



Design Complex Parts

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- ▶ Create Advanced Sketch-Based Features
- ▶ Multi-Sections Solid
- ▶ Create Advanced Drafts
- ▶ Advanced Dress-Up Features
- ▶ Use the Multi-Body Method
- ▶ Create Multi-Model Links



8 hours





Case Study: Design Complex Parts

The case study for this lesson is the Bottom Cover of a CD jewel case, as shown below. The focus of this case study is the creation of features that incorporate the design intent for the part. The jewel case will consists of pads, pockets, ribs, solid combines, and Boolean operations, which can be accessed using the Part Design workbench.

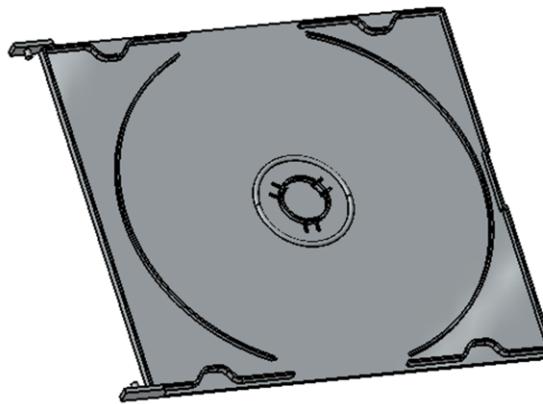


Handwritten notes area for the student guide.

Design Intent (1/2)

The CD jewel case must meet the following design intent requirements:

- ▶ Base feature must include overall dimensions supplied.
 - Two sketches outlining the overall shape of the model are supplied. These sketches can be used to create a solid combine.
- ▶ Create each support as single feature.
 - Creating each support as a rib feature will avoid using multiple features to create the final support geometry.
- ▶ Create a cut to simulate the logo.
 - This cut can be created using a removed multi-sections solid.

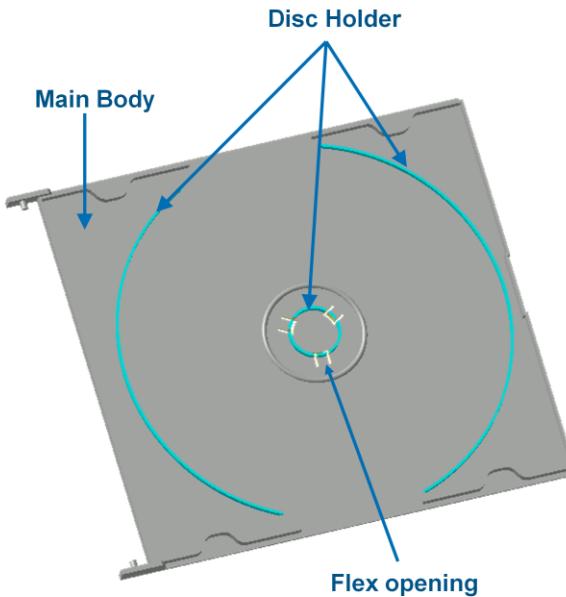


Handwritten notes or sketches can be made here.

Design Intent (2/2)

The CD jewel case must meet the following design intent requirements (continued):

- ▶ Links must be created to the disk holder and flex opening models to ensure conformance to standards.
 - Multi-model links independently copy features from one file to another. By linking to disk holder and flex opening models, any changes that occur in the original source files will update in this file.
- ▶ Linked features must be kept in separate bodies.
 - The bodies copied from other files can be included in the model using Boolean operations. This will keep the features in separate bodies and help with organization.
- ▶ Do not display indented logo when it goes for manufacturing.
 - The removed face tool can remove the logo can be removed from the model without deleting the feature.



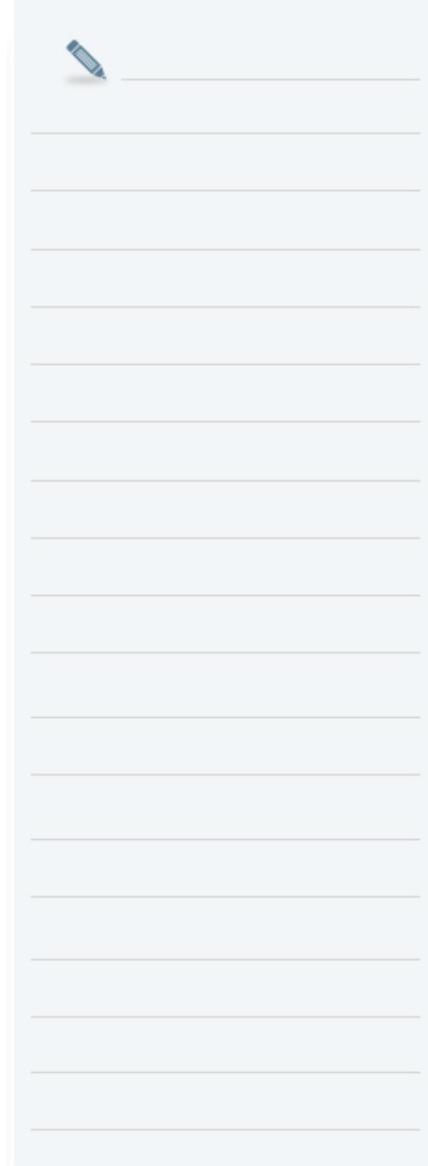
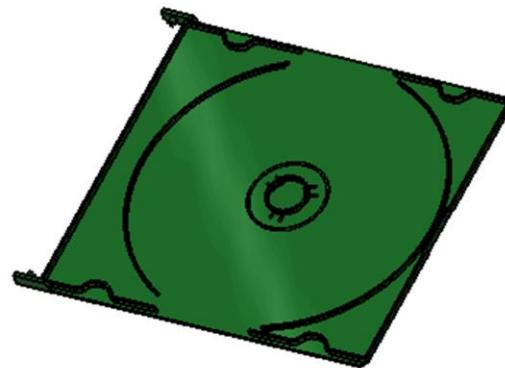
Handwritten notes or sketches can be made here.



Stages in the Process

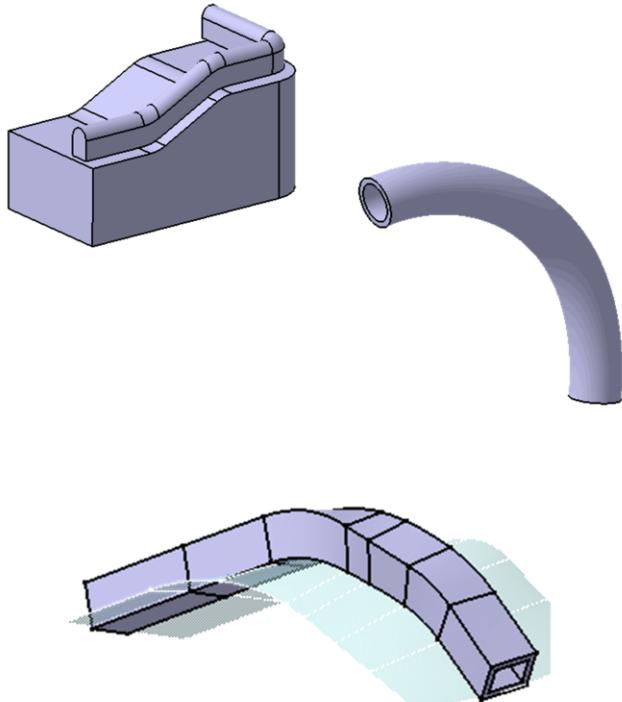
Use the following steps to create the CD jewel case:

1. Create sketched-based features.
2. Create dress-up features.
3. Use the Multi-Body method.
4. Create multi-model links.





Creating Advanced Sketch-Based Features



Here are the topics to be covered:

1. Creating Advanced Sketch-Based Features
2. Creating Multi Section solids
3. Creating Advanced Drafts
4. Creating Advanced Dress-Up features
5. Using the Multi-Body Method
6. Creating Multi-Model Links

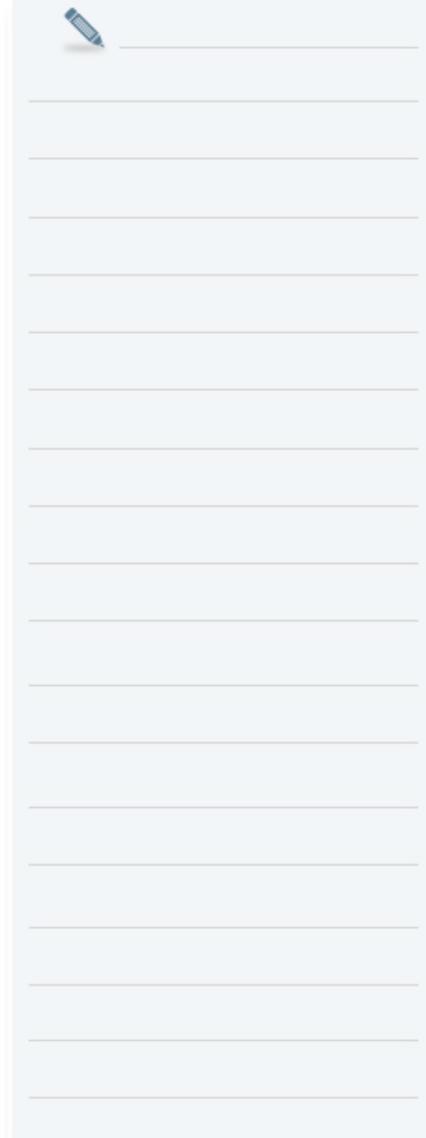
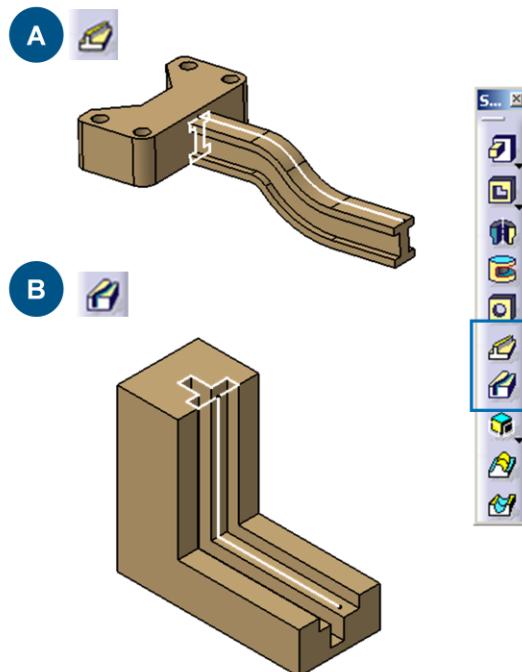


About Ribs and Slots

A rib is a positive (i.e., add material) solid that is generated by sweeping a profile along a center curve.

A slot is a negative (i.e., subtract material) solid that is generated by sweeping a profile along a center curve.
To create a rib, you must have the following:

- A profile that can be a planar open or closed loop sketch.
- A center curve that can be a planar sketch or planar/non-planar continuous wireframe element.

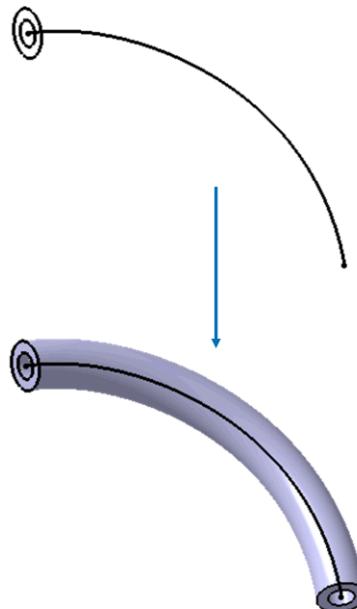


Using Ribs and Slots

Consider using a rib or slot feature when you need to extrude a profile along a non-linear trajectory.

Ribs and slots are used to create complex walls with many details. Using a rib or slot feature enables you to control the complexity of the sketch. It enables you to create, in one feature, what may take many other features (such as pads and pockets).

Ribs can be used to create a pipe feature by sweeping two closed loop profiles, created in the same sketch, along a center curve.

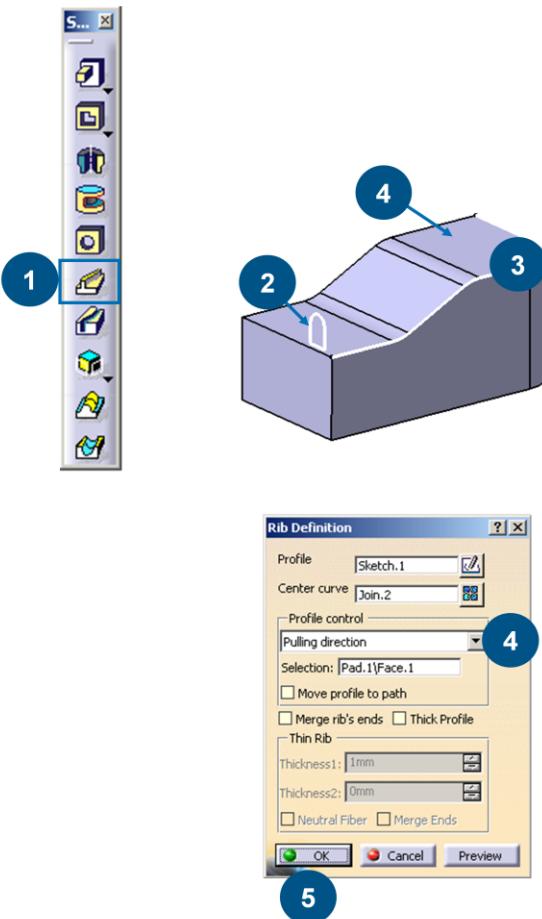
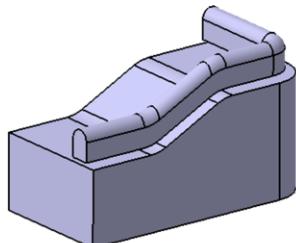




Creating a Rib

Use the following steps to create a rib feature:

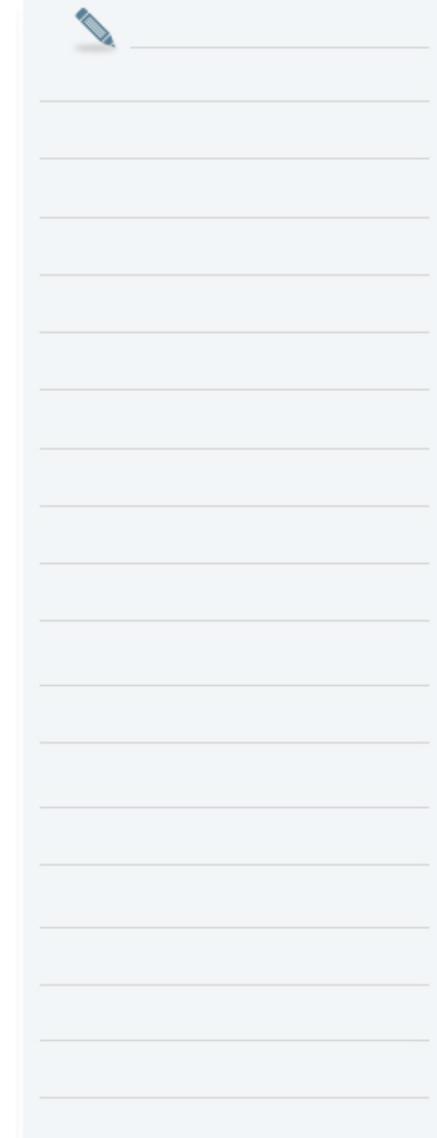
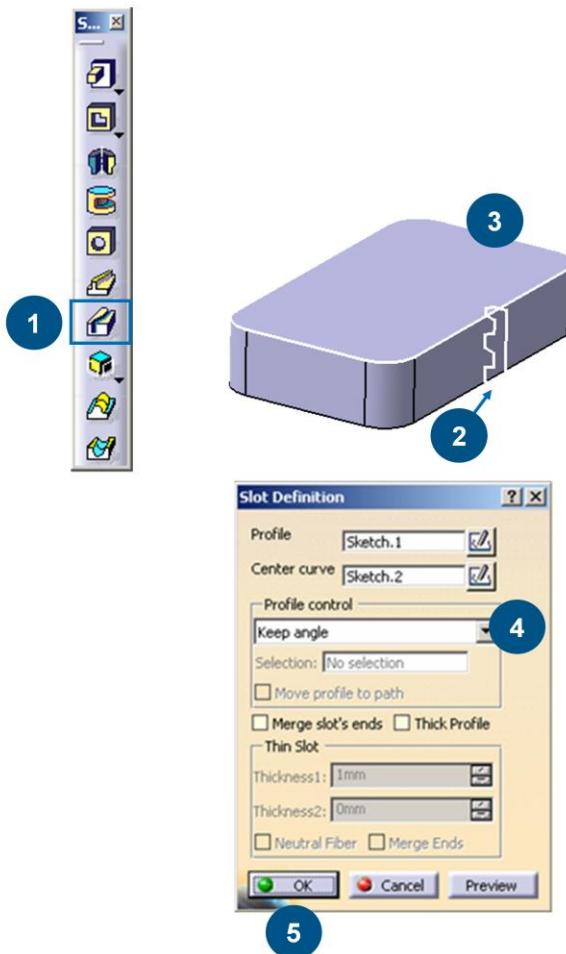
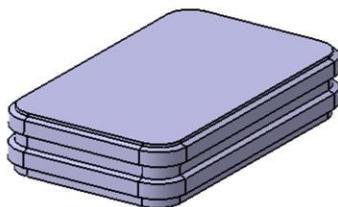
1. Click the **Rib** icon.
2. Select the profile to be swept.
3. Select the center curve to sweep the profile along. In this example, the center curve is a 3D curve created in the Wireframe and Surface Design workbench.
4. Select the appropriate **Profile Control** option. In this example, **Pulling direction** is selected and the top surface of the base feature is selected as the reference.
5. Click **OK** to complete the rib feature.



Creating a Slot

Use the following steps to create a slot feature:

1. Click the **Slot** icon.
2. Select the profile to be swept.
3. Select the center curve to sweep the profile along.
4. Select the appropriate **Profile Control** option. In this example, the default option, **Keep Angle** is selected.
5. Click **OK** to complete the rib feature



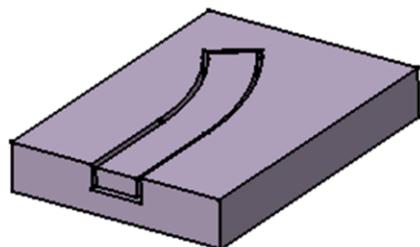
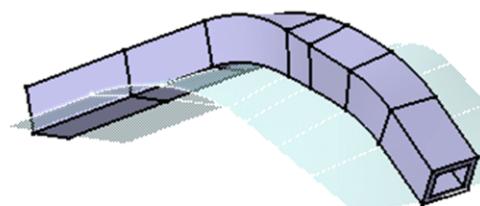
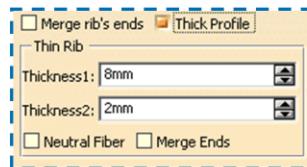
Creating Thin Ribs and Slots (1/3)

Ribs and slots can be created as thin features using the **Thick Profile** option.

Using the **Thick Profile** option, thickness is added to one or both sides of the profile.

To create a thin profile, select the **Thick Profile** option and enter the thickness(es) in the appropriate field.

Use the **Neutral fiber** option to add material to both sides of the profile equally.

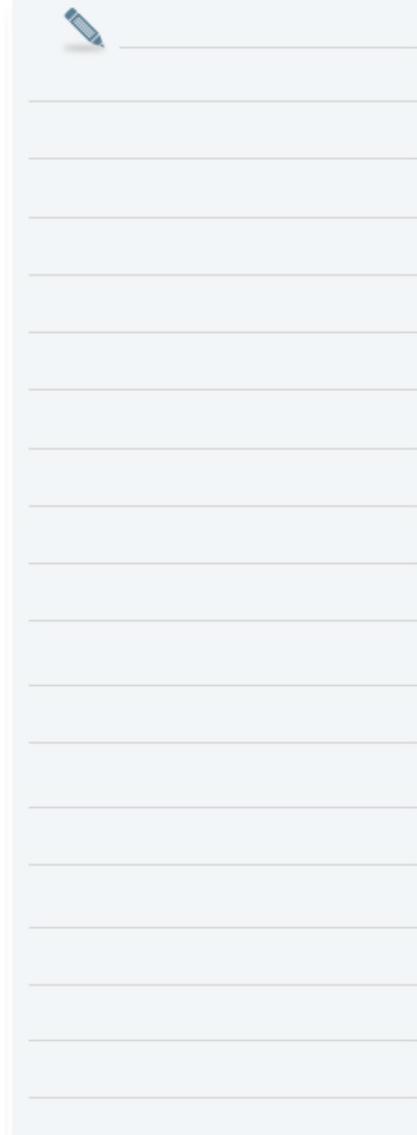
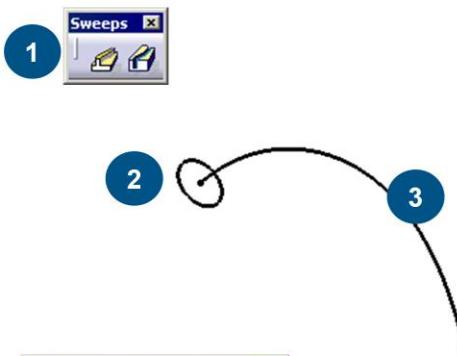




Creating Thin Ribs and Slots (2/3)

Use the following steps to create a thin rib or slot feature:

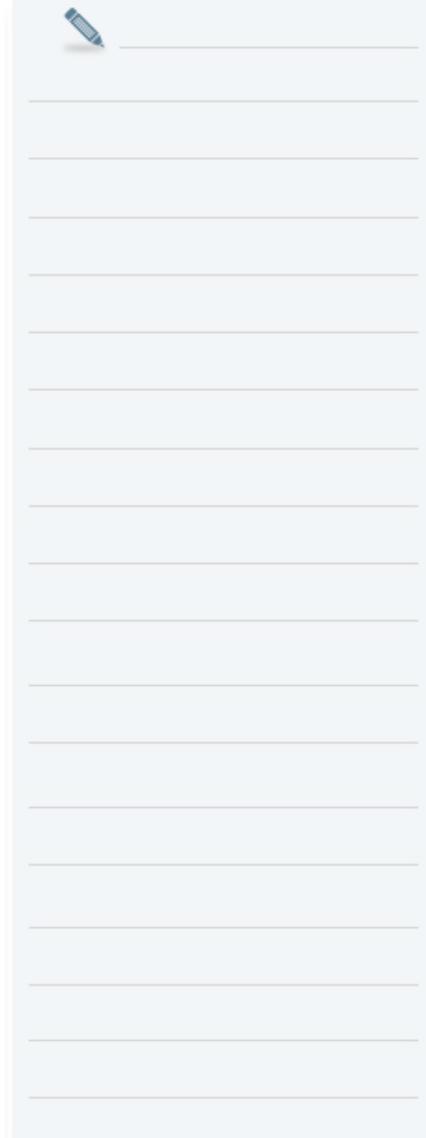
1. Click the feature icon. In this example a rib will be created.
2. Select the profile.
3. Select the center curve.
4. Define the Profile control. In this example, the default option **Keep angle** is selected.
5. Select the **Thick Profile** option.



Creating Thin Ribs and Slots (3/3)

Use the following steps to create a thin rib or slot feature
(continued):

6. If needed, select the **Neutral Fiber** checkbox to add material equally to both sides of the profile.
7. Enter the thickness as required. If the **Neutral fiber** option is selected, only the **Thickness1** field will be available.
8. If required, select the **Merge Ends** option to trim the feature to the existing material.
9. Click **OK** to complete the thin feature.



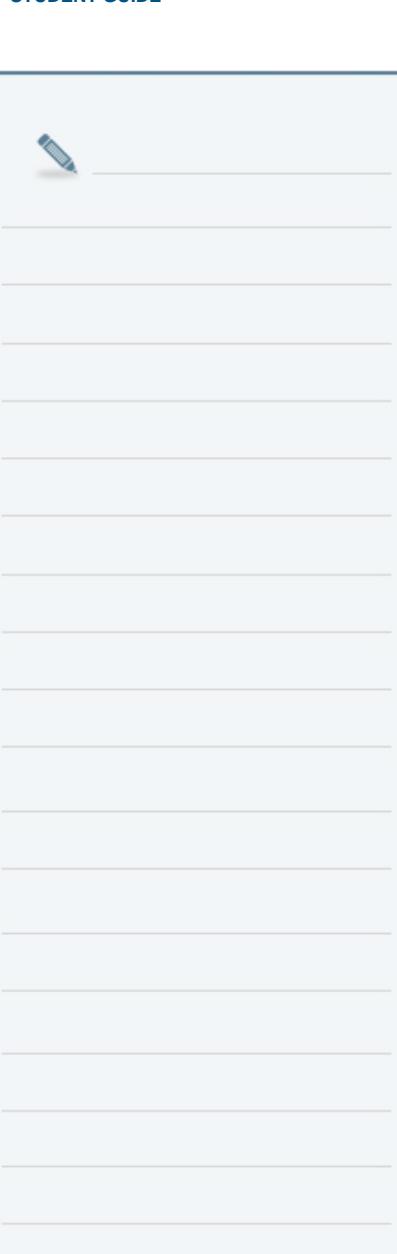
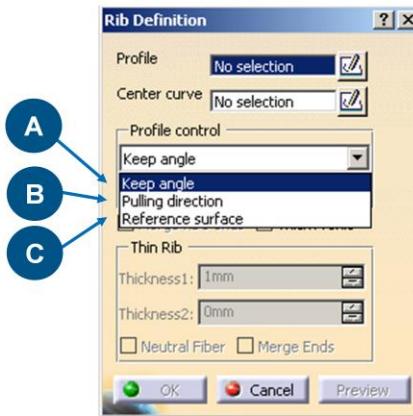


Setting the Rib and Slot Options (1/2)

Profile control and Merge Ends options can be used to help control the Rib or Slot.

The profile of the feature is controlled using options from the Profile control pull-down menu.

- A. The Keep angle option maintains a constant angle between the profile's sketch support and the tangent of the center curve.
- B. The Pulling direction option causes the profile to be swept along the center curve with respect to a specified direction. The direction can be defined using a plane or an edge.
- C. The Reference surface option causes the profile to remain at a constant angle to a selected reference surface.



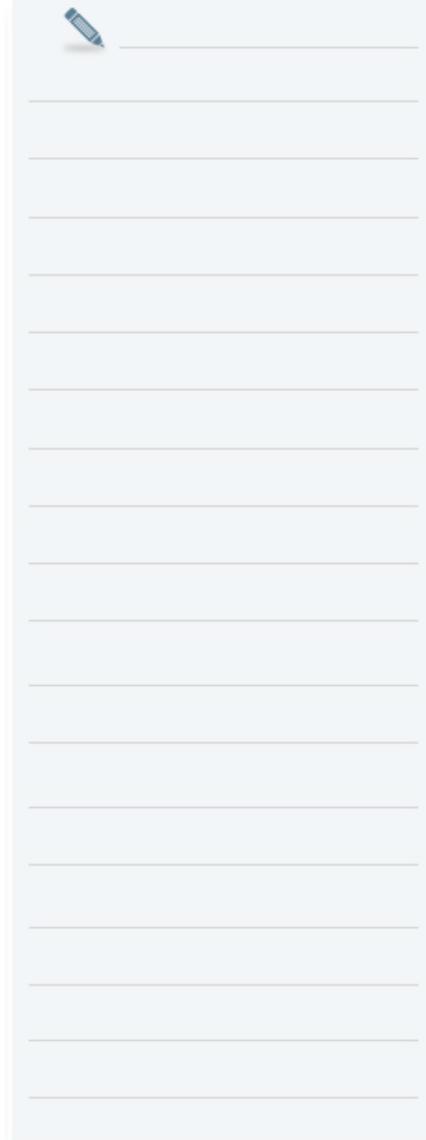
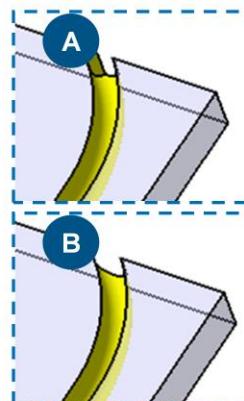
Setting the Rib and Slot Options (2/2)

Profile control and Merge ends options can be used to help control the Rib or Slot (continued).

The **Merge slot's ends** and **Merge rib's ends** options can be used to extend or shorten the feature to its proper wall.

- A. When the option is cleared the feature terminates at the end of the center curve. In the example shown, the feature does not fully extend to the edge of the base feature when the option is cleared.

- B. When the option is selected, the feature is either extended or shortened, to blend into the existing material. In the example shown, the profile is extended to fully intersect the base feature.

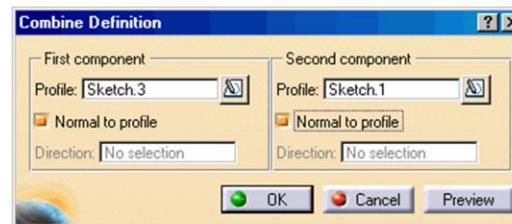
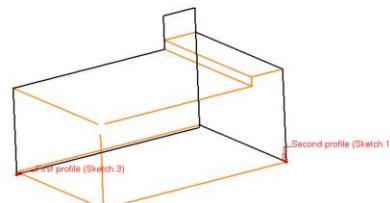
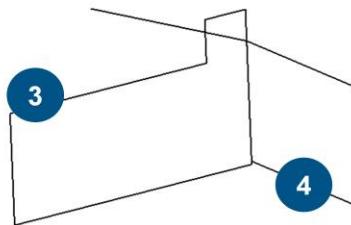


Creating Solid Combines (1/2)

A Solid Combine feature is created by the intersection of two extruded profiles.

Use the following steps to create a solid combine:

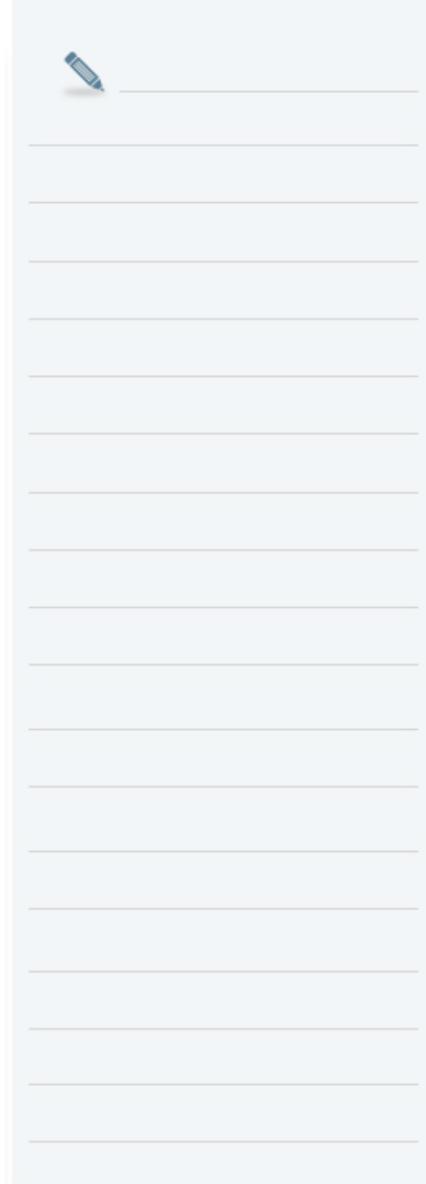
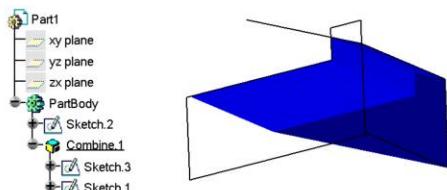
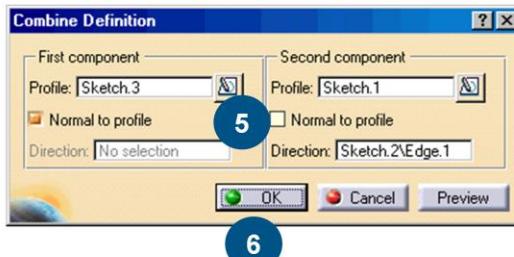
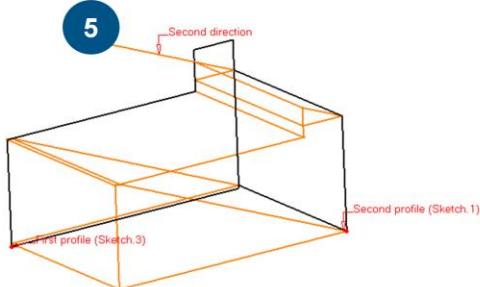
1. Create the sketched profiles. The sketches must contain closed profiles.
2. Click the **Solid Combine** icon.
3. Select the first sketch.
4. Select the second sketch.



Creating Solid Combines (2/2)

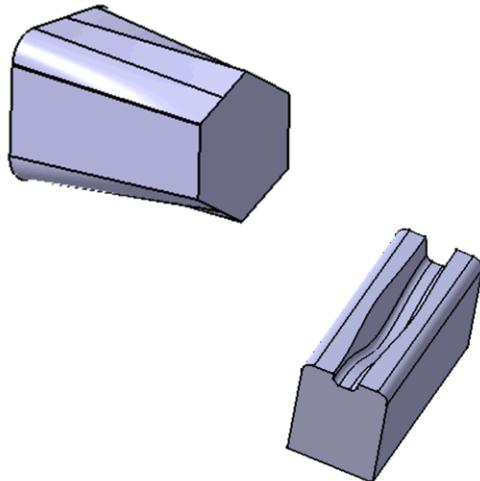
Use the following steps to create a solid combine (continued):

5. By default, profiles are extruded normal to the sketch support. To change the direction, clear the **Normal to Profile** option and select a geometrical element to indicate the extrude direction.
6. Click **OK** to create the feature. The solid combine is the intersection of these profiles when they are extruded.





Creating Multi-Sections Solids



Here are the topics to be covered:

- 1. Creating Advanced Sketch-Based Features
- 2. **Creating Multi Section solids**
- 3. Creating Advanced Drafts
- 4. Creating Advanced Dress-Up features
- 5. Using the Multi-Body Method
- 6. Creating Multi-Model Links

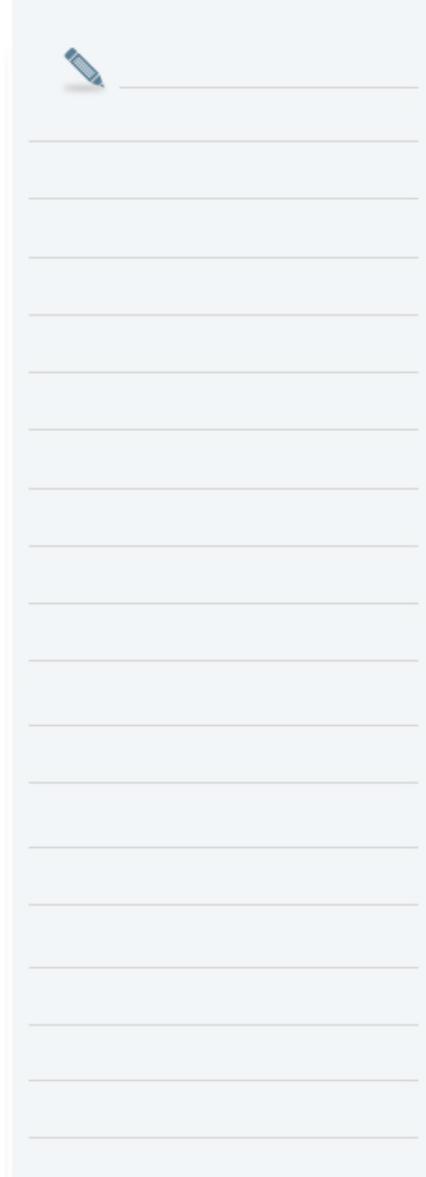


About Multi-Sections Solids

The Multi-Sections solid can be a positive (i.e., add material) or negative (i.e., subtract material) solid that is generated by two or more planar profiles swept along a spine.

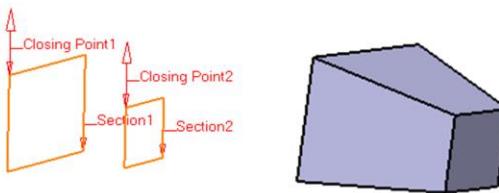
Common uses for Multi-Sections solids are to create complex solids and transition geometry between two existing solids.

Like the Multi-Section solids, removed Multi-Sections solids are used to subtract a transitional surface from an existing solid.



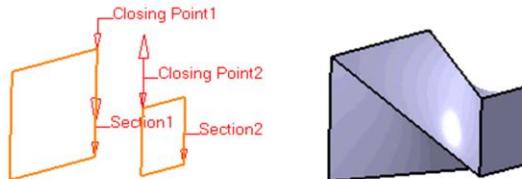
About Closing Point and Orientation (1/4)

While defining the multi-sections solid, the closing points are displayed on a vertex in each of the selected profiles. These closing points indicate how the system will connect the vertices.



The directional arrow indicates the direction of the next aligned vertices. Ensure that the arrow points in the same direction for each section.

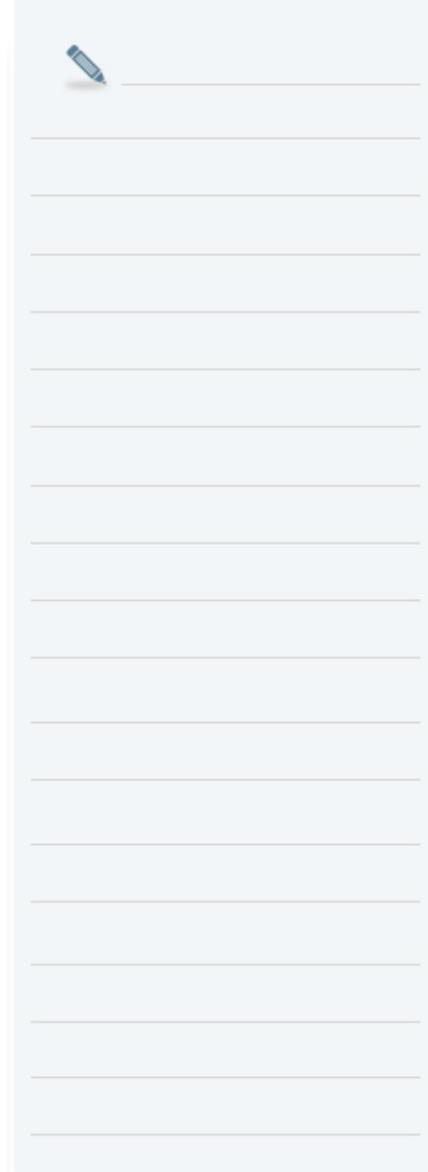
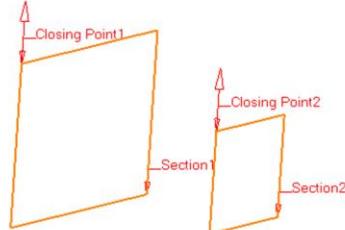
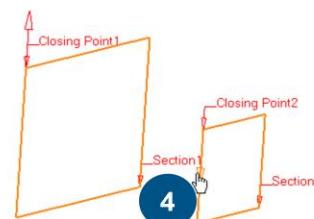
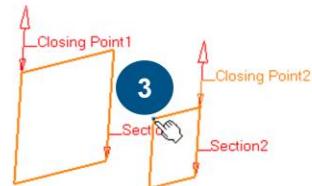
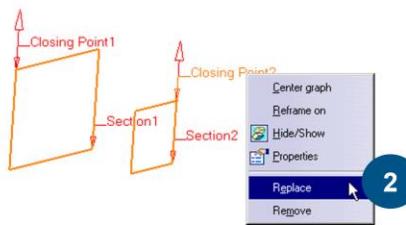
Closing points must be aligned for proper orientation of the sections, else the multi-sections solid will be twisted.



About Closing Point and Orientation (2/4)

Use the following steps to replace the Closing Point location:

1. Right-click the existing Closing Point.
2. Click **Replace** from the contextual menu.
3. Select the replacing vertex.
4. To change the direction of the arrow, click it.

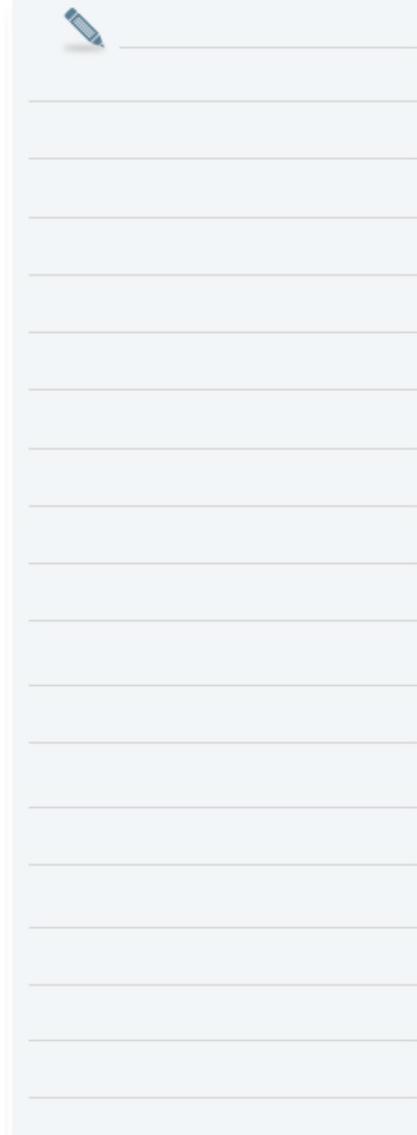
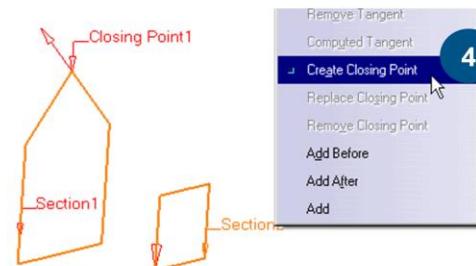
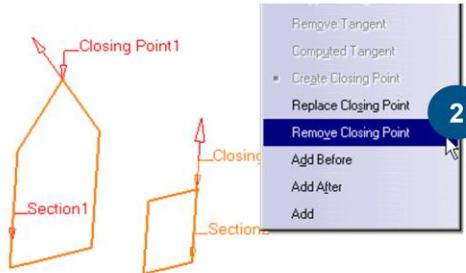


About Closing Point and Orientation (3/4)

If there is no vertex in the required location for the closing point, you can create a closing point while in the feature operation.

Use the following steps to create a Closing Point:

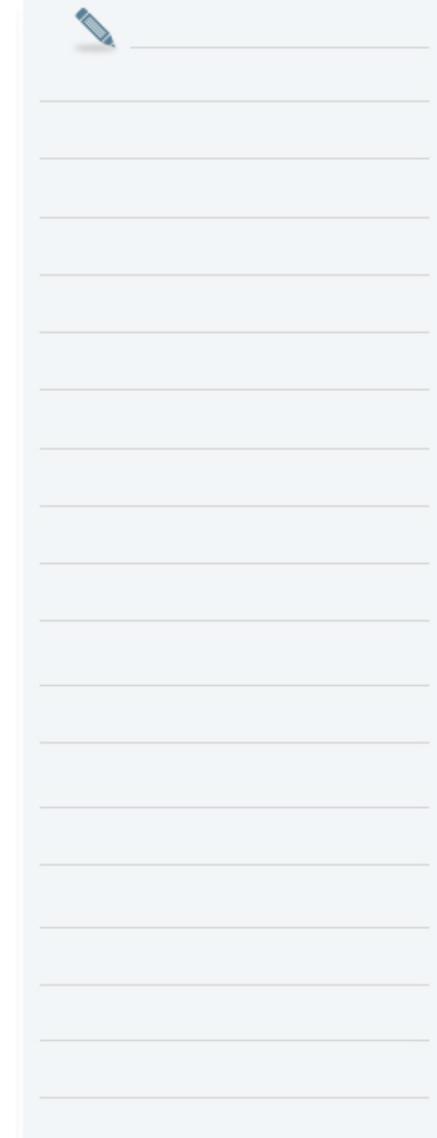
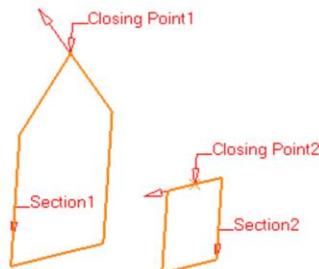
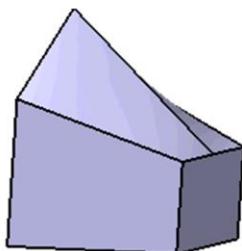
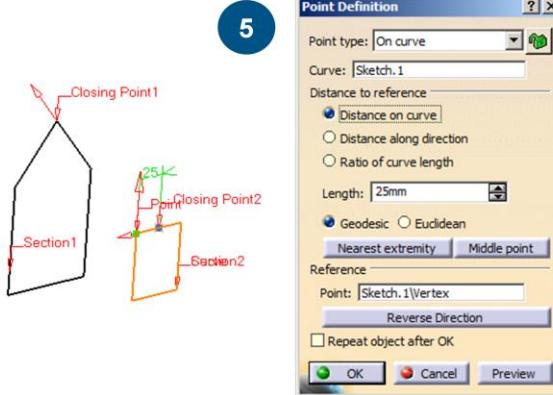
1. Right-click the section.
2. Click **Remove Closing Point**.
3. Right-click again on the section
4. Click **Create Closing point**.



About Closing Point and Orientation (4/4)

Use the following steps to create a Closing Point
(continued):

5. Define the point location using the **Point Definition** dialog box.
6. Click **OK** to generate the Closing Point and return to the Feature Definition.

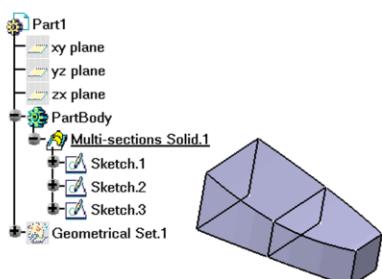
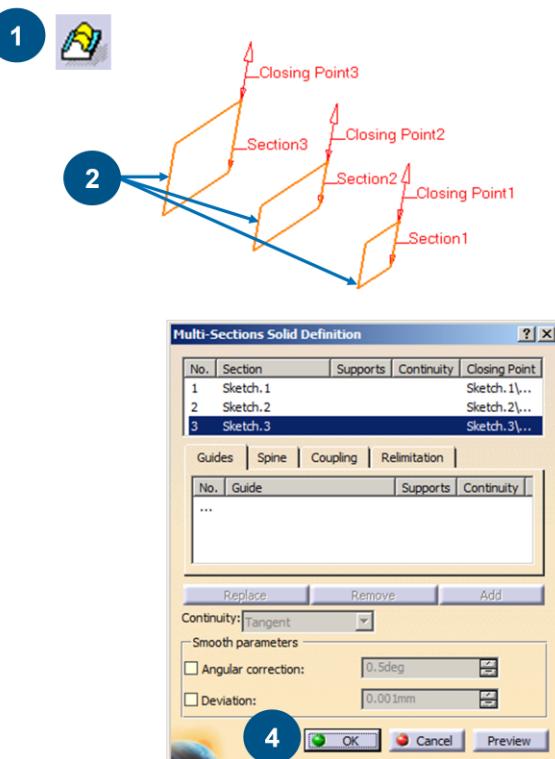




Creating a Simple Multi-Sections Solid

Use the following steps to create a simple multi-sections solid:

1. Click the **Multi-sections Solid** icon.
2. Select the section through which the features will pass. The order of selection is important as it defines the order of connection between the sections.
3. Ensure the location and direction of the Closing Points are correct.
4. Click **OK** to generate the feature.

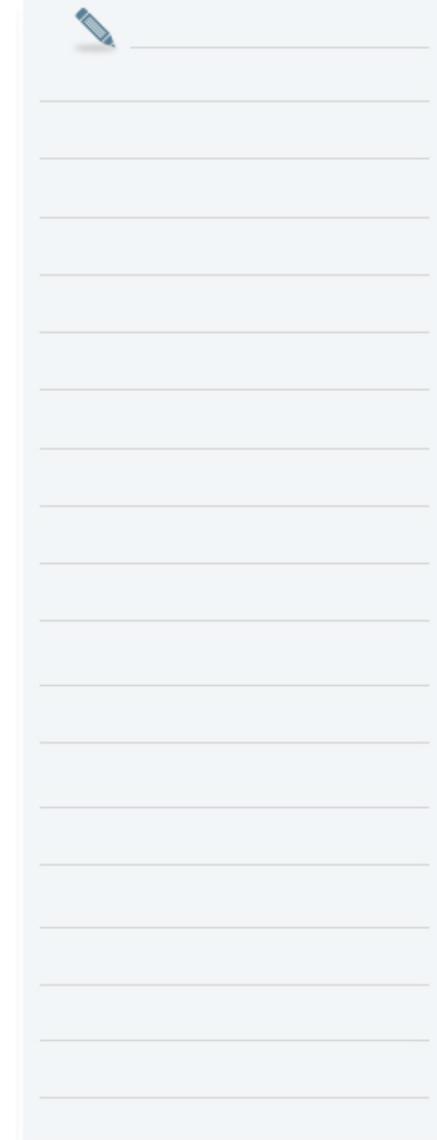
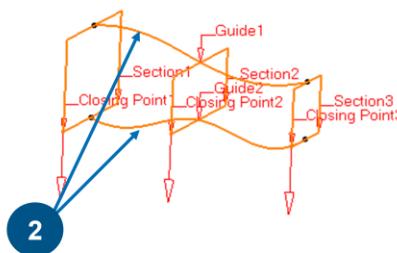
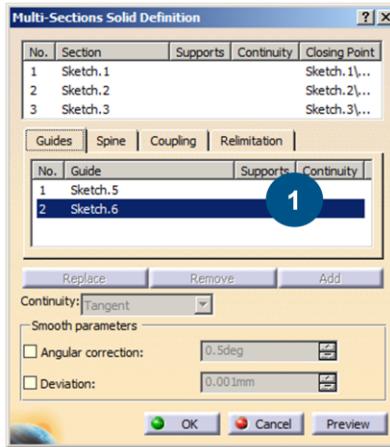
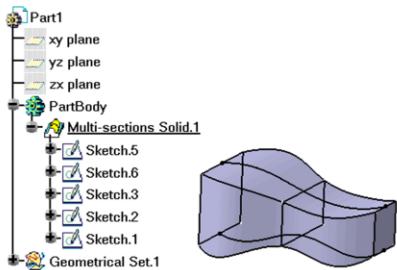


About Guides

Guides are used to help control the shape of the multi-section solid as it transitions between the profiles. Guides must intersect all sections of the feature.

From the feature dialog box use the following steps to add guides:

1. Select the **Guides** tab.
2. Select the guides. One or more guides can be used to control the shape of the feature.

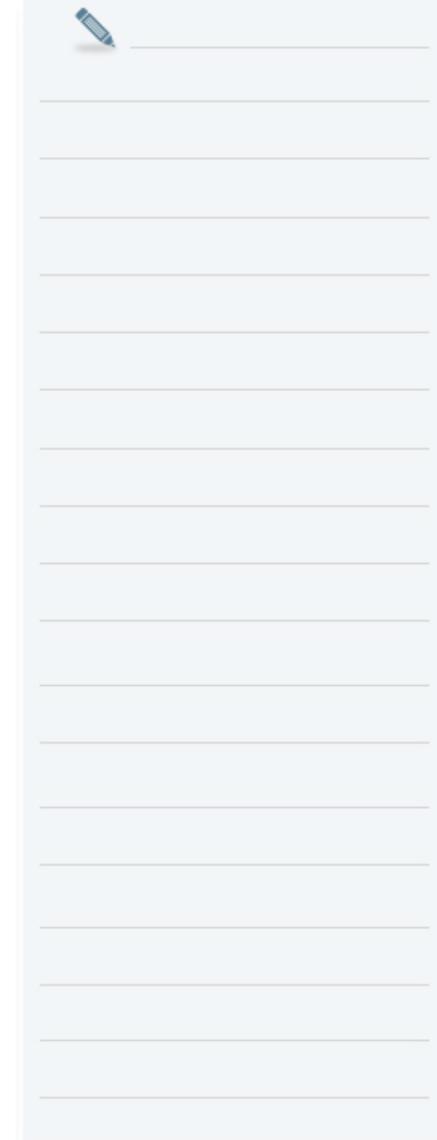
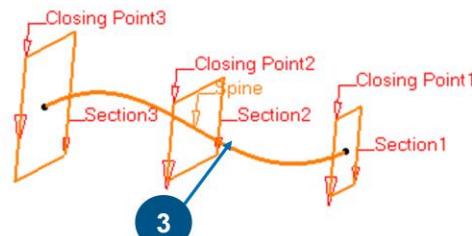
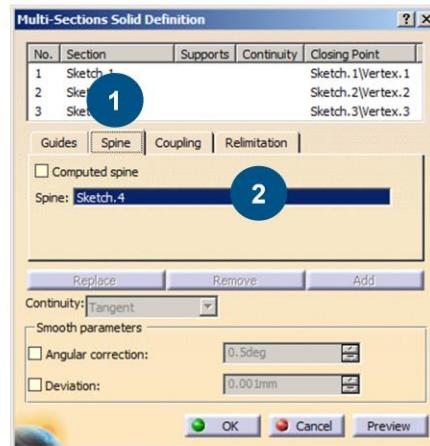
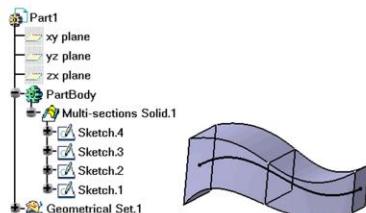


About Spine

A Spine controls shapes of the features between profiles. As the feature transitions between the sections, it must always remain perpendicular to the Spine. A spine is automatically computed when creating a solid. If required, you can use a user-defined Spine.

From the feature dialog box use the following steps to add a user-defined spine.

1. Select the **Spine** tab
2. Select the **Spine** field.
3. Select the spine.



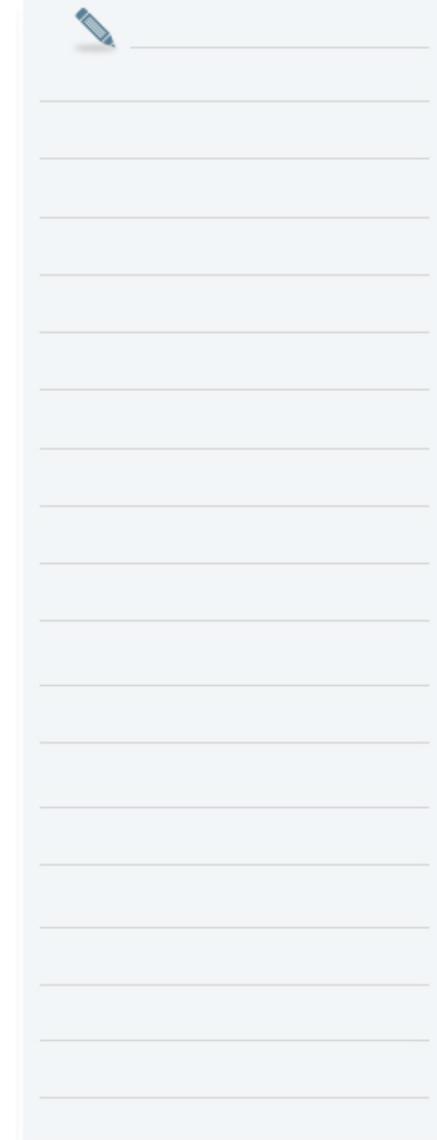
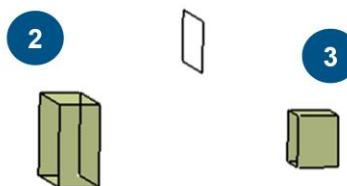
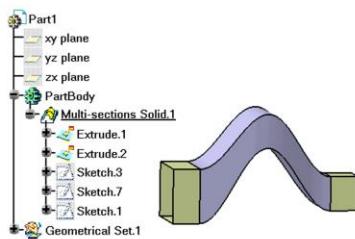
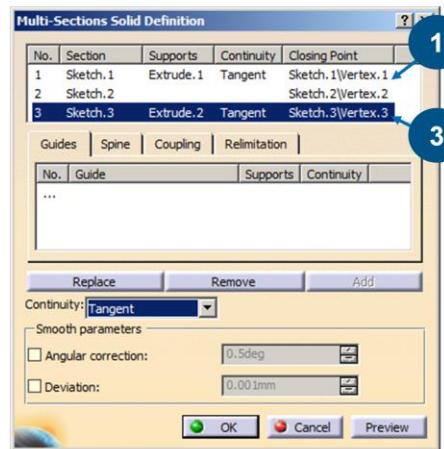


About Tangent Surfaces

When multi-sections solids are used as transitional features, it must be tangent to the adjoining solid.

Use the following steps to apply tangency:

1. From the feature dialog box select the section.
2. Select the Tangent surface.
3. Repeat steps 1 and 2 for each section requiring tangency. In this example, both the first and last sections have tangency constraints applied to them.





About Couplings

A Coupling refers to the way the profiles are connected.

The following are several Coupling options available:

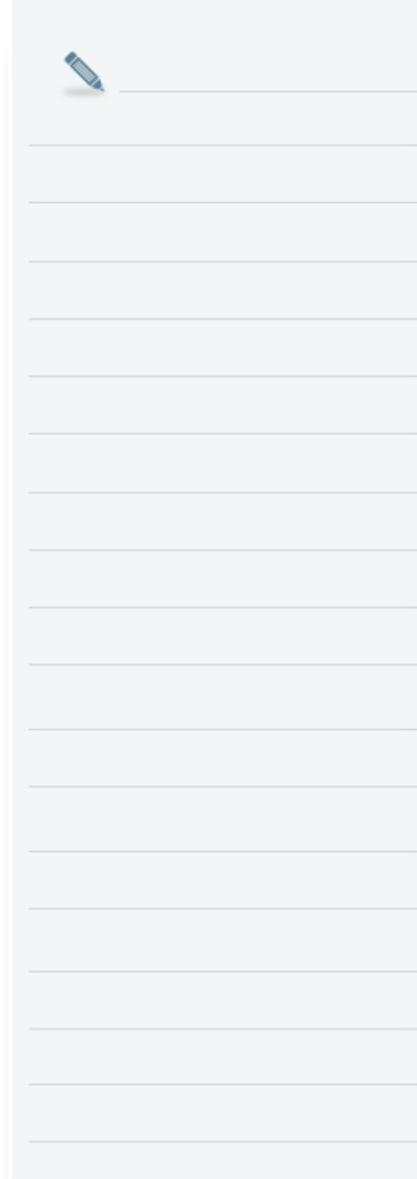
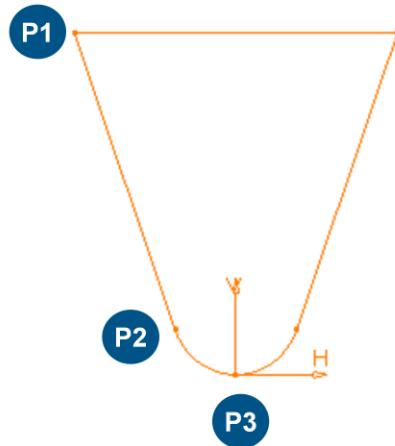
- Using the **Ratio** option, the curves are coupled according to a ratio of the total length of each section.
- Using the **Tangency** option, the curves are coupled at their tangency discontinuity points. To use this option the same number of tangency discontinuity points must exist in all the sections.
- Using the **Tangency then curvature** option, the curves are coupled at their tangency discontinuity points first and then later at their curvature discontinuity points. To use this option the same number of tangency discontinuity point and curvature discontinuity points must exist in all the sections.
- Using the **Vertices** option, the curves are coupled at their vertices. To use this option the same number of vertices must exist in all the sections.



About Points of Continuity in a Coupling

To effectively illustrate the points of continuity concept consider the profile shown. This profile has several types of Continuity:

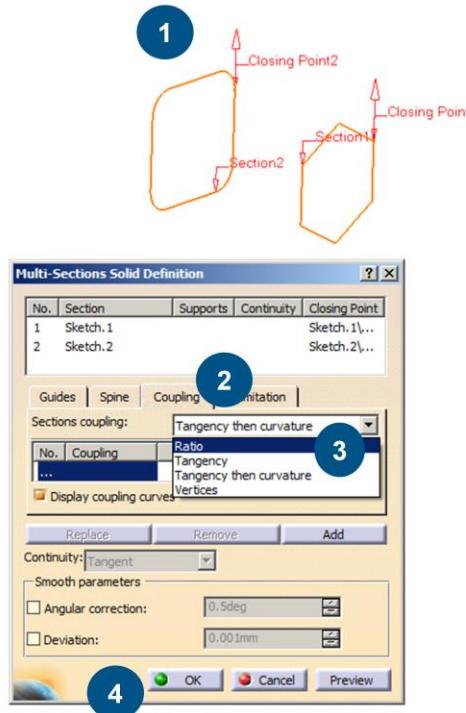
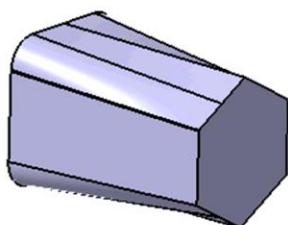
Points on profile	Point Continuity	Tangency Continuity	Curvature Continuity
P1	✓	✗	✗
P2	✓	✓	✗
P3	✓	✓	✓



Modifying a Coupling

Use the following steps to change the Coupling option:

1. Select and orient the profiles.
2. Select the **Coupling** tab.
3. Select the type of **Sections coupling**. In this example, the **Ratio** option is selected. **Ratio** is selected because the number of vertices in each section is not equal.
4. Click **OK** to generate the feature.

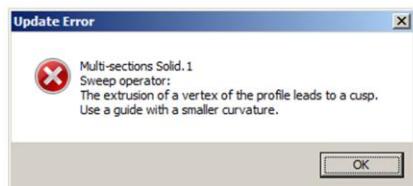
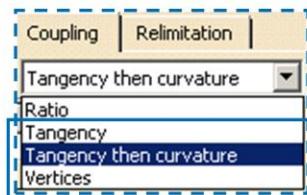
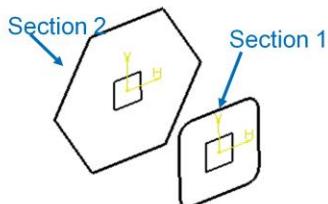


Displaying Uncoupled Points (1/2)

An error will display if CATIA cannot couple the profiles automatically.

For each Coupling mode, the points that could not be coupled are displayed in the geometry with specific symbols.

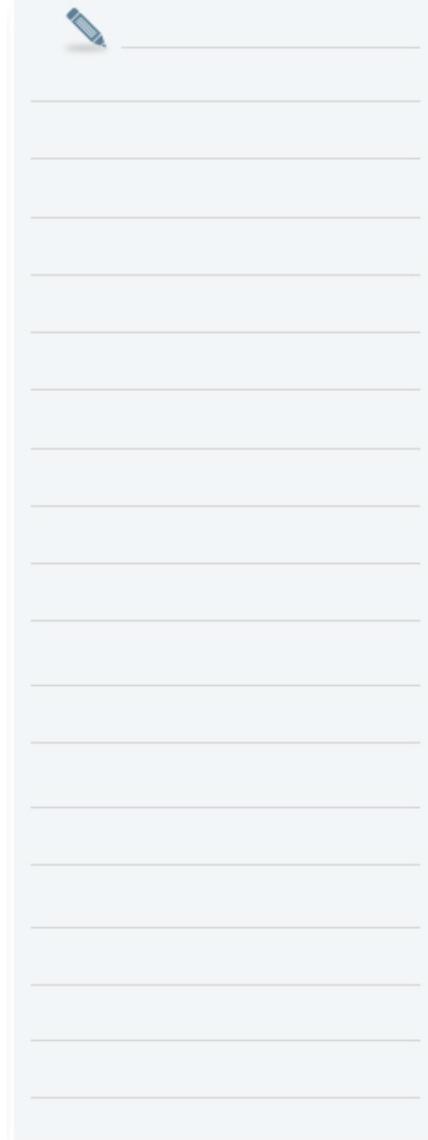
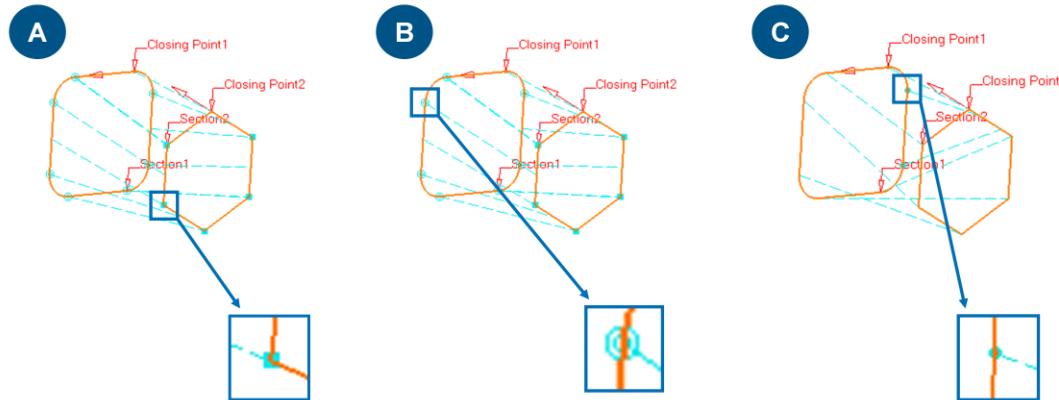
For example, if a hexagon profile is transitioned to a square profile with rounded edges an error message will display indicating that the current Coupling mode cannot be applied for the Coupling options of **Tangency**, **Tangency then curvature**, and **Vertices**.



Displaying Uncoupled Points (2/2)

For each Coupling mode, the points that could not be coupled are displayed in the geometry with specific symbols:

- A. Uncoupled tangency discontinuities are represented by a square.
- B. Uncoupled Curvature discontinuities are represented by an empty circle.
- C. Uncoupled vertices are represented by a full circle.



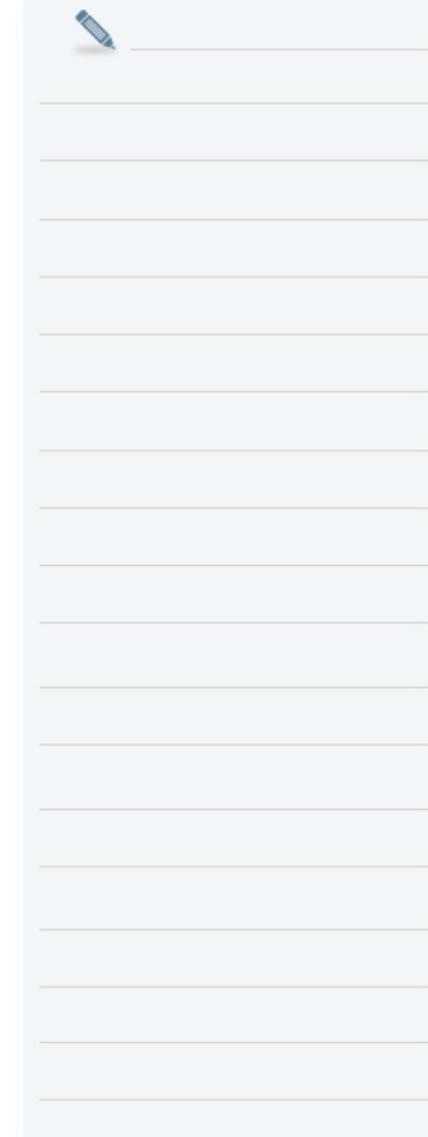
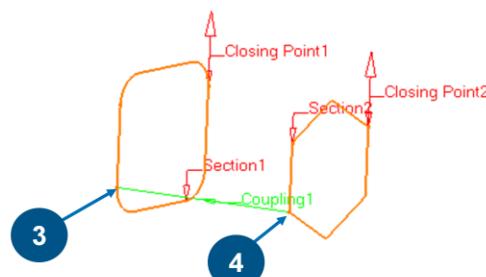
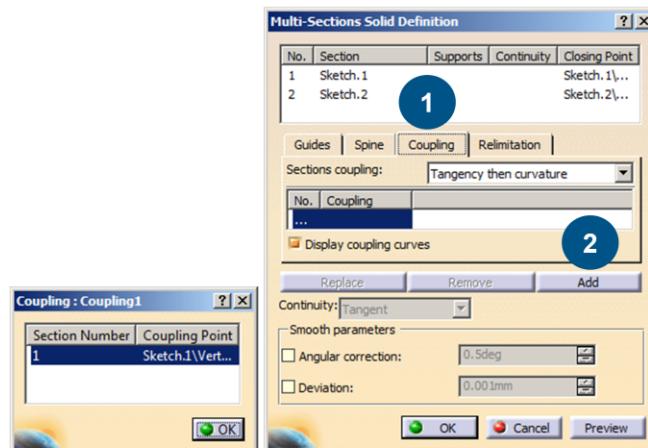


About Manual Couplings (1/2)

If the sections in the multi-sections solid (or removed multi-sections solid) do not have the same number of vertices you can define the Coupling manually.

From the feature definition use the following steps to manually couple the sections:

1. Select the **Coupling** tab.
2. Click **Add**. If the **Add** button is unavailable, select inside the coupling window to activate it.
3. Select a point on the first section.
4. Select the corresponding point on each of the other sections. Remember to select the points in the correct order or the feature will fail.

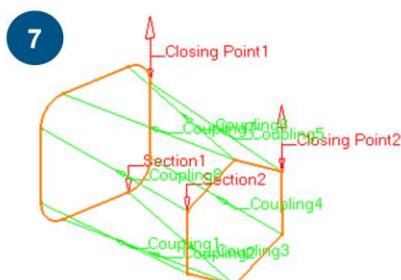
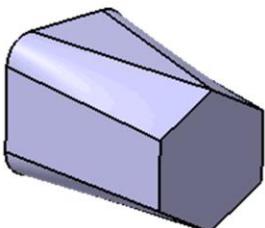
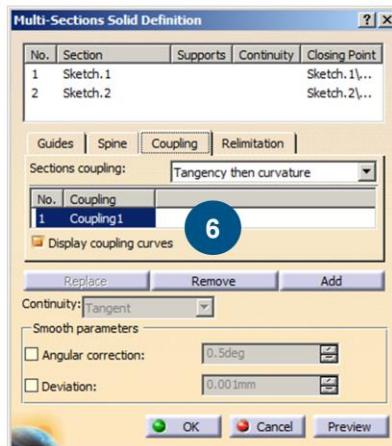




About Manual Couplings (2/2)

From the feature definition use the following steps to manually couple the sections (continued):

5. Once the Coupling points for each section have been defined, the **Coupling** dialog box automatically disappears.
6. Click inside the **Coupling** dialog box to make the Add button available.
7. Repeat steps 2 – 6 for each Coupling.



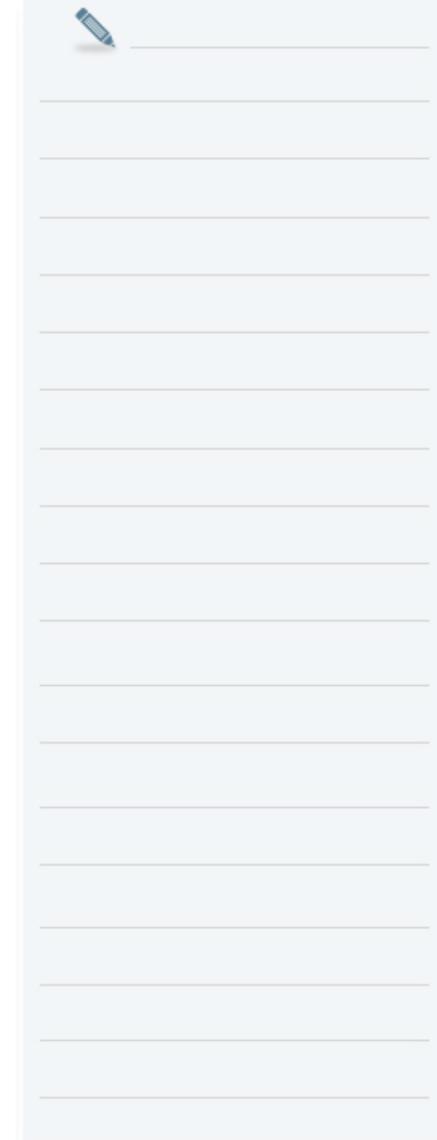
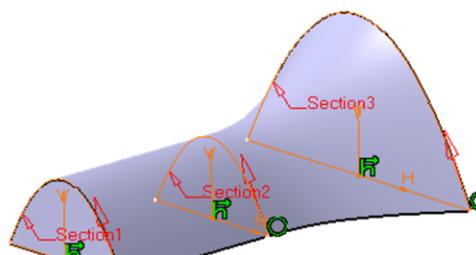
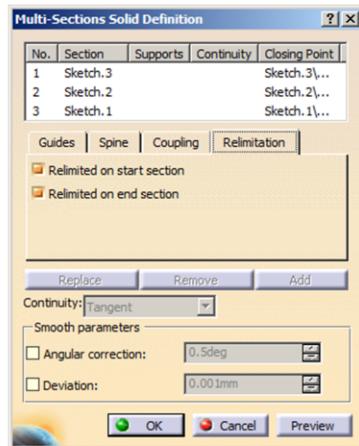


Relimiting Multi-Sections Solid (1/3)

By default, Multi-sections Solids and removed Multi-sections Solids are limited by the start and end sections. You can choose to change the limit of the feature to the length of a user-defined Spine or Guides.

You can limit the start or the end section of the feature by clearing the appropriate option on the Relimitation tab.

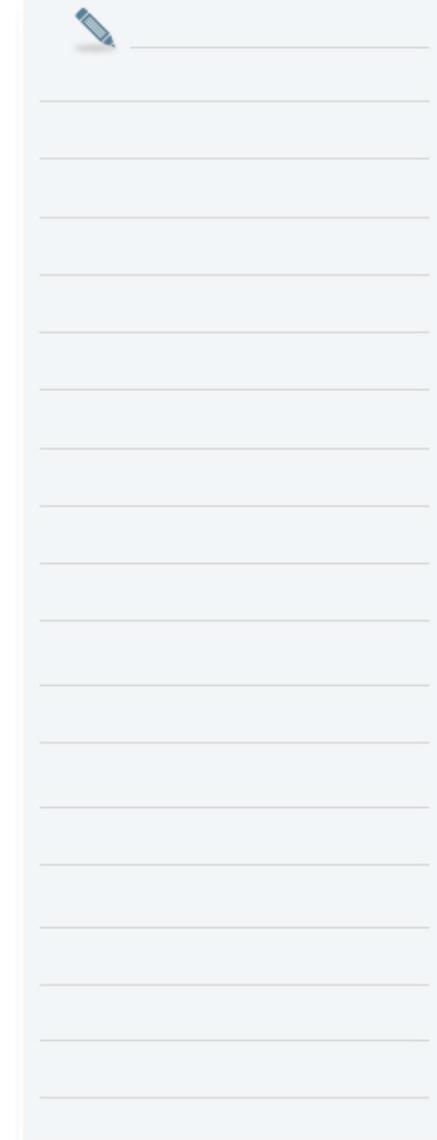
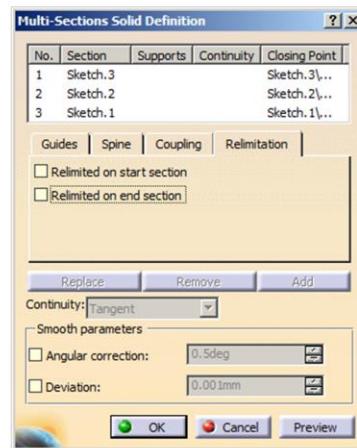
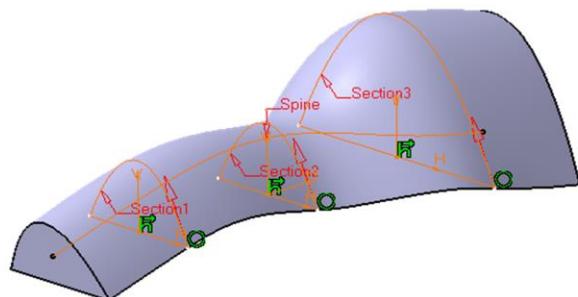
For example, when a multi-sections solid is created using three sections and the Relimited options are selected, the feature will be limited by the start and end sections.



Relimiting Multi-Sections Solid (2/3)

When the Relimitation options are cleared, the feature will be limited by either the Spine or a Guide Curve, whichever is the shortest.

For example, a multi-sections solid is created using three sections with a Spine that extends past the first and last sections. If the **Relimited on start section** and **Relimited on end section** options are cleared, the feature will extend past the start and end sections of the multi-sections solid to the start and endpoints of the spine.

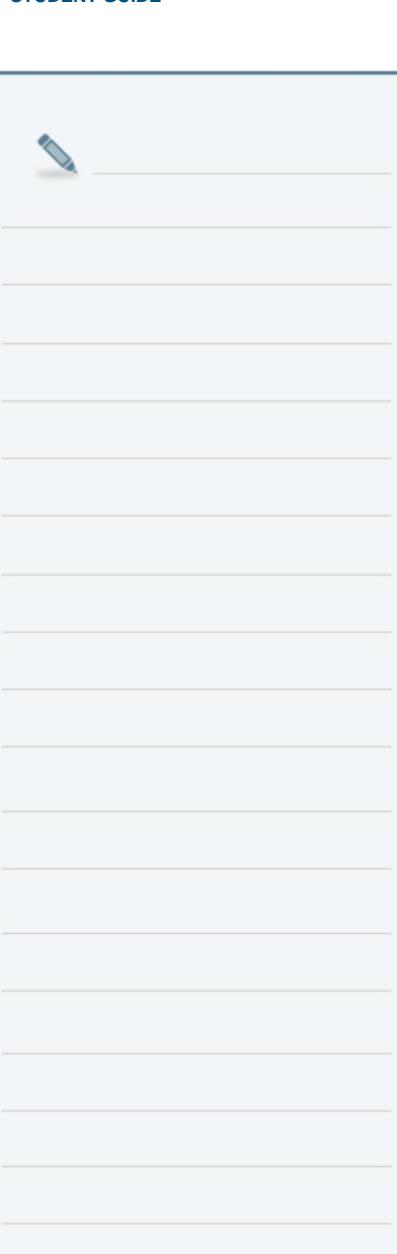
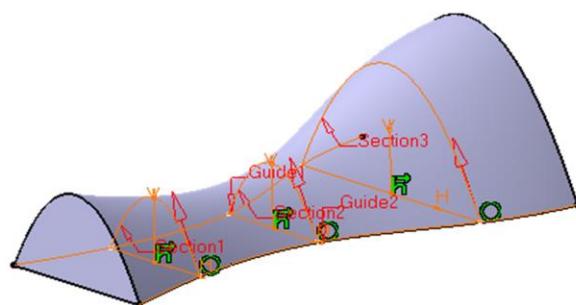
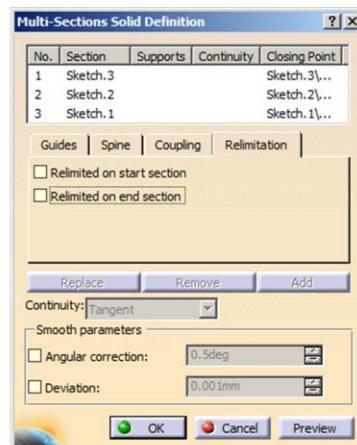


Relimiting Multi-Sections Solid (3/3)

If a user-defined Spine and Guides are defined, the feature will be limited by the shortest curve.

For example, a multi-sections solid is constructed through three sections using a Spine and Guide curves to control the transitions surfaces.

If the Relimited options are cleared, the feature will be limited by the shortest curve. In this example, the shortest guide will limit the feature.



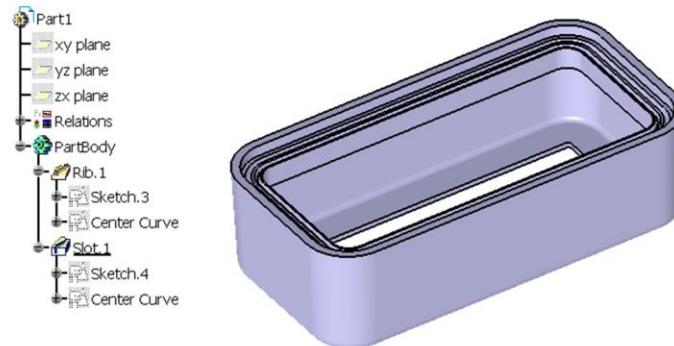


Exercise: Create a Rib and Slot

In this exercise, you will create a new model and use the tools learned in the lesson to create a rib and a slot feature. High-level instruction is provided for this exercise.

By the end of this exercise you will be able to:

1. Create a Rib Feature
2. Create a Slot Feature



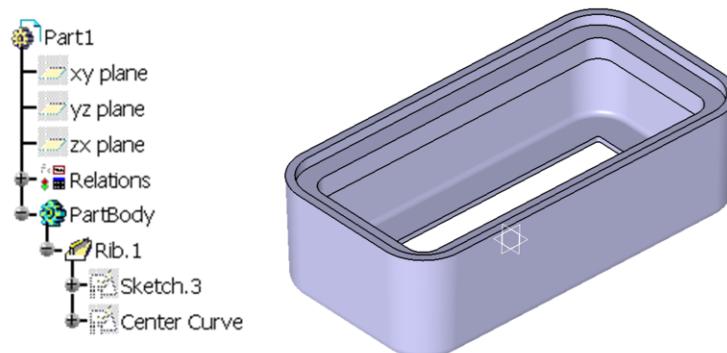
15 minutes





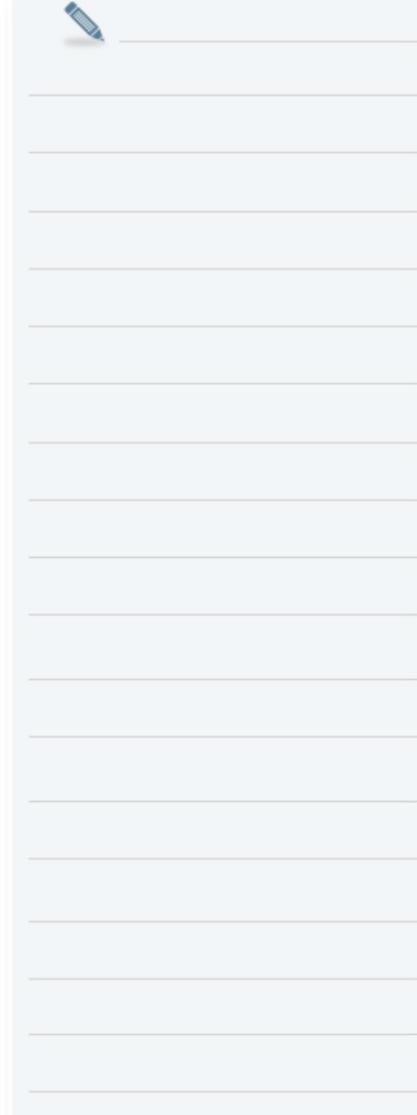
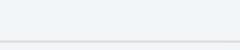
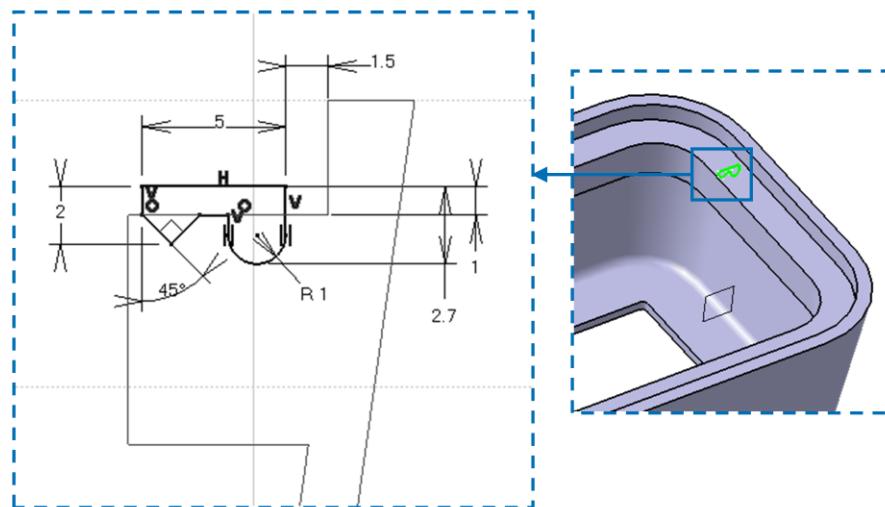
Create a Rib Feature

1. Open the Ex8B.CATPart.
2. Create the rib feature.
 - a. Use the center curve and profile sketch to create a rib feature.



Create a Sketch

1. Create a profile sketch for the slot.
 - a. Create a positioned sketch as shown for the slot profile.

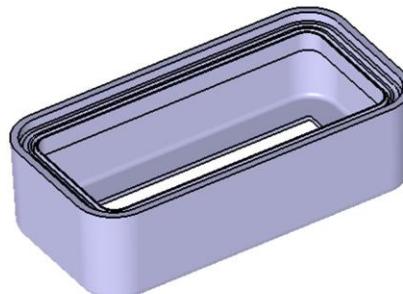
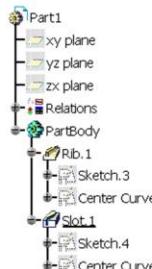




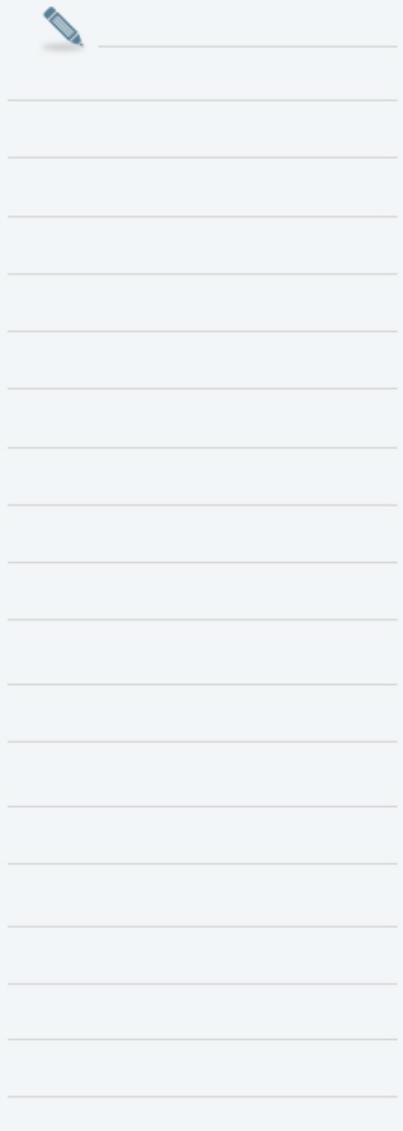
Create a Slot Feature

1. Create a slot feature.

Create a slot feature using the sketch created in the last step as the profile and the Center Curve sketch as the trajectory.



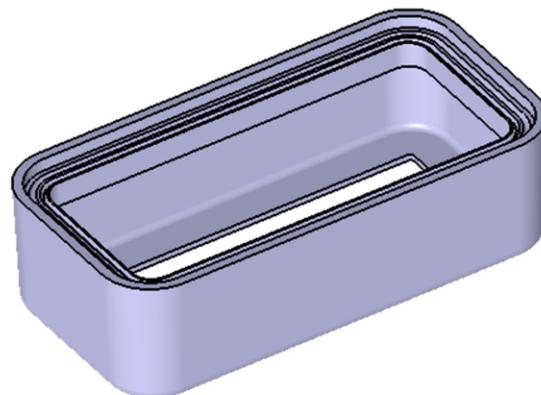
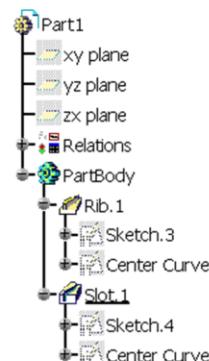
2. Close the file without saving it.



Recap: Create a Rib and Slot

In this exercise you have:

- Created a rib
- Created a slot



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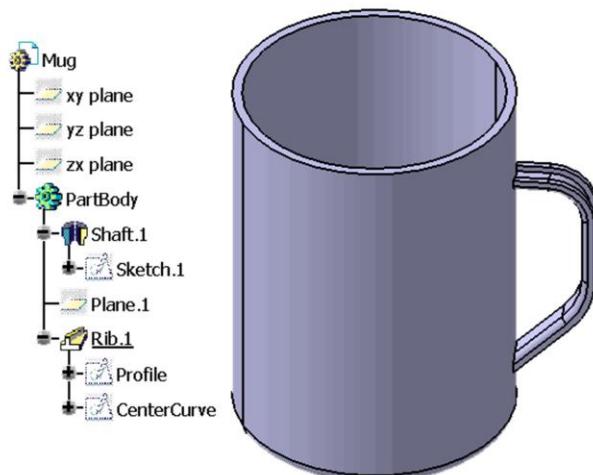


Exercise: Create a Thin Rib

In this exercise, you will open an existing model and use the tools learnt in the lesson to create a rib feature that will attach to the existing geometry. You will create the profiles for the rib feature. Detailed instructions are provided for the new topics present in this exercise.

By the end of this exercise you will be able to:

- 1.Create a thin Rib\Feature
- 2.Use Merge Ends Option



15 minutes



Create a Thin Rib (1/4)

1. Open the part file.

Open Mug.CATPart.

2. Sketch the rib center curve.

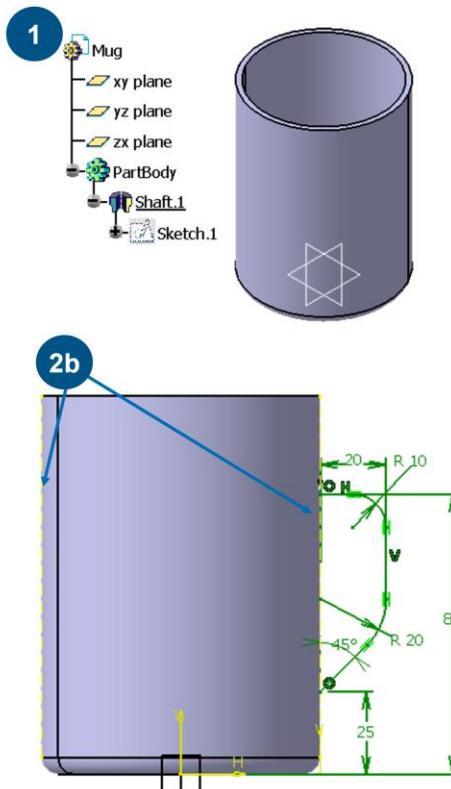
Sketch the center curve for the handle rib feature.

Create a sketch on the yz plane.

Project the silhouette edges of the mug onto the sketch support using Construction Elements.

Sketch and constrain the profile.

Once complete, exit sketcher and rename the sketch to [Center Curve].



Create a Thin Rib (2/4)

3. Create a reference plane.

Create a reference plane that will be used to sketch the rib profile.

Select the **Plane** icon.

Select the **Normal to curve** Plane type.

Select the CenterCurve sketch.

Select the point shown.

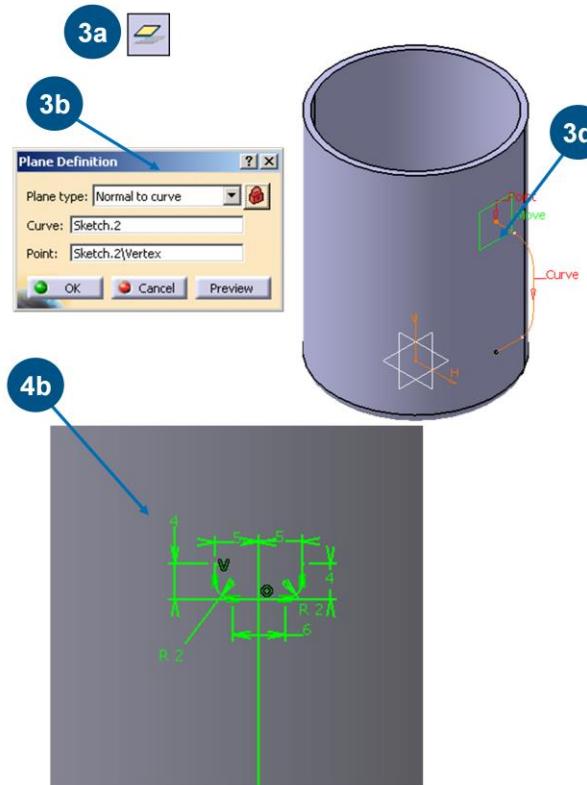
4. Sketch the rib profile.

Sketch the profile for the handle rib feature. The rib will use the **Thick Profile** option so that the profile remains open.

Create a sketch on Plane.1.

Sketch and constrain the profile. Be sure to make the horizontal line element coincident with the CenterCurve sketch.

Rename the sketch to [Profile].



Create a Thin Rib (3/4)

5. Create a thin rib.

Create a rib feature using the Profile and Center Curve sketches you have just created.

Select the **Rib** icon.

Select the **Thick Profile** option.

Select the Profile sketch for the **Profile**.

Select the CenterCurve sketch for the **Center curve**.

Select **Keep angle** from the Angle control pull-down menu.

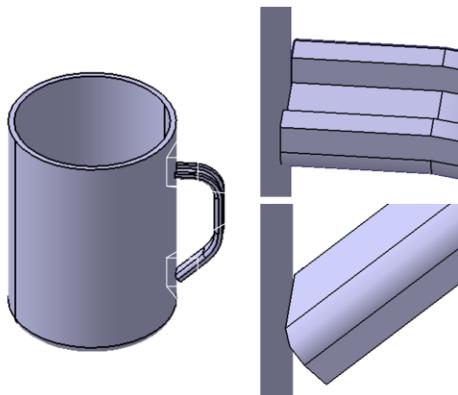
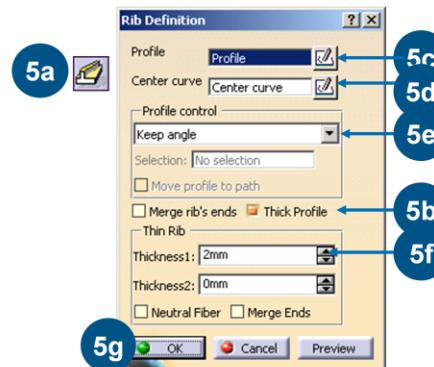
Enter [2mm] for Thickness1.

Click **OK**.

6. Investigate the rib feature.

Click **Tools > Hide > All Planes**.

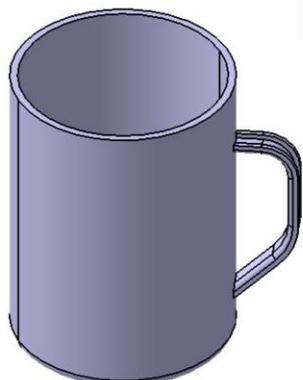
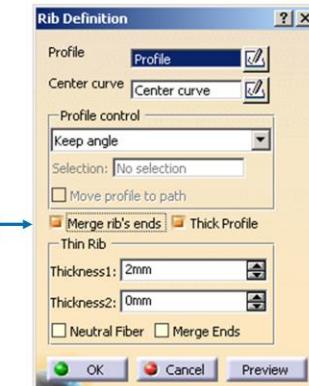
Zoom in on the ends of the rib feature. Due to the shape of the mug and the angle of the center curve, the rib feature does not attach properly to the mug surface.





Create a Thin Rib (4/4)

7. Modify the rib feature.
 1. Modify the rib feature to use the Merge rib's ends option so that the rib will properly attach to the mug.
Modify Rib.1.
Select the **Merge rib's ends** option.
Click **OK**.
8. Save and close the file.

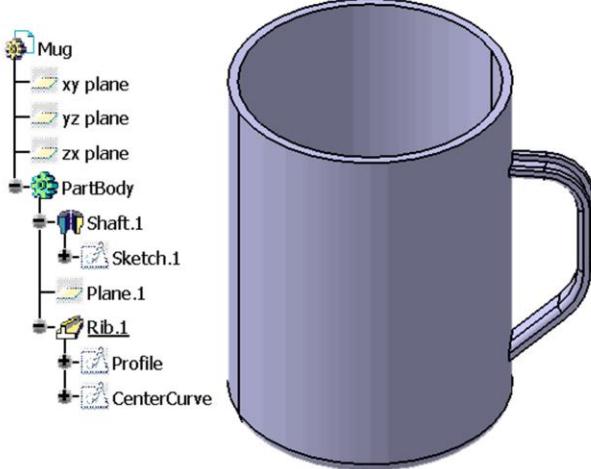




Recap: Create a Thin Rib

In this exercise you have:

- Created a thin rib feature
- Used the Merge Ends option

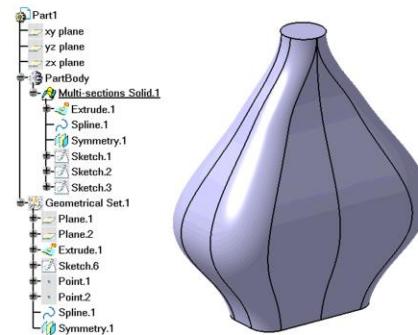




Exercise: Create a Multi-Sections Feature

In this exercise, you will open an existing model and use the tools learnt in this lesson to create a multi-sections solid. These solids will be created using the existing sketches and 3D wireframe and surface elements. Detailed instructions are provided for the new topics present in this exercise.

By the end of this exercise you will be able to create a Multi-Section Solid Feature



20 minutes



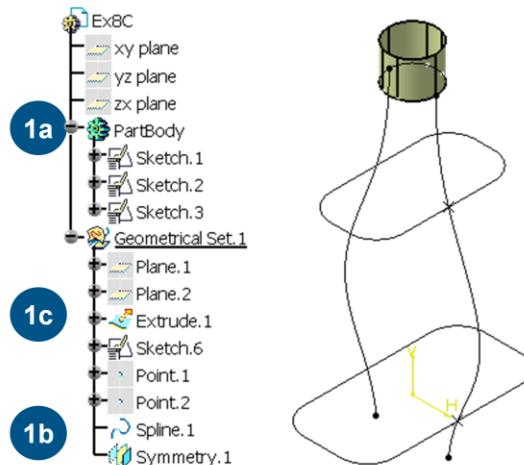


Create a Multi-Sections Feature (1/12)

1. Load Ex8C.CATPart.

Load Ex8C.CATPart.

- a. Notice the sketches in the PartBody. These three sketches are the profiles for the multi-sections solid.
- b. Notice the Spline and symmetry feature in Geometrical Set.1. These features are the guides for the feature.
- c. Notice the extruded surface in Geometrical Set.1. The multi-sections solid has to be tangent to this surface.

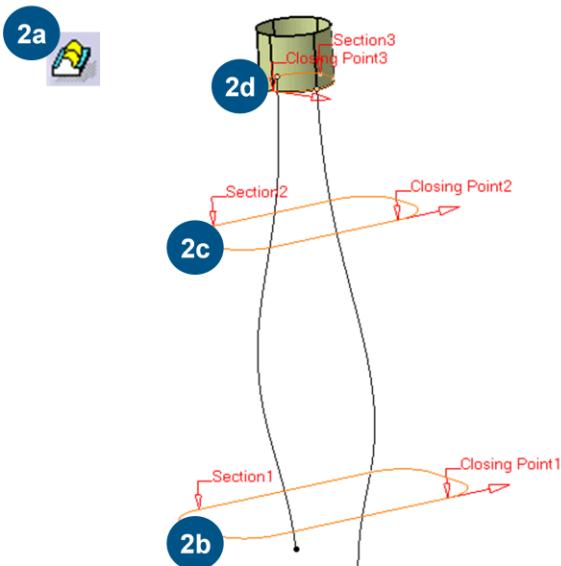


Create a Multi-Sections Feature (2/12)

2. Create multi-sections solid.

Create a simple multi-sections solid.

- Select the multi-sections solid icon.
- Select Sketch.1as the first profile.
- Select sketch.2 as the second profile.
- Select sketch.3 as the third profile.

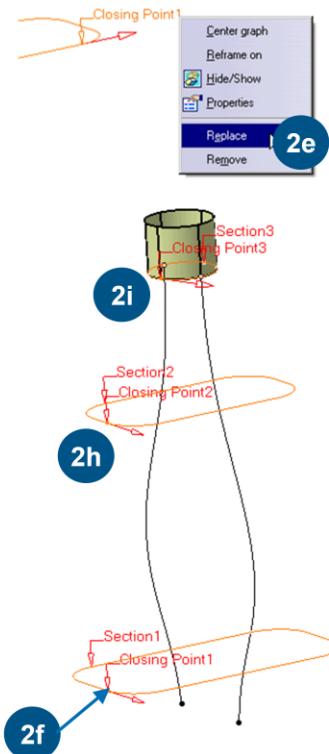


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Create a Multi-Sections Feature (3/12)

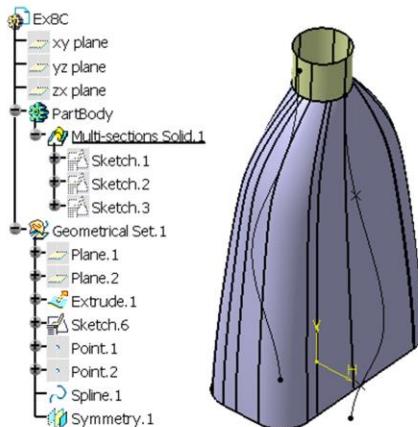
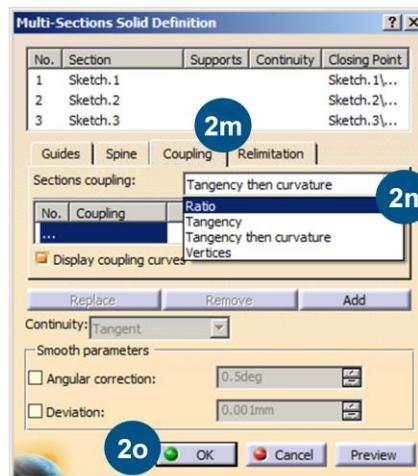
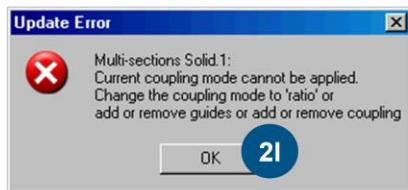
2. Create multi-sections solid (continued).
 - e. Right-click on the Closing Point for the first profile and click **Replace** from the contextual menu.
 - f. Select the vertex shown.
 - g. Ensure that the directional arrow for the first closing point is correct. If needed, click on the arrow to change its direction
 - h. Move the closing point of the second profile to the vertex shown.
 - i. Ensure that the closing point for the third profile is in the correct location and direction.





Create a Multi-Sections Feature (4/12)

2. Create multi-sections solid (continued).
 - j. Click **OK**.
 - k. An update error occurs. Read the error. Why did the feature fail?
 - l. Click **OK** to the Update Error.
 - m. Select the Coupling tab.
 - n. From the Sections coupling pull-down select 'Ratio'.
 - o. Click **OK** to generate the feature.



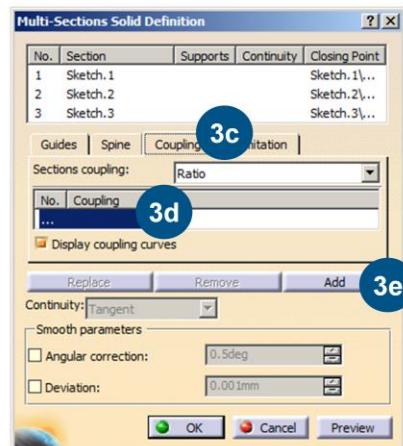


Create a Multi-Sections Feature (5/12)

3. Redefine the multi-sections solid.

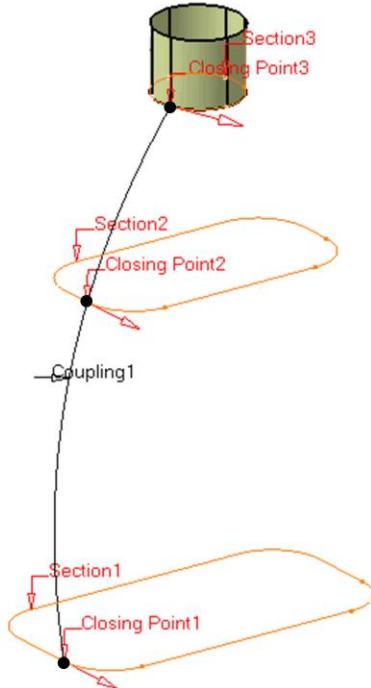
Currently, the feature is coupled based on a ratio, change this to specific locations by manually coupling the feature.

- a. Show Sketch.1, Sketch.2, and Sketch.3.
- b. Double-click the multi-sections solid from the specification tree or directly on the model to redefine the feature.
- c. Select the **Coupling** tab.
- d. Click inside the Coupling field to activate the Add button.
- e. Select **Add**.



Create a Multi-Sections Feature (6/12)

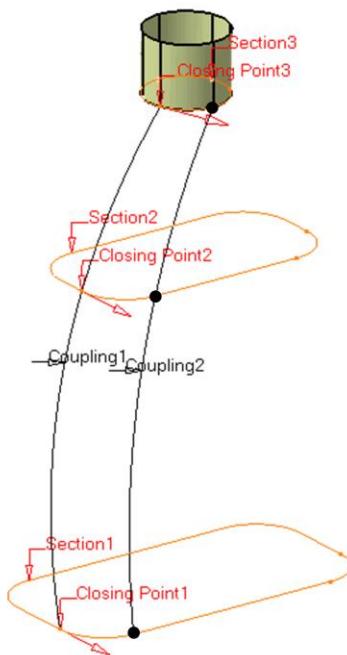
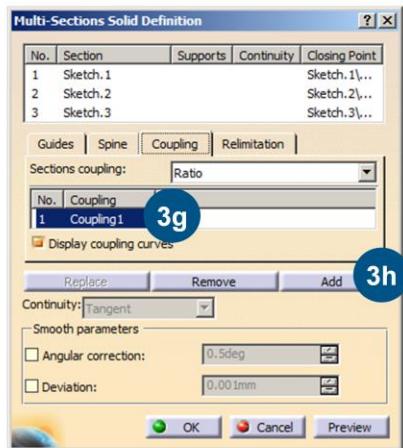
3. Redefine the multi-sections solid (continued).
 - f. Select the vertices shown. It is important to select the vertices in order (i.e., select the vertex from profile 1, then profile 2, then profile 3). This coupling connects the closing points of all three sections.



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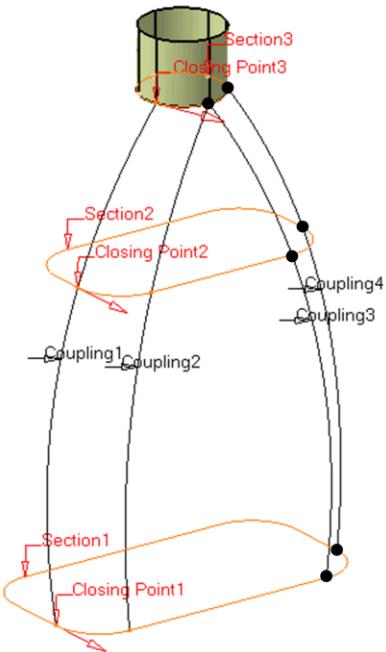
Create a Multi-Sections Feature (7/12)

3. Redefine the multi-sections solid (continued).
 - g. Click inside the coupling field to re-activate the **Add** button.
 - h. Select the **Add** button.
 - i. Create a second coupling as shown.
Remember to select the vertices in the correct order.



Create a Multi-Sections Feature (8/12)

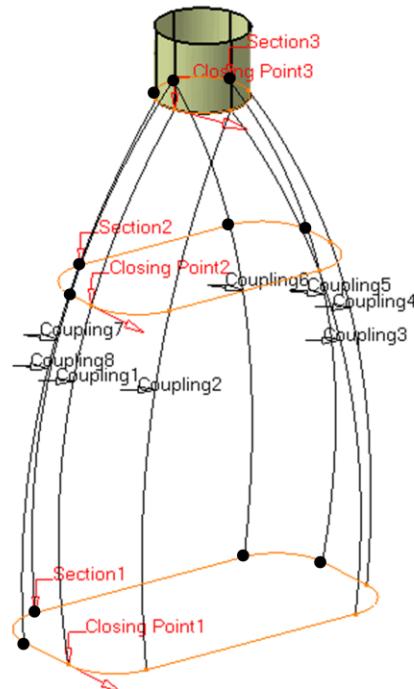
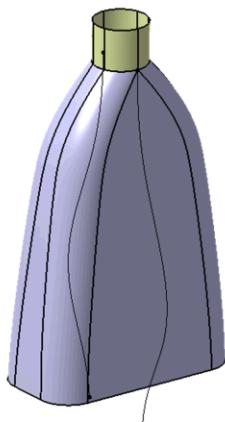
3. Redefine the multi-sections solid (continued).
 - j. Create the coupling for the second corner as shown.



Handwritten notes or sketches can be made here.

Create a Multi-Sections Feature (9/12)

3. Redefine the multi-sections solid (continued).
 - k. Couple the vertices for the last two corners using the same technique as the front corners.
 - l. Click **OK** to confirm the changes.



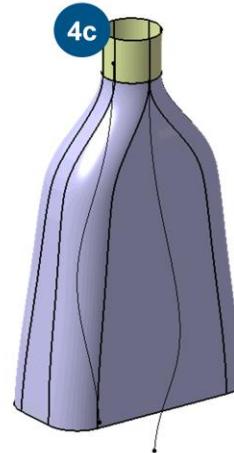


Create a Multi-Sections Feature (10/12)

4. Apply Tangency.

Redefine the feature to apply tangency to the third profile.

- a. Double-click on the multi-sections solid to edit its definition.
- b. Select the third profile from the profile window.
- c. Select the extrude surface. The feature is now tangent to this surface.
- d. Click **OK** to apply the changes.



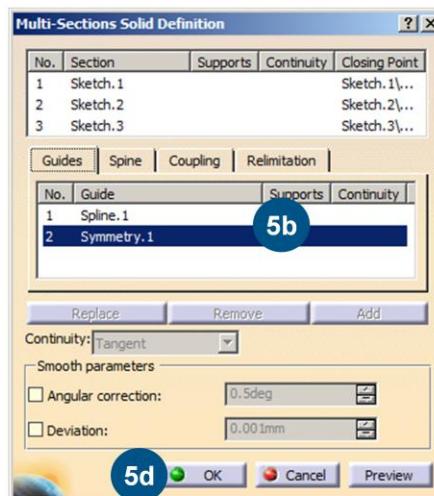


Create a Multi-Sections Feature (11/12)

5. Add Guides.

Redefine the feature and apply guides to define the shape of the multi-sections solid between the sections.

- a. Double-click on the multi-sections solid to edit its definition.
- b. Select in the Guides window.
- c. Select Spline.1 and Symmetry.1 as the guides.
- d. Click OK to apply the changes.





Create a Multi-Sections Feature (12/12)

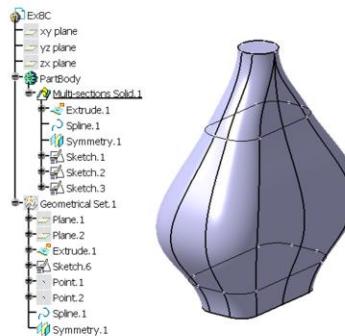
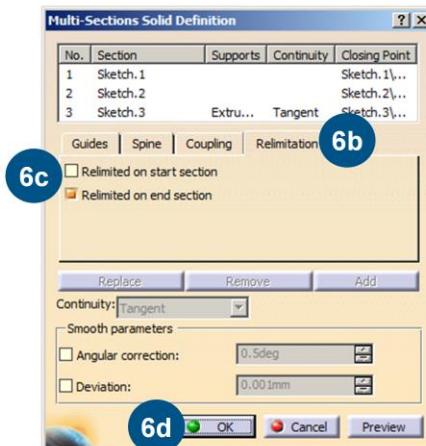
6. Change the relimitation options.

Redefine the feature and change the feature so that it begins at the start of the guide lines and not the first profile.

- Double-click multi-sections solid to edit its definition.
- Select the Relimitation tab.
- Clear the **Relimited on start section** option.
- Click **OK** to apply the changes.

7. Close the file without saving it.

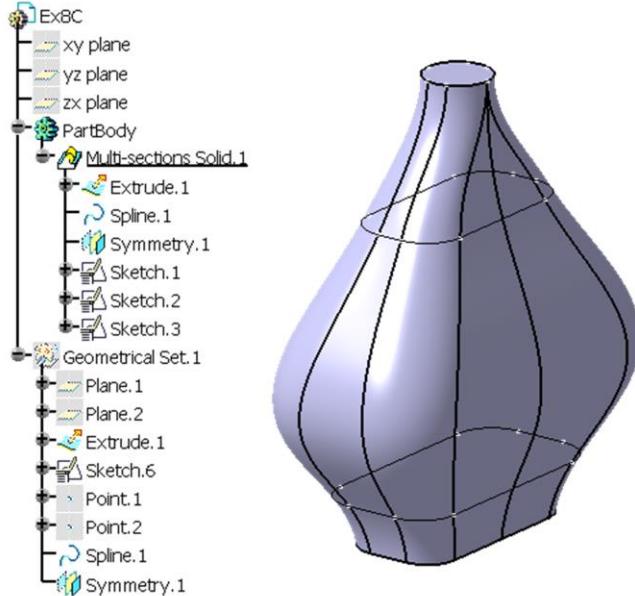
Hide Geometrical Set.1 and close the file without saving it.



Recap: Create a Multi-Sections Feature

In this exercise you have:

- Created multi-sections solid feature



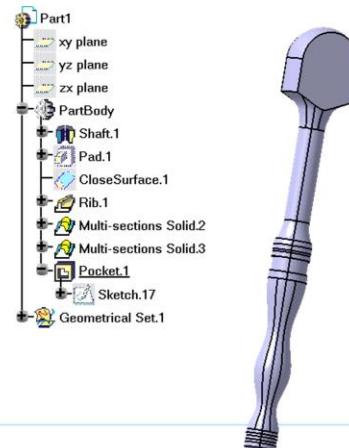


Exercise: Create Rib and Multi-section Solid

In this exercise, you will open an existing model and use the tools learnt in the lesson to create rib and Multi-sections Solid features. High-level instruction is provided for this exercise.

By the end of this exercise you will be able to:

1. Create a Rib Feature
2. Create a Multi-sections Solid Feature

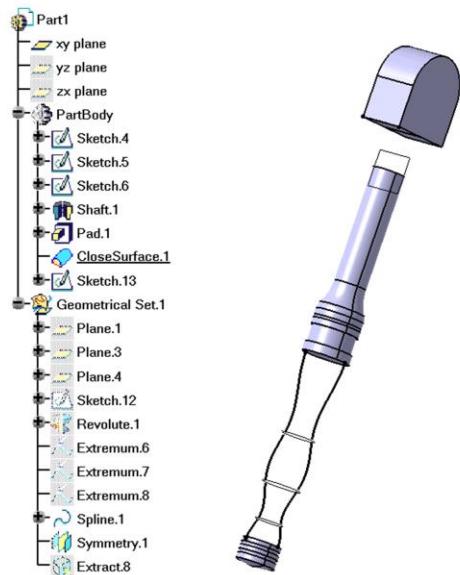


15 minutes



Open the Part

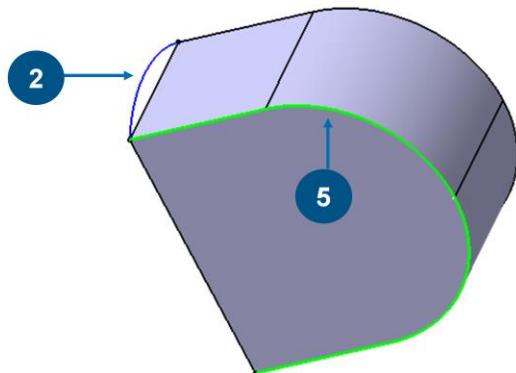
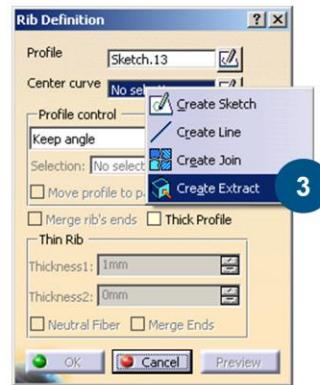
1. Open Wrench.CATPart. Notice some features have already been created.



Create a Rib (1/2)

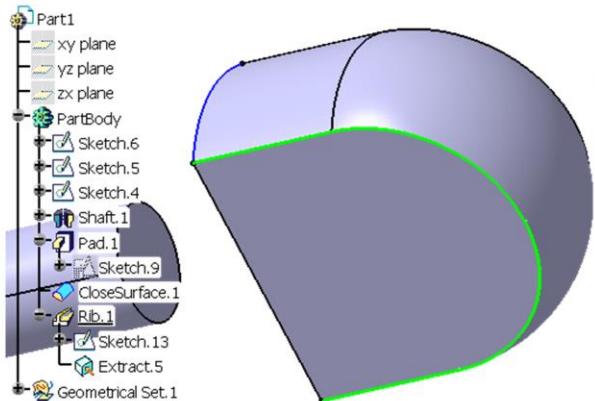
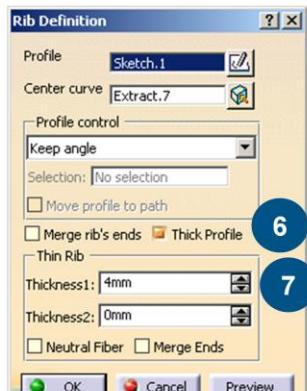
Use Sketch.13 as the profile for a rib feature.

1. Access the **Rib Definition** dialog box.
2. Select *Sketch.13* as the profile.
3. Right-click the **Center Curve** field.
4. Select **Create Extract** from the contextual menu.
5. Select the edge shown.



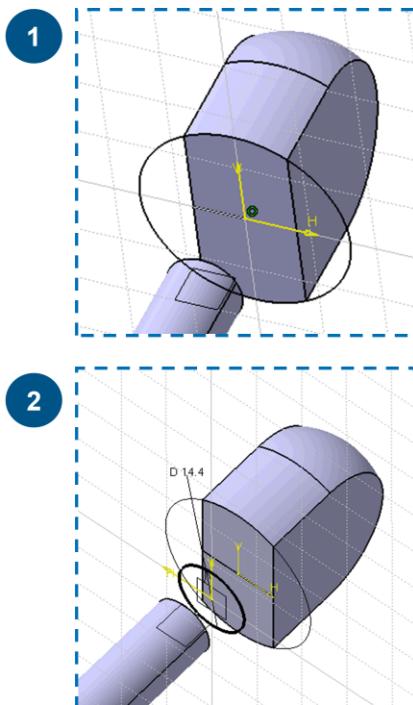
Create a Rib (2/2)

6. Select the **Thick Profile** option.
7. Type [4 mm] in the **Thickness1** field.
8. Complete the feature.



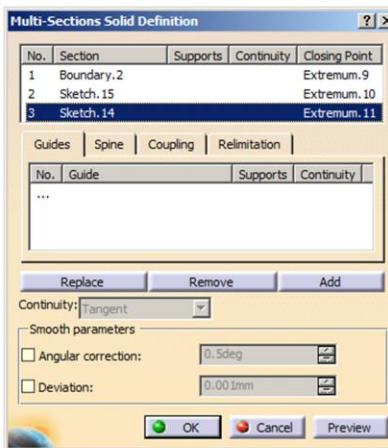
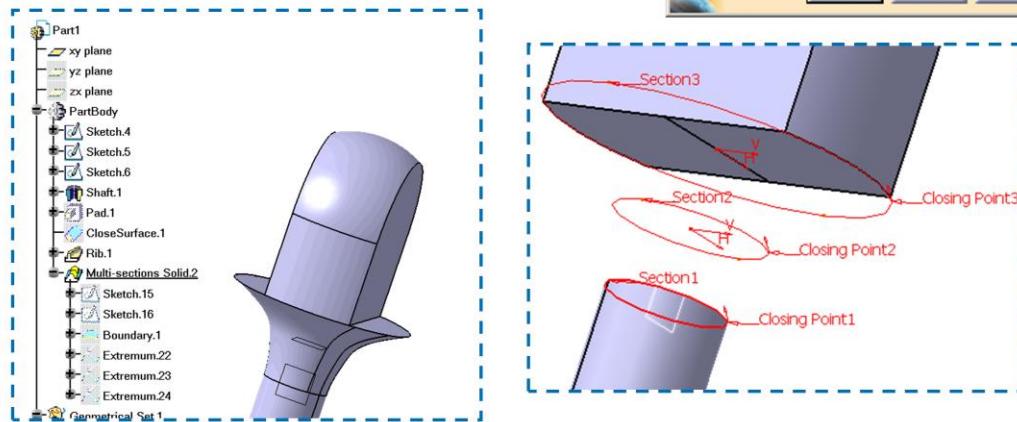
Create Profiles for a Multi-Section Solid

1. Create the profile as shown using the lower face of the pad as the sketch support.
2. Create a second profile for the Multi-sections solid.
 - a. Create a reference plane offset [7mm] from the lower surface of the pad.
 - b. Create the sketch shown using the reference as the sketch support. The diameter of the sketched circle is [14.4mm].



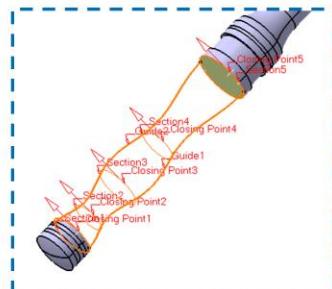
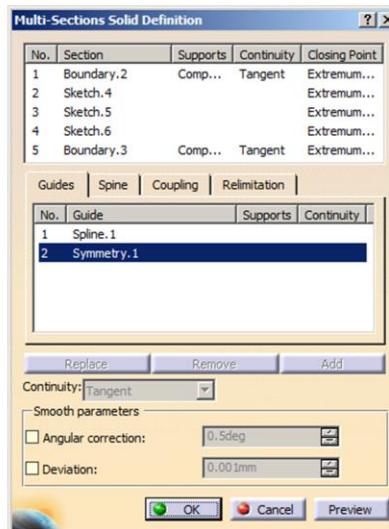
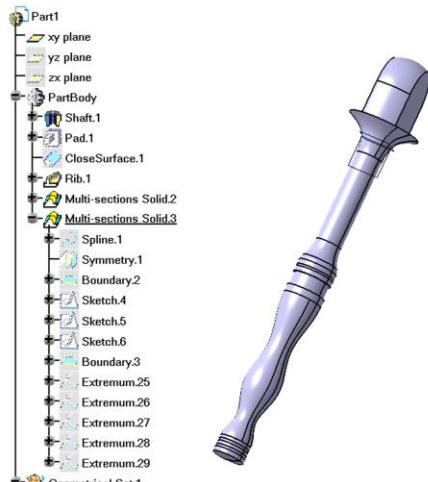
Create Multi-Section Solid (1/2)

1. Use the profiles and the lower surface of the shaft feature as the profiles for the feature.
Notice that the feature is automatically tangent to the shaft.



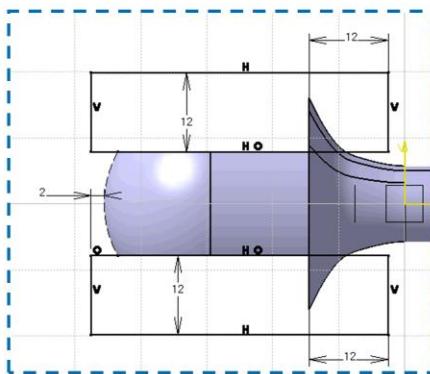
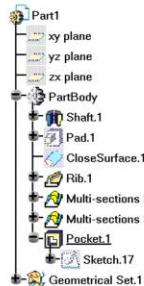
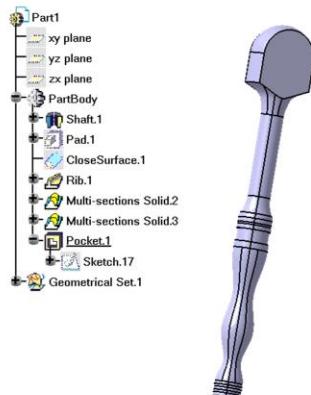
Create Multi-Section Solid (2/2)

2. Create a second multi-sections solid to complete the handle.
3. Use appropriate surface of the shaft, sketch.4, sketch.5, and sketch.6 as the profiles. Use Spine.1 and Symmetry.1 as guide curves for the feature.



Finish the Design

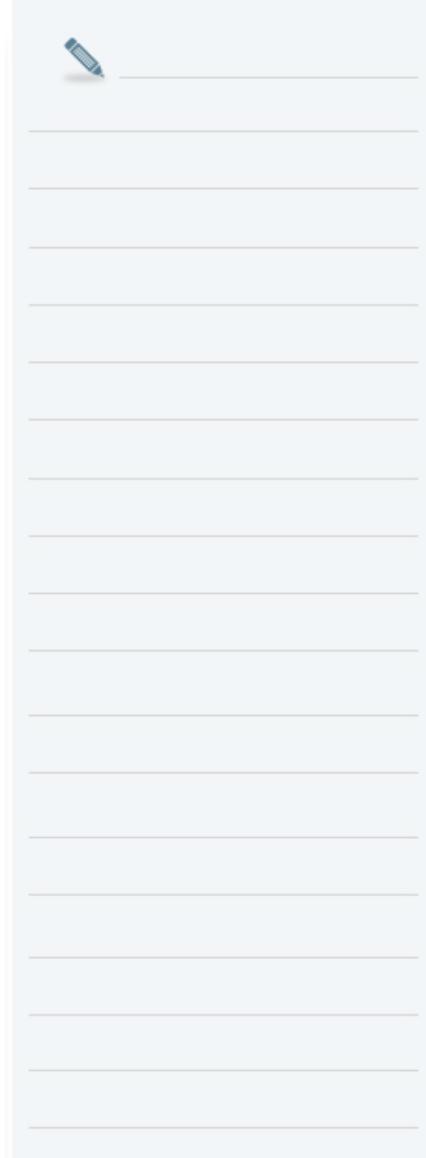
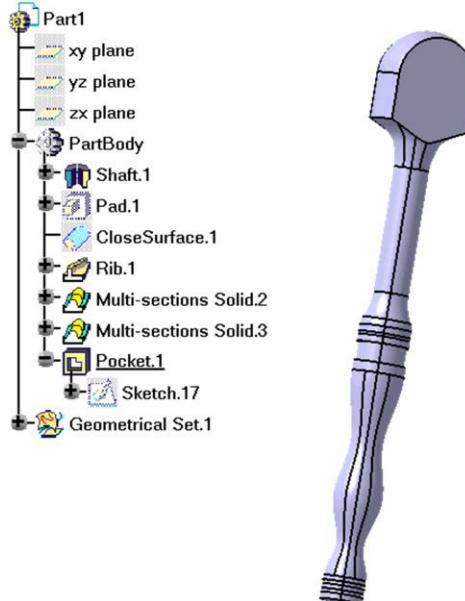
1. Create a pocket feature.
 - a. Create a pocket feature to trim away the excess material from the top of the wrench. Use the XY plane as the sketch support for the pocket feature.
2. Clarify the display, save, and close the model.
 - a. Hide all wireframe and surface elements. Save and close the model.



Recap: Create Rib and Multi-section Solid

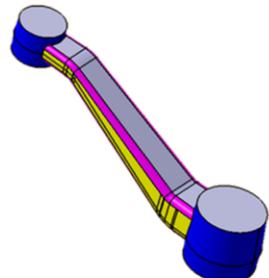
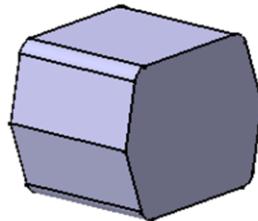
In this exercise, you have:

- Created a rib
- Created a multi-section solid





Creating Advanced Drafts



Here are the topics to be covered:

- 1. Creating Advanced Sketch-Based Features
- 2. Creating Multi Section solids
- 3. **Creating Advanced Drafts**
- 4. Creating Advanced Dress-Up features
- 5. Using the Multi-Body Method
- 6. Creating Multi-Model Links



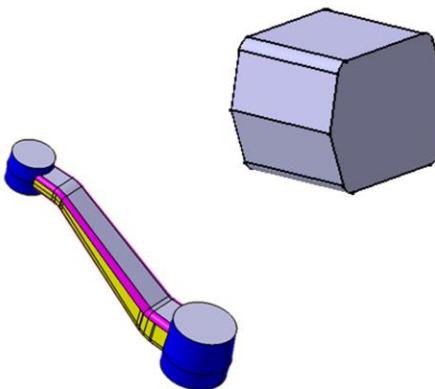
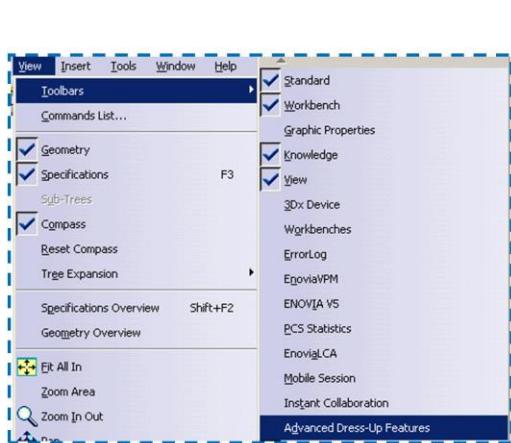


Introduction to Advanced Drafts (1/2)

The Advanced Drafts tool allows you to add complex draft angles to existing solids.

Advanced Drafts can be used to create basic and reflect line drafts as well as drafts with two different angle values for complex parts.

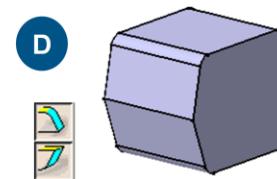
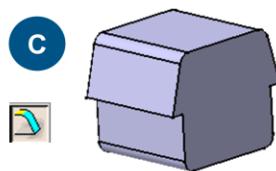
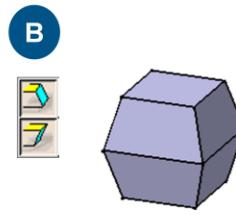
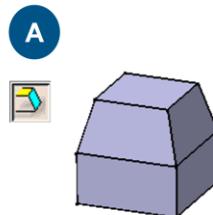
By default, the Advanced Dress-Up Features toolbar is not displayed in the Part Design workbench. To display the Feature on the toolbar, click **Views > Toolbars > Advanced Dress-up Features**.



Introduction to Advanced Drafts (2/2)

Using the Advanced Draft tool you can create:

- A. A standard 1st side draft
- B. A standard 2nd side draft
- C. A draft using a reflect line
- D. A draft using two reflect lines



Select the appropriate button(s) at the top of the Advanced Draft definition dialog box to create a draft.



Creating an Advanced Draft (1/5)

Use the 1st Side tab to define the characteristics of the draft angle for the selected faces.

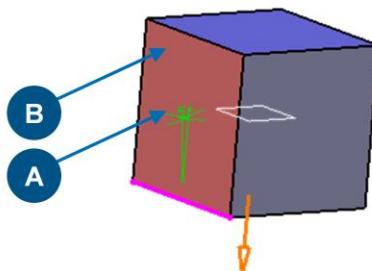
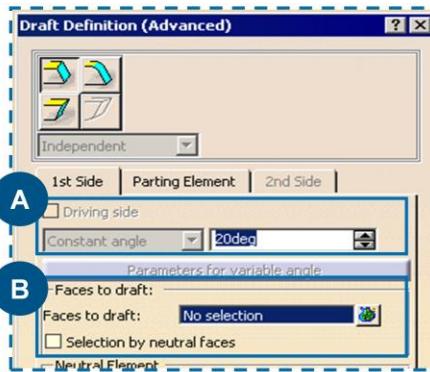
The following 1st side characteristics must be defined:

A. Draft angle

The draft angle is an angle that the draft faces make with the pulling direction from the neutral element. This angle may be defined for each face.

B. Faces to draft:

These are the surfaces where the draft will be applied.

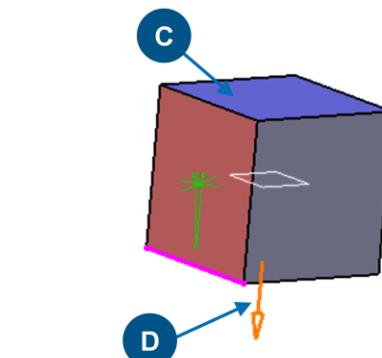
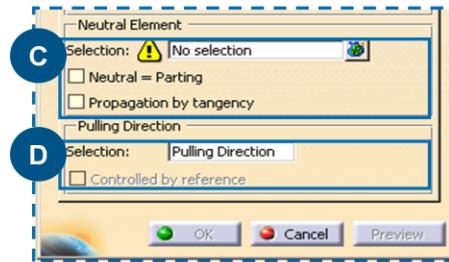


Creating an Advanced Draft (2/5)

The following 1st side characteristics must be defined
(continued):

C.Neutral Element

- The Neutral Element is used to define the pivot hinge for the drafted surfaces. The drafted surfaces pivot about a neutral curve, the hinge, where it intersects the Neutral Element. The Neutral Element, usually a plane or face, can be the same reference that is used to define the pulling direction.



D.Pulling Direction

- The Pulling Direction defines the direction from which the draft angle is measured. It derives its name from the direction in which the sides of a mold are pulled to extract the molding.
- Using Advanced Draft, both sides of a face can be drafted to achieve different pulling directions.



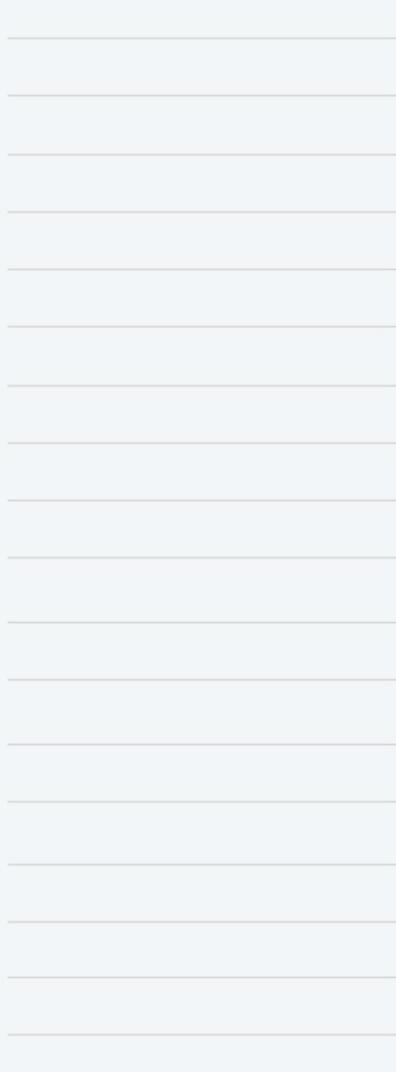
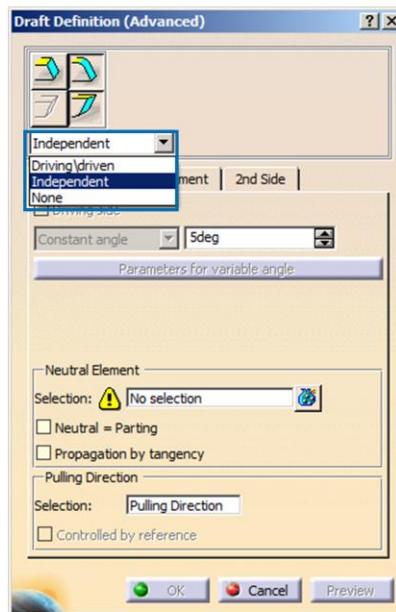


Creating an Advanced Draft (3/5)

While creating a two-sided draft using a reflect line, the Dependency menu becomes available.

This menu enables you to define the dependency of the draft angle.

- ▶ With the **Independent** option, draft is created where both the 1st & 2nd side draft angles must be defined.
- ▶ With the **Driving\driven** option, the angle specified for the driving side controls the angle specified for the driven side.



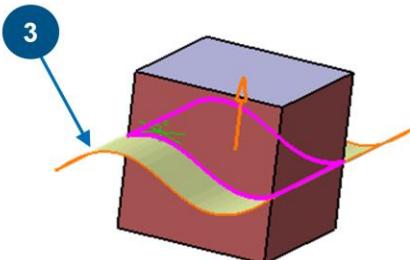
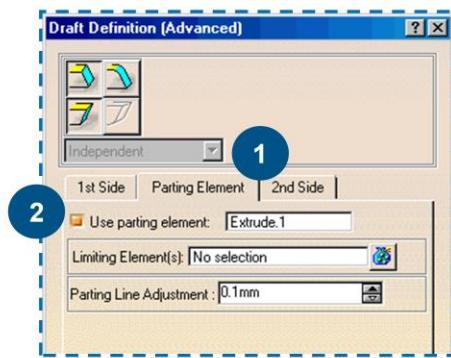
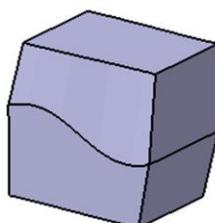
Creating an Advanced Draft (4/5)

A Parting Line represents the location where two halves of a mold meet.

Use the following steps to define a Parting Element:

1. Select the **Parting Element** tab.
2. Select the **Use parting element** option from the Parting Element tab.
3. Select the parting element from the model.

The parting element can be a plane, a surface, or a face.



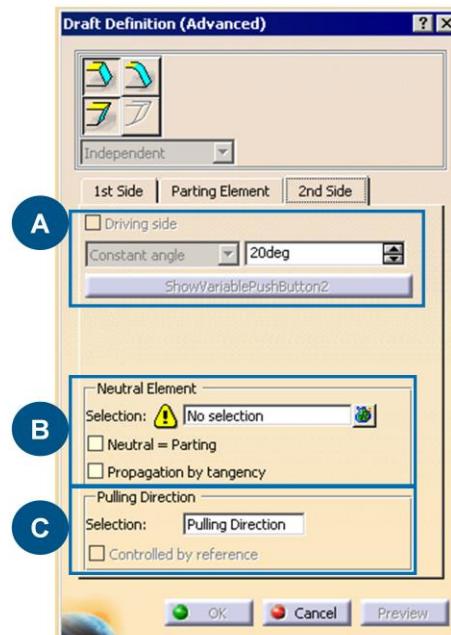


Creating an Advanced Draft (5/5)

To define a second draft angle, select the appropriate 2nd Side option from the dialog box and from the 2nd side tab, define the second draft.

Many of the options necessary to define the 2nd Side of the draft are the same as those that defined the 1st Side of the draft.

- A. Draft Angle Value
- B. Neutral Element
- C. Pulling Direction



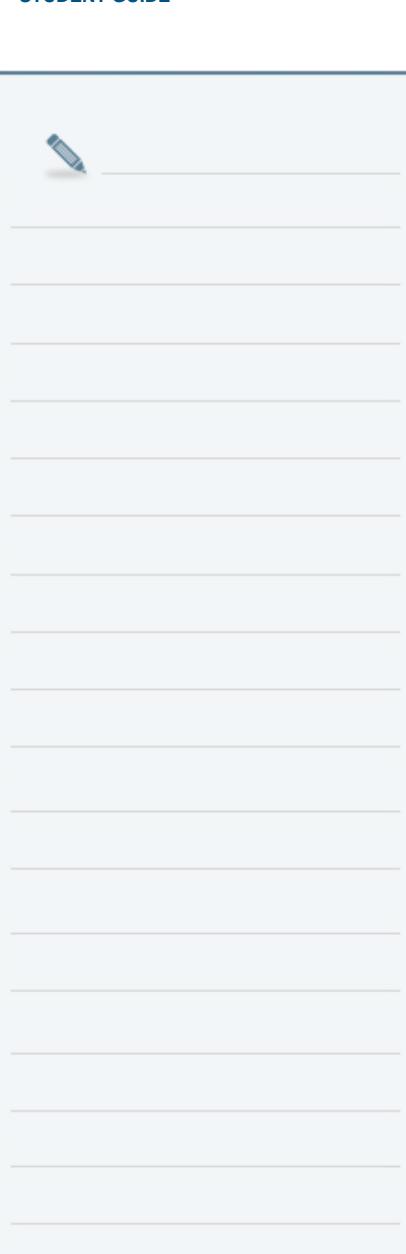
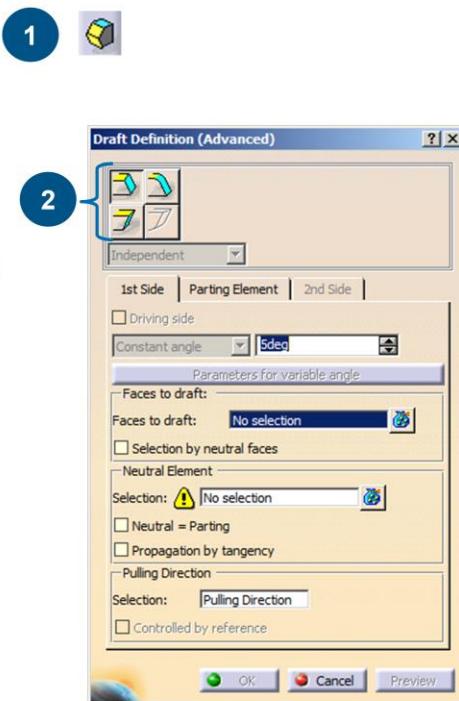


Creating a Draft on Both Sides (1/5)

In the following example, a standard two-sided draft is created.

Use the following steps to create an Advanced Draft feature:

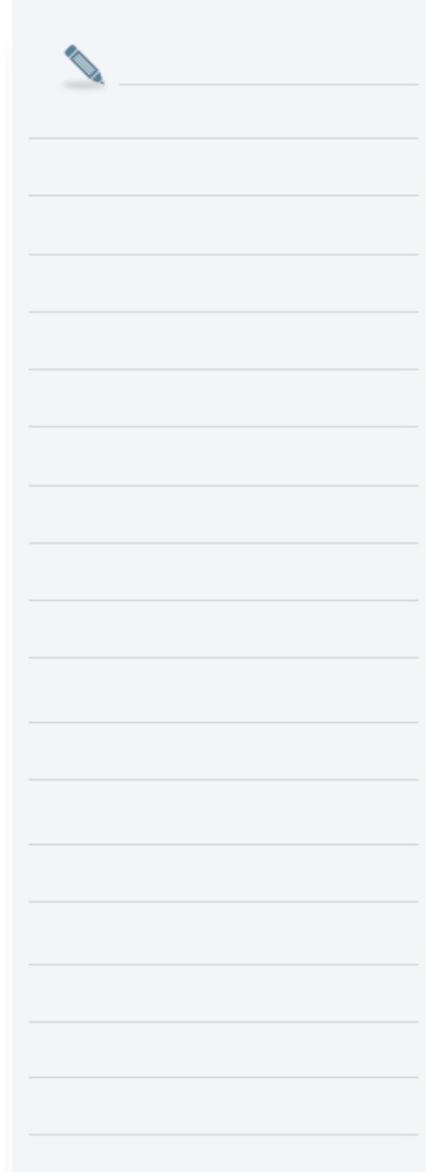
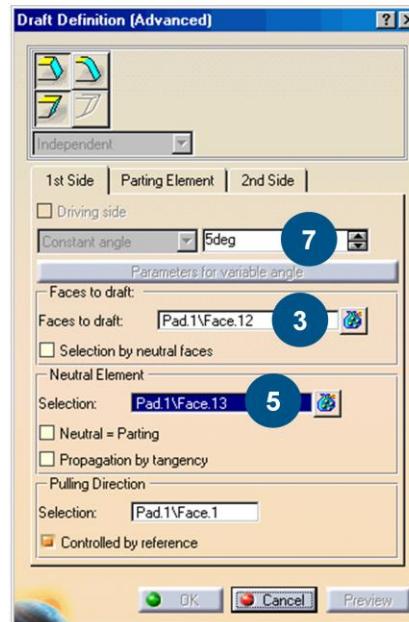
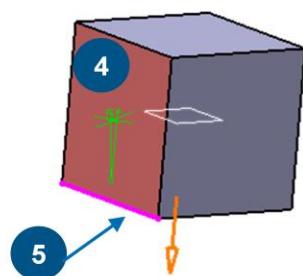
1. Click the **Advanced Draft** icon.
2. Activate the **Standard Draft (1st Side)** and **Standard Draft (2nd Side)** options.



Creating a Draft on Both Sides (2/5)

Use the following steps to create an Advanced Draft feature (continued):

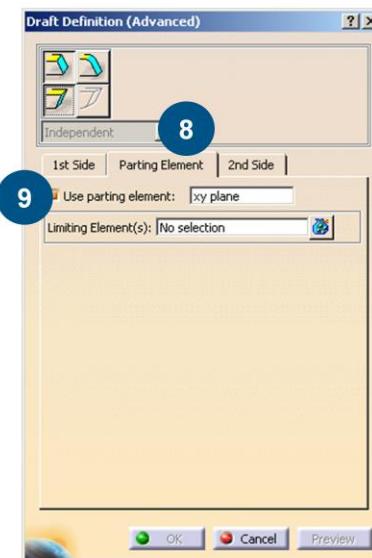
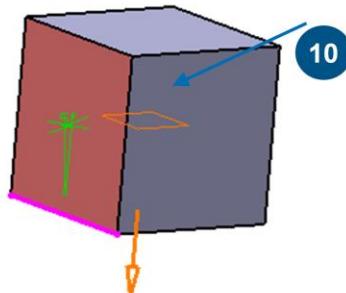
3. Select the **Faces to draft** selection field
4. Select the faces to be drafted.
5. Select the **Neutral Element** selection field.
6. Select the Neutral Element(s).
7. Enter the draft angle for the first side.



Creating a Draft on Both Sides (3/5)

Use the following steps to create an Advanced Draft feature (continued):

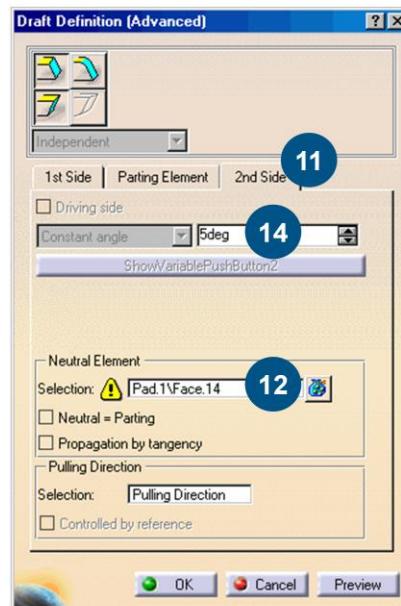
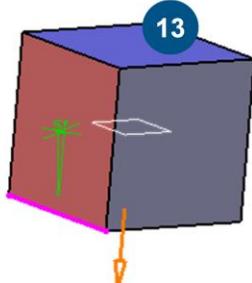
8. Select the **Parting Element** tab.
9. Select the **Use Parting Element** option.
10. Select the parting element from the model.



Creating a Draft on Both Sides (4/5)

Use the following steps to create an Advanced Draft feature (continued):

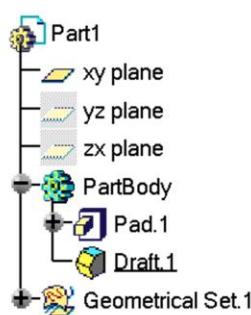
11. Select the 2nd side tab.
12. Select in the **Neutral Element** field.
13. Click the Neutral Element(s) for the second side.
14. Enter the draft angle for the second side.



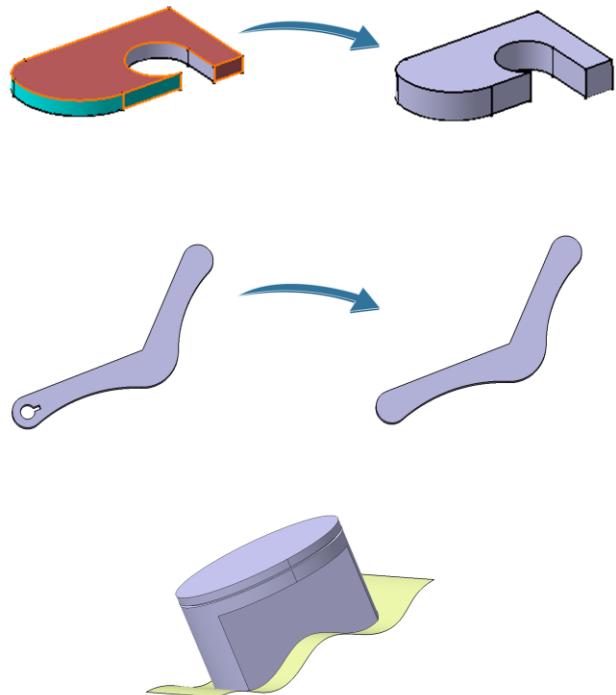
Creating a Draft on Both Sides (5/5)

Use the following steps to create an Advanced Draft feature (continued):

15. Click **Preview**.
16. Click **OK** to generate the draft.



Creating Advanced Dress-Up Features



Here are the topics to be covered:

- 1. Creating Advanced Sketch-Based Features
- 2. Creating Multi Section solids
- 3. Creating Advanced Drafts
- 4. **Creating Advanced Dress-Up features**
- 5. Using the Multi-Body Method
- 6. Creating Multi-Model Links



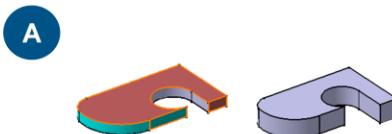


Introduction to the Advanced Dress-Up Features

The following tools allow you to dress-up existing solids.

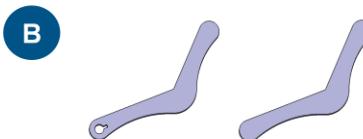
A. Thickness:

Use this tool to add a thickness to a face.



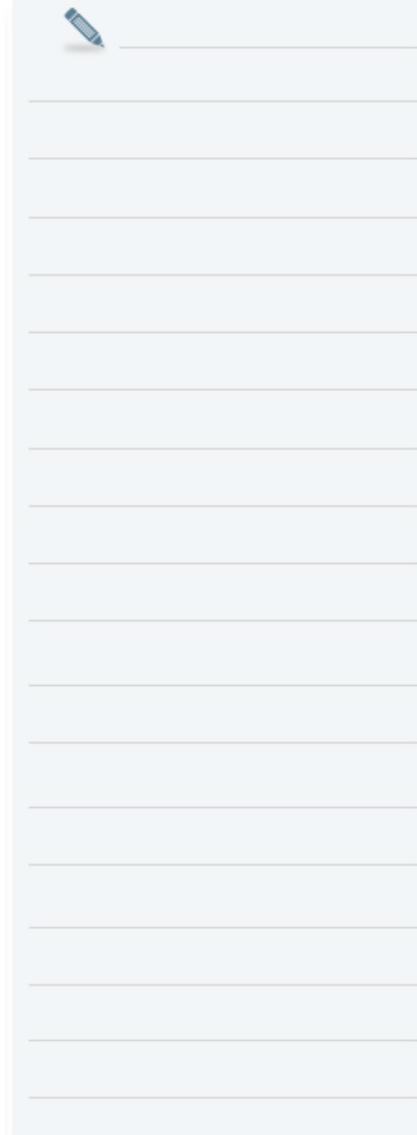
B. Remove faces:

Use this tool to simplify the geometry of a part for a down stream processes.



C. Replace a face with a surface:

Use this tool to replace a planar solid surface with a surface.



About Thickness (1/2)

Thickness is applied to a model to enhance productivity during solid model creation.

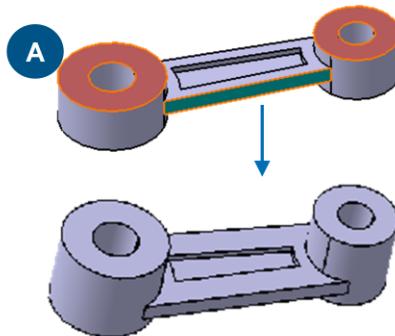
The thickness feature is often used to add or remove material before machining a part. Thickness enhances the design intent and allows for rapid modifications.

Material can be quickly added or removed from various faces of a part to accommodate machining or other manufacturing operations. For instance, you might add thickness to account for additional material necessary to cast the part.

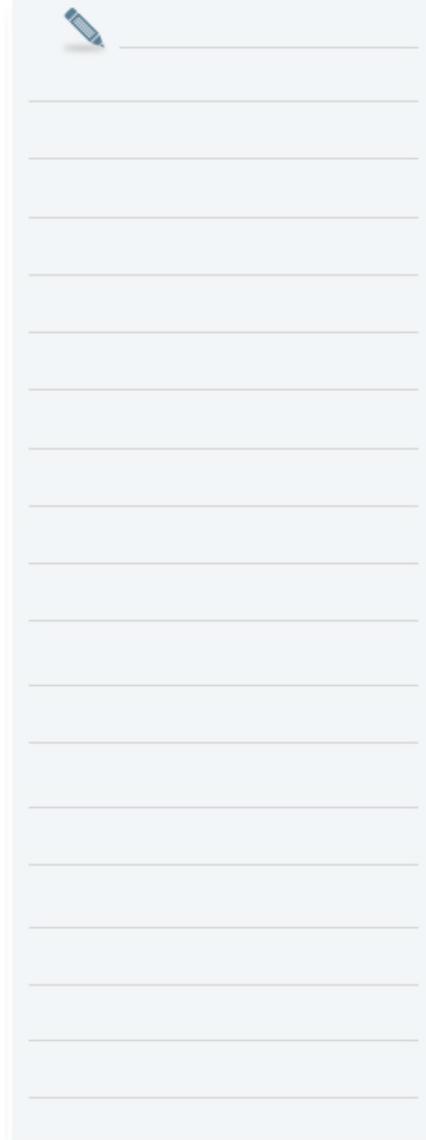
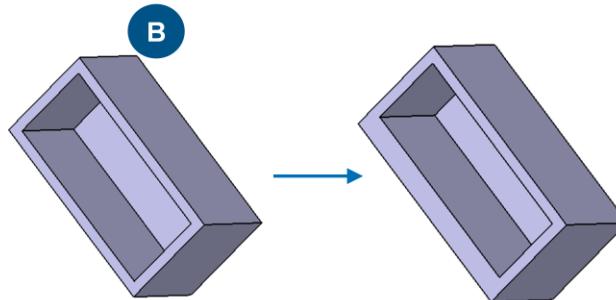


About Thickness (2/2)

- A. The **Thickness** tool adds material to the pad while considering the other features. Use the **Thickness** tool adding material be done quickly and efficiently.



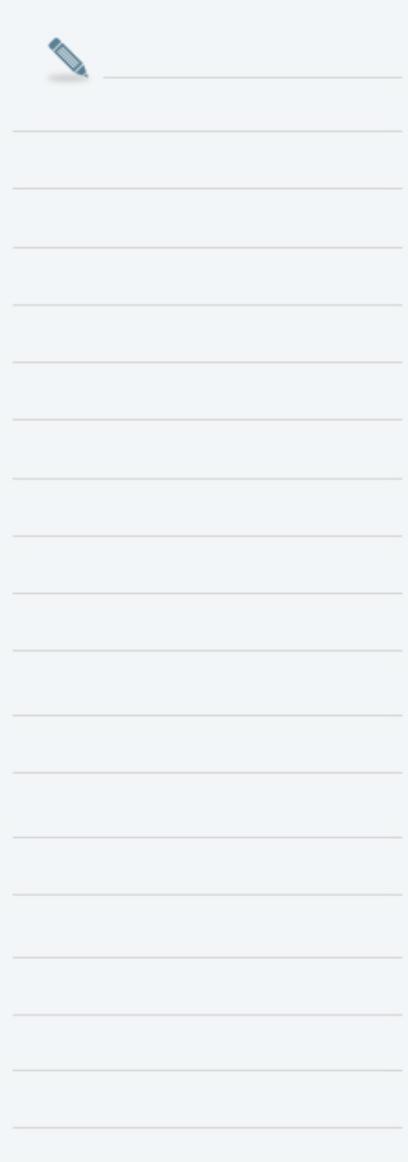
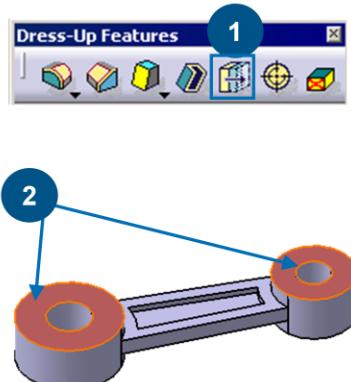
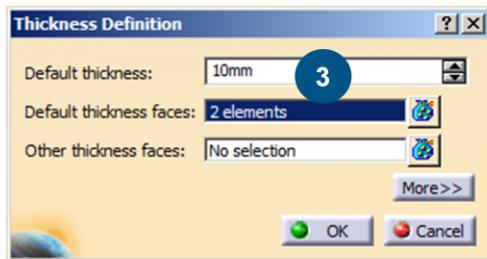
- B. Another common use of the **Thickness** tool is to apply thickness to select walls of a model that has been shelled.



Creating Thickness (1/2)

Use the following steps to apply thickness to a model:

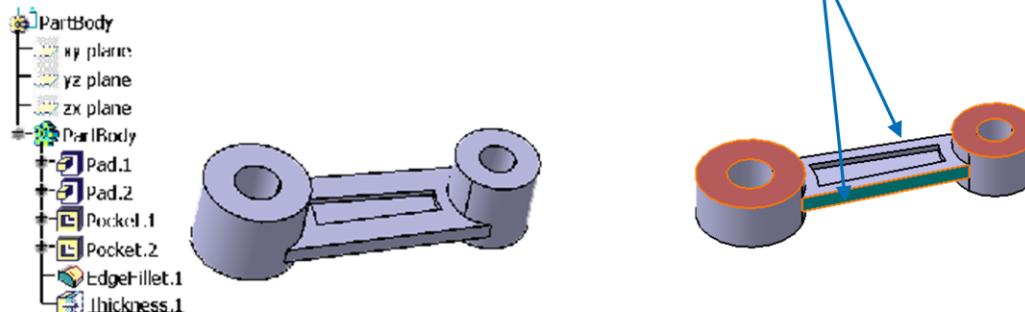
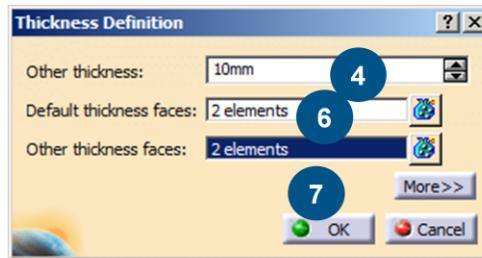
1. Select the **Thickness** icon from the Dress-up Features toolbar.
2. Select the faces to which thickness has to be applied.
3. Enter the thickness value.



Creating Thickness (2/2)

Use the following steps to apply thickness to a model
(continued):

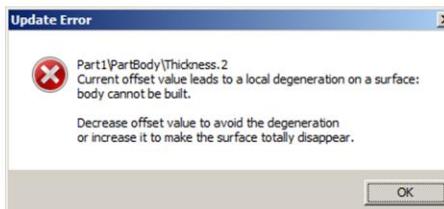
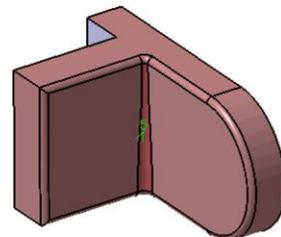
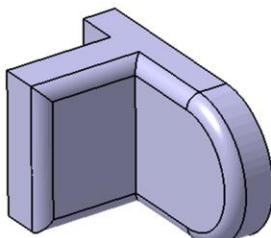
4. Select in the Other thickness field.
5. Select the faces to which a different thickness value will be applied.
6. Enter the thickness value for those faces.
7. Click **OK**.



Ignoring Faces While Creating Thickness

In some cases, when you apply a thickness, an error message appears indicating that some of the bodies cannot be built properly. After closing the window, another message appears prompting you to ignore the problem faces. If you select **Yes**, the thickness is created and the face causing the issue is removed.

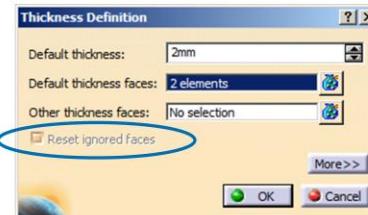
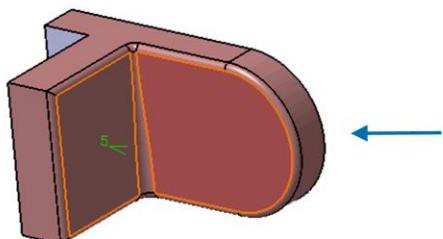
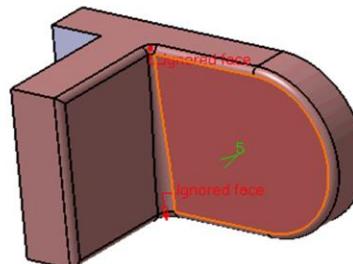
For example, if the inside face of the model shown in the top image on the right-hand side is offset, an error message will appear. CATIA is unable to offset the filleted surface. Select **Yes** to create the thickened body as shown in the bottom image on the right-hand side.



Resetting Ignored Faces Option for Thickness Tool

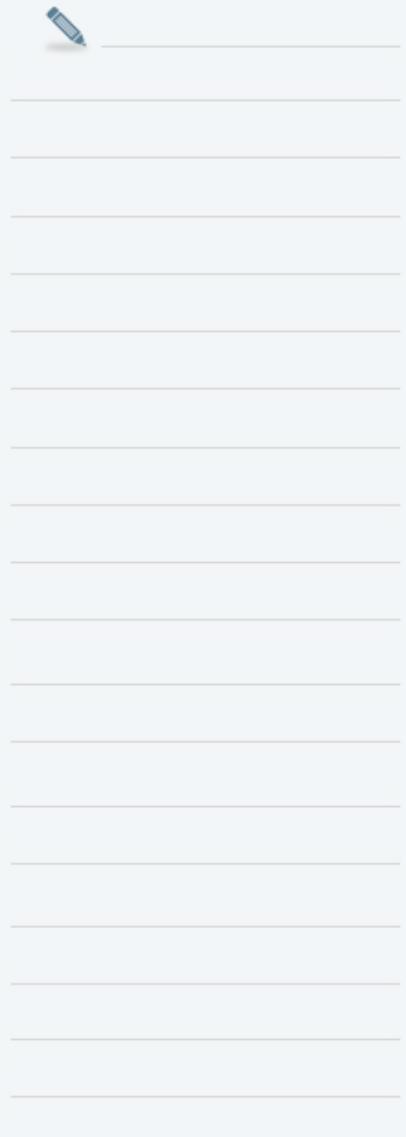
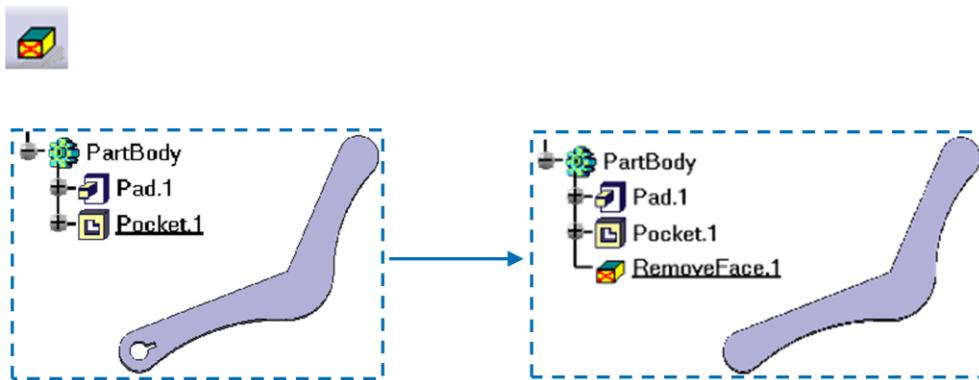
If a thickness feature has been created with some faces ignored, the ignored faces are previewed when you edit the thickness from the specification tree, as shown in the top image on the right-hand side.

The option **Reset ignored Faces** appears in the **Thickness Definition** Dialog box. After selecting this option, the ignored faces are reinitialized and the Ignored Face note is removed from the geometry.



Removing Faces (1/2)

To simplify the part for a finite element analysis, you can remove some of its faces or features to simplify the geometry using the **Remove Face** tool.

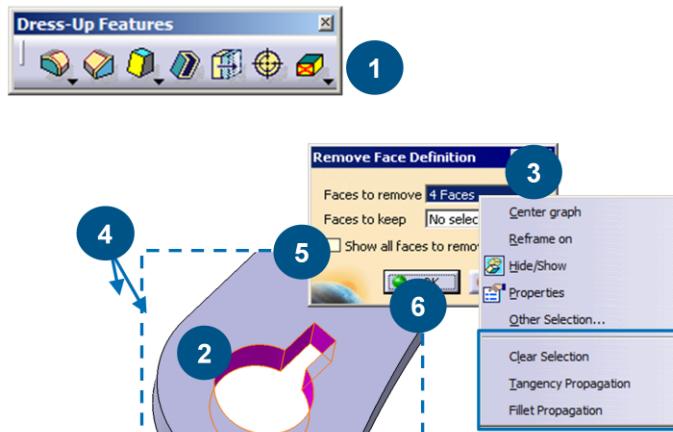




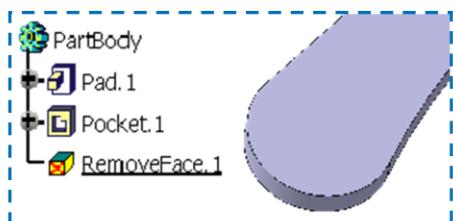
Removing Faces (2/2)

Use the following steps to remove faces:

1. Select the **Remove Face** icon.
2. Select the internal faces you want to remove.
3. Select in the Faces to keep field.
4. Select the faces to be kept.
5. Select the **Show all faces to remove** option to preview all faces that will be removed during the operation.
6. Click **OK** to complete the feature.
The selected faces are removed and a new feature is added to the specification tree.



Contextual menu of Faces to remove field allows to select tangency propagation option which automatically removes all faces tangent to the selected ones.



If the Remove Face operation is unsuccessful, you can deactivate it by selecting Yes in the message box displayed. Later in your design, you can edit it to provide the valid input element.

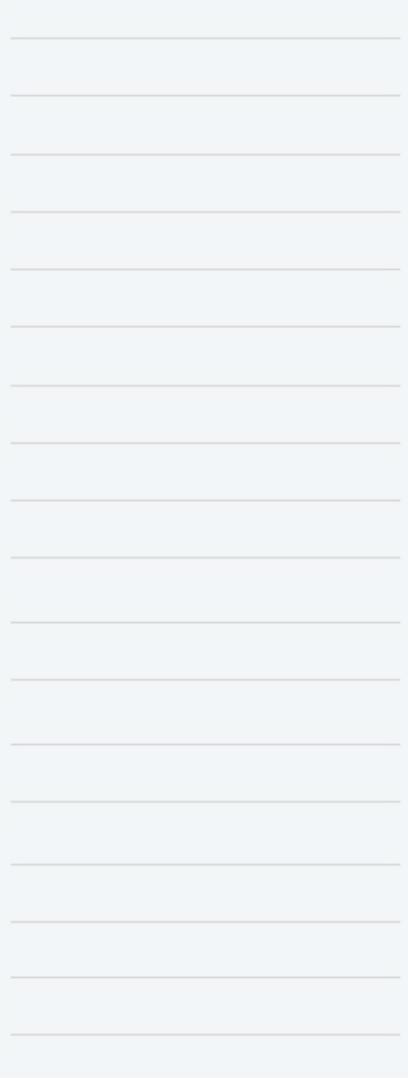
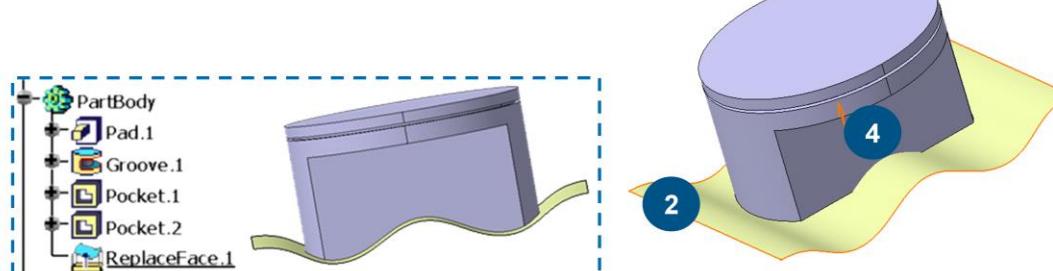
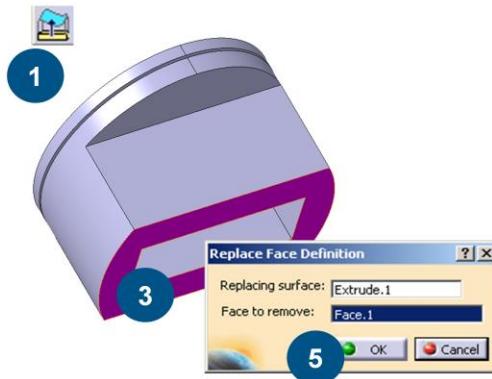


Replacing a Face

The **Replace Face** tool is used to extrude a solid face up to a surface.

Use the following steps to extrude a solid face up to a surface:

1. Select the **Replace Face** icon.
2. Select the replacing surface.
3. Select the face you want to extrude.
4. Ensure that the arrow points in the direction of the kept material. Click on the arrow to change its direction.
5. Click **OK** to complete the feature.

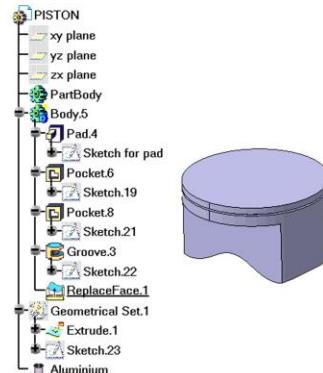




Exercise: Replace a Model Face

In this exercise, you will open an existing part that contains a solid model and a surface feature. You will use the Replace Face tool to extrude the solid model to the surface. Detailed instructions are provided for this exercise.

By the end of this exercise you will be able to replace the Model face



05 minutes

Replace a Model Face (1/2)

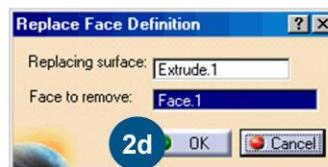
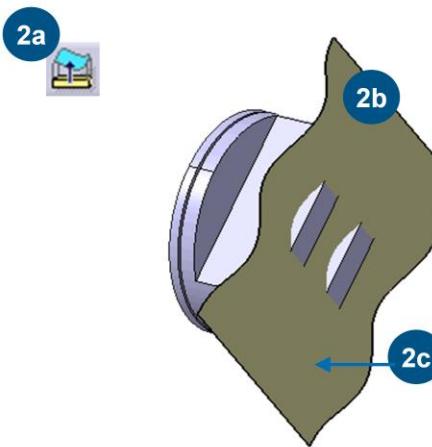
1. Open the part ReplaceFace.CATPart.

Open the ReplaceFace model. The solid geometry and the surface have already been created for you.

2. Replace the Pad face with the surface.

Use the **Replace Face** tool to replace the solid feature face with the existing surface.

- a. Select the **Replace Face** icon.
- b. Select the extruded surface feature
- c. Select the bottom surface of the cylinder head.
- d. Select **OK** to complete the operation.



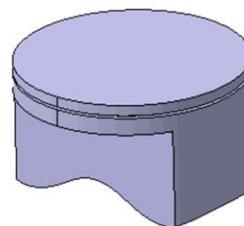
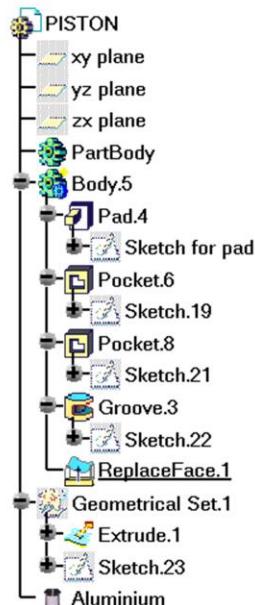
Replace a Model Face (2/2)

3. Clear the display.

For clarity, hide the geometrical set.

1. Right mouse click on Geometrical Set.1 in the specification tree
2. Click Hide/Show from the contextual menu.

4. Save and close the model.

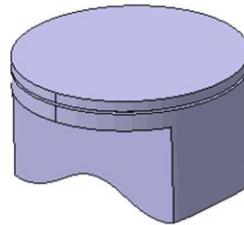
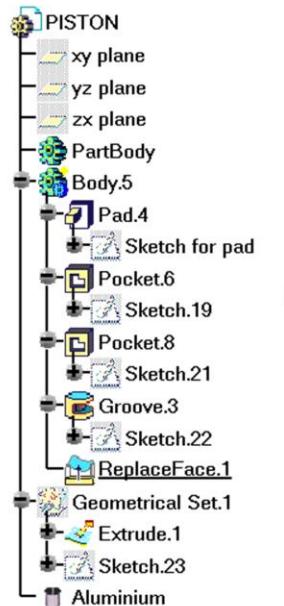


Handwritten notes area:

Recap: Replace a Model Face

In this exercise you have:

- Replaced a model face



Handwritten notes area for the student guide.

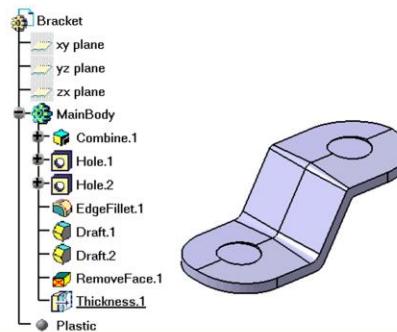


Exercise: Create Solid Combine and Advanced Draft

In this exercise, you will open an existing part that contains two sketches and use them to create a solid model. As revision, you will create holes and fillets. An advanced draft is then applied. To prepare the model for more advanced applications, faces are removed and thickness will be applied. Detailed instructions are provided for the new topics present in this exercise.

By the end of this exercise you will be able to:

1. Create a solid combine
2. Apply an advanced draft
3. Remove faces
4. Apply thickness to the model





Create Solid Combine and Advanced Draft (1/9)

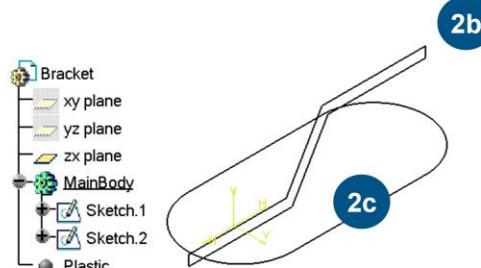
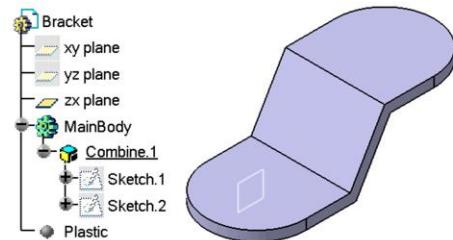
1. Open the part Bracket.CATPart.

Open the bracket model. Two sketches have been created for you.

2. Create a solid combine feature.

Use the two sketches provided to create a solid feature.

- a. Select the Solid Combine icon.
- b. Select Sketch.1 as the first profile.
- c. Select Sketch.2 as the second profile.
- d. Click **OK** to complete the feature.

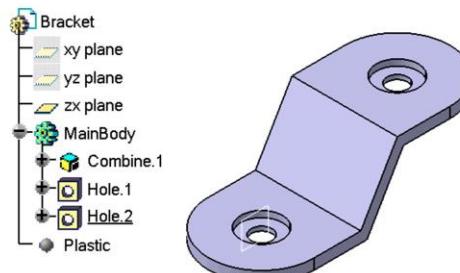
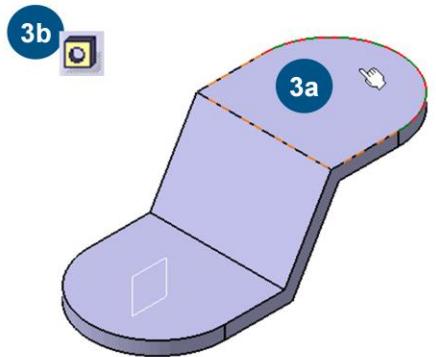


Create Solid Combine and Advanced Draft (2/9)

3. Create two counter-bored holes.

Create two counter-bored holes on the model.

- To create a concentric hole, multi-select the arc edge of the top horizontal face and the top horizontal surface.
- Select the **Hole** icon.
- Use the **Extension** tab to create a hole of [25mm] diameter, having depth **Up to Last**.
- Use the **Type** tab to create a counter-bore of [50mm] diameter and [7.5mm] depth.
- Create another counter-bored hole on the bottom horizontal surface. Use the same dimensions.

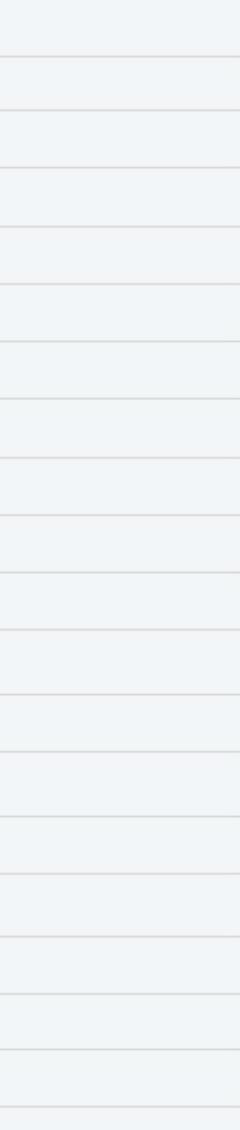
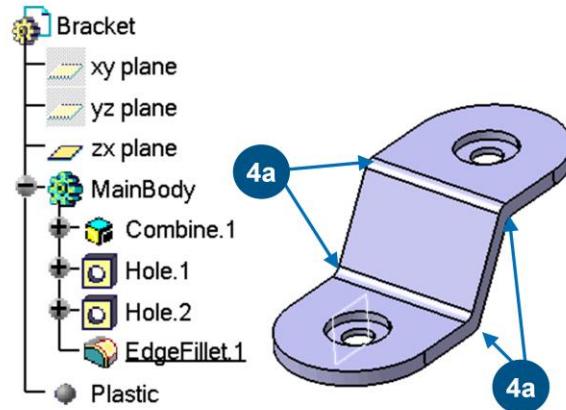


Create Solid Combine and Advanced Draft (3/9)

4. Create fillets.

Create fillets to smooth the edges.

- Create [10 mm] fillets on the four edges shown.

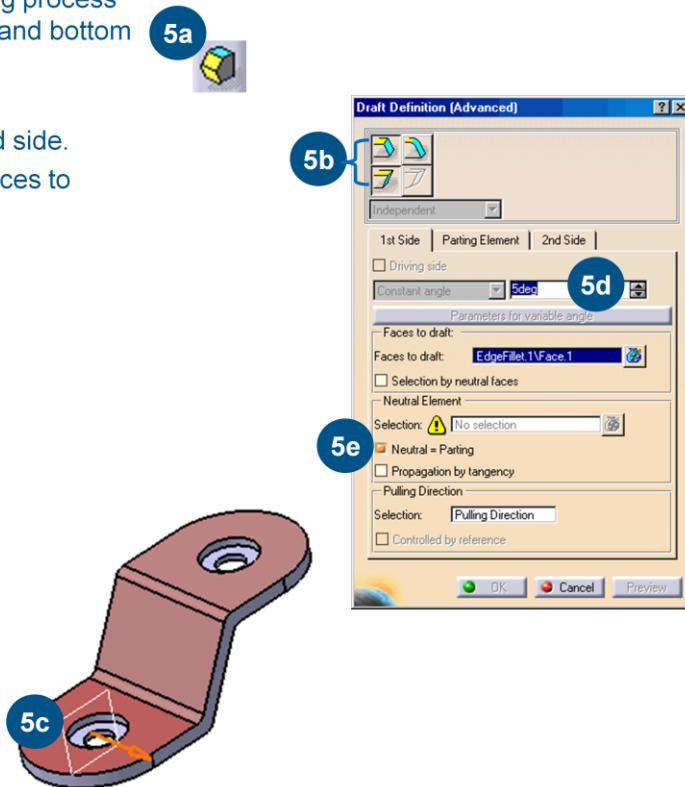


Create Solid Combine and Advanced Draft (4/9)

5. Create an advanced draft.

Prepare the model for the manufacturing process by creating advanced drafts on the top and bottom of the model.

- Select the **Advanced Draft** tool.
- Specify standard draft first and second side.
- Select lower horizontal faces as the faces to draft.
- Enter a draft angle of [5 deg].
- Select the Neutral = Parting option.

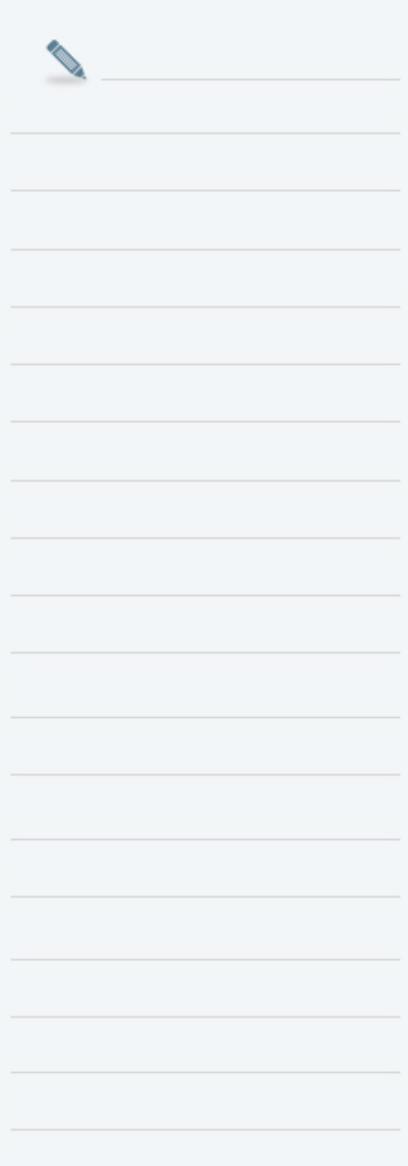
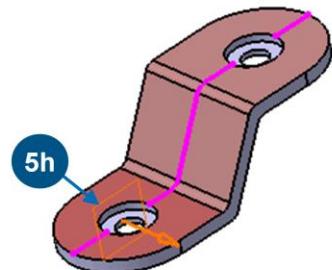
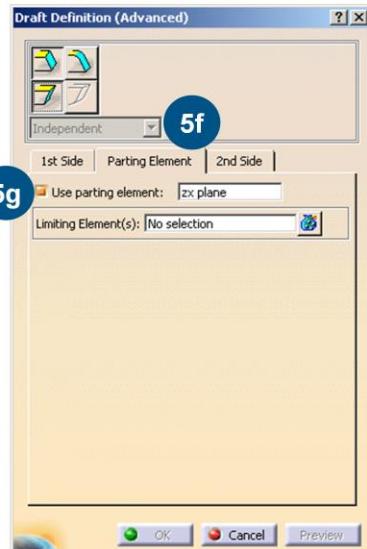


Create Solid Combine and Advanced Draft (5/9)

5. Create an advanced draft (continued).

Prepare the model for the manufacturing process by creating advanced drafts to the top face of the model.

- f. Select the Parting Element tab.
- g. Select the Use parting element option.
- h. Select the ZX plane as the parting element.

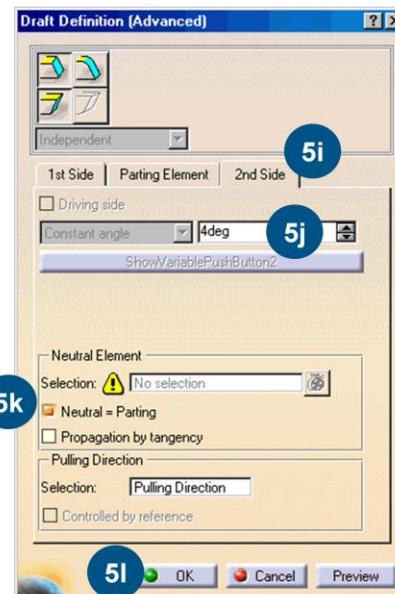
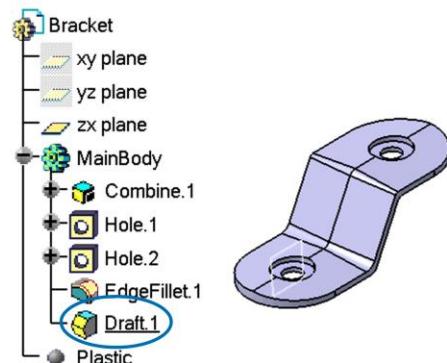


Create Solid Combine and Advanced Draft (6/9)

5. Create an advanced draft (continued).

Prepare the model for the manufacturing process by creating advanced drafts to the top face of the model.

- i. Select the 2nd side tab.
- j. Specify a draft angle of [4deg].
- k. Select the **Neutral = Parting** option.
- l. Click **OK** to generate the draft.



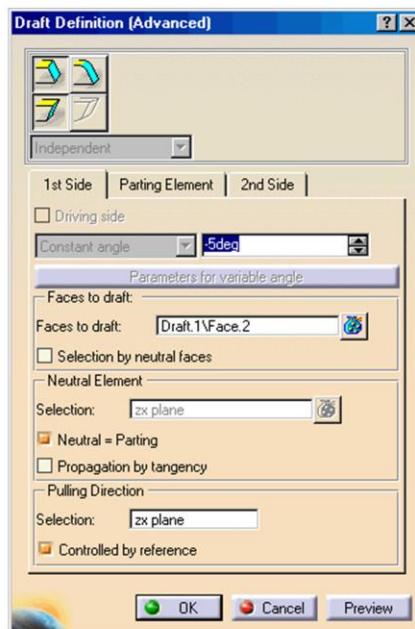
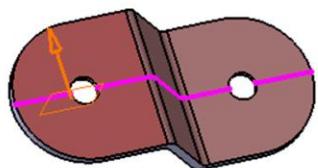
Create Solid Combine and Advanced Draft (7/9)

6. Create an advanced draft .

Prepare the model for the manufacturing process by creating advanced drafts to the bottom face of the model.

- Create a second advanced draft feature for the bottom surface. This time use the following parameters:

- 1st side draft angle: [- 5 deg]
- 1st side Neutral element: **Neutral = Parting**
- 1st side pulling direction: ZX plane
- Parting element: ZX plane
- 2nd side draft angle: [- 4deg]
- 2nd side Neutral element: Neutral = Parting
- Leave pulling direction for the 2nd side as the default.

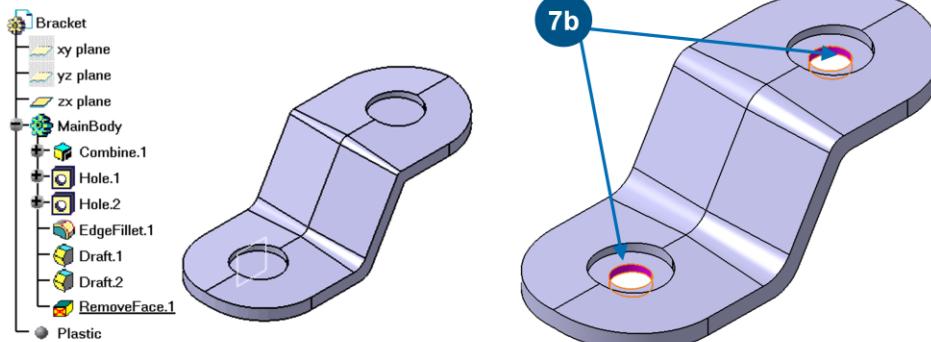
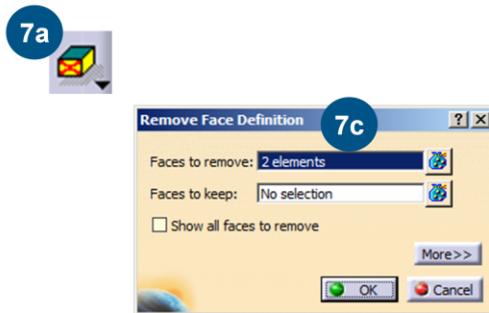


Create Solid Combine and Advanced Draft (8/9)

7. Remove faces.

Remove the bottom faces of the counter-bored holes. These faces are not to be considered in the analysis process.

- Select the Remove Face icon.
- Select the inside faces of the two holes. Do not select the counter-bored portion of the hole.
- Click OK to remove the faces.



Create Solid Combine and Advanced Draft (9/9)

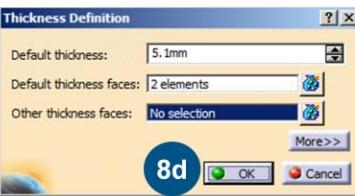
8. Apply thickness.

Add thickness to the counter-bored section of the holes.

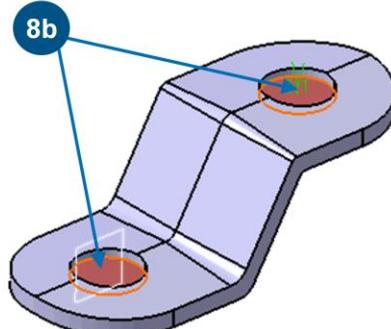
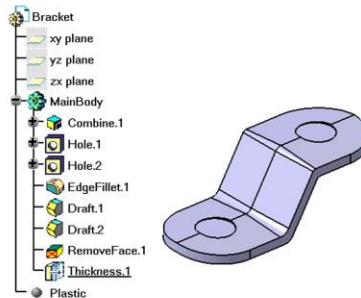
- Select the **Thickness** icon.
- Select the bottom faces of the two holes.
- Apply a [5.1mm] thickness.
- Click **OK**.

9. Save and close the model.

- For clarity, hide the ZX plane.



8d

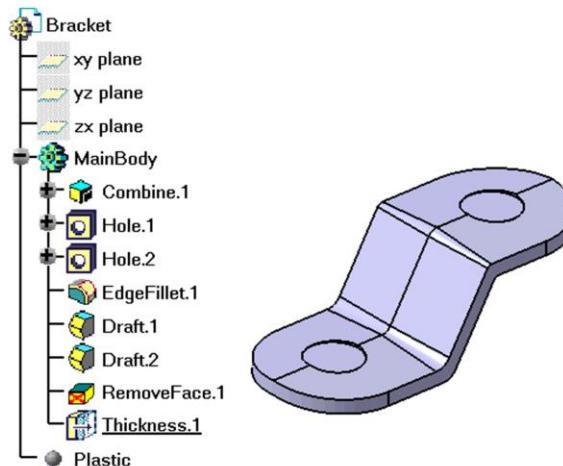


8b

Recap: Create Solid Combine and Advanced Draft

In this exercise you have:

- ✓ Created a solid combine
- ✓ Applied advanced draft
- ✓ Removed face
- ✓ Applied thickness

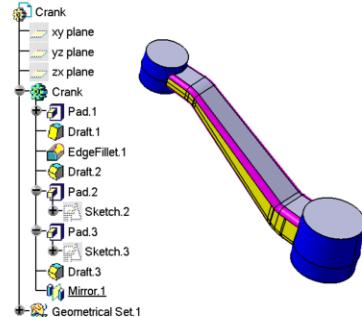




Exercise: Apply Advanced Draft

In this exercise, you will open an existing part that contains sketched wireframe elements and a surface feature. To complete this model you will have to create several advanced draft features. You will also use pads, variable fillets, and the mirror operation to complete this model. High-level instruction is provided for this exercise.

By the end of this exercise you will be able to Apply advanced draft features.

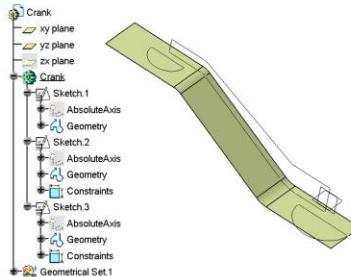


20 minutes



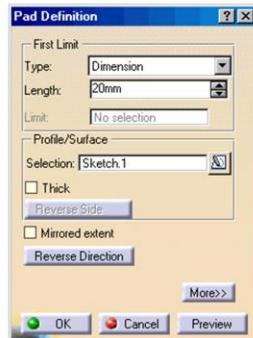
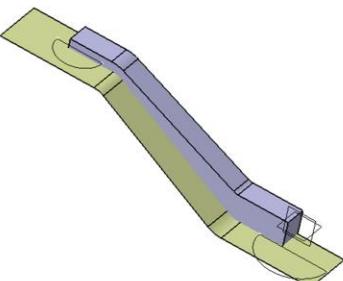
Create a Pad

1. Load Ex8F.CATPart.



2. Create a pad Feature.

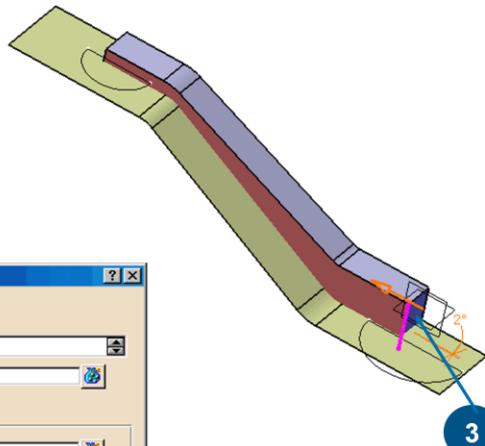
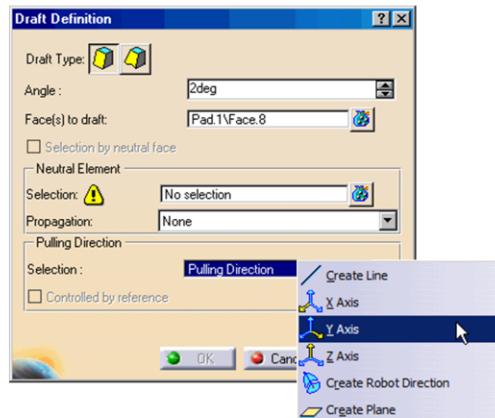
Use Sketch.1 to create a pad feature with a depth of [20mm].



Create a Draft

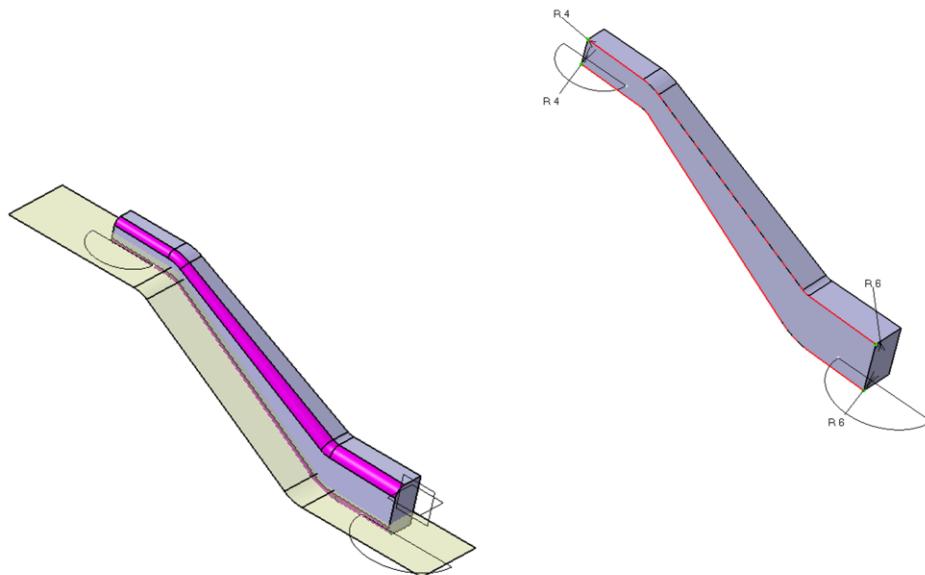
Create draft on the outside vertical wall.

1. Use a draft angle of 2 degrees.
2. Use the positive Y direction as the pull-direction.
3. Use the right vertical face as the neutral plane.



Create a Variable Fillet

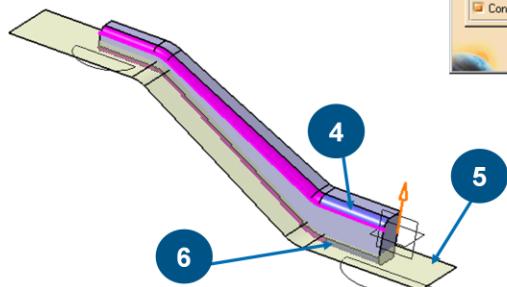
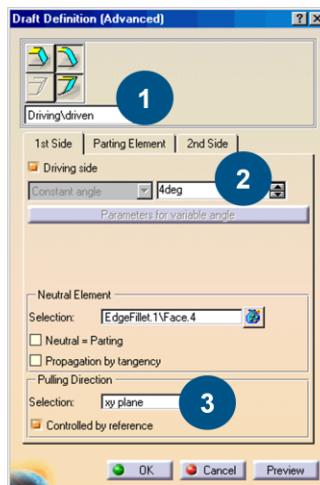
1. Apply a variable radius fillet to the top and bottom outside edges.
 - a. Create the fillet from [4mm] to [6mm] along each side.



Apply Advanced Draft (1/2)

Create a two-sided reflect draft.

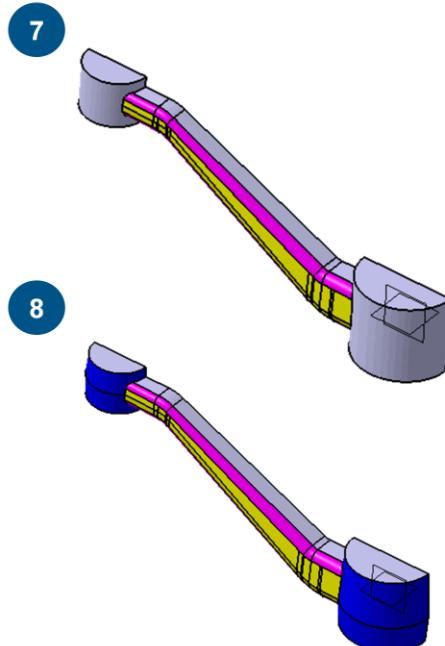
1. Use the **Driving/Driven** dependency option.
2. Set the draft angle to 4 degrees.
3. Use the XY plane as the pulling direction for the first side.
4. Use the top fillet as the neutral element for the side one.
5. Select the Extruded surface as the parting element.
6. Use the bottom fillet as the neutral element for the side two.



Apply Advanced Draft (2/2)

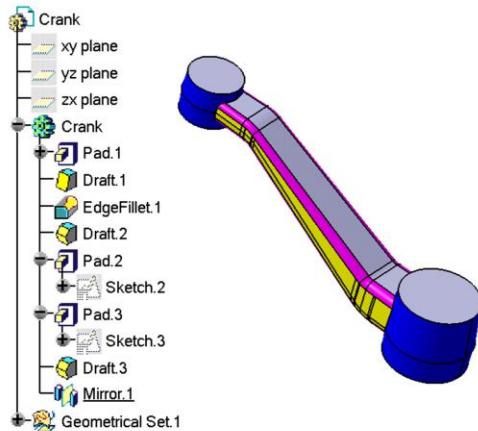
7. Create two pad features.
 - a. Use Sketch.2 to create a pad feature with a depth of [30mm].
 - b. Use Sketch.3 to create a pad feature with a depth of [50mm].

8. Apply an advanced draft feature.
Apply an advanced draft feature to the two pads.
 - a. Create the draft with a 4 degree draft angle on the first side.
 - b. Use the XY plane as the pulling direction for side one.
 - c. Use a 6 degree draft angle on the second side.
 - d. Use Extrude.1 as the parting element.
 - e. Set the Neutral element on both sides equal to the parting element.



Finalize the Model

1. Mirror the model.
 - a. Complete the model by mirroring the part body about the YZ plane.
2. Clear the model, save and close it.
 - a. Hide all wireframe and surface elements and save the model.

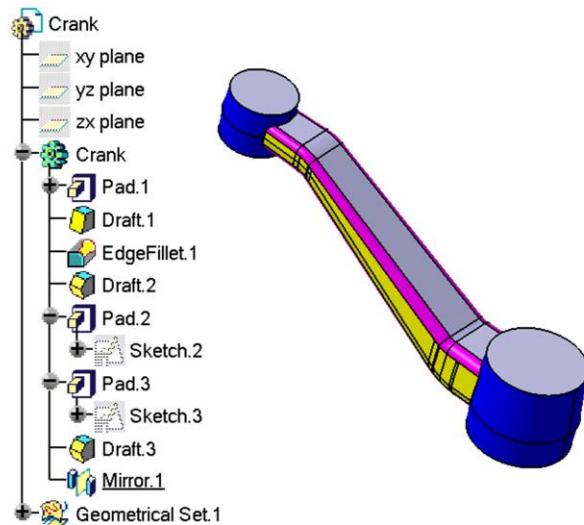


Handwritten notes area for the student guide.

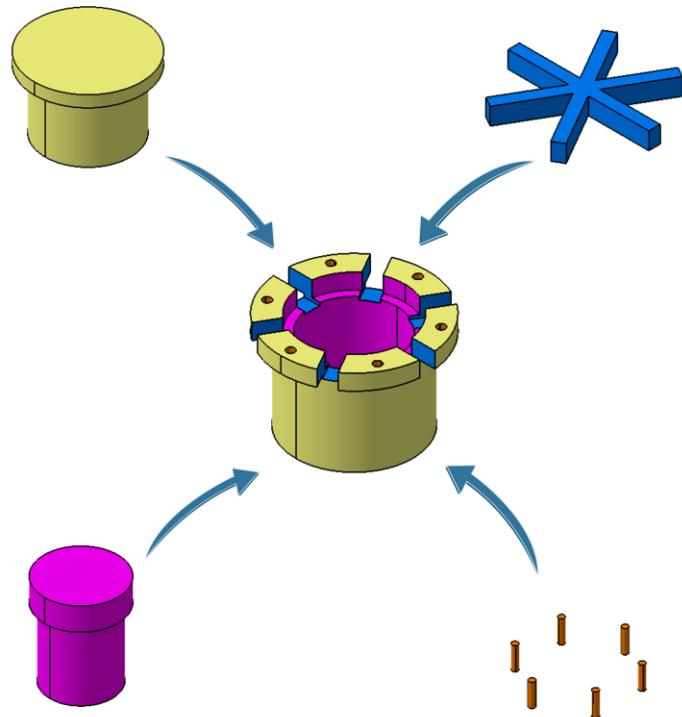
Recap: Apply Advanced Draft

In this exercise, you have:

- Applied an advanced draft



Using the Multi-Body Method



Here are the topics to be covered:

- 1. Creating Advanced Sketch-Based Features
- 2. Creating Multi Section solids
- 3. Creating Advanced Drafts
- 4. Creating Advanced Dress-Up features
- 5. **Using the Multi-Body Method**
- 6. Creating Multi-Model Links

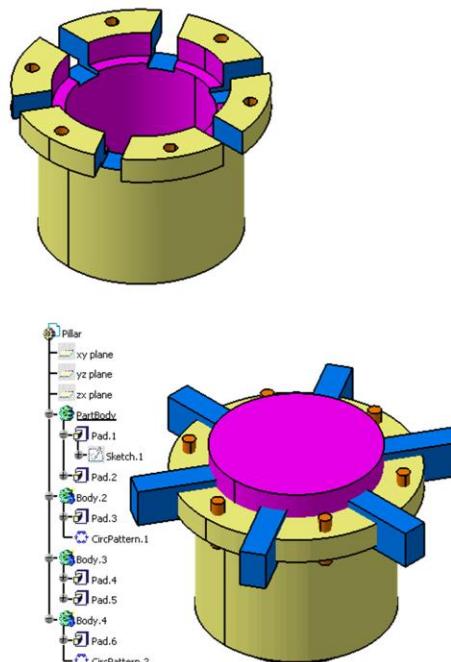
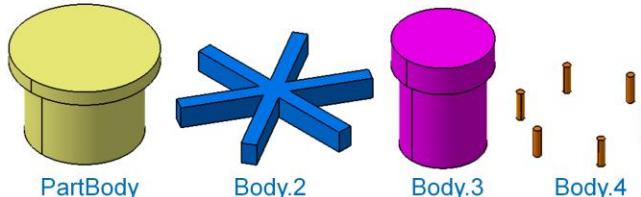


Handwritten notes area with horizontal lines for writing.

About Multi-Body Method

The Multi-Body Method allows you to organize discrete areas of geometry within a complex model into different bodies. Each geometry area is created in a separate body. Each body acts independently in the model.

In the images on the right-hand side, the Pillar model is divided into four bodies. The geometry in the bodies is modeled using positives. The bodies are then combined using Boolean Operations to create the completed model shown in the top right-hand image.

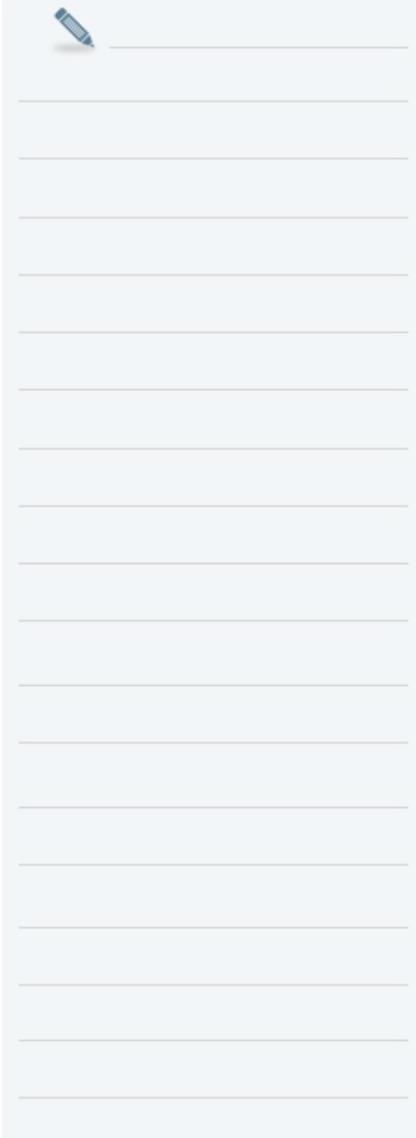
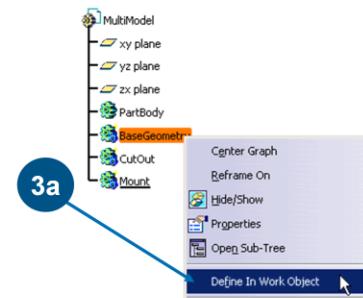
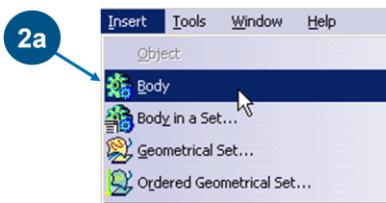




Using the Multi-Body Method (1/2)

Use the following steps to apply the Multi-Body Method:

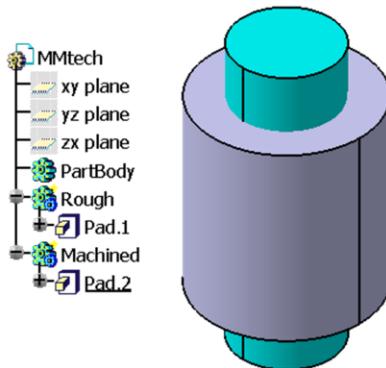
1. Break the model into bodies
 - a. Consider the areas of the model that should be contained in a separate body.
 - b. These can be functional areas, for example a complex cutout or area of the model.
 - c. Try to combine features that can similar design intent into the same body.
 - d. Create as many bodies as required.
2. Define the body structure.
 - a. Select **Insert > Body** to add a new body to the model.
 - b. Bodies should be descriptively named so that the design intent is clear.
3. Insert features into the bodies.
 - a. To activate a body, select it from the specification tree and select **Define In Work Object** from the right mouse button contextual menu.
4. Combine the bodies using Boolean Operations.



Using the Multi-Body Method (2/2)

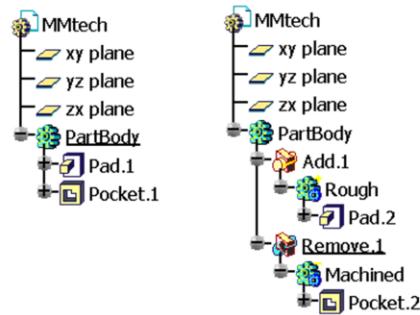
Advantages

- ▶ Provides an organized approach to modeling complex parts.
- ▶ Solid features within a body can be hidden independently of the rest of the model.
- ▶ Groups of geometry can be de-activated by de-activating the body.
- ▶ Complex geometry is easier to create within a focused area of the model.
- ▶ Model will update faster due to the organized structure.



Disadvantages

- ▶ The number of operations in a multi body method is greater than the number of operations in the pure feature modeling method.





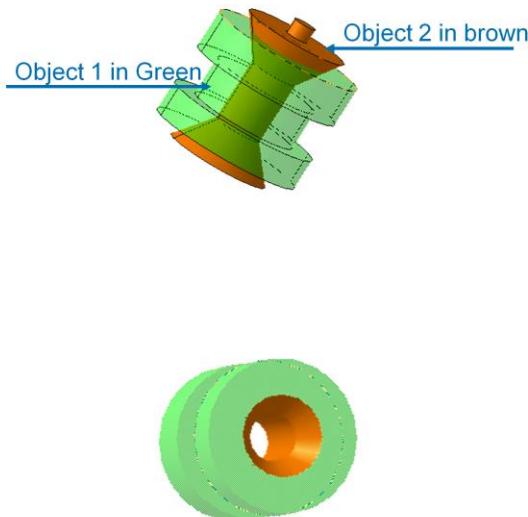
About Boolean Operations

Boolean operations enable you to use a Multi-Body approach to modeling. Using multiple bodies in a model or in different models, you can use Boolean operations to manipulate the bodies to achieve different results. For example, casting models can be developed using bodies to represent cast and machined features.

In the images on the right-hand side, a Boolean operation is performed on the two bodies to subtract the volume of the second body from the second where they intersect.

The Boolean operations are:

- A. Assemble
- B. Add
- C. Remove
- D. Intersect
- E. Union Trim
- F. Remove Lump

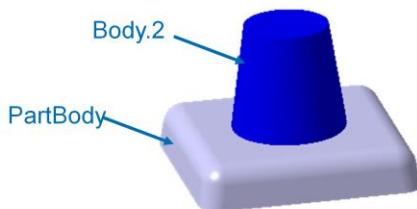


Assembling Bodies (1/4)

In this example, Body.2 will be assembled into PartBody. When Body.2 is assembled to PartBody, the operation between the bodies is a union. An Assemble operation will respect the “nature” of features. If Body.2 contains Pocket feature (permissible) as its first node, the Assemble operation will remove material from Body.1.

Use the following steps to perform the Assemble operation:

1. Right-click the body to be assembled. In this example, Body.2 will be assembled into the PartBody.
2. From the contextual menu select **x.object > Assemble**.

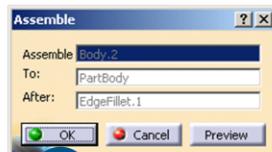




Assembling Bodies (2/4)

Use the following steps to perform the Assemble operation (continued):

3. By default, the selected body will be assembled to the active body as the last feature. If required, select another body to which the selected body will be assembled.
4. Select **OK** to finalize the operation. Notice that Body.2 contains a groove. As the groove features remove material, the result of the union removes material.



4



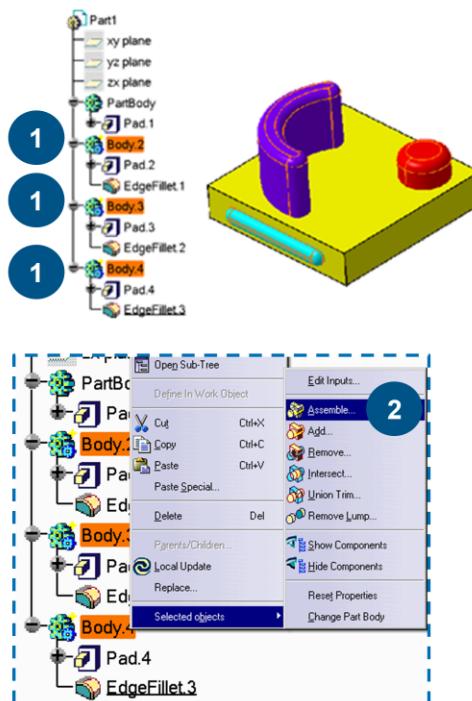


Assembling Bodies (3/4)

To work more efficiently, a single operation can be performed on multiple bodies.

Use the following steps to Assemble multiple bodies into one:

1. Pre-select all the bodies using the <Ctrl> key. In this example, Body.2, Body.3, and Body.4 are all pre-selected.
2. Right-click and select **Selected objects > Assemble** from the contextual menu.

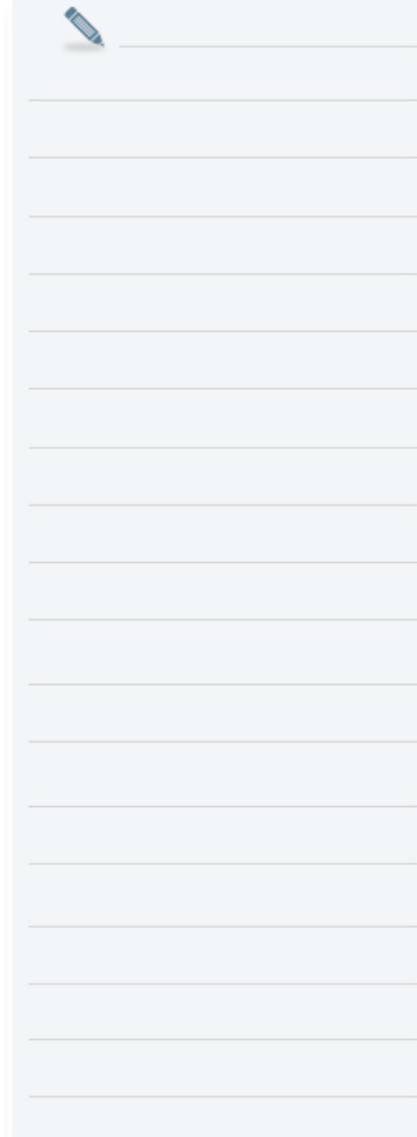
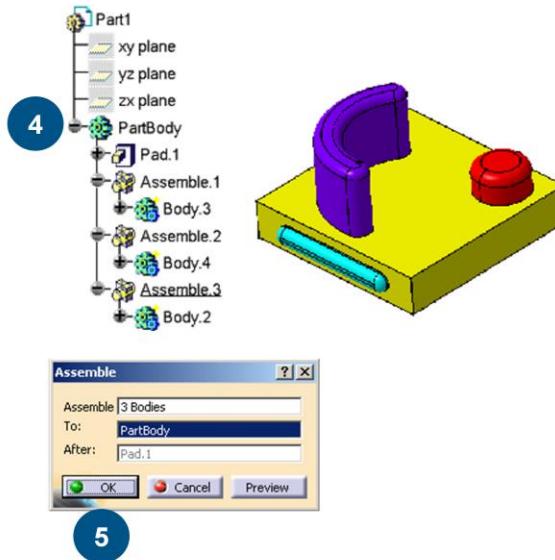


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Assembling Bodies (4/4)

Use the following steps to Assemble multiple bodies into one (continued):

4. Select the body in which other bodies will be inserted.
5. Click **OK** to complete the operation.

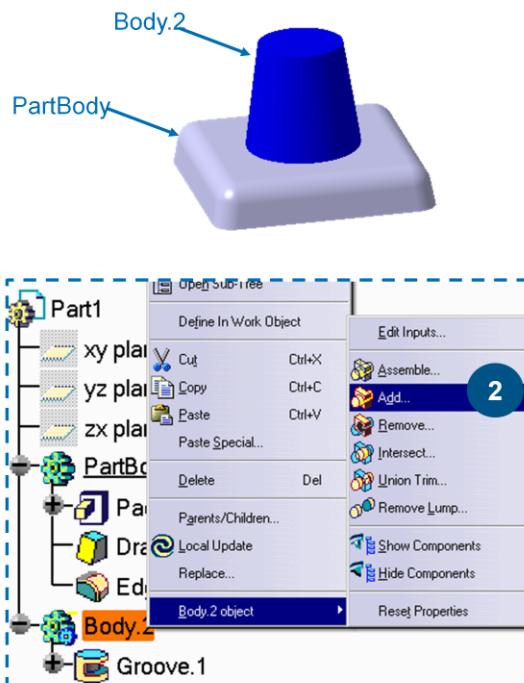


Adding Bodies (1/2)

In this example, Body.2 will be added to the PartBody. The Add operation also creates a union between the two selected bodies. The difference between an Add and an Assemble is that if Body.2 contains a pocket feature as its first node, using an Add operation the pocket will be seen by PartBody as a pad.

Use the following steps to perform an Add operation:

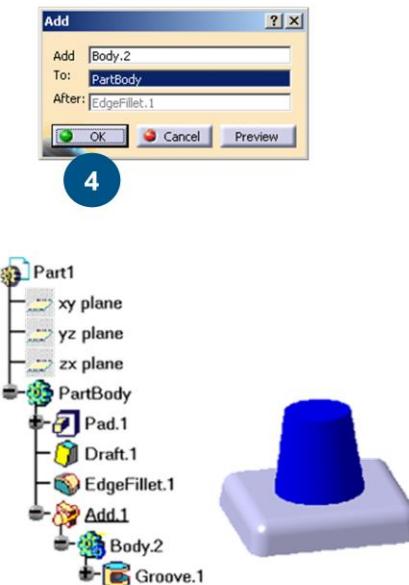
1. Right-click the body to be added. In this example, Body.2 will be added to the PartBody.
2. From the contextual menu select **x.object > Add**.



Adding Bodies (2/2)

Use the following steps to perform an Add operation
(continued):

3. By default, the selected body will be added to the active body as the last feature. If required, select another body to which the selected body will be added.
4. Click **OK** to finalize the operation. Notice that Body.2 contains a groove; however, using the add operation the feature remains as it was before the operation.

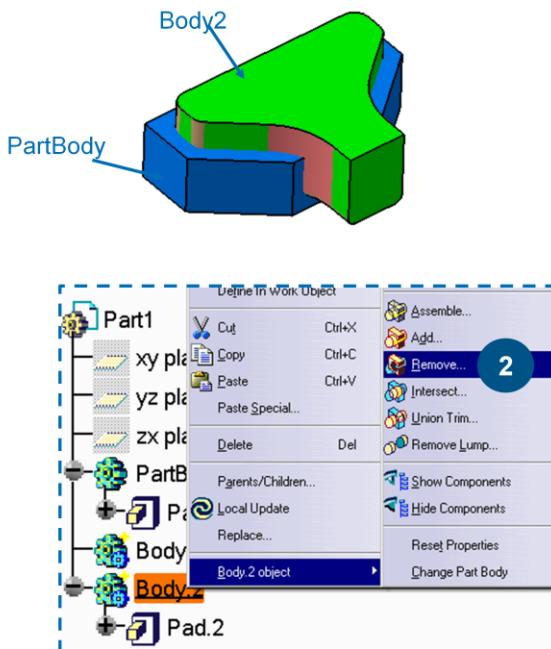


Removing Bodies (1/2)

In this example, Body,2 will be removed from the PartBody. If Body2 is Removed from PartBody, the operation is PartBody minus Body2.

Use the following steps to perform a remove operation:

1. Right-click on the body to be added. In this example, body,2 will be added to the PartBody.
2. From the contextual menu select **x.object > Remove**.

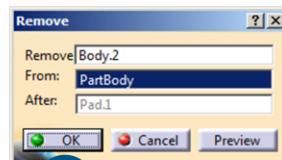


Removing Bodies (2/2)

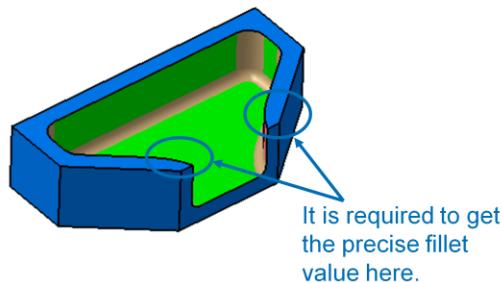
Use the following steps to perform a remove operation
(continued):

3. By default, the selected body will be removed from the active body. If required, select another body from which the selected body will be removed.

4. Click **OK** to finalize the operation.



4

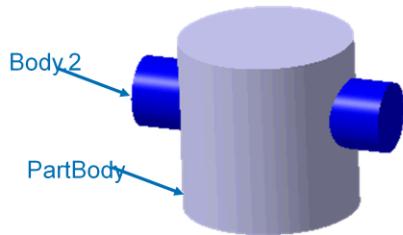


Creating Body Intersects (1/2)

In this example, Body.2 will intersect PartBody. The resulting solid is the material common between the two intersecting bodies.

Use the following steps to perform an Intersect operation:

1. Right-click the body to be added. In this example, Body.2 will be added to the PartBody.
2. From the contextual menu select **x.object > Intersect**.

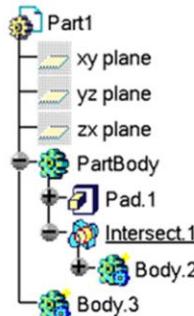
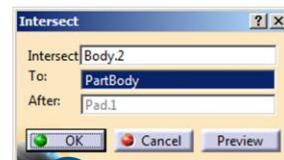




Creating Body Intersects (2/2)

Use the following steps to perform an Intersect operation (continued):

3. By default, the selected body will intersect the active body. If required, select another body to intersect.
4. Click **OK** to finalize the operation.



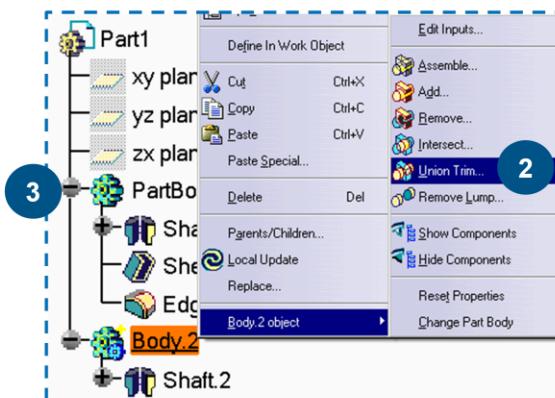
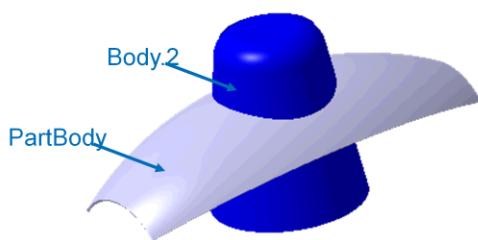


Creating a Union Trim (1/2)

In this example, Body.2 will be used to trim the PartBody using the Union Trim operation. This operation is a union of two bodies with an option to remove or keep one side. One face is selected to remove the lower section of Body.2, while keeping the outer section of the PartBody, while the other face is selected to keep only the upper section of Body.2. For the union trim operation to work, the geometry must have sides that are clearly defined.

Use the following steps to perform a Union Trim operation:

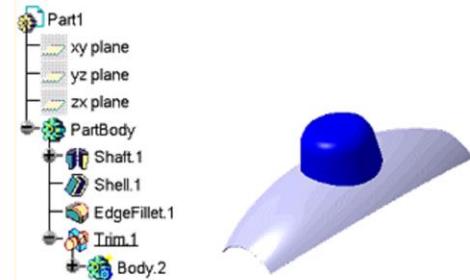
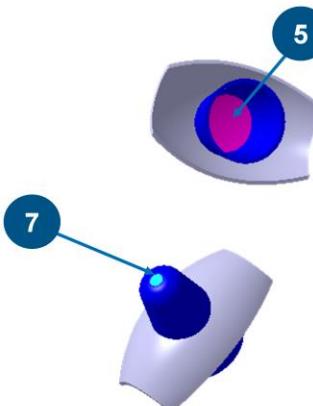
1. Right-click the body to be added. In this example, body.2 will be added to the PartBody.
2. From the contextual menu select **x.object > Union Trim**.
3. Select another body. This body will be trimmed by the body you have already selected. In this example, PartBody is selected.



Creating a Union Trim (2/2)

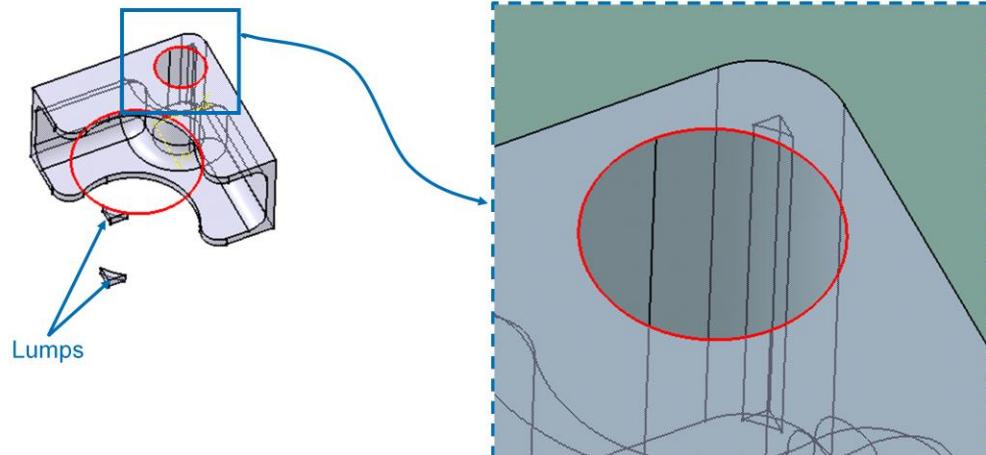
Use the following steps to perform a Union Trim operation (continued):

4. Select in the **Faces to remove** field.
5. Select the faces to be removed by the operation.
6. Select in the **Faces to keep** field.
7. Select the faces to be kept by the operation.
8. Click **OK** to finalize the operation.



Removing Lumps (1/2)

Lumps and cavities may appear in the model after certain operations. These elements can be removed using the Remove Lump tool. The previous options work between two bodies. The Remove Lump option works on geometry within a specific body. A lump is a material that is completely disconnected from other parts within a single body. You can delete any lump as a single entity even if the lump is a combination of features.

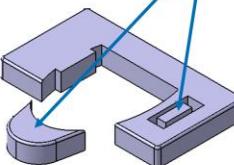
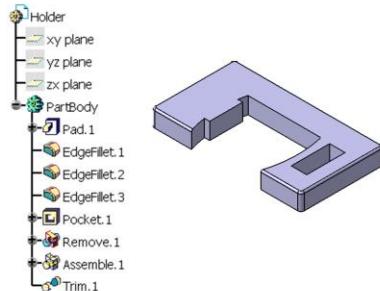
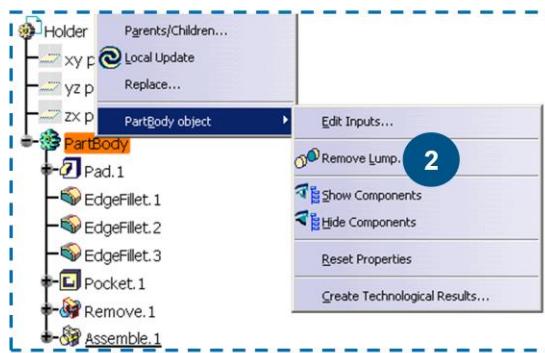


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Removing Lumps (2/2)

Use the following steps to remove Lump and cavities from a model:

1. Right-click the body from which the Lumps and Cavities are to be removed. In the given Lump and Cavities have to be removed from the PartBody.
2. Select **x.object > Remove Lump** from the contextual menu.
3. Select in the **Faces to remove** field.
4. Select the Lumps.
5. Click **OK**.

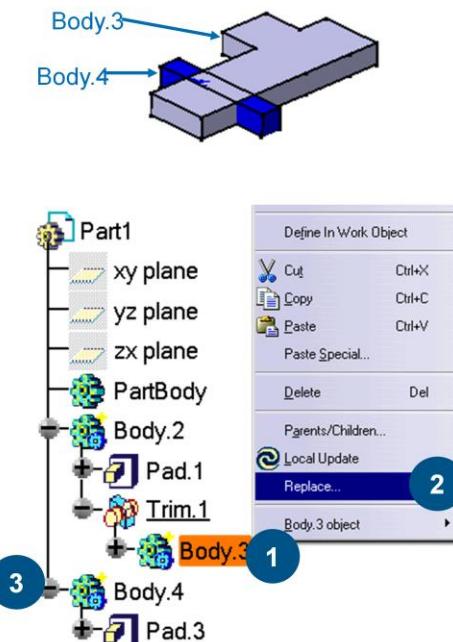


Replacing a Body (1/3)

The Replace tool can replace a body used for an operation by another body. This eliminates the need to delete the operation and redo it with the correct body.

Use the following steps to replace the body used in an operation:

1. Right-click the body to be replaced.
2. Select **Replace** from the contextual menu.
3. Select the replacement body.

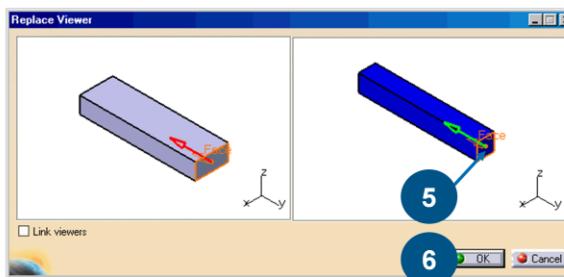
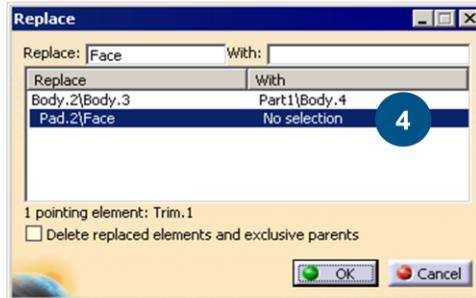




Replacing a Body (2/3)

Use the following steps to replace the body used in an operation (continued):

4. In this example, additional references are required to replace the bodies. From the Replace dialog box click on the second field.
5. A Replace Viewer dialog box displays the reference. Select the appropriate reference in the replacing body. In this example, the missing reference is the face that is to be removed during the Union Trim operation.
6. Click **OK** to close the Replace Viewer dialog box.

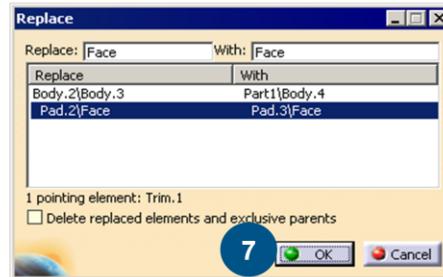
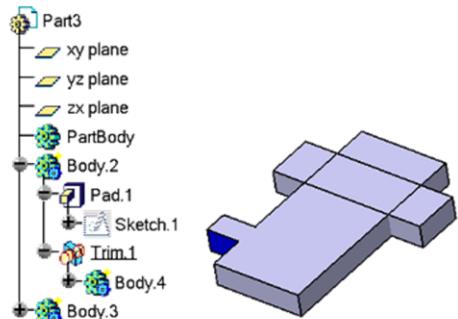




Replacing a Body (3/3)

Use the following steps to replace the body used in an operation (continued):

7. Click **OK** to complete the operation.
8. If necessary, update the part by selecting the **Update All** icon.



Changing the Boolean Operation Type (1/2)

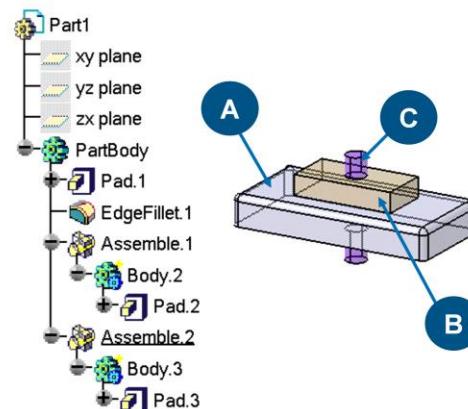
The type of Boolean operation can be changed without deleting the operation and recreating it.

Consider the following example.

Three bodies are constructed:

- A. PartBody
- B. Body.2
- C. Body.3

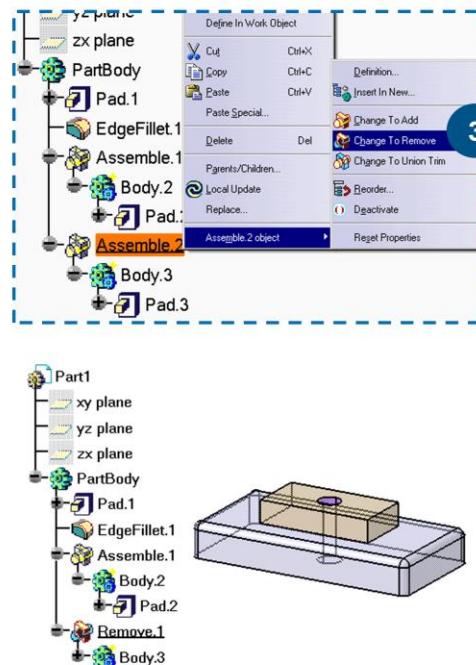
Currently, Body.2 and Body.3 have been assembled into the PartBody. However, Body.3 should be removed from the PartBody.



Changing the Boolean Operation Type (2/2)

Use the following steps to change a Boolean operation:

1. Right-click the operation to be replaced. In this example, the Assemble operation is to be replaced.
2. A list of operations, which the current operation can be converted to are shown in the contextual menu.
3. Select the appropriate operation. Here, the **Change to Remove** option is selected.





Working with Boolean

In this section, you will be given a recommendation to help during working with Boolean operations.



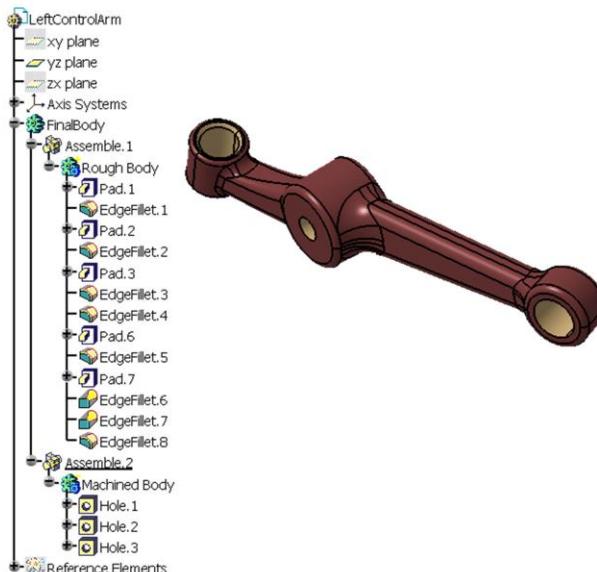


Maintaining a Flat Specification Tree Structure (1/2)

It is recommended that you maintain a flat specification tree structure while working with Boolean operations.

Flat specification tree structure enables you to:

1. Easily understand the way in which the part is designed.
2. Easily locate the failed feature, in case of feature failure.
3. Reduce the update time.



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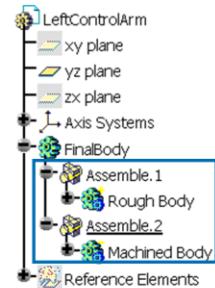
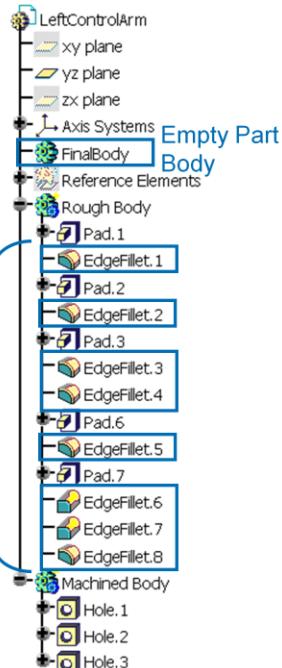


Maintaining a Flat Specification Tree Structure (2/2)

In order to maintain a flat specification tree :

1. Perform the Boolean operations in an empty PartBody.
2. Create dress-up features (Draft, Fillet, Shell, etc.) as close as possible, to the solid primitive.

Basic features are close to solid primitives

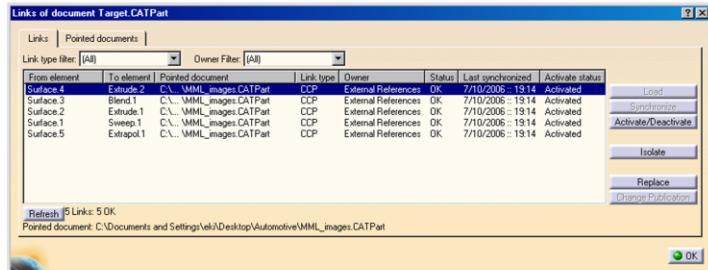


Boolean operations are done in empty body





Creating Multi-Model Links



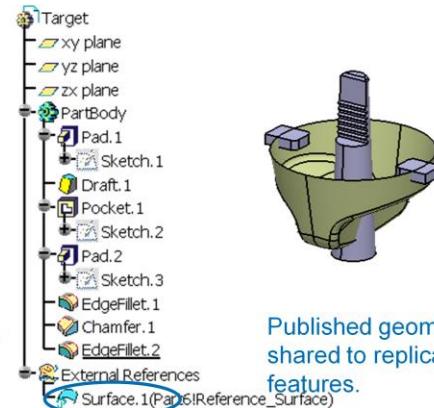
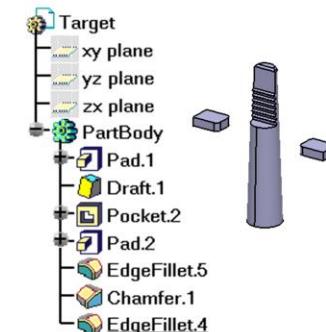
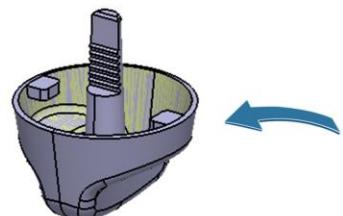
Here are the topics to be covered:

- ✓ 1. *Creating Advanced Sketch-Based Features*
- ✓ 2. *Creating Multi Section solids*
- ✓ 3. *Creating Advanced Drafts*
- ✓ 4. *Creating Advanced Dress-Up features*
- ✓ 5. *Using the Multi-Body Method*
- ✓ 6. ***Creating Multi-Model Links***

About Multi-Model Links (1/3)

Using multiple bodies while designing can give the model added flexibility. Boolean operations allow complex models to be created by adding, removing, or intersecting simple geometric shapes.

Geometry can be shared between models to quickly replicate features in a number of parts. However, in order to share geometry, it is recommended that the elements must be published. In this case the shared geometry can be restricted to published elements only.



Published geometry is shared to replicate the features.



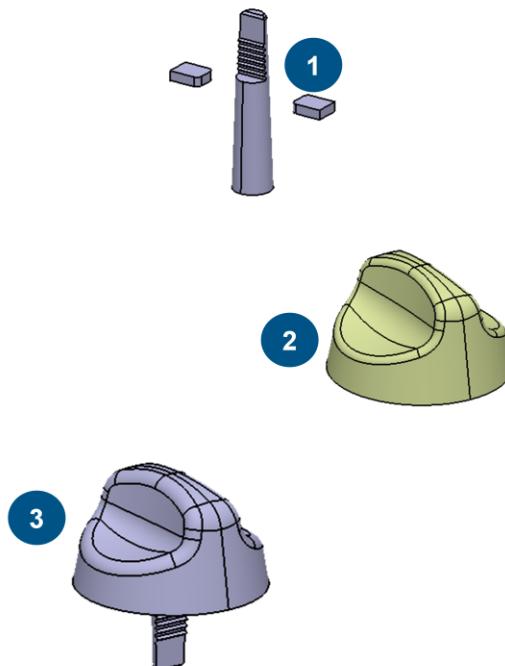
About Multi-Model Links (2/3)

In the context of the concurrent engineering, Multi-Model Links enable you to design a model using elements from another model.

Using Multi-Model Links you can copy bodies created in different files into your own part. This enables to automatically update the part when changes occur in the source files.

For example,

1. Part A is created by Designer A.
2. Part B is created by Designer B.
3. Using Multi-Model links, Part B is copied into Part A.

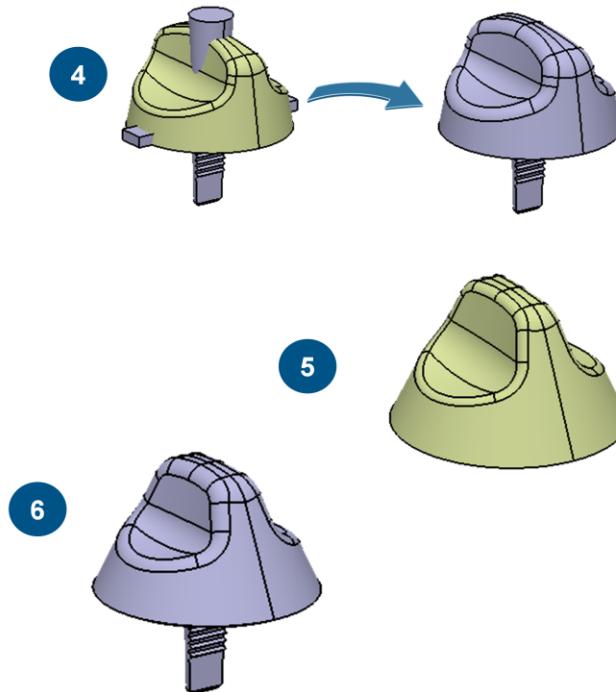


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About Multi-Model Links (3/3)

For example (continued):

4. Using a Boolean operation, Part B is added to Part A.
5. Part B is modified by Designer B.
6. Because of the Multi-model link, Part A is automatically updated to reflect the changes in Part B.

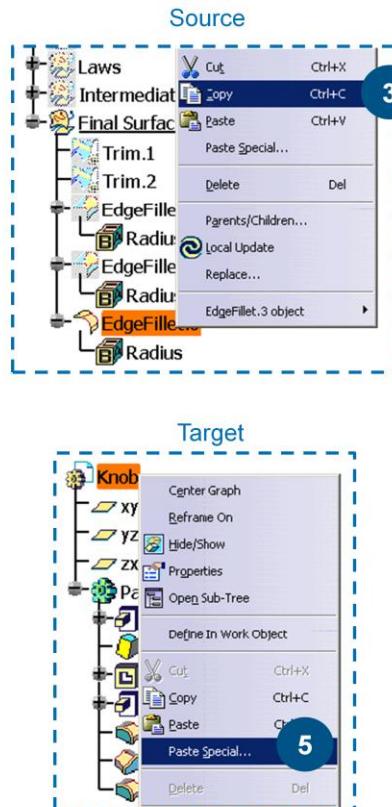


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Establishing Multi-Model Links (1/3)

Use the following steps to establish a Multi-Model link:

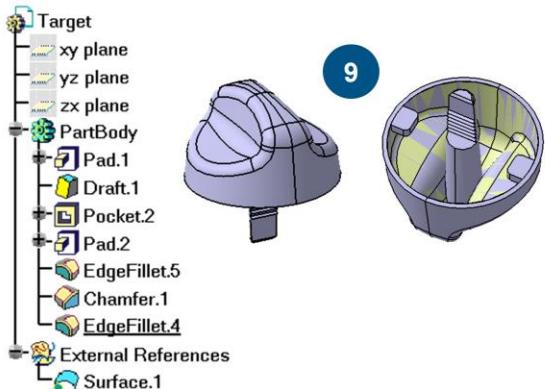
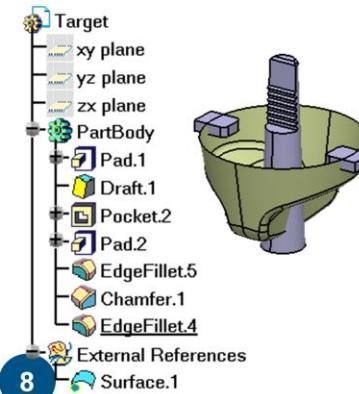
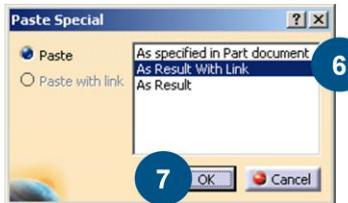
1. Open both the source and target files.
2. From the source model, right-click on the feature to be copied.
3. From the contextual menu click **Copy**.
4. From the target model, right-click on the Part as shown.
5. Click **Paste Special** from the contextual menu.



Establishing Multi-Model Links (2/3)

Use the following steps to establish a Multi-Model link
(continued):

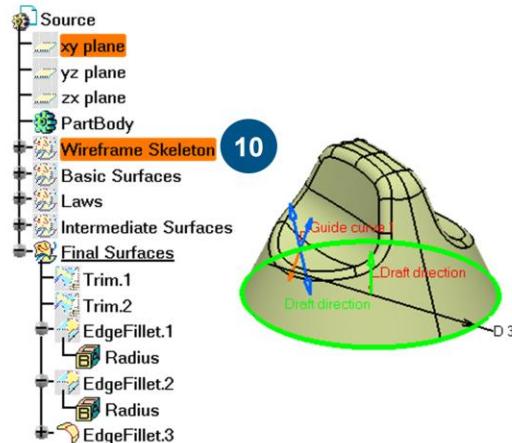
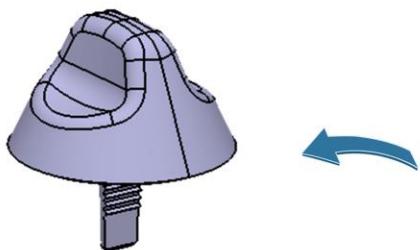
6. From the **Paste Special** dialog box select **As Result with Link**.
7. Click **OK**.
8. The Source PartBody is copied into the target model.
9. Complete the model.



Establishing Multi-Model Links (3/3)

Use the following steps to establish a Multi-Model link (continued):

10. If required, make changes to the source model. In this example, several features have been added to the source body.
11. Update the target model. The model updates to reflect the changes.

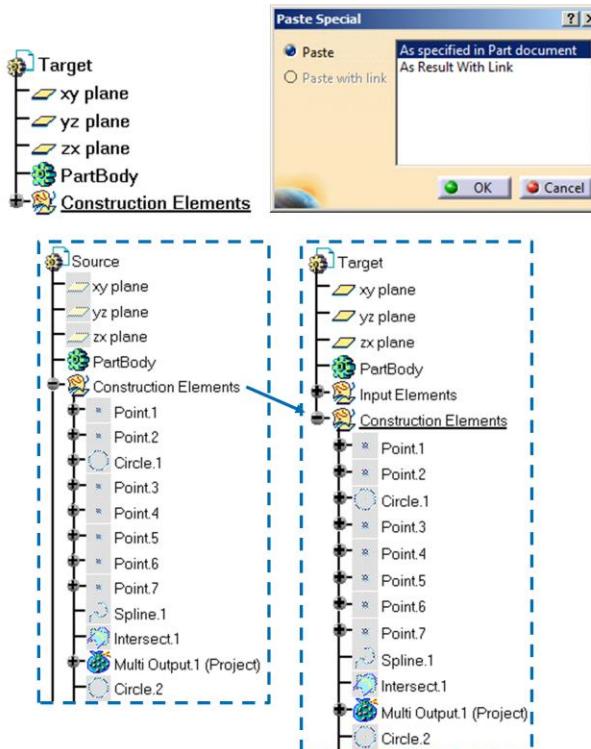


Using Paste Special (1/3)

Many features can be reused in other documents. This reduces the need to recreate features that are commonly used in files and also aids in concurrent engineering.

Several paste special options are available, choose the option that best meets the requirements of your design:

- A. The **As specified in Part document** option copies the element(s) with their design specifications. Each feature is recreated in the target model and can be edited. There is no link to the source model. In this example, the 'Construction Elements' Geometrical Set from the source document is copied into the Target model. A second set is added to the model containing all the copied elements and their design specifications.

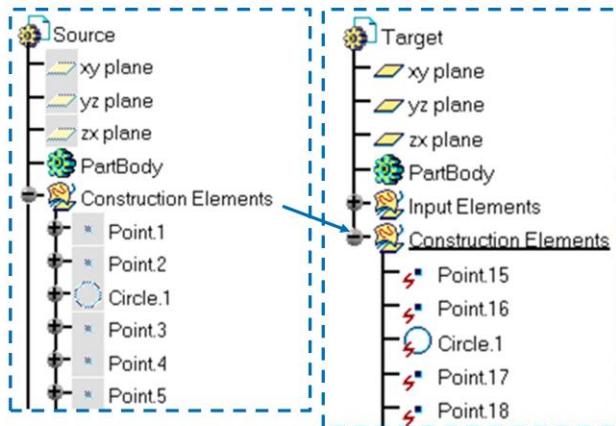




Using Paste Special (2/3)

- B. The **As Result** option copies the elements without their design specifications and link. This option is useful when you do not want to show the feature information or make changes to the copied elements in the target document.

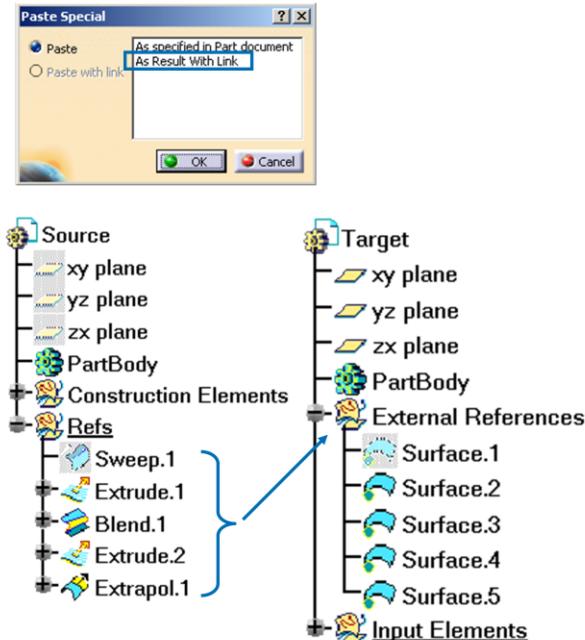
In the example, when the 'Construction Element' Geometrical Set is copied from the Source model and pasted in the Target model it creates datum surfaces. The red lightning bolt against the elements denote that the link has been isolated.





Using Paste Special (3/3)

- C. The **As Result with Link** option can be used for copying the individual features and not on the entire Geometrical set. It copies the element(s) without their design specifications and links the copied element(s) to the original object(s). When changes occur in the source document they will update in the target document. Notice that when the surface in the target model is copied using this option, it creates a single surface with a green dot, indicating that there is a link between the source and target documents.





Managing Multi-Model Links (1/4)

When you use the Paste Special option **As result with Link** you create a link between the source document and the target document.

Copied links display a number of symbols depending on their status:

Icon	Icon (working with Publications)	Description
		The pointed element is loaded and synchronized
		The pointed document is not loaded
		Link has been isolated. This icon will also appear if the source document has been copied using the As Result paste special option.
		Source document has been modified. Target document is not up to date.
		Source document is not found.



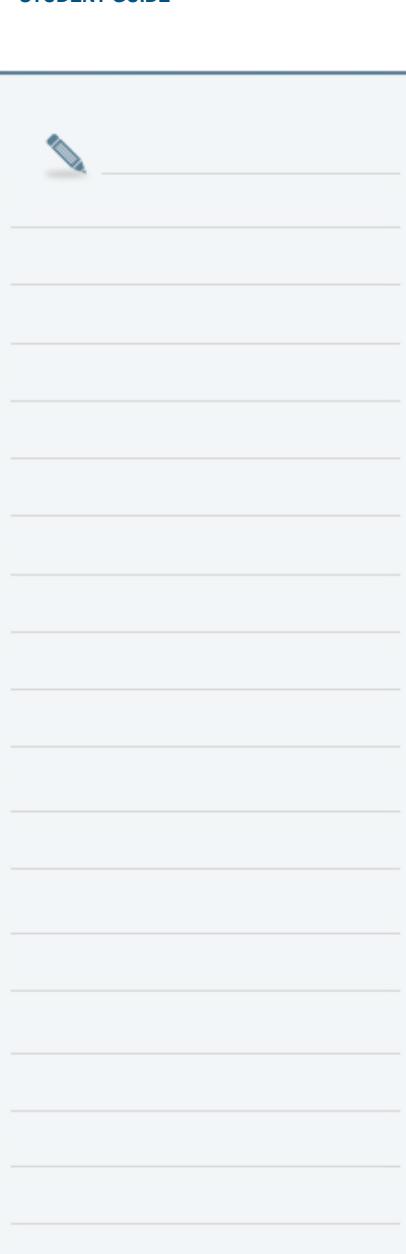
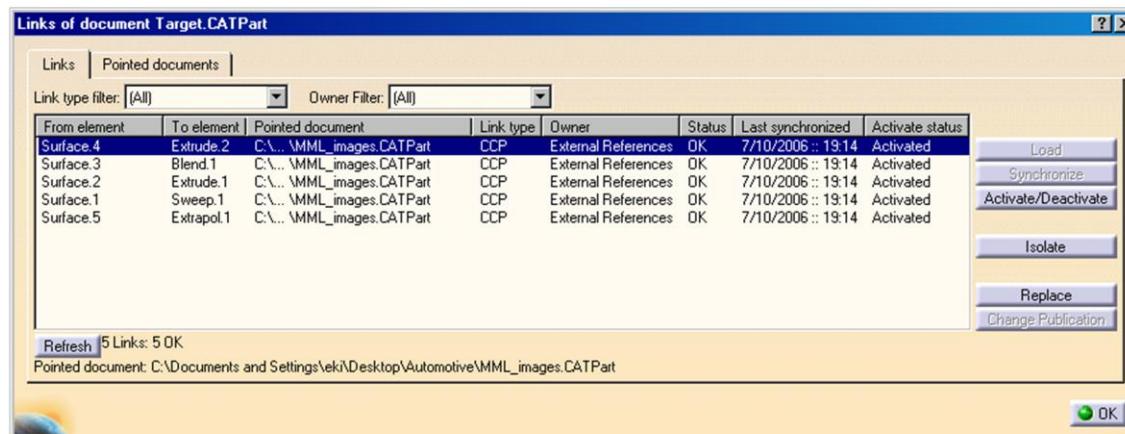


Managing Multi-Model Links (2/4)

Using the Link Panel, you can determine which document the model points.

To access the links dialog box click **Edit > Links**. The links document lists all links referenced by the correct document and their status.

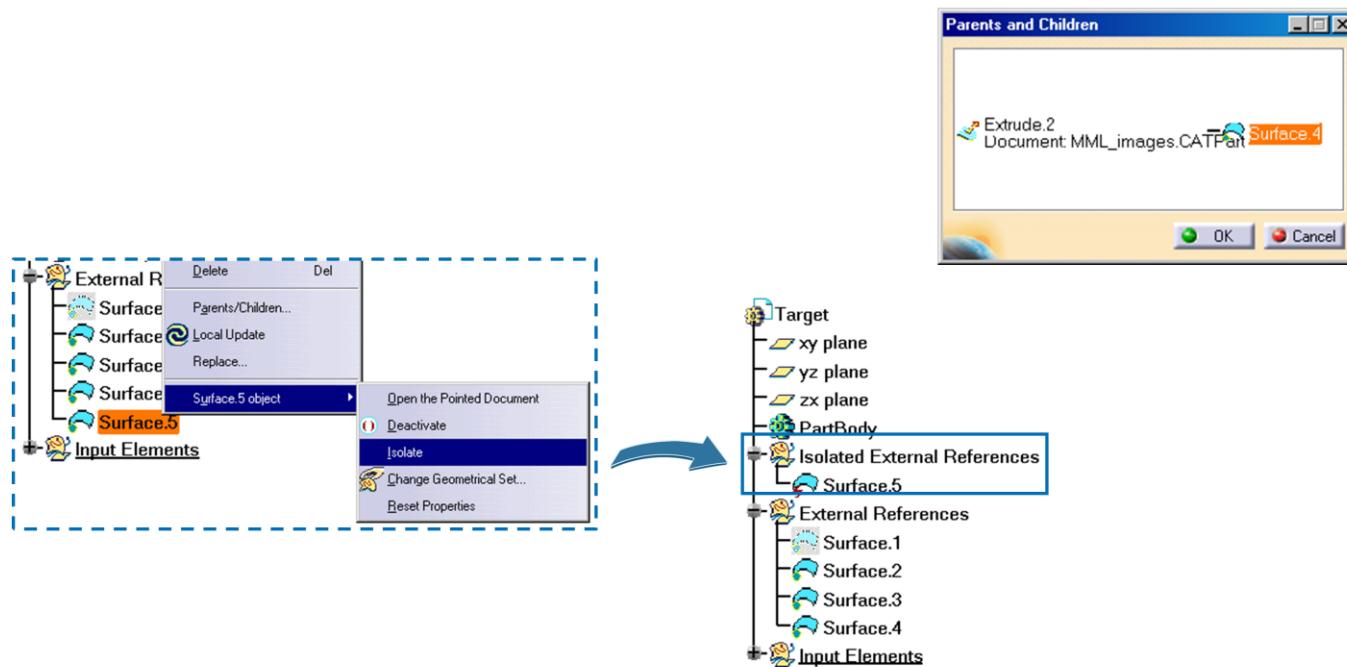
Use the links dialog box to Load, Synchronize, Activate/Deactivate, Isolate, or Replace referenced documents.



Managing Multi-Model Links (3/4)

Another way to determine the source document for a copied surface is to click **Parents/Children**. A dialog box displays the source document.

If you no longer want the target document to update changes to the source, you can break the link from the contextual menu. Click **Isolate** to break the link between the source and target documents. New Geometrical set is created called 'Isolated External Reference' the Isolated element is moved in to this Geometrical set.



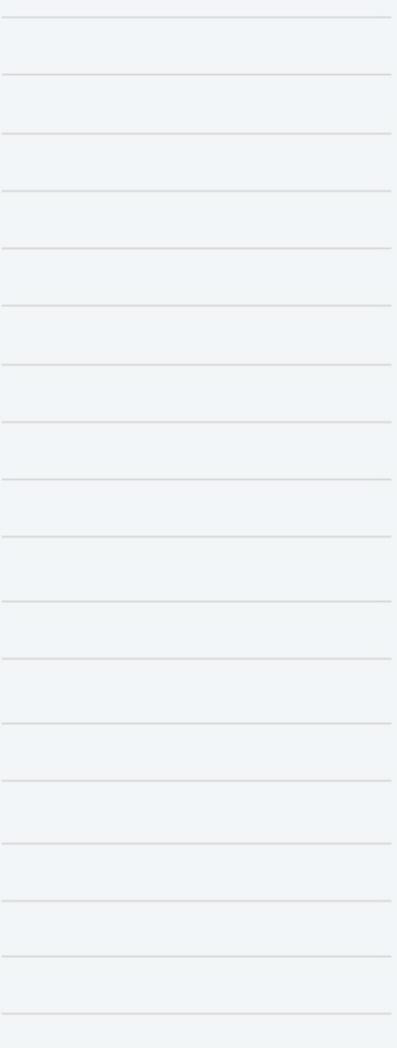
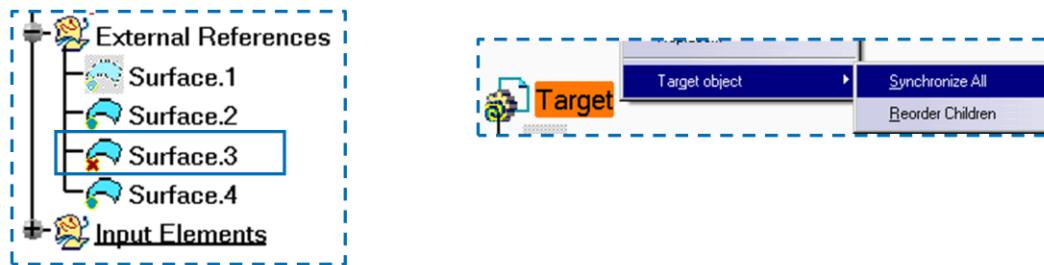


Managing Multi-Model Links (4/4)

When changes occur to the source document, the linked document will display a red X in its icon. This indicates that the link is not up to date. You can update the link using the links dialog box.

Another way to update a link without opening the Link dialog box is using the contextual menus. To update an individual link without opening the links dialog box, right-click on the solid and click **Solid.1 object > Synchronize** from the contextual menu.

To update all links in a model at the same time, right-click on the part and click **Part2 object > Synchronize All** from the contextual menu.



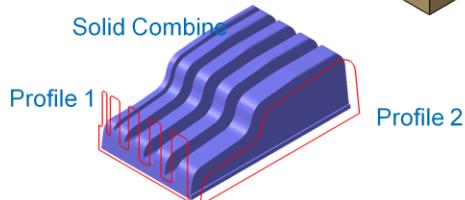
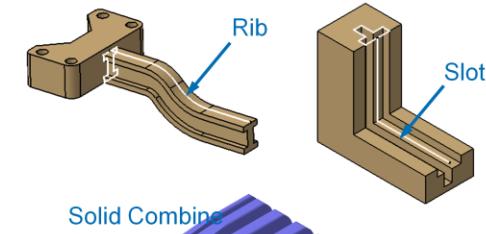
Lesson Summary (1/7)

Create Advanced Sketch-Based Features

Ribs and slots are created by sweeping a profile along a center curve.

A Solid Combine feature is created by the intersection of two components. These can be:

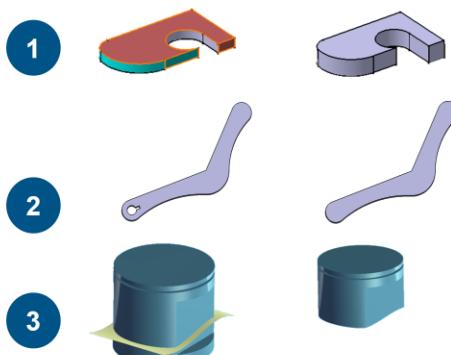
- ▶ Sketches
- ▶ Surfaces
- ▶ Sub-elements of sketches
- ▶ 3D Planar curves



Create Advanced Dress Up Features

The advanced dress up features are:

1. Thickness: Adds an over thickness to a face; used before machining the part.
2. Remove Faces: Used to simplify the geometry for downstream applications e.g. machining.
3. Replace Faces: Used to replace the planar solid surface with the surface.



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Lesson Summary (2/7)

Create Advanced Draft

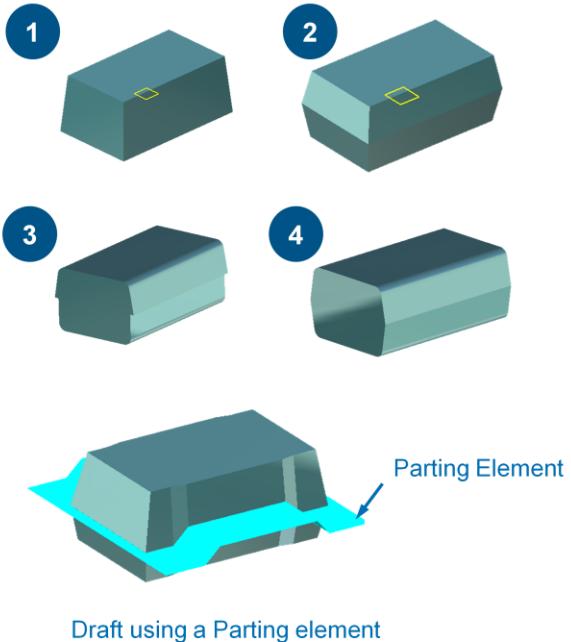
Advanced Drafts can be used to create basic line and reflect line drafts as well as drafts with two different angle values for complex parts.

Different types of advanced Drafts are possible:

1. A Standard draft with one side draft
2. A Standard draft with two sides draft
3. A draft using a reflect line
4. A draft using two reflect lines

While creating advanced drafts, the parting element can be selected. A Parting line represents the location where two halves of the mold meet.

A Parting element can be a line, surface or a face.



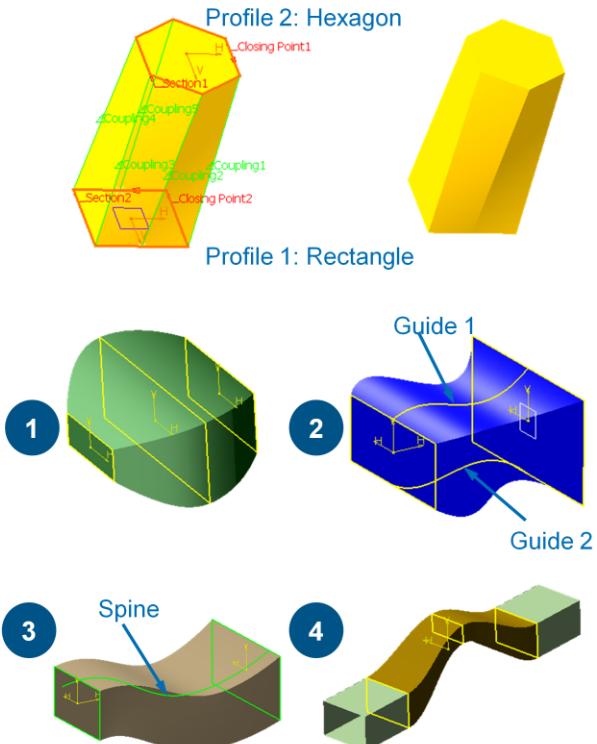
Lesson Summary (3/7)

Create Multi-Sections Solids (1/2)

A Multi-Sections solid can be positive (i.e., add material) or negative (i.e., subtract material). It is generated by two or more planar profiles swept along a spine.

Various types of Multi-Sections Solid are:

1. Simple Multi-Sections Solid: The selection order of the sections controls the shape of the result.
2. Multi-Sections Solid using Guide curve: The guide curves control the shape of the solid between the profiles. They must intersect the profile.
3. Multi-Sections Solid using Spine: The spine curve controls the shape of the features between the profiles.
4. Multi-Sections Solid Tangent to adjacent surfaces: The multi-sections solid is tangential to the adjacent solids / surfaces. Here the multi-sections solid acts as a transitional feature.





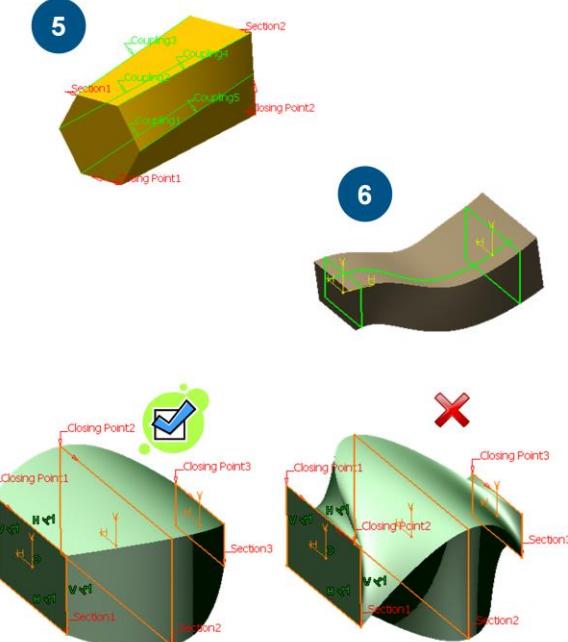
Lesson Summary (4/7)

Create Multi-Sections Solids (2/2)

5. Multi-Sections Solid using Couplings: The curves are coupled according to different criteria. These are as follows:
 - a. Ratio: The ratio of each section's length.
 - b. Tangency: Uses tangency discontinuity points.
 - c. Tangency then Curvature: uses the tangency discontinuity points first and then later the curvature discontinuity points.
 - d. Vertices: Uses the section's vertices.
 - e. Manual coupling: Used when various sections do not have the same number of vertices.
6. Multi-Sections Solid using Relimitations: By clearing the Relimitation options in the Relimitation tab, the result can be extended to the length of the spine or the guide curves.

Recommendations to avoid twisted surfaces:

- ▶ Choose appropriate Closing Points.
- ▶ Keep consistent directions.



Closing Point and Direction is correct
Closing Point selected is incorrect



Lesson Summary (5/7)

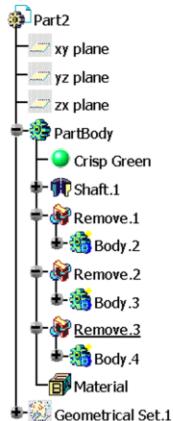
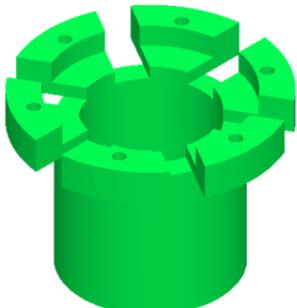
Use the Multi-Body Method

The Multi-Body Method allows you to design a complex part using simple bodies. Each body acts independently in the model. The final part is obtained by combining these bodies using Boolean operations.



The advantages of using the Multi-Body method are as follows:

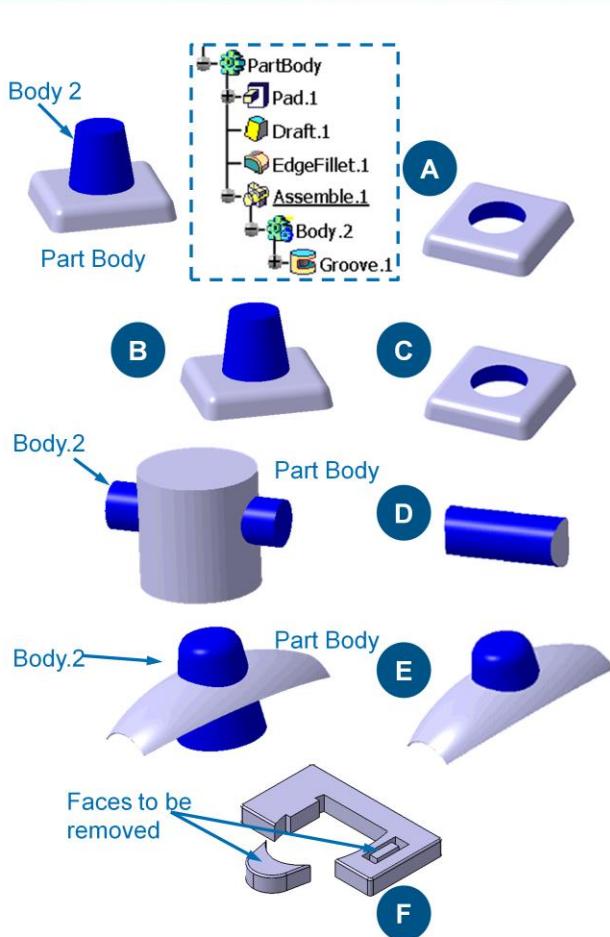
- ▶ It provides an organized approach to modeling complex parts.
- ▶ Solid features within a body can be hidden independently of the rest of the model.
- ▶ Groups of geometry can be de-activated by de-activating the body.
- ▶ Complex geometry is easier to create within a focused area of the model.
- ▶ The model will update faster due to the organized structure.



Lesson Summary (6/7)

Boolean Operations

- A. Assemble: The result will depend on the polarity of Body.2. A negative feature (pocket or groove), will remove material from the PartBody, a positive feature will add material.
- B. Add: A union of Body.2 and PartBody.
- C. Remove: Body.2 will cut PartBody.
- D. Intersect: The resulting solid is the material common to the intersecting elements.
- E. Union Trim: This operation is a union of the two bodies with the option to remove or keep selected faces.
- F. Remove Lump: A lump is material that is completely disconnected from the remainder of a single body, and may appear after certain operations. This operation is used select the faces to remove.





Lesson Summary (7/7)

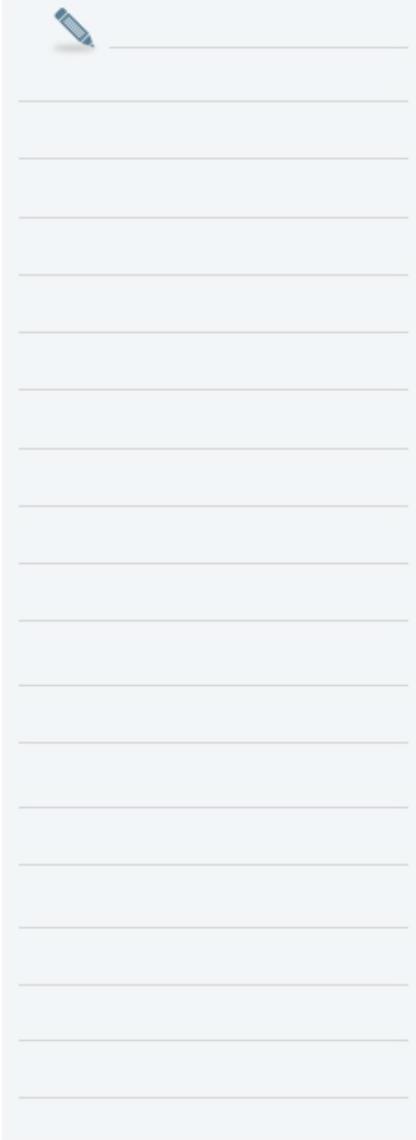
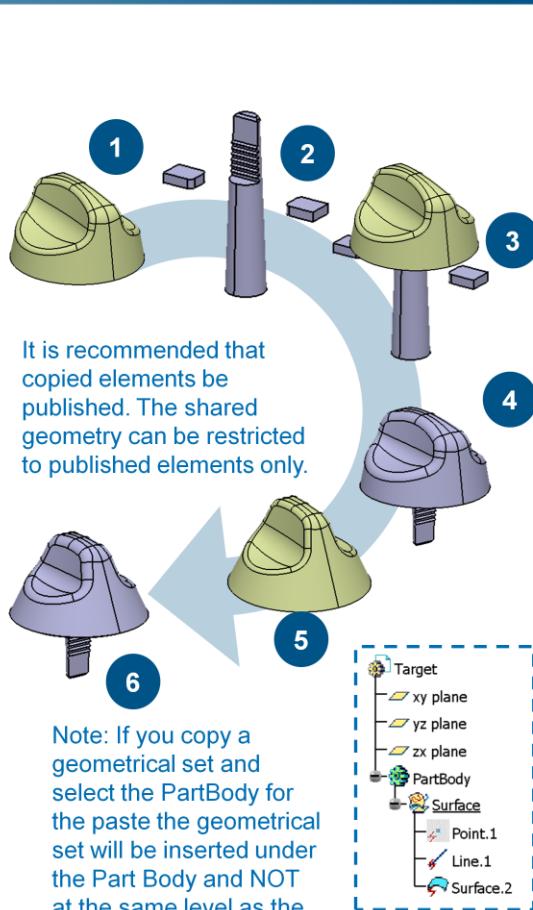
Create Multi-model Links

The use of Multi-Model Links enable you to design a model using elements from another model. This will enable you to update the part automatically if changes occur in the source model. To create links:

1. Copy a body in the source model.
2. In the target model, right-click on the Part and click Paste Special from the contextual menu.
3. Select As Result with Link. The Source PartBody is copied into the target model.
4. Complete the target model with the new body.
5. Modify the source model.
6. The target model is updated to take into account changes to the source model.

Choose the Paste Special option that best meets your design requirements:

- ▶ As Specified in the Part Document: The copied elements can be edited separately in the target part. A surface cannot be pasted in this way.
- ▶ As Result: The copied elements cannot be edited in the target part and are not linked to the source part.
- ▶ As Result with Link: The copied elements cannot be edited in the target part but are linked to the source part. A geometrical set cannot be pasted in this way.

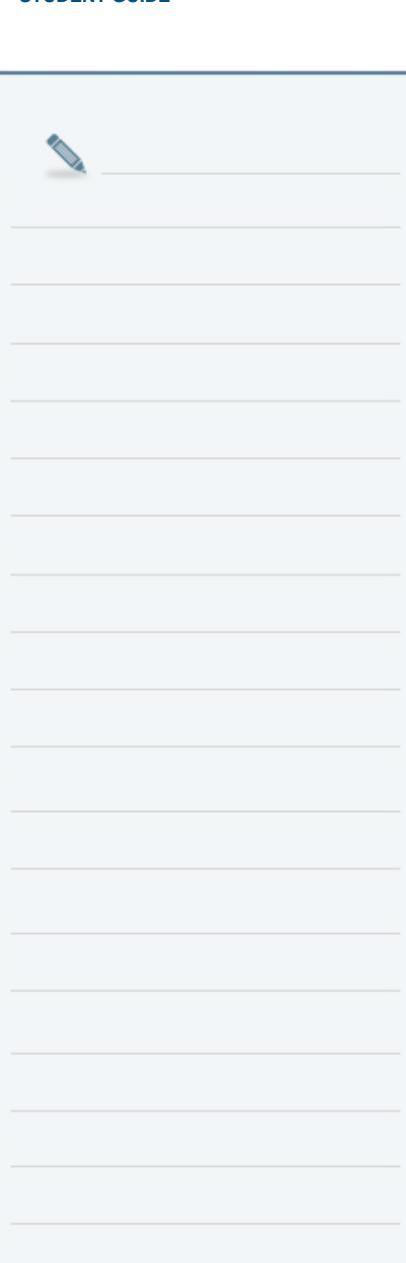
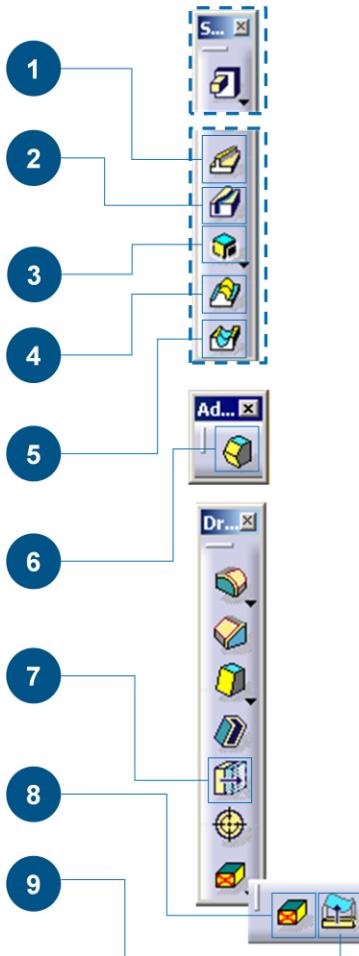
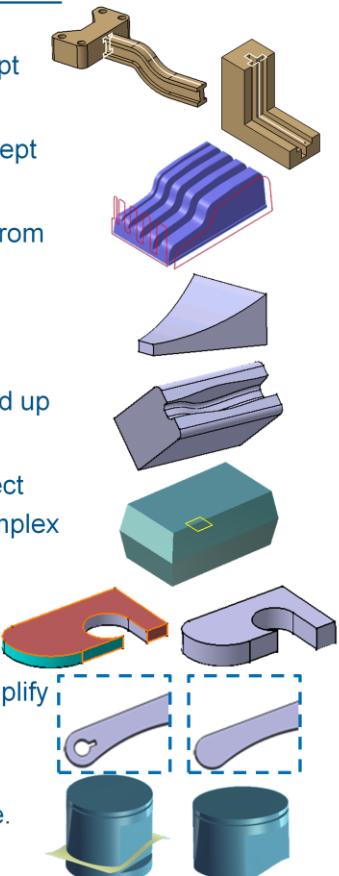




Main Tools (1/3)

Advanced Sketch Based Features

- 1 **Rib:** Creates a positive solid from a profile swept along the center curve.
- 2 **Slot:** Creates a negative solid from a profile swept along the center curve.
- 3 **Solid Combine:** Creates an intersection solid from the two extruded profiles.
- 4 **Multi-sections Solid:** Creates a positive solid joining multiple sections.
- 5 **Remove Multi-sections Solid:** Extrudes a solid up to a surface.
- 6 **Advanced Draft:** Creates a basic line and reflect line drafts with two different draft angles for complex parts.
- 7 **Thickness:** Adds / Removes thickness to a selected face or a surface.
- 8 **Remove Face:** Removes selected faces to simplify the geometry for finite element analysis / downstream applications.
- 9 **Replace Face:** Extrudes a solid up to a surface.

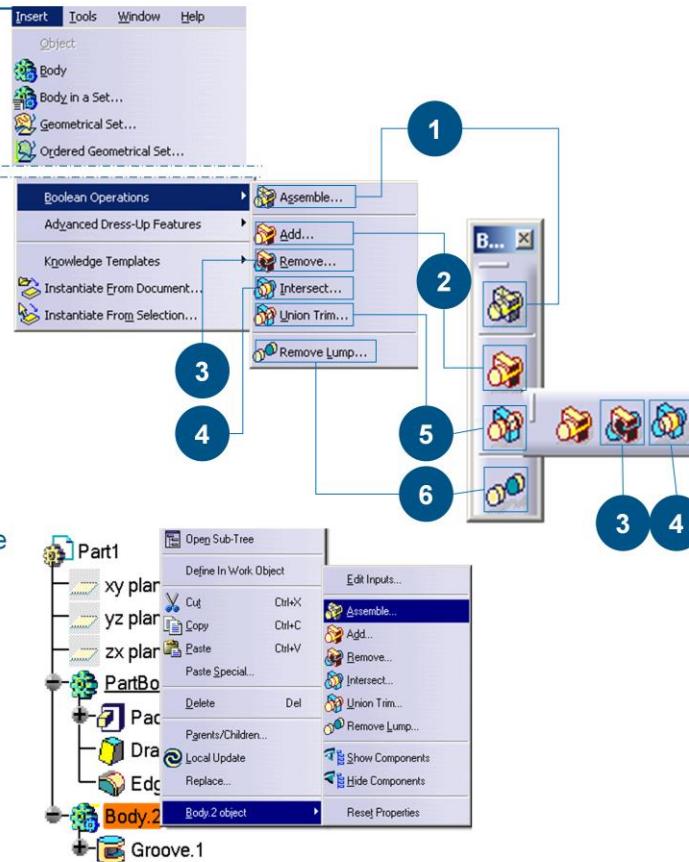




Main Tools (2/3)

Boolean Operations

- 1 **Assemble:** Creates a union of two bodies, the union respects the true nature of the bodies. (Positive features add material, negative features remove material).
- 2 **Add:** Creates a union of two bodies.
- 3 **Remove:** Removes selected body from the PartBody.
- 4 **Intersect:** Creates an intersection solid from the selected bodies.
- 5 **Union Trim:** Creates an intersection solid from the selected bodies with an option to remove or keep one side.
- 6 **Remove Lump:** Removes selected faces (lumps and cavities).

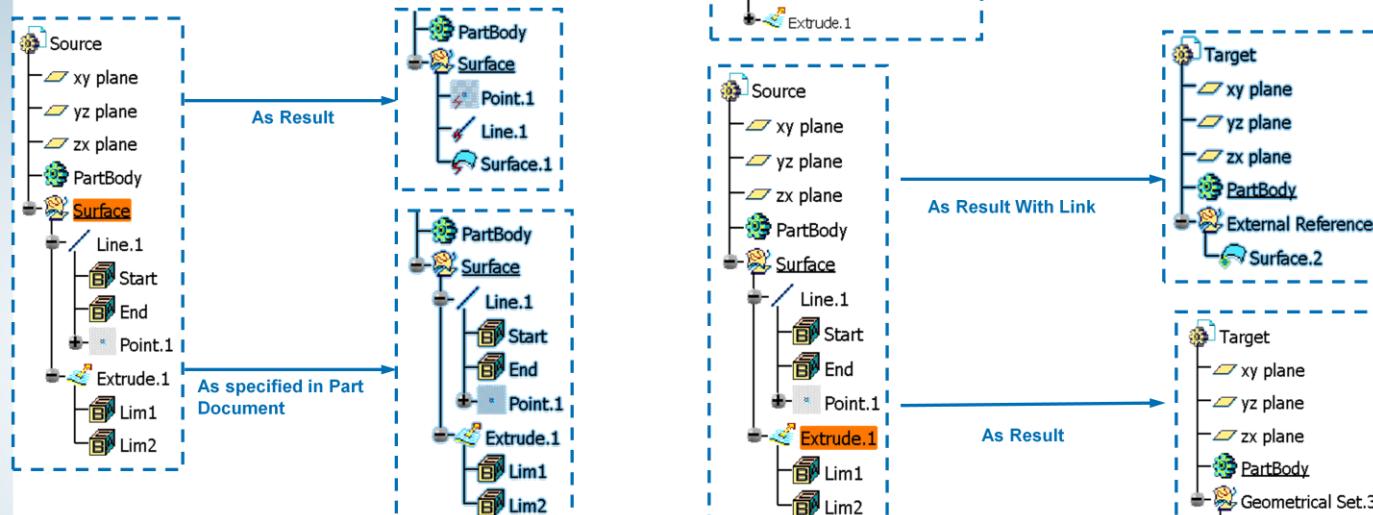




Main Tools (3/3)

Multi-Model Links

- 1 **Copy:** Copies the selected features.
- 2 **Paste Special:** Pastes the selected features into the destination.



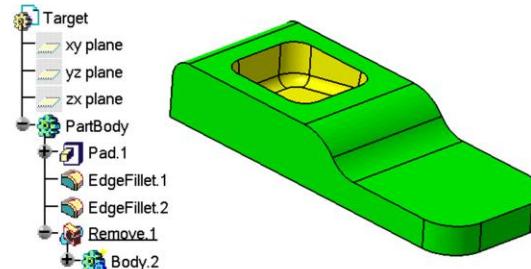


Exercise: Create a Part Using Multi-Body Method

Here, you will open an existing part, containing a single feature, and use the tools learnt in the lesson to insert a body from another model. You will use Boolean operations to remove the copied body from the main part. Detailed instructions for this exercise are provided.

By the end of this exercise you will be able to:

1. Create Multi-Model links.
2. Perform Boolean Operations.
3. Modify Multi-Linked Models.



20 minutes

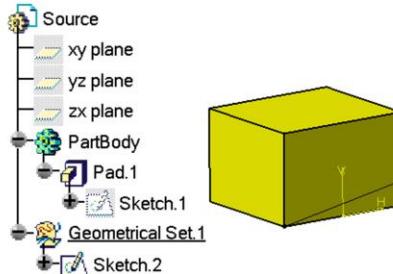




Create a Part Using Multi-Body Method (1/4)

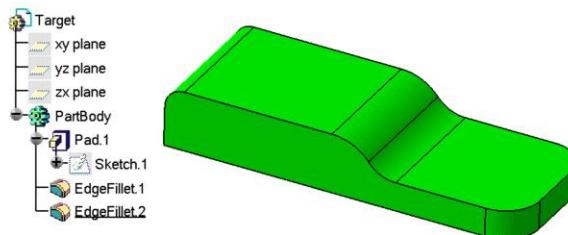
1. Open the source part file.

Open the existing part file, Source.CATPart. This file contains the body that will be removed from the main body.



2. Open the target part.

Open the existing part file, Target.CATPart. This file contains the main body for the model.



Handwritten notes area for the student guide.

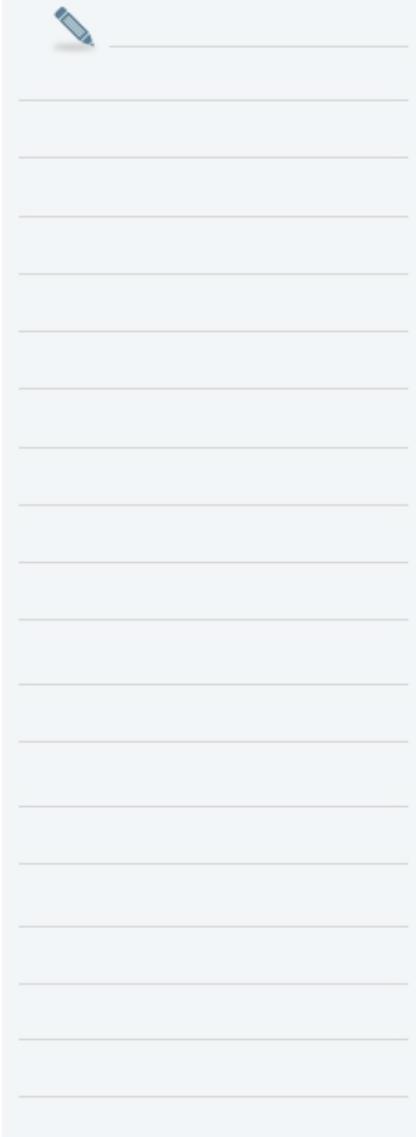
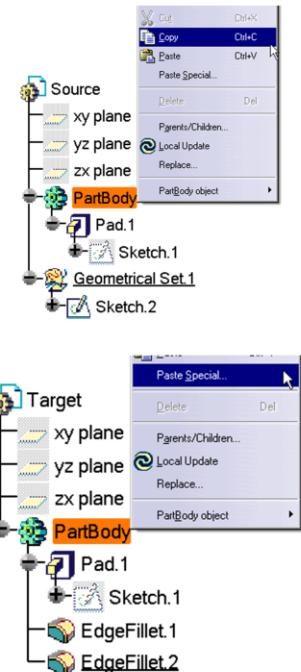
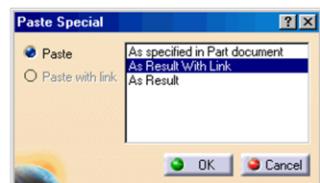


Create a Part Using Multi-Body Method (2/4)

3. Copy a body.

Copy the body from the source file to the target file.

- a. Activate the source part file.
- b. Right-click on the PartBody from the specification tree.
- c. Click **Copy** from the contextual menu.
- d. Activate the target part file.
- e. Right-click on the PartBody in the specification tree.
- f. Click **Paste Special** from the contextual menu.
- g. Select **As Result With Link** from the Paste Special dialog box.
- h. Click **OK**.

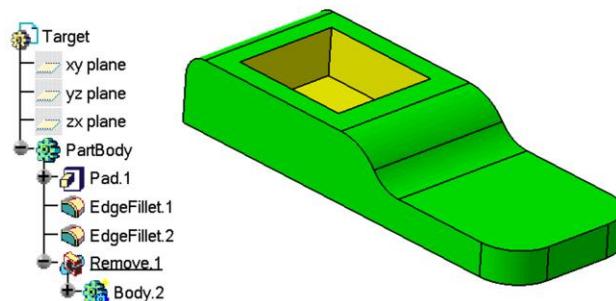
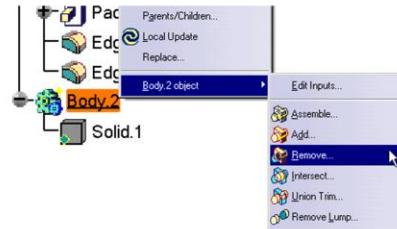


Create a Part Using Multi-Body Method (3/4)

4. Remove the body.

Remove the copied body from the main body.

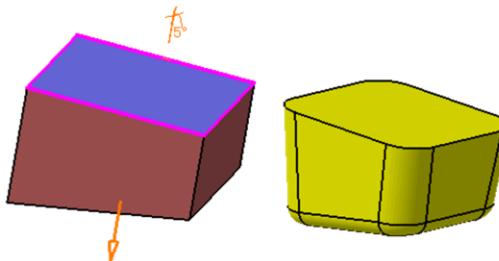
- a. Right click on the copied body.
- b. Click Body.2 object > Remove from the contextual menu. Since the only other body in the model is the PartBody, the body is automatically removed from it.



Create a Part Using Multi-Body Method (4/4)

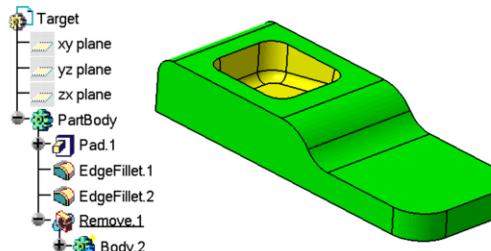
5. Modify the source document.

- Add fillets and draft to the source document.
- Activate the source document.
 - Create a pocket using Sketch.2.
 - Apply a 5 degree draft to the sides of the model. Use the top surface as the neutral element.
 - Add 8mm edge fillets to the four corners
 - Add 4mm edge fillet to the bottom edge.



6. Update the target file.

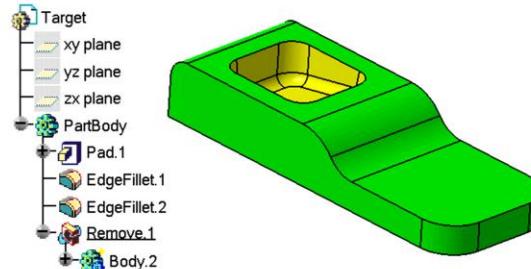
- Update the target file and to include the changes made to the source document.
- Activate the target document.
 - Click Edit > Update.



Recap: Create a Part Using Multi-Body Method

In this exercise you have:

- ✓ Created multi-model links
- ✓ Performed a remove operation
- ✓ Modified multi-model link models



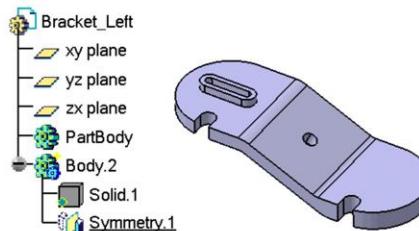


Exercise: Create a Part Using Multi-Body Method

In this exercise, you will open an existing part that contains a single feature. You will use the tools learned in this lesson to perform a Boolean operation, and create a multi-model link. High-level instructions for this exercise are provided.

By the end of this exercise you will be able to:

1. Create Multi-Model links.
2. Perform Boolean Operations.
3. Modify Multi-Linked Models.



Create the Part (1/2)

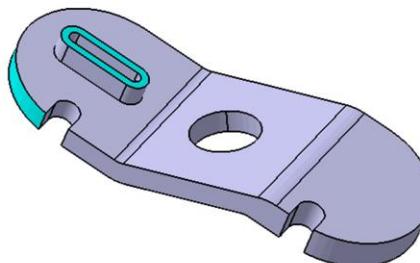
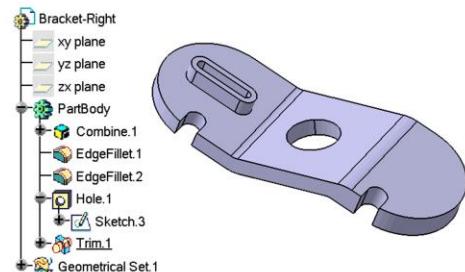
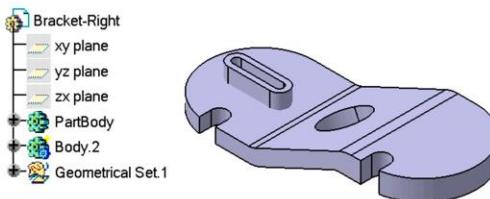
1. Open the part file.

Open the existing part file, Bracket_right.CATPart.

There are two bodies in this file.

2. Perform a **Union Trim** operation on the PartBody using Body.2.

Use the **Union Trim** operation to trim Body.2 from the PartBody. Keep the top surface of Body.2 and the cylindrical surface from the PartBody.



Create the Part (2/2)

3. Create a new part file.

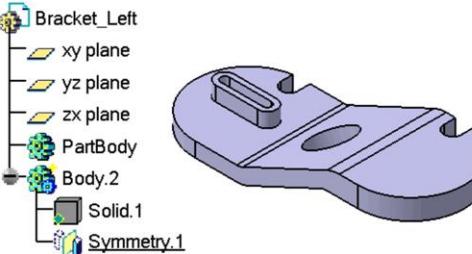
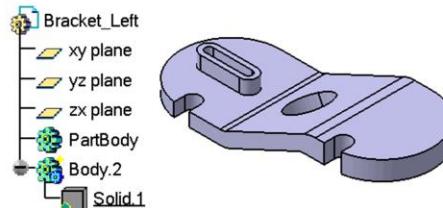
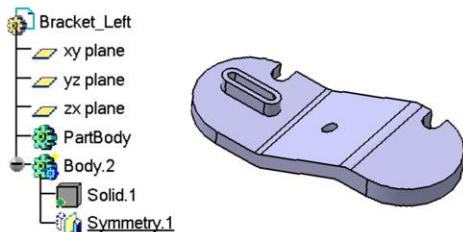
Create a new PartBody called Bracket_Left.
Create a multi-model link to the PartBody in
Bracket_right.

4. Transform features.

Use the **symmetry** tool to transform the notches in
the Bracket_Left model. Perform the symmetry
operation about the YZ plane.

5. Modify the hole in Bracket_Right.

Modify the hole dimension in Bracket_Right to [5
mm] and update both the models.

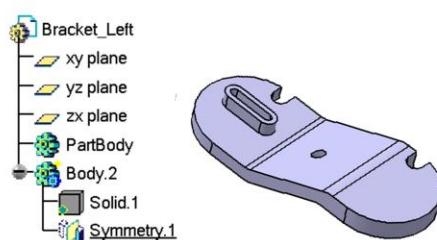
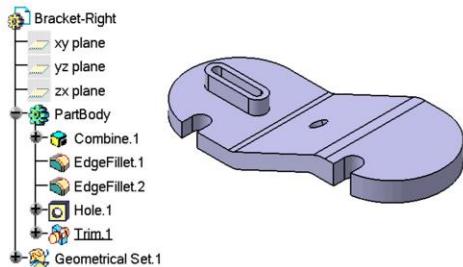




Recap: Create a Part Using Multi-Body Method

In this exercise you have:

- ✓ Created multi-model links
- ✓ Performed a union trim operation
- ✓ Modified multi-model link models



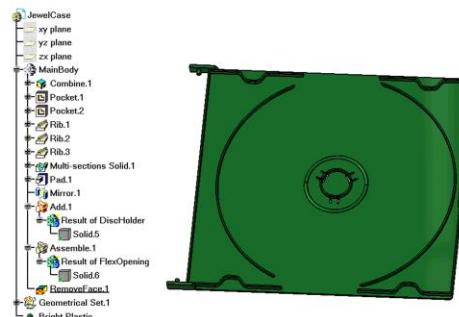


Case Study: Design Complex Parts

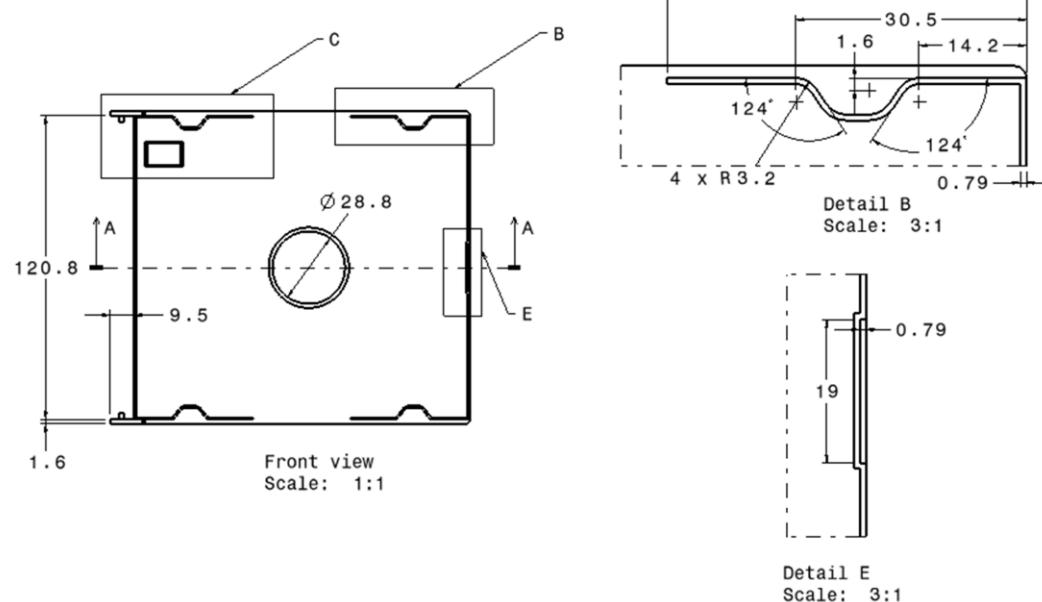
In this exercise you will create the case study model. Recall the design intent of this model:

Using the techniques you have learnt in this and previous lessons, create the model without detailed instructions.

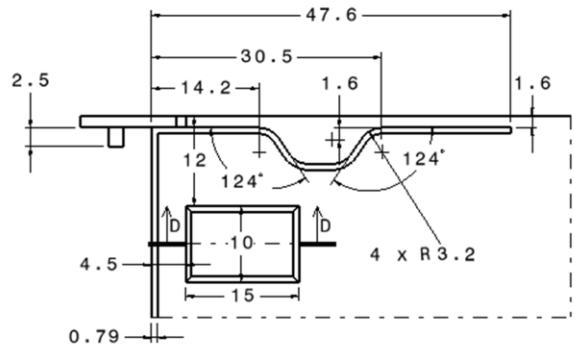
- ▶ Base feature must include overall dimensions supplied
- ▶ Create each support as a single feature
- ▶ A cut is to be created to simulate the logo. The cut profile varies
- ▶ Links must be created to the Disk holder and the flex opening models to ensure conformance to standards.
- ▶ Linked features must be kept in separate bodies.
- ▶ An indented logo should not be displayed when it goes for manufacturing



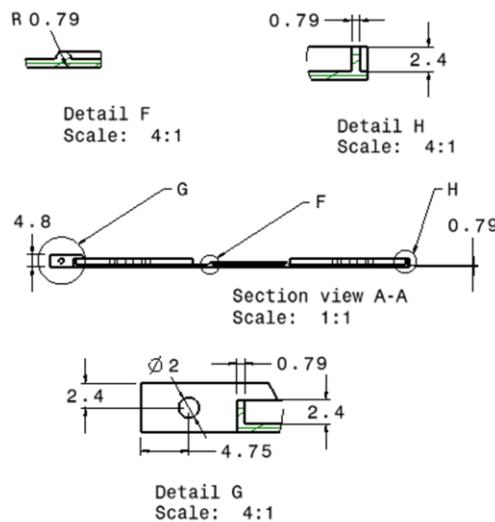
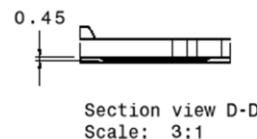
CD Jewel Case Reference Drawings (1/2)



CD Jewel Case Reference Drawings (2/2)



Detail C
Scale: 3:1



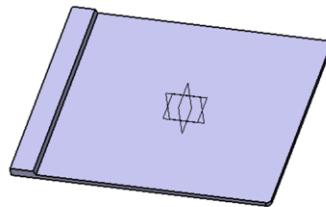
Design the Part (1/5)

You must complete the following tasks with the help of the drawing attached at the end:

1. Create a solid combine.

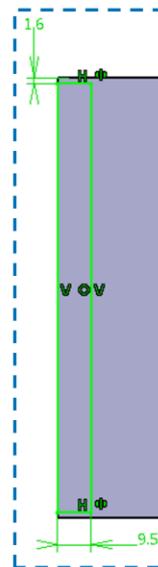
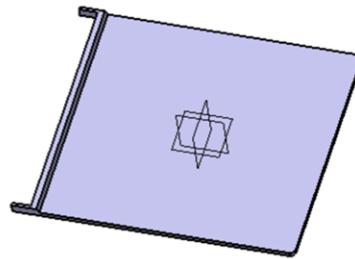
Load JewelCase.CATPart

Use the two sketches supplied to create a solid combine feature.



2. Create a pocket.

Create a pocket using the dimensions shown in the image.

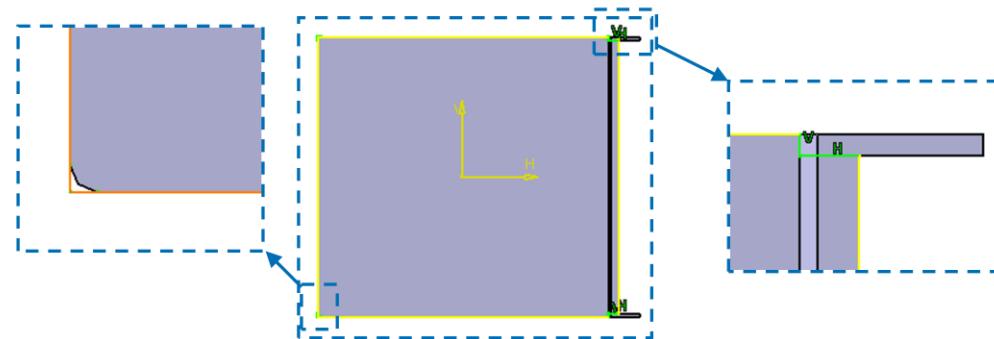
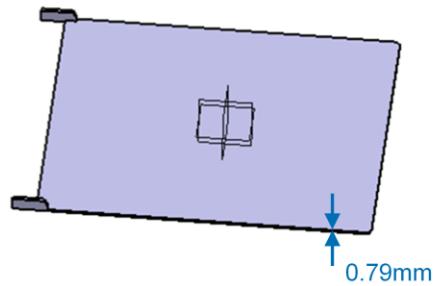


Design the Part (2/5)

You must complete the following tasks (continued):

3. Create a pocket.

Create a second pocket using the dimensions shown. The cut is symmetrical about the ZX plane. This pocket needs to cut the material such that only a 0.79mm thickness is left.



Notes:



Design the Part (3/5)

You must complete the following tasks (continued):

4. Create a rib feature.

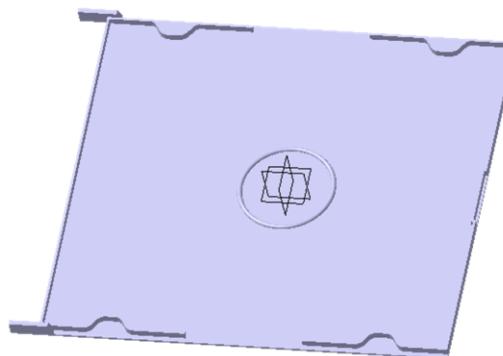
Create a rib feature using the dimensions shown on Detail view C and G of the drawing. The rib is symmetric about the ZX plane.

5. Create a second rib feature.

Create a rib feature using the dimensions shown on Detail view B, E and H of the drawing. The rib is symmetric about the ZX plane.

6. Create a third rib feature.

Create a rib feature using the dimensions shown on detail view F and the front view of the drawing.

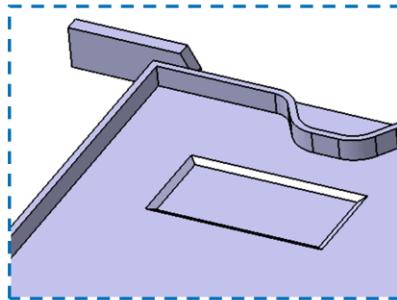


Design the Part (4/5)

You must complete the following tasks (continued):

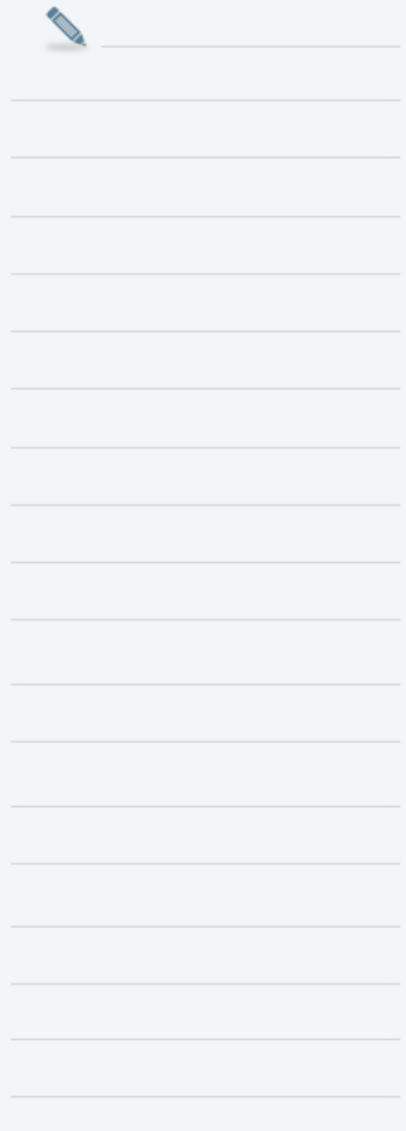
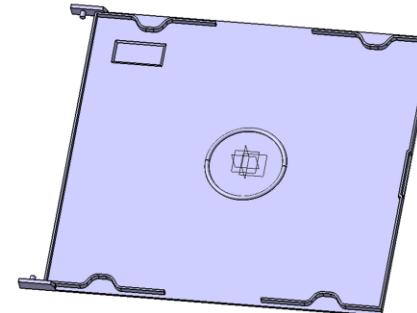
7. Create a removed multi-sections solid.

Create the logo using a removed multi-sections solid. The lower profile is created on a reference plane that is offset 0.45mm below the top surface of the case. Use Detail view C and Section view D-D for the dimensions.



8. Create two pad features.

Create two pad features using the dimensions shown on Detail views C and G. Consider creating only one Pad feature and mirroring it to create the other.



Design the Part (5/5)

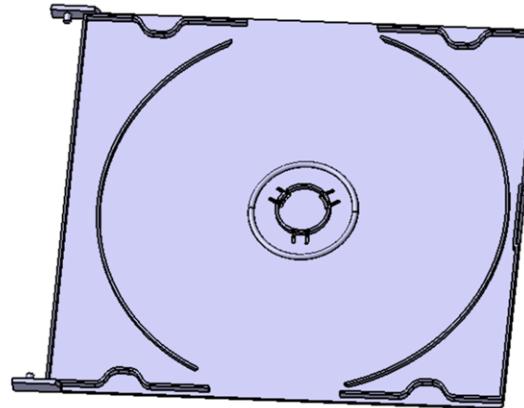
You must complete the following tasks (continued):

9. Copy the DiskHolder and FlexOpening bodies.

Copy the DiskHolder and the FlexOpening bodies from JewelCaseSubPart.CATPart using the Paste Special option As Result With Link.

10. Assembly the FlexOpening body to the main body.

11. Use the remove face tool to remove the logo from display.



Handwritten notes area: [This section is a large, empty white space for handwritten notes.]

Recap: Design Complex Parts

In this exercise you have:

- ✓ Created a solid combine
- ✓ Created rib features
- ✓ Created a removed multi-sections solid
- ✓ Created multi-model links
- ✓ Performed Boolean operations
- ✓ Removed a face.

