transition from that key point to the key point in the direction you moved the point. Likewise, the duration of the transition of the camera path is increased from the key point moved and the point you moved away from.



- ▶ Play the animation and observe how the edits to the camera path affected the animation. After the animation plays, reset the animation by stopping the animation and then clicking the Go to Start command.



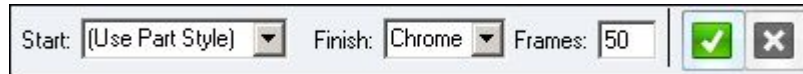
### Step 3

Add an appearance event to the time line.

- ▶ Click the Appearance command



- ▶ Select *a\_case.par* and *a\_backplate.par* as the parts for the appearance change, and then click Accept.
- ▶ Set the Start style to Use Part Style and the Finish style to Chrome. Set the Frames duration to 50, and then click Accept. Click Finish.



- ▶ Find the appearance and play the animation from the beginning through the first 50 frames. Observe the change to chrome for the parts selected.
- ▶ Play the animation and observe how the selected parts transition to chrome. After the animation plays, reset the animation by stopping the animation and then clicking the Go to Start command.



#### Step 4

Edit the event bars along the time line.

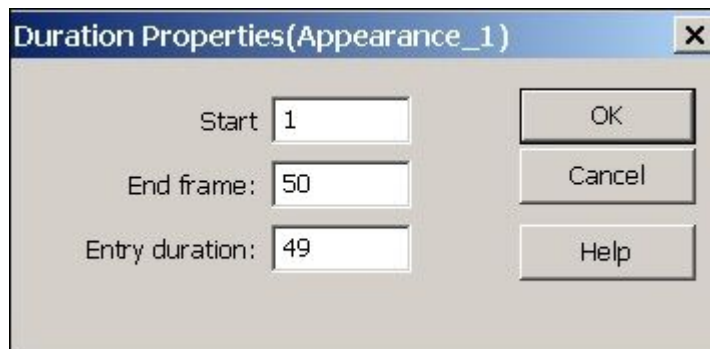
#### Note

An event group consists of events occurring either simultaneously or sequentially. The duration of an event group is defined by the extent of all the events within that group. Actions such as mirroring and copying of events must incorporate all the events making up a group.

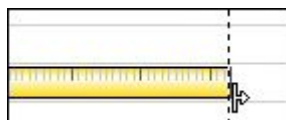
- ▶ Right-click the Appearance\_1 event bar, and then click Properties. Examine the values, and then click OK.

#### Note

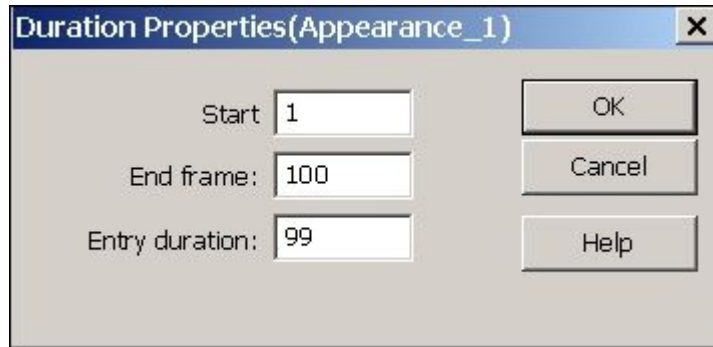
The values can be edited in the Duration Properties dialog box if desired.



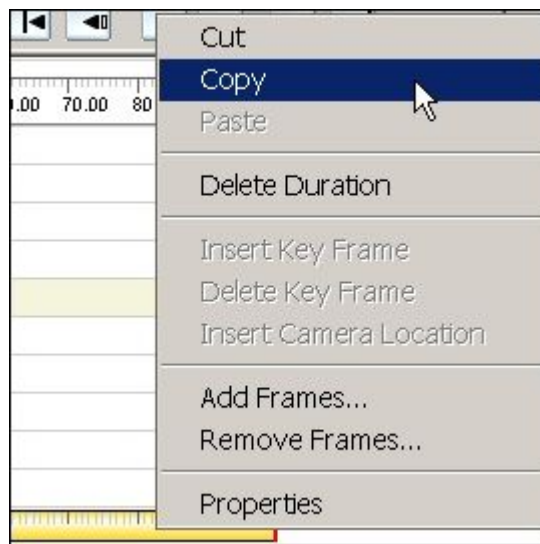
- ▶ Click and drag the right side of the Appearance\_1 event bar to frame 100.



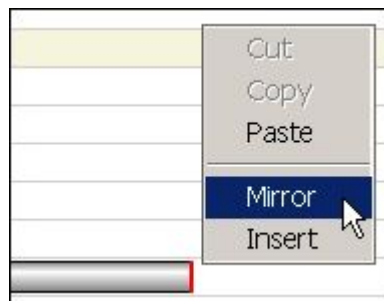
- ▶ Right-click the Appearance\_1 event bar, and then click Properties as you did previously. Notice the changed values.



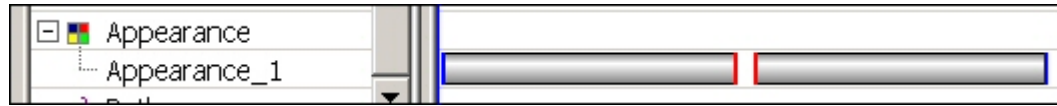
- ▶ Right-click the Appearance\_1 event bar and choose Copy.



- ▶ Right-click on the time line after the Appearance\_1 event and choose Mirror.



- ▶ Observe the new event bar on the Appearance\_1 event.



**Note**

Copying and mirroring an event reverses the effect caused by that event. By copying and mirroring the appearance event, as the animation plays, the selected parts transition into chrome and return to their original part styles. Mirroring is also valid for explosion events. Mirrored event bars can be moved, or shortened to make the transition happen faster, or lengthened to make the event happen slower.

- ▶ Play the animation to observe the changes you made. After stopping the animation, click Go to Start to reset the animation.
- ▶ Click Save Animation to save your edits.



*Step 5*

In the following steps, you will use what you learned in the previous steps to continue editing the animation. You will not be directed to edit specific events, however you will edit events you choose based on the general directions given.

- ▶ Lengthen an explode event by dragging the end of the event bar for that event.
- ▶ Copy and mirror the explosion event.
- ▶ Reposition an explosion event that has a sequence of cascading events. Shorten some of the events and lengthen others.
- ▶ Generate a new camera path based on saved views.

*Step 6*

Save an animation.

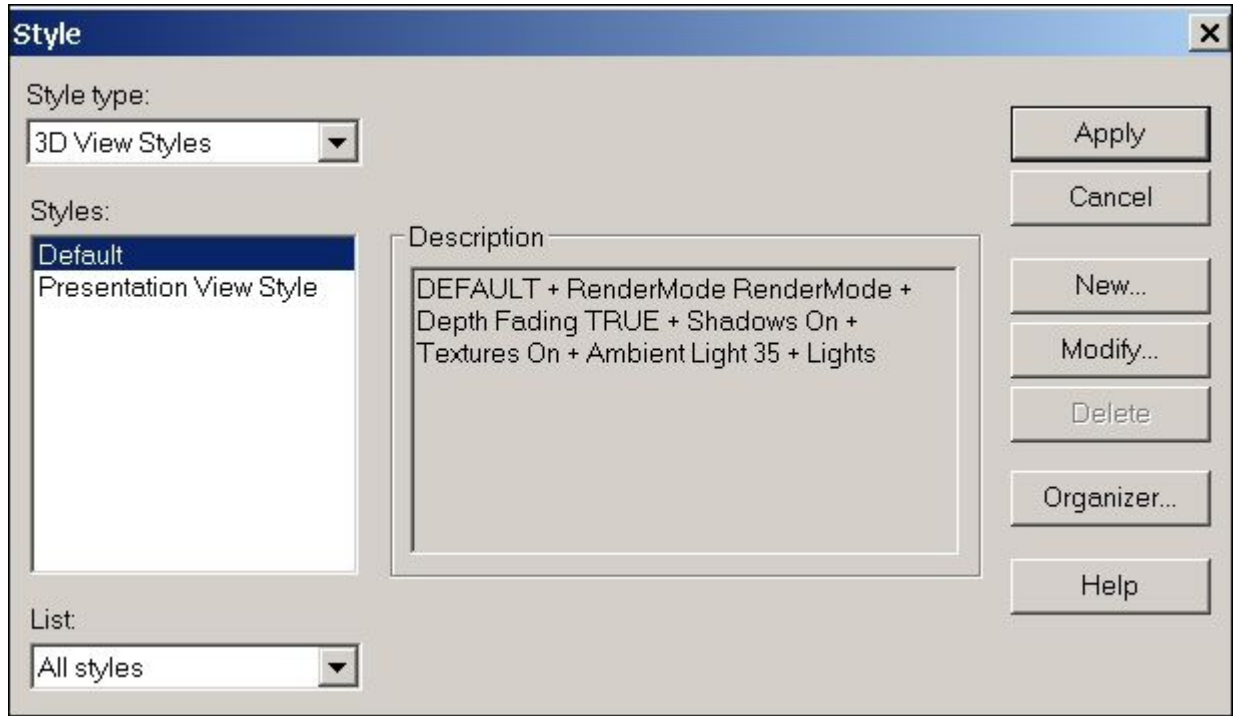
**Note**

Animations are saved as *.avi* formatted movies. There are many different players available from many different sources. A video codec is a device or software that enables video compression and or decompression for digital video. The list of codecs that are available to be used for creating an animation may differ from one computer to the next. To choose the codec which best works for the animations you are trying to create needs to be determined by experimentation.

- ▶ Animation views can be generated from previously generated 3D view styles. To see the current 3D view styles available, choose View tab@ Style group@ Styles



- ▶ Set the Style type to 3D View Styles.



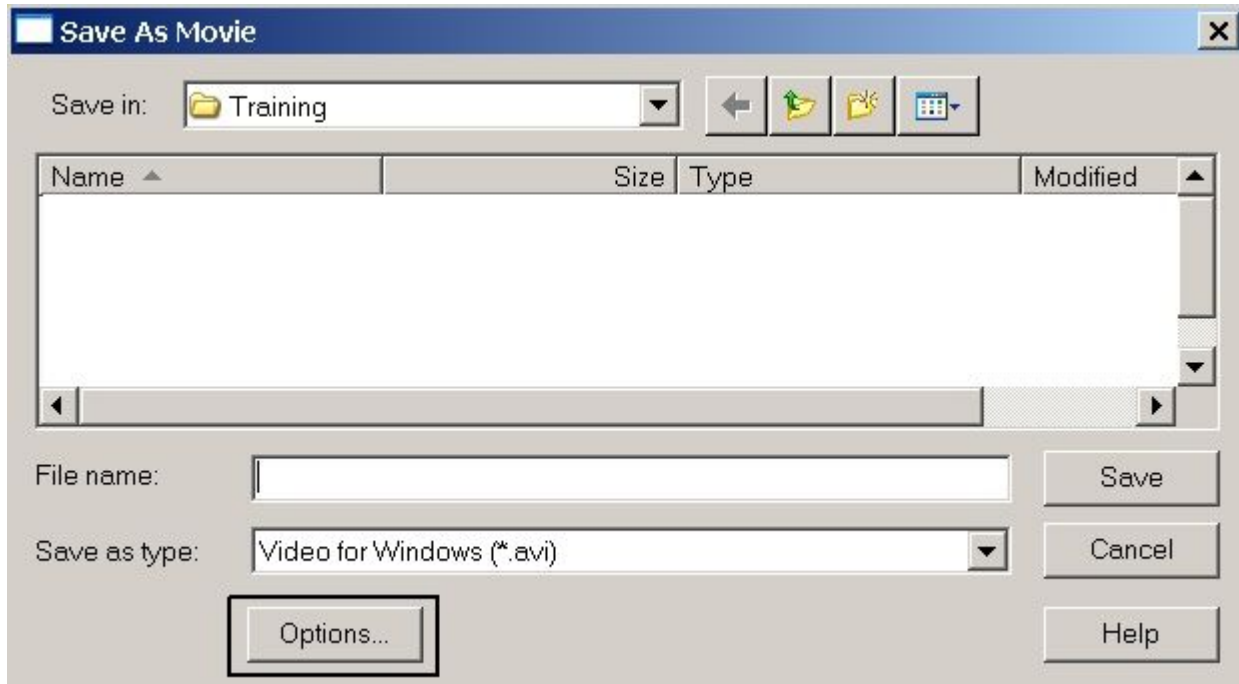
**Note**

Existing 3D view styles can be modified or new 3D view styles can be created to suit your needs.

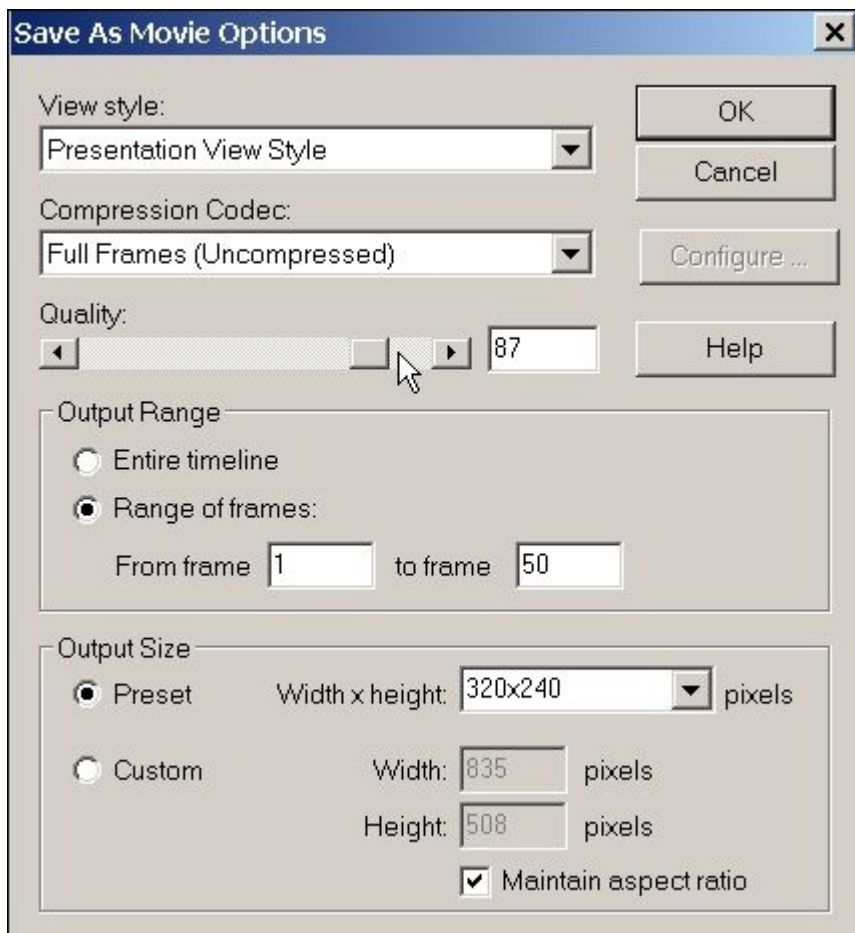
- ▶ Click Apply to close the Style dialog box.
- ▶ Reopen the Animation Editor in Explode-Render-Animate.
- ▶ Reset the animation to the beginning. Click the Save as Movie command



- ▶ In the Save As Movie dialog box, click Options.



- ▶ Examine the options for creating an animation.



**Note**

Creating an animation can take considerable system resources depending on the options chosen. It is good practice to run a small number of frames rather than the complete animation to preview the results. Once the settings are satisfactory, then you can run the complete animation.

Notice the view style selection set is the same as the 3D view styles shown in the previous steps.


Set the quality to 100, and click OK.

- ▶ Save an animation to a folder of your choice and give it a name of your choice.

**Note**

To create a rendered animation, render the view just prior to saving the animation and the animation will use the rendering settings. Rendered animations typically take more processing time to create.



- To exit the Animation Editor, click the Animation Editor command .
- This completes the activity. Click Close ERA to exit the Explode-Render-Animate application. Save the assembly.



**Note**

To create a rendered animation, render the view just prior to saving the animation and the animation will use the rendering settings. Rendered animations typically take more processing time to create.

This completes this activity. Save and exit the assembly.

*Activity summary*

In this activity, you learned how to create and edit an animation. The animation consisted of events defined by exploded views, camera movement, appearance events, and rendering. Using editing commands, the time line for each event was able to be edited to produce the desired effect. You created an *.avi* movie showing the animation.

**Lesson review**

Answer the following questions:

1. How is a camera path represented in an animation?
2. How is an appearance event used in an animation?
3. The frame scale can be displayed in what two types of units?
4. Can explosion or appearance event duration bars be copied and mirrored?
5. What are the different types of event duration bars that appear in the animation editor time line?
6. What command creates an *.avi* movie file?



**Answers**

1. How is a camera path represented in an animation?

A camera path is represented as a curve. Key points along the curve can be edited to change camera viewing angle and view previews of the camera display for that point in the time line.

2. How is an appearance event used in an animation?

Use appearance events to change part colors and visibility during an animation. For example, you can create fade in and fade out effects for parts in the animation.

3. The frame scale can be displayed in what two types of units?

The frame scale can be represented in frames or seconds.

4. Can explosion or appearance event duration bars be copied and mirrored?

Explosion and appearance event duration bars can be copied and mirrored.

5. What are the different types of event duration bars that appear in the animation editor time line?

Types of duration bars are:

- Explode events
- Paths
- Appearance
- Motors
- Camera
- What command creates an *.avi* movie file?

The save as movie command creates an *.avi* movie file.

**Lesson summary**

In this lesson, you learned how to create and edit an animation. The animation consisted of events defined by exploded views, camera movement, appearance events, and rendering. Using editing commands, the time line for each event was able to be edited to produce the desired effect. You created an *.avi* movie showing the animation.

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**Lesson**

# *17 XpresRoute (tubing)*

## Introduction

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
- **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
- **spse01670**—Designing in the context of an assembly
- **spse01675**—Assembly features
- **spse01680**—Inspecting assemblies
- **spse01685**—Alternate assemblies
- **spse01686**—Adjustable parts and assemblies

- **spse01690**—Virtual components in assemblies
- **spse01691**—Exploding assemblies
- **spse01692**—Rendering assemblies
- **spse01693**—Animating assemblies
- **spse01695**—XpresRoute (tubing)
- **spse01696**—Creating a Wire Harness with Harness Design
- **spse01424**—Working with Solid Edge Embedded Client

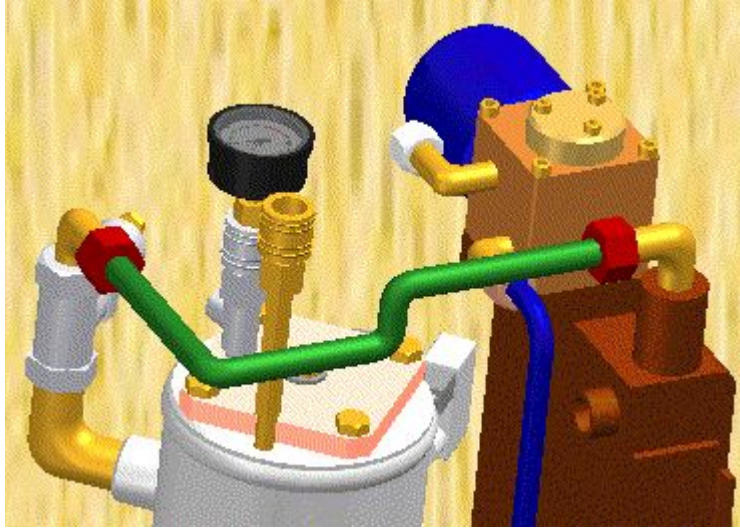
### **Start with the tutorials**

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.

## Tube design workflow

### Tube design overview

You can use XpresRoute to create path segments and tubes in an assembly. To access the commands for tubing, while in Assembly, choose Tools® Environs® XpresRoute .



Tube parts are designed in the context of an assembly, so you can directly model them within an assembly, using existing part and assembly geometry to ensure accurate fit and function. Tube parts are fully associative and update with the parts to which they are connected. Tube wire parts are directed parts. They conform to the path segment and the options you use to construct the part. When you make changes to the assembly that cause the path to change, the part will also change.

### Tube design workflow

#### 1. Create a path

Use the PathXpres command to automatically create a 3D path for the tube.

To learn how, see [Create a tube path with PathXpres](#).

Use the Line Segment or Arc Segment command to manually draw the path for the tube.

To learn how, see [Create a tube path](#).

#### 2. Create the tube

Use the Tube command to assign pipe attributes and fittings to a path segment that defines the route the pipe should follow.

To learn how, see [Create a tube](#).

## Creating path segments

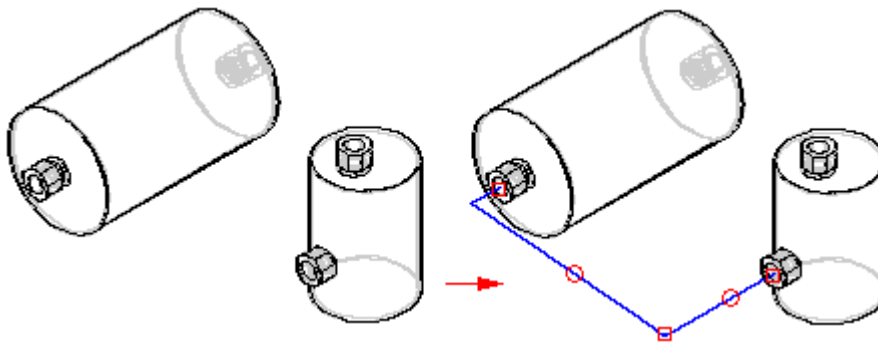
Tubes and Pipes are created along a path segment. You can use the PathXpres command in XpresRoute to create the path without manually drawing the individual lines of the path or you can use the Line Segment command or Arc Segment command to manually draw the path.

### Note

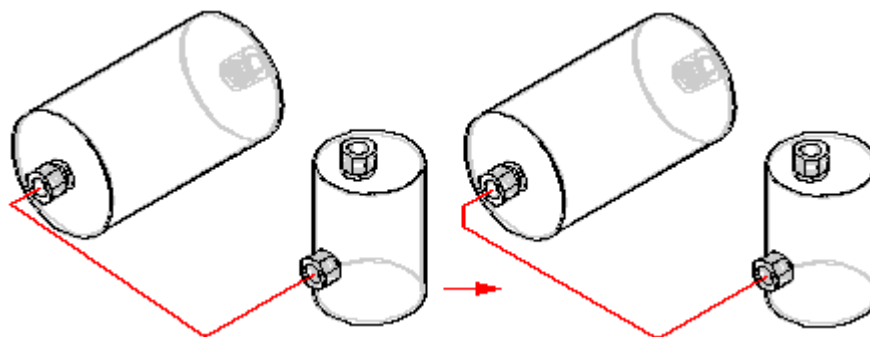
In addition to drawing the path segment, you can use the sketch geometry of an assembly layout to define the path.

## Creating the path with PathXpres

Use the PathXpres command to create a 3D path for a tube or pipe without having to manually draw the individual lines of the path. PathXpres generates a path between two points that is orthogonal to the default reference planes. These points must be circular or elliptical element, the endpoint of a segment, or the endpoint of a sketch element.



In cases where more than one way for the path exists, you can use the Next and Previous button on the PathXpres command bar to display alternative paths. The order of the paths goes from the simplest path, with the least number of segments, to the most complex path. The maximum number of segments in a path that PathXpres generates is five.



## Drawing a Path Manually

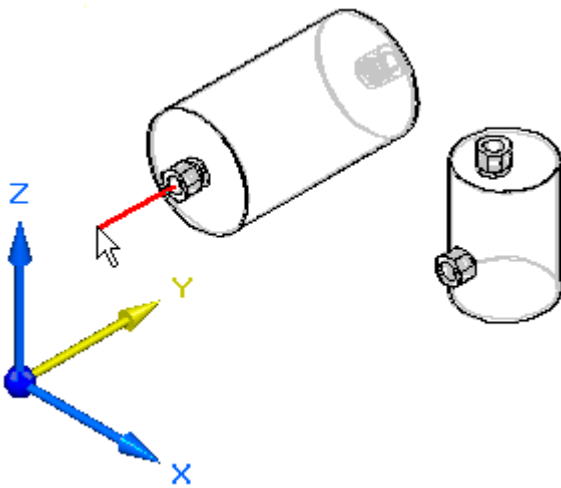
You can use the Line Segment command, Arc Segment command, or Path command in XpresRoute to manually draw the path for the tube or pipe. You can connect arc segments to line segments or other arc segments.

**Note**

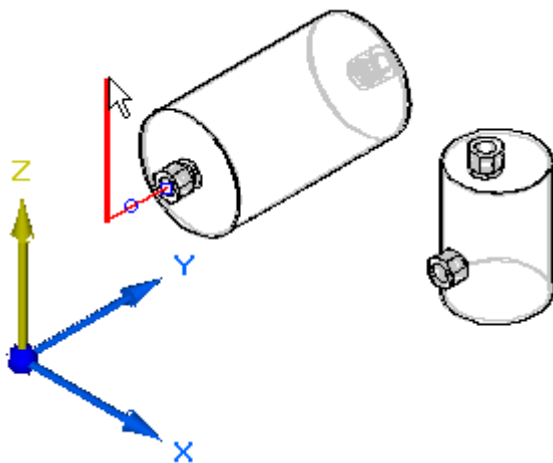
The part must be activated before you can attach a line segment to the part. If the part is not active, you can use the Activate Part button on the Line Segment command bar to activate the part.

**OrientXpres**

You use the **OrientXpres** tool to assist you in drawing lines and arcs in 3D space when drawing a path manually. As you draw the line or arc segments, use OrientXpres to lock the orientation of the element parallel to an axis or plane as you draw it. For example, after you define the start point for a line segment, you can use OrientXpres to lock the orientation to the y axis.

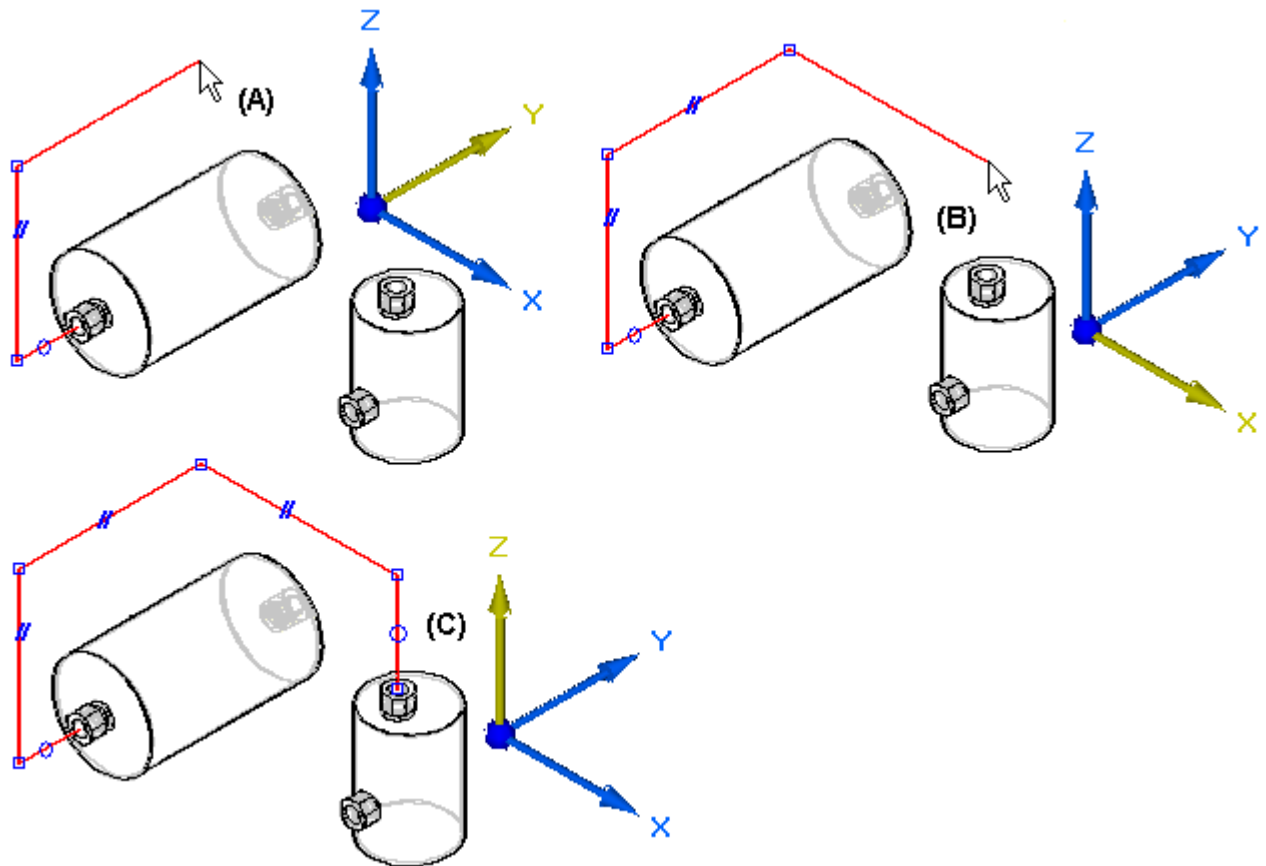


When you click to define the second point for the line, you can then use OrientXpres to lock the orientation to the z axis.





You can continue to lock the axis or plane to assist you in defining the path (A), (B), (C).

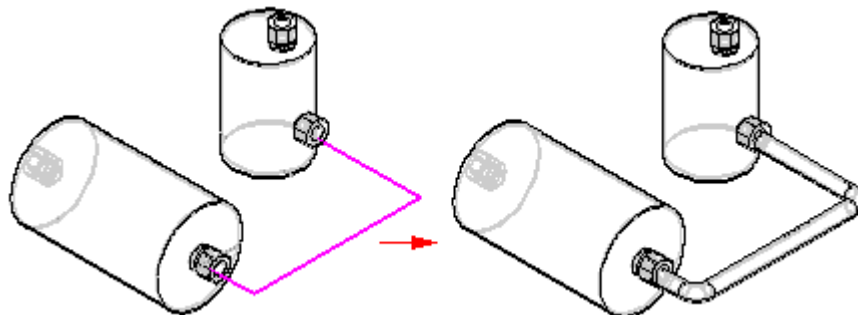


If you make an error when you draw a path segment, you can click the Undo button to undo the unwanted action. You can then continue drawing the path segment.

If you mistakenly undo the wrong action, you can click the Redo button to restore the action.

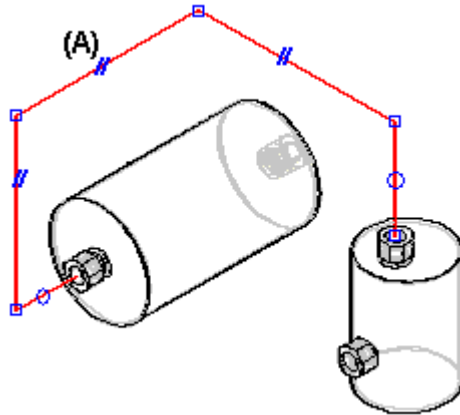
### Creating paths with assembly layouts

In addition to drawing the path, you can use sketch geometry that exists in an assembly layout as input for the path segment.



### Applying relationships and dimensions to path segments

As you add segments to the tube path, relationship handles are displayed on the segments (A) to indicate the relationships that you are creating. You can display or hide the relationship handles with the Relationship Handles command.

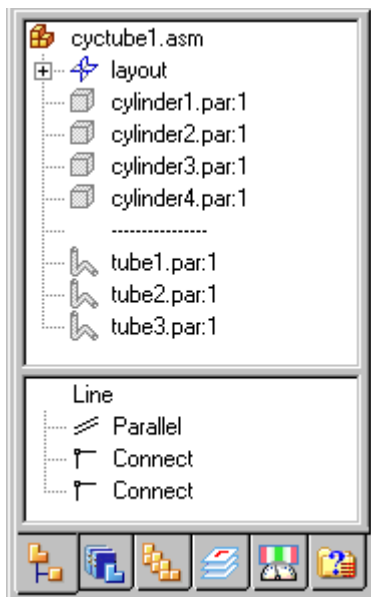


There are four types of geometric relationships for tube parts:

- Connect relationships
- Coaxial relationships
- Parallel relationships
- Tangent relationship

The PathFinder tab displays the tube path segment relationships.

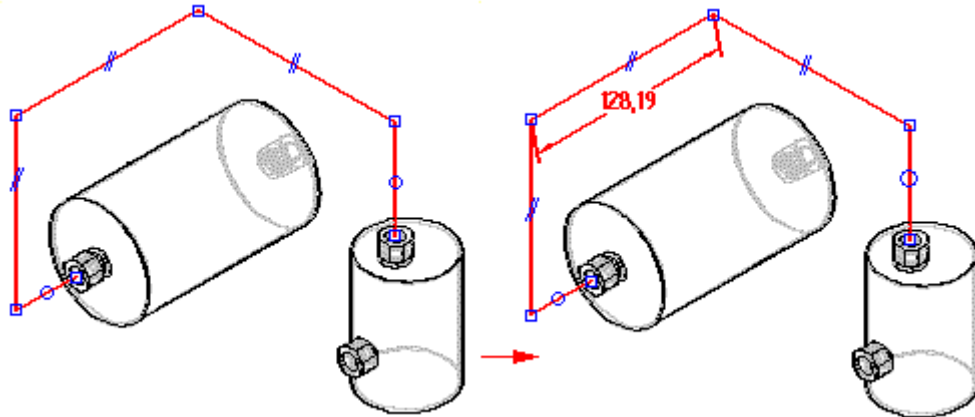
You can delete any relationship by deleting its handle in the graphic window or deleting the relationship in PathFinder.



Notice the dashed line separating cylinder4.par and tube1.par. This dashed line indicates that parts below the line are directed parts.

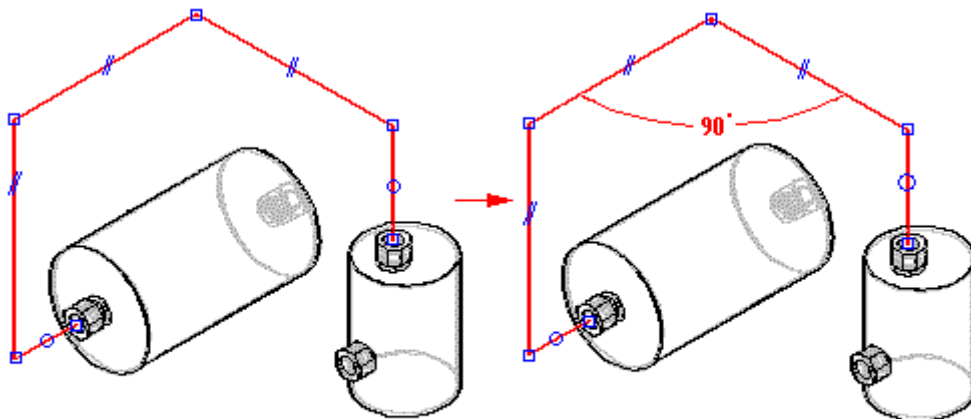
### Creating linear dimensions on paths

You can use the Axis Dimension command to create a dimension along a principal axis between a path segment and a reference element. The reference element can be another path segment, a principal plane, or a part edge.



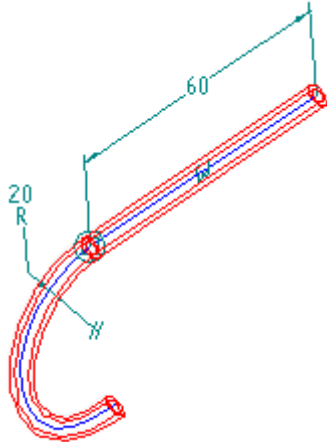
### Creating angular dimensions on paths

You can use the Angle command to place a dimension that measures the angle between two endpoint-connected tube path segments.



### Dimensioning path segments

You can use the SmartDimension command to dimension the length of a linear path segment or the radius of an arc path segment.



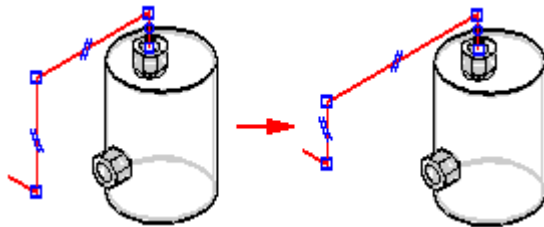
## Modifying path segments

You can modify path segments by:

- [Moving the path segment](#)
- [Splitting the path segment](#)
- [Creating a 3D curve from the path segment](#)

### Moving path segments

You can use the Move Segment command to move a path segment along the plane to which it is attached. With the command, you simply click the segment you want to move, drag it to a new position, then release the mouse button. This command maintains any existing relationships on the path segment and any of the adjacent tube segments.



Suppose you place a dimension on a segment and later decide you want to change the dimension value. All you have to do is click the dimension you want to change, and type the new value in the Dimension Value on the XpresRoute command bar.

### Splitting path segments

You can use the Split Segment command to split a path segment into two separate segments. With the command, you click the segment at the point you want to split.

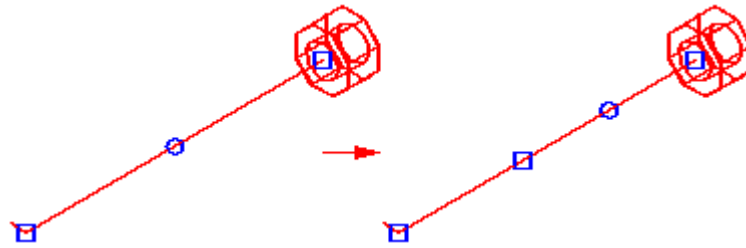


### Relationships and split segments

When you use the Split Segment command, the relationships on the split segment are maintained on the new segment. In addition, a connect relationship is applied to the split point of the new segments. The following cases describe how the Split Segment command works with relationships.

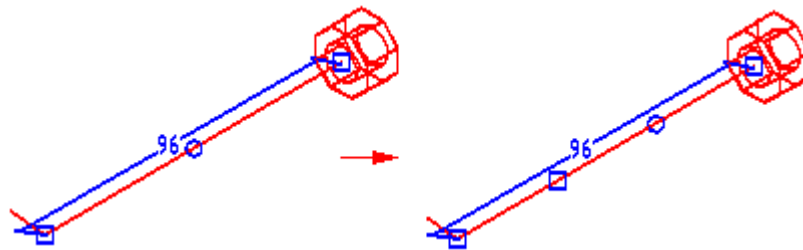
#### Segment coaxial relationship with a port

When you split a segment that is coaxially aligned with a port, the new segment attached to the port retains the coaxial relationship. The segment that is not directly attached to the port has no additional relationships other than the connect relationships.



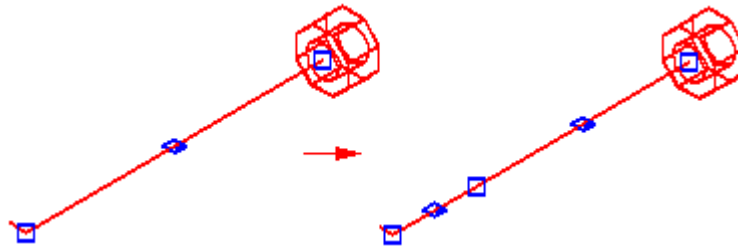
**Segment with an axis dimension**

When you split a segment containing an axis dimension relationship, the original axis dimension is maintained between the endpoints of the segment.



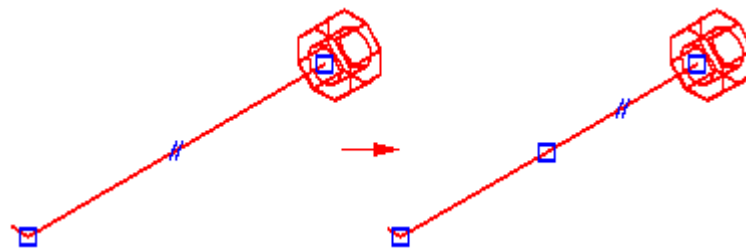
**Segment with a planar relationship**

When you split a segment containing a planar relationship, both new segments retain the planar relationship.



**Segment with a parallel relationship**

When you split a segment containing a parallel relationship, the new segment maintains the parallel relationship.

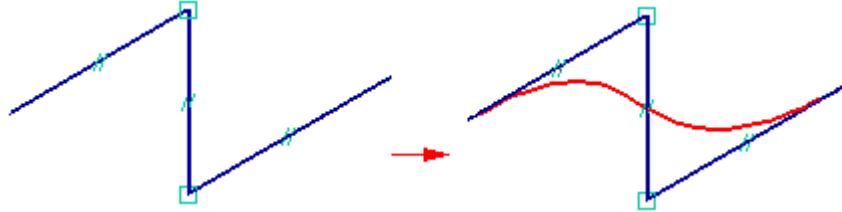


**Creating curve segments**

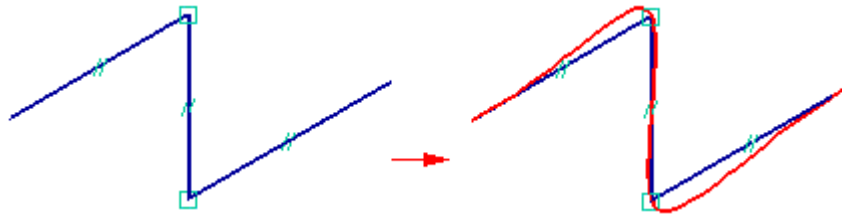
You can use the Curve Segment command to create a 3D curve based on a set of endpoint-connected path segments. The curve is always tangent to the first and last segments in the select set and it passes through the first and last points of the path.

There is an option on the Curve Segment command bar that allows you to define the points for the curve. You can specify that the curve pass through the

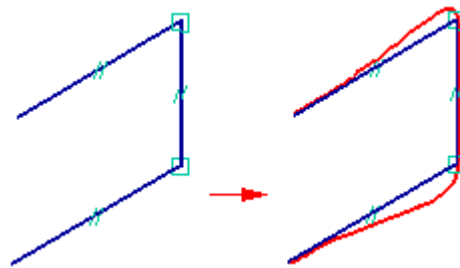
- midpoints of the line segments,



- the endpoints of the line segments,



- or all points of the input segments.

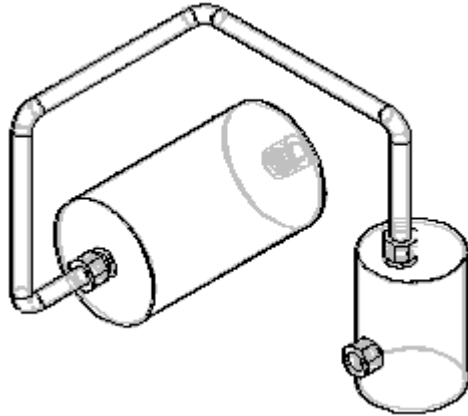


### Note

You can use the Hide Input Path and Show Input Path commands on the shortcut menu to control the display on the path used to create the curve segment.

## Creating the tube

Once you have drawn a tube path, use the Tube command to create a tube along the path segment. With the Tube command you can select a single segment or a chain of segments as the tube path. You can also define tube extents to both ends of the tube path.



When creating a tube part, you can use the Tube Options dialog box to define parameters such as material, outside diameter, bend radius, and wall thickness for the part. To access the Tube Options dialog box, click the Tube Options button on the Tube command bar.

## Defining the tube material

You can use the Material option on the Tube Options dialog box to specify the material for the tube part. The list of materials is populated from the Material table.

### Note

Prior to ST4, specifying the material simply applied an assembly-level style override to the tube part and did not apply physical properties such as density.

If a material is specified in the template you use to create the tube, it is the default material for the tube. If no material exists in the template, the default material is the material used when the last tube was created. If there was no last used material, the default material is Copper, if Copper exists in the Material Table. If Copper does not exist in the Material table, the default material is None.

### Note

If you click OK on the Tube Options dialog box with Material set to None, a warning dialog box indicates that the tube file has no density, which causes the physical property calculations for the assembly to be inaccurate. To correct the problem, you can select a material with a defined density or you can leave the material without a defined density.

## Editing the tube material

You can edit the tube material:

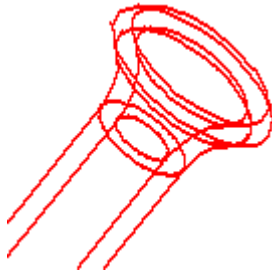
- With the Material Table command while in the tube part file.



- With the Tube Options dialog box while editing the tube definition in XpresRoute.
- With the Tube Properties command while editing the tube definition in XpresRoute.

### **Defining end treatments**

You can use the End Treatment Options dialog box to apply treatment types to the end of the tube.



The list of available end treatments include: None, Expand, Reduce, Close, and Flange. To display the End Treatment Options dialog box, on the Tube command bar, click the End Treatment Options button. You cannot apply end treatment to a curve segment.

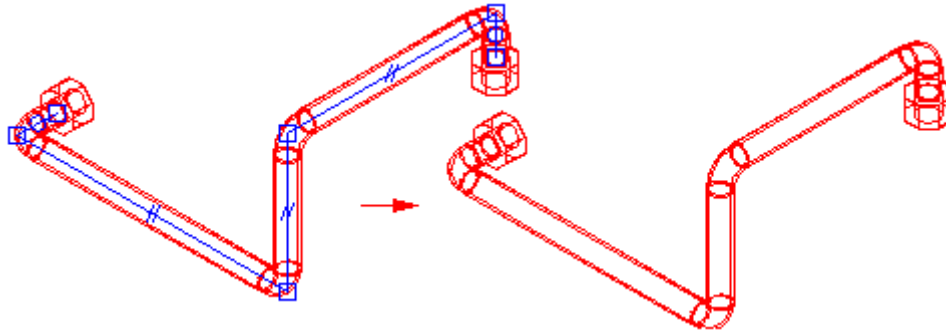
## **Opening and editing tube parts**

The Edit Definition button on the Select Tool command bar displays the XpresRoute command bar so you can edit the tube part. You can use the Tube Options dialog box to make changes to such parameters as the material, bend radius, and wall thickness. You can also make changes to path segments and edit the extents for the ends of the tube.

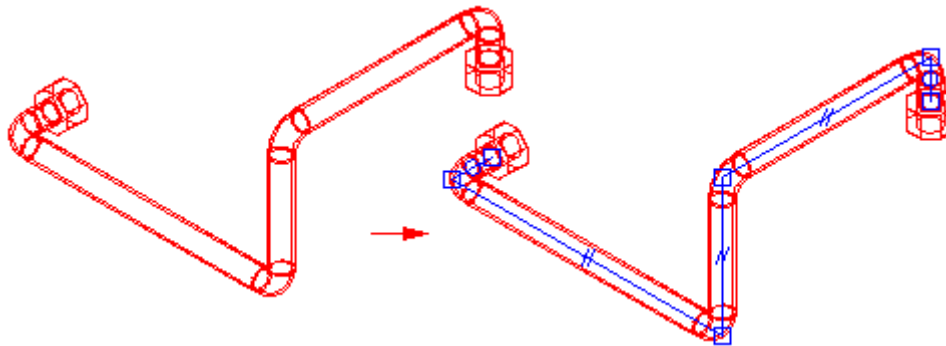
The Open command on the shortcut menu activates the tube part in the Part environment so you can add features to the tube. To save your changes, you can use the Close command to change the part and return to the XpresRoute environment.

## Managing path display

When working with tube parts, it is often useful to manage the path display. Solid Edge makes it easy to hide and display the part paths so you can work more efficiently. To hide a tube path, right-click the tube part containing the path and click Hide Path on the shortcut menu.



To display a hidden path, right-click the part containing the path and click Show Path on the shortcut menu.



## Displaying tube center lines

When you place tube parts in a draft document, you can display the tube center lines in the drawing view. To display the center lines, on the Annotation tab of the View Properties dialog box, select Show Centerlines.

## **Generating tube information**

You can use tube properties to extract information to create such reports as bend tables, tube reports, and parts lists.

## **Outputting bend information**

You can use the Bend Table command to create an ASCII text file that contains information about how to manufacture the tube. You can output the tubes as a select set or output them all at once. The information consists of columns of data such as feed length, rotation angle, bend radius, and bend angle.

## **Creating tube reports and parts list**

You can include tube properties defined on the Tube Options dialog box in reports or parts lists. When you create a tube file, these tube properties are automatically stored in the tube file and are exposed so that they can be included in reports and parts lists.

For tube parts created in version 12 and earlier, the tube properties are added to the tube file when the tube is recomputed. You can use the `WriteTubeFilePropertiesForReportsAndPartsList` method on the Tube object in a Visual Basic program to automatically populate the tube files with the tube properties.

## **Transferring tubes to another assembly**

You can use the Transfer command to transfer a tube to a new or existing assembly. When you transfer a tube, the system copies the path for the tube into the target assembly and creates an associative link between the tube and the copied path. The path in the target assembly is not associative to the path in the source assembly, so you can edit the new path without affecting the path in the source assembly.

All of the parameters of the tube are copied to the new assembly.

Any relationships applied to the path segments in the source assembly are reestablished in the target assembly. If the part that contains the port remains below the target assembly, the relationships are reestablished. If the part that contains the port is transferred to an assembly that is not below the target assembly no warning is displayed and the associativity of the port is broken. You can use the Hide Previous Level command to make sure the part containing the port is below the target assembly. To do this, in-place activate into a subassembly and select the Hide Previous Level command. If you can see the part in the graphics window, it is at the current level or below in the tree structure. If the part disappears when you select the Hide Previous Level command, the part is above the active assembly level. If you want to move the tube down the tree structure, make sure that the port part will remain at or below the target level. If the target assembly is below the current level of the port part, you should transfer the port part to the level of the target assembly before transferring the tube.

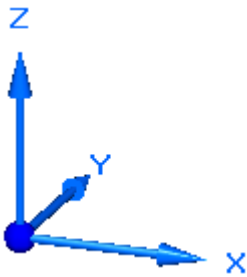
If the tube being transferred refers to sketch elements, a new sketch is created in the target assembly. The sketch will contain only the 2D elements referenced by the tube being transferred.

## OrientXpres tool

The OrientXpres tool is an interactive design aid for drawing lines, arcs, and curves in 3D space, and for editing the position of BlueDots in 3D space. OrientXpres appears automatically when creating or editing elements which require its capabilities. For example, OrientXpres appears when drawing line segments in the XpresRoute and Frame applications, and when editing BlueDots in the Part and Sheet Metal environments.

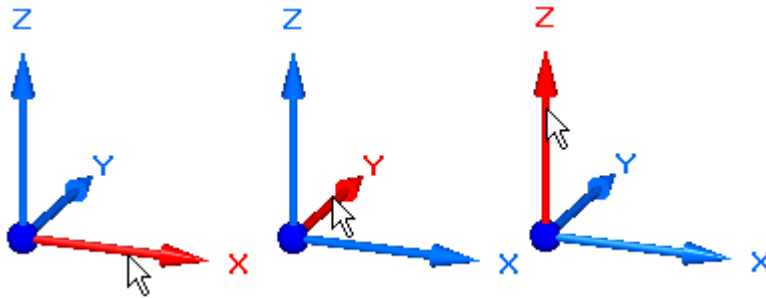
### Note

BlueDots are only available in the ordered modeling environment.

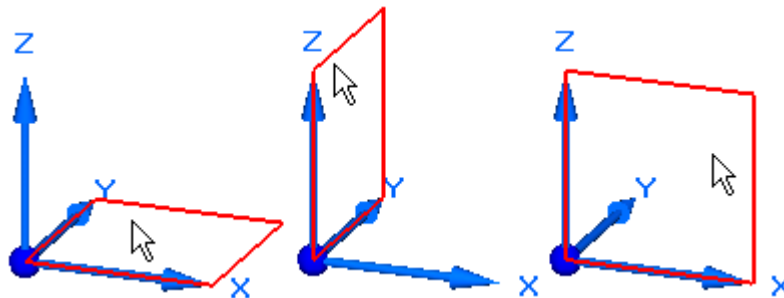


When working in 3D space, you often need to restrict the placement or movement of elements to be parallel to a particular axis or plane. The OrientXpres tool provides that capability. You can do the following using OrientXpres:

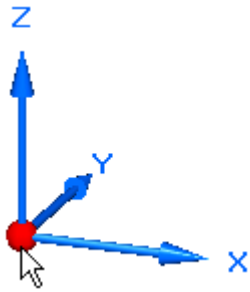
- To restrict movement parallel to an axis, select one of the three axes (X, Y, or Z). You can also cycle through the axes by typing Z key on the keyboard.



- To restrict movement parallel to a plane, select one of the three planes (XY, YZ, or XZ). You can also cycle through the planes by typing X key on the keyboard.



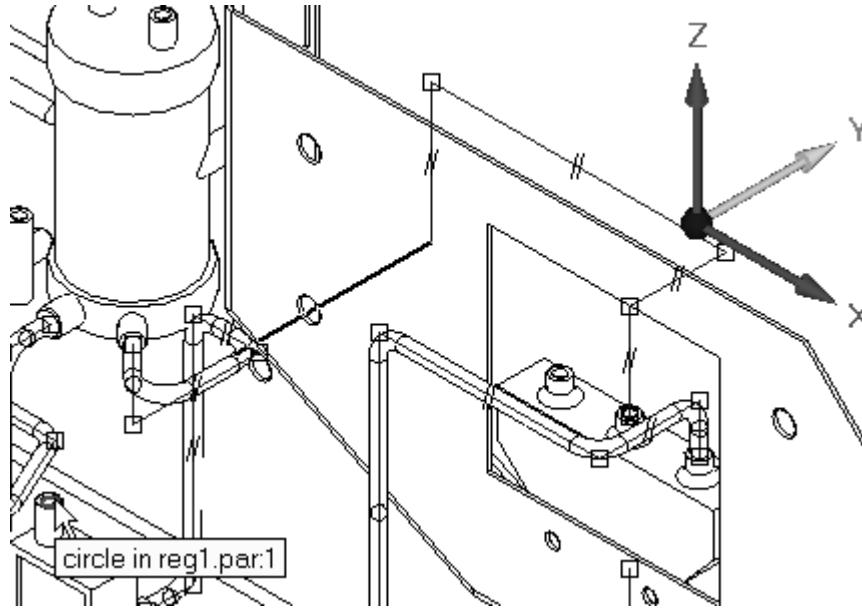
- To move the OrientXpres tool to a more convenient location, select the origin, and drag it to a new location.



- You can press C to clear any locks to planes or axes.

### Activity: Placing tubes in assembly with XpresRoute

Activity guides you through the process of using XpresRoute, and OrientXpres to route tubing components in an assembly, and then create a bend table of the tubes.





## **Activity: Placing tubes in assembly with XpresRoute**

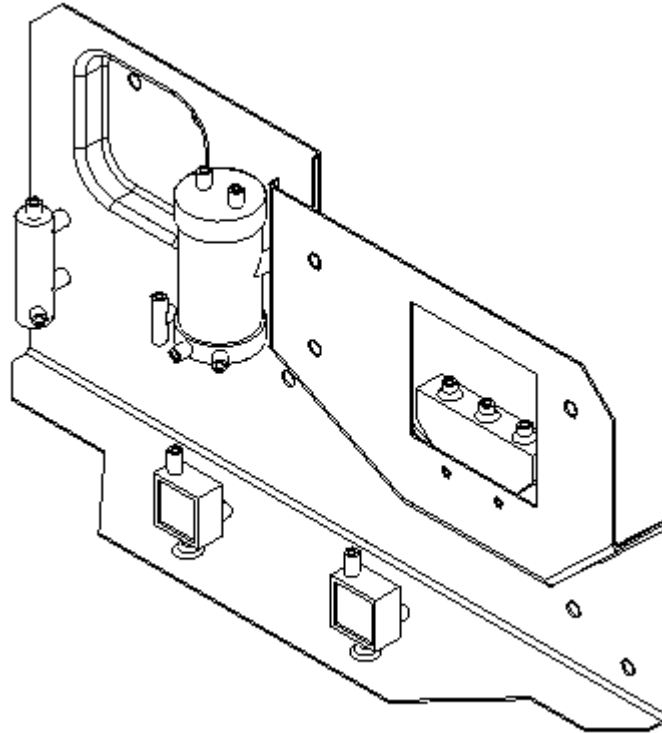
### **Overview**

When you complete this activity, you will be able to:

- Add tubes to the design using PathXpres.
- Add tubes by manually creating tube paths.
- Add end treatments to tubes.
- Modify tube paths and update the tube part.
- Edit the tube part after the tube is created.
- Output a Bend Table of the tube parts for manufacturing.

**Use PathXpres to automatically route the first tube path**

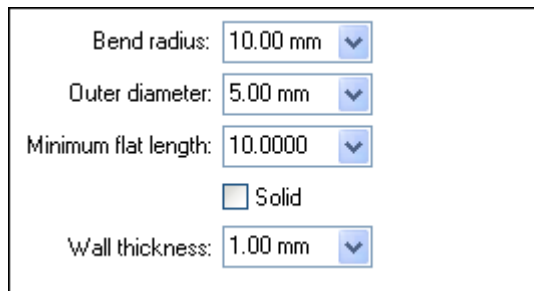
- ▶ Open *Xpres.asm* with all the parts active.



- ▶ Click the Tools tab. In the Evirons group, click XpresRoute.



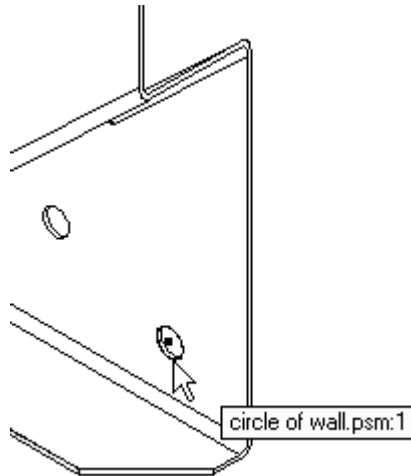
- ▶ Click the Application button. Click Solid Edge Options, and then click the Tube Properties tab. Set the tube properties values as shown, and click OK.



- ▶ Click the PathXpres command.



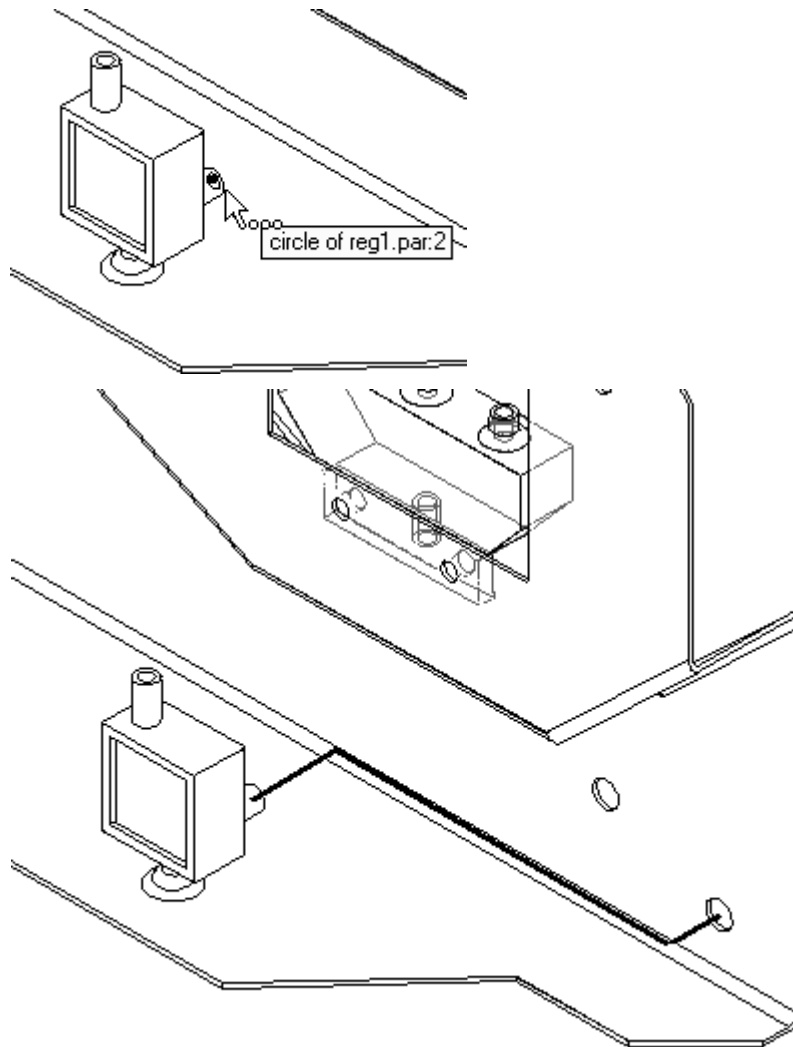
- For the first port, on the part *wall.psm*, click the front edge of the far right hole.



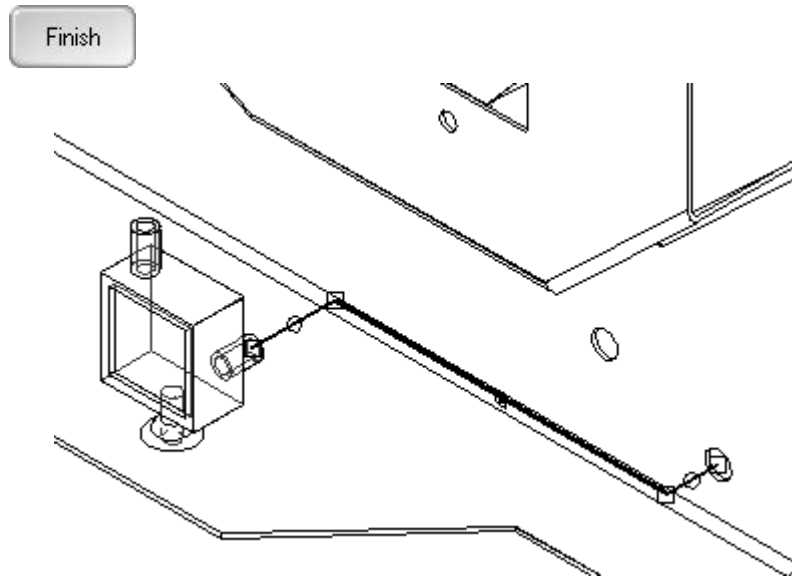
**Note**

The front edge is selected so the tube will extend toward the front of the assembly. If you select the rear edge of the hole, the tube path tries to project toward the rear of the assembly.

- ▶ For the second port, select the rear inlet on the blue part *reg1.par* as shown. After you click the rear port, the tube path should highlight.



- Accept the tube path by clicking Finish. The tube path changes to the profile color and displays the relationships applied along the path.



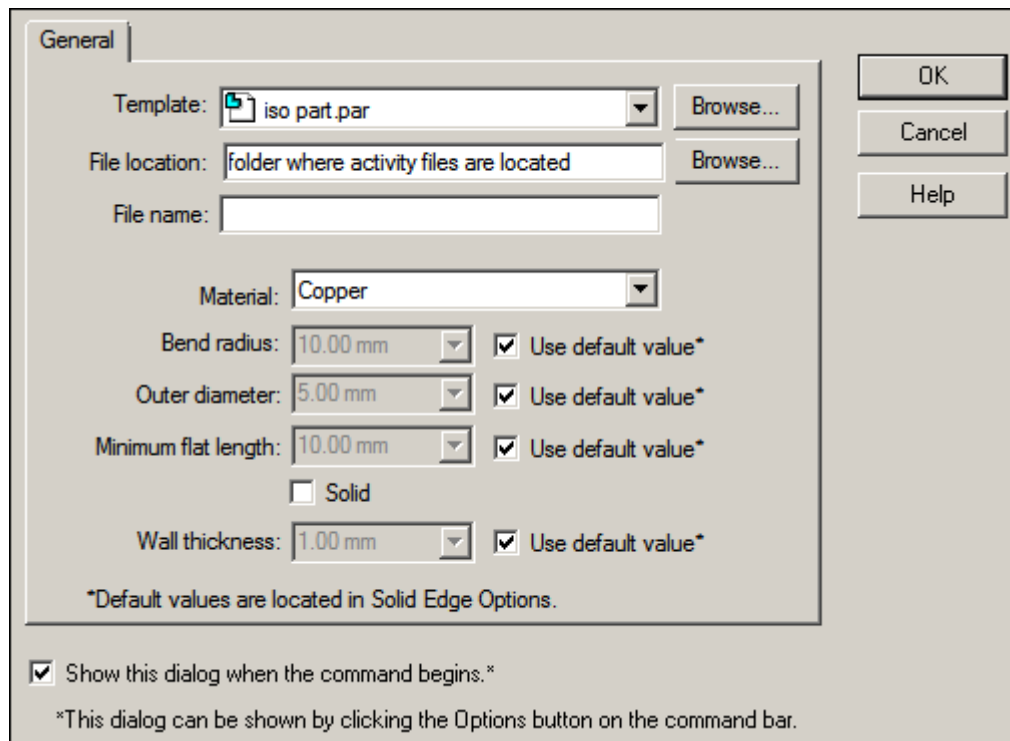
### Place the tube on the tube path

- On the Home tab, in the Tubing group, click the Tube button. If the Tube Options dialog is not displayed automatically, click the Tube Options button.



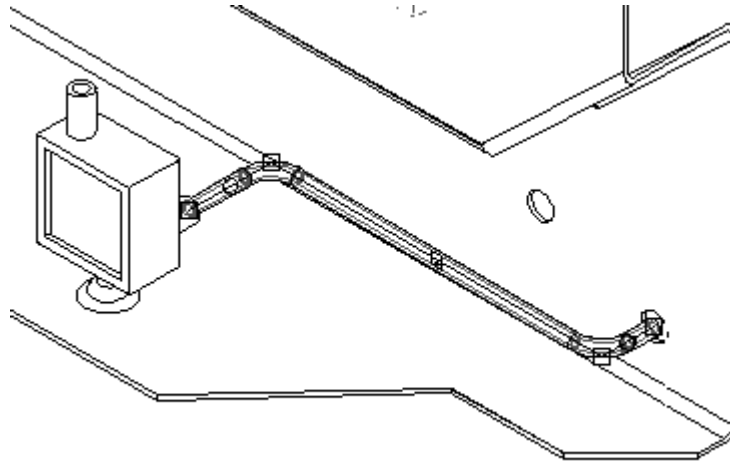
#### Note

The Tube Options dialog box should display as shown. Check the values against the values in the image below and make any necessary adjustments.



- To select the folder where the part file for this tube will be stored, click the browse button and select the folder location where the activity files are located.
- On the dialog box, clear the show this dialog when the command begins option. To see this box in the future, click the tube options button on the command bar.
- Type *tube001* as the new file name and click OK.
- Select the tube path just created.
- In the Name box, *tube001* is displayed. Click the Accept button to accept the path.

- ▶ The result should look like the following illustration.



- ▶ To accept this result, click Finish.

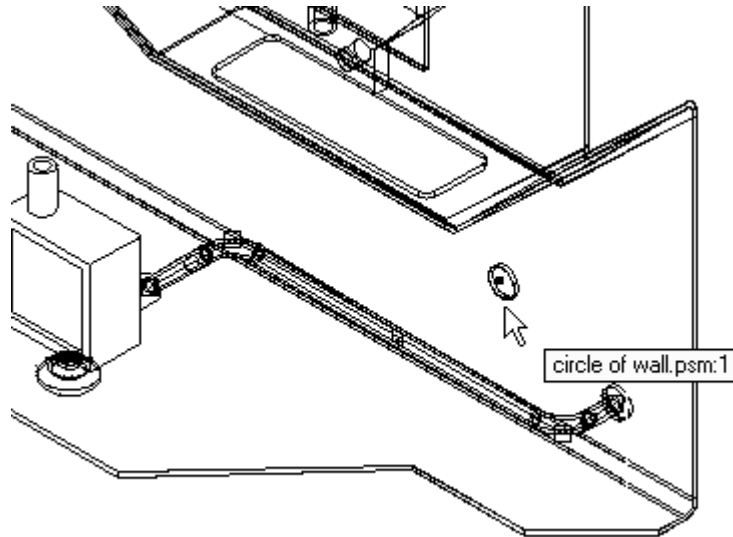


**Create a new path and a new tube**

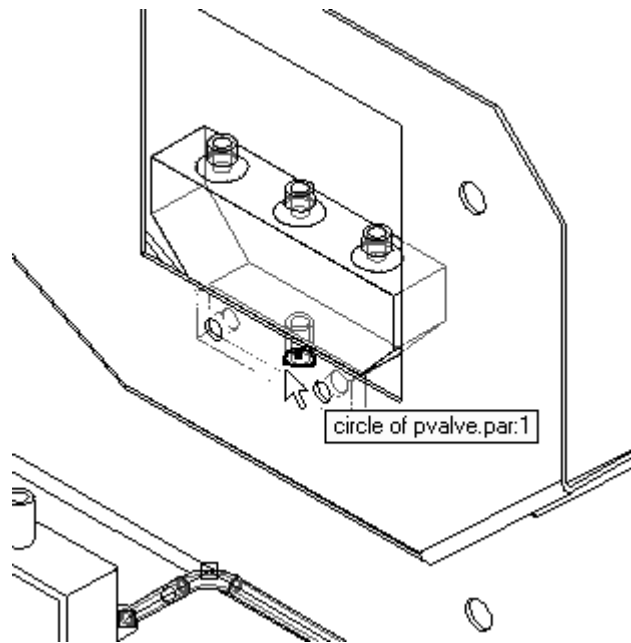
- ▶ To construct a second path, click PathXpres.



- ▶ For the first port of this new path, click the hole to the left of the first hole selected on *wall.psm*.

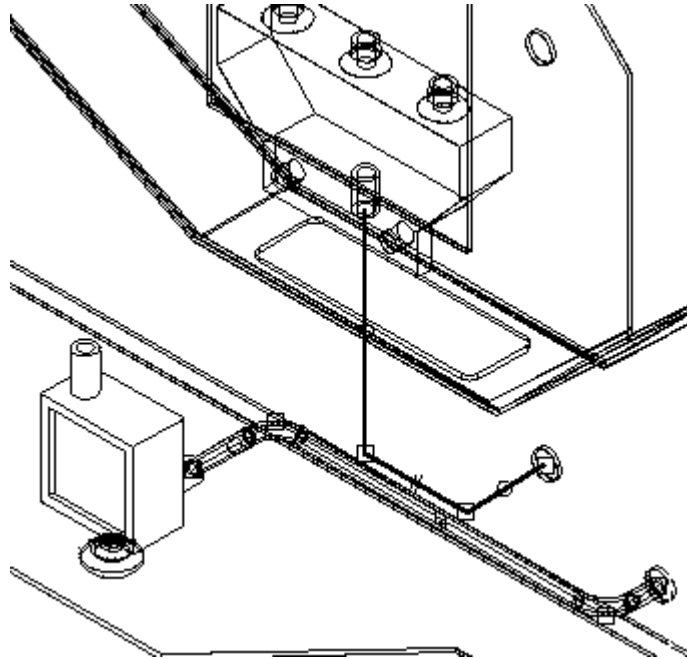


- ▶ For the second port, click the bottom port on the valve body *pvalve.par*.





- ▶ To accept this tube path, click Finish on the command bar.

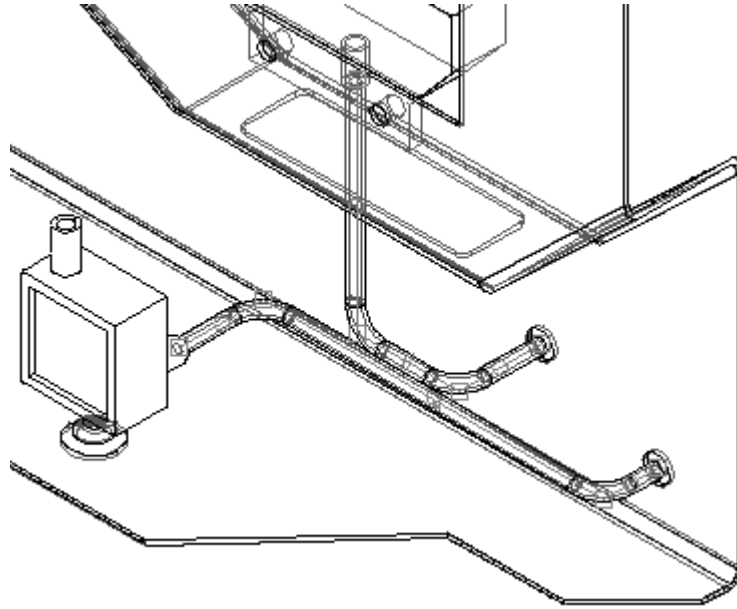


- ▶ To place a tube using this path, click the Tube command.



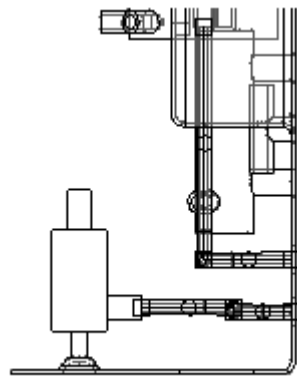
- ▶ Select the path just created.
- ▶ In the name box, type the name *tube002*.

- ▶ Click Preview.



**Note**

In the 3D window, it looks like the two tubes intersect each other. By checking the window with the right view of the assembly, you can see that the two tubes do not interfere with each other.



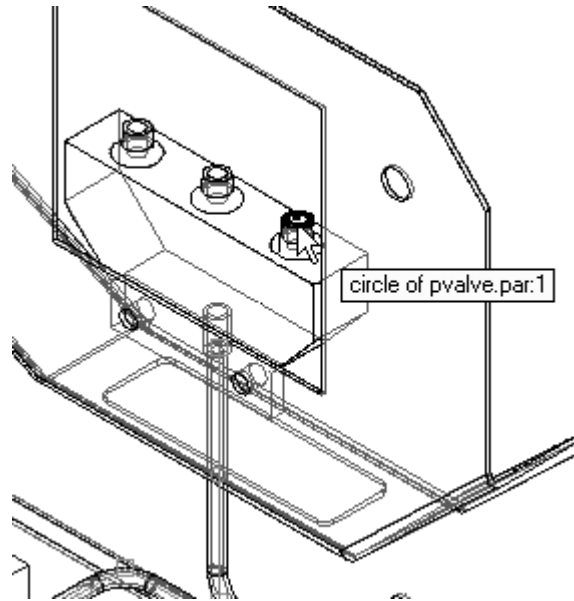
- ▶ Click Finish to complete the placement of the tube part.

**Create another new path and tube**

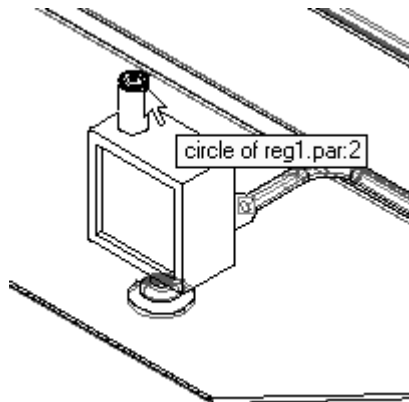
- ▶ Click PathXpres.



- ▶ Select the far right port on *pvalve.par* as shown.



- ▶ Select the top port on the blue *reg1.par* as shown.

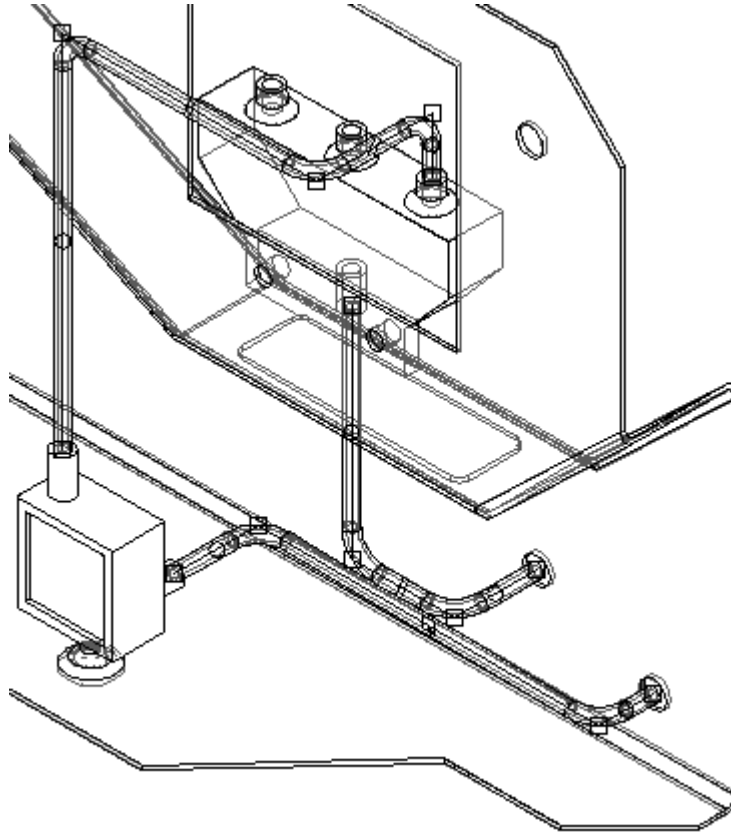




- ▶ To construct a tube part from this path, click the Tube command.



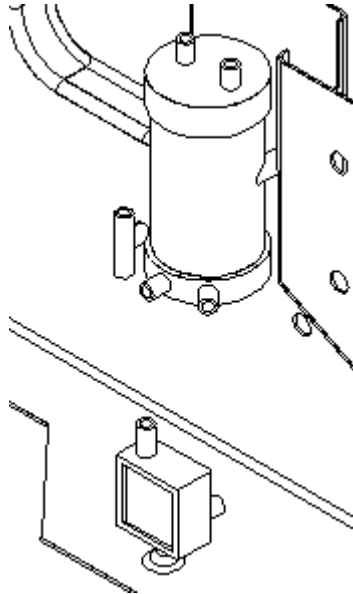
- ▶ Select the path just created, type tube003 for the tube part name, and click Preview to view the part. The result should look like the following illustration.



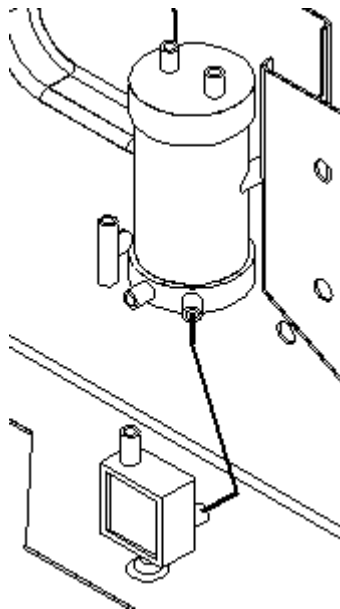
- ▶ Click Finish.

**Create a new path, then move a segment to maintain a clearance**

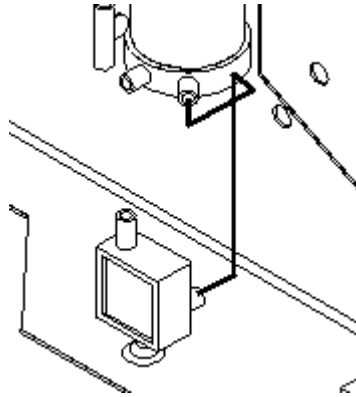
- ▶ The fourth path will be created in the middle of the assembly. Adjust the view as shown.



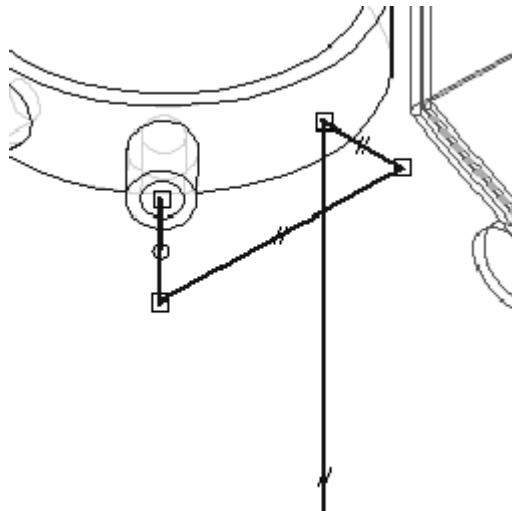
- ▶ Click PathXpres and construct a tube path from the bottom right port of the *tank2.par* to the rear port of the green *reg1.par*. Do not click finish.



- ▶ Cycle through the tube path solution options to get a tube path as shown by clicking the right blue arrow on the command bar.



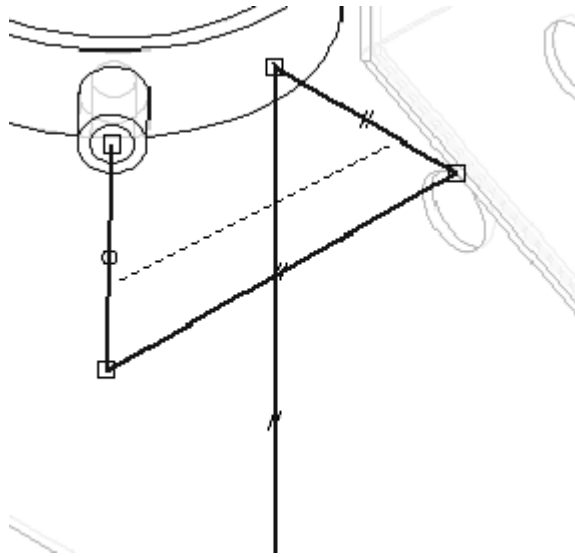
- ▶ Click Finish.
- ▶ Zoom the area on the top portion of the tube path as shown.



- ▶ On the Home tab, in the Segments group, click the Move Segment command.



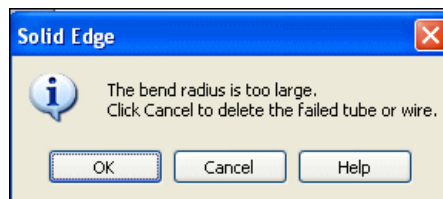
- ▶ Hold down the left mouse button over the second segment in the path, and drag the path segment to the right. Use additional windows to make sure you do not cause an interference with the other parts in the assembly.



- ▶ Using this modified path, construct a tube part named tube004. Click the Tube command, and select this path.

**Note**

Depending on the new segment position, the software may display a warning dialog box stating that one of the segments has violated either the minimum flat length or the bend radius as specified in the options settings. This is not an error message but a notification that this tube does not meet the requirements as specified in the Tube Options dialog box. Take the time to study whether or not the tube path is still valid, and modify the tube path accordingly. The tube part will not be placed into the assembly if you click OK.

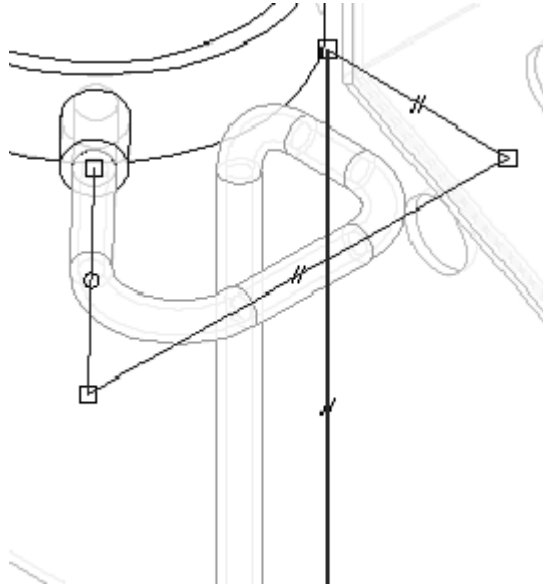


- ▶ If this warning is displayed, click OK to dismiss the dialog box.
- ▶ Click the Move Segment button.

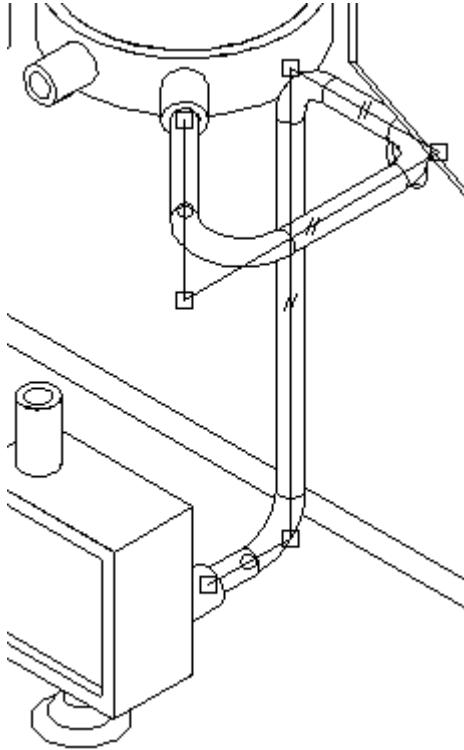
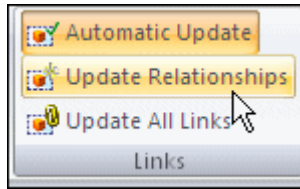




- ▶ Select the tall vertical segment, and drag it towards the rear of the assembly.



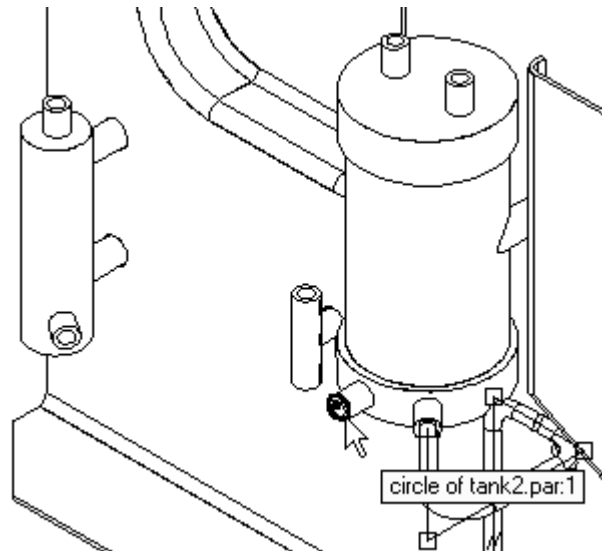
- ▶ To force the tube part to reflect the new tube path, click the Tools tab and then in the Links group, click Update Relationships. Notice that the tube recalculates and now resides on the newly modified path.



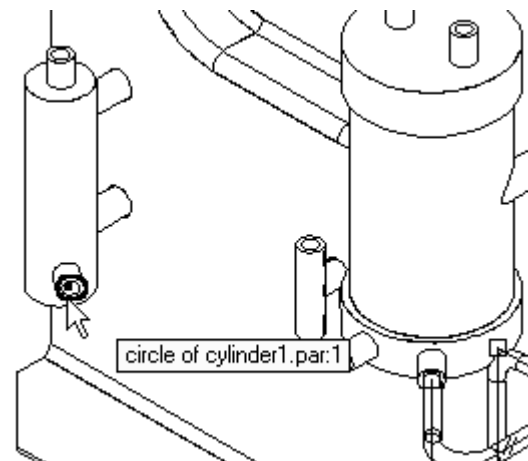
- ▶ To construct and modify another tube path, click PathXpres.



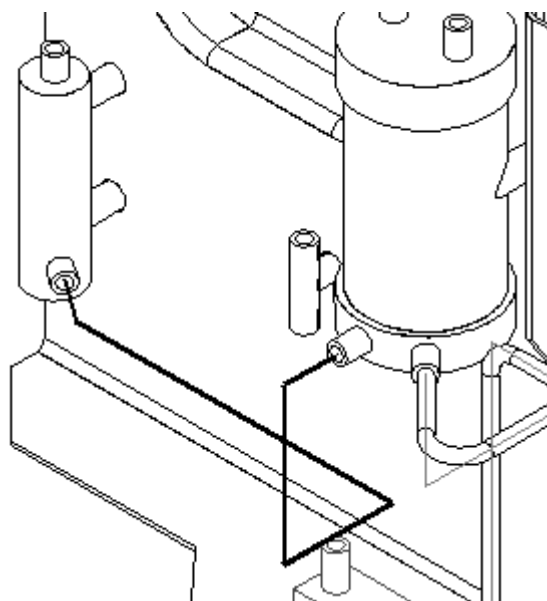
- ▶ For the first port, select the remaining bottom tube port on *tank2.par*.



- ▶ For the second port, select the bottom port on *cylinder1.par*.



- ▶ Cycle through the path options until a path is found that resembles the path as shown.



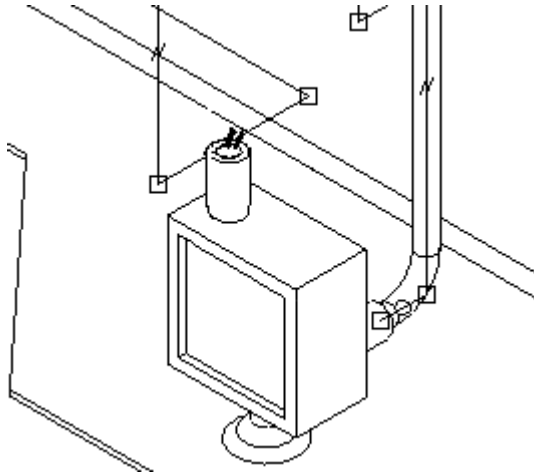
- ▶ Click Finish.

**Modify the tube path and add end treatments**

- ▶ Click the Select tool.



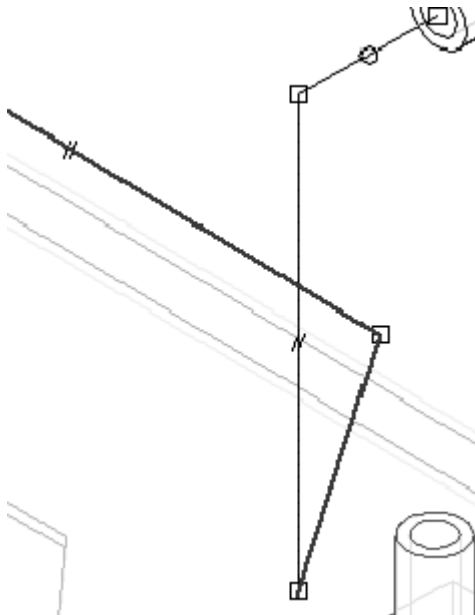
- ▶ As shown, select the parallel relationship on the line segment.



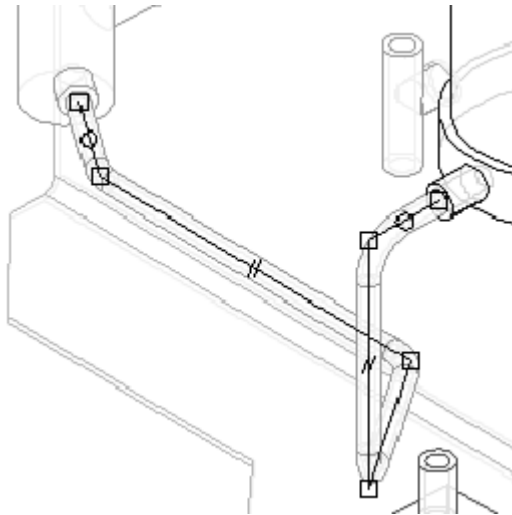
- ▶ Press the Delete key to delete it.
- ▶ Click the Move Segment button.



- ▶ Select the endpoint of this line segment closest to the rear of the part, and drag it to the left to make it longer.



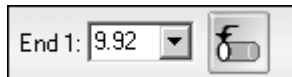
- Click the Tube button and construct a tube part named tube005 from this modified path.



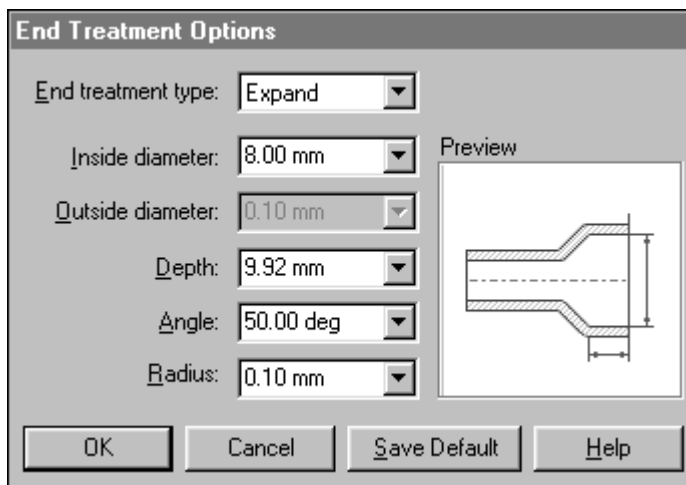
- Once *tube005* has been placed, click the Select tool and select *tube005.par*. Double-click *tube005.par* in PathFinder to edit the definition, and then edit the tube by clicking the End Treatment step.



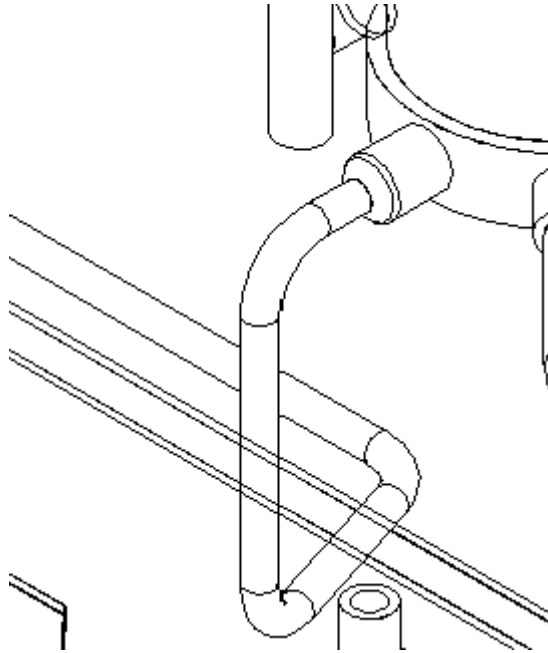
- Increase the extent of End 1 by typing 9.92 mm.



- Click End Treatment Options for end 1, and edit as shown in the illustration. Then click OK on the End Treatment Options dialog box.

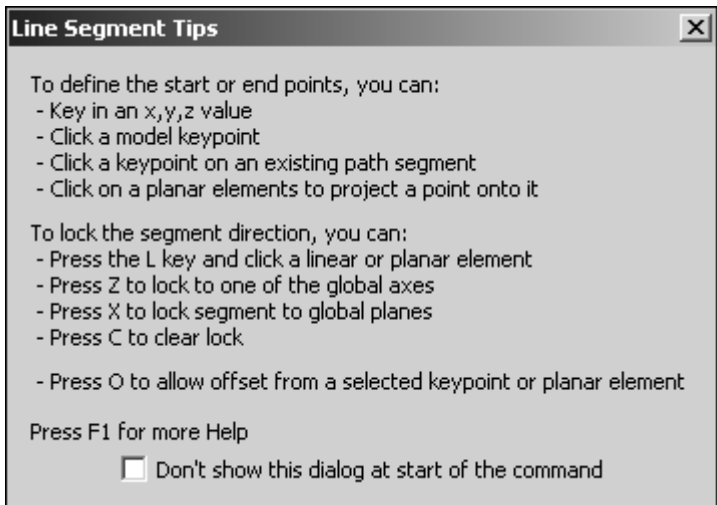


- ▶ Click Preview, and then click Finish. The end of tube005 is shown.

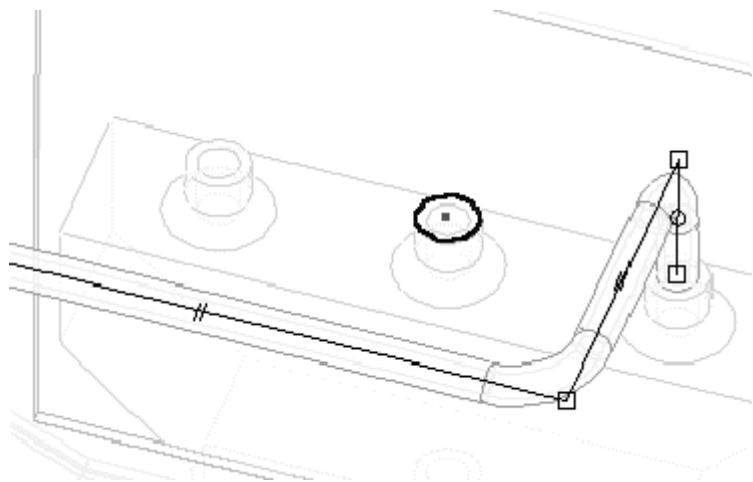


**Use OrientXpres to manually route a tube path and place a tube**

- ▶ On the Home tab, in the Segments group, click the Line Segment button. You may see the Line Segment Tips box shown.

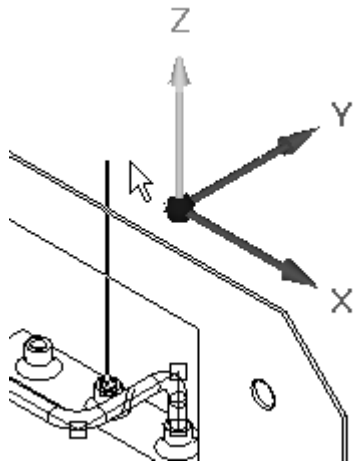


- ▶ As the starting point for the new path, select the middle port on *pvalve.par*. Adjust the view angle to get a better view.





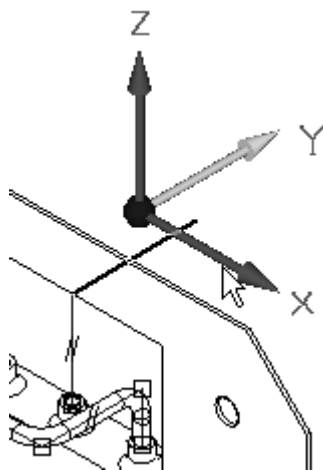
- ▶ To lock this line segment to this axis, select the vertical axis on OrientXpres.



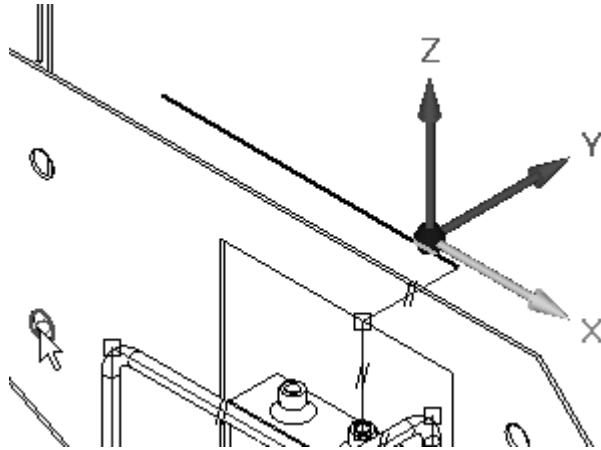
**Note**

To move the OrientXpres triad to another screen location, click and grab the triad by the origin.

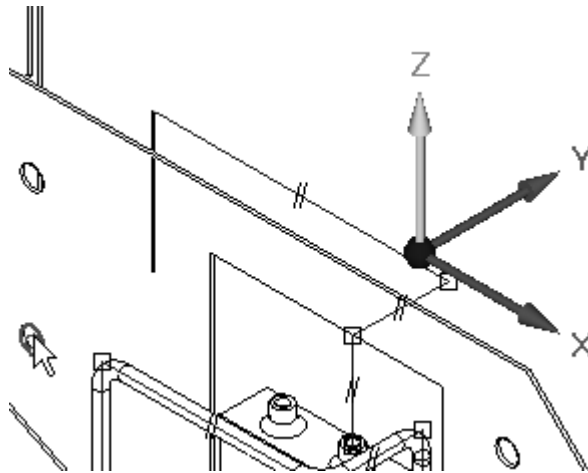
- ▶ Extend this vertical line 40 mm and then click.
- ▶ Click the axis as shown, and extend the line 40 mm toward the rear of the part and then click.



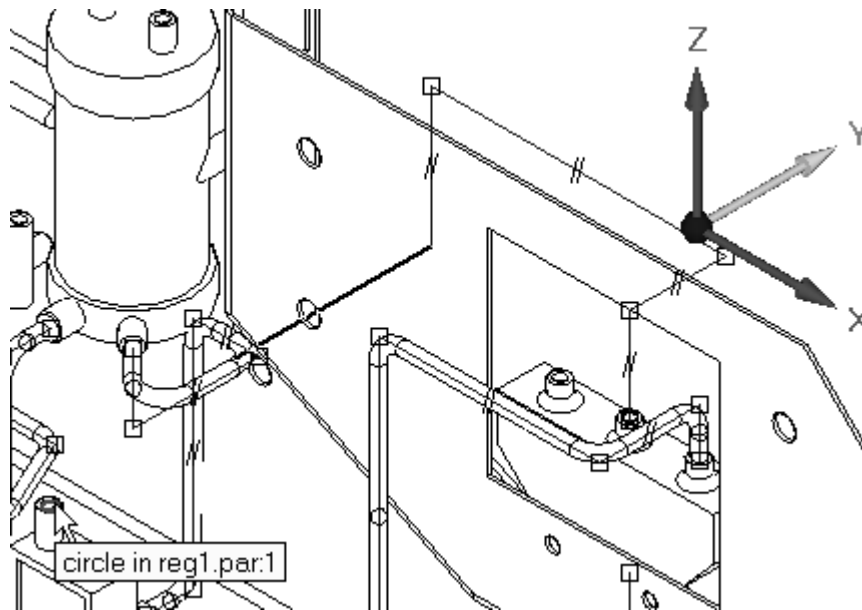
- ▶ Extend the next line segment the distance to the center of the hole in *wall.par*. Do this by selecting the axis shown and then positioning the cursor over the hole feature. When the center of the hole feature highlights, left-click. The line will extend to that distance horizontally.



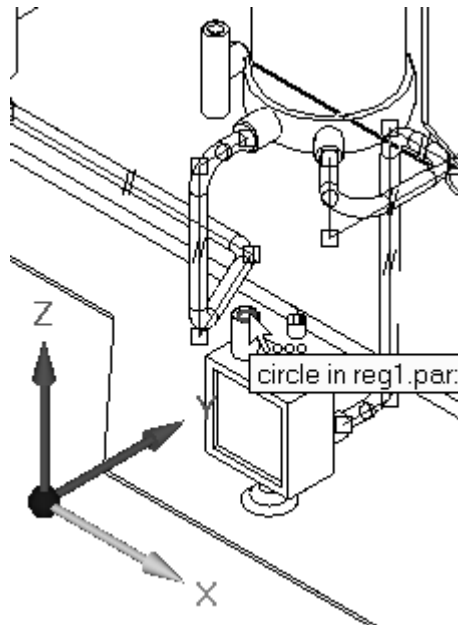
- ▶ Select the axis and extend the line down to the hole's center as in the previous step. Highlight the center, and accept this as the distance.



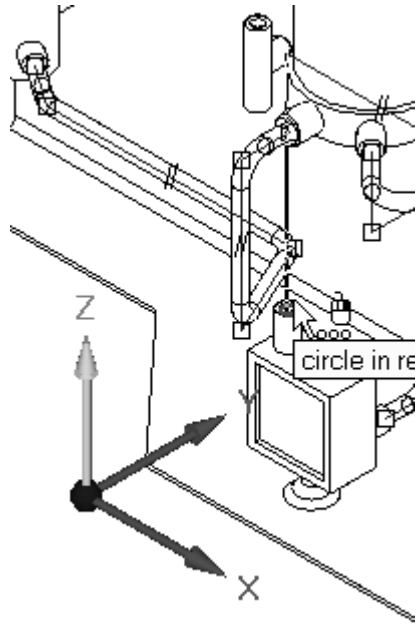
- ▶ Select the axis shown in the illustration, and extend the line to the center of the top port on green *reg1.par*.



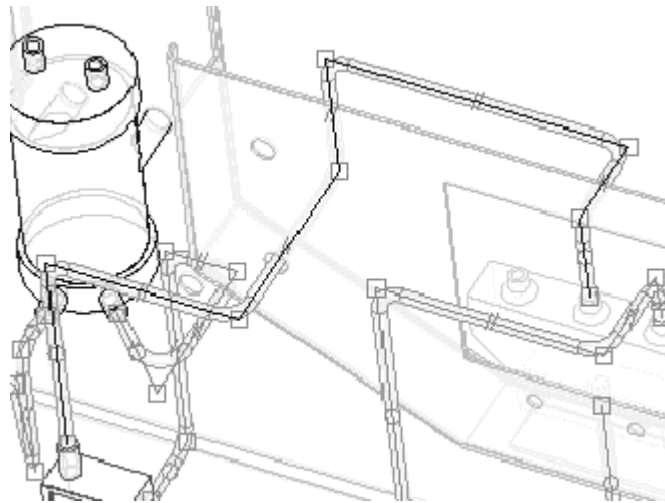
- ▶ Select the axis as shown, and extend the line to the port center.



- ▶ Select the axis as shown, and extend the line to the port center.



- ▶ To end the line segment command, right-click. The path is now complete.
- ▶ From this path, construct a tube part named *tube006*.

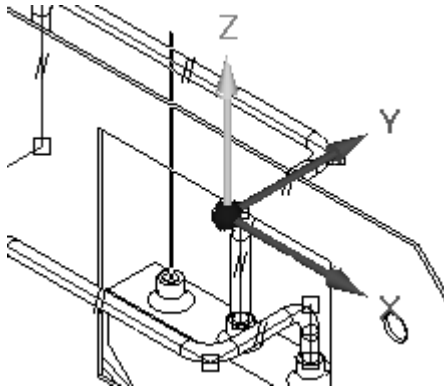


### Place a tube path manually

- ▶ To construct a second path manually, on the XpresRoute toolbar click the line segment button.



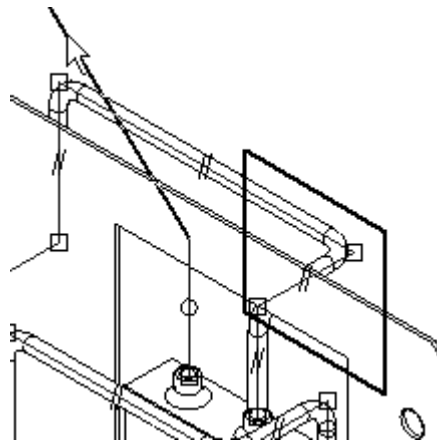
- ▶ Select the remaining port on *pvalve.par* as the starting port. Lock the OrientXpres into the vertical axis as shown.



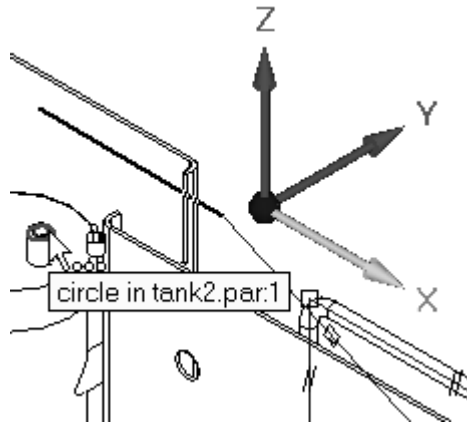
- ▶ Extend this line segment by 50 mm and click.
- ▶ Lock the next line segment to the plane as shown (click the plane with the mouse when it highlights), and extend the line 100 mm towards *tank2.par*. Use the other views to determine a reasonable height for the line segment. Right-click to restart the line command.

#### Note

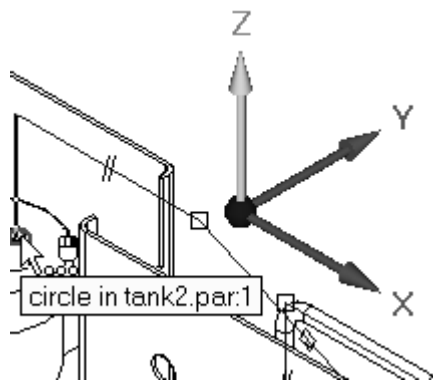
Locking to the plane allows construction of an angled tube segment.



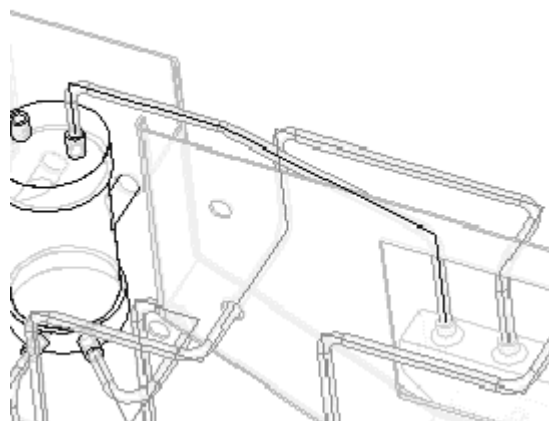
- ▶ Start the next line segment at the end of the previous line segment. Lock the next segment as shown, and extend it to one of the port centers on *tank2.par*.



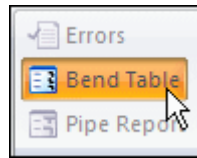
- ▶ Extend the final line segment to the port center by locking the line to the vertical axis.



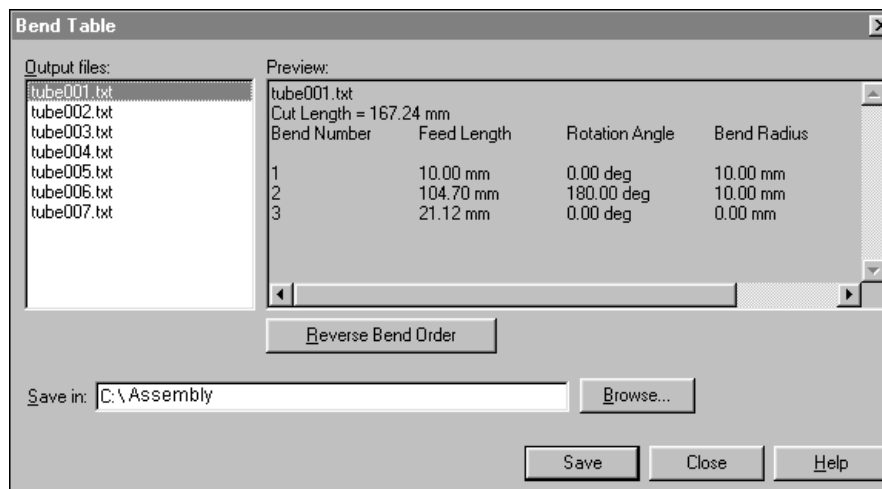
- ▶ Right-click to exit the line segment command.
- ▶ From this tube path, construct a tube part named *tube007*.



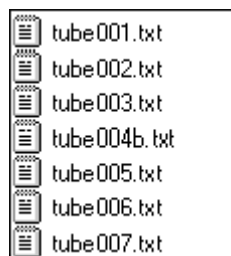
- ▶ On the Tools tab, in the Assistants group, click Bend Table.



- ▶ Click the Select All Tubes button and accept. All the tubes in the assembly should highlight.
- ▶ The bend information for the highlight tube part displays in the right portion of the dialog box. To view other tube part information, highlight the tube part file on the left portion of the dialog box.



- ▶ Click Save. Solid Edge will construct .txt files for each tube part in the folder specified in the Bend Table dialog box.



- ▶ Save and close this file. This completes the activity. However, if time permits, use the remaining ports to practice constructing additional tubes in the assembly.

### **Lesson review**

Answer the following questions:

1. What is the difference between PathXpres and OrientXpres?
2. What defines a tube?
3. If a path segment is moved, does the tube change to reflect the new position of the path segment?
4. After deleting a tube part, can the tube name be reused?
5. What information is contained in a bend table?



## **Answers**

1. What is the difference between PathXpres and OrientXpres?

PathXpres finds as many valid solutions for routing between two points. You can view each solution and choose the one you desire.

OrientXpres allows you to route the path manually. Tools exist in path express to lock the direction of the path segments to either planes or axes.

2. What defines a tube?

A tube is defined by a path which routes the tube, and a part file which contains the geometry of the tube. The part file shows up as a part in the assembly pathfinder.

3. If a path segment is moved, does the tube change to reflect the new position of the path segment?

Paths are associatively linked to the tube part. Changes in the path cause the tube to change. The Tools>update all relationships command will refresh the tube geometry.

4. After deleting a tube part, can the tube name be reused?

A tube name can be assigned to another path if the tube part file is first deleted. If not, when creating the new tube, a message showing the part file exists will display.

5. What information is contained in a bend table?

The bend table contains the information needed to manufacture the tube. Segment properties such as length, rotation, radius, and angle format is contained in the bend table. Bend tables can also be formatted for X,Y, and Z radius format.

## Lesson summary

In this lesson you learned how create tube paths both automatically and manually using OrientXpres. Once created, the tube paths were edited and tubes placed on the different paths. A bend table was created for the tube paths created.

- The XpresRoute provides the basic tools for creating tubes for fluid and air transfer. Other tube part features can be modeled by in-place activating to the part environment.
- XpresRoute provides editing, modification, and output tools for creating and manufacturing these types of tube parts.
- PathXpres is an automated method of generating a 3D tube path between two ports. Solid Edge calculates as many solutions as possible for connecting the two ports, where the solution solves a maximum of five (5) segments.
- OrientXpres activates automatically when you click the line segment command. This tool allows PathXpres to lock the line segment direction or planar orientation along a specific vector regardless of the mouse position on the screen. OrientXpres is activated by default, but its options are not active.
- The Tube command is used to construct a tube part file from a tube path. The tube path must exist before the tube part can be constructed. When the tube command is clicked, the command bar updates to show the options and steps required for tube part creation.
- The Bend Table command is accessed on the Tools tab. Bend table allows the extraction of tube information to an ASCII text file for use on the shop floor.