ASCENT®
CENTER FOR TECHNICAL KNOWLEDGE

CATIA: Advanced Surface Design

Version 5, Release 20

Student Guide

Revision 1.0
June 2011

ASCENT - Center for Technical Knowledge®
CATIA: Advanced Surface Design
Version 5, Release 20, Revision 1.0

Prepared and produced by:

ASCENT Center for Technical Knowledge
1001 E. Market Street, Suite 102
Charlottesville, VA 22902

866-527-2368
www.ascented.com

ASCENT - Center for Technical Knowledge is a division of RAND Worldwide Inc., providing custom developed knowledge products and services for leading engineering software applications. ASCENT is focused on specializing in the creation of education programs that incorporate the best of classroom learning and technology-based training offerings.

We welcome any comments you may have regarding this training manual, or any of our products. To contact us please email: education@ASCENTed.com.

CATIA is a registered trademark of Dassault Systèmes.

Table of Contents

Chapter 1 Surface Design Overview 1-1

1.1 Surfacing Tools in CATIA V5 1-3

1.2 Generative Shape Design Workbench 1-4
Access the GSD Workbench 1-4
GSD User Interface 1-4

1.3 Surface Design Review 1-5

1.4 Design Intent 1-10
Surface Modeling Methods 1-10
Boundary Representations 1-13

1.5 Tree Organization 1-15
Groups 1-15
Mask 1-18

Exercise 1a Hybrid Surface Modeling 1-21

Exercise 1b Front Quarter Panel 1-37

Exercise 1c Using a Mask 1-46

Chapter 2 Advanced Wireframe Elements 2-1

2.1 Generative Shape Design Workbench 2-3

2.2 Reference Geometry 2-4
Extremum 2-4
ExtremumPolar 2-6

2.3 Projected Geometry 2-10
Combine 2-10
ReflectLine 2-13

2.4 Offset Geometry 2-16
3D Curve Offset 2-16

2.5 Curve Geometry 2-19
Spine 2-19
Isoparametric Curve 2-21

2.6 Circle-Conic Geometry 2-23
Conic 2-23

2.7 Trim (Pieces) 2-27

2.8 Create Multi-Extract Elements 2-28
Extract vs Multi-Extract 2-31

2.9 Extrapolate Operation 2-33

Exercise 2a Spine Curve 2-35

Exercise 2b Wing Surface **2-40**
Exercise 2c Earbud Parting Line **2-49**
Exercise 2d Isoparametric Curve (Optional) **2-57**

Chapter 3 Curve Analysis and Repair **3-1**
3.1 Terminology Review **3-3**
Curvature 3-3
Inflection Point 3-3
Degrees of Continuity 3-3
3.2 Analysis and Repair **3-6**
3.3 Analyzing Curves **3-7**
Connection Between Curves 3-7
Projection On a Curve 3-7
Connection Between a Curve and a Surface 3-8
3.4 Porcupine Curvature Analysis **3-14**
3.5 Repair Curve Geometry **3-21**
3.6 Manual Curve Repair Tools **3-31**
Untrim Curve 3-31
Disassemble 3-32
Exercise 3a Curve Analysis **3-34**
Exercise 3b Earbud Curve Repair **3-47**
Exercise 3c Known Element Method **3-63**

Chapter 4 Blend Surfaces **4-1**
4.1 Blended Surfaces Overview **4-3**
4.2 Optional Blend Elements **4-5**
Trim Supports 4-5
Tangent Borders 4-6
Tension 4-7
Coupling 4-8
Developable 4-13
Exercise 4a Simple Blend **4-17**
Exercise 4b Diablo **4-26**

Chapter 5 Adaptive Sweeps **5-1**
5.1 Adaptive Sweep **5-3**
Exercise 5a Adaptive Sweep Surfaces I **5-9**
Exercise 5b Adaptive Sweep Surfaces II **5-15**
Exercise 5c Rear View Mirror **5-19**

Exercise 5d Air Intake Tube 5-26

Chapter 6 Additional Sweep Types 6-1

6.1 Swept Surface Types 6-3

6.2 Line Profile 6-7
Two Limits 6-7
With Draft Direction 6-8
With Tangency Surface 6-10

6.3 Circle Profile 6-11
Center and Two Angles 6-11
Two Guides and Tangency Surface 6-11

6.4 Conic Profile 6-13
Two Guide Curves 6-13
Five Guide Curves 6-14

Exercise 6a Remote Control Sweeps 6-16

Exercise 6b Helix Sweep 6-21

Exercise 6c Exhaust Diffuser 1 6-33

Chapter 7 Optional Elements - Sweeps 7-1

7.1 Sweep Optional Elements 7-3

7.2 Explicit Sweep Optional Elements 7-14
Anchor Points 7-14
Position Profile 7-19

Exercise 7a Explicit Swept Surface 7-21

Exercise 7b Using Spines to Build Surfaces 7-30

Exercise 7c Twisted Areas Management 7-48

Chapter 8 Advanced Filleting 8-1

8.1 Surface Fillet Overview 8-3

8.2 Chordal Fillet 8-5

8.3 Tritangent Fillet 8-7

8.4 Trim Options 8-9
Trim Supports 8-9
Trim Ribbons 8-10

8.5 Fillet Orientation Options 8-11
Spine/Circle 8-11
Hold Curve 8-12

8.6 Fillet Boundary Options 8-13
Edges to Keep 8-13
Limiting Element 8-14

Blend Corner 8-14

8.7 Alternative Methods of Filleting 8-16
Sweeps 8-16
Blends 8-17
Multi-Sections Surface 8-17
Fill 8-17

Exercise 8a Advanced Fillet Options 8-18

Exercise 8b Conic Fillets 8-26

Exercise 8c Curvature Continuous Fillet 8-35

Exercise 8d Complex Fillets 8-43

Chapter 9 Laws 9-1

9.1 Law Definition 9-3

9.2 Creating a Law Feature 9-8
X-parameter on Definition 9-11

9.3 Using Laws 9-15
Sweeps 9-15
Parallel Curves 9-17
Helix 9-18
Shape Fillets 9-18

Exercise 9a Law Driven Shape Fillet 9-21

Exercise 9b Using Laws 9-33

Exercise 9c Advanced Laws 9-38

Exercise 9d Drill Auger 9-45

Chapter 10 Duplication Tools 10-1

10.1 Duplication Tools 10-3

10.2 Repetition 10-4
Object Repetition 10-4
Points and Planes Repetition 10-5
Planes Repetition 10-5

10.3 Patterning Surface Features 10-7

10.4 Duplicate a Geometrical Set 10-8

Exercise 10a Surface Patterns 10-12

Exercise 10b Copying Geometrical Sets 10-19

Exercise 10c Duplicate Geometrical Set (Optional) 10-27

Exercise 10d Speaker Cover 1 10-35

Chapter 11 Knowledge Templates 11-1

11.1 Knowledge Templates 11-3

11.2 Create a PowerCopy 11-4
Preview 11-9

11.3 Instantiate a PowerCopy 11-11

11.4 Catalogs 11-15

Exercise 11a PowerCopy Creation 11-18

Exercise 11b Thread PowerCopy 11-28

Chapter 12 Surface Analysis and Repair 12-1

12.1 Analysis and Repair 12-3

12.2 Surface Error Display 12-4
3D Accuracy 12-4
Display Mode 12-4
Color 12-5
WYSIWYG Mode 12-6

12.3 Analyzing Surfaces 12-7

12.4 Porcupine Curvature 12-15

12.5 Surface Curvature Analysis 12-19

12.6 Temporary Analysis 12-27

12.7 Repairing Surface Geometry 12-29

12.8 Element Direction and Inversion 12-34
Curve/Surface Orientation 12-34
Identification 12-35
Invert Orientation 12-35
Downstream Effects 12-35

12.9 Tips for Rebuilding Geometry 12-36

Exercise 12a Distance Analysis 12-37

Exercise 12b Problem Pipe 12-44

Exercise 12c Imported Surface 12-54

Exercise 12d Repairing Surfaces 12-59

Chapter 13 Offset Surfaces 13-1

13.1 Offset Overview 13-3

13.2 Offset Smoothing 13-5
Smoothing - None 13-5
Smoothing - Automatic 13-6
Smoothing - Manual 13-7

13.3 Sub-elements to Remove 13-8
Exercise 13a Plastic Top 13-11

Chapter 14 Advanced Surfacing Projects 14-1
14.1 Surfacing Overview 14-2
Exercise 14a Exhaust Diffuser 14-5
Exercise 14b Offset Surface Project 14-10
Exercise 14c Surfacing for Solids Project 14-12

Chapter 1

Surface Design Overview

This chapter serves as an overview of wireframe and surface design fundamentals and the typical surface design process.

This chapter introduces:

- ✓**Surfacing Tools in CATIA V5**
- ✓**Generative Shape Design Workbench**
- ✓**Surface Design Review**
- ✓**Design Intent**
- ✓**Tree Organization**

1.1 Surfacing Tools in CATIA V5

Surface features introduce additional flexibility to a model. Surfaces can be used to define a complete part or to integrate a complex shape into the solid part (in the Part Design workbench). As learned previously, many parts can be designed directly using solid geometry. However, when shapes become more complex, surface shapes must often be used.

CATIA has several workbenches available to create surface geometry. The primary workbench is Generative Shape Design (GSD).

The GSD workbench contains the required tools for wireframe and surface design. Other surfacing workbenches available in CATIA are shown in Figure 1–1.

Figure 1–1

The Wireframe and Surface Design workbench has basic wireframe and surfacing capabilities.

The Freestyle workbench provides design tools for an Industrial Designer. Non-parametric wireframe and surface geometry can be quickly developed and optimized around conceptual drawings and models.

The Imagine & Shape workbench contains non-parametric surface creation tools. The surfaces are NURBS based, enabling pushing and pulling on a mesh. Organic and freeform type shapes can easily be created.

1.2 Generative Shape Design Workbench

Access the GSD Workbench

Until now, you normally worked in the Mechanical Design area of CATIA to develop part and surface models. Solid geometry was created in the Part Design workbench. To access the advanced surfacing functionality in CATIA, you must use the GSD workbench. To access this workbench, select **Start > Shape > Generative Shape Design**. The workbench symbol changes to  .

GSD User Interface

The interface for the GSD workbench is similar to the Part Design workbench. The primary difference is that the toolbar options change to GSD-specific tools, as shown in Figure 1–2.

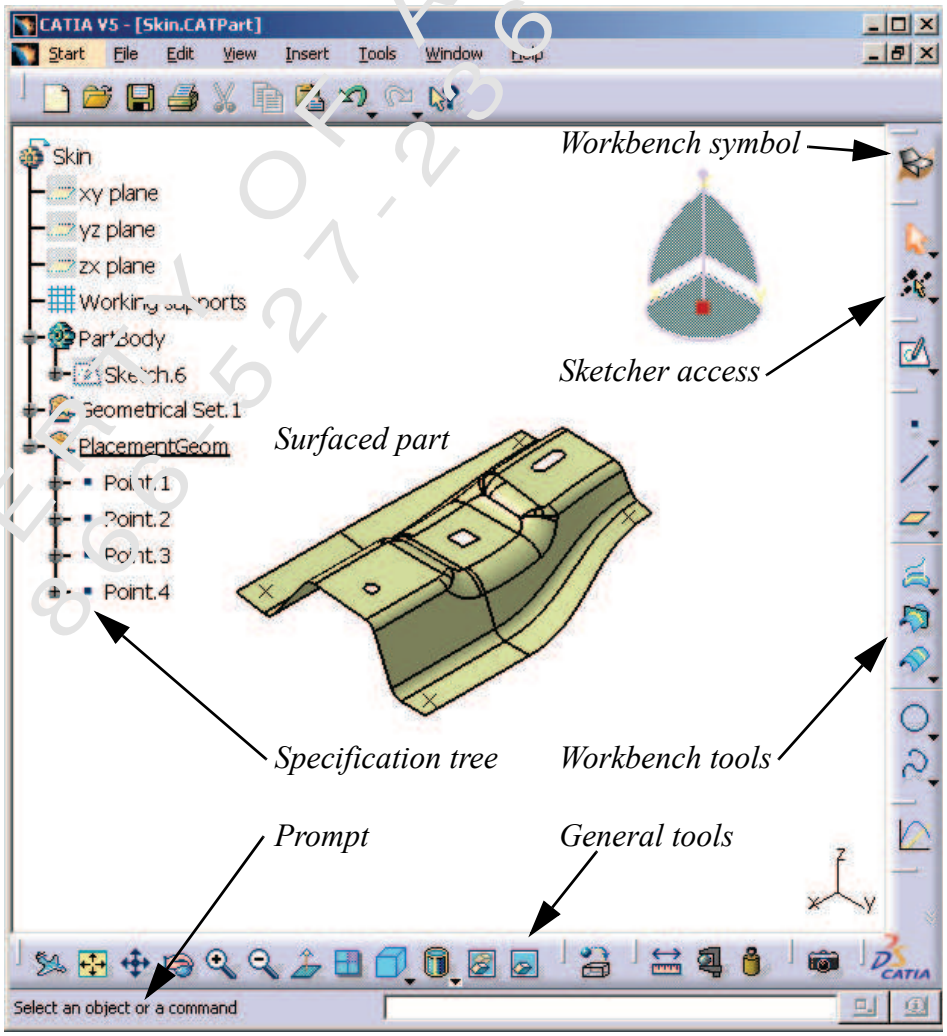

Figure 1–2

1.3 Surface Design Review

Designing surface features differs slightly from designing solid features in CATIA. With solid modeling, you can often visualize the final shape of a feature and create the geometry with individual construction features (e.g., Pad, Pocket, Fillet, or Hole). With surfaces, you must frequently create reference geometry (e.g., points and curves) before creating a single surface feature. Planning ahead is essential to achieving the desired results when designing surfaces.

When creating solid features, all faces of the feature must be defined in one step. The intersections of solid features are immediately calculated and consumed in the model. With surfaces, the contours of the model can be individually created as separate features. These multiple surfaces can then be joined to generate a final solid feature. As a result, simple surfaces have the benefit of being able to yield complex results.

General Steps

Use the following general steps to create a model from surface features:

1. Create wireframe geometry.
2. Create surface geometry.
3. Perform surface operations.
4. Create solid geometry.

Step 1 - Create wireframe geometry.

Wireframe geometry is the backbone on which the surface features of the model are created. You can use wireframe features to define construction elements, intersections, and common boundaries of the surfaces that define the shape of the model. Wireframe geometry can consist of simple features, such as sketches, points, lines, and planes, as well as more complex geometry, such as splines or helixes.

Figure 1–3 shows an example of a group of wireframe features that are used to develop a surface model.

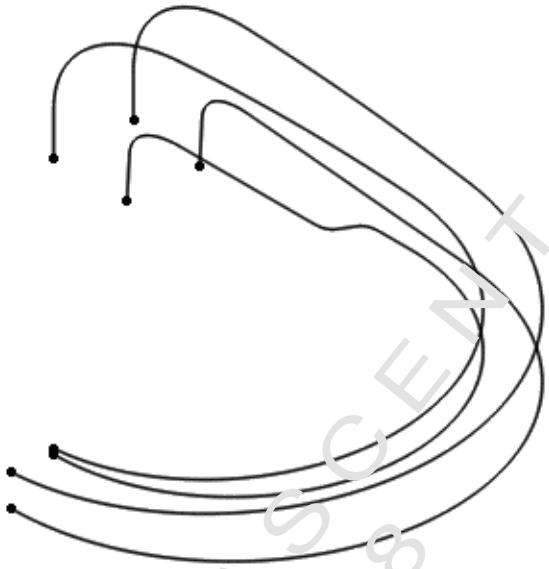

Figure 1–3

Step 2 - Create surface geometry.

Once the wireframe geometry has been created, surface features can be created to define the internal and external boundaries of the model. Surface features can be created independent from the rest of the model or by using existing wireframe and surface geometry as a reference. Common surface types include:

- **Extruded**
- **Revolved**
- **Spherical**
- **Offset**

The boundaries of the model shown in Figure 1–4 are completely defined. Although the surfaces appear to form a closed shape, they still need to be joined before making the solid model.

Figure 1–4

Step 3 - Perform surface operations.

The advantage of using surface features is being able to control how individual surface features connect and interact with other parts of the model. Surface operations are used to control this interaction by enabling you to manipulate existing surface features. Common ways to manipulate surfaces include:

- **Join**
- **Split**
- **Trim**
- **Extract**
- **Transform**

The model shown in Figure 1–5 uses the **Join** option to define the resulting surface shape.

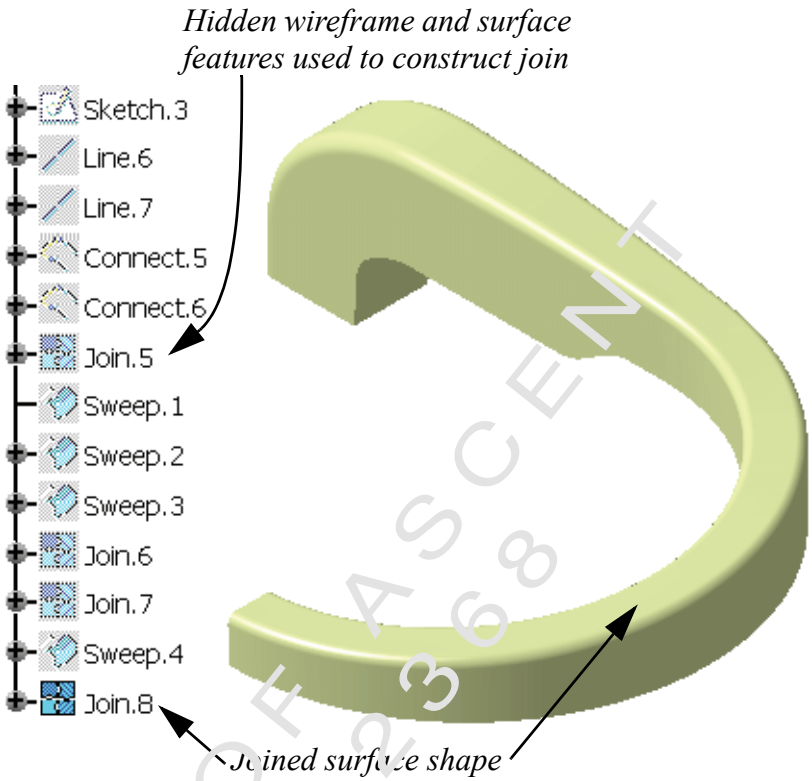

Figure 1–5

Step 4 - Create solid geometry.

When you finish creating surface features, you are ready to create solid geometry. At this point, your model contains a surface representation of the 3D model. This can be a closed surface or an open surface depending on your design intent.

If the mass properties of your model are required or you intend on performing additional solid modeling, you must create solid geometry from the surfaces. You must take the model back to the Part Design workbench and use the Surface-Based Features toolbar options to create solid geometry from the skin. These options include **Close Surface**, **Thick Surface**, **Split**, and **Sew**.

The solid geometry is added to the PartBody, as shown in Figure 1–6.

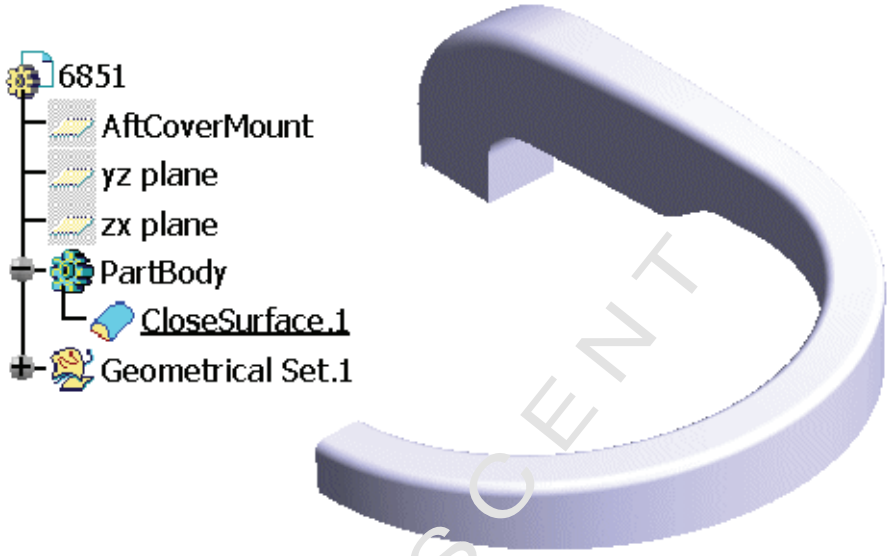

Figure 1–6

1.4 Design Intent

The key to building parametric, feature-based, surface models is to construct them so that their behavior is flexible and predictable. This process is known as capturing design intent.

You should already be familiar with several methods of capturing the design intent with a solid part model. These methods are applicable to the creation of surface features, which include the following:

When designing surfaces. decisions should always be made to drive the design intent of the finished model.

- Dimension sections and features so that your model updates based on your design intent.
- Select **Depth** options for your features to drive your design intent.
- When applicable, use symmetry conditions to drive the design intent.

Surface Modeling Methods

Surface features introduce additional flexibility into the model because of the different ways in which a surface model can be created. Modeling methods include the following:

- Boundary
- Slab
- Hybrid

Boundary

The geometry of a model can be defined by a series of curves, known as a curve network. The curve network is the backbone on which many surfaces are created. It can be used to define the intersections or common boundaries of surfaces that define the shape of the model. A curve network is used to create the model shown in Figure 1–7.

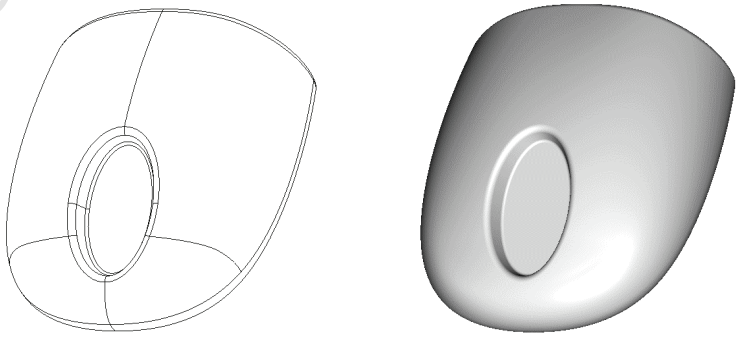

Figure 1–7

The advantages of boundary modeling are:

- Enables precise control over shape and continuity.
- More complex shapes can be created from the curve network.

The disadvantages of boundary modeling are:

- Longer setup time for required reference geometry.
- Can be time consuming to update necessary design changes.

Slab

Models can be defined by creating simple surfaces. Once these surfaces have been created, they can be trimmed, split, or joined to create the desired shape. This method of construction is frequently used when the surface geometry can easily be broken into simple geometric shapes. These surfaces are typically called Slab or Blocked surfaces. Simple slab surfaces are shown in Figure 1–8. A more complex example is shown in Figure 1–9.

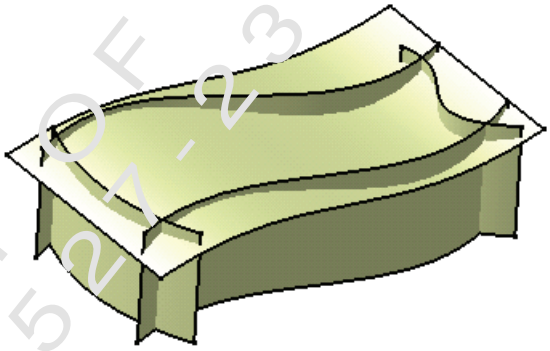

Figure 1–8

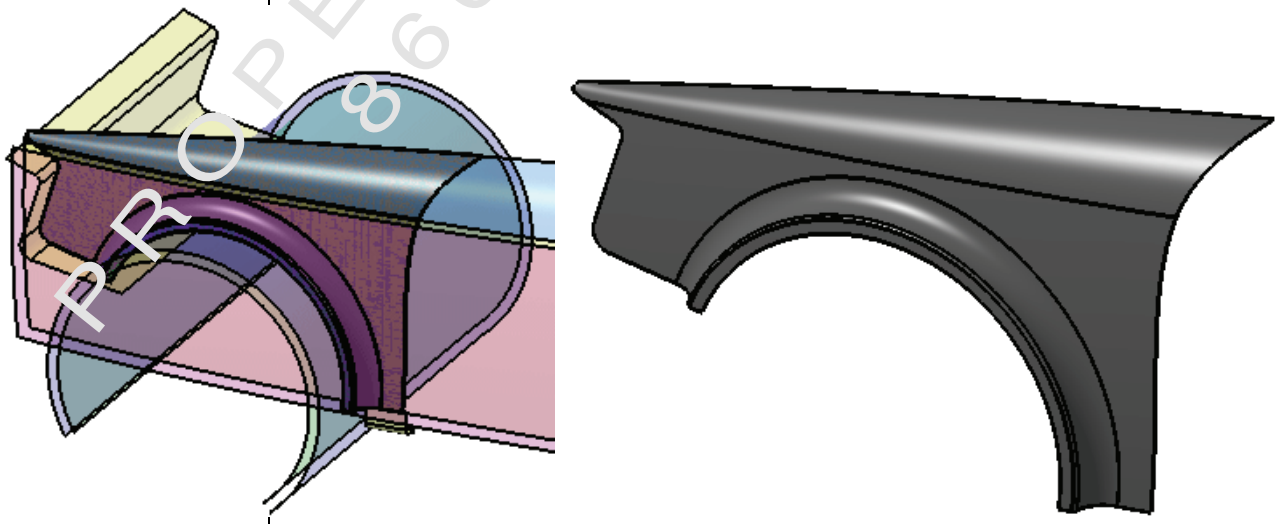

Figure 1–9

The advantages of slab modeling are:

- Simple surfaces take less time to create than a network of curves.
- Slab surfacing is easier to make design changes to downstream, than curve networks.

The disadvantages of slab modeling are:

- The order in which the surface operations is performed can create undesired results.
- Cannot always capture the design intent with simple surfacing techniques.

Hybrid

Models can be defined by combining the boundary and slab surface methods. This is the most common method of creating surface models in CATIA. Slab surfaces are generally used to start the model and curve networks are generated, as needed, to complete the geometry. Three simple, extruded, slab surfaces and a curve network are shown in Figure 1–10. This example shows the two techniques used to generate the required geometry.

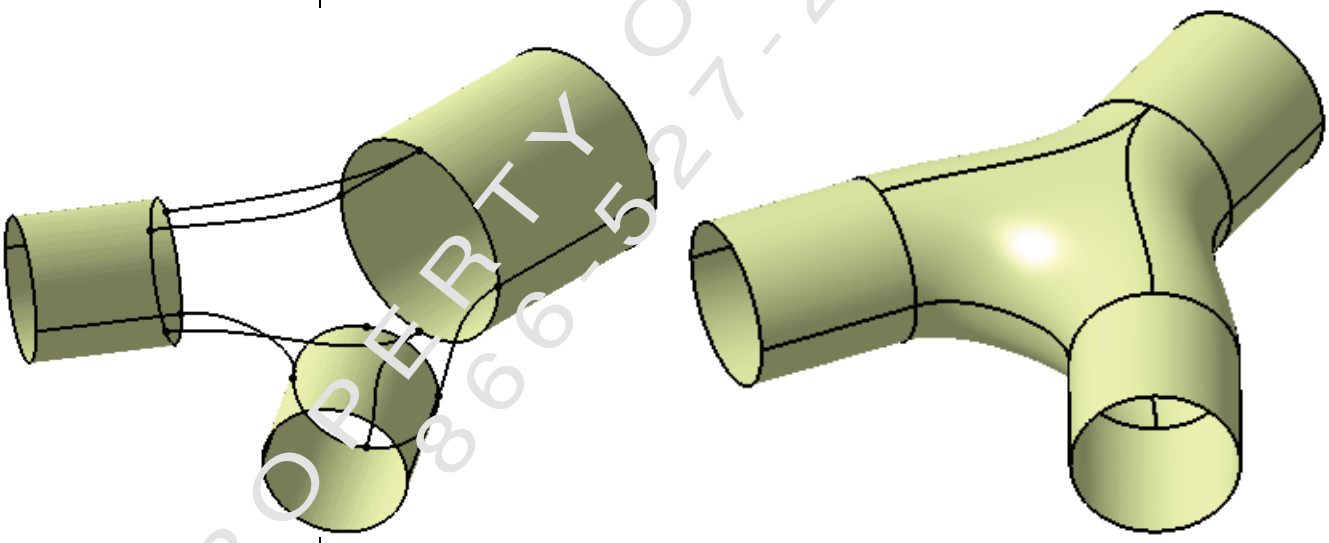

Figure 1–10

Boundary Representations

A boundary representation refers to the selection of any sub-element when defining references for a feature. This means that a feature in the specification tree is not directly referenced. Instead an entity belonging to a feature, such as the edge or vertex of a surface, is selected. Whenever a boundary representation is selected, CATIA reports the reference using **<feature>\<sub-element>**. For example, **Sweep.1\Edge** is shown in Figure 1–11.

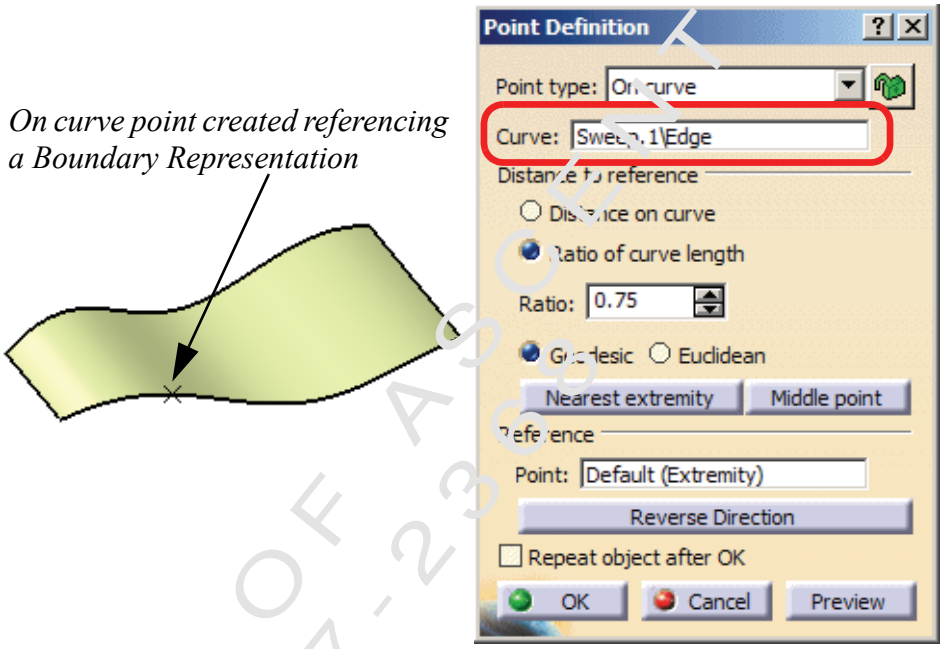

Figure 1–11

Selecting a boundary representation makes the model less stable. Modifications to features can cause the deletion of a sub-element, resulting in errors. The best practice is to reference non-boundary representations, also known as explicit elements. These display directly in the specification tree. Referencing explicit elements promotes a more stable model that you have more control over.

Recommended practices

- Create Extract and Boundary curves instead of directly referencing an edge.
- Use Sketch Profiles and Sketch Outputs when a sketch element needs to be referenced.
- Create points manually instead of selecting a vertex.
- Use the **Replace** tool when design modifications are needed.

- Use contextual menus to easily create proper reference geometry as shown in Figure 1–12.

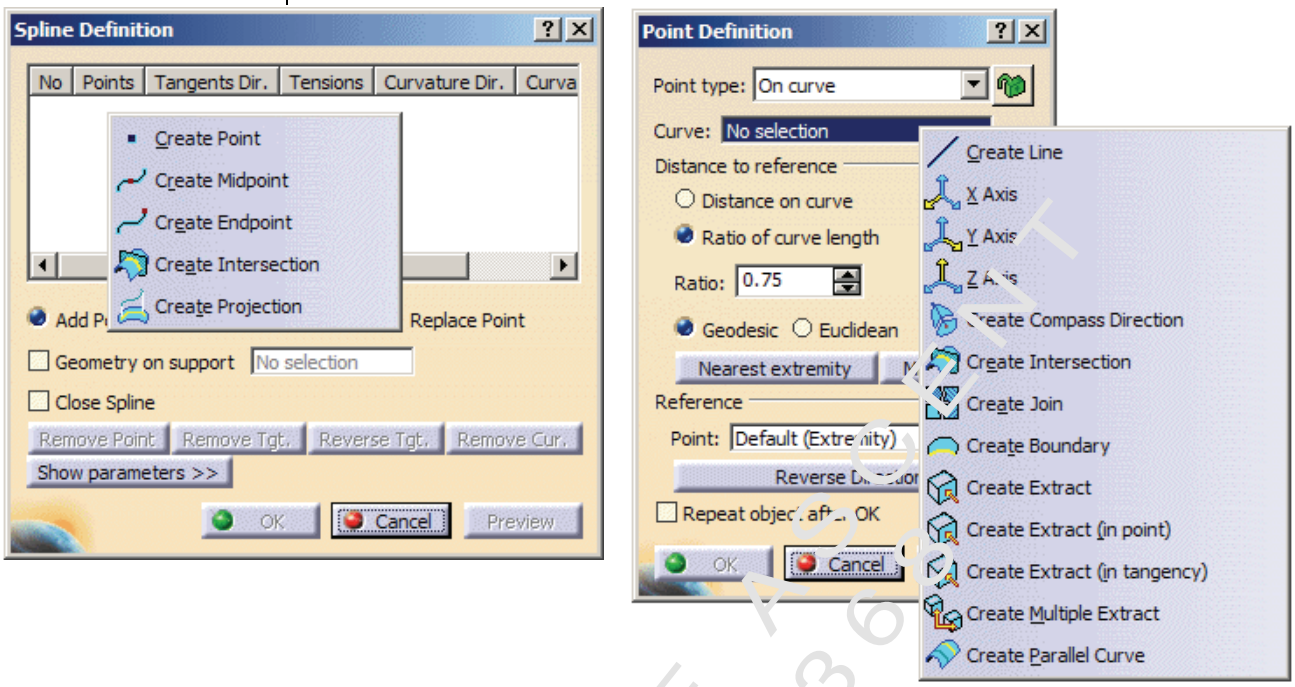

Figure 1–12

1.5 Tree Organization

The specification tree can become very long and messy in large models. You should already be familiar with some techniques to organize the tree, including inserting and changing geometrical sets to help separate and order like elements. Renaming features and geometrical sets also makes updating and investigating a model easier. The **Autosort** command is also helpful in managing the feature order in a geometrical set. Other tools exist to help organize the specification tree.

Groups

Groups enable you to control the display of features within the specification tree and can be used as a filter. Groups are created from an existing geometrical set. Features from that geometrical set are added to the group. When the Group is collapsed, only features added to the group are shown in the specification tree. When the Group is expanded, all features from the geometrical set are displayed as shown in Figure 1–13.

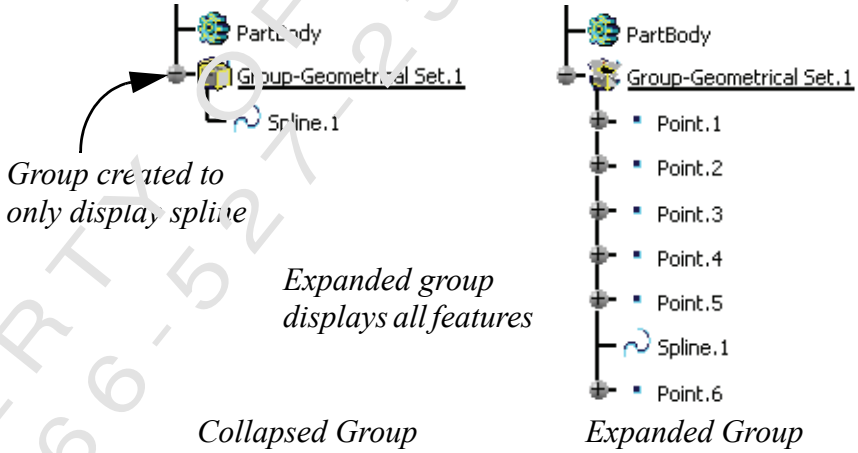

Figure 1–13

Use the following steps to create a group:

1. Right-click on a geometrical set from which to create the group.
2. Select **&/ object > Create Group** in the contextual menu.
3. Enter a name for the group.
4. Select features from the geometrical set to place in the group. Any added feature is shown in the group when collapsed in the tree. All other features of the geometrical set are not shown in the specification tree.

5. Click OK to complete the creation of the group.

To collapse or expand a group, right-click the group in the specification tree and select **&/ object > Expand Group** or **&/ object > Collapse Group.**

Grouping is a very useful option. With Groups, insignificant construction or reference elements can be hidden in the specification tree. This makes the specification tree very concise and easy to review, which improves the effectiveness of the editing process.

If a group was the active geometrical set, any new feature added to the model would not be displayed in the specification tree until the group was expanded. For example, a point is added to the model, but not as an input to Group-Geometrical Set.1. The point is not displayed unless the group node is expanded.

There is an option that enables you to automatically add new features as input to the active group. To activate this option, select **Tools > Options > Shape > Generative Shape Design**, select the General tab, and enable the **Integration of created feature as group inputs** option, as shown in Figure 1–14.

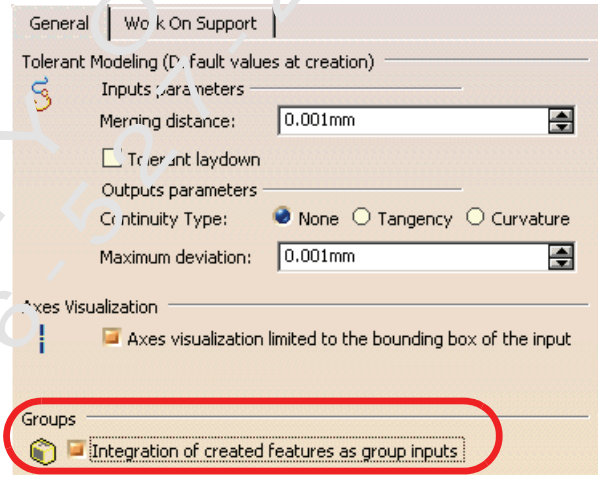

Figure 1–14

All new features are now added to the active group. Features can be removed from the group by right-clicking on the group node and selecting **Edit Group**. This only applies to features created in the Generative Shape Design workbench.

AutoSort

AutoSort is a command that organizes the features in a geometrical set. When **AutoSort** is performed on a geometrical set, the features are reordered by the order in which CATIA updates the features.

Use the following steps to use the **AutoSort** command:

1. Select a geometrical set, as shown in Figure 1–15.

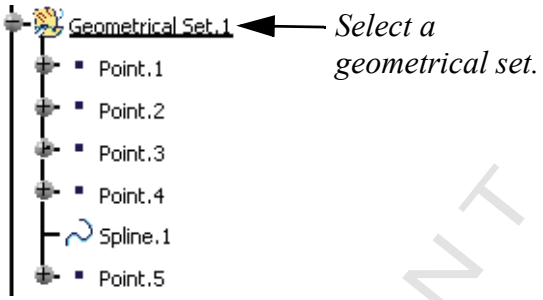

Figure 1–15

2. Right-click and select **&/ object > Autosort** as shown in Figure 1–16.

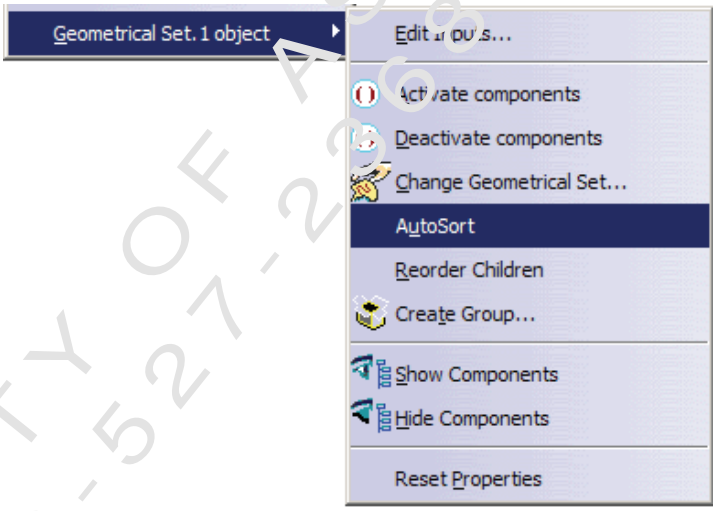

Figure 1–16

3. The result displays as shown in Figure 1–17. Spline.1 was moved below Point.5. This is because Point.5 was used to create Spline.1.

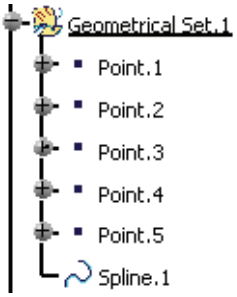

Figure 1–17

Mask

The ability to create a mask was added to the R16 version of the Generative Shape Design workbench. A mask is used to simplify the display of a part model.

 (Mask) has been added to the Tools toolbar, as shown in Figure 1–18.

Figure 1–18

Alternatively, select **Tools > Mask** in the menu bar, as shown in Figure 1–19.

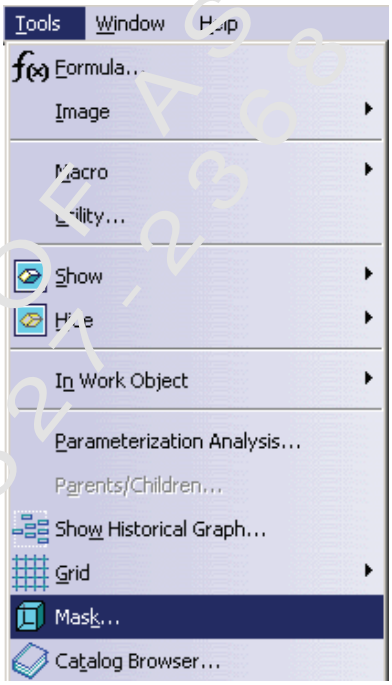

Figure 1–19

CATIA displays a bounding box around the part when a mask is created, as shown in Figure 1–20. The box has six sides, each of which has a green dot. The dot represents a handle that can be moved by selecting and dragging it with the cursor.

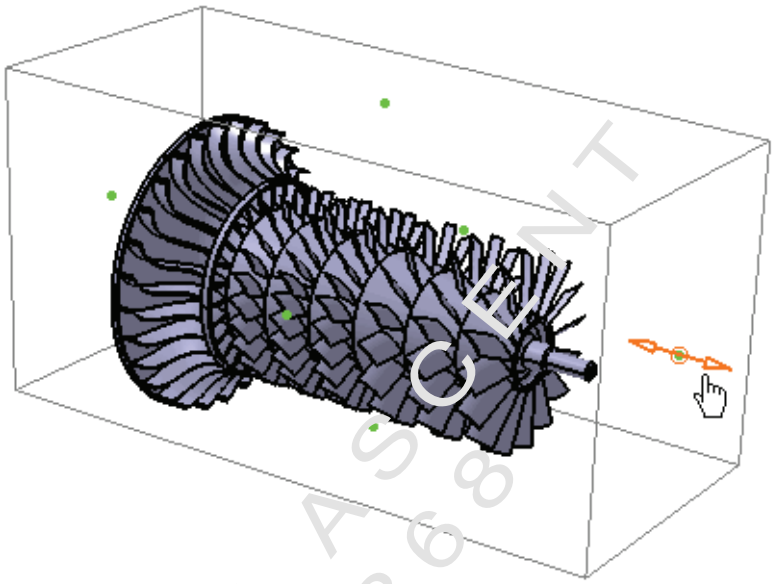

Figure 1–20

The part updates to exclude any surface or reference geometry that is completely outside of the bounding box, as shown in Figure 1–21.

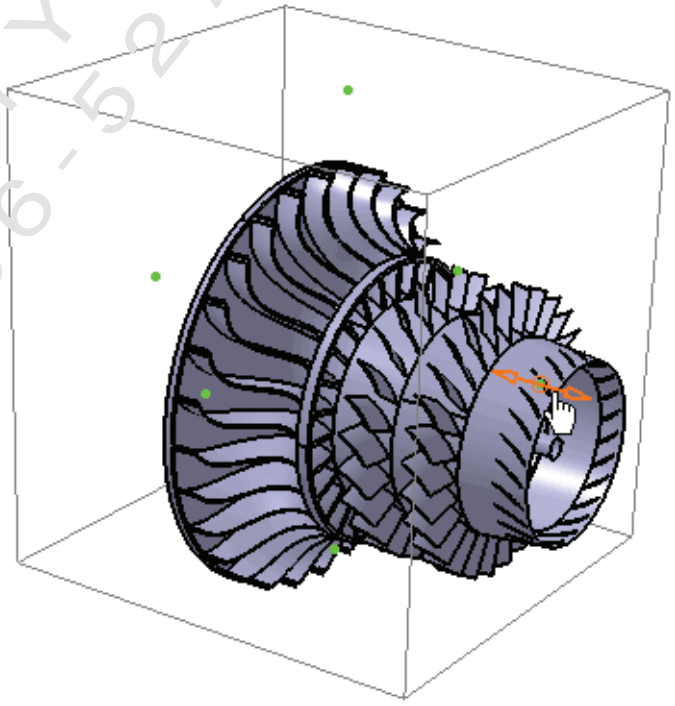

Figure 1–21

You can place the bounding box using the **Coordinates** or **Position & Dimensions** option in the Mask Type pull-down menu, as shown in Figure 1–22. With the **Coordinates** option, you place the box relative to selected points or vertices from the model.

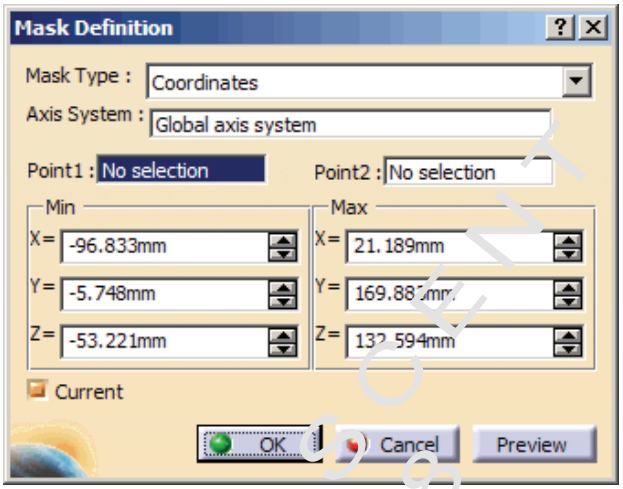

Figure 1–22

You can create multiple masks for a single part. CATIA lists all masks under the Masks node of the specification tree, as shown in Figure 1–23.

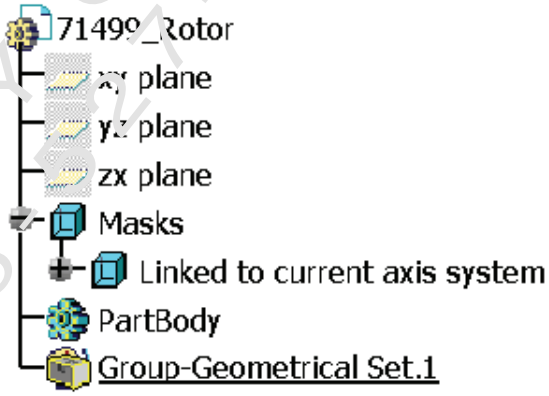

Figure 1–23

Only one mask can be active at a time. The symbol of an active mask displays in red. The active mask is circled in Figure 1–24.

The symbol of an active mask displays in red and is visible within CATIA.

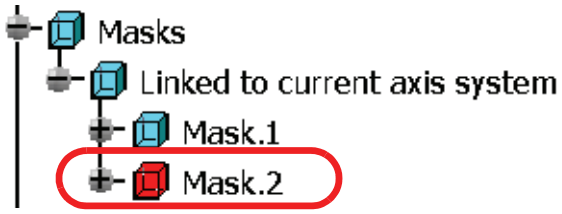

Figure 1–24

Exercise 1a

Hybrid Surface Modeling

In this exercise, you will investigate tangency conditions of curves and surfaces. The Boundary method of surfacing will be used. The slab method of modeling produced the surfaces provided for you to start with. You will work through a typical surface-based modeling process, from wireframe to the creation of solid geometry from a surface.

The completed model is shown in Figure 1–25.

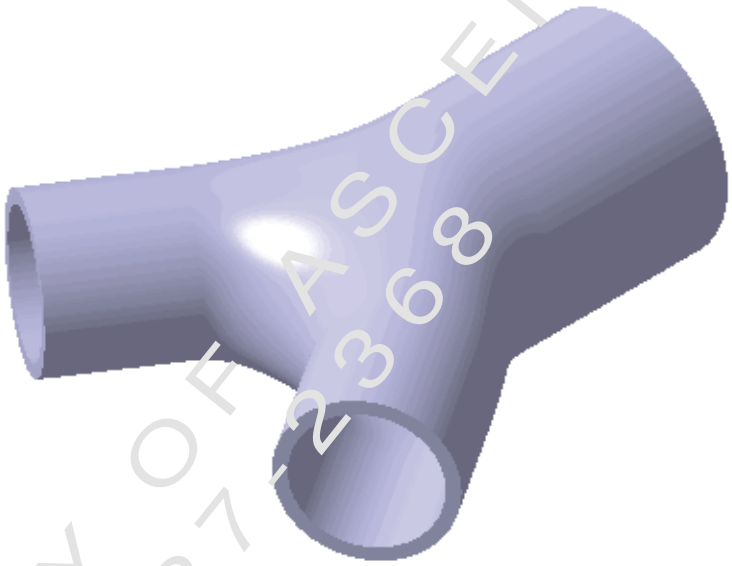

Figure 1–25

Goal

After you complete this exercise, you will be able to:

- ✓ **Investigate curve tangency conditions**
- ✓ **Create a curve with tangency**
- ✓ **Create a surface from curves**
- ✓ **Define tangency conditions for a surface**
- ✓ **Organize surface data**
- ✓ **Create a solid from surfaces**

Task 1 - Open a part file.

1. Open **Y_Pipe.CATPart**. The model displays as shown in Figure 1–26.

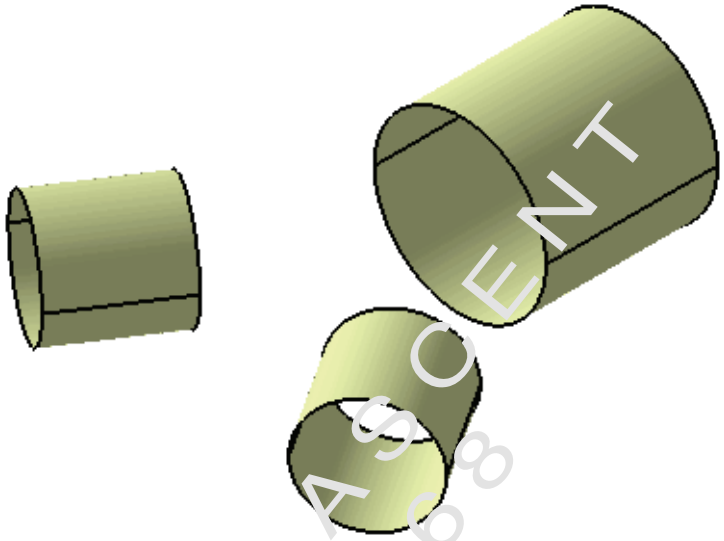

Figure 1–26

2. In the specification tree, note that the geometry has been organized into geometrical sets. Also note that **NewSurfaces** is the work object as shown in Figure 1–27.

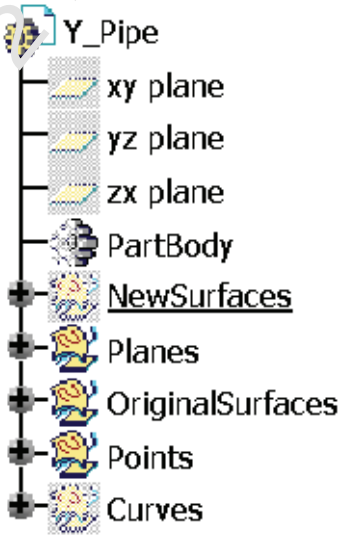

Figure 1–27

Task 2 - Change the display of bodies.

1. Hide the **OriginalSurfaces** geometrical set.
2. Show the **Curves** geometrical set. The specification tree will display as shown in Figure 1–28.

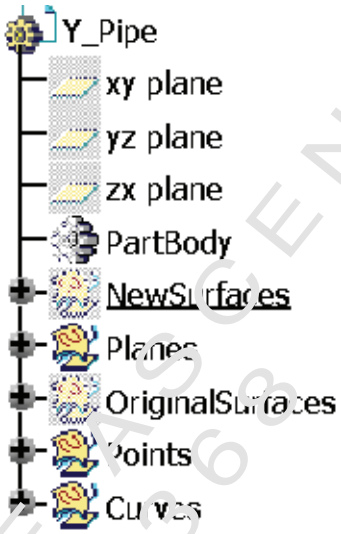

Figure 1–28

The model displays as shown in Figure 1–29.

Design Considerations

The visible geometry can be referred to as a curve network. Creating a curve network provides a wireframe structure that permits the creation of surface features.

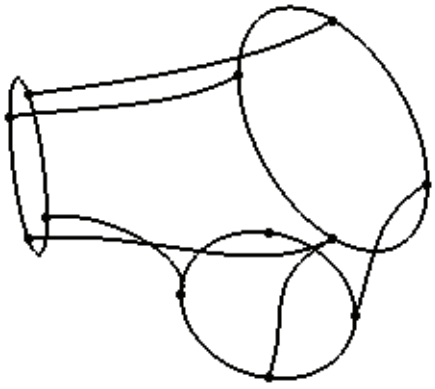

Figure 1–29

3. Show **OriginalSurfaces** as shown in Figure 1–30.

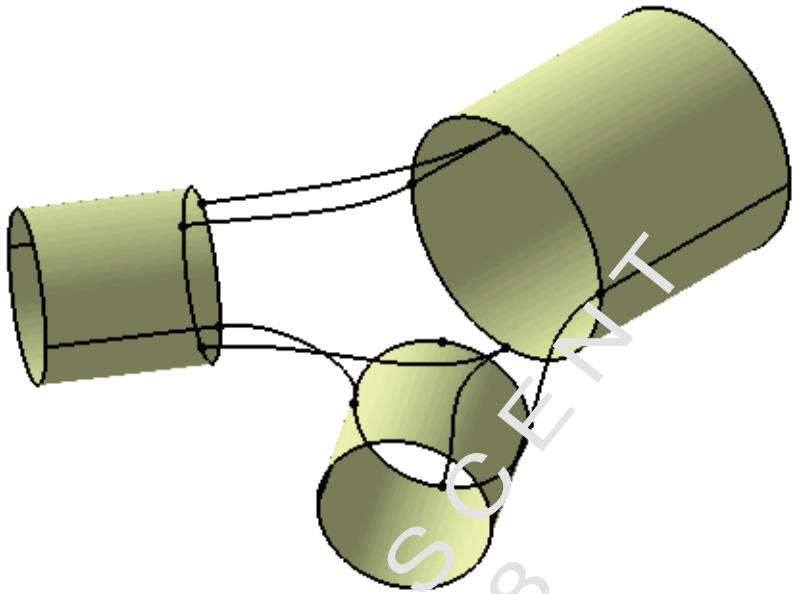

Figure 1–30

4. Show **NewSurfaces** as shown in Figure 1–31.

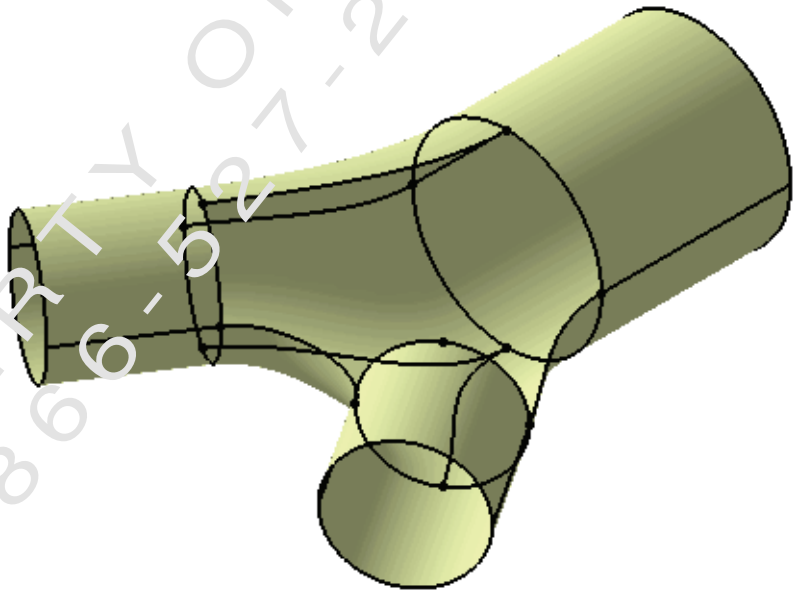

Figure 1–31

Design Considerations

Currently, the **NewSurfaces** geometrical set contains four fill surfaces. Later in the exercise, you will develop additional surfaces to create a complete skin for the Y-Pipe.

5. Hide **NewSurfaces**.

6. Hide **OriginalSurfaces** as shown in Figure 1–32.

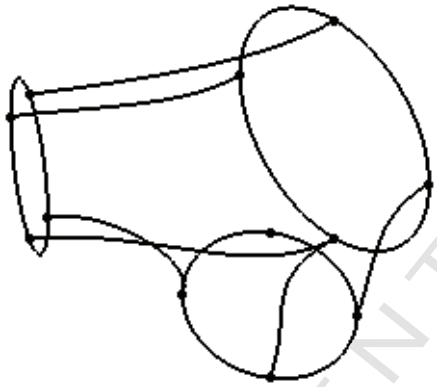

Figure 1–32

7. Hide the appropriate curves so that only the seven shown in Figure 1–33 are visible.

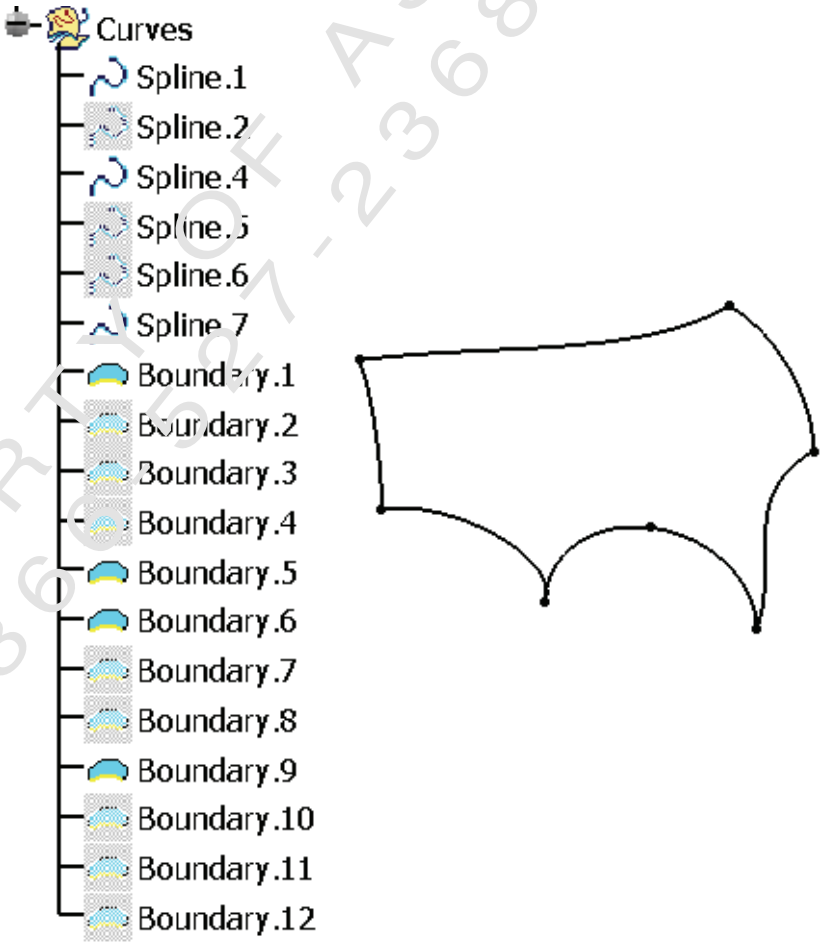

Figure 1–33

Task 3 - Investigate tangency options for a curve.

1. Select the **yz plane** in the specification tree and show it.
2. Edit the curve shown in Figure 1–34.

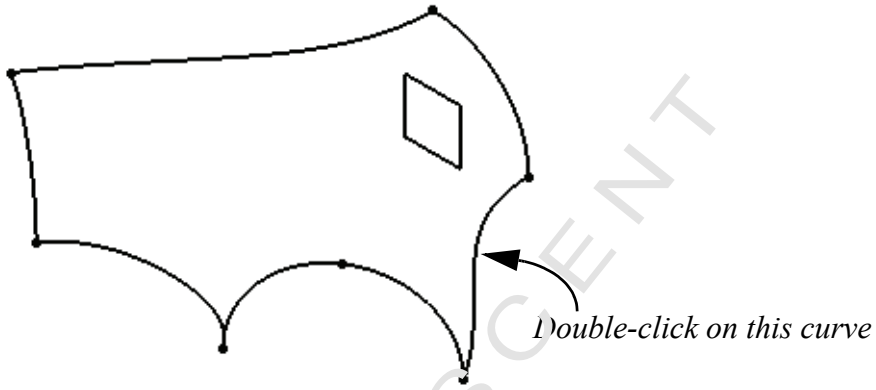

Figure 1–34

3. The curve is a spline curve with a tangency condition applied to both ends. This is shown in the *Tangents Dir* column, as shown in Figure 1–35.

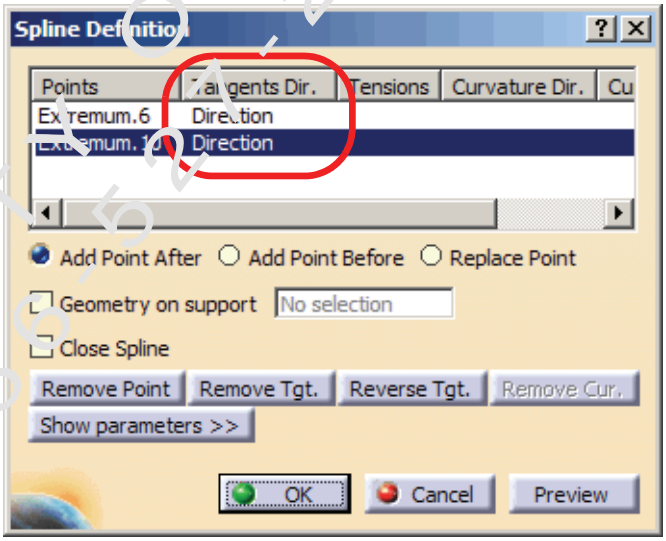

Figure 1–35

4. Click Show parameters >> to view the tangency references.

Design Considerations

5. Select **Extremum.6** in the *Points* column in the Spline definition dialog box, as shown in Figure 1–36.

For this end of the curve, a plane is used to defined tangency. If a plane is used as a Tangents Dir reference, the curve is normal to the selected plane at that end.

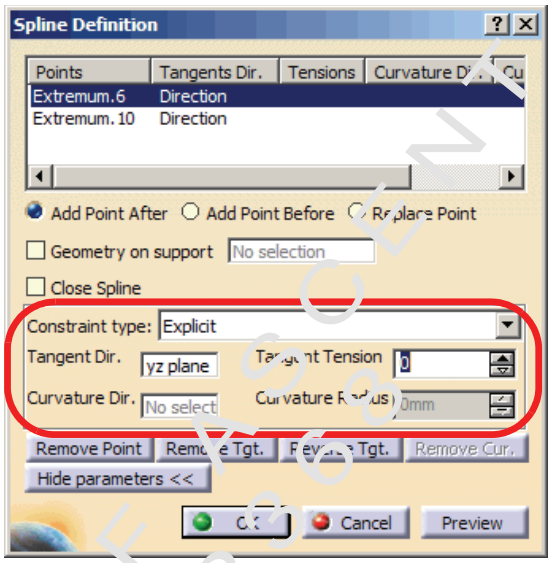

Figure 1–36

6. Select **Extremum.10** in the *Points* column.

7. Right-click on **Plane.3** in the *Tangents Dir* field and select **Hide/Show**, as shown in Figure 1–37.

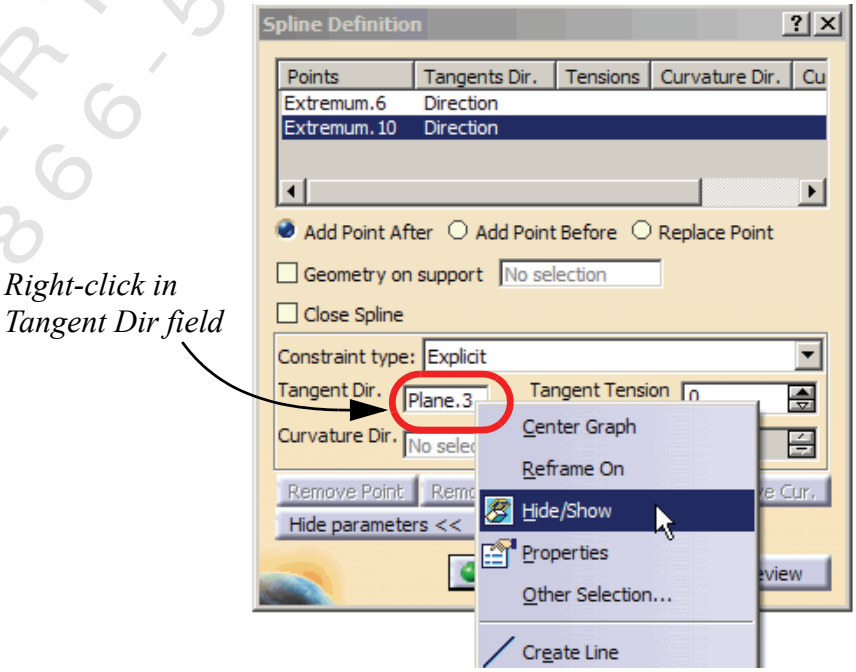

Figure 1–37

8. Click OK.

9. Ensure that the two reference planes are visible in the model, as shown in Figure 1–38.

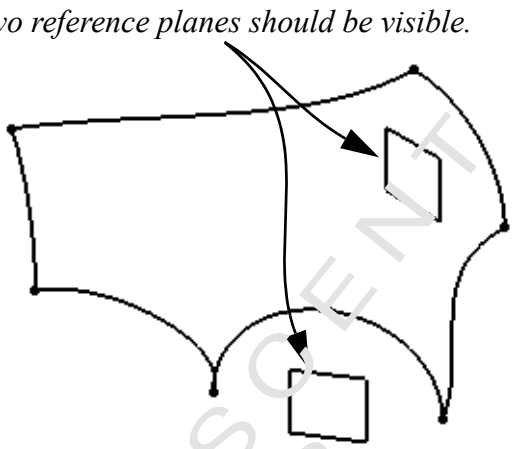

Figure 1–38

Task 4 - Create a spline curve with tangency.

1. Show the following from the **Points** geometrical set, as shown in Figure 1–39:

   - Extremum.1
   - Extremum.8

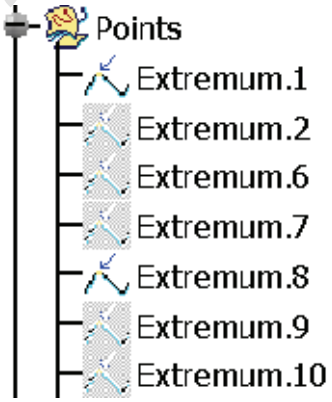

Figure 1–39

2. Show the **NewSurfaces** geometrical set.

3. Ensure that **Curves** is the active work object. The specification tree and model display as shown in Figure 1–40.

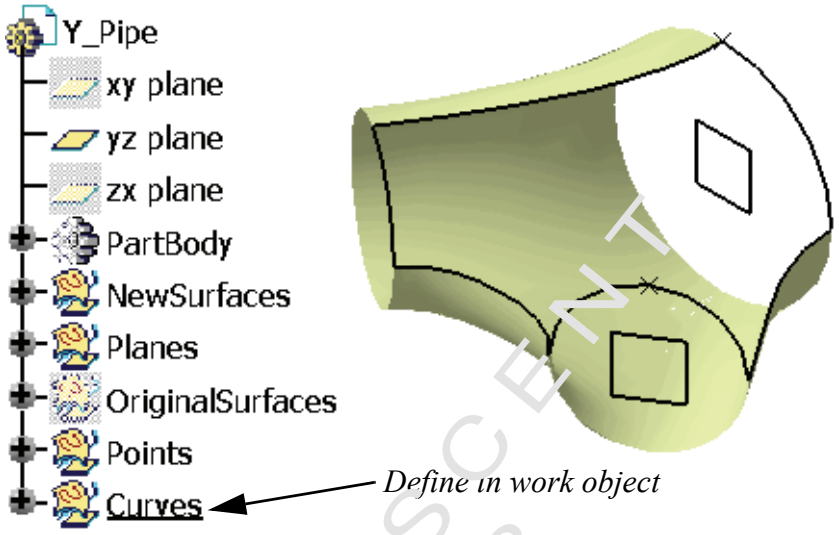

Figure 1–40

4. Click (Spline) and select the two Extremum points that have been shown.

5. Select the appropriate reference planes to define tangency for each of the two points, as shown in Figure 1–41.

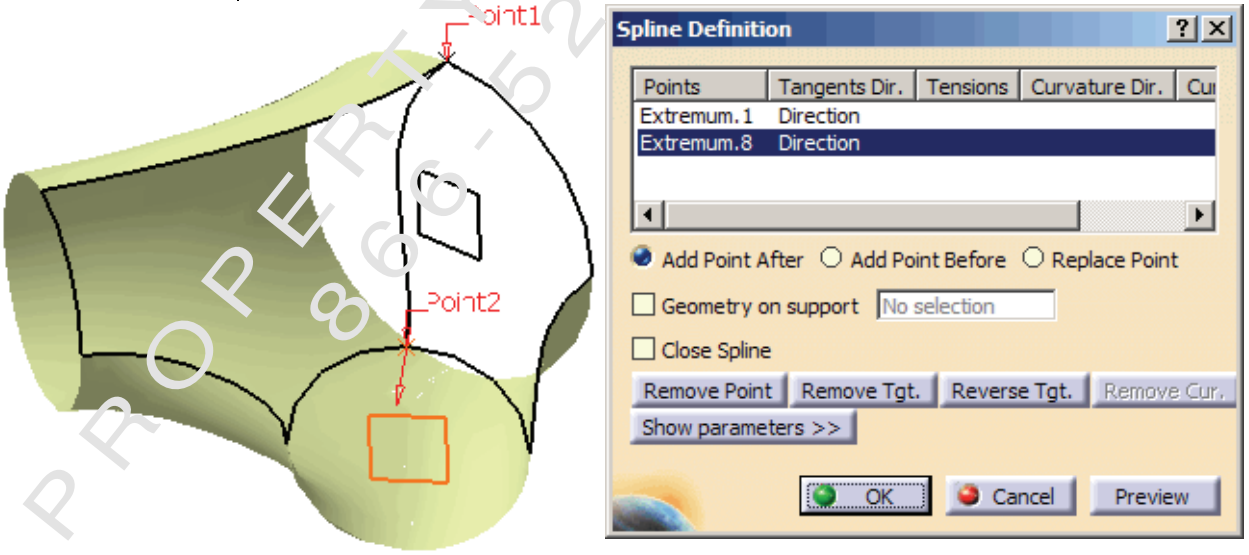

Figure 1–41

6. Click OK.

Task 5 - Create a surface from curves.

1. Define **NewSurfaces** to be the work object, as shown in Figure 1–42.

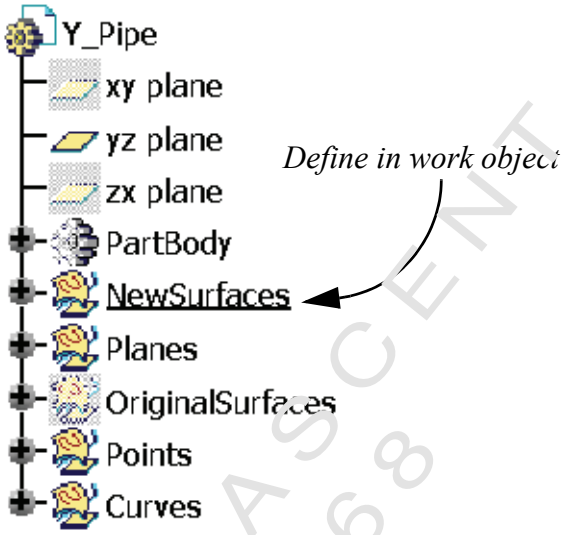

Figure 1–42

2. Click (Fill) and select the four boundary curves shown in Figure 1–43.

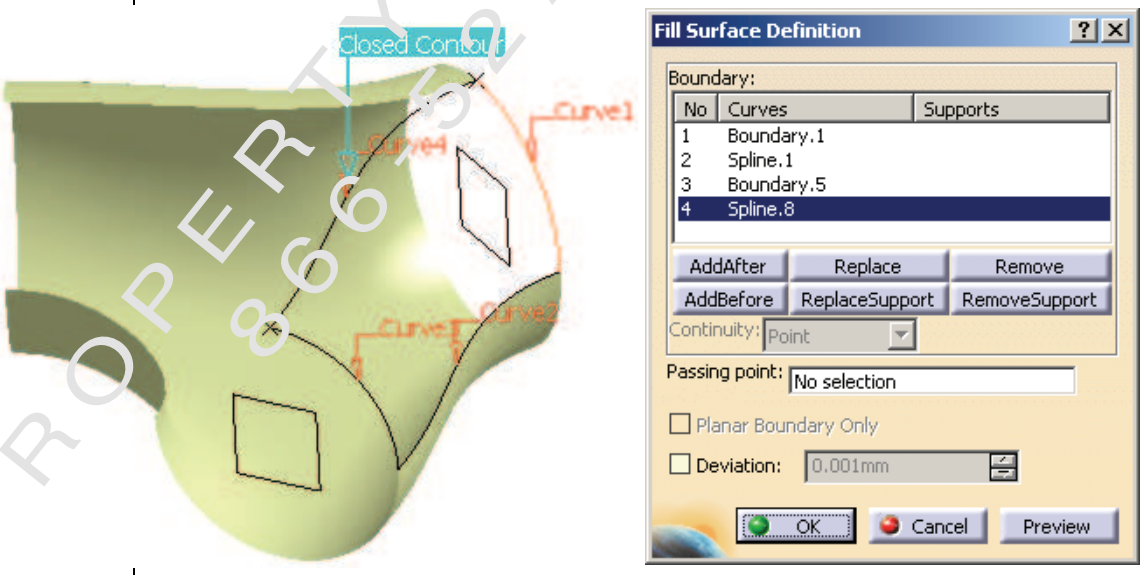

Figure 1–43

3. Show **OriginalSurfaces**.

4. Define the support for the three fill boundaries, as shown in Figure 1–44.

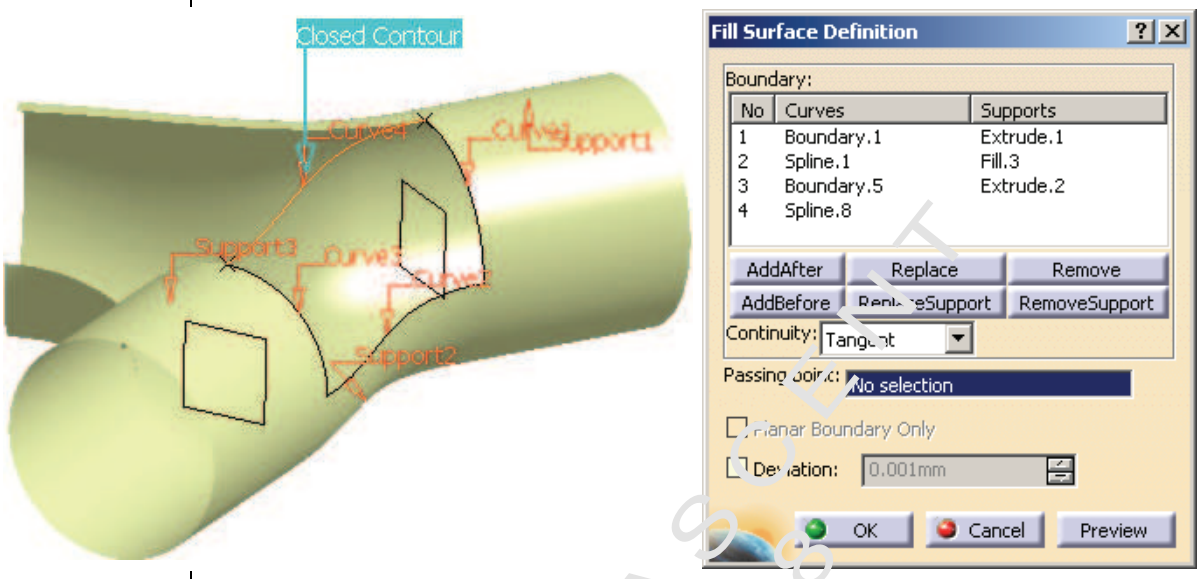

Figure 1–44

5. Complete the fill surface. The model displays as shown in Figure 1–45.

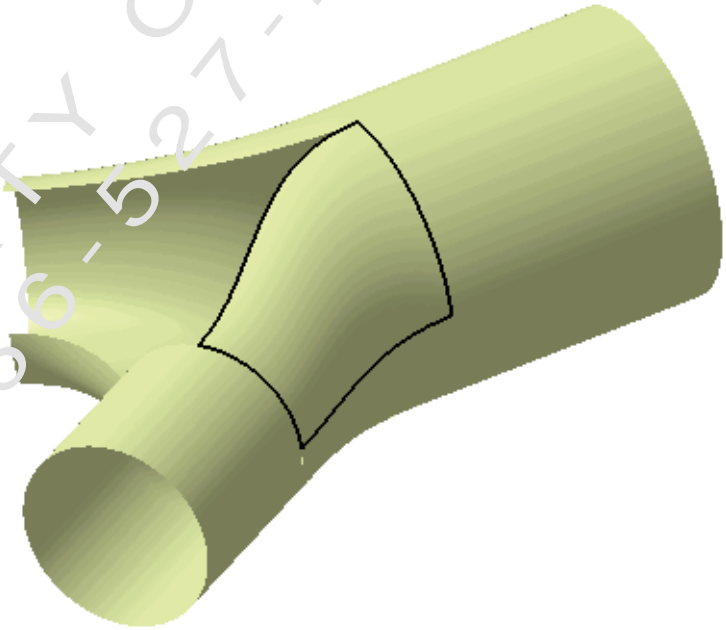

Figure 1–45

6. Create another fill surface using the five boundaries shown in Figure 1–46. Ensure that the adjacent surfaces are selected as support references. Define **Tangent** as the continuity type.

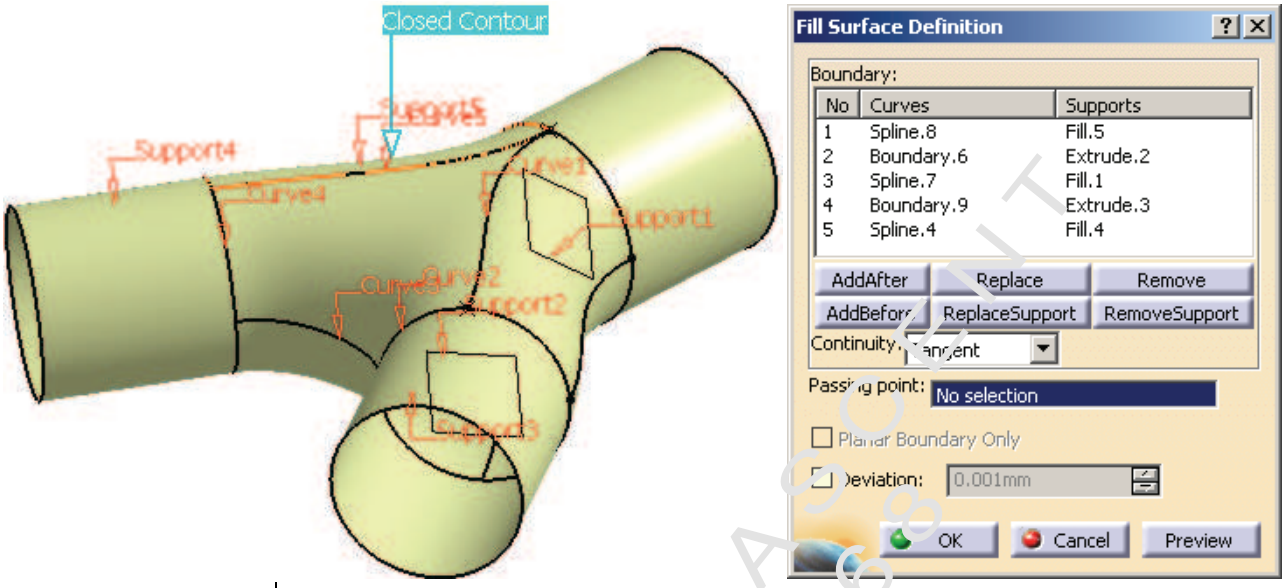

Figure 1–46

7. Complete the fill surface.

Task 6 - Create a single surface feature.

Design Considerations

Since your goal is to create solid geometry from a surface model, all surfaces (extrude and fill) need to behave as a single element for a thick surface feature creation to be successful. To help keep the geometry organized, you will insert a new geometrical set to hold the Join feature that will be created in this task.

1. Select **Insert > Geometrical Set** and enter the following as shown in Figure 1–47:
   - Name: **JoinedSurfaces**

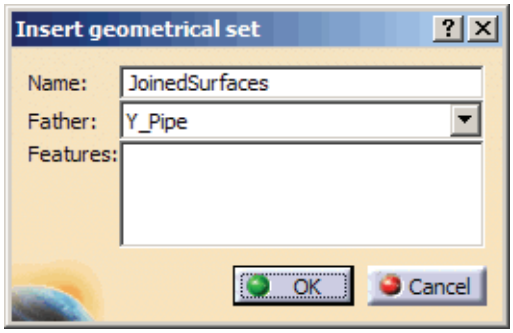

Figure 1–47

Design Considerations

2. Click OK.

The new geometrical set is positioned directly beneath the previously activated geometrical set.

3. Click (Join).
4. Select any of the surfaces of the model.
5. Right-click in the *Elements to Join* field and select **Distance Propagation**, as shown in Figure 1–48.

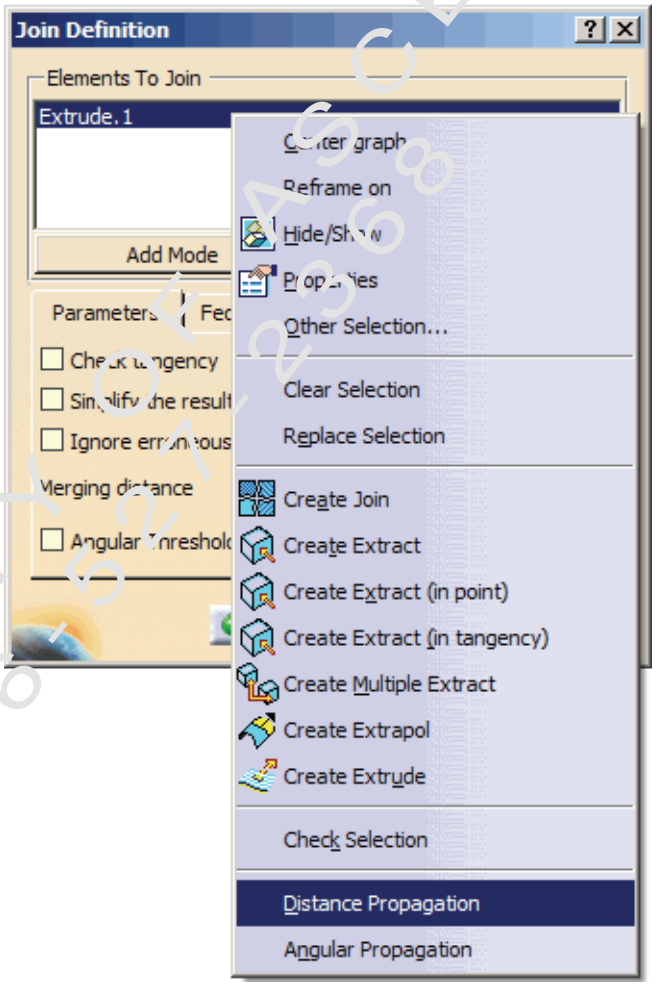

Figure 1–48

Design Considerations

This propagation tool enables the system to automatically select all surfaces that have boundaries that are within the Merging distance parameter value.

All nine surfaces should be listed in the Join dialog box, as shown in Figure 1–49. (Listing order might be different in your model.)

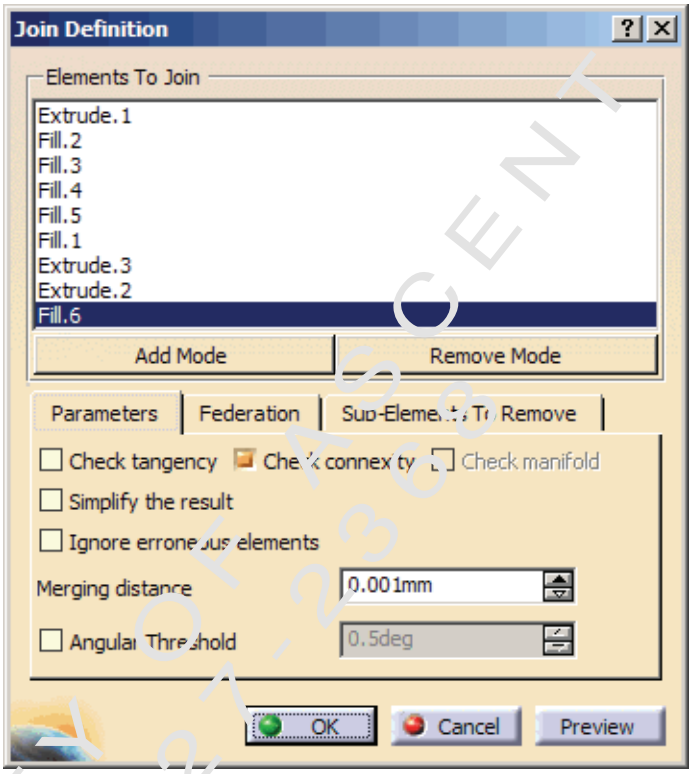

Figure 1–49

6. Click OK. The join is added to the JoinedSurfaces body as shown in Figure 1–50.

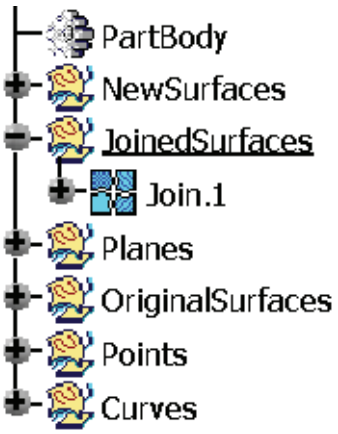

Figure 1–50

7. Hide all geometrical sets except for **JoinedSurfaces** as shown in Figure 1–51.

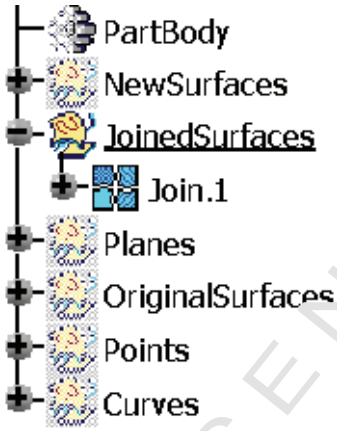

Figure 1–51

Task 7 - Create solid geometry.

1. Activate the Part Design workbench.
2. Define the PartBody to be the work object.
3. Use  (Thick Surface) to create a 2mm thick solid from Join.1 in the direction shown in Figure 1–52.

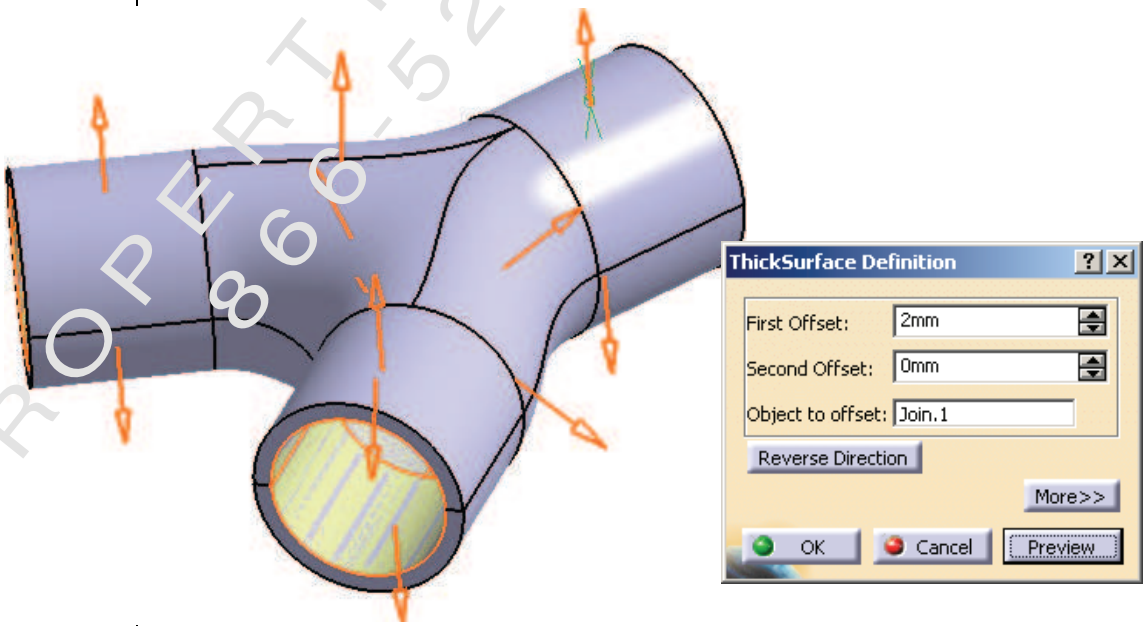

Figure 1–52

4. Select **Tools > Hide > All Geometrical Sets** to hide all wireframe and surface geometry that is still shown. The completed model displays as shown in Figure 1–53.

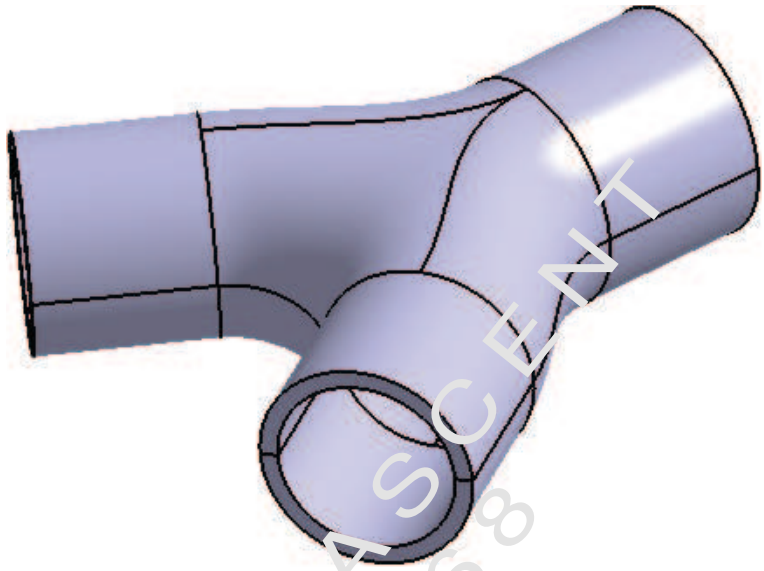

Figure 1–53

5. Save the file and close the window.

Exercise 1b

Front Quarter Panel

In this exercise, you will create the front quarter panel of an automobile as shown in Figure 1–54.

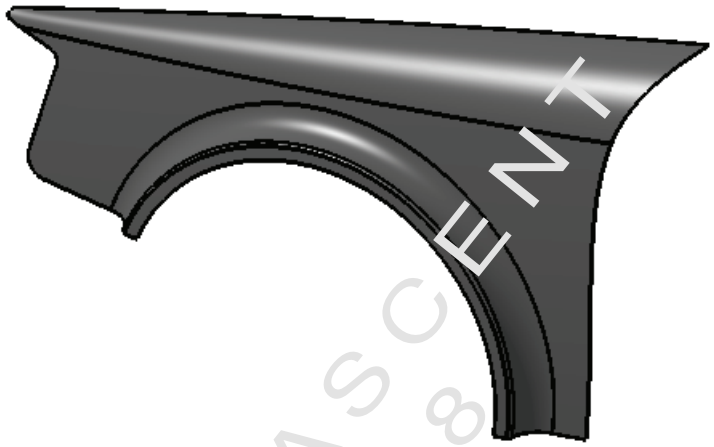

Figure 1–54

Goal

After you complete this exercise, you will be able to:

- ✓ **Use Slab surface modeling techniques**
- ✓ **Perform surface operations**
- ✓ **Create surface fillets**
- ✓ **Project curves**
- ✓ **Organize wireframe and surface data**

Task 1 - Open a part file.

1. Open **Front_Panel_Start.CATPart**. The model displays as shown in Figure 1–55.

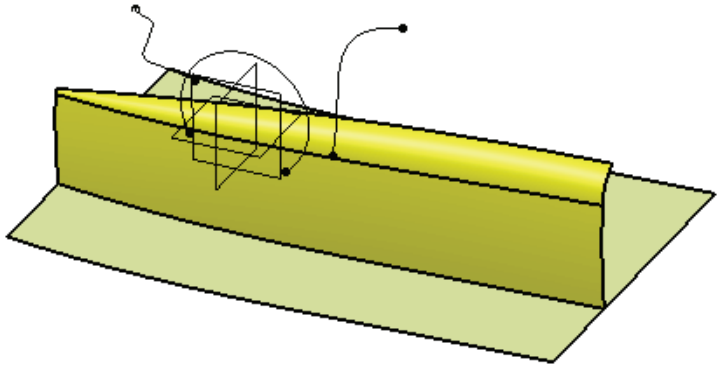

Figure 1–55

2. Ensure that the model units are set to [mm].
3. Hide the following elements:
   - Base
   - 1/4 Panel - Door Profile
   - Front Project Sketch

Task 2 - Create wheel surfaces.

 *(Extrude) is located in the Surface toolbar.*

1. Create an extruded surface using the following specifications:
   - Profile: **Wheel Well Sketch**
   - Direction: **ZX plane**
   - Limit 1: **1010mm**
   - Ensure that the direction of the extrude is created towards the existing surfaces.
2. Rename the completed extrude as [Wheel Extrude]. The model displays as shown in Figure 1–56.

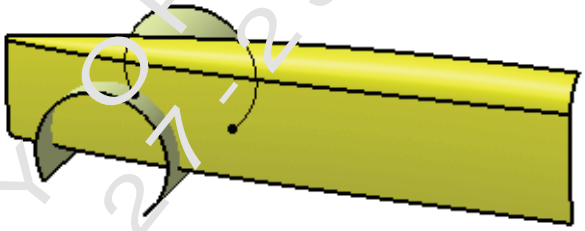

Figure 1–56

3. Hide **Wheel Well Sketch**.

 *(Offset) displays in the Surfaces toolbar.*

4. Create an offset surface using the following specifications:
   - Surface: **Swept Surface.1**
   - Offset: **45mm**

   The Offset direction displays as shown in Figure 1–57.

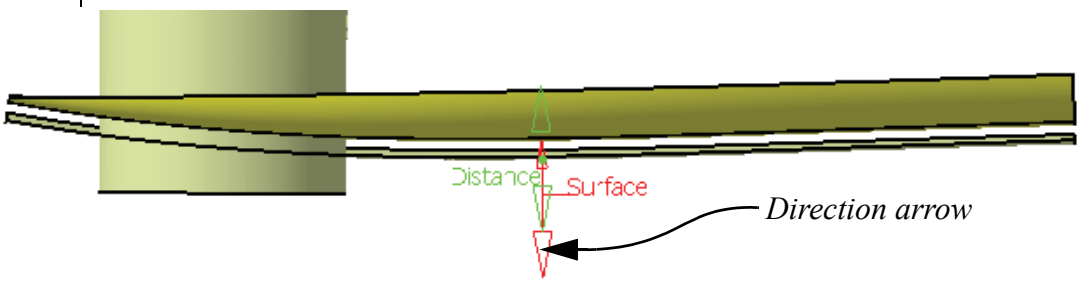

Figure 1–57

5. Rename the completed offset surface as [Body Offset]. The model displays as shown in Figure 1–58.

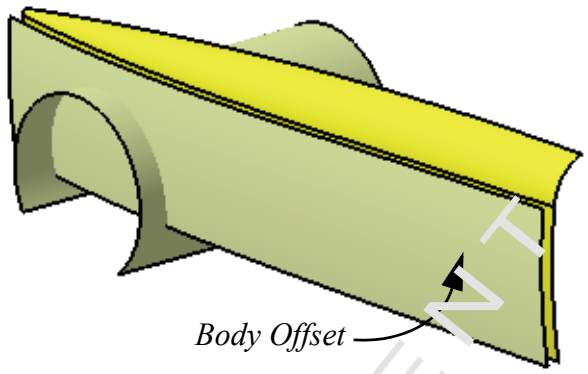

Figure 1–58

6. Create another offset surface using the following specifications:

   - Surface: **Wheel Extrude**
   - Offset: **25mm**

   The offset direction needs to point outside of the Wheel Extrude surface.

7. Rename the completed offset surface as [Wheel Offset]. The model displays as shown in Figure 1–59.

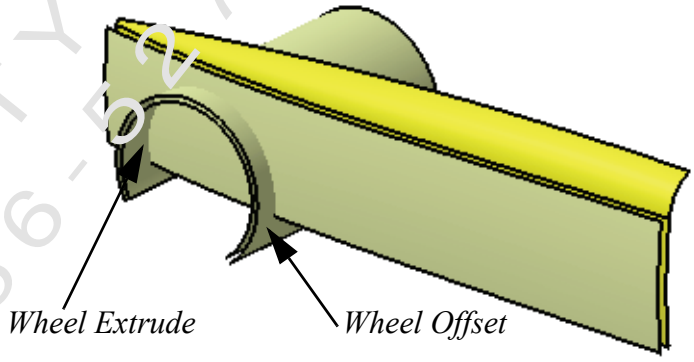

Figure 1–59

Task 3 - Split surfaces.

1. Hide **Body Offset**.

Click Other Side *to toggle which portion of the surface is kept in the* ***Split*** *operation.*

2. Click  (Split). Select **Wheel Offset** as the element to cut. The cutting elements are **Body Offset**, **Swept Surface.1**, and **Base**. The completed split displays as shown in Figure 1–60.

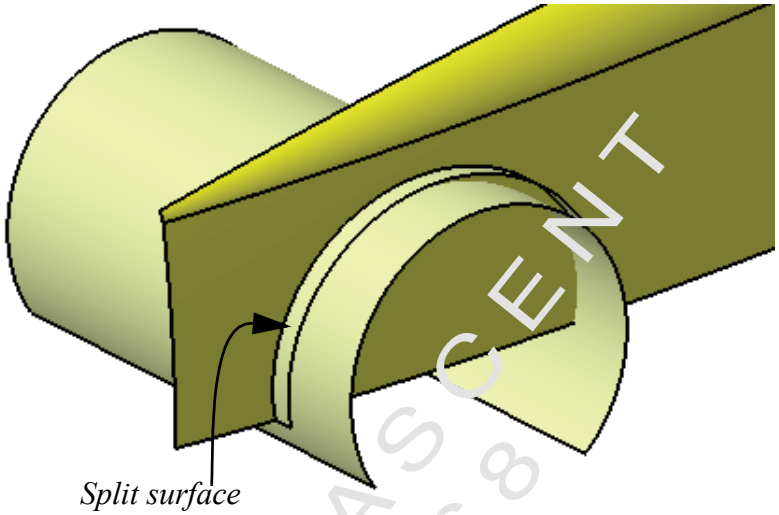

Figure 1–60

3. Rename the split as [Wheel Outside Split].
4. Split **Wheel Extrude**. The cutting elements will be **Body Offset**, **Swept Surface.1**, and **Base**. The completed split displays as shown in Figure 1–61.
5. Rename the split as [Wheel Inside Split].

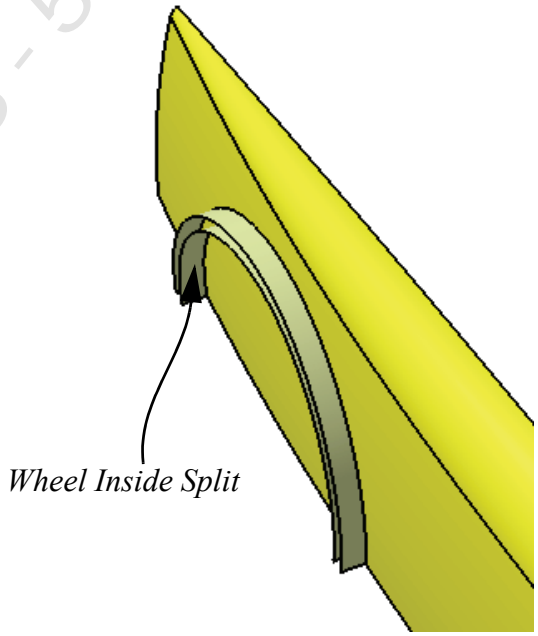

Figure 1–61

6. Show **Body Offset** and **Base**.
7. Split **Body Offset**. The cutting elements will be **Wheel Outside Split**, **Wheel Inside Split**, and **Base**. The completed split displays as shown in Figure 1–62.
8. Rename the split as [Wheel Front Split].

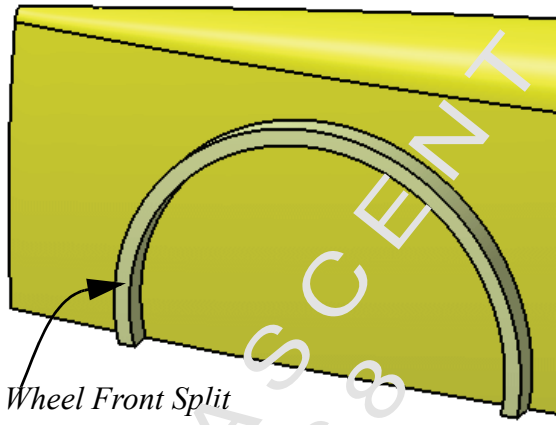

Figure 1–62

9. If not already done, hide **Body Offset**, **Base**, and **Wheel Inside Split**.
10. Split **Swept Surface.1**. The cutting element is **Wheel Outside Split**. The completed split displays as shown in Figure 1–63.
11. Rename the split as [Body Split].

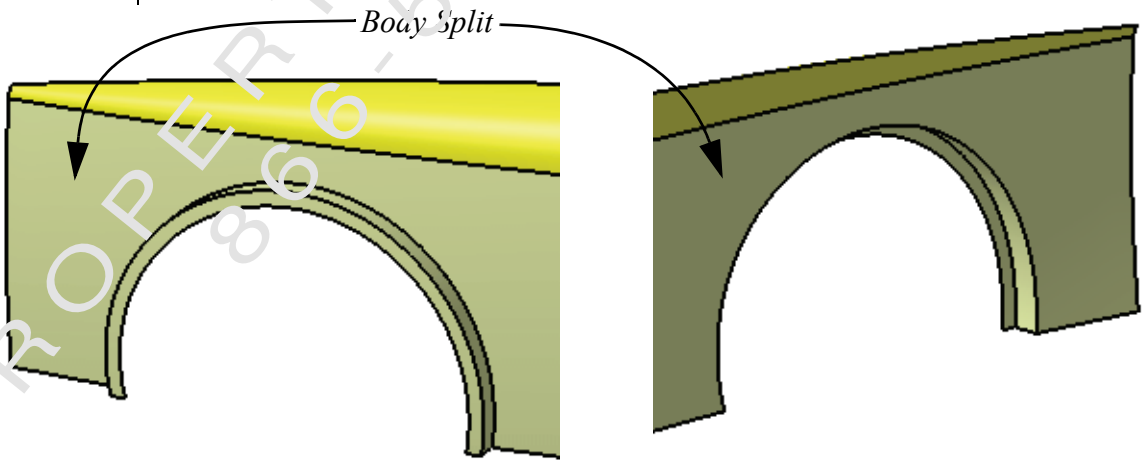

Figure 1–63

 *(Join) displays in the Operations toolbar.*

 *(Projection) displays in the Wireframe toolbar.*

Task 4 - Use Project curves to split.

1. Join **Swept Surface.2**, **Wheel Outside Split**, **Wheel Front Split**, and **Body Split**.
2. Rename the Join as [Surface Join].
3. Show **1/4 Panel - Door Profile** and **Front Project Sketch**.
4. Create a projected curve using the following specifications:
   - Projection type: Along a direction
   - Projected: 1/4 Panel - Door Profile
   - Support: Surface Join
   - Direction: Y-Component or the ZX plane
5. Rename the completed project as [Projected Door Curve]. The model displays as shown in Figure 1–64.

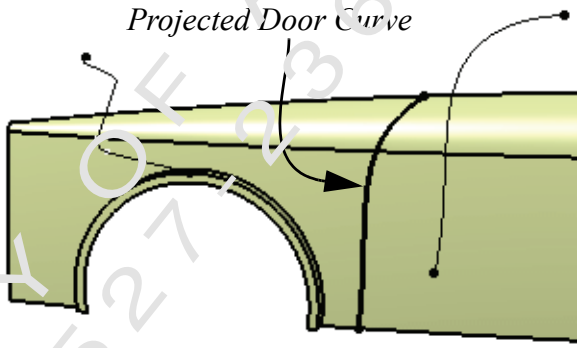

Figure 1–64

6. Repeat Step 4 using **Front Project Sketch**. Rename the projected curve as [Projected Front Curve]. The model displays as shown in Figure 1-65.

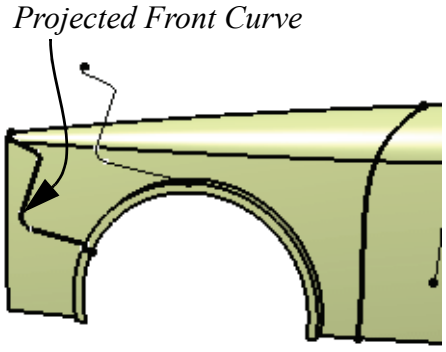

Figure 1–65

7. Hide **1/4 Panel - Door Profile** and **Front Project Sketch**.

8. Split **Surface Join** using **Projected Door Curve** and **Projected Front Curve** as the cutting elements. The completed split displays as shown in Figure 1–66.

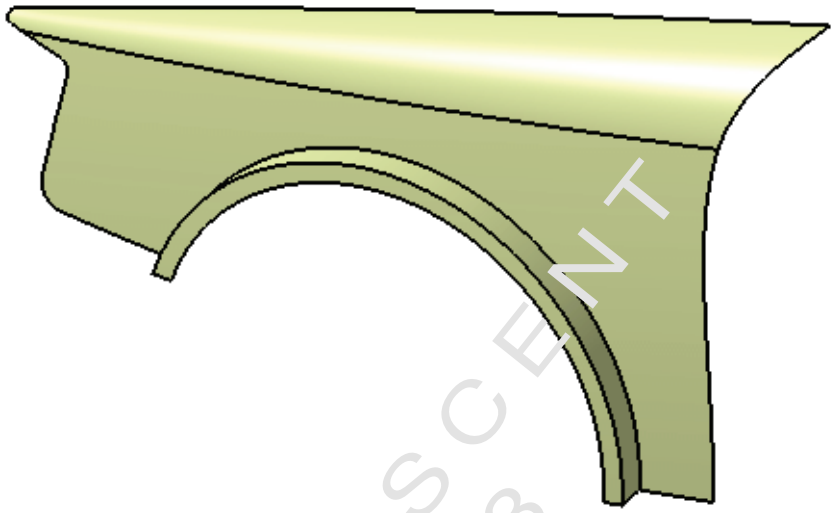

Figure 1–66

*If a color does not display in the list, select **More Colors**.*

9. Using the graphic properties, color the surfaces gray.
10. Save the model.
11. Using the **Edge Fillet** tool place fillets on the edges of the wheel well as shown in Figure 1–67.

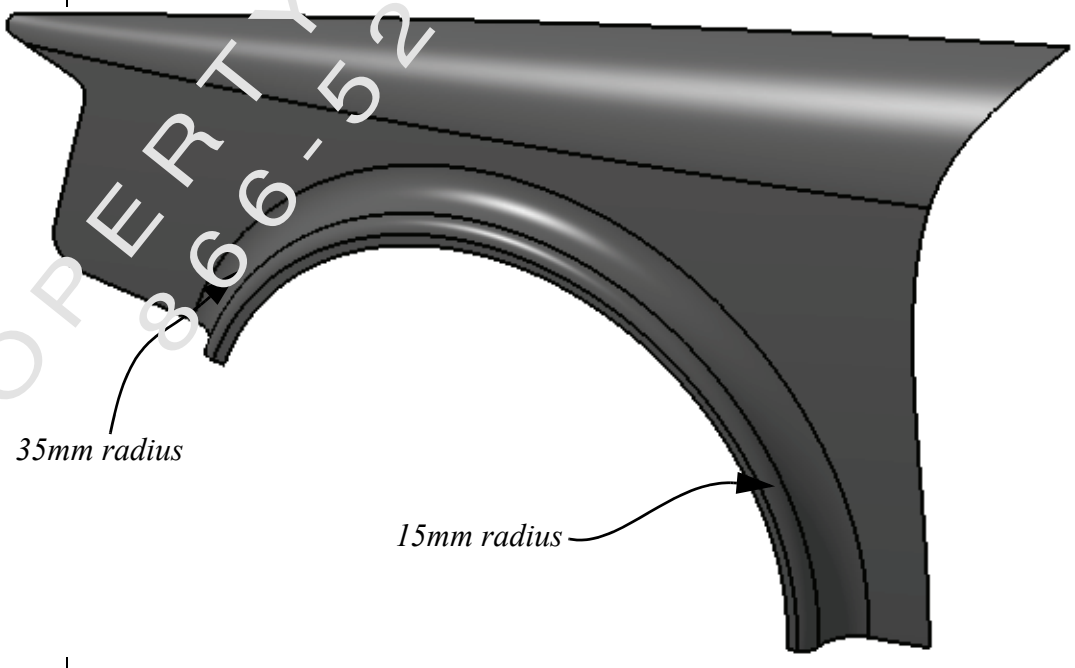

Figure 1–67

Task 5 - Organize the specification tree.

1. Right-click on Geometrical Set.1 and select **Geometrical Set.1 object > Create Group.** The Group dialog box opens as shown in Figure 1–68.

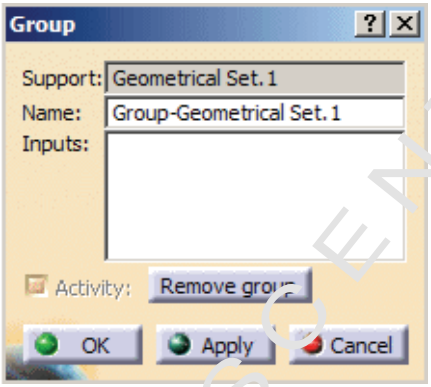

Figure 1–68

2. Select **Wheel Extrude** in the specification tree and every feature after it, as shown in Figure 1–69.

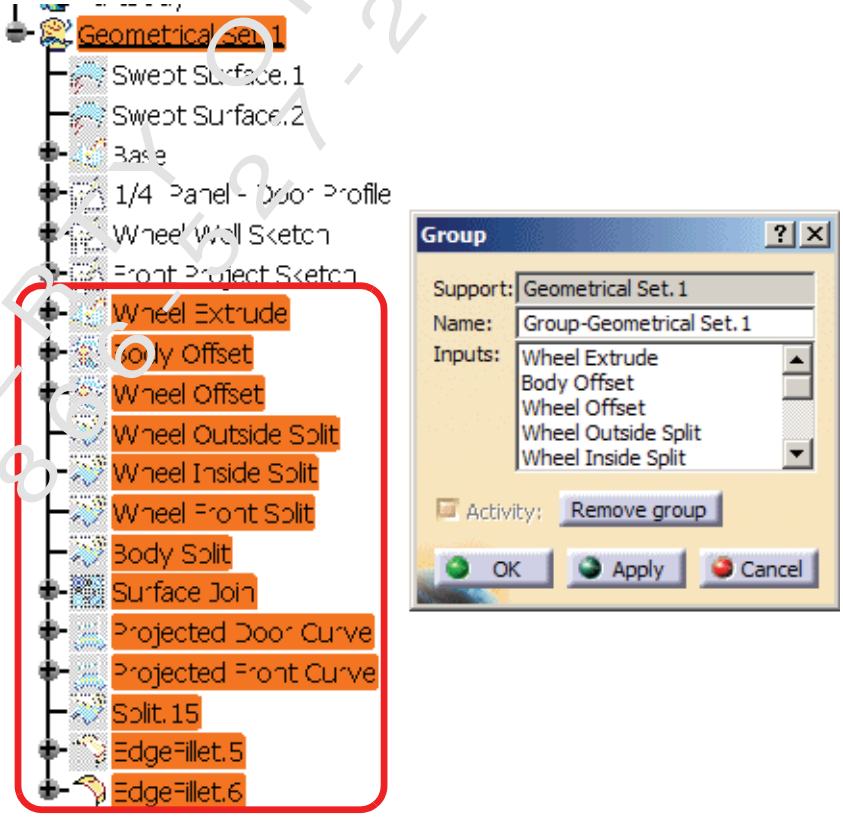

Figure 1–69

3. Click OK in the Group dialog box. The specification tree displays as shown in Figure 1–70. Note that only the selected features display in the group. This tool simplifies the specification tree.

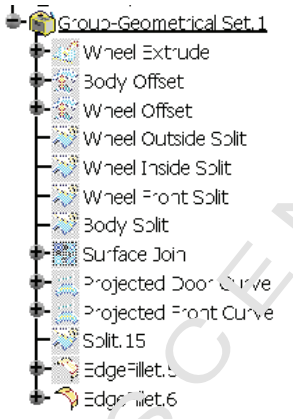

Figure 1–70

4. The wireframe and surfaces not in the group can be shown by right-clicking on Group-Geometrical Set.1 and selecting **Geometrical Set.1 object > Expand Group**. The specification tree displays as shown in Figure 1–71.

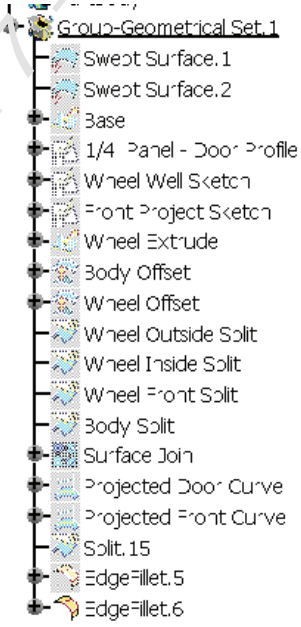

Figure 1–71

5. Right-click on Group-Geometrical Set.1 and select **Geometrical Set.1 object > Collapse Group.** The specification tree only displays the features placed in the group.

6. Save the model and close the file.

Exercise 1c

Using a Mask

In this exercise, you will open a part that contains imported surface data. You use the mask functionality to create two mask states of the model. The third and final mask that you create will be used to return the model to a state that displays all geometry in the part.

Goal

After you complete this exercise, you will be able to:

- ✓ **Create masks for a part file**

Task 1 - Open a part file.

1. Open **71499_Rotor.CATPart**. The model displays as shown in Figure 1–72.

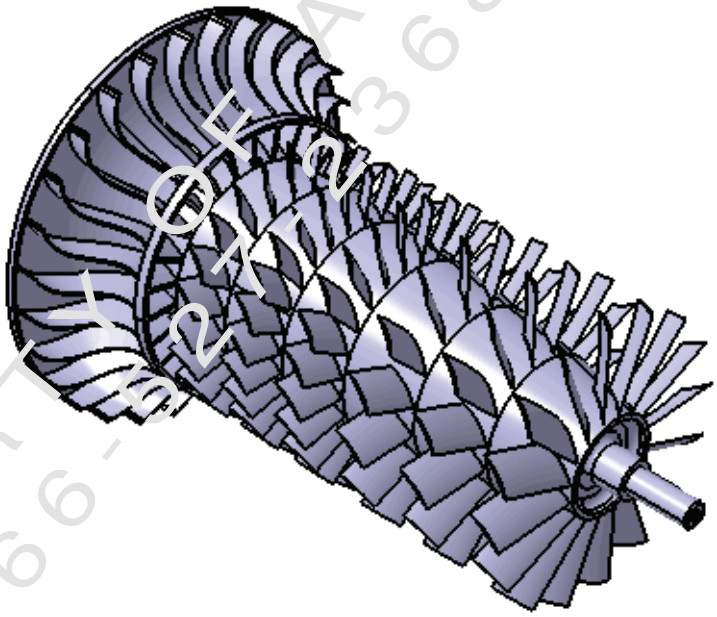

Figure 1–72

Task 2 - Create a mask.

1. Click  (Mask) in the Tools toolbar, as shown in Figure 1–73.

Figure 1–73

A bounding box displays around the model. Each side of the box has a green handle, as shown in Figure 1–74.

Your bounding box could have a different orientation then the one shown. The bounding box is displayed relative to the orientation of the model.

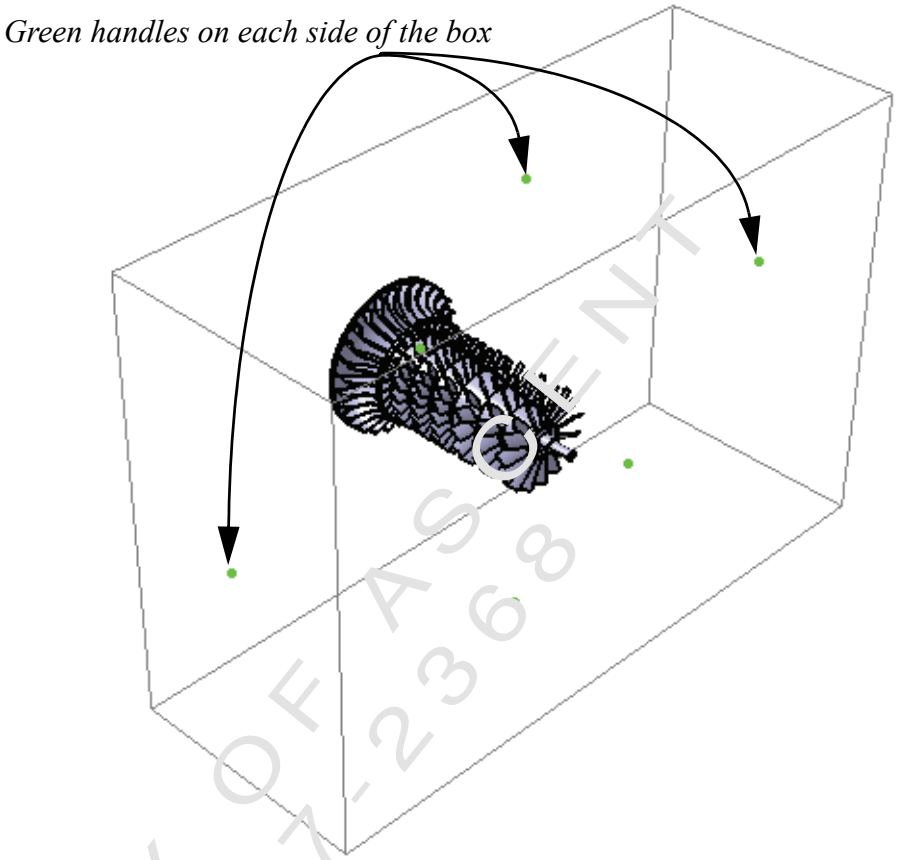

Figure 1–74

2. Use the cursor to select the handle shown in Figure 1–75.

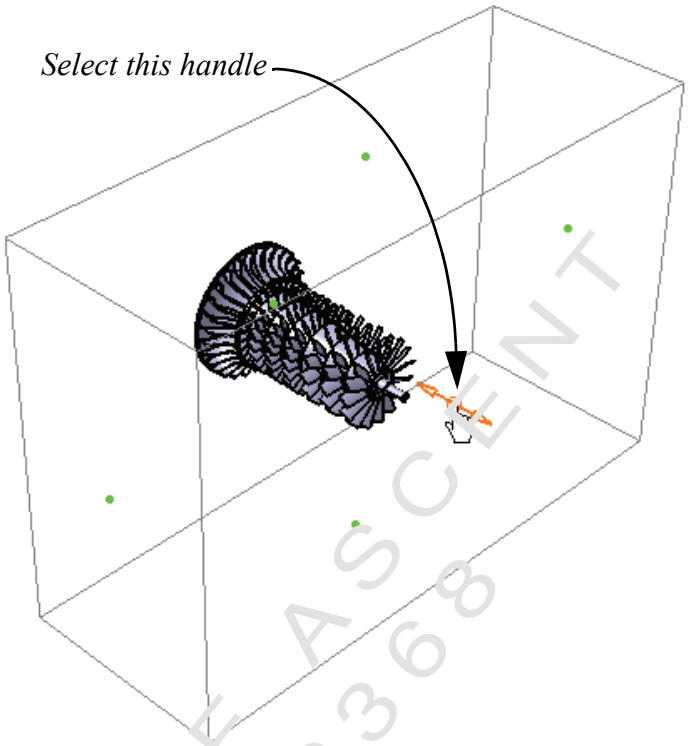

Figure 1–75

3. Drag the handle to the location shown in Figure 1–76.

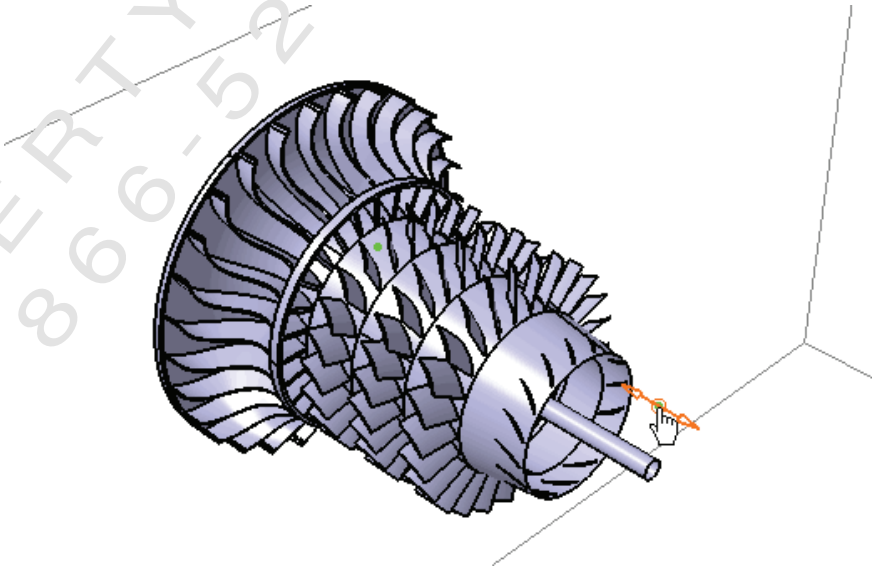

Figure 1–76

4. Click OK. Note that Mask.1 has been added to the specification tree.

Task 3 - Create a second mask.

1. Create another mask that results in the display shown in Figure 1–77.

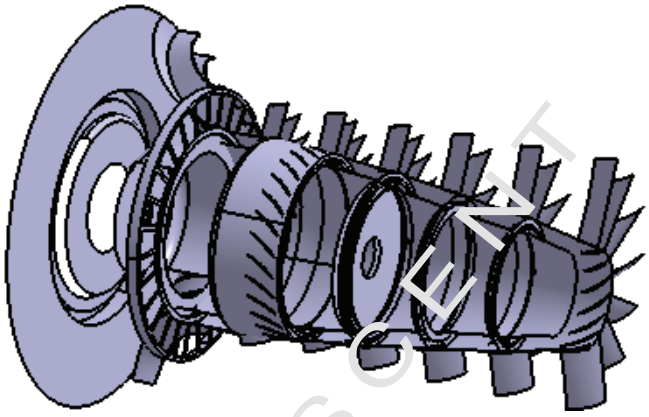

Figure 1–77

2. Click OK to complete the second mask.

Task 4 - Set a mask to be current.

1. Right-click on Mask.1 and select **Mask.1 object > Set As Current**. An active mask is indicated with a red icon, as shown in Figure 1–78.

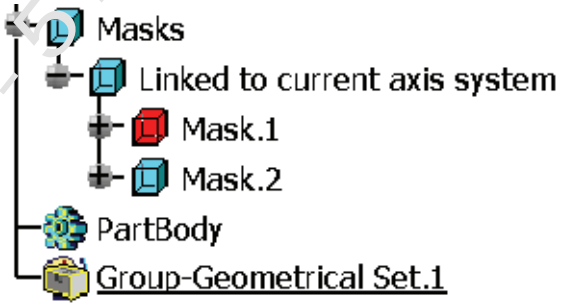

Figure 1–78

The model display updates as shown in Figure 1–79.

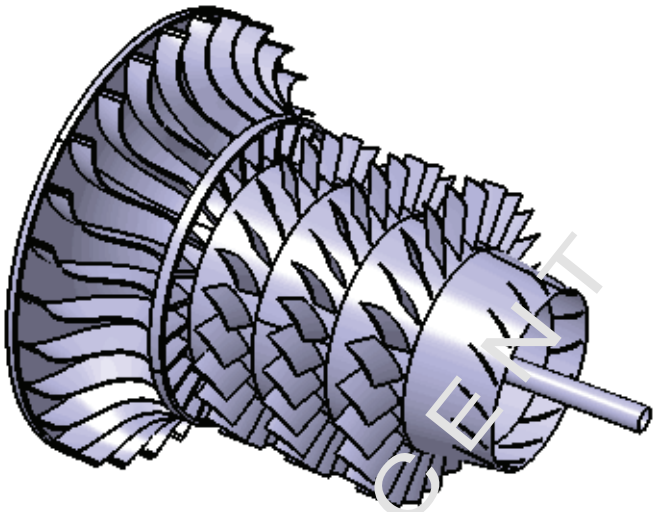

Figure 1–79

Task 5 - Set a mask to not be current.

1. Right-click on Mask.1 and select **Mask.1 object > Set As Not Current**. The model displays as shown in Figure 1–80.

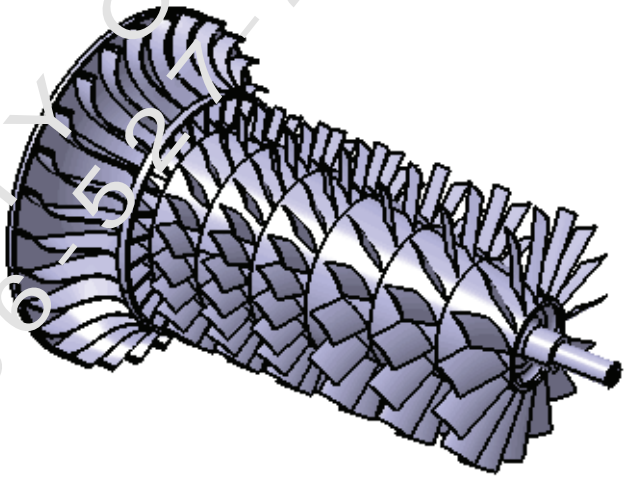

Figure 1–80

2. Save the model and close the file.