

Assembly features

Assembly features

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SOLID EDGE
VELOCITY SERIES

...with Synchronous Technology

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Lesson

1 *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
- **spse01541**—Explode-Render-Animate
- **spse01545**—Creating detailed drawings
- **spse01546**—Sheet metal design
- **spse01550**—Practicing your skills with projects
- **spse01560**—Modeling a Part Using Surfaces
- **spse01610**—Solid Edge frame design
- **spse01640**—Assembly patterning
- **spse01645**—Assembly systems libraries
- **spse01650**—Working with large assemblies
- **spse01655**—Revising assemblies
- **spse01660**—Assembly reports
- **spse01665**—Replacing parts in an assembly

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

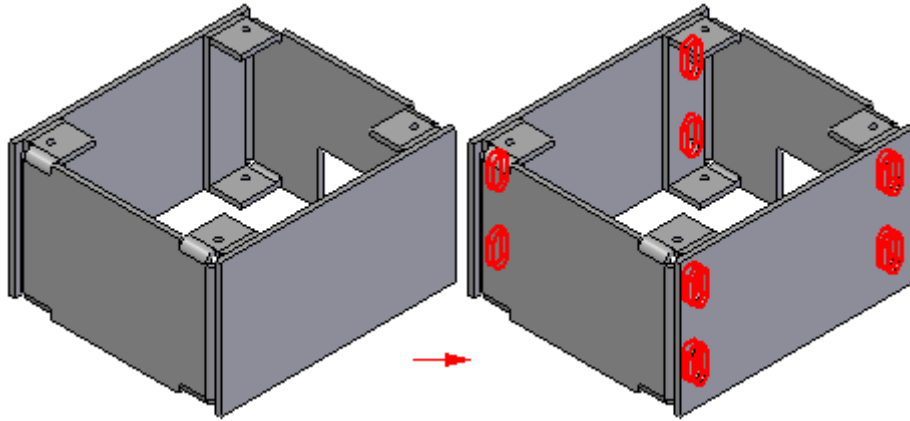
Start with the tutorials

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

Lesson

2 *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.

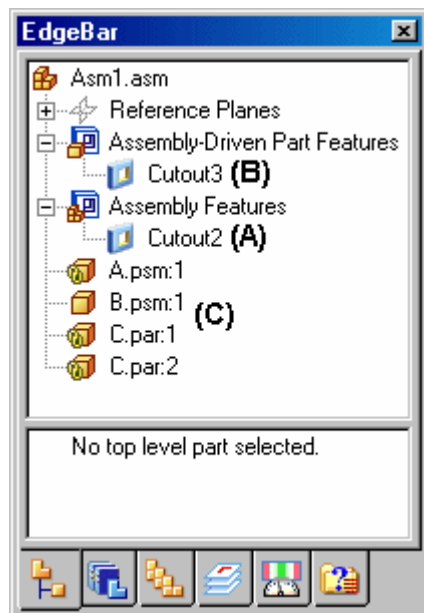
Selecting the Parts

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.

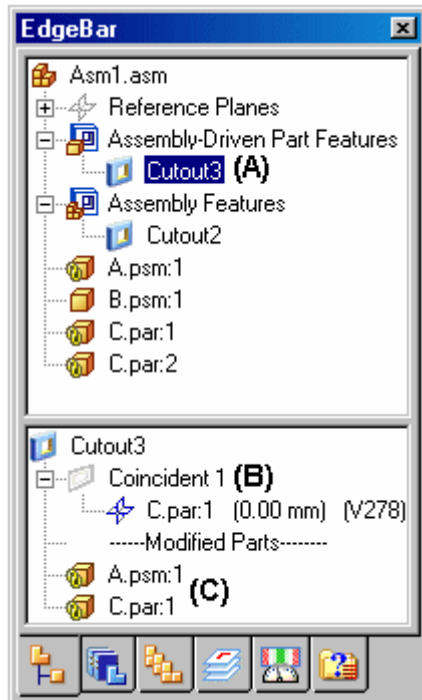
You can select a subassembly as the select set of parts to modify, and all parts that intersect the feature are modified. If parts are added to the subassembly later, the new parts are not added to the feature scope. If you want the new parts to participate in the feature, you must edit the feature and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

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

Assembly Features and Simplified Parts

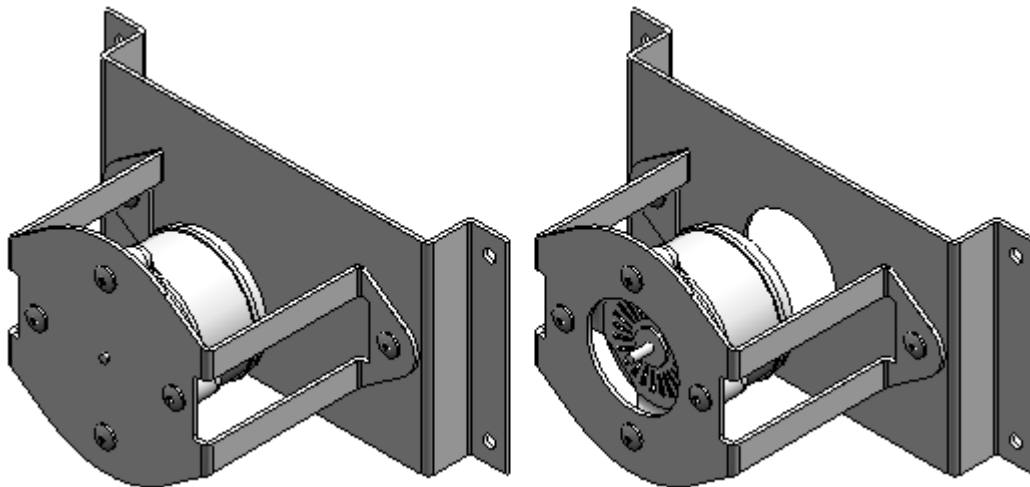
When you create an assembly feature (assembly-based feature or assembly-driven part feature), any simplified parts that are displayed in simplified mode in the assembly are redisplayed in design mode when you select them to be included in an assembly feature.

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.

Lesson

3 *Activity: Assembly features*

Activity guides you creating a cutout traversing multiple parts in an assembly.



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

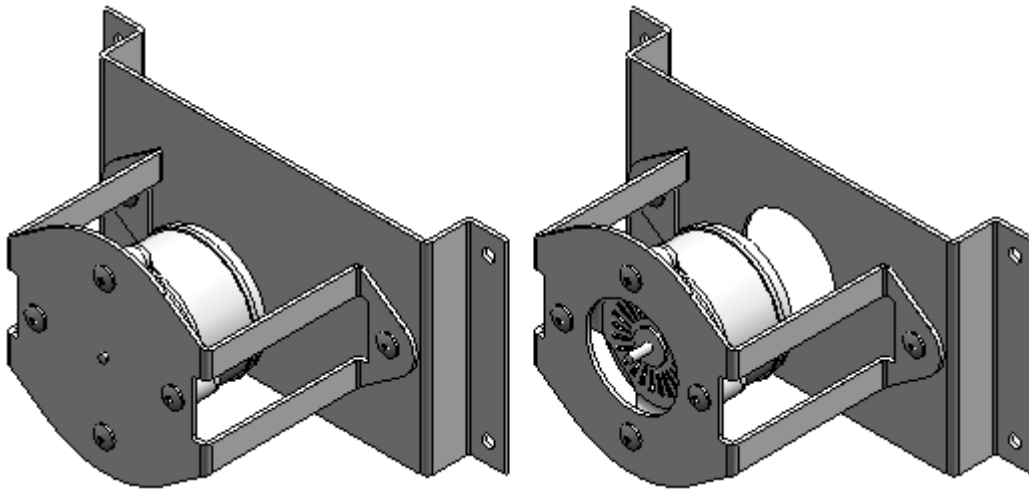
A *Activity: Assembly features*

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.

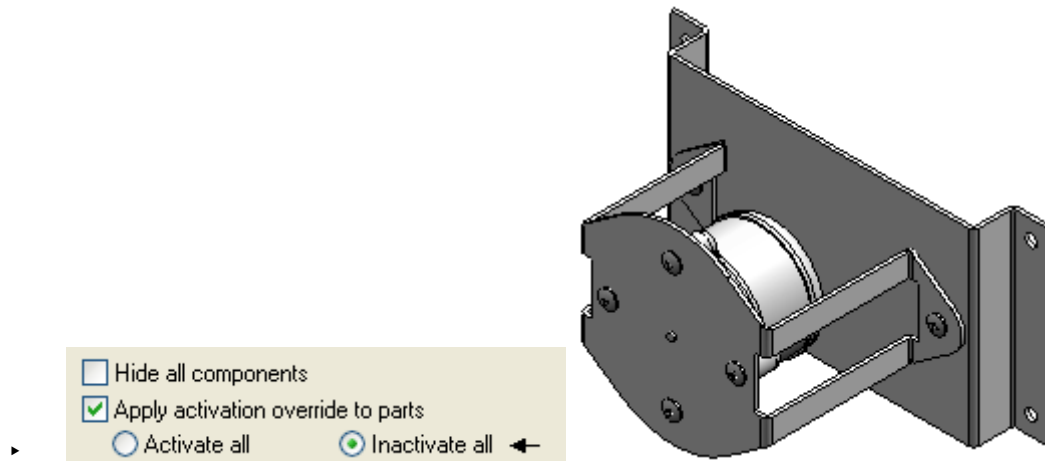
Objectives

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.

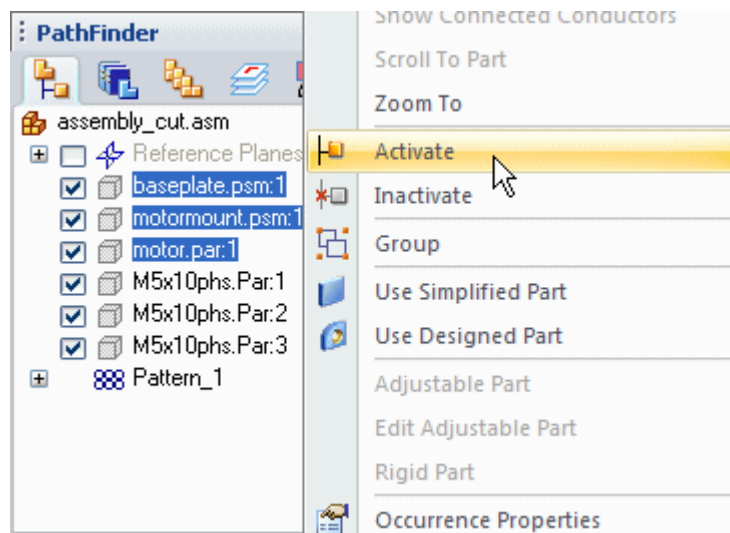


Set the Inter-Part options

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

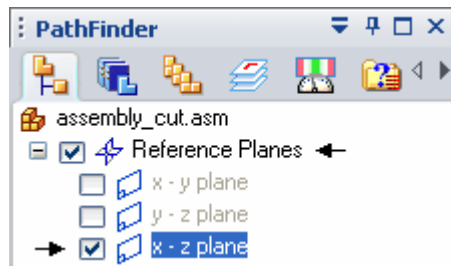
Activate parts in the assembly in order to select them for modification

- 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.




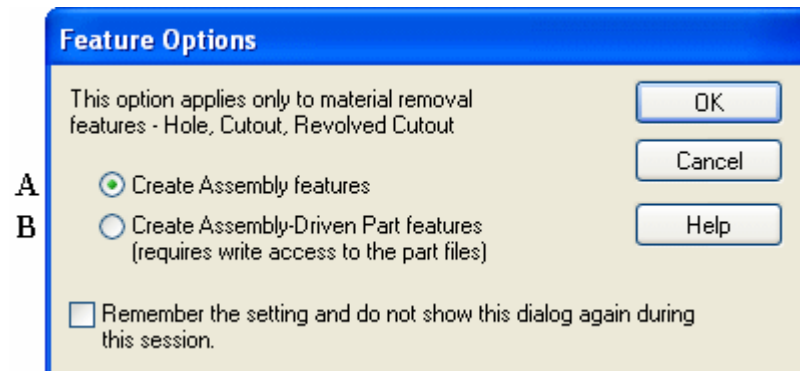
Turn on the display of one of the reference planes

- Turn on the display of the x-z plane. In PathFinder, click the check box on the x-z plane and on 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.



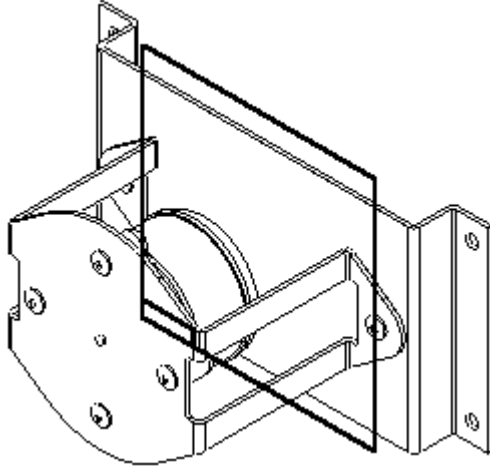
Option A

Create Assembly features—creates the cutout feature in the assembly but does not affect the part files.

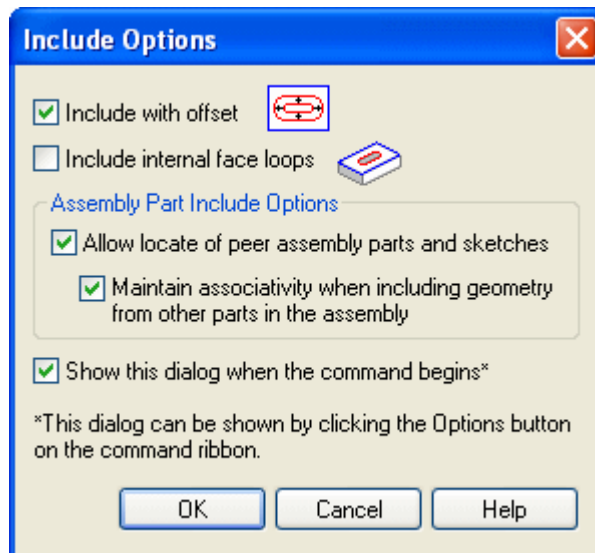
Option B

Create Assembly-Driven Part features—creates the cutout feature by modifying the part files directly.

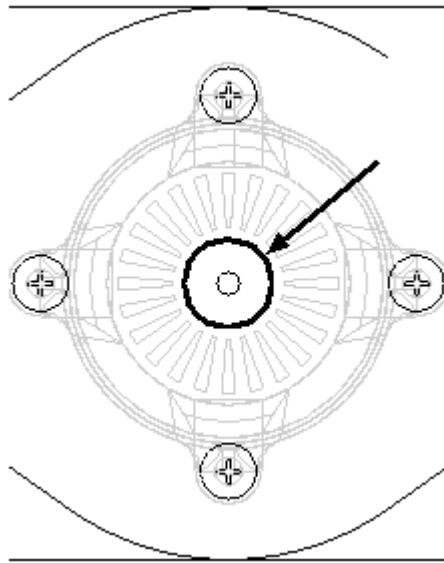
- ▶ Select the assembly reference plane as shown.



- ▶ 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.

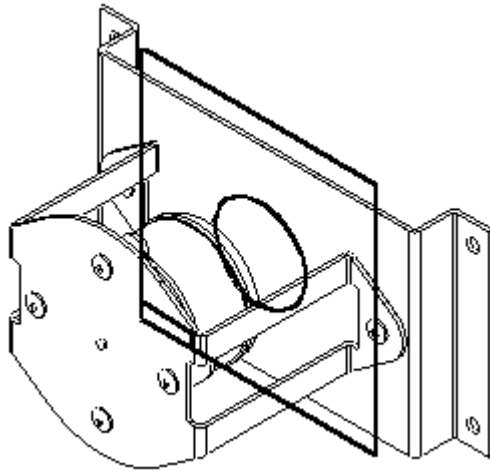
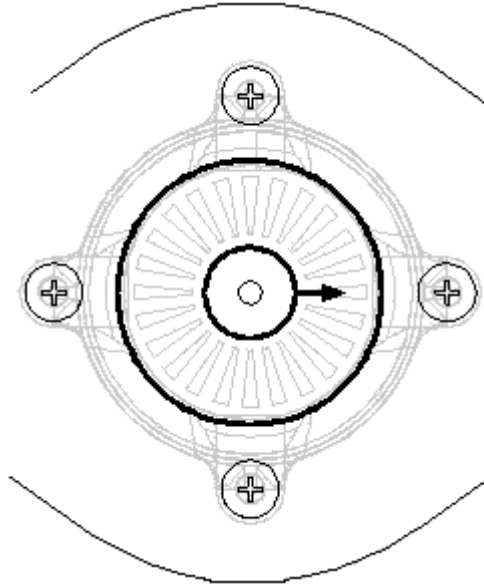


- ▶ Select the circle on the motor.

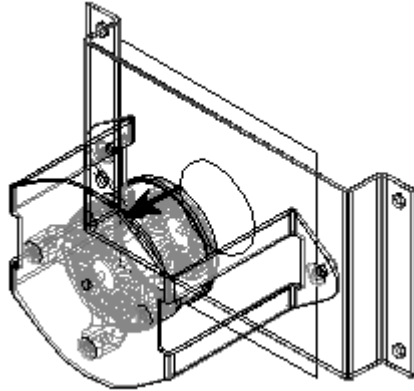


- ▶ Click the Accept button.

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



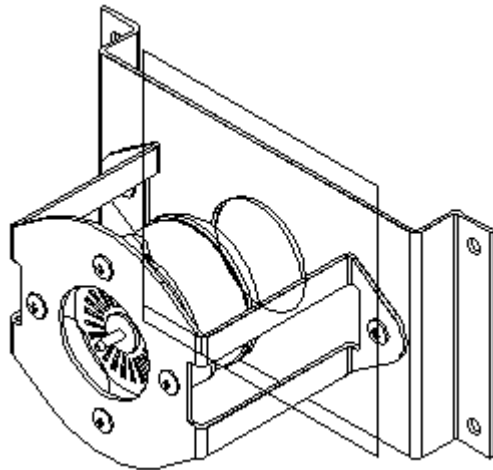
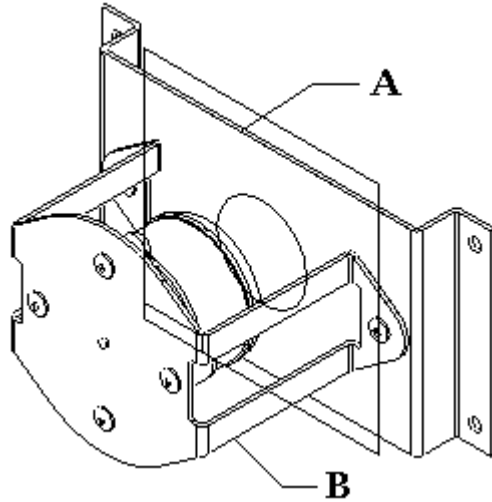
- ▶ On command bar, click the Through All extent option. Position the cursor so the arrow points in the direction as shown, and click to accept.



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

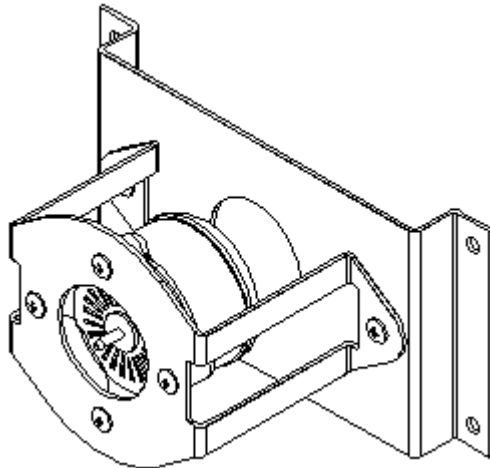


- In PathFinder, select the files *baseplate.psm* (A) and *motormount.psm* (B) for the parts to cut. Click the Accept button, and then click Finish. Both cutouts, driven by the circle and are placed in the assembly only.



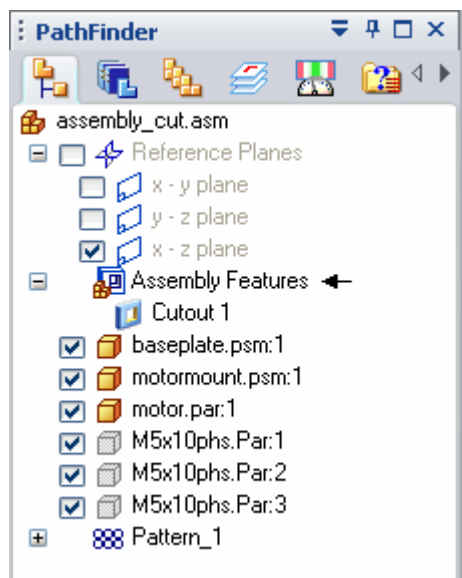
Turn off the reference plane display

- ▶ Click the Select tool. Right-click in the working window to display the shortcut menu as shown, and then click Hide All Reference Planes.

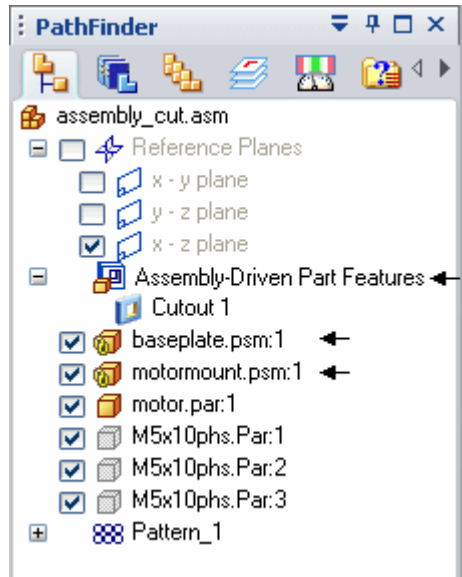


Observe the changes in Pathfinder

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



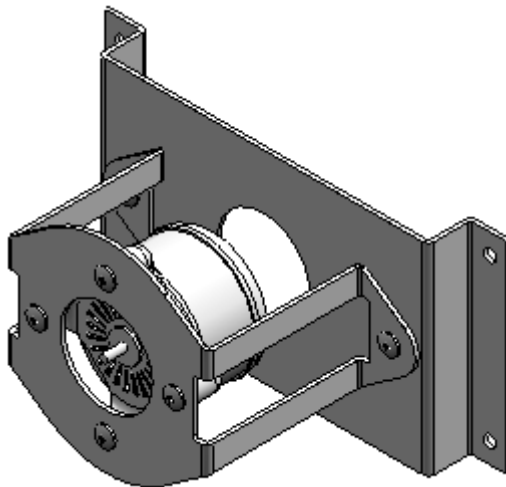
- ▶ If the Create Assembly-Driven Part features option was selected, then PathFinder would show the following.



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.