CATIA V5 Surface Design

DASSAULT SYSTEMES

See what you mean

CATIA V5 Surface Design

Student Handbook

Version 5 Release 19

24 hours

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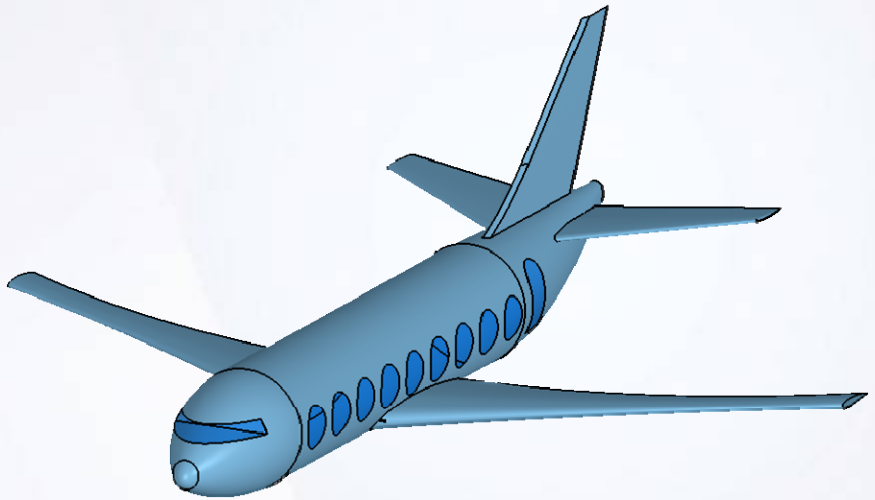
Introduction 1

Learning Objectives:

Upon completion of this lesson you will:

- ✓ Be introduced to Generative Shape Design.
- ✓ Learn how to manage features in Specification Tree.



2.5 hours

Case Study

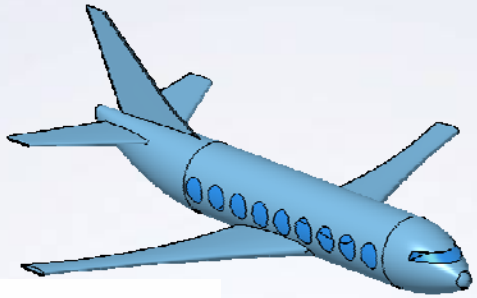
The case study for this lesson is to understand how to access Surface Design workbench and to manage few basic tools.

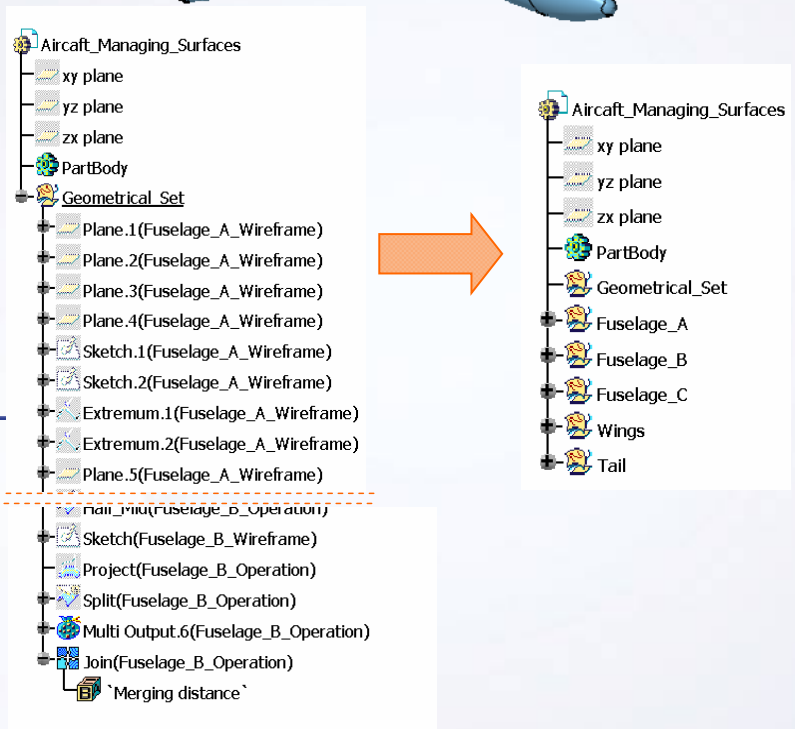
Design Intent

- ✓ Get familiar with the user interface.
- ✓ Create separate containers for Surfaces, Wireframes and Operations for different parts of the Aircraft.
- ✓ Finally Group the features from each Geometrical Set (which are referred in the model), thus minimizing the tree length of the tree.

Stages in the Process

1. Access the Generative Shape Design workbench.
2. Scan the model to better understand the modeling sequence.
3. Create the Geometrical Sets.
4. Group the features.



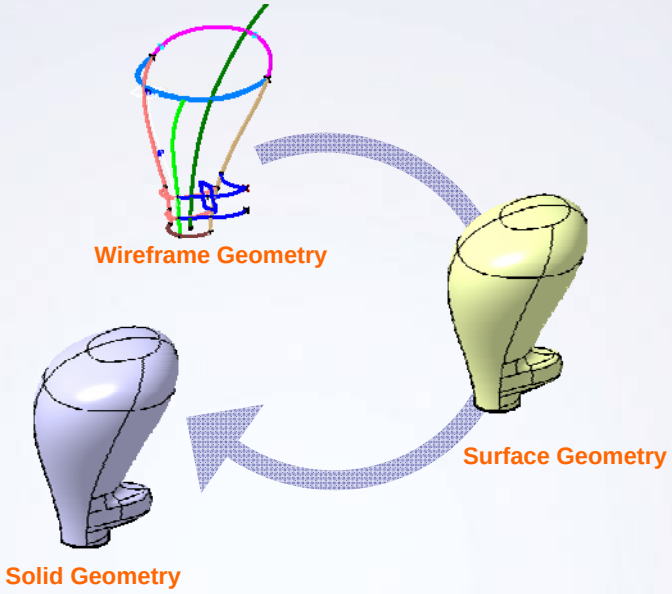


Introduction to Generative Shape Design

Generative Shape Design workbench allows you to define models with complex shapes using wireframe and surface geometry. Surface geometry may be integrated into the solid model to capture its design intent.

Use the following general steps while creating a surface-based feature:

- ✓ Access the Generative Surface Design workbench.
- ✓ Create the Wireframe geometry.
- ✓ Create the Surface geometry.
- ✓ Trim and join the body surfaces.
- ✓ Access the Part Design workbench.
- ✓ Create a part body.
- ✓ Modify the geometry as needed.

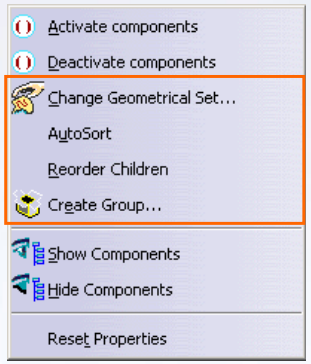


Managing Features in Specification Tree

In complex models, features must be structured in a logical way. It helps in better understanding of the designing process and reduction in tree size. Structuring can be done using a Geometrical Set or an Ordered Geometrical Set.

Geometrical Set (GS): This is a default container for wireframe and surface elements. The features are not displayed according to the logical update order.

Ordered Geometrical Set (OGS): It takes into account the update order of the features. OGS are equivalent to part design bodies.



Commonly used Feature Management Tools

Exercise 1B

Recap Exercise

15 min

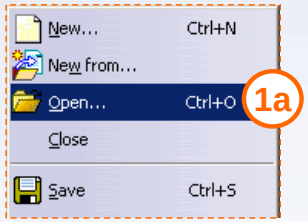
In this exercise, you will observe the basic difference in the characteristics of a Geometrical set (GS) and an Ordered Geometrical set (OGS). You will also observe the characteristics of GS and OGS in the hybrid design environment. High-level instructions for this exercise are provided.

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

- **Understand the behavior of OGS and GS when implied with reorder command.**
- **Understand the Parent/Child structure under a Hybrid design environment.**

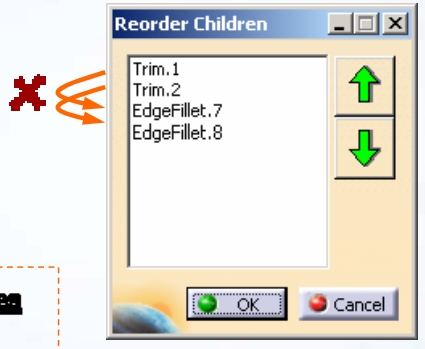
Exercise 1B (1/3)

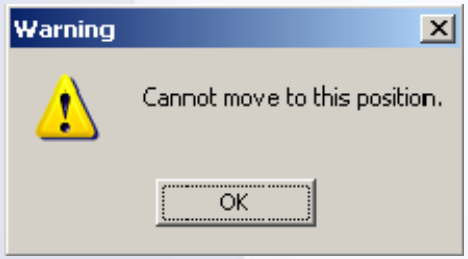
1. **Open the part.**
   - Open an existing part file. This file contains surface and wireframe features in the Geometrical set and in the Ordered Geometrical set.
     a. Browse and open the part: Exercise_1B_Start.CATPart

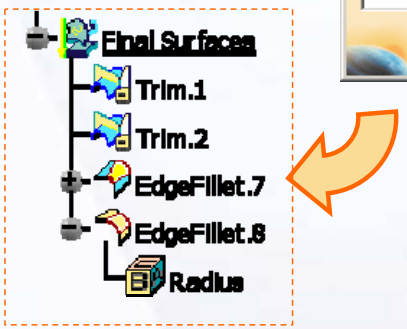


2. **Scan the Ordered Geometrical set.**
   - You will study the model to understand its sequence of modeling.

3. **Reorder the features in the Ordered Geometrical set.**
   - The Ordered Geometrical set will not allow you to reorder the features, as the features are arranged in the order of their creation.



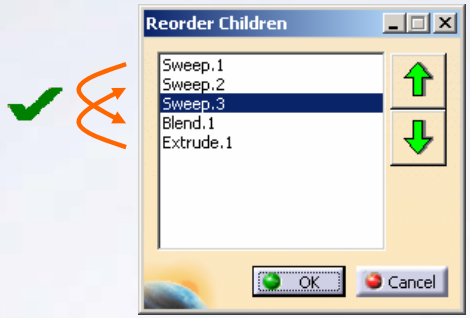




Exercise 1B (2/3)

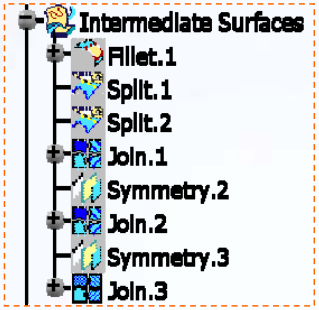
4. Reorder the features in the Geometrical set.

- You can reorder the features in the geometrical set, as the features are not arranged in the order of creation.



5. Reuse the parents in Geometrical set and Ordered Geometrical set.

- In a geometrical set, when a new operation is performed the parents are automatically hidden. You can reuse the parents available in No show to perform another operation. In the geometrical set 'Intermediate Surfaces', you can reuse the parent surfaces of Split.1 or Split.2 to create a new feature which is not associated with the Split.1 or Split.2 features.
- In an Ordered Geometrical set (OGS), the parents are consumed by the operation and can not be reused or seen in the hide mode. This allows the OGS to maintain the sequence of operation in which the features were created. Here in the model, the parents of the feature in the OGS are consumed by the operation and not seen in the hide mode.



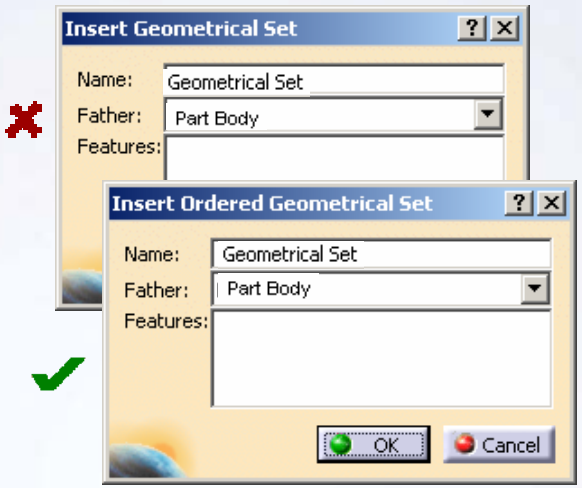
Parents get hidden after each operation

Parents get consumed after each operation

Exercise 1B (3/3)

6. Insert a new Geometrical set.

- Try to insert the geometrical set inside the part-body. You will not be allowed.
- Try to insert an Ordered Geometrical set inside the part-body. You will be allowed.

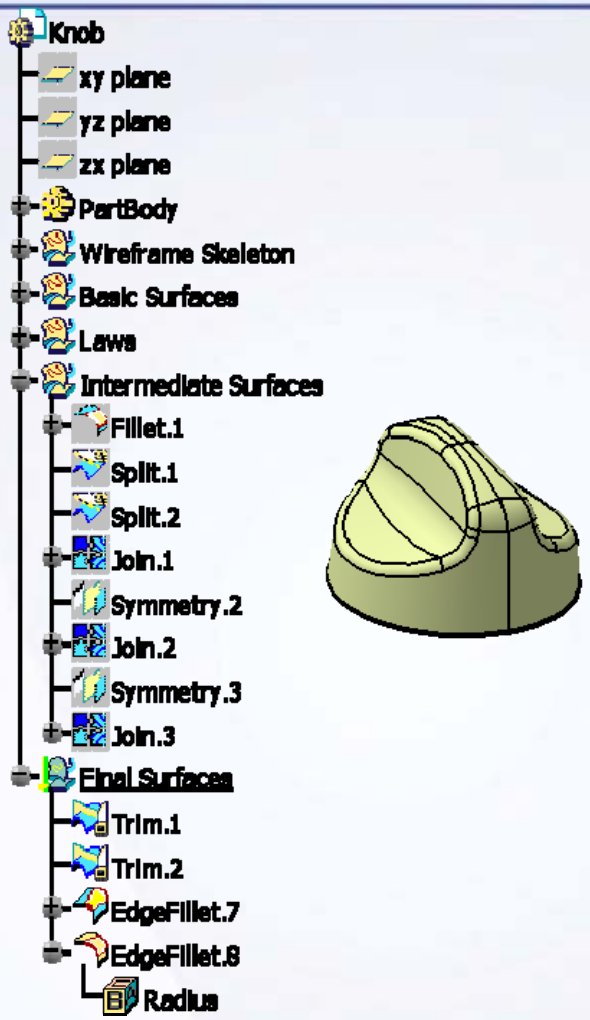


Observations

1. The above action is possible as the Hybrid mode has been activated while creating the part.
2. While in the Hybrid mode, you an create surface and wireframe features inside a part body.

Exercise 1B: Recap

- ✓ Understand the behavior of OGS and GS when implied with reorder command.
- ✓ Understand the Parent/Child structure under a Hybrid design environment.



Case Study: Introduction to GSD

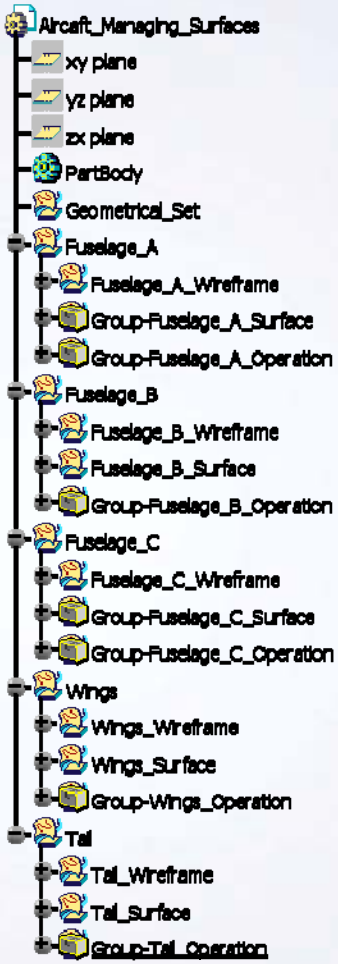
Recap Exercise

30 min

In this exercise, you will practice feature management in the specification tree and will be able to reorganize the specification tree when it becomes too complex or too long.

- ✓ *Open the given part consisting of the Air Craft model in the **Generative Shape Design** Workbench.*
- ✓ *Get familiar with the user interface.*
- ✓ *Study the Surface, Wireframe and Operations.*
- ✓ *Create separate containers for Surfaces, Wireframes and Operations for different parts of the Aircraft.*
- ✓ *Finally Group the features from each Geometrical Set (which are often referred in the model), thus minimizing the tree's length.*

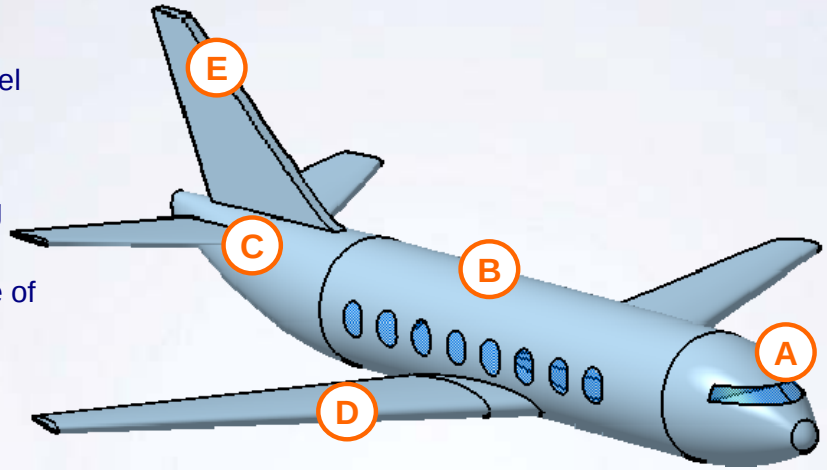
Using the techniques mentioned in this lesson and tips from the previous exercises, create the model without detailed instruction.



Do It Yourself: Introduction to GSD

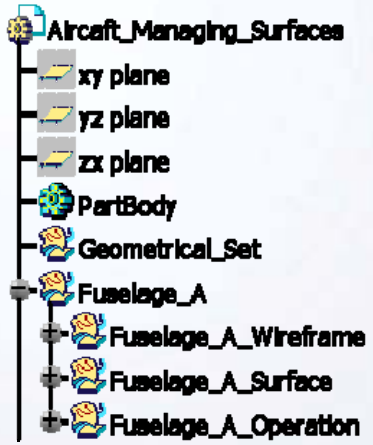
1. **Open the given part 'Air Craft' model in the Generative Shape Design Workbench.**
   a. Browse through the files and open the model Case_Study_Air_Craft_Start.CATPart

2. **Scan the model to understand the modeling sequence.**
   a. Study the model according to the sequence of its operation, which is used to build the surface, with the help of the **Scan** tool.
   b. Observe the nomenclature given to each feature inside the Geometrical set.



3. **Create separate containers for different parts of the Aircraft. Regroup the features according to their names.**
   a. Create individual Geometrical Set for each part of the Aircraft.
   b. Under the Geometrical Sets, create sub-sets for separating the Wireframes, Surfaces and Operations (as shown in the Specification tree).

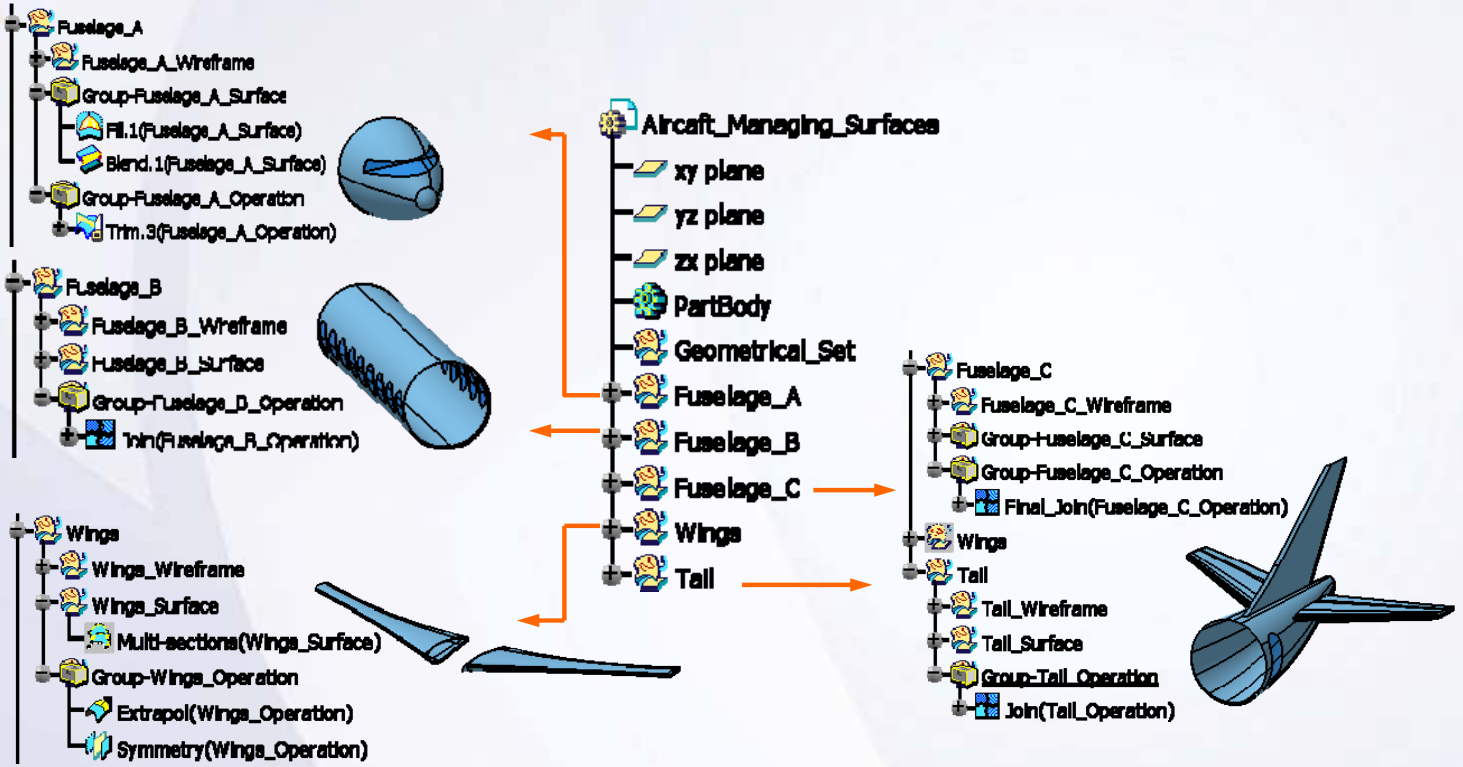
A. Fuselage_A
B. Fuselage_B
C. Fuselage_C
D. Wings
E. Tail



Do It Yourself: Introduction to GSD

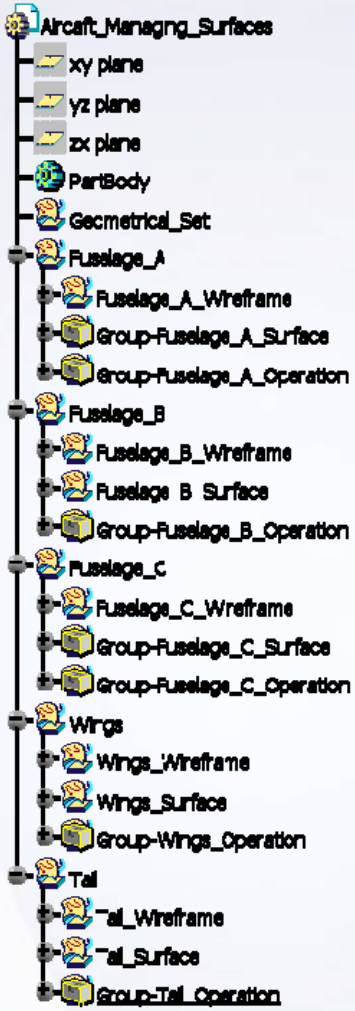
4. Group the features.

a. Create a **Group** of the features which are often referred during the modeling process. By this unwanted surfaces are not seen in the Specification tree and the tree's length reduces.



Case Study: Introduction to GSD Recap

- ✓ Open the given part ‘Air Craft’ model in the Generative Shape Design Workbench.
- ✓ Familiarize with the user interface.
- ✓ Study the Surface, Wireframe and Operations.
- ✓ Create separate containers for Surfaces, Wireframes and Operations for different parts of the Aircraft.
- ✓ Group the features from each Geometrical Set (which are often referred in the model) minimizing the tree’s length.

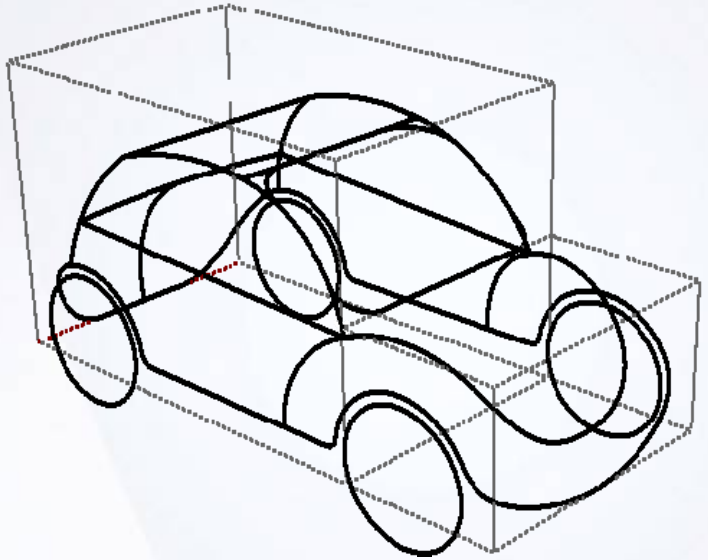


Wireframe Creation 2

Learning Objectives:

Upon completion of this lesson you will be able to:

- ✓ Create Reference Geometry
- ✓ Create 3D Curve
- ✓ Manage Curve Continuity
- ✓ Remove Unhealed Defects of the Curve



2.5 hours

Case Study

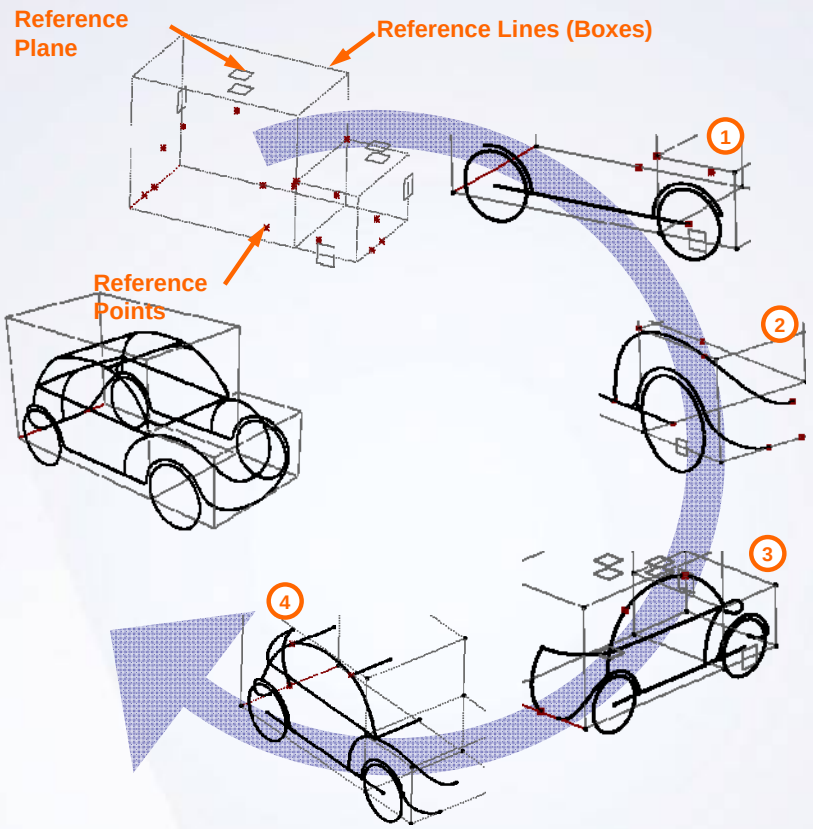
The case study for this lesson is to create a wireframe model of a toy car as the first stage of concept designing.

Design Intent

- ✓ Create a quick model of the car considering all the dimensions of the car. This is to understand the overall shape of the model while designing it.
  - Create reference boxes of the size required, using a curve mesh.
- ✓ Create feature lines on the model to understand the shape and visual characteristics of the car.
  - Create a spline and connect a curve to form 3D feature lines

Stages in the Process

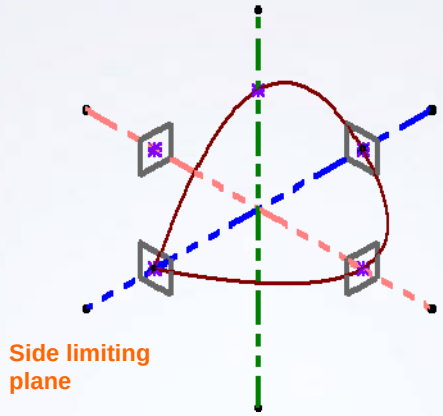
1. Create the wheel features of the car.
2. Create bonnet feature curves.
3. Create lower door features.
4. Create roof features.



Reference Geometry Creation

Reference geometries are the basic elements (planes, points, lines, axis), which provide a stable support to geometry. They can be used to design more intricate wireframe and surface geometries. A reference geometry can be used to limit and control the overall size of the part. They can also be renamed, based on its functionality in the model.

CATIA uses a fixed coordinate system called the Absolute Axis System. Any point in the model always has coordinates specific to this axis system. You can also define an arbitrary coordinate system located anywhere in three dimensional space and oriented in any direction. This user-defined axis system is called as Local Axis System. There can be multiple axis systems in a single part.



3D Curve Creation

A curve is said to be Continuous when the vertices of two curves join to form a single curve. These are of following type:

A. Point Continuity:

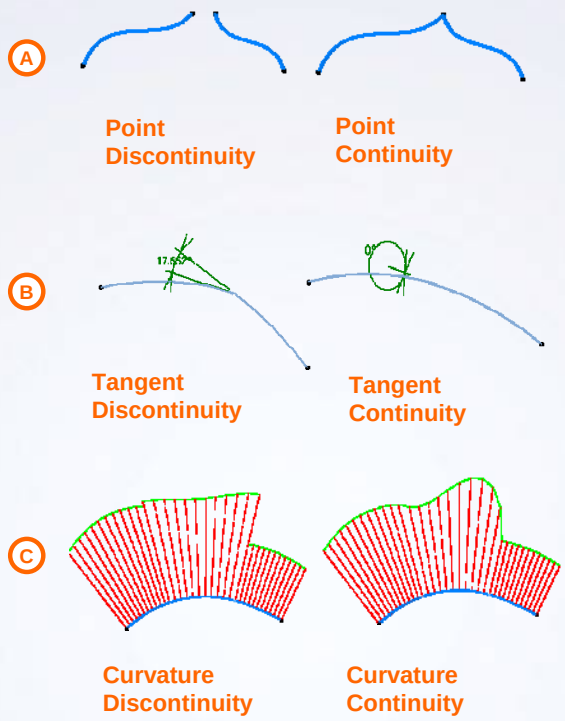
When the distance between two vertices of the connecting curve is within (less than) the specified CATIA V5 tolerance.

B. Tangent Continuity:

When angle between two normal curves at the connecting points is equal to zero or 180deg.

C. Curvature Continuity:

This is the rate of change of the angle of a tangent.



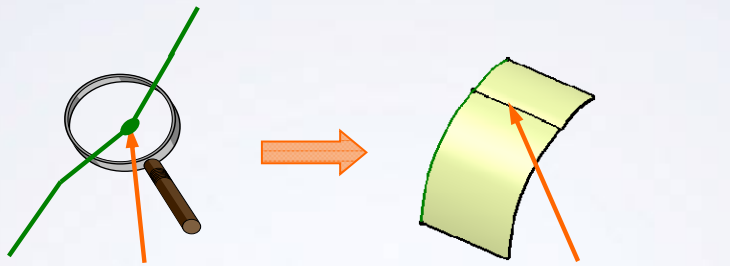
Curve Continuity Management

A surface derives many of its characteristics from the wireframe used to generate it. A defective surface will propagate the defect in downstream operations such as prototyping, machining, tooling, etc thus affecting the final product. Hence care must be taken while constructing a wireframe.

Tools used to detect geometrical discontinuities of curves are:

- ✓ Connect Checker Analysis
- ✓ Porcupine Curvature Analysis

The Curve Smooth tool allows you to correct the discontinuities in a curve up to a required extent by specifying the Threshold value. This value sets the upper limit of the discontinuity acceptance. The Maximum deviation value allows to set acceptable deviation between input curve and the smoothened curve. This tool repairs flaws such as Point, Tangent and Curvature discontinuity of the curve.



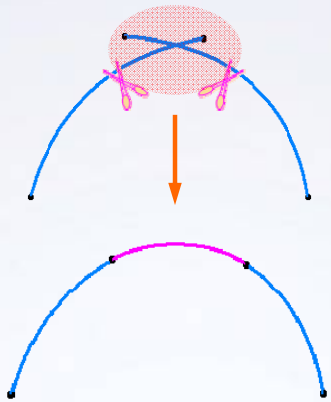
Curve with small flaw, used to make a surface

Curve will always transmit flaw to the surface

Removing Unhealed Defects of the Curve

The erroneous area of the curve such as self intersecting or overlapping cannot be healed using curve smoothen tool. In such situations you must remove the problem area and reconnect the curve to achieve a smooth result.

You can crop the erroneous area of the curve using the Split-Trim tools.



Two intersecting curves can be split and reconnect using connect curves.

Main Tools

Wireframe Toolbar

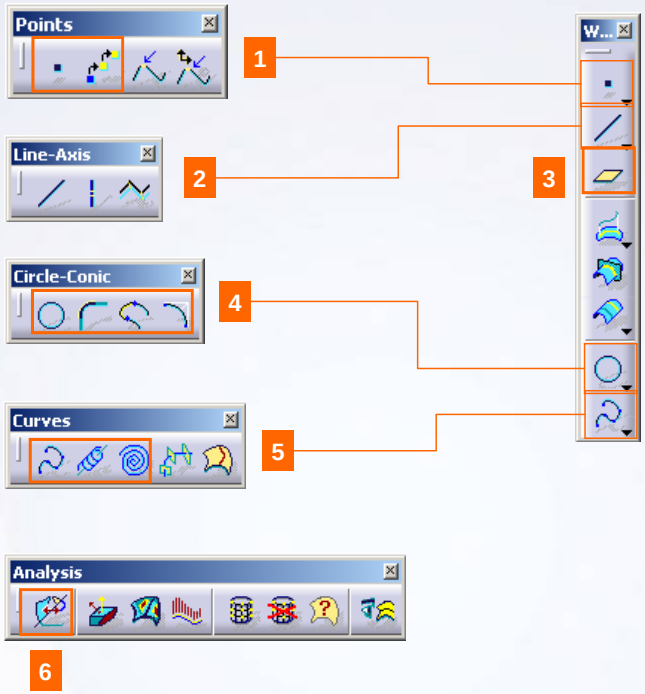
1 **Points:** Creates a point or multiple points.

2 **Line-Axis:** Creates lines, axis or polyline.

3 **Plane:** Creates planes using different options.

4 **Circle Conic:** Creates 3D curves.

5 **Curves:** Creates 3D curves like Spline, Helix and Spiral.



Analysis Toolbar

6 **Connect Checker Analysis:** Performs connection analysis of curves and surfaces.

Exercise 2B

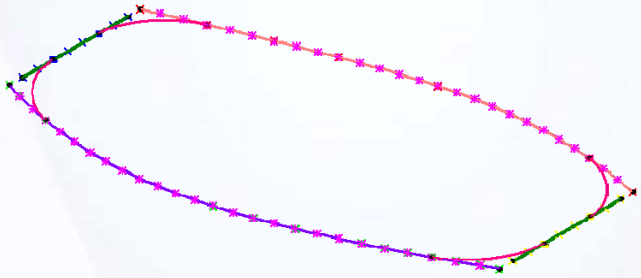
Recap Exercise

20 min

In this exercise, you will practice on some of the tools in 3D curves creation of GSD workbench.You will be provided with a set of points of a mirror shell model. You will be creating a basic curve using the points data. High-Level instructions for this exercise are provided.

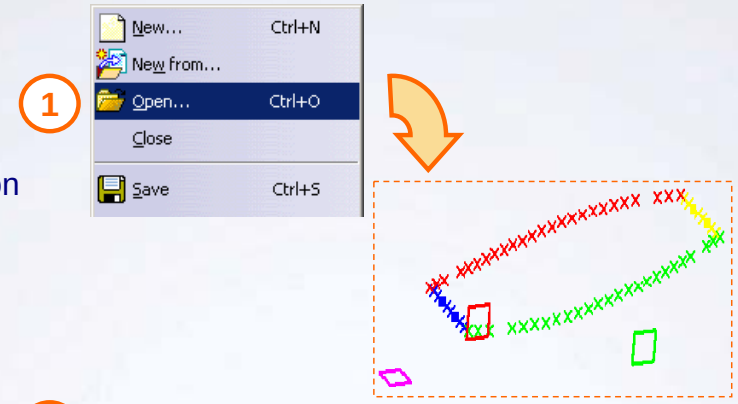
By the end of this exercise you will be able to:

- **Create a 2D Spline and an Arc.**
- **Create a 3D Spline and a Connect curve**
- **Create a Combine curve and a Curve project.**

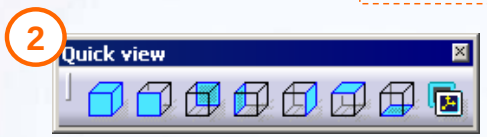


Exercise 2B (1/5)

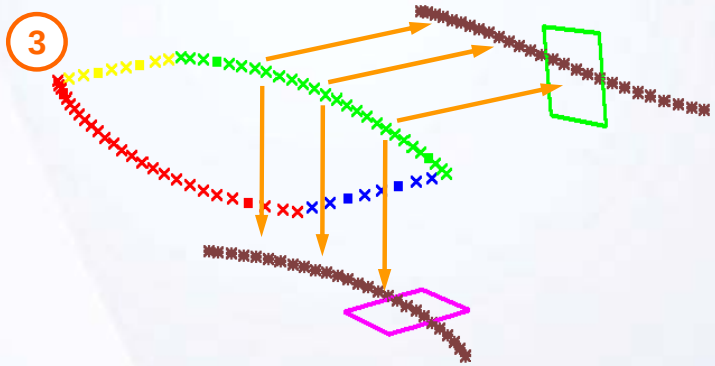
1. **Open the part.**
   - Open an existing part file Exercise_2B_Start.CATPart
   - The file consists of Scanned points and construction planes.



2. **Inspect the Point data with 'Quick view' Icons.**
   - Select the FRONT, LEFT and TOP icons. You should be able to determine that the green and red set of points are three dimensional, and the yellow and blue set of points are linear.

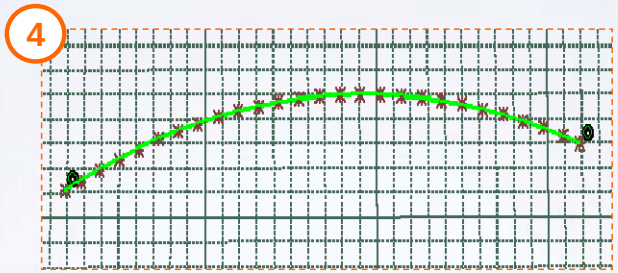


3. **Project the points on the given planes.**
   - Project the Green points on the Green and Pink Construction planes.

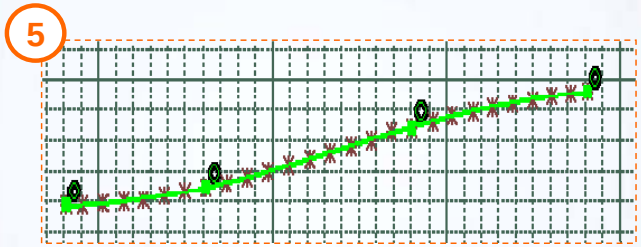


Exercise 2B (2/5)

4. **Create a partial curve on the Pink plane in a Sketcher using the points projected.**



5. **Create a sketch with a Spline using the projected points.**



6. **Repeat steps 3 ,4 and 5 for Red set of points.**
   - Project the Red points on the Pink and Red plane
   - Using these projected points create partial circles on the Pink plane, and Spline on the Red plane using a sketcher.

Exercise 2B (3/5)

7. Combine 2D curve into 3D shapes.

- Use the Combine tool to create 3D curve from the first and second sketched curve.
- Similarly, combine the curves third and fourth to create a 3D curve.

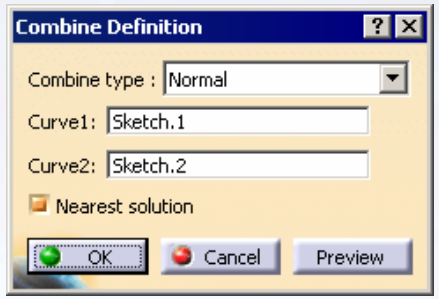
8. Create Line over Yellow and Blue set of points.

- Create a Line using the point to point option.

9. Create a point on 3D Curve.

- Create a point on both the combine curves at a distance ratio of 0.1 from both the ends of the curve.

7



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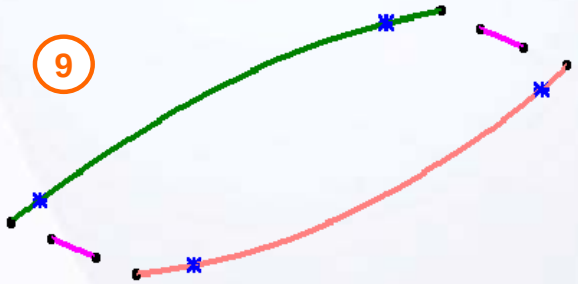
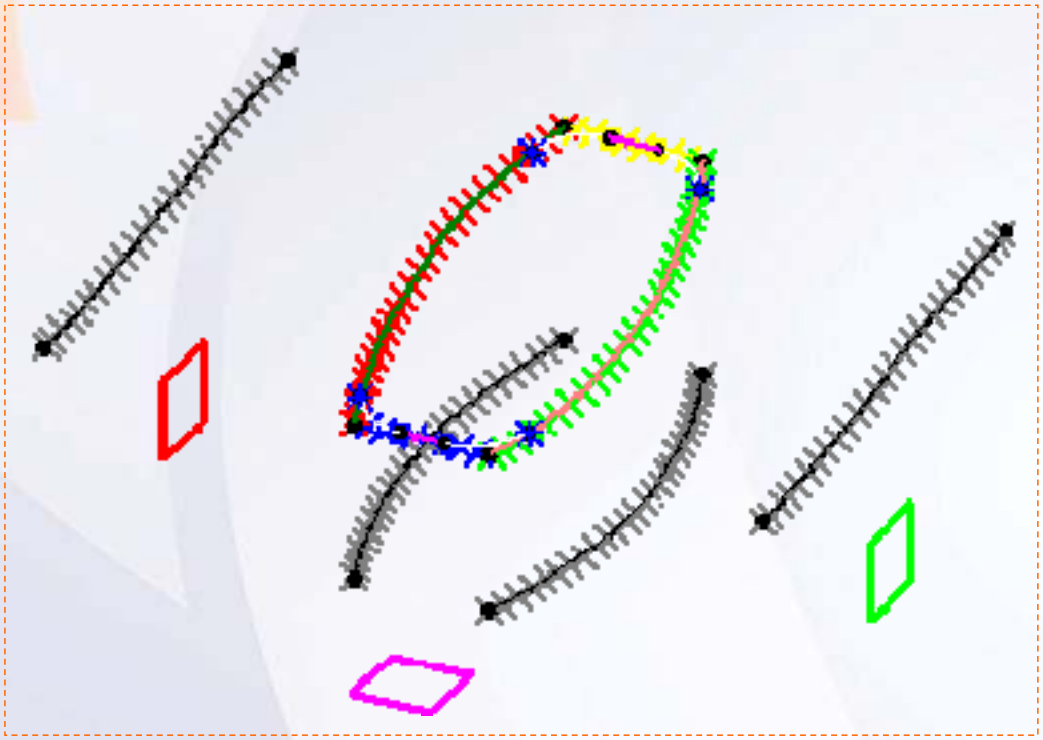


Exercise 2B (4/5)

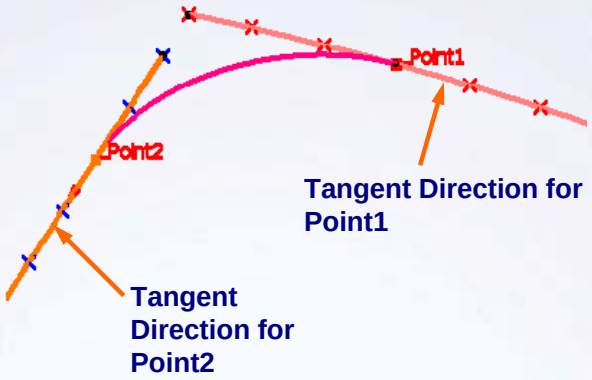
At this stage the part should look like this.



Exercise 2B (5/5)

10. Create a 3D Spline.

- Create a **Spline** at the corner where the Red and Blue points meet.
- Similarly, create a Spline at the corner where the Green and Blue points meet.

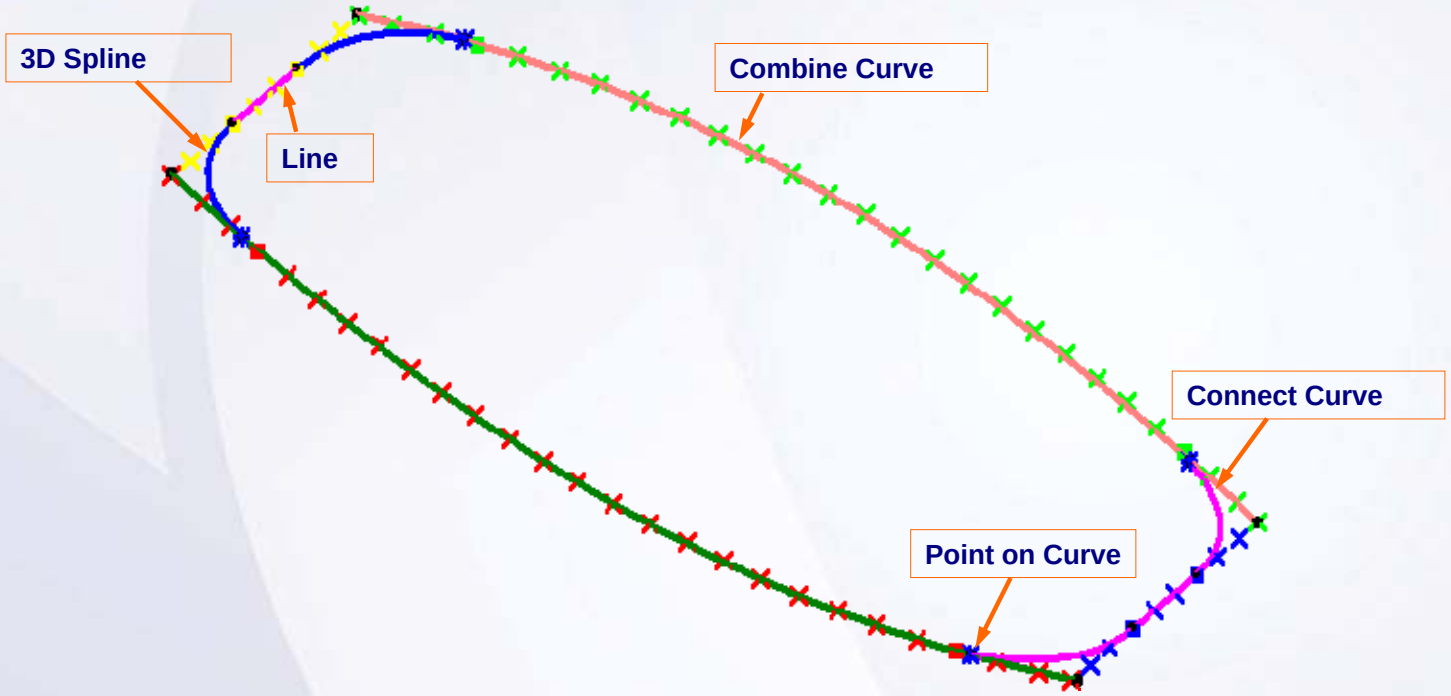


11. Create a Curve Connect.

- Create a **Connect Curve** at the corner where the Red and Yellow points meet.
- Similarly, create a Connect Curve at the corner where the Green and Yellow points meet.

Exercise 2B: Recap

- ✓ Create a 2D Spline and an Arc.
- ✓ Create a 3D Spline and a Connect curve.
- ✓ Create a Combine curve and a Curve project.



Exercise 2C

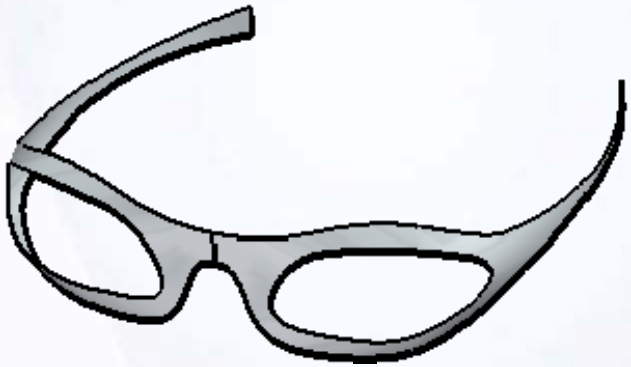
Recap Exercise

15 min

In this exercise, you will practice how to create 3D wireframes with surface support. You will be given a datum surface of sunglasses. You will be creating the feature profiles of sunglasses using the surface support. High-level instructions for this exercise are provided.

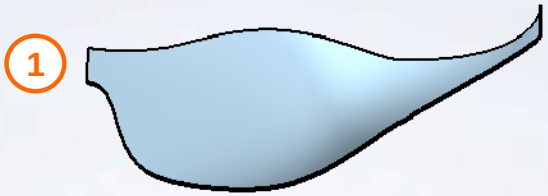
By the end of this exercise you will be able to:

- **Extract Boundary of an existing surface.**
- **Create a parallel curve using the surface support.**
- **Create a Corner using the surface support.**
- **Create points on the surface.**
- **Create 3DSpline on a surface.**

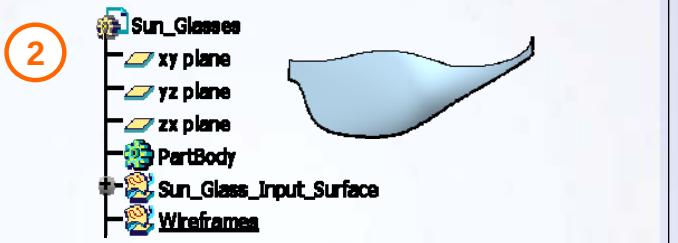


Exercise 2C (1/3)

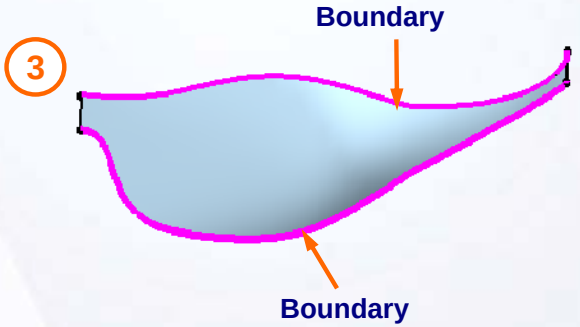
1. **Open the part.**
   - Open an existing part file. The file consists of datum surface of sunglasses.
     a. Browse and open part Exercise_2C_Start.CATPart



2. **Create a Geometrical Set.**
   - Insert a Geometrical set and rename it as Wireframes.



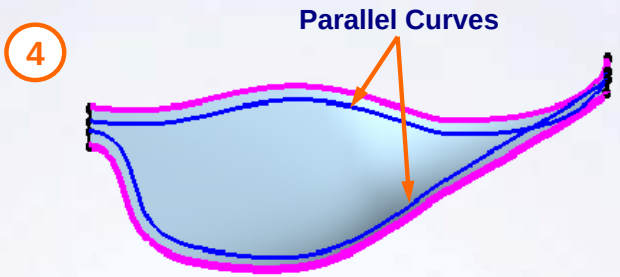
3. **Create a boundary**
   - Create a boundary at the top and bottom edge of the sunglass surface (as shown) using the boundary tool.



Exercise 2C (2/3)

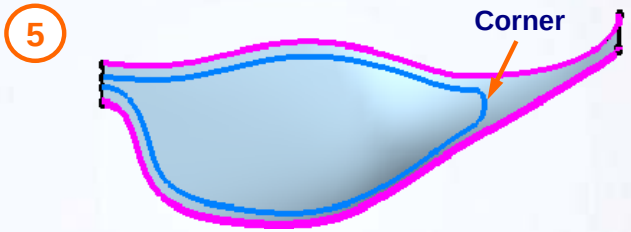
4. Create a Parallel curve

- Create a parallel curve on the sunglass surface from boundary curves using the Parallel curve tool (4mm offset).



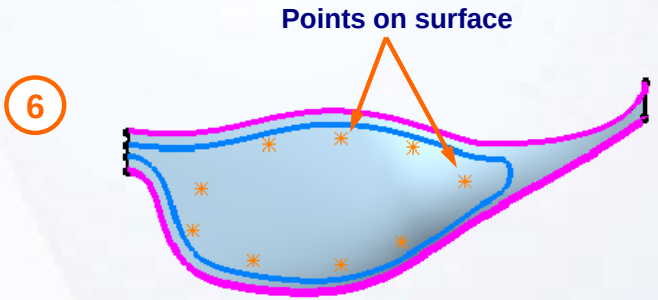
5. Create a Corner on the surface

- Create a corner between the two parallel curves at their intersection point using the Corner tool (5mm radius)



6. Create Points on the surface

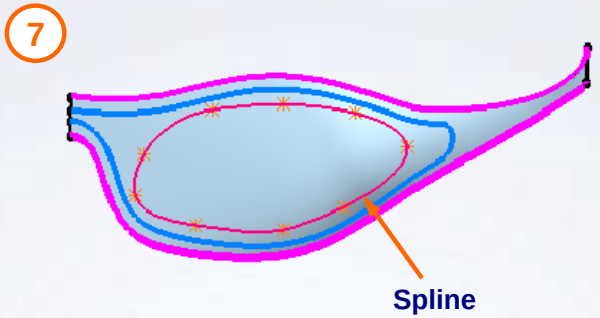
- Create nine points on the surface (use more points if required) using the Point tool.



Exercise 2C (3/3)

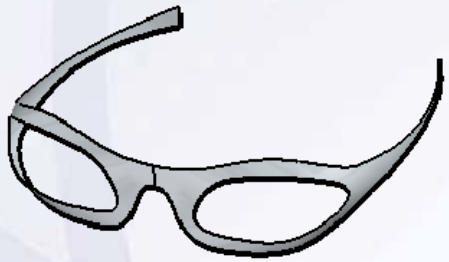
7. Create Spline on the surface

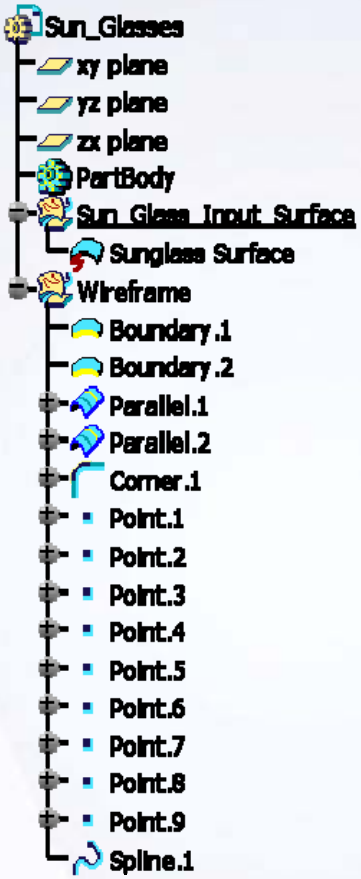
- Create a Spline by joining all the previously created points on the sunglass surface.



Exercise 2C: Recap

- ✓ Extract Boundary of an existing surface.
- ✓ Create a parallel curve using the surface support.
- ✓ Create a Corner using the surface support.
- ✓ Create points on a surface.
- ✓ Create 3DSpline on a surface.





Exercise 2E

Recap Exercise

15 min

In this exercise, you will be provided with a curve which has to be analyzed for its continuity and connections. No instructions will be provided for this exercise.

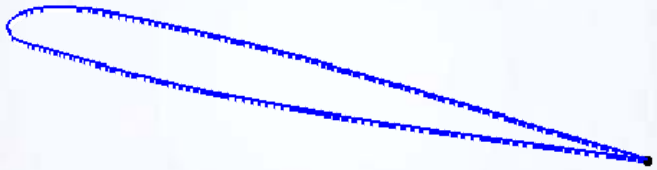
By the end of this exercise you will be able to:

- **Check continuity of a curve using Connecter checker.**
- **Perform the Porcupine analysis on a curve to check its Curvature Continuity.**

Exercise 2E

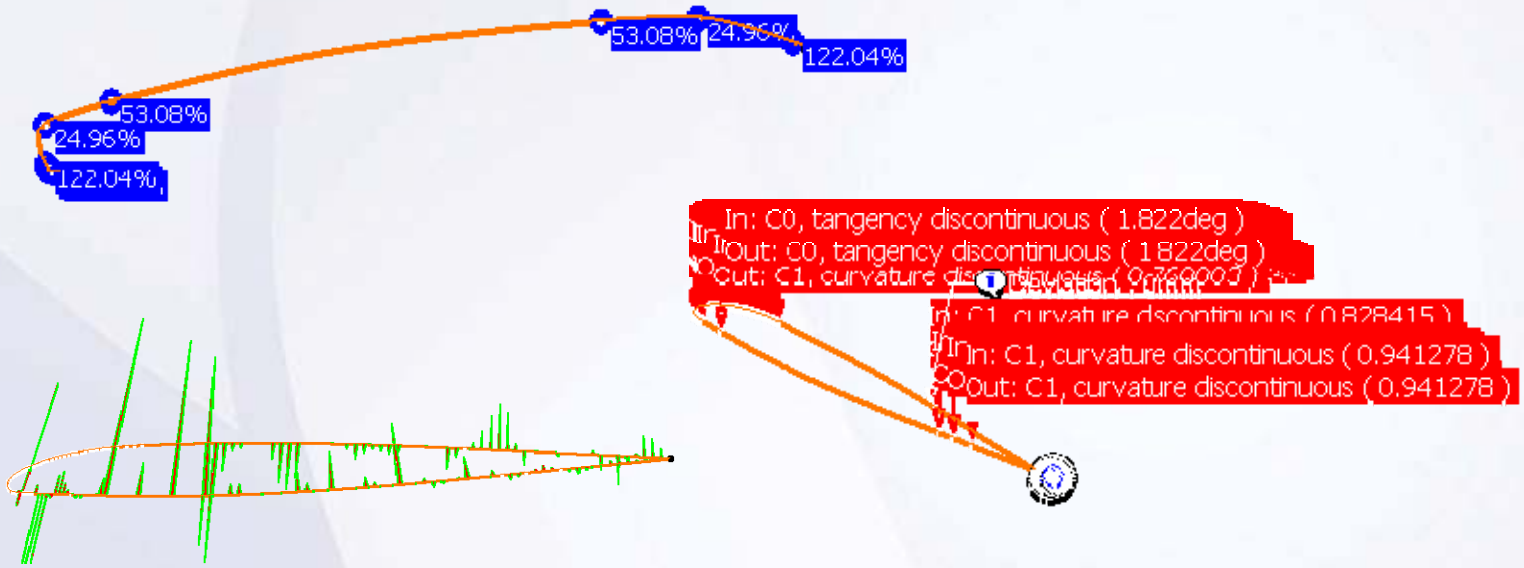
1. **Load the start data.**
   - Exercise_2E_Start.CATPart
2. **Analyze the Section Curve of a car's bumper for its geometrical discontinuities.**
   - Perform Connect Checker analysis to understand the discontinuities on the curve.
   - Perform the Porcupine analysis to check the inflection points and curvature continuities.
   - Smoothen the curve using the Curve Smooth tool.

3. **Analyze the Airfoil curve for its connections and continuities using Curve Connect Checker.**
   - Perform connect checker analysis to understand the discontinuities.
   - Smoothen the curve using the Curve Smooth tool.



Exercise 2E: Recap

- ✓ Check continuity of the curve using Connect checker.
- ✓ Perform the Porcupine analysis on a curve to check its curvature continuity.



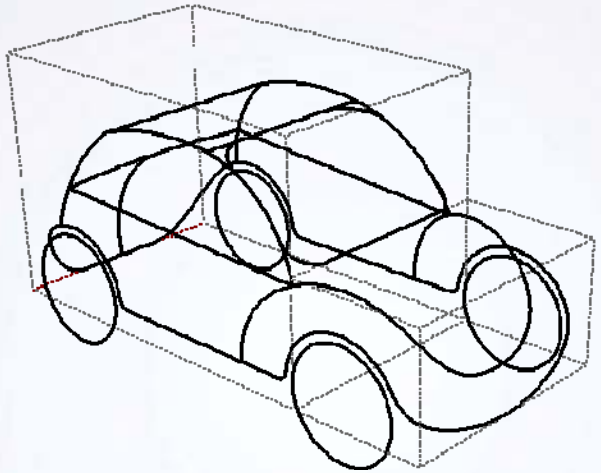
Case Study: Wireframe Creation

Recap Exercise

30 min

In this exercise you will practice with the Wireframe creation tools.

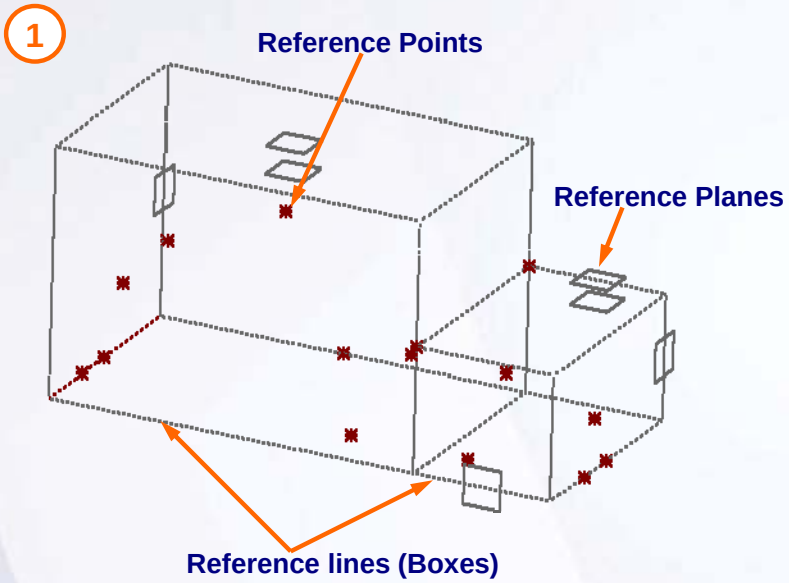
- ✓ Create a quick model of the car considering the overall dimensions of the car.
- ✓ Create feature lines on the model to understand the shape and visual characteristics of the car.



Using the techniques you have learnt in this lesson and previous exercises, create the model without detailed instruction.

Do It Yourself: Wireframe Creation (1/7)

1. **Open the given part consisting of reference elements to build a wireframe of the model of a Toy Car.**
   a) **Browse** through the files and open the model Case_Study_Start.CATPart



Do It Yourself: Wireframe Creation (2/7)

2. Create a front and back wheel curve.

a. Create a Circle at the point shown with a plane support of radius 60mm

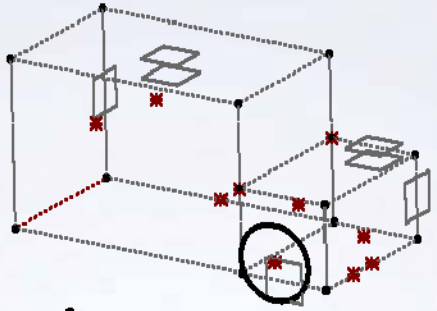
b. Create a part Circle with the same inputs from 0 deg to 155 deg. and Radius = 70mm.

a. Create a Line from the center of the circle, length 375mm along Y axis.

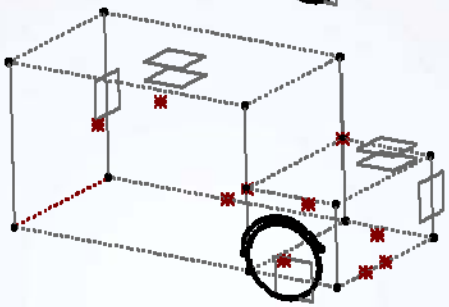
b. Create a Circle at the end point of the line as a center of radius 60mm (with initial plane support)

c. Create a part Circle with the end point of Line as center with start angle 75 deg and End angle 180 deg. Radius = 70 mm (with initial plane support)

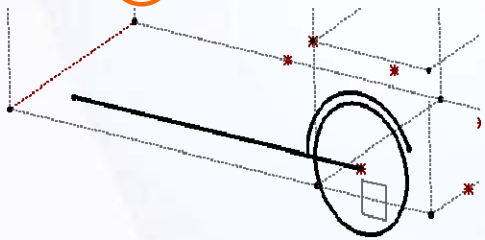
2a



2b



2c



2d

Do It Yourself: Wireframe Creation (3/7)

3. Create a Corner between elements.

a. Create a corner between the first part circle and line as shown. Specify the radius 20mm

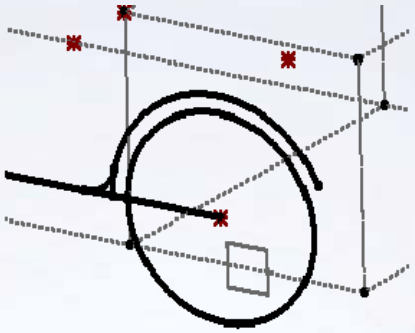
b. Create a corner between the second part circle and line as shown. Specify the radius 20mm

4. Create a Connect Curve between first part circle and point on reference curve.

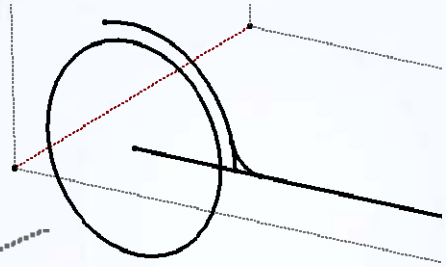
a. Specify the tension at both the ends (if required) to achieve desired shape.

b. Specify curvature continuity option.

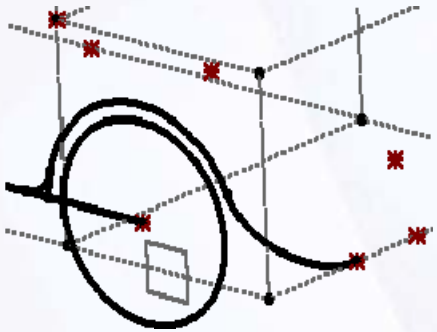
3a



3b

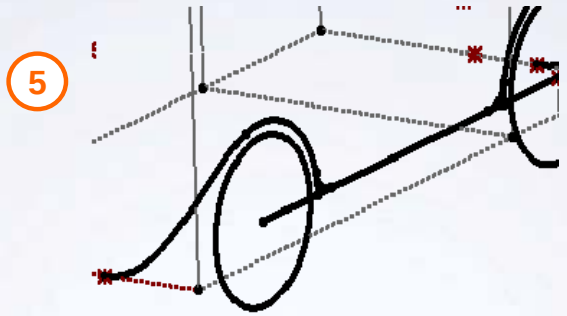


4a

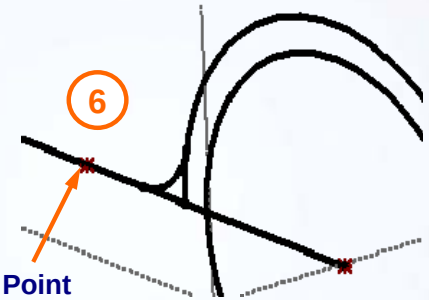


Do It Yourself: Wireframe Creation (4/7)

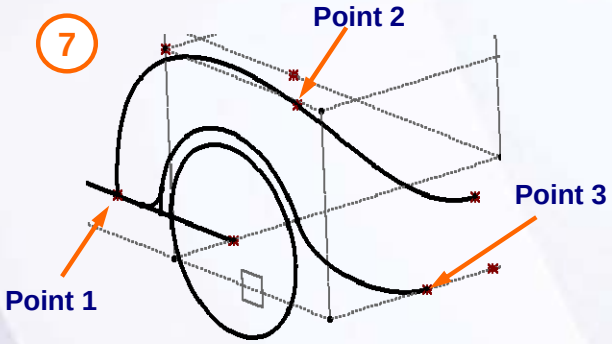
5. **Create a Connect Curve between Second part circle and point on reference curve.**
   a. Specify the tension at both the ends (if required) to achieve the desired shape.
   b. Specify the curvature continuity option.



6. **Create a Point on the line as shown.**
   a. Create a point at a distance ratio of 0.3.



7. **Create a Spline to form a Bonnet feature line.**
   a. Create a Spline passing through the points shown in the image.
   b. Specify the tangent direction and modify the tension values to attain the desired shape.



Do It Yourself: Wireframe Creation (5/7)

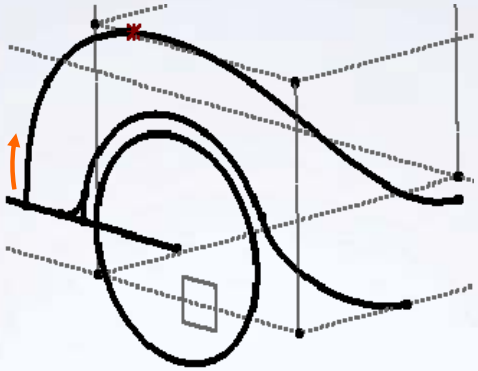
8. Create a body shell.

a. Create a Point on Spline created in previous operation at a distance ratio of 0.4.

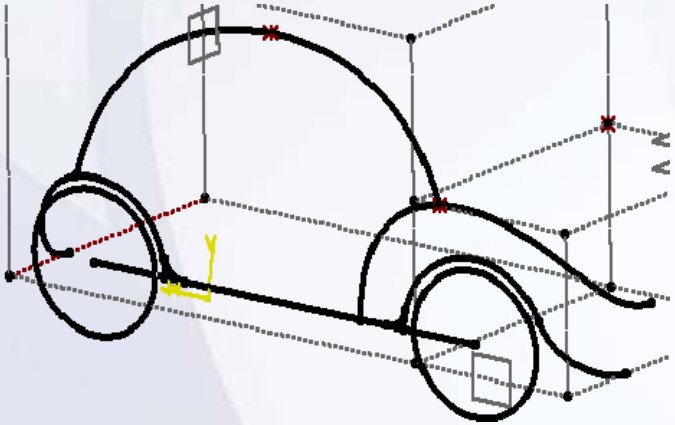
b. Create a Plane passing through three points as shown.

c. Create a sketch of a three-point arc passing through these three points.

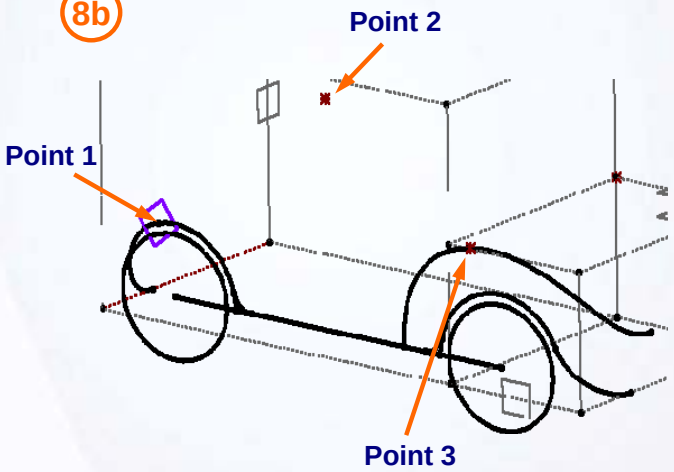
8a



8c



8b



Do It Yourself: Wireframe Creation (6/7)

9. Create a window and wind shield features.

a. Create a Point on the Spline created in the previous step at a distance ratio of 0.3.

b. Create a Point on the Spline created in the previous point at a distance ratio of 0.7.

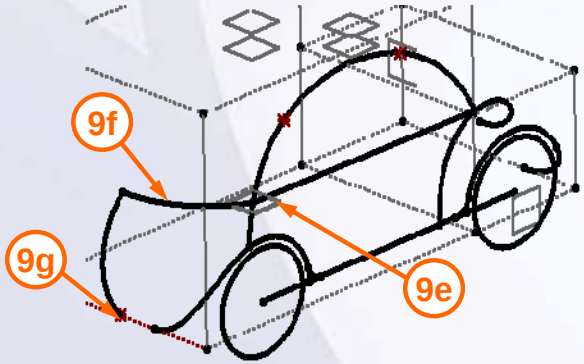
c. Create a Point on the Spline created in the previous point at a distance ratio of 0.9.

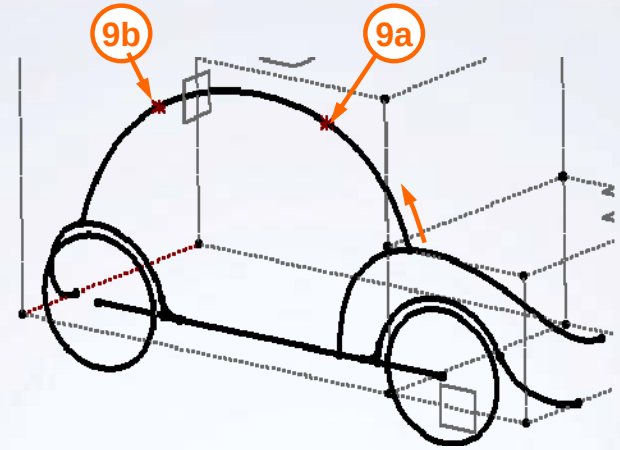
d. Create a line between two points as shown.

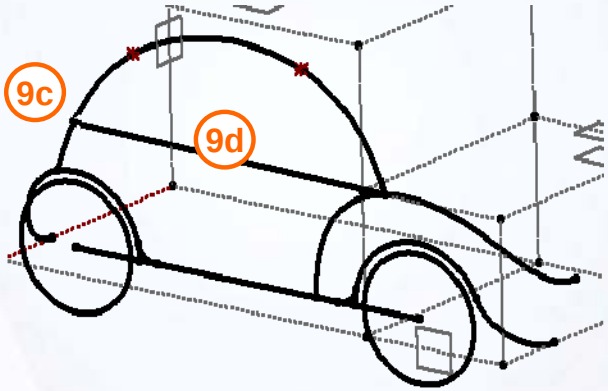
e. Create a Plane parallel to XY plane at the end point of the line previously created.

f. Create a sketch on the plane created in the previous step as shown.

g. Create a part circle joining the end point of the sketched curve and mid-point of the reference curve. Specify radius 100mm.







Do It Yourself: Wireframe Creation (7/7)

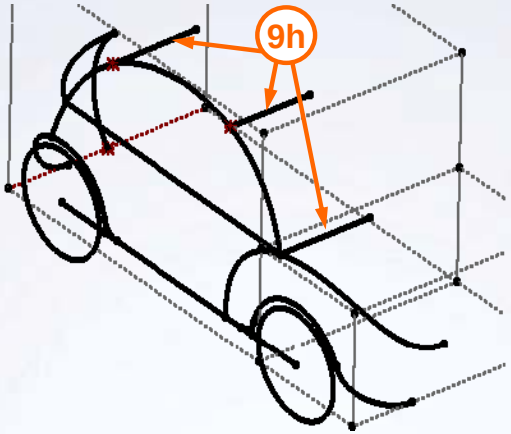
9. Create a window and wind shield features (continued…).

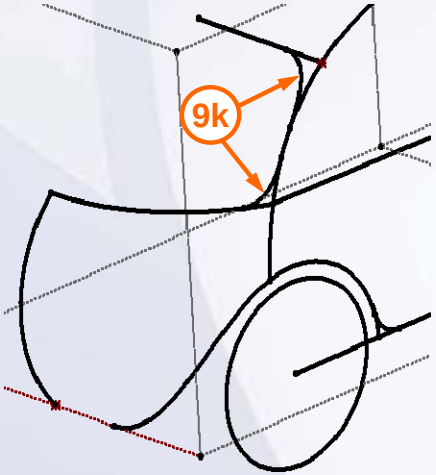
h. Create the lines up to YZ planes from a specified point.

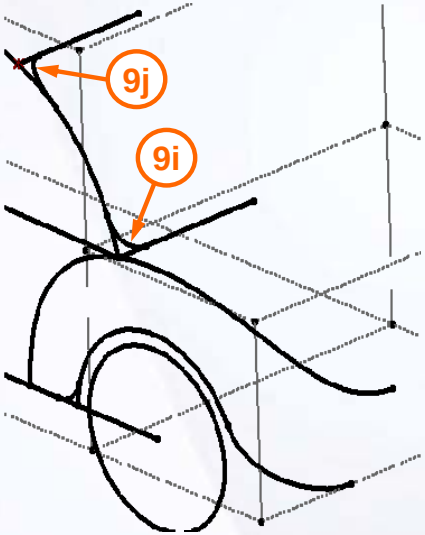
i. Create a 3D Corner at left corner of front wind shield. Specify radius of 20mm.

j. Similarly, create a 3D corner at top-left corner of the front wind shield. Specify radius 20mm.

k. Similarly, create a 3D corner at both the corners of the back wind shield. Specify radius 20mm.

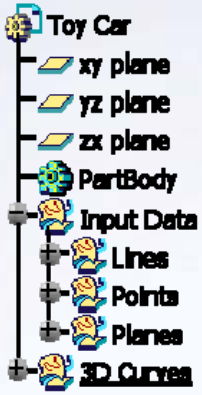


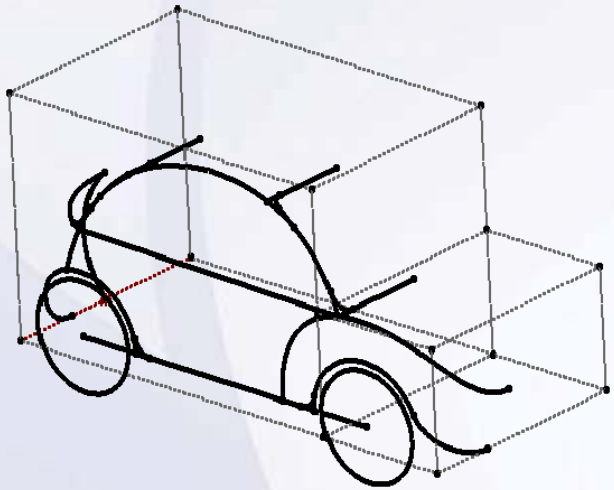


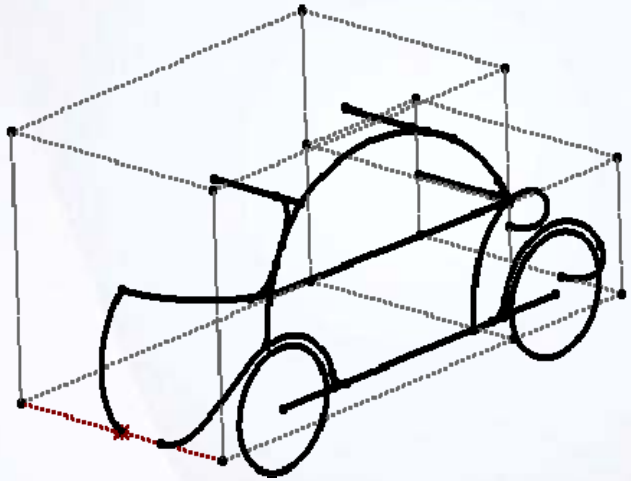


Case Study: Wireframe Creation Recap

- ✓ Create a quick model of the car considering the overall dimensions of the car.
- ✓ Create feature lines on the model to understand the shape and visual characteristics of the car.







Surface Creation 3

Learning Objectives:

Upon completion of this lesson you will be able to:

- ✓ Choose a Surface
- ✓ Sweep a Profile
- ✓ Extrude or Revolve a Profile
- ✓ Create a Multi-Section Surface
- ✓ Create an Adaptive Sweep Surface

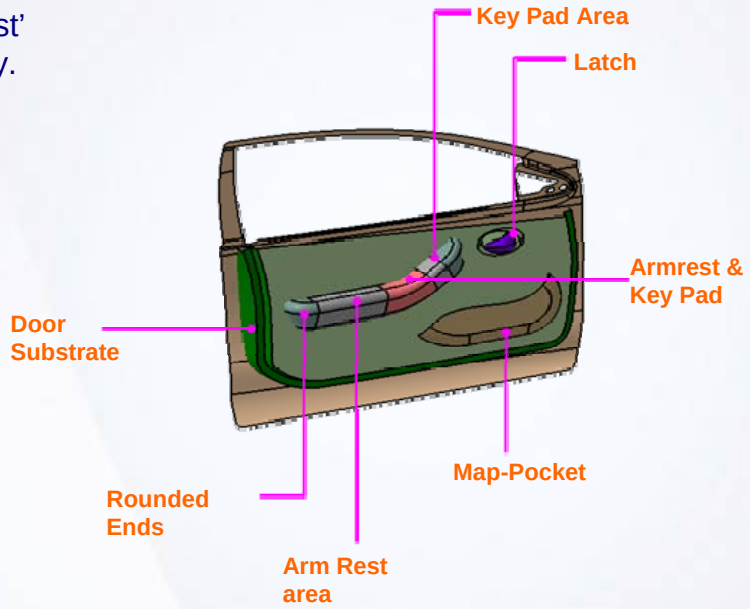
2.5 hours

Case Study

The case study for this lesson is to create surfaces for design feasibility study of the given components.

Design Intent

- ✓ The substrate profile needs to be adaptable for design modification and changes without replacing the original input.
- ✓ Create a broad cross-section surface for an ‘Arm rest’ attached to the front door, for design feasibility study.
- ✓ Create a cross-section surface for ‘Key-pad’ (for Electronic control) at a measured distance from the Arm rest ankle point.
- ✓ Attain a single merged part using Arm rest and the keypad component.
- ✓ Close the end of the Arm rest and Key-pad with rounded ends.
- ✓ Design the door latch.
- ✓ Design a Map-Pocket with the rounded edges.



Stages in the Process

1. Create an Adaptive Swept surface.
2. Sweep a Profile.
3. Revolve a Surface.
4. Create a Multi-Sections Surface.

Choice of Surface

You can choose a surface depending upon their characteristics and function:

- ✓ The surfaces are based on a profile and a direction or revolution axis: Extrude, Cylinder, Revolve and Sphere
- ✓ The surface is defined by a few pre-existing sections: Multi-Sections Surface
- ✓ A profile (predefined or not) is swept along a guide curve: Sweep or Adaptive Sweep (in order to manage the shape of the profile along the guide curve)
- ✓ To fill a gap: Fill
- ✓ To simulate a thickness on an existing surface: Offset

You can also choose surface with regards to the wireframe features available. The table shows wireframe required for each type of surface.

Tools \ Inputs	Profile	Direction/Axis	Guide Curve	Section	Spine
Extrude	✓	✓	✗	✗	✗
Sphere	✗	✓	✗	✗	✗
Cylinder	✗	✓	✗	✗	✗
Revolve	✓	✓	✗	✗	✗
Loft ✓	✗	✗	Optional	✓	Optional
Sweep	✓	✗	✓	✗	✓

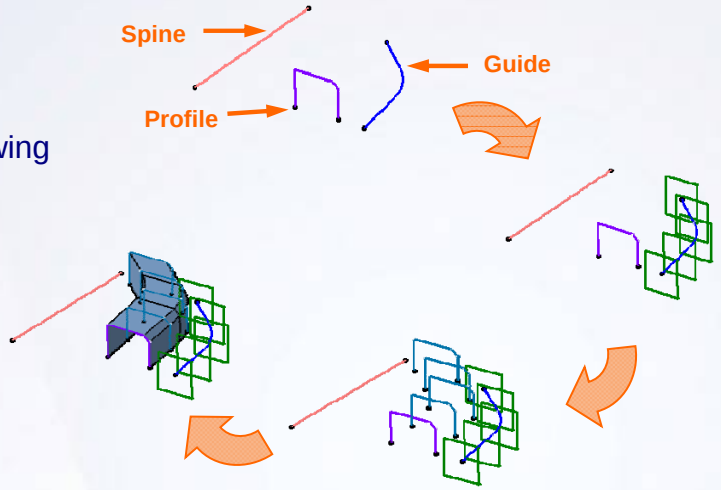
Mandatory Not Applicable Loft = Multi-Sections Surface

Sweeping a Profile

Sweep is a surface generated by sweeping a profile along a guide curve with respect to a spine. The profile can be user-defined or pre-defined. The shape and quality of the sweep depends upon the spine.

In GSD the Sweep tool can be used to sweep following profile types:

- ✓ Explicit
- ✓ Line
- ✓ Circle
- ✓ Conic



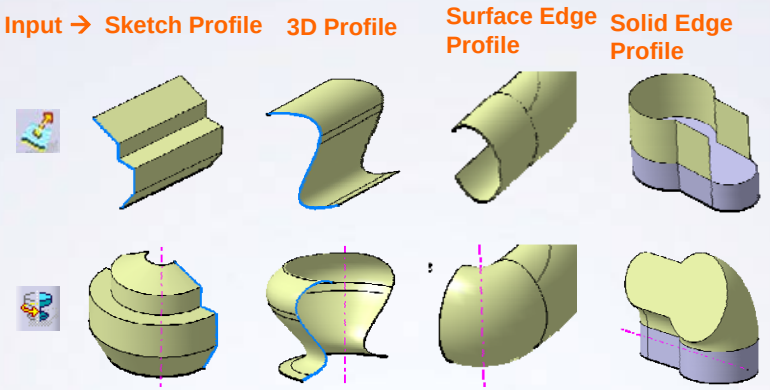
Sweeping a Profile along a Guide Curve with respect to a Spine

Extruding or Revolving a Profile

An Extrude or a Revolve tool uses a Sketch, a 3D curve, an edge of an existing surface or a solid. However self intersecting profiles or profile which intersect with the axis cannot be used. When a sketch is used to create these features, the normal plane is automatically detected (as with an axis, if included in the sketch).

The characteristics of an Extrude and the Cylinder command are similar when a profile is circular.

The characteristic of a Revolve and Sphere command are similar when a profile is circular.

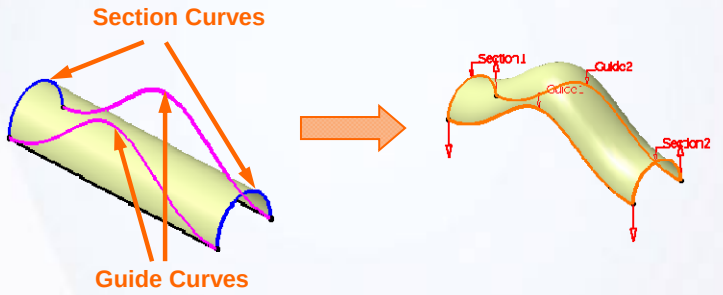


Profiles That can be Used

Create a Multi-Section Surface

A surface computed using two or more consecutive sections along a guide curve is called Multi-Section Surface. The guide curve defines the shape of the surface between two sections.

This correspondence between the sections (vertices) can also be specified using coupling points on the sections. During the surface generation, the coupling points of one section are automatically connected to the corresponding coupling points of the consecutive section to attain a guided flow between two or more sections.



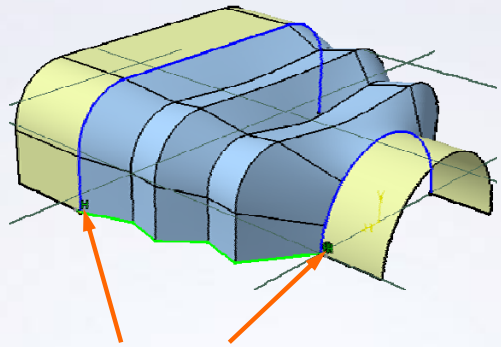
Sections and Guide Curves

Create an Adaptive Sweep Surface

An Adaptive Sweep Surface is a surface which can adapt to changing dimensions of the parent profile along the defined path.

While creating an Adaptive Sweep remember the following key points:

- ✓ The sketch of an Adaptive surface has to be a constrained sketch. A constraint is created between the guide curve and the sketch origin so that the sketch origin is always placed on the guide curve throughout the length of the sweep.
- ✓ To make the dimensions of the sketch vary dimension the sketch during its creation.
- ✓ To ensure relative positioning of the sections with reference elements (parallelism, angle, offset) constrain the sketch on the intersections of the sketch plane and the reference elements.
- ✓ To use angle constraints, rather than tangency or perpendicularity constraints.



Main Tools

Surface Toolbar

1 **Extrude:** Extrudes a user-defined profile in a specified direction.

2 **Revolve:** Revolve a user-defined profile around an axis.

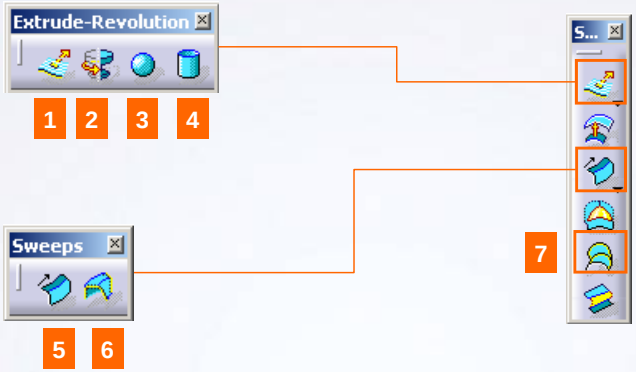
3 **Sphere:** Creates full or partial spherical surface.

4 **Cylinder:** Extrudes an implicitly circular profile in a specified direction.

5 **Sweep:** Sweeps a profile along a path.

6 **Adaptive Sweep:** Sweeps a parametric profile along a path, allowing the parameters to evolve along the path.

7 **Multi-Section Surface:** Surface passing through multiple sections.



Exercise 3B

Recap Exercise

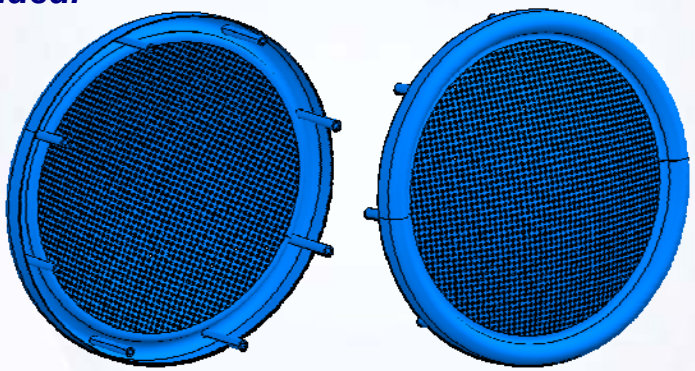
15 min

In this exercise, you will practice on the sweep sub-options to create a speaker grill of an automotive door. You will practice the sub-options of Circle and Conic type of sweep.

High level instructions for this exercise are provided.

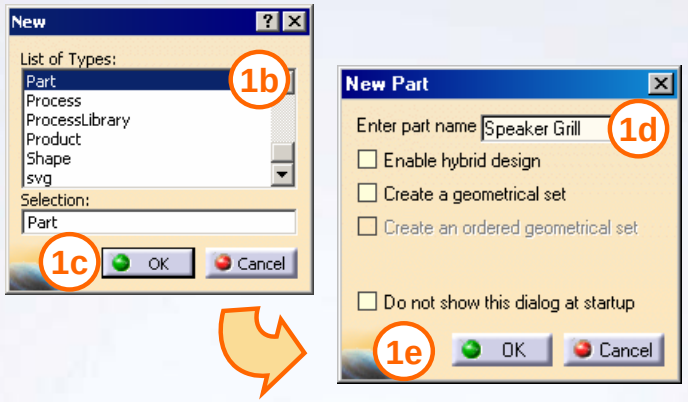
By the end of this exercise you will be able to:

- **Extrude a profile in a direction.**
- **Create a Fill surface.**
- **Create a swept surface using the Two Guides and Tangency surface available in Circle option.**
- **Create a swept surface using Two Guides available in Conic option.**

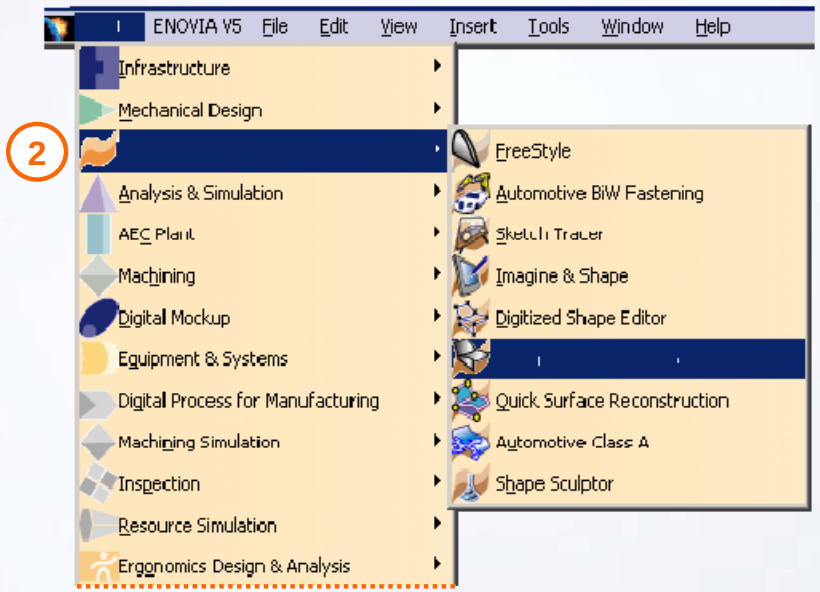


Exercise 3B (1/8)

1. **Create a new part file.**
   - To create a new part file, select Part from the New dialog box.
     a. Click **File > New.**
     b. Choose Part from the New dialog box.
     c. Click OK.
     d. Enter the new part name as – Speaker Grill
     e. Click OK.

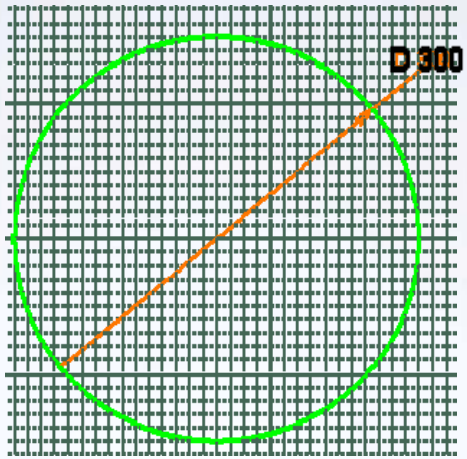


2. **Access Generative Shape Design workbench.**

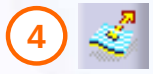


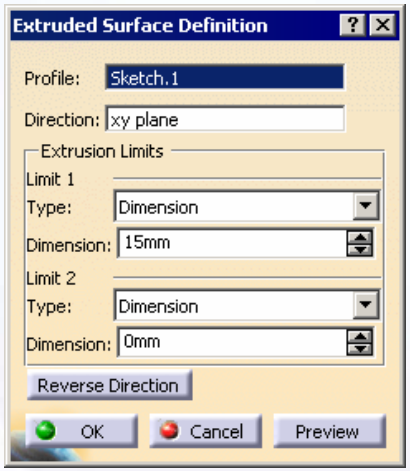
Exercise 3B (2/8)

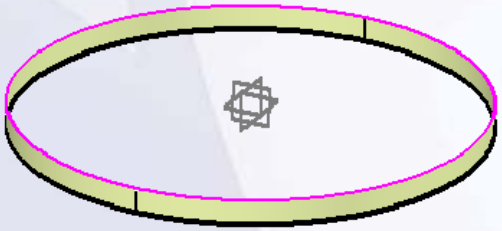
3. **Create a Sketch.**
   - Create a Circle of diameter 300mm in a sketch on XY plane with center at absolute co-ordinate system.



4. **Extrude a surface.**
   - Create an extruded surface in the –Z axis direction using the previously created sketch.

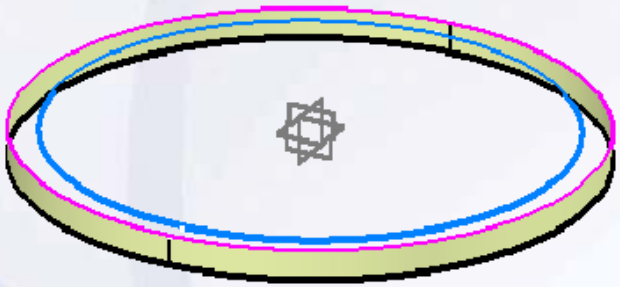


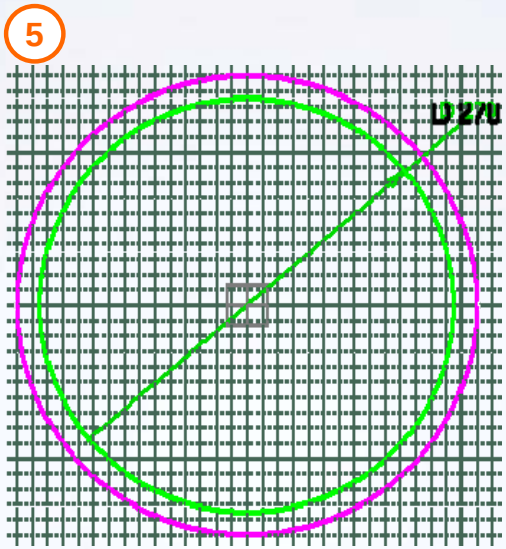




Exercise 3B (3/8)

5. **Create a Sketch.**
   - Create a circle of diameter 270mm on XY plane with center at absolute co-ordinate system.

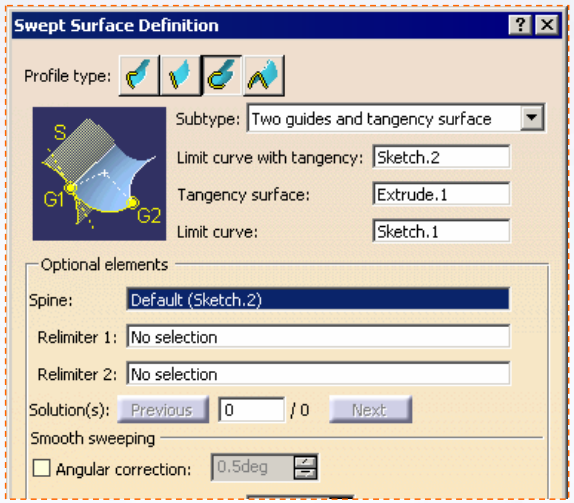




Exercise 3B (4/8)

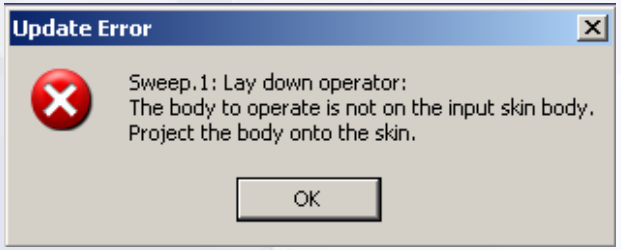
6. Create a Swept Surface

- Create a swept surface using the sub-option, Two Guide and Tangency surface, available in the Circle sweep option.



Observation:

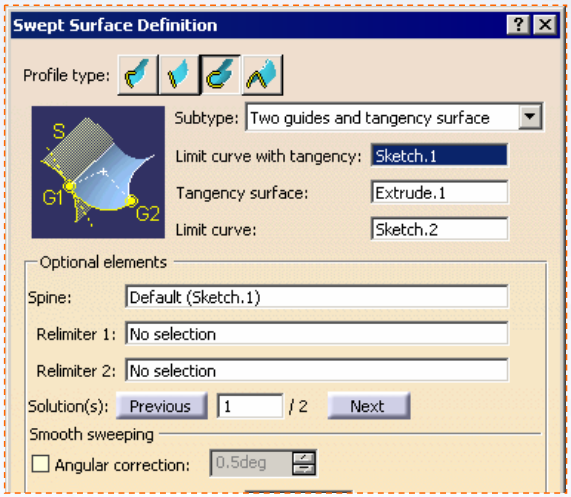
It is mandatory that the first guide curve selected should lie on the specified Tangency surface or plane. The swept surface which is generated maintains tangency with the specified surface. If not, an error is displayed.



Exercise 3B (5/8)

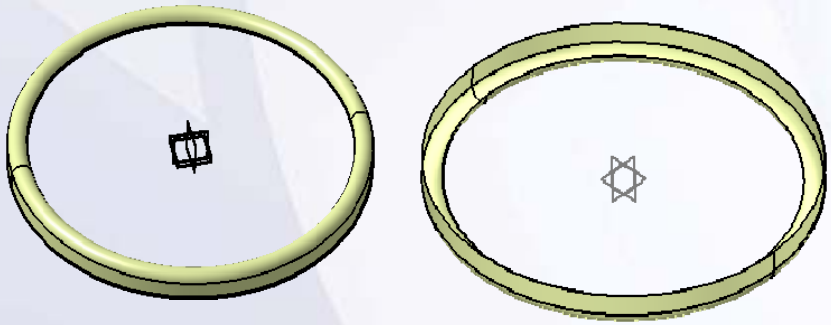
7. Re-create a Swept surface by changing the inputs.

- Create a swept surface using the sub-option, Two Guide and Tangency surface, available in the Circle Sweep option.



Observation:

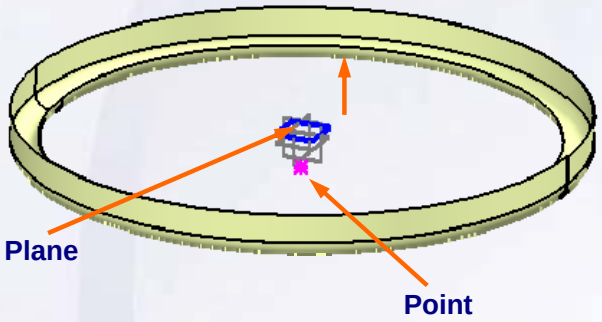
Sketch.1 lies on the specified tangency surface. The surface generated will be tangent to the surface and within the limits of the guide curve.

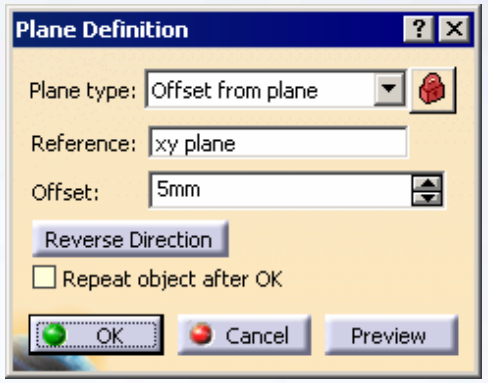


Exercise 3B (6/8)

8. Create a Plane.

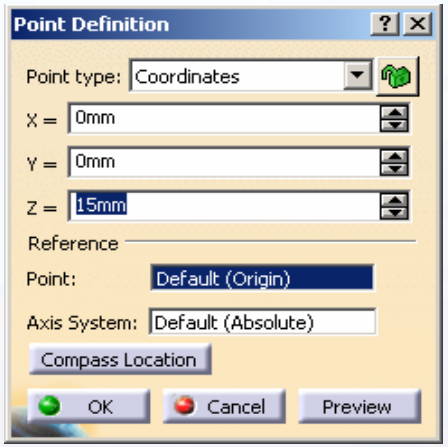
- Create a Plane parallel to XY plane at a distance of 5mm.





9. Create a Point.

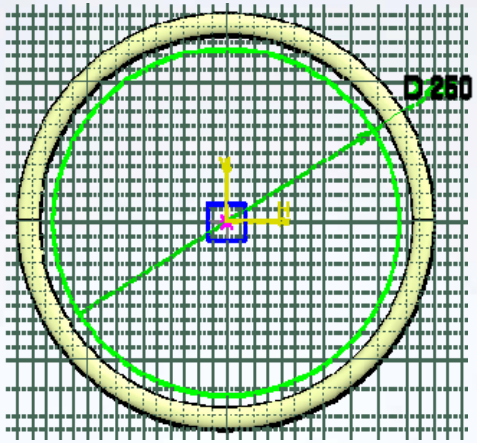
- Create a Point at a distance of 15mm along Z axis from the absolute coordinate system.



Exercise 3B (7/8)

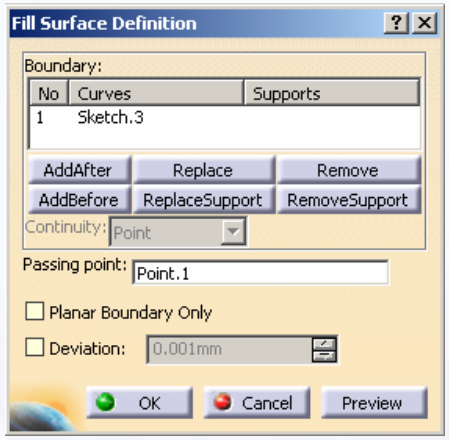
10. Create a Sketch.

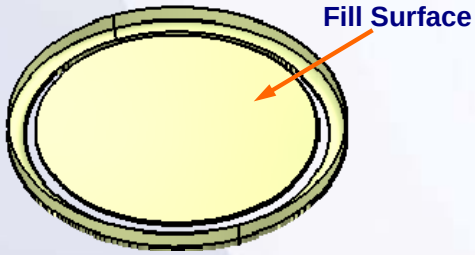
- Create a Sketch on Plane 1.



11. Create a Fill Surface.

- Create a fill surface using Sketch.3 passing through Point.1

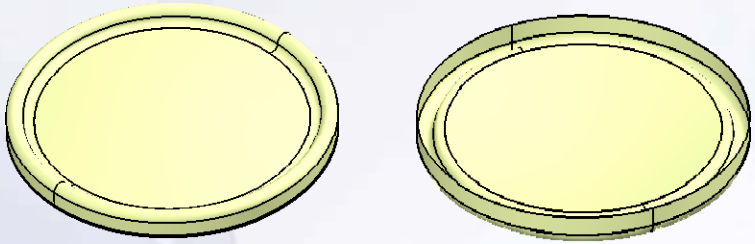


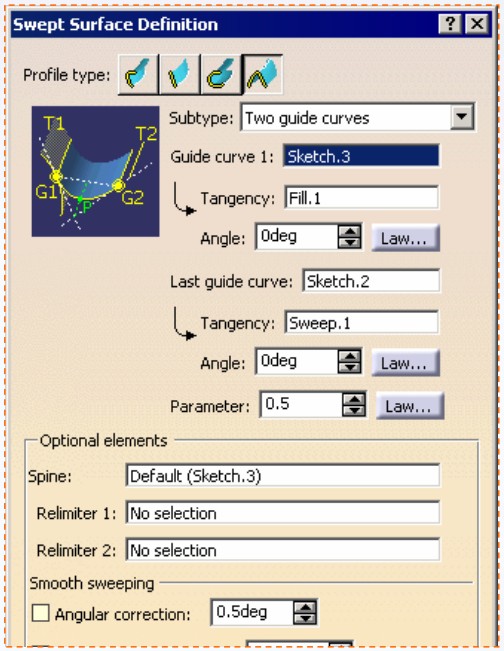


Exercise 3B (8/8)

12. Create a Swept surface

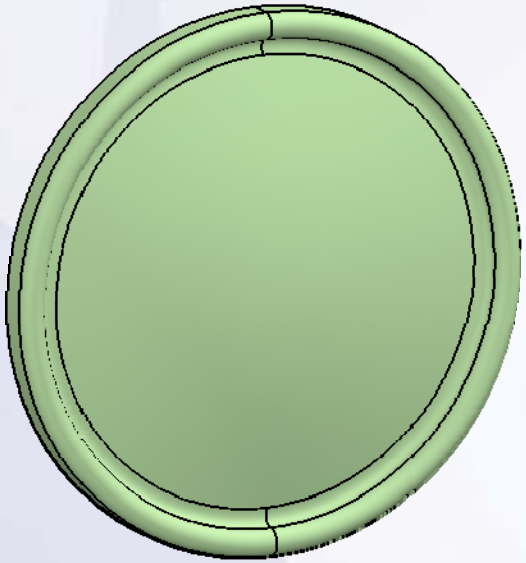
- Create a swept surface using the sub-option, Two Guide curves, available in Conic sweep option.

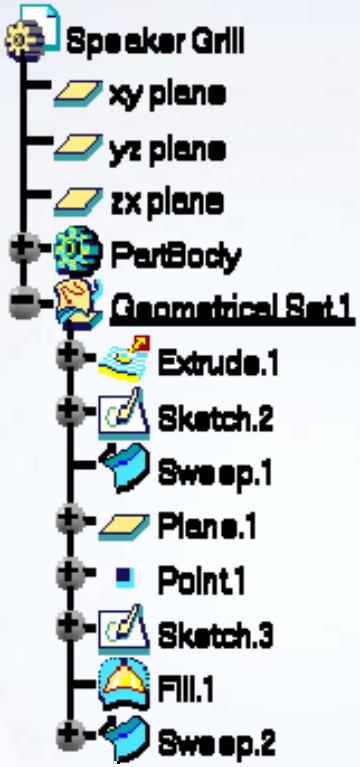




Exercise 3B: Recap

- ✓ Extrude a profile in a direction.
- ✓ Create a Fill surface.
- ✓ Create a swept surface using the Two Guides and Tangency surface sub-type available in the Circle option.
- ✓ Create a swept surface using Two Guides available in the Conic option.





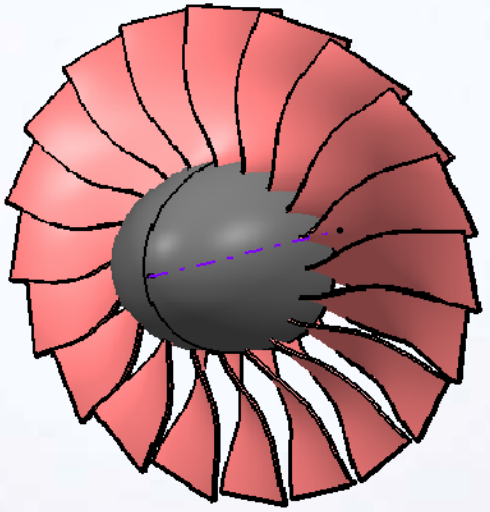
Exercise 3C

Recap Exercise

15 min

In this exercise, you will practice how to create swept surface using a law. You will design a base surface of turbine blade. You will be provided with the basic curves. High level instructions for this exercise are provided.

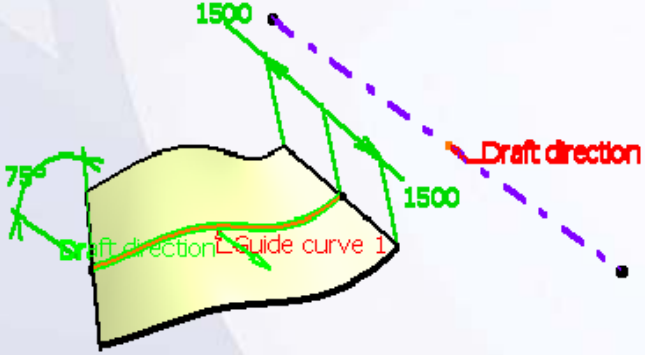
By the end of this exercise you will be able to create a swept surface using a law.

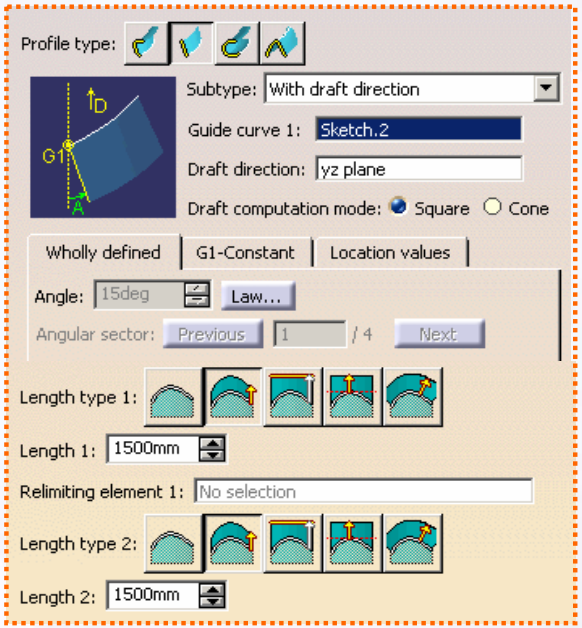


Exercise 3C (1/2)

1. **Open the part file.**
   - Open an existing part file. The file consists of Section curves for blade model.
     a. Browse and open part:
     Exercise_3C_Start.CATPart

2. **Create a Line Sweep.**
   - Create a blade surface using the Line sweep option.
   - Specify the given curve as guide and YZ plane for draft direction.
   - The orientation of the blade changes along the guide curve at a constant angle. To attain this specify a linear law for angular dimension. The angle varies from 15 deg to 75 deg.

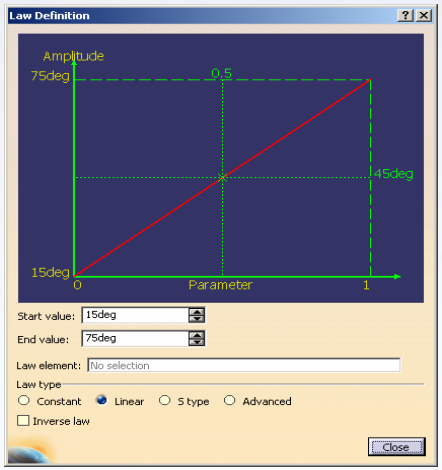




Exercise 3C (2/2)

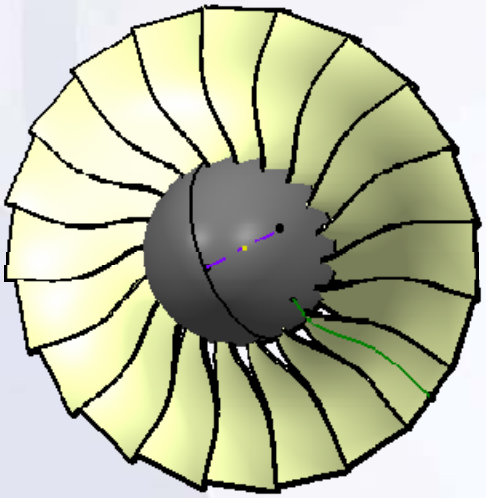
3. Define a Law

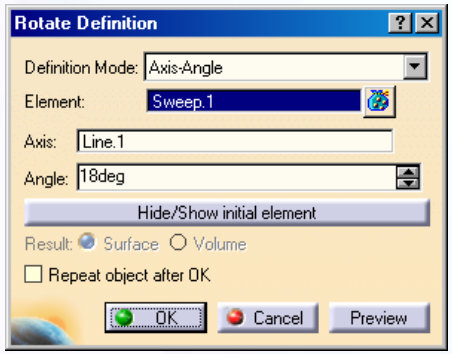
- Use the law type as Linear.
- Specify the start angle as 15 deg and end angle as 75 deg.

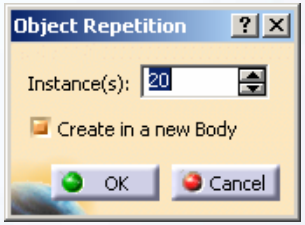


4. Rotate the Surface

- Rotate a surface along the given axis line at an angle of 18 deg.
- Specify the object instance to 20 numbers.

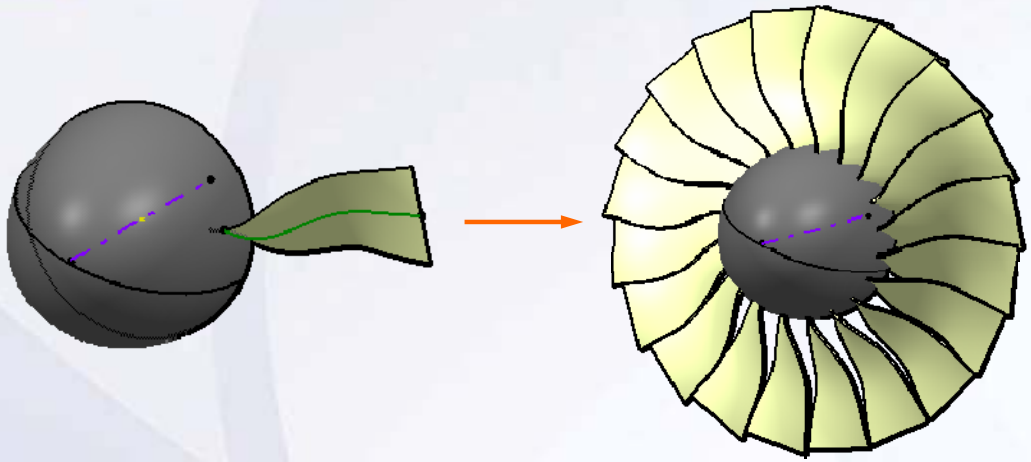


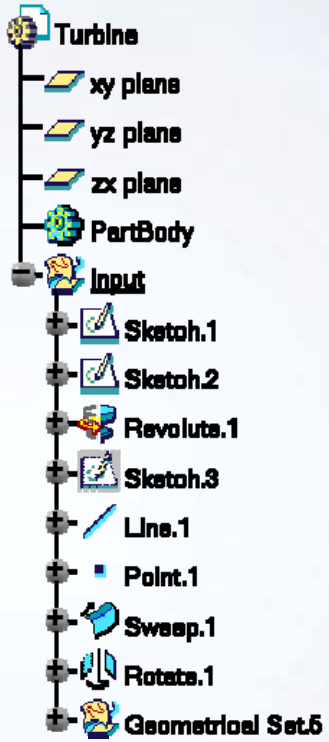




Exercise 3C: Recap

✓ Create a sweep using a law.





Exercise 3D

Recap Exercise

15 min

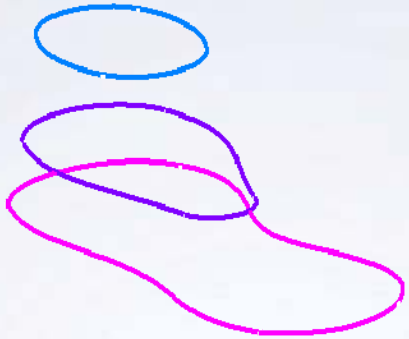
In this exercise, you will practice how to create a Multi-Section surface. You will be given a set of section curves of a shoe. You will create model of a shoe. You will also understand the different coupling options of Multi-Section surfaces. High-level instructions for this exercise are provided.

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

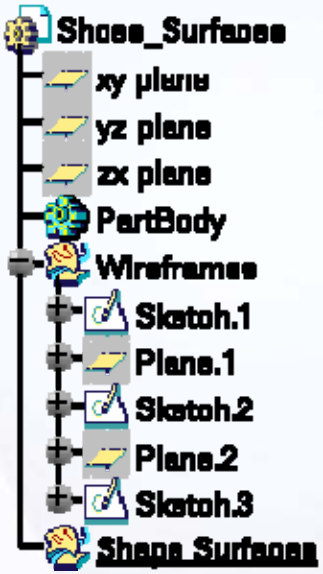
- **Create a Multi-Section surface.**
- **Use different coupling options.**

Exercise 3D (1/5)

1. **Open the part.**
   - Open an existing part file. The file consists of Section curves for creating the model of a shoe.
     a. Browse and open part: Exercise_3D_Start.CATPart



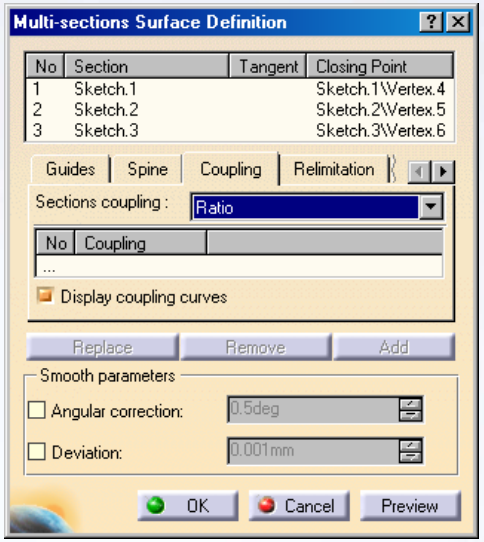
2. **Create a Geometrical Set.**
   - Insert a Geometrical set and rename it as Shape Surfaces.



Exercise 3D (2/5)

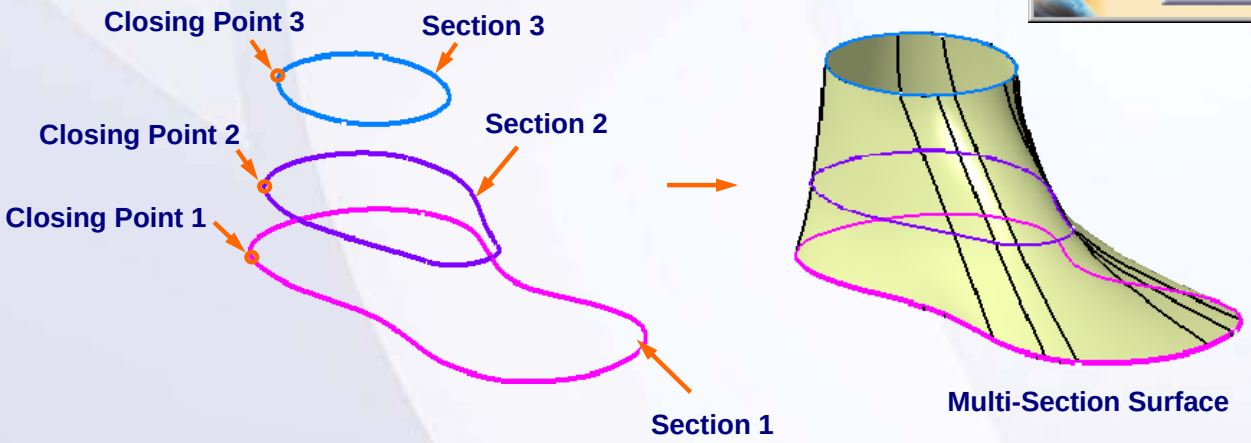
3. Create a Multi-Section Surface.

- Create a Multi-Section surface using the three closed sections without guide curve.



Observation:

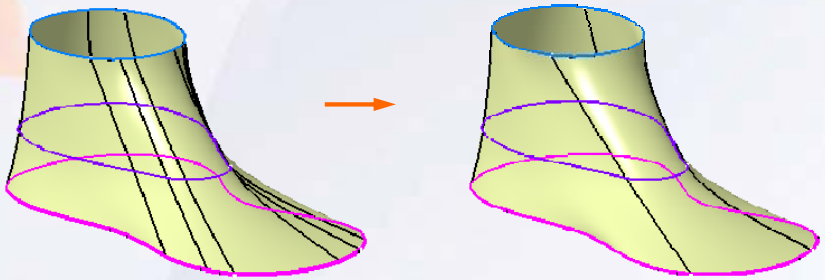
The results obtained are computed using the default mode of coupling (Ratio).



Exercise 3D (3/5)

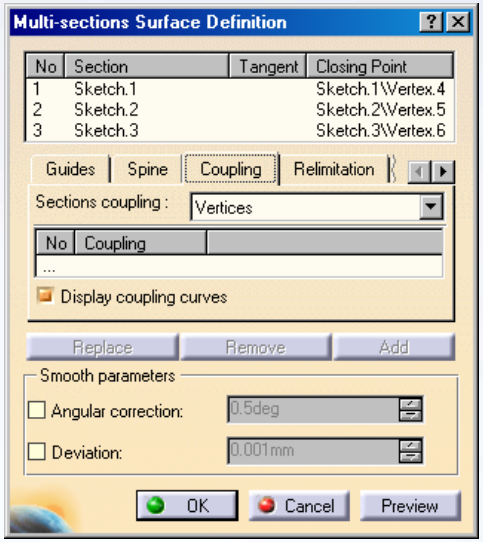
4. Change the Coupling Mode to Vertices.

- The segmentation of the surface is changed.



Surface Computed using Ratio Coupling

Surface Computed using Vertices Coupling



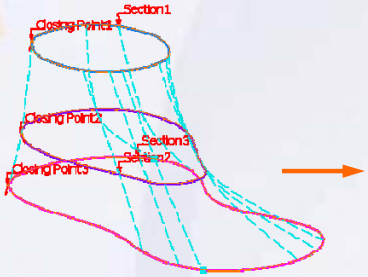
Observation:

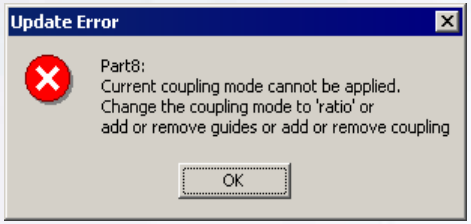
You will observe that the segmentation of the surfaces changes. CATIA couples the surface with corresponding vertices of different sections.

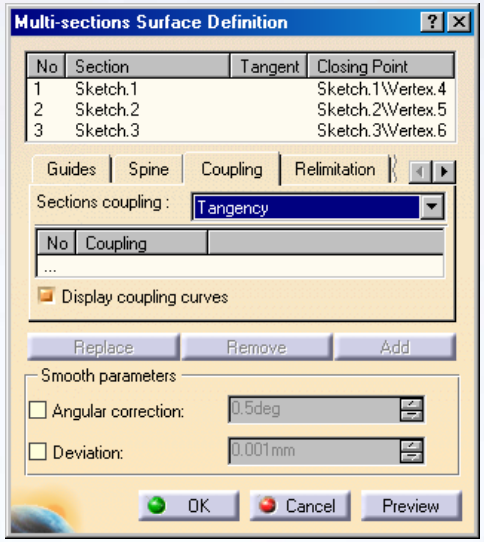
Exercise 3D (4/5)

5. Change the Coupling Mode to Tangency.

- A message is shown that the tangency option of coupling is not feasible to generate the surface, as the coupling points on each section are unequal.







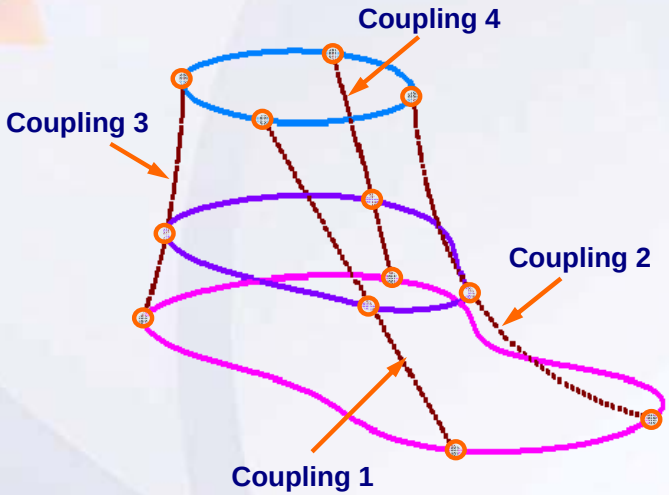
Observation:

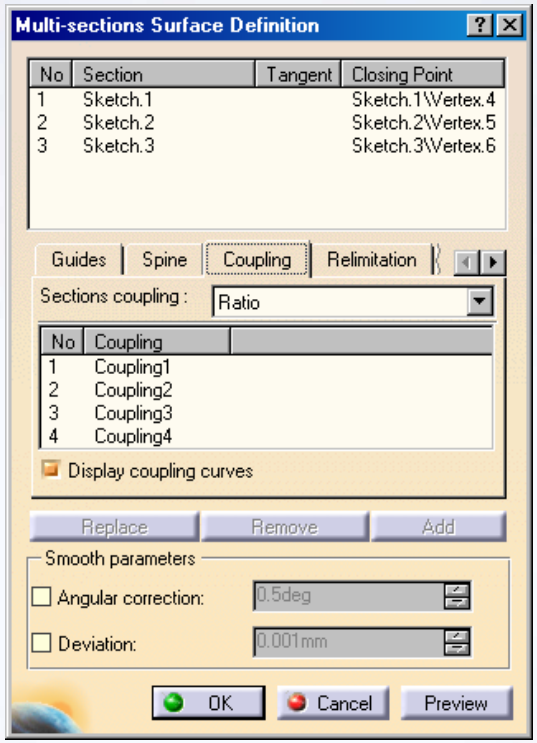
You will observe that the tangency discontinuous points which act as coupling points on each section are unequal.

Exercise 3D (5/5)

6. Couple the sections manually.

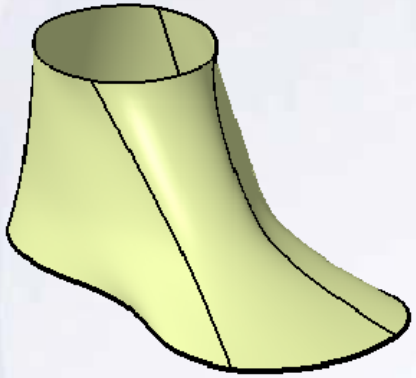
- You will create the coupling between three sections manually to obtain the desired segmentation on the surface.

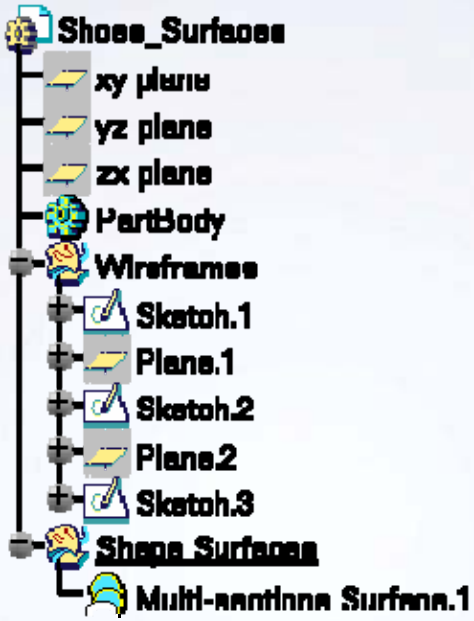




Exercise 3D: Recap

- ✓ Create a Multi-Section surface.
- ✓ Use different coupling options.





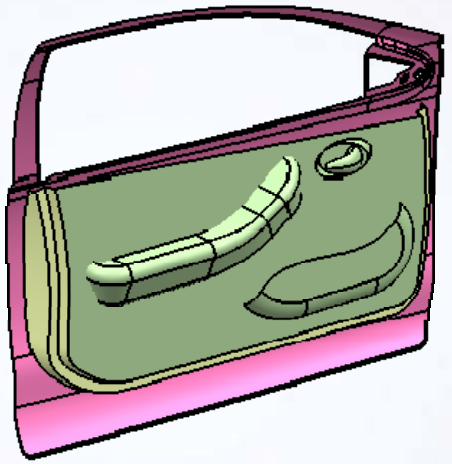
Case Study: Surface Creation

Recap Exercise

30 min

In this exercise you will create the Door inner components.
Recall the design intent of this model:

- ✓ Create a Door Substrate. The substrate profile needs to be adaptable for design modifications and changes without replacing the original input.
- ✓ Create a broad cross-section surface for an ‘Arm rest’ attached to the front door for design feasibility study.
- ✓ Create a cross-section surface for ‘Key-pad’ (for Electronic control) at a measured distance from the Arm rest ankle point.
- ✓ Create a single merged part by using Arm rest and the key pad components.
- ✓ Close the ends of the Arm rest and Key-pad with rounded ends.
- ✓ Design the door latch.
- ✓ Design a Map-Pocket with the rounded edges.



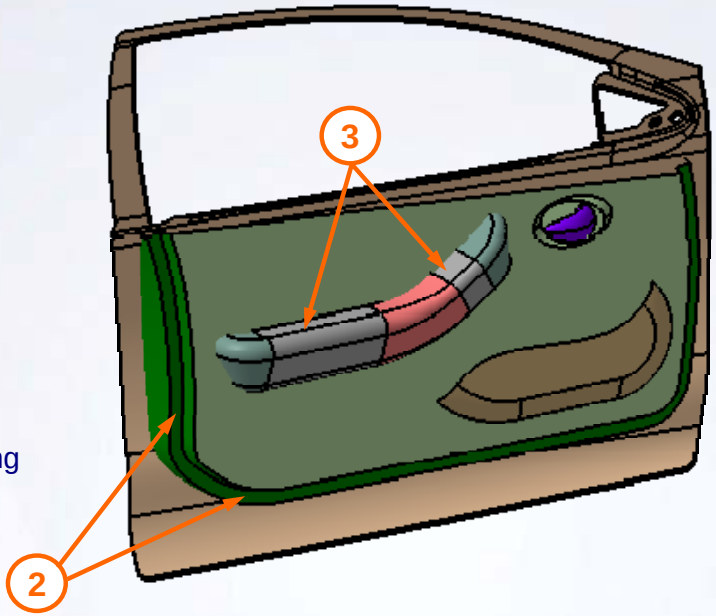
Using the techniques mentioned in this lesson and tips from the previous exercises, create the model without detailed instructions.

Do It Yourself: Surface Creation (1/3)

The following steps offer hints to guide you through the creation of door part surfaces.

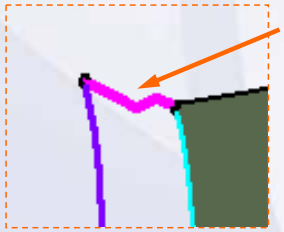
1. **Open the given part consisting of the wireframes of 'Car Door' model, in Generative Shape Design workbench.**

   Browse through the files and open the model Lesson_3_Case_Study_Start.CATPart



2. **Create an Adaptive Sweep surface.**

   Create a door substrate from the guides provided using an Adaptive swept surface.



3. **Create a Sweep surface.**

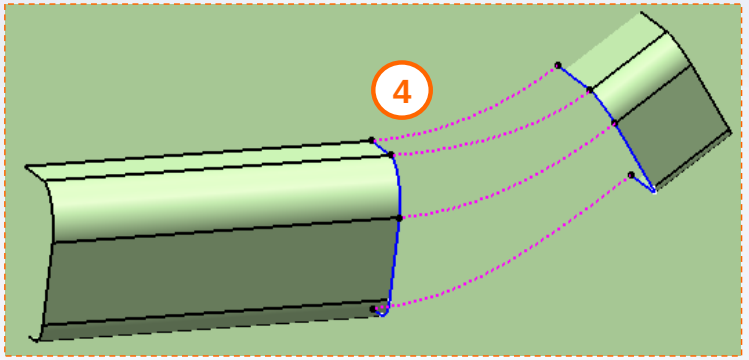
   Create a swept surface at the Arm rest and Key pad areas using the Profile and Guide Curve.

Do It Yourself: Surface Creation (2/3)

4. **Create a single merged part by using Arm rest and the key pad component.**

   Create a Multi-Section Surface between Arm Rest surface and Key Pad surface.

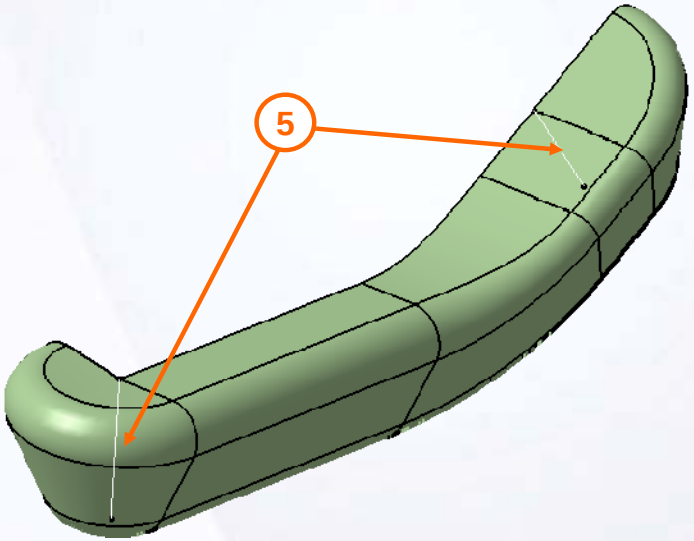
   Extract the boundary of the swept surfaces and build a Multi-Section surface between the two boundaries as shown.



5. **Close the end of the Arm rest and Key-pad with rounded ends.**

   Revolve the boundaries of the swept surface to attain the rounded ends as shown.

   Extract the boundary of the swept surface and create an axis line to revolve the surface.

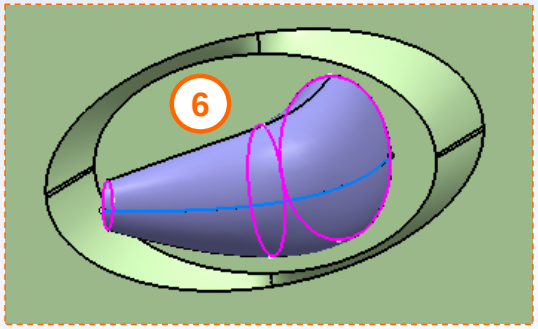


Do It Yourself: Surface Creation (3/3)

6. Create a 'Door latch'.

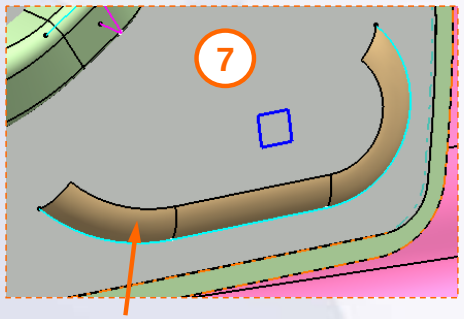
Create a Multi-Section surface using the sections and guide provided.

Allow the deviation of 0.01mm in smoothing parameters.

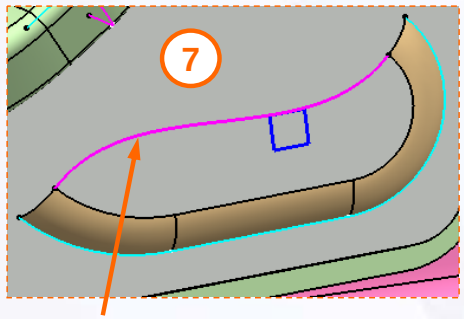


7. Create a 'Map Pocket feature'.

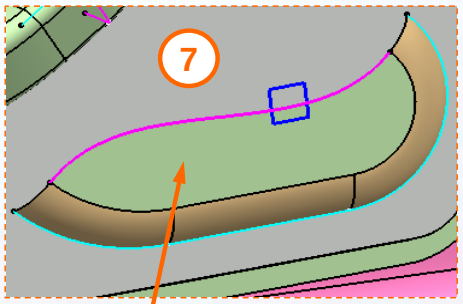
Using the circle sweep option create lower half of the pocket. This surface should be tangent to the blue plane along the Guide curve. Connect the edges of the resultant surface and fill the side part of the pocket.



Use the shown plane to create the circle sweep of radius 50mm



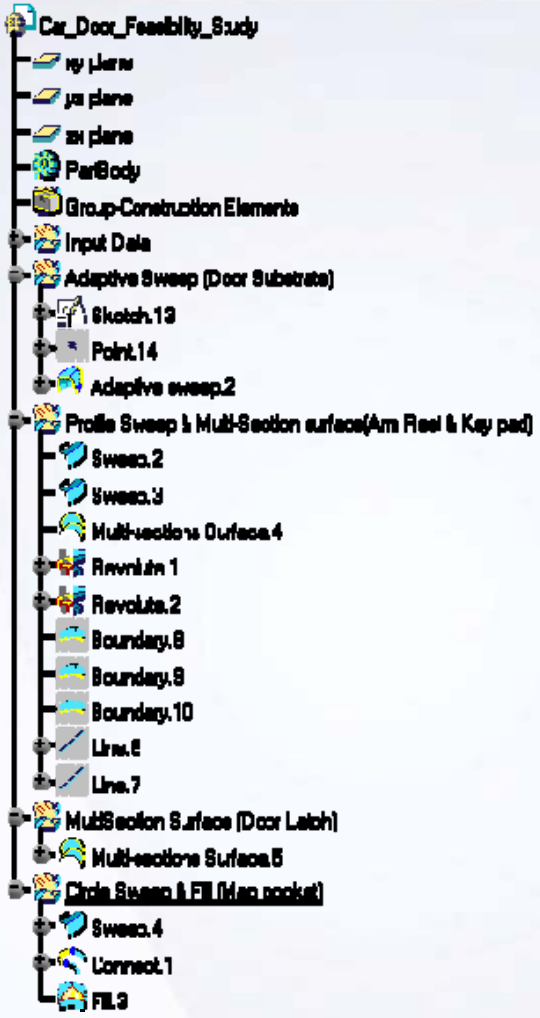
Connect the edges of the resultant surface



Fill the side part of the pocket

Case Study: Surface Creation Recap

- ✓ Create a Door Substrate. The substrate profile needs to be adaptable for design modification and changes without replacing the original input.
- ✓ Create a broad cross-section surface for an 'Arm rest' attached to the front door for design feasibility study.
- ✓ Create a cross-section surface for 'Key-pad' (for Electronic control) at a measured distance from the Arm rest ankle point.
- ✓ Create a single merged part by using Arm rest and the key pad component.
- ✓ Close the end of the Arm rest and Key-pad with rounded end.
- ✓ Design the door latch.
- ✓ Design a Map-Pocket with the rounded edges.



Surface Re-limitation and Connection 4

Learning Objectives:

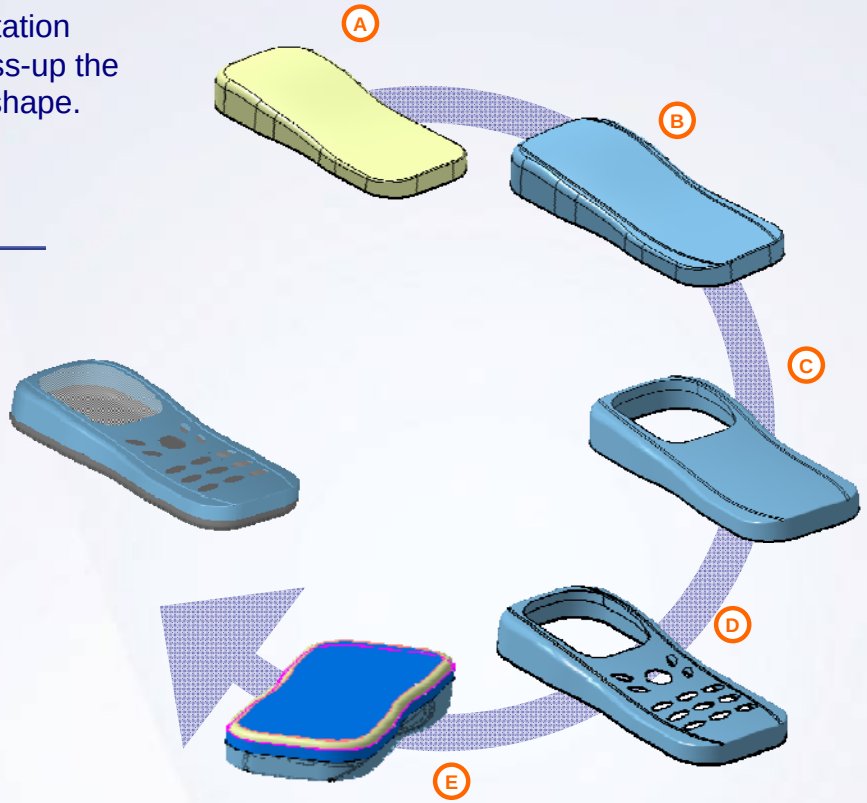
Upon completion of this lesson you will able to:

- ✓ Re-limit the Surfaces
- ✓ Connect the Surfaces Smoothly
- ✓ Assemble the Surfaces

4.5 hours

Case Study

The case study for this lesson is to use the re-limitation and connection tools of GSD workbench, and dress-up the Mobile Phone model to achieve the final finished shape.



Design Intent

A. Create a smooth blended edge at the top edge which varies with respect to thickness of the phone.
B. Create a feature groove on the top face of the phone to beautify it or to make it look attractive.
C. Create a display screen with the flange at inside to mount the display card.
D. Create key pad holes on the top face of the phone to incorporate number keys.
E. Create a smooth blended edge on the lower case of the phone.

Stages in the Process

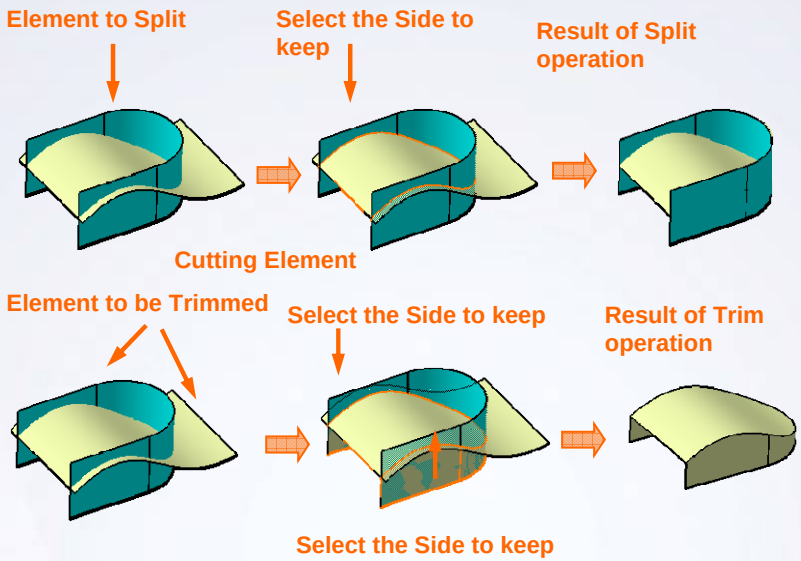
1. Access the Generative Shape Design workbench.
2. Scan the model for a better understanding of the modeling sequence.
3. Create Geometrical Sets.
4. Group the features.

Re-limit the Surfaces

Operations such as Split and Trim help to convert raw surfaces and construction elements into finished geometry. Elements involved in an operation are kept in the history, but are hidden.

The image shows the difference between Split and Trim. Splitting is breaking all the geometries at the intersection with the cutting element and then removing the unwanted portion. During splitting, cutting element does not get affected.

Trimming is cutting all the geometries with respect to one another to get the required shape.



Connect the Surfaces Smoothly

You can connect two existing surfaces smoothly using following ways:

- ✓ Radius Driven connections - Fillets
- ✓ Tension Driven connections - Blend

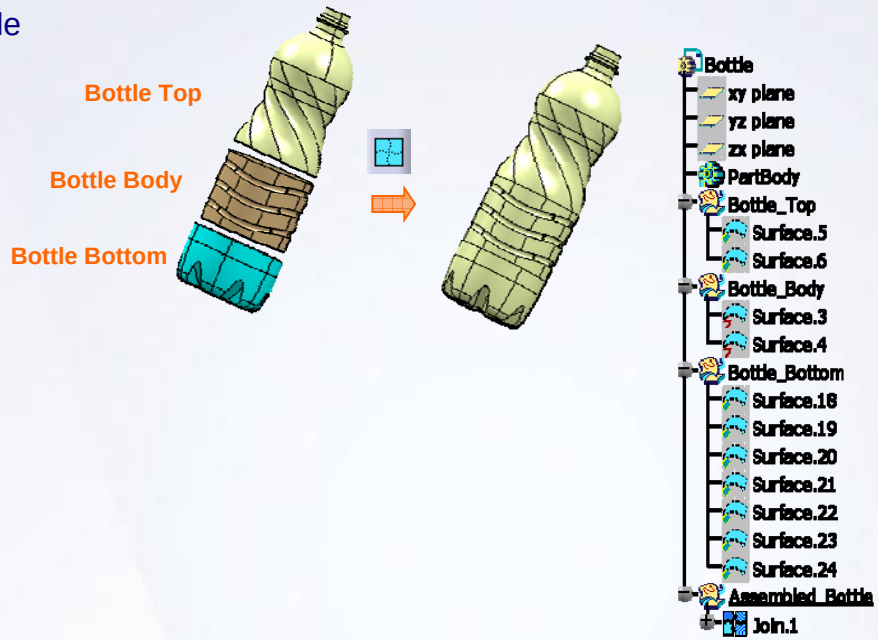
The choice of the blend depends upon the functional requirement and aesthetics of the part. Fillets are used for mechanically connecting surfaces with tangent continuity. Blend are used to create a curvature (G2) connection between the surfaces and obtain a more aesthetic result.

Assemble the Surfaces

Join operation is used to concatenate or logically group adjacent surfaces / wireframes into a single element that can be used for future operations.

The merging distance is given by a threshold value (similar to smooth tool) this makes Join a tolerant modeling tool (allows the creation of non-continuous features that can be accepted in certain cases).

Similarly some other tools which offer the possibility to assemble features on the fly are Connect Curve, Corner, Trim, Fillet, Blend and Extrapolate.



Main Tools

Operations Toolbar

1 **Split:** Splits one surface with other surface. Only one surface gets affected.

2 **Trim:** Trims surfaces involved in the operation with respect to one another.

3 **Extrapolate:** Extends surface boundaries with required continuity.

4 **Fillet:** Mechanically connects surfaces with tangent continuity.

5 **Join:** Logically groups adjacent surfaces or wireframe.

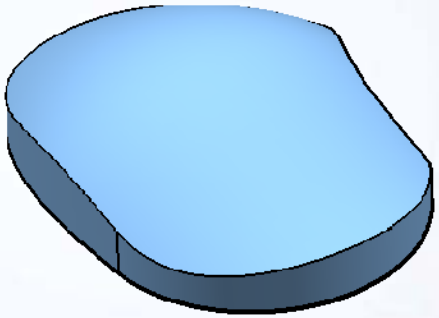
Exercise 4B

Recap Exercise

20 min

In this exercise, you will perform Trim and Split operations on a computer mouse model. You will perform these Split and Trim operations on the same set of surfaces to understand the differences in the operation. High-level instructions for this exercise are provided.

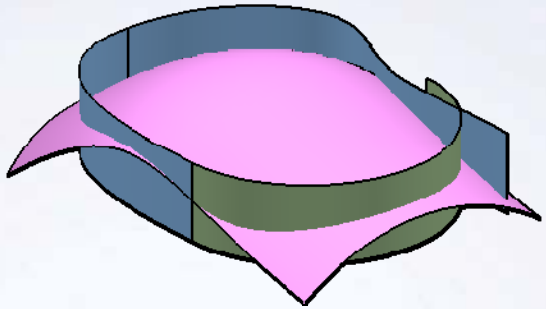
By the end of this exercise you will be able to understand the difference between Split and Trim operations.



Exercise 4B

1. Open the part.

- Open an existing part file. The file consists of surfaces of the model of the mouse. Browse and open the part: Exercise_4B_Start.CATPart



2. Split the intersecting surfaces to get the finished shape.

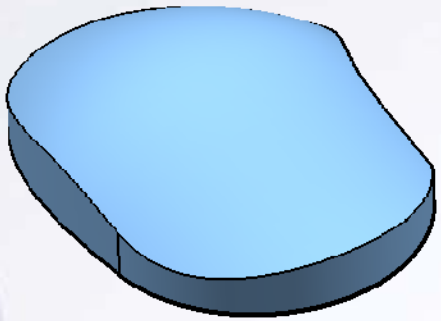
- Split the Green and Blue surfaces one by one.
- Join the resulting surface
- Split between Pink and Joint surface.

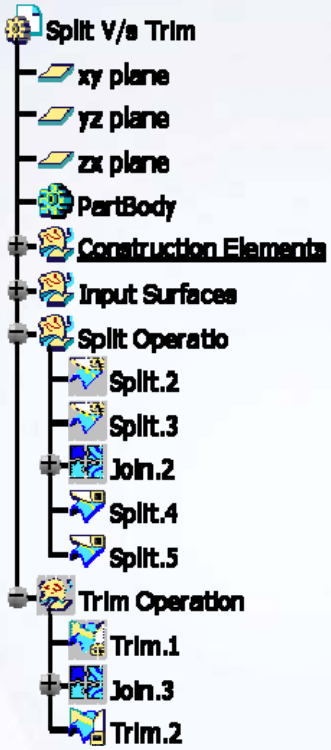
3. Perform Trim operation on the same set of parent surfaces.

- Trim between Green and Blue surfaces. The resultant surface would be a single entity.(Unhide the parent surfaces).
- Split the resulting surface with the Pink surface to get the final model.

Exercise 4B: Recap

- ✓ Split and Trim operations on surfaces and wireframe elements.





Exercise 4D

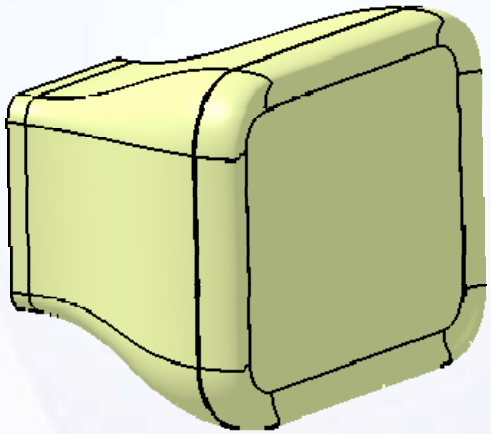
Recap Exercise

20 min

In this exercise you will practice the Surface Blend and Filleting tool. You will understand the difference between a blended surface and a fillet by analyzing the results. High-Level instructions for this exercise are provided.

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

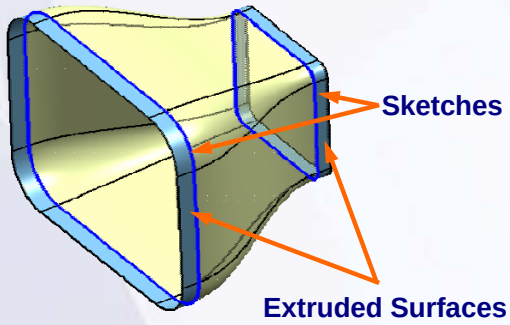
- **Create a blended surface between two existing surfaces.**
- **Create a shape fillet between two intersecting surfaces.**
- **Join two or more surfaces**
- **Split the surfaces using a curve.**

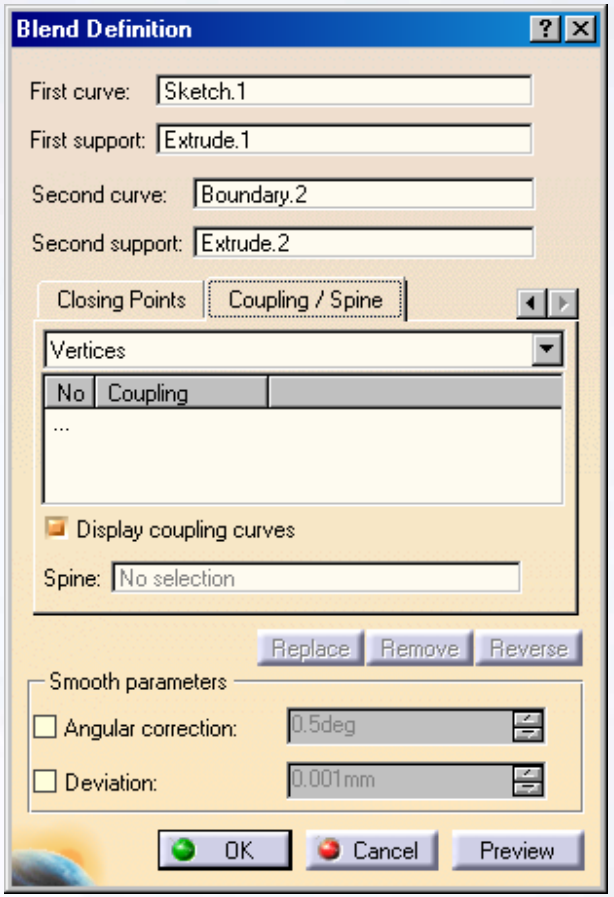


Exercise 4D (1/6)

1. **Open the part.**
   - Open an existing part file. The file consists of extruded surfaces.
     a. Browse and open part: Exercise_4D_Start.CATPart

2. **Create a Blend surface between two extruded surfaces.**
   - Create a blend using the two sketches.Use the extruded surfaces as support.

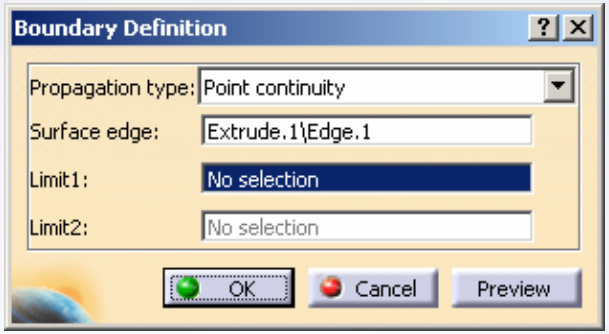


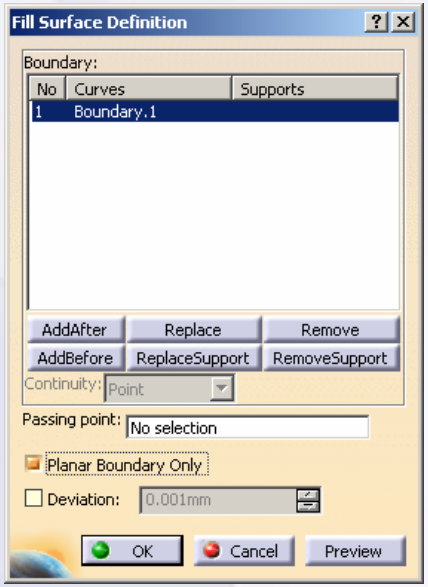


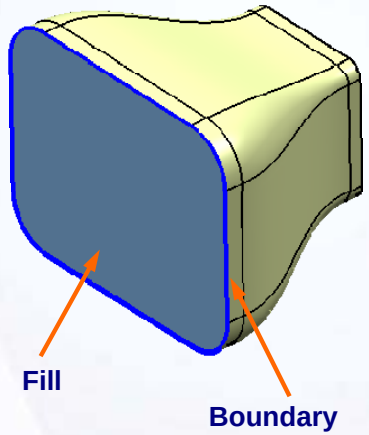
Exercise 4D (2/6)

3. **Create a boundary.**
   - Create a boundary of Extrude.1 surface.

4. **Fill the boundary.**
   - Select the Planar Boundary Only check box to fill the boundary.



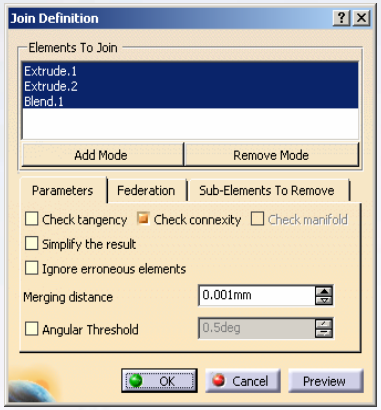


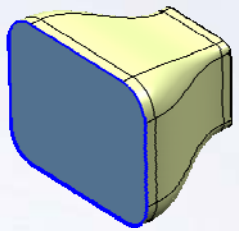


Exercise 4D (3/6)

5. Join surfaces.

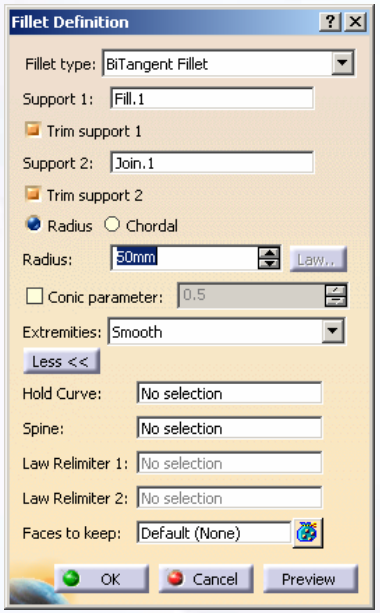
- Join blended surface and the extruded surfaces.

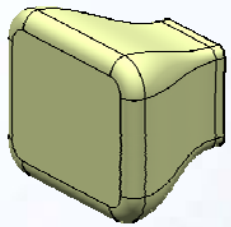




6. Create a Shape Fillet.

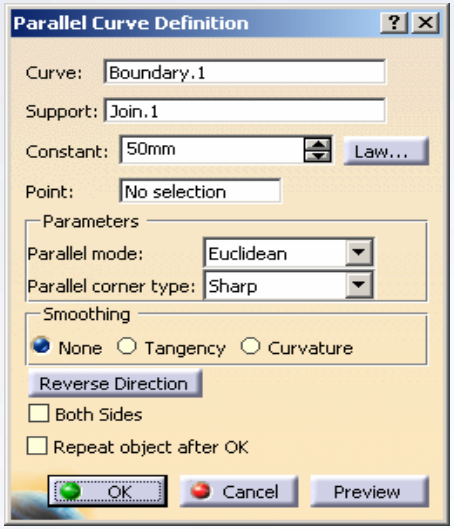
- Create a fillet between joint surface and the filled surface.

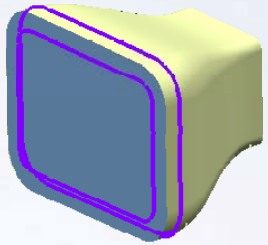




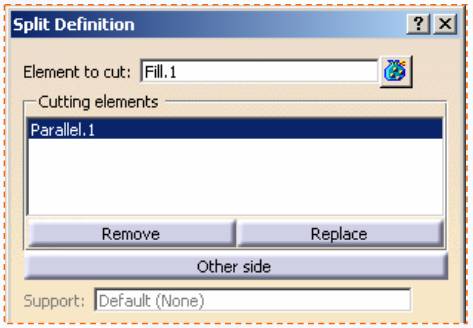
Exercise 4D (4/6)

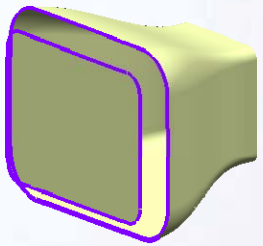
7. **Create Parallel curves.**
   - Hide this fillet and unhide the join and fill surface from the specification tree.
   - Create parallel curves on join and fill surface using a boundary curve (You will be reusing these surfaces to create Blend).





8. **Split the surfaces.**
   - Split the fill and join surface using respective parallel curves.

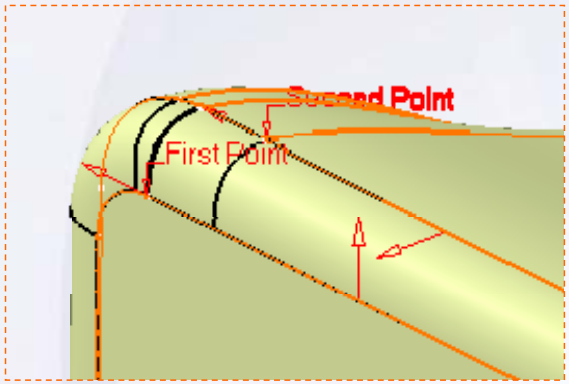


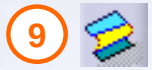


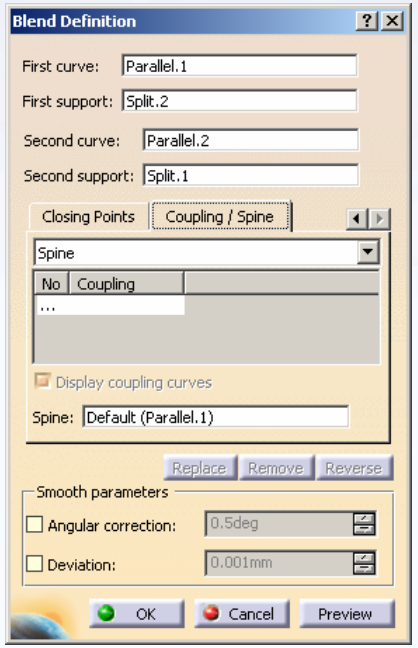
Exercise 4D (5/6)

9. Create a blend surface between two split surfaces.

- Create a blend between two split surfaces.

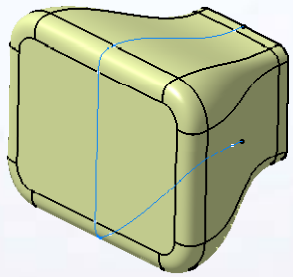






10. Create two join surfaces.

- Join all the surfaces which include filleted surface (but not the blend surface created in step 9).
- Join all the surfaces which include blend surface (but not the filleted surface created in step 6).
- Intersect the two joins created with ZX plane. (This intersection curve will be used for Porcupine analysis in the next step).

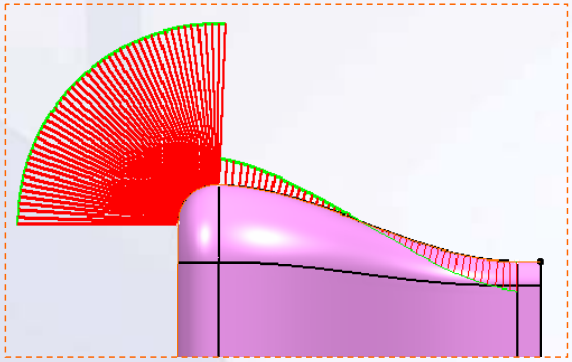


Exercise 4D (6/6)

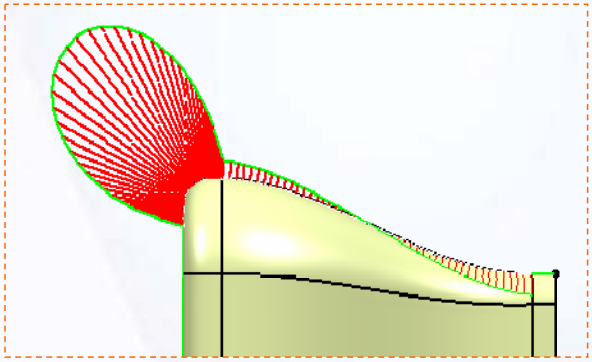
10. Porcupine analysis results of filleted surface and blend surface.

- Filleted surface
  a. Surface produced by Fillet tool is tangent continuous surface.
  b. The shape of the fillet is invariable.
  c. It is a radius driven shape.

- Blend surface.
  a. Surface produced by Blend tool can be Point, Tangent or Curvature continuous surface.
  b. The shape of the surface is variable.
  c. It is a tension driven shape.



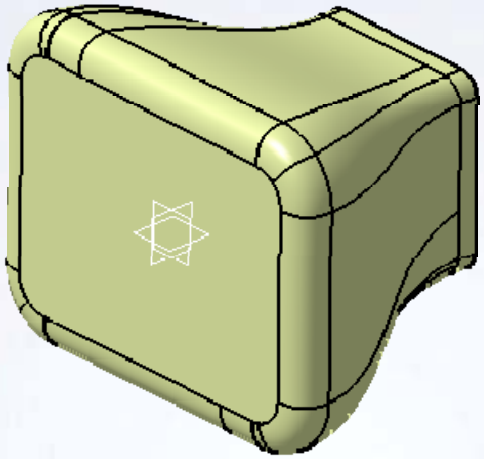
Filleted surface with tangent continuity



Blended surface with Curvature continuity

Exercise 4D: Recap

- ✓ Create a blend surface between two existing surfaces.
- ✓ Create a shape fillet between two intersecting surfaces.
- ✓ Join two or more surfaces
- ✓ Split surfaces using a curve.



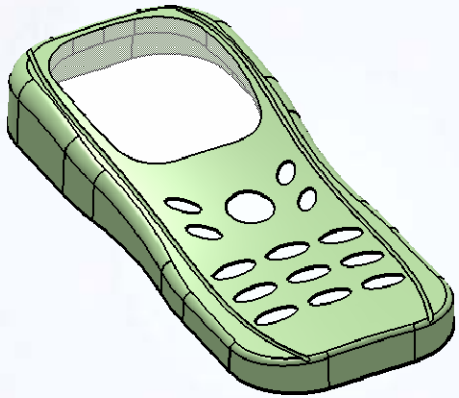
Case Study: Surface Re-limitation and Connection

Recap Exercise

30 min

In this exercise you will practice how to use the surface Re-limitation and Connection tools.

- ✓ Create a smooth blended edge at the top, this edge varies with respect to thickness of the phone.
- ✓ Create a feature groove on the top face of the phone to add aesthetics.
- ✓ Create a display screen with a flange on the inside to mount the display card.
- ✓ Create key pad holes on the upper part of the phone for number keys.
- ✓ Create a smooth blended edge on the lower case of the phone.



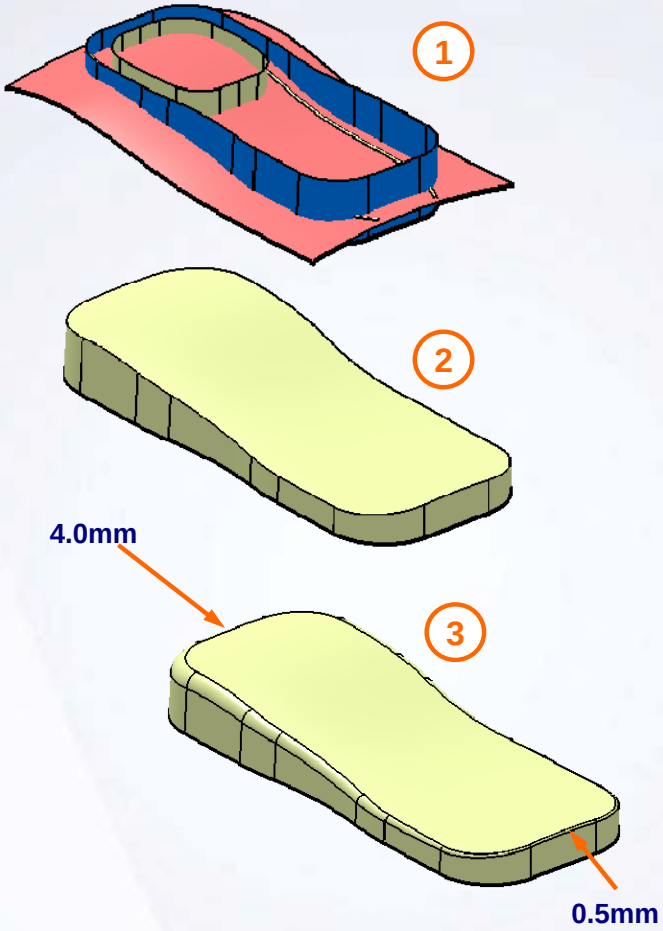
Using the techniques you have learned in this lesson, and with tips from the previous exercises, create the model without detailed instruction.

Do It Yourself: Surface Re-limitation and Connection (1/4)

1. **Open the given part consisting of base surfaces of mobile phone model in the Generative Shape Design Workbench.**
   a. Browse through the files and open the model Case_Study_Start.Catpart
   b. Study the part.

2. **Trim between the top and the side surfaces**
   a. Trim between the Pink and Blue surfaces.

3. **Create a smooth blended edge at the top with respect to thickness of the phone.**
   a. Create a Variable Fillet along the top edge as shown.
   b. The radius of the fillet should vary from 0.5mm to 4.0mm.



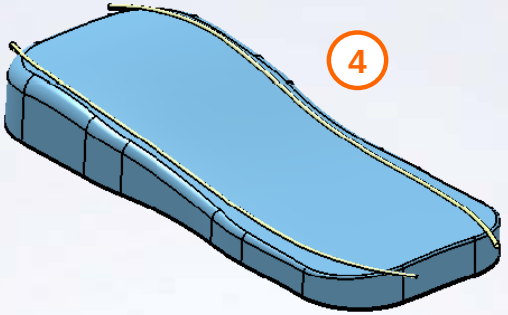
Do It Yourself: Surface Re-limitation and Connection (2/4)

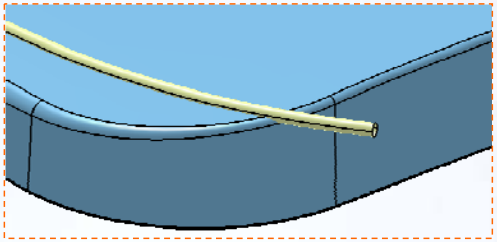
4. Create a feature groove on the top face of the phone to add aesthetics.

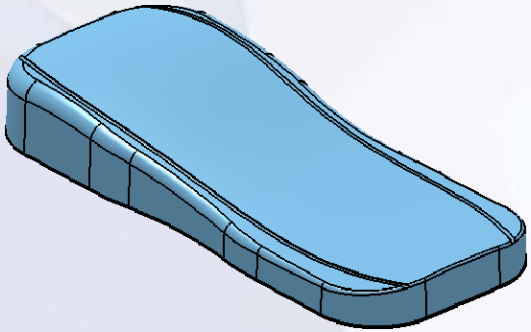
a. Trim the sweep surface from the top face as shown.

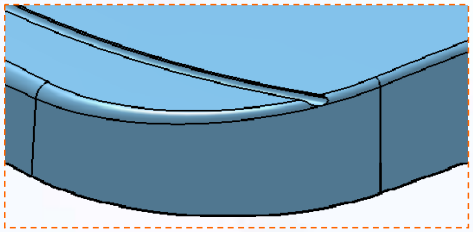
b. Keep the surface to form a groove on the main body.

c. Create the groove at both the locations specified.







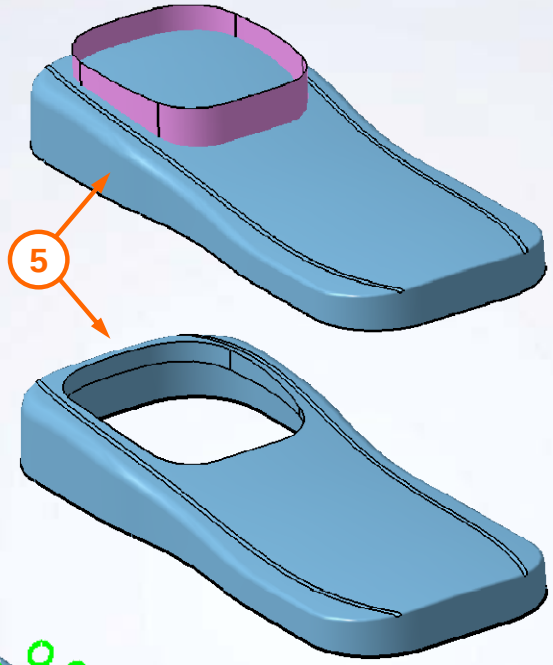


Do It Yourself: Surface Re-limitation and Connection (3/4)

5. Create a display screen with a flange inside.

a. Create a trim between the main surface and the extruded surface as shown.

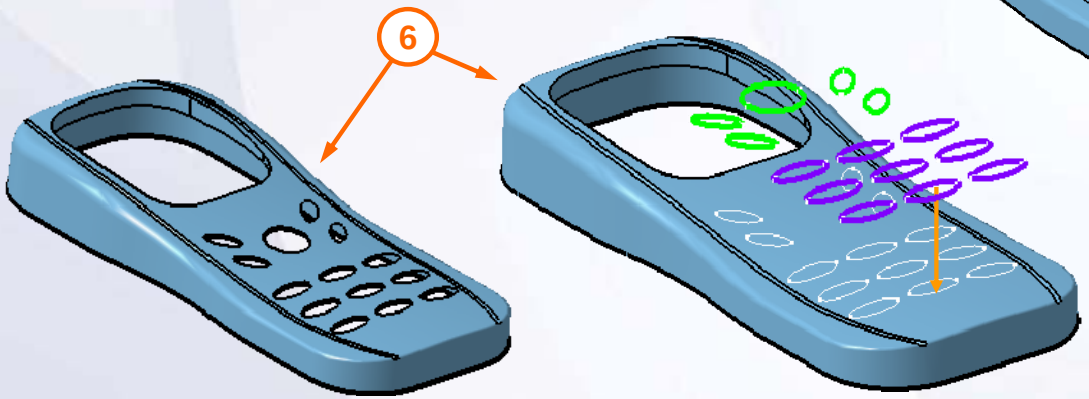
b. Keep the inner portion of the extruded surface to create a flange like feature.



6. Create a key pad holes on the upper part of the phone for number keys.

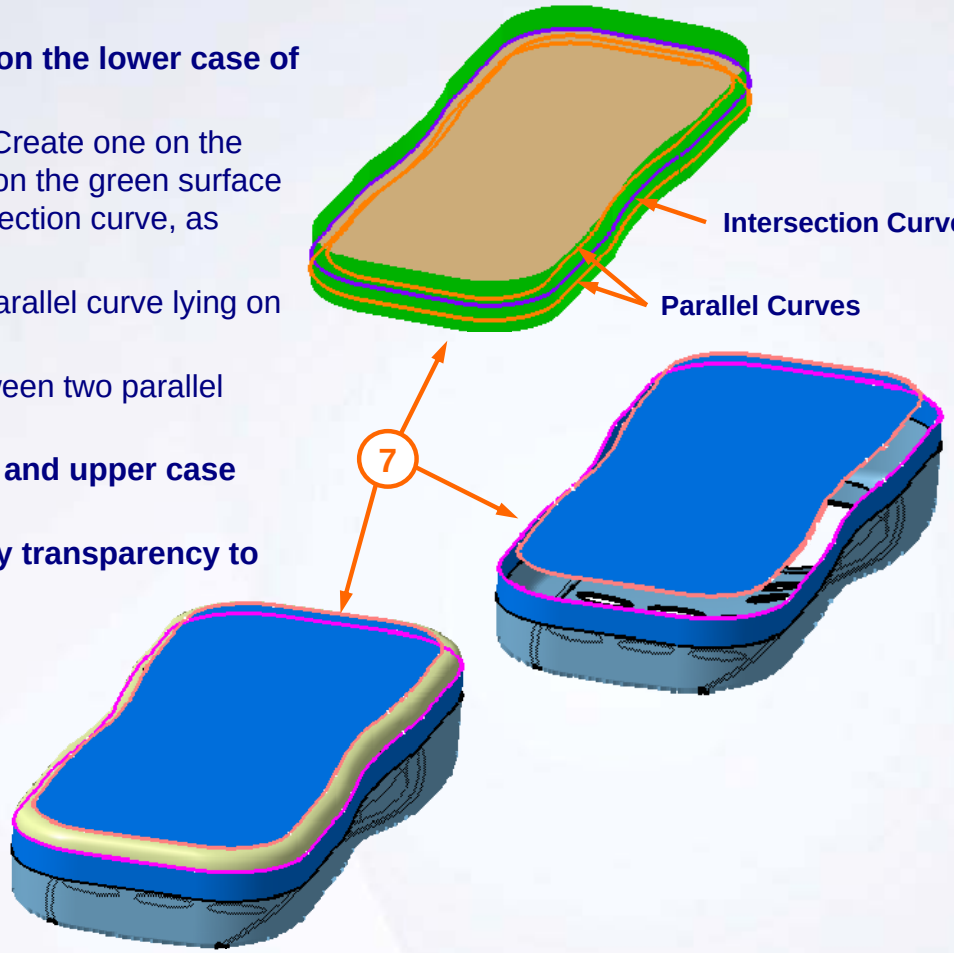
a. Project the sketch consisting of key profiles on the main surface. Each profile in a sketch is an output feature.

b. Split the main surface with a projected curve as shown.



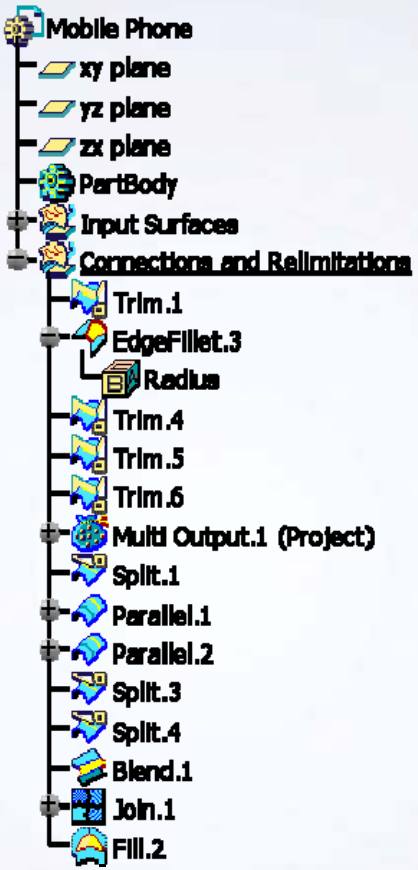
Do It Yourself: Surface Re-limitation and Connection (4/4)

7. **Create a smooth blended edge on the lower case of the phone.**
   a. Create two parallel curves. Create one on the brown surface and another on the green surface at 3mm distance from intersection curve, as shown.
   b. Split the surfaces with the parallel curve lying on them.
   c. Create a blend surface between two parallel curves.
8. **Join the surfaces of lower case and upper case separately.**
9. **Fill the display screen and apply transparency to the surface.**



Case Study: Surface Re-limitation and Connection Recap

- ✓ Create a smooth blended edge at the top, the edge varies with respect to thickness of the phone.
- ✓ Create a feature groove on the top face of the phone to add aesthetics.
- ✓ Create a display screen with a flange on the inside to mount the display card.
- ✓ Create key pad holes on the upper part of the phone for number keys.
- ✓ Create a smooth blended edge on the lower case of the phone.

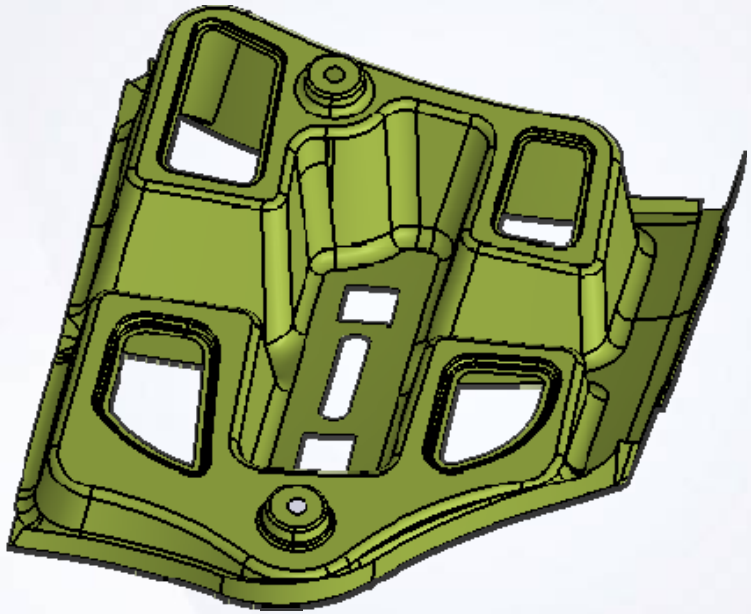


Surface Check Tools 5

Learning Objectives:

Upon completion of this lesson you will be able to:

- ✓ Check Surface Continuity
- ✓ Use Geometric Connection Tools
- ✓ Correct Defects in Surfaces
- ✓ Check Surfaces Moldability



4.5 hours

Case Study

The case study for this lesson is to analyze the geometric connections between the surfaces of a plastic part using different surface analysis tools.

Design Intent

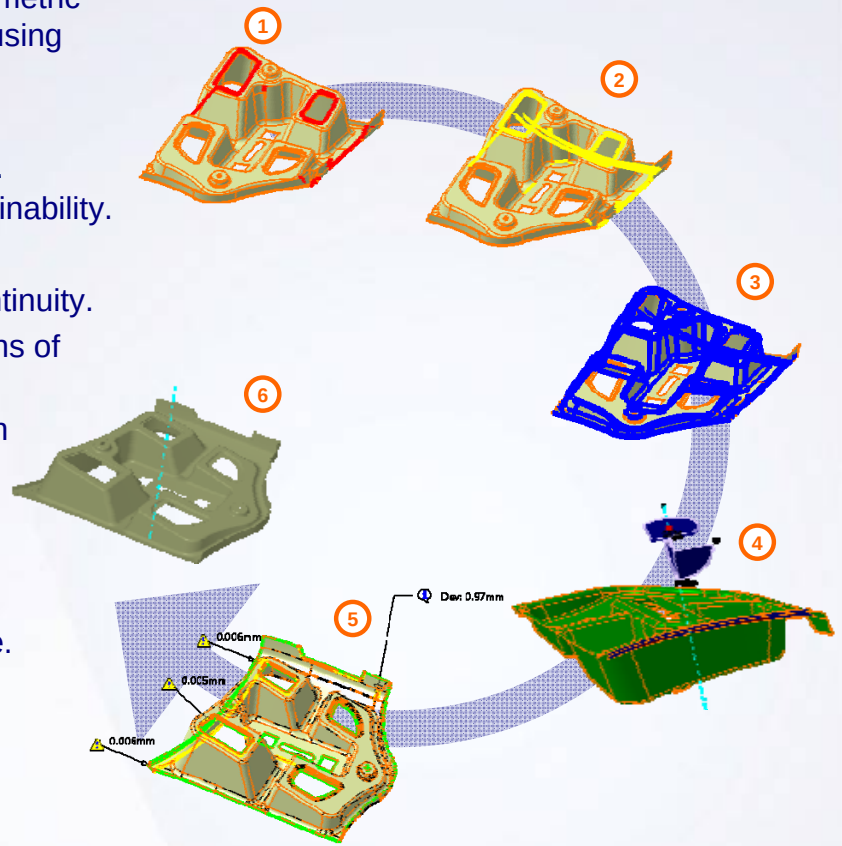
A. Check the surfaces for gaps to ensure easy machinability.
   - Use Connect Checker.

B. Check the surface for tangency and curvature continuity.
   - Use the Tangent and Curvature analysis options of Connect Checker.

C. Check the draft on the surface for better extraction of the part from the mold.
   - Use Draft Analysis tool to measure the draft angle on the part features.

D. Heal the discontinuities between the surfaces.

E. Replace the undercut faces by positive draft angle.



Stages in the Process

1. Check the surface for surface gaps.
2. Check the surface for tangency discontinuities.
3. Check the surface for curvature discontinuities.
4. Perform the draft analysis.
5. Heal the surface from discontinuity problems.
6. Replace the undercut defective surface by a drafted surface.

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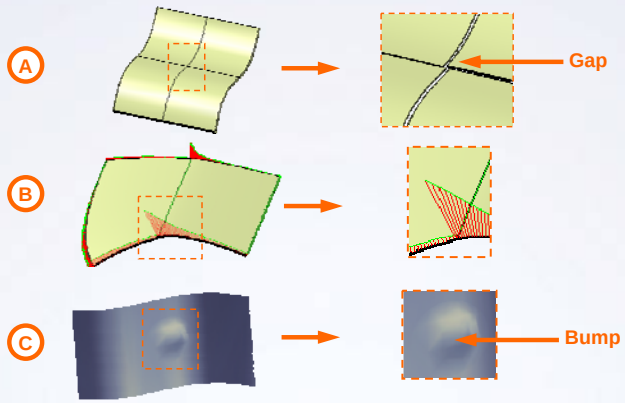
Surface Continuity Check

We need to perform surface connection analysis when surfaces from CATIA V4 or other CAD packages are imported into CATIA V5. This ensures that good quality surface is provided for further processing.

Following are the flaws that can be detected between surfaces:

A. Surface continuity faults like gaps, overlapping surfaces and tangency and curvature discontinuities.
B. Irregular surface boundaries.
C. Surface Inflections.



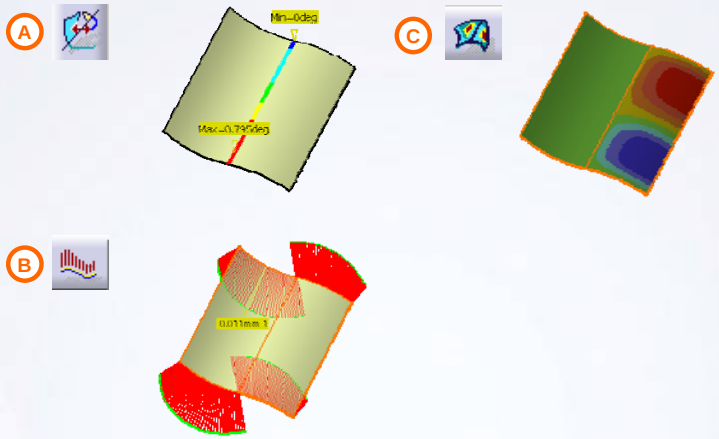
Tools to Detect Geometric Connection

Following are the tools to detect geometrical connection faults:

A. **Connect Checker**: It is used to check connections between two or more surfaces. The tool gives a measure of Distance (mm), Tangency (degrees), and Curvature (percentage) between the edges of the surfaces.

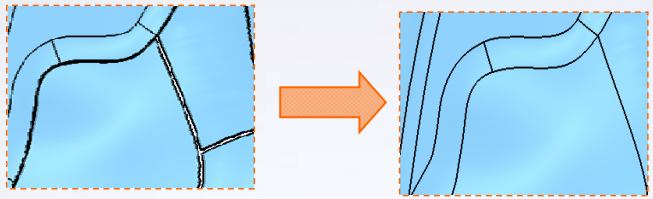
B. **Porcupine Analysis**: It is used to detect the imperfections in the boundaries of a surface which cannot be seen with the naked eye. The result is shown in the form of spikes.

C. **Surfacic Curvature Analysis**: It is used to detect curvature changes on a surface or group of surfaces.



Correcting Defects in Surfaces

The Heal tool is used to correct the defects (gaps, continuity problems) found by performing an analysis between the surfaces. Heal deforms the surface to fill up the gaps and reduces the tangency and the curvature discontinuities.



Before Healing

After Healing

To heal the surfaces, you need to enter healing parameters. These parameters are threshold values and must be deduced from the results of the analysis. Parameters allow you to:

A. Define the discontinuities which are important and must be healed: Merging distance and Tangency angle parameter values must be greater than the maximum value you get from the connect checker analysis.

B. Define the discontinuities which are not important and must not be healed: The magnitude of distance objective or tangency objective parameters must be set less than the discontinuity that you desire to heal. In other words, discontinuities less than these values are not considered as gap.

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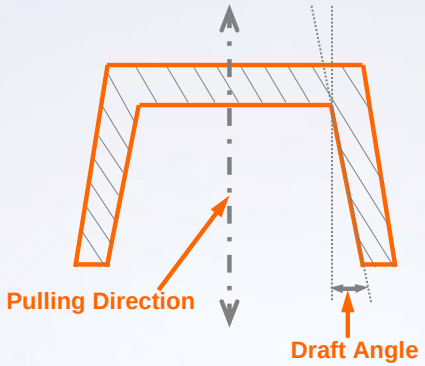
Surface Moldability Check

A certain value of draft angle is provided on the walls of a part to be molded. This is done for easy extraction of the part from the mold. Using the Draft Analysis tool, you can check whether the part is extractible from the mold at the given draft angle.

In case of huge parts reflect line helps you to divide a part into two or more smaller parts which can be easily retractable. It helps in determining the parting line of a part.



Main Tools

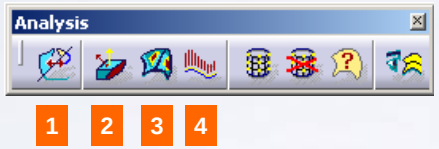
Analysis Toolbar

1 **Connect Checker Analysis:** Checks connections between two or more surfaces.

2 **Feature Draft Analysis:** Analyzes the draft direction on selected surfaces.

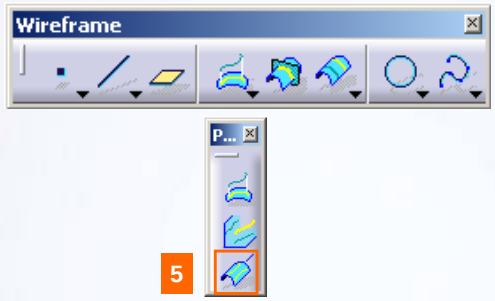
3 **Surfacic Curvature Analysis:** Analyzes the Gaussian curvature on shape.

4 **Porcupine Curvature Analysis:** Detects the imperfections in boundaries of the curve.



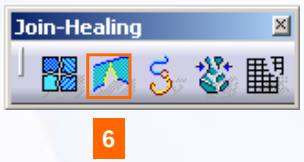
Wireframe Toolbar

5 **ReflectLine:** Creates a reflect line.



Operations Toolbar

6 **Healing:** Heals the surface.



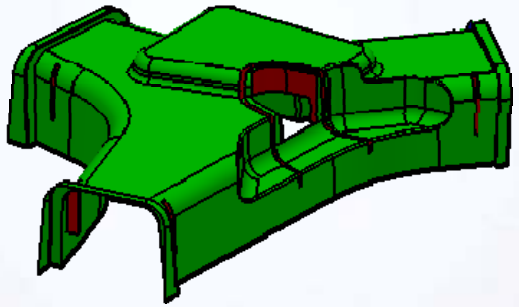
Exercise 5B

Recap Exercise

20 min

In this exercise, you will practice how to analyze the draft angle of molded parts using draft analysis tool in Generative Shape Design workbench. High-level instructions are provided to perform this exercise.

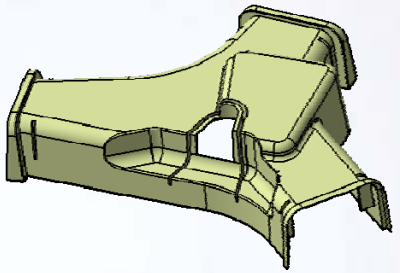
By the end of this exercise you will be able to perform draft analysis on a apart using the Draft Analysis tool.

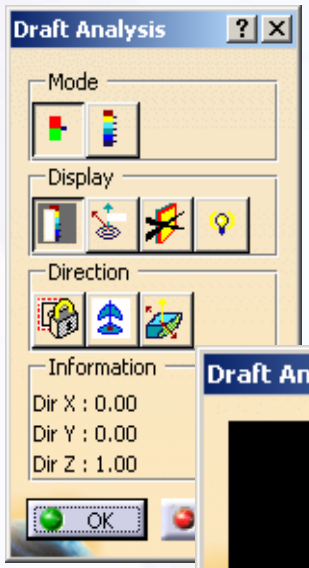


Exercise 5B (1/2)

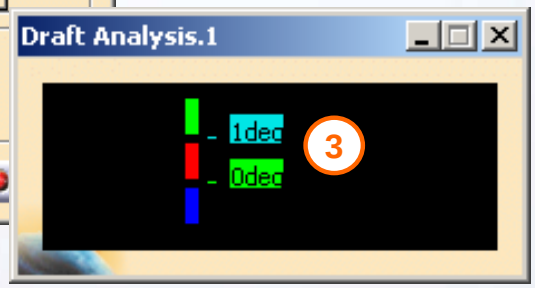
1. **Open the part.**
   - Open an existing part file. The file consists of V4 surfaces of a molded part.
     a. Browse and open part: Exercise_5B_Start.CATPart

2. **Check the surface for their extractability using the Draft Analysis tool.**
   - Perform a draft analysis on the joined surface (Join.2 in start part).



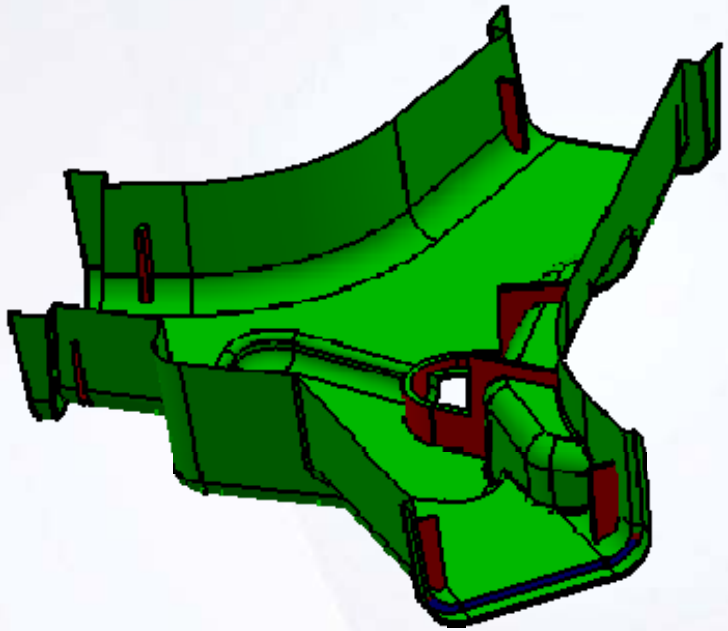


3. **Check for surfaces having draft less than 1deg.**



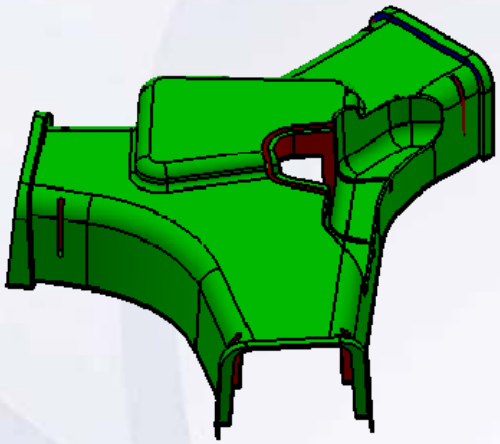
Exercise 5B (2/2)

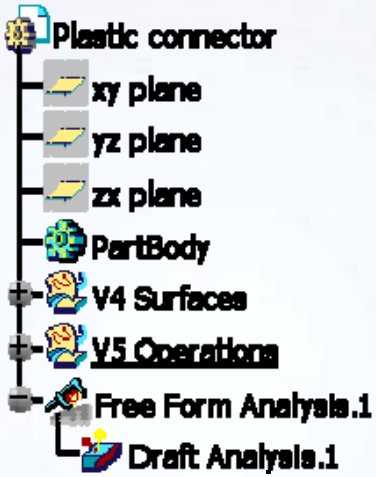
4. **Study the results of the draft analysis.**
   a. The red color signifies that these surfaces have the draft less than 1 deg and greater than 0deg.
   b. The blue surface has draft less than 0deg (negative draft).
   c. The green surface has draft greater than 1deg.



Exercise 5B: Recap

- ✓ Analyze the draft on the part using the Draft Analysis tool.





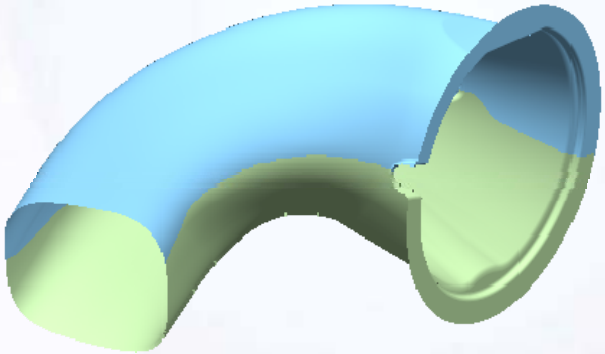
Exercise 5C

Recap Exercise

20 min

In this exercise, you will determine the parting line for a given surface using the Reflect Line tool. With the given set of surfaces, you will be able to divide the surface into separate parts which have to be molded independently. High-level instructions are provided to perform this exercise.

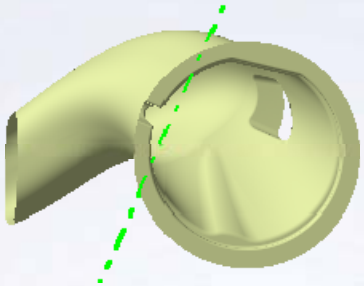
By the end of this exercise you will be able to determine the parting line of a given surface.



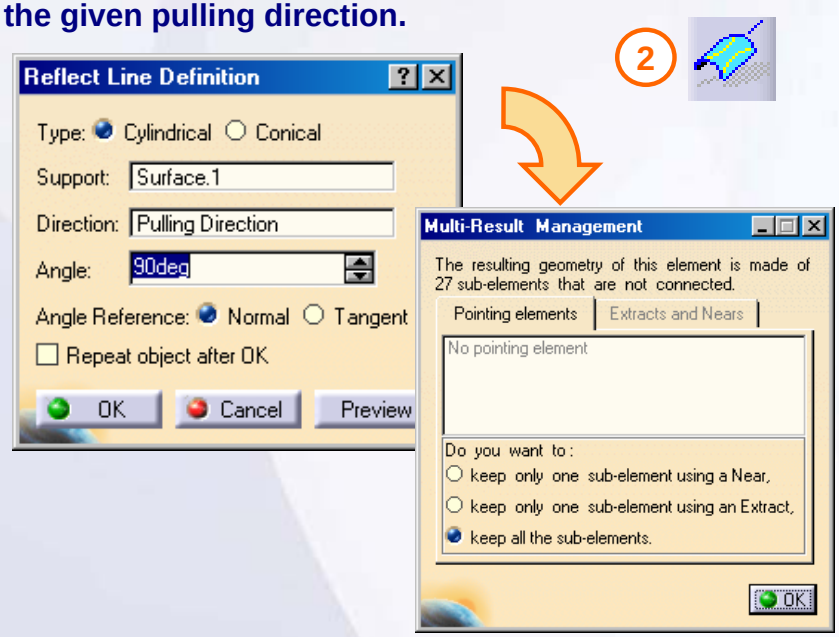
Exercise 5C (1/3)

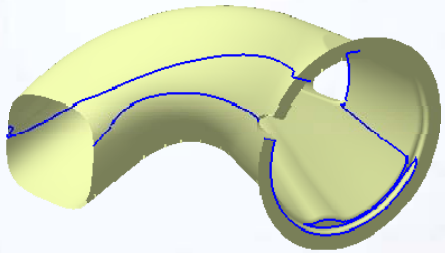
1. Open the part.

- Open an existing part file. The file consists of V4 surfaces of a molded part.
  a. Browse and open part: Exercise_5C_Start.CATPart



2. Compute the reflect line on the surface using the given pulling direction.

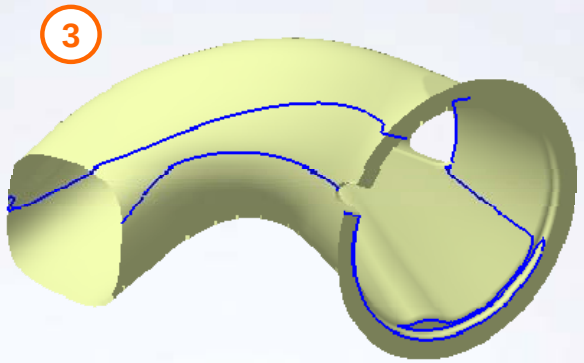


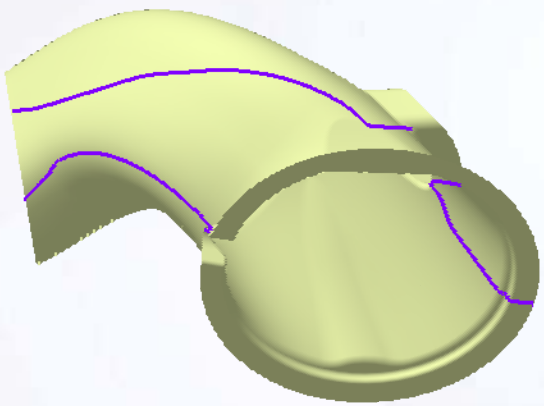


Exercise 5C (2/3)

3. Edit the reflect line to get a smooth parting line.

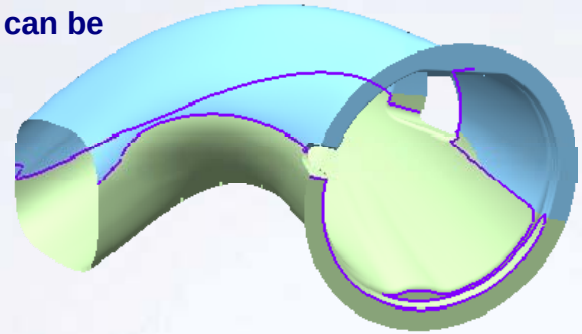
- Extract the curves from the resultant reflect line and edit them. You will edit the reflect line when it has more than one sub-element. The editing consists of splitting the erroneous area of the curve and filling the gap between the curves.
- As shown below, create three curves. These curves are later used to split the surface into two parts.
- Join the three curves.

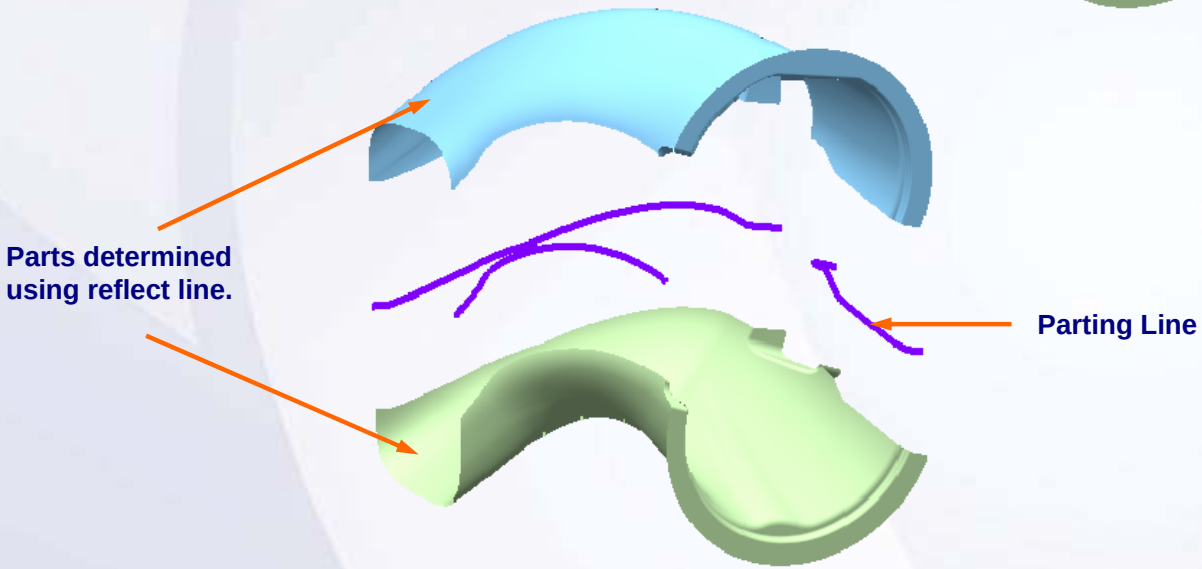




Exercise 5C (3/3)

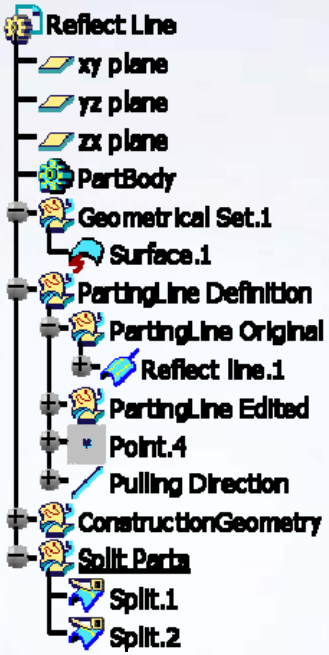
4. Split the surface using the edited reflect line. You will divide the surface into two parts so that they can be molded separately.





Exercise 5C: Recap

✓ Determine the parting line using Reflect line.



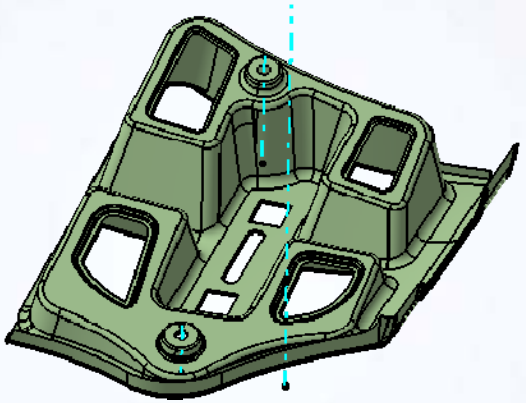
Case Study: Surface Check Tools

Recap Exercise

20 min

You will practice what you learnt, by completing the case study model.
In this exercise, you will perform different surface checks and rectify the defects.

- ✓ Check the surfaces for gaps to ensure easy machinability.
- ✓ Check the surface for tangency and curvature continuity.
- ✓ Check the draft on the surface for better extraction of the part from the mold.
- ✓ Heal the discontinuities between the surfaces.
- ✓ Replace the undercut faces by a positive draft angle.



Using the techniques you have learned in this lesson, and with tips from the previous exercises, create the model without detailed instruction.

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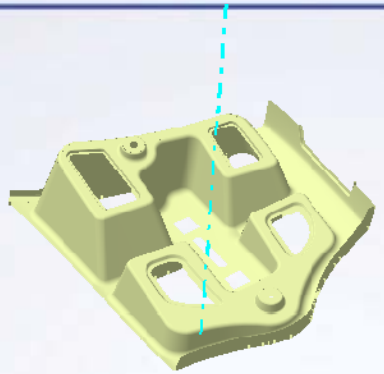
Do It Yourself: Surface Check Tools (1/5)

The following steps offer hints to guide you through the creation of door part surfaces.

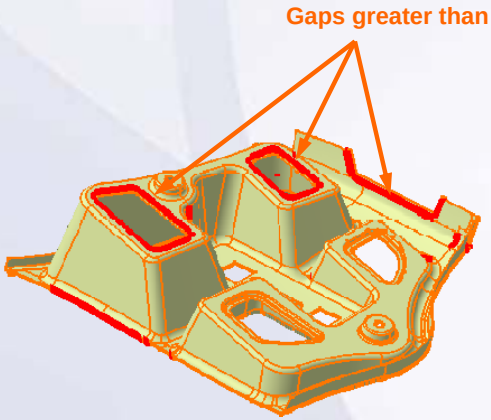


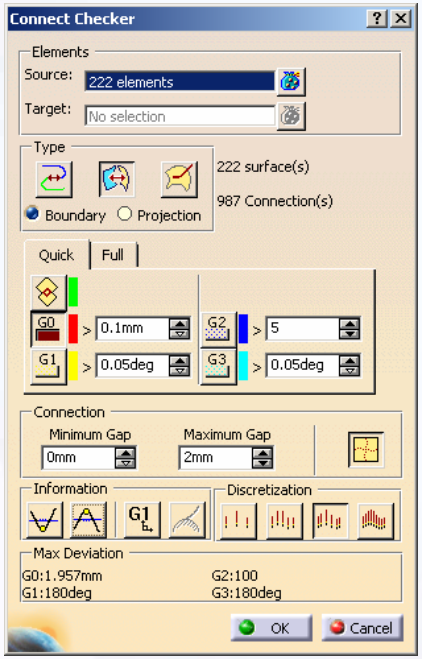
1. **Open the given part consisting of the CATIAV4 surfaces data.**

   Browse through the files and open the model Case_Study_Start.CATPart

2. **Check the point connection between the surfaces using surface connect checker.**

   Use the quick analysis option and study the gap between the surfaces.





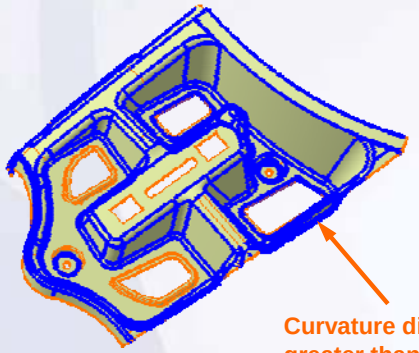
Do It Yourself: Surface Check Tools (2/5)

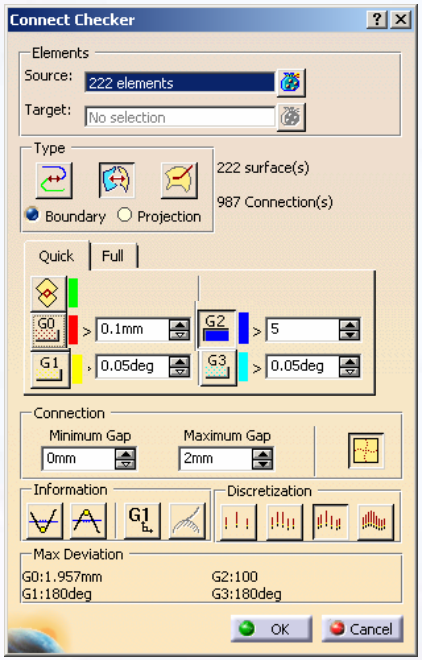
3. **Check the Tangency connection between the surfaces.**

   Study the tangency discontinuity between the surfaces under Tangency option.

4. **Check the Curvature connections between the surfaces under curvature option.**

   Study the curvature discontinuity between the surfaces under Curvature option.

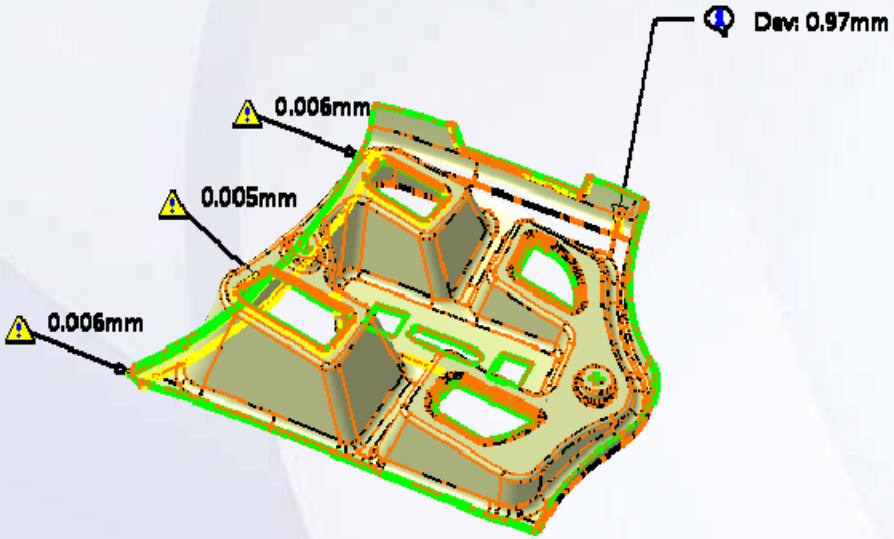


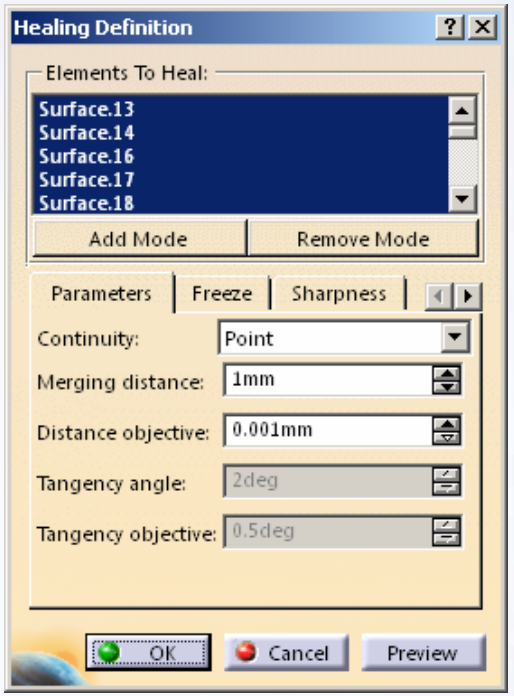


Do It Yourself: Surface Check Tools (3/5)

5. Create a clean topology with the surfaces using the healing tool to correct the gaps.

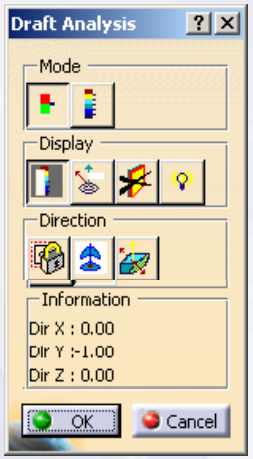
You will be healing all the gaps which are more than 0.001mm and less than 1.0mm.

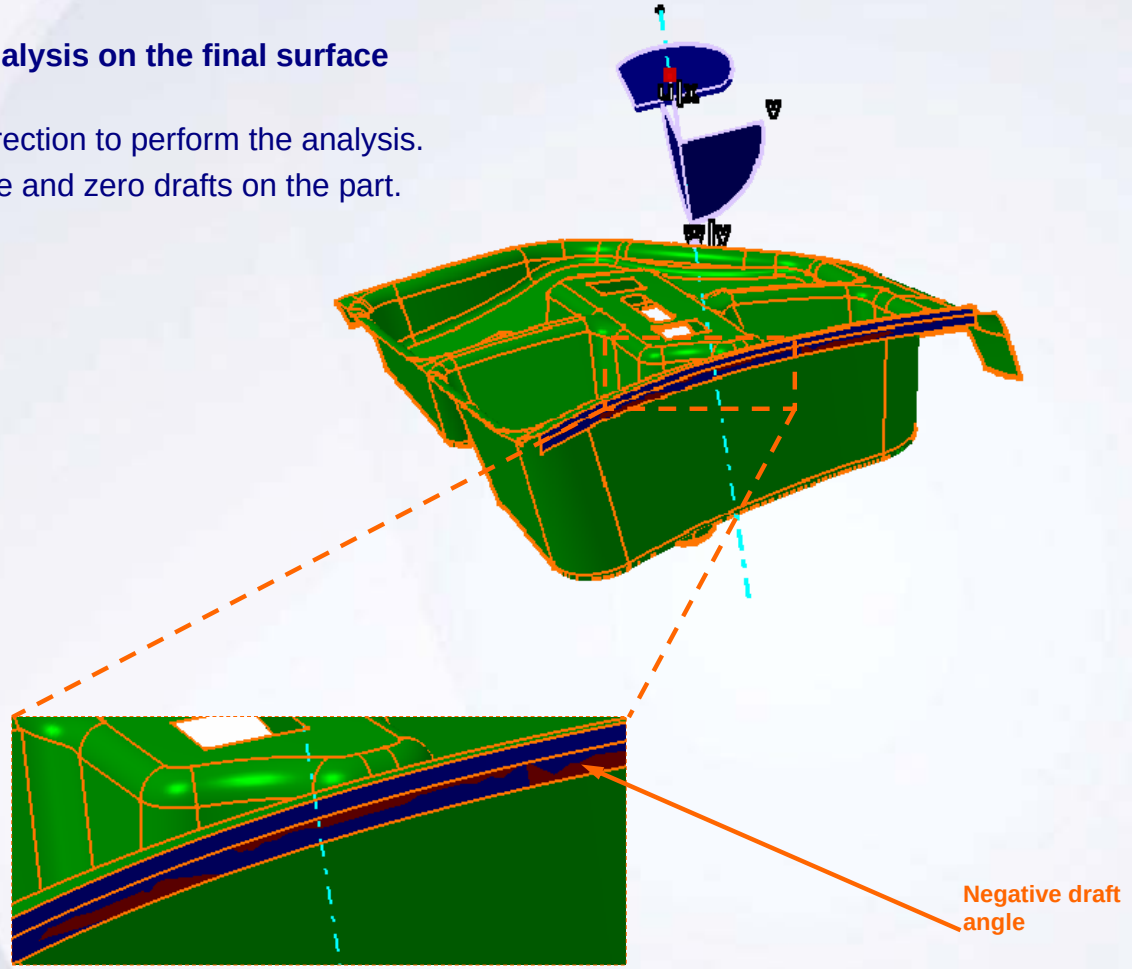




Do It Yourself: Surface Check Tools (4/5)

6. **Perform the Draft analysis on the final surface (Healed surface).**
   - Use the pulling direction to perform the analysis.
   - Check for negative and zero drafts on the part.

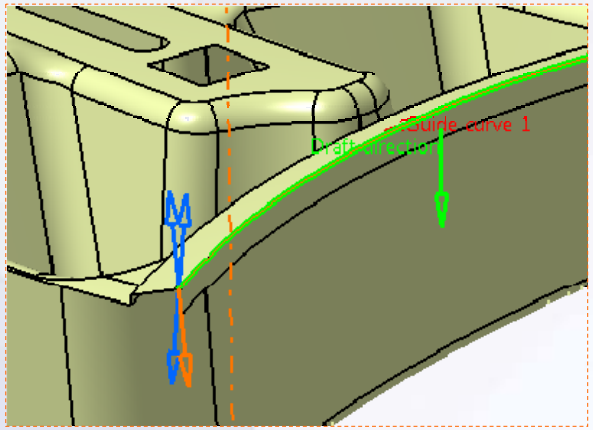




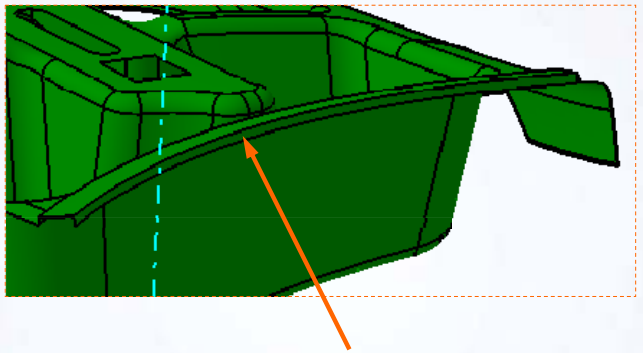
Do It Yourself : Surface Check Tools (5/5)

7. Remove the defective surface.

- Remove the defective surface from the healed surface.
- Create a new drafted surface of 3deg in the same place.
- Join the new surface and the previously healed surface.
- Check for negative and zero drafts on the part.



Create new drafted surface in place of defective surface.



Check for negative and zero draft on the part.

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Case Study: Surface Check Tools Recap

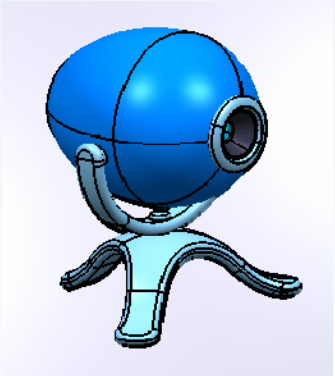
- ✓ Check the surfaces for gaps to ensure easy machinability.
- ✓ Check the surface for tangency and curvature continuity.
- ✓ Check the draft on the surface for better extraction of the part from the mold.
- ✓ Heal the discontinuities between the surfaces.
- ✓ Replace the undercut faces by a positive draft angle.

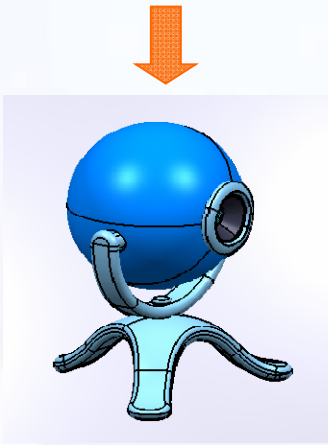
Working in Multi-Model Environment 6

Learning Objectives:

Upon completion of this lesson you will learn about:

- ✓ Surface and Wireframe Publication
- ✓ How to use Published Surface in Product Context





2.5 hours

Case Study

The case study for this lesson is to create a web camera in a multi- model environment using publication tool, in assembly context.

Design Intent

A. Build the top part using lower half of the camera without using Publication.

B. Replace the lower (referenced part) by its variant part, which has a different shape.

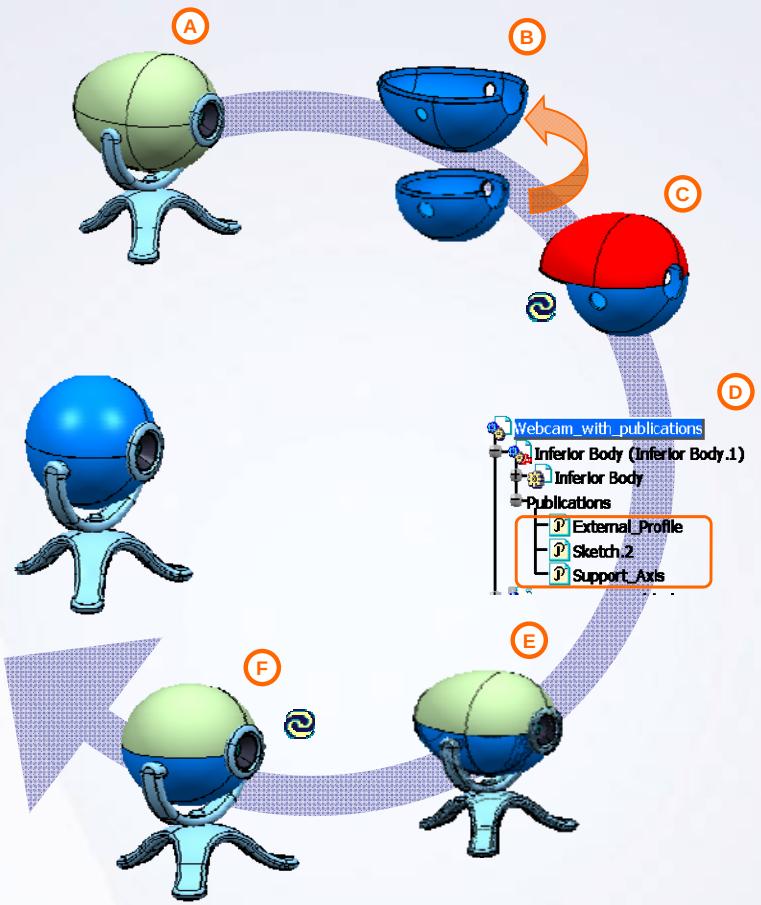
C. Updating the model with new variant will not be possible with out re-routing the child features.

D. Publish the required surfaces of reference lower half of the camera case.

E. Build the top part again using the published surfaces.

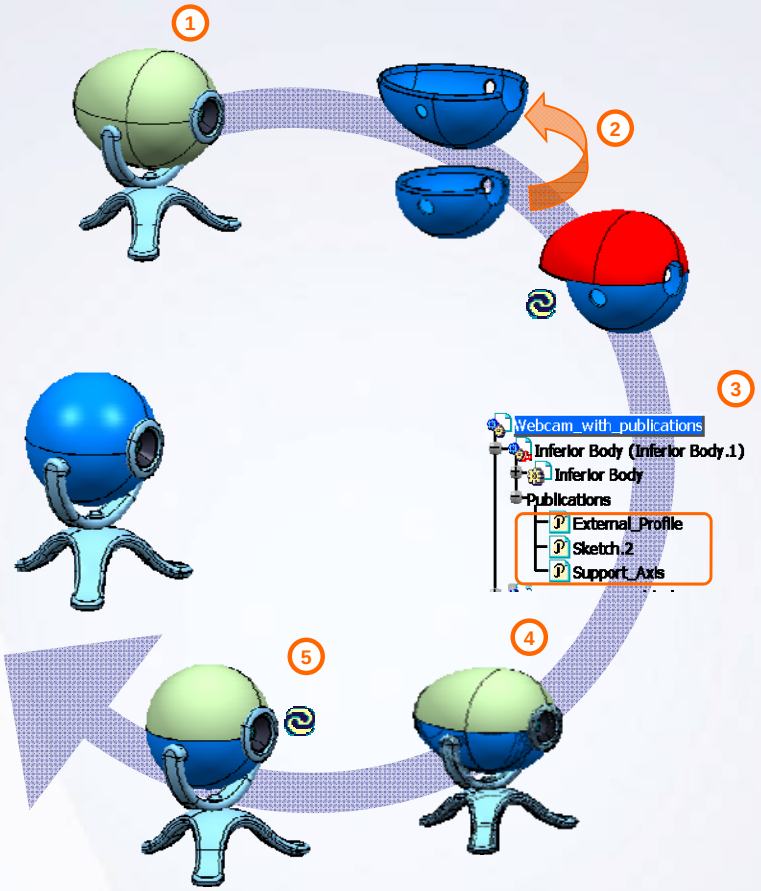
F. Replace the lower (referenced part) by its variant part.

G. Update the model. You will be able to update without any re-routing.



Stages in the Process

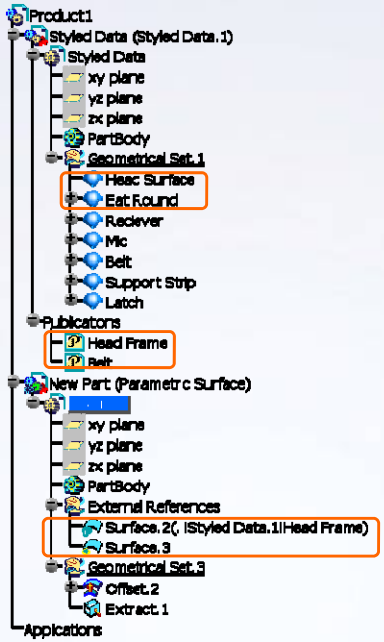
1. Build the Top part referring to the Inferior body without using Publication.
2. Replace the Inferior body with its variant and update the assembly. You will have to re-route links.
3. Publish the reference part.
4. Rebuild the top part using published elements.
5. Replace Inferior body with its variant (with published element) and update the assembly. Assembly gets updated without manual re-routing.



Surface and Wireframe Publication

Publication helps in managing design iterations during product development cycle and provides easy availability of geometric elements to different users. Wires and surfaces are published at part level. Following points must be kept in mind:

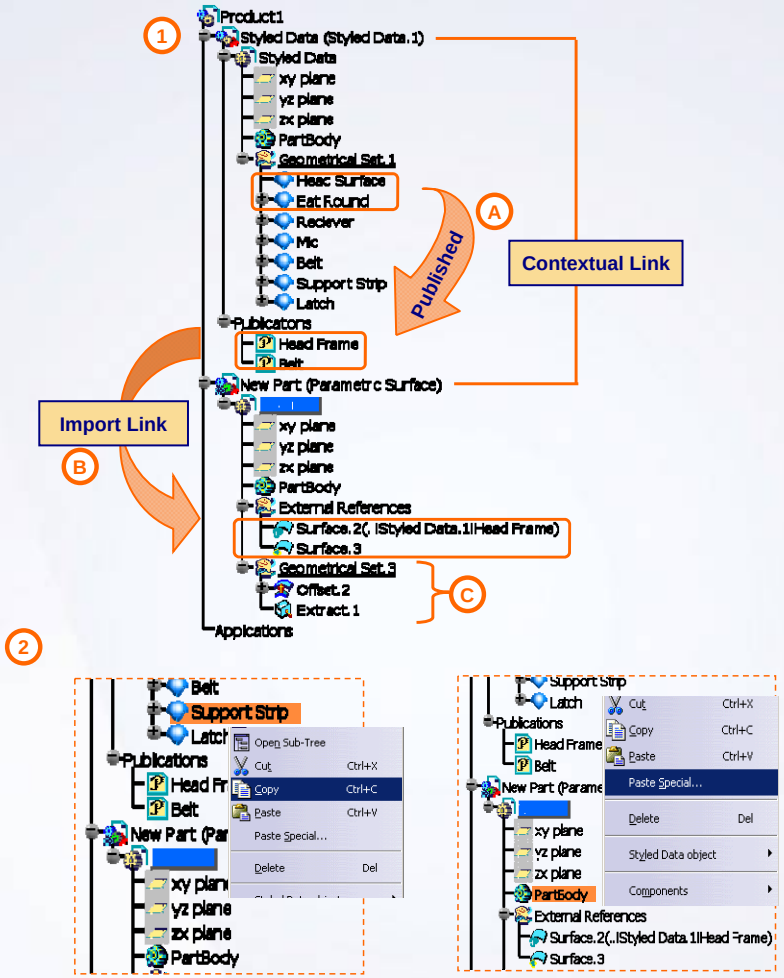
- ✓ The name of the published element must be exactly similar to that in source part and child part.
- ✓ Easily recognizable names must be given to the published elements.
- ✓ Geometry is grouped to give easier access in the Specification Tree.
- ✓ An option is available that allows to select, only the published elements as an external reference.



Use Published Surface in Product Context

There are two methods to use published geometry of a part in a multi-model environment.

1. Geometry Selection during design process.
   A. Publishing of Elements.
   B. Creating External References in a New part.
   C. Creating parametric surface using external referenced geometry.
2. Copy/Paste published elements: You can also copy/paste the published elements with links, to place the external references in “New Part” before you need them. The result is the same even in the tree.



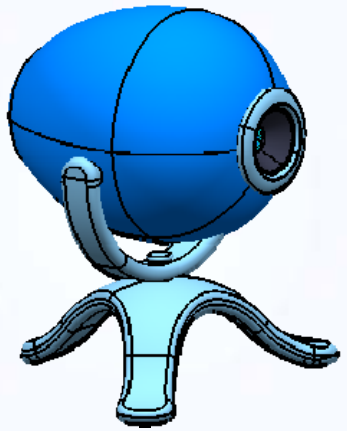
Case Study: Multi-Model Environment with Surfaces

Recap Exercise

20 min

In this exercise, you will practice how to apply multi-model link using external references in the assembly context. You will understand the merits of using publication in an multi-model environment.

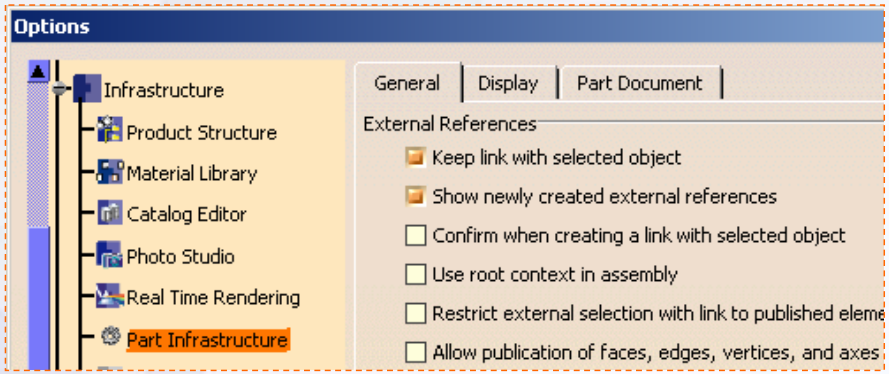
- ✓ Build the top part with the help of the lower half of the camera without using Publication.
- ✓ Replace the bottom (referenced part) by its variant part which has a different shape.
- ✓ Reroute the links in order to update the model with new variants.
- ✓ Publish the required surfaces of reference bottom half of the camera case.
- ✓ Build the top part again using the published surfaces.
- ✓ Replace the bottom (referenced part) by its variant part.
- ✓ Update the model. You will be able to update without re-routing the links.

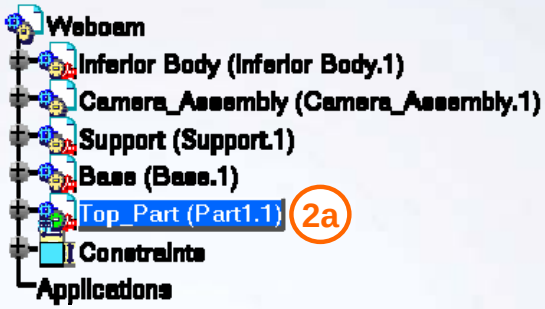


Using the techniques you have learned in this lesson and previous exercises, create the model without detailed instruction.

Do It Yourself: Multi-Model Environment with Surfaces (1/4)

1. **Open the Assembly of 'Web cam'.**
   a. Browse through the files and open the model 'Webcam_start.CATProduct.
   b. Select **Tools>Options>Infrastructure>Part Infrastructure> General** and select the options in the External Reference.





2. **Create a new part named 'Top_Part'**
   a. Position it on an inferior body by creating coincident constraints between xy, yz and zx planes.

Do It Yourself: Multi-Model Environment with Surfaces (2/4)

3. Design the cover in context with 'Inferior_Body'.

a. Create a 360deg revolve using 'External Profile' Sketch of 'Inferior Body'. Use X direction as the revolution axis.
(Observe that after selecting of the sketch from 'Inferior Body' an External Reference Geometrical Set is formed in 'Top_Part').

b. Split the revolute surface using xy plane.

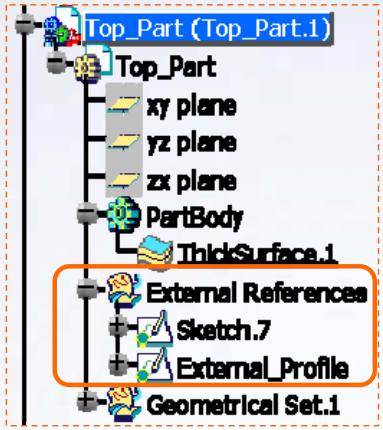
c. Project 'Sketch.2' of 'Inferior Body' on revolved surface along the X direction.

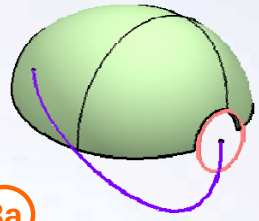
d. Split the revolute surface with a projected curve.

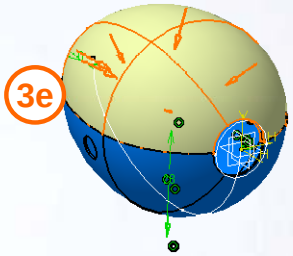
e. Thicken the surface by 2mm towards the inside as shown.

f. Save the product

g. Open the Spherical_Body_Unpublished.CATPart.

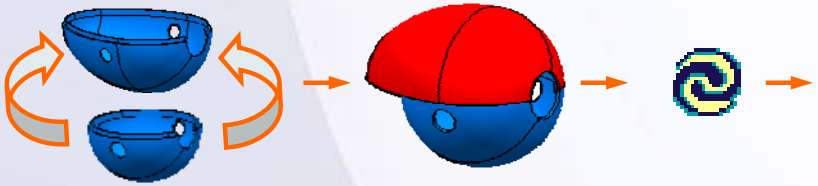


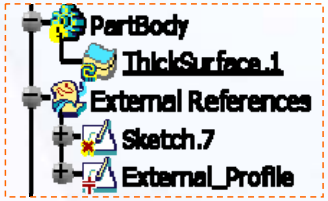


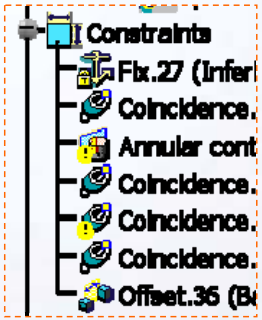


4

4. Replace the 'Inferior Body' with 'Spherical_ Body_Unpublished' and update the product.







Notice that some of the assembly constraints are broken. An error is displayed. These are the constraints that were pointing to the 'Inferior_body'.

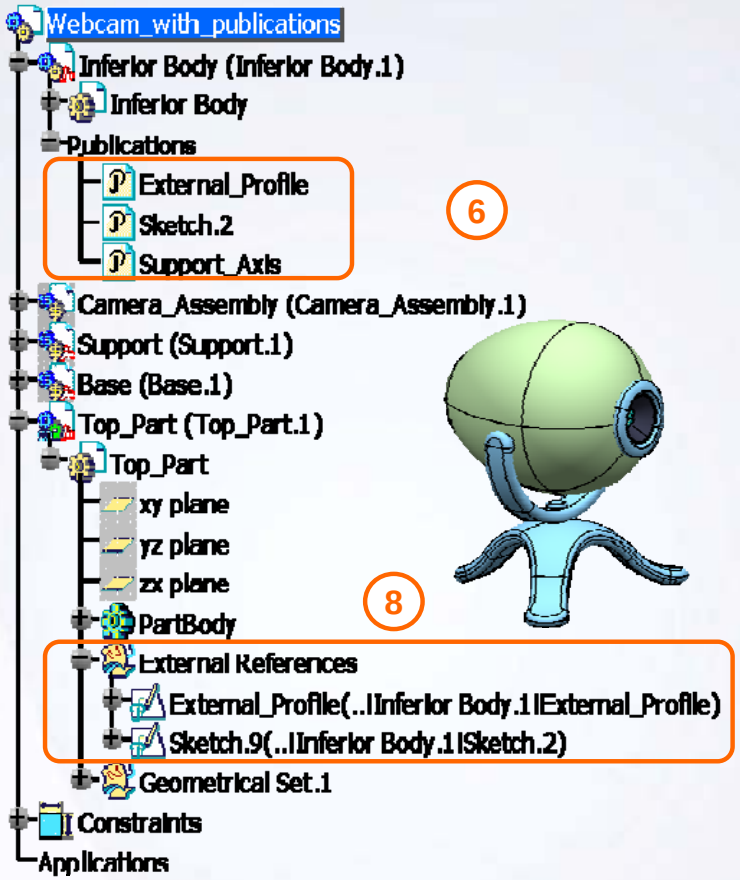
Do It Yourself: Multi-Model Environment with Surfaces (3/4)

5. **Close the product without saving the modifications.**

6. **Reopen the Web Cam assembly Webcam_start.CATProduct and publish the geometries of Inferior Body, which are referred to build the 'Top_Part'.**

7. **Rename the product as Webcam_with_publications.CATProduct.**

8. **Rebuild the 'Top_Part' using the published elements of 'Inferior Body'.**
   a. Insert the new part and follow the same process used in step 3, but this time use published elements of 'Inferior Body'.
   b. Recreate the constraints that need to be connected to newly published elements.
   c. Save the modified files through Save Management.



Do It Yourself: Multi-Model Environment with Surfaces (4/4)

9. **Open Spherical_Body_Published.CATPart and check that it contains publication with exactly the same names as in 'Inferior Body'. If not, rename the publications of 'Inferior Body' and save the product again.**

10. **Replace the 'Inferior Body' by 'Spherical_Body_Published'.**

11. **Update the product.**

Observations: Notice that this time the geometry of 'Top_Part' has adapted to spherical shape of a new inferior body: the external reference of the Top_Part has been automatically reconnected to the published elements of replacing part.

Also notice that the axis of the Support is coincident with the axis of the Inferior Body. The constraints have automatically reconnected due to Publication.

Case Study: Multi-Model Environment with Surfaces Recap

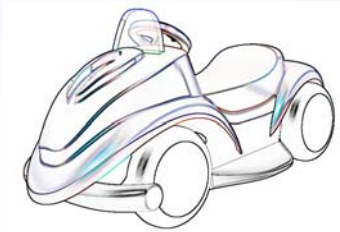
- ✓ Build the top part using the lower half of the camera without applying Publication.
- ✓ Replace the bottom (referenced part) by its variant part which has a different shape.
- ✓ Reroute the links in order to update the model with new variants.
- ✓ Publish the required surfaces of reference bottom half of the camera case.
- ✓ Build the top part again using the published surfaces.
- ✓ Replace the bottom (referenced part) by is variant part.
- ✓ Update the model. You will be able to update without re-routing the links.

Master Project

Toy Car

4 hours

The objective of the project is to model a Toy Car. The model is to be created from scratch in Generative Shape Design (GSD) workbench. You will be provided with the conceptual design created by a stylist. The stylist's ideas are in the form of 2D sketches and a 3D model.

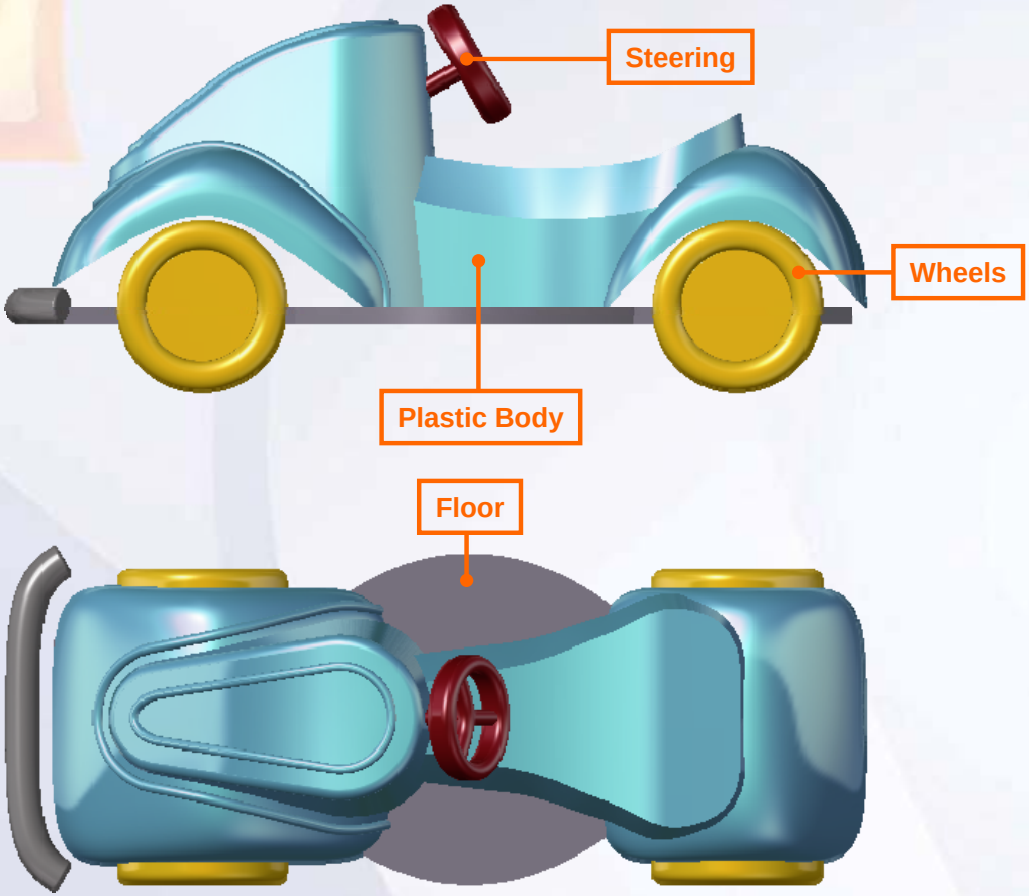


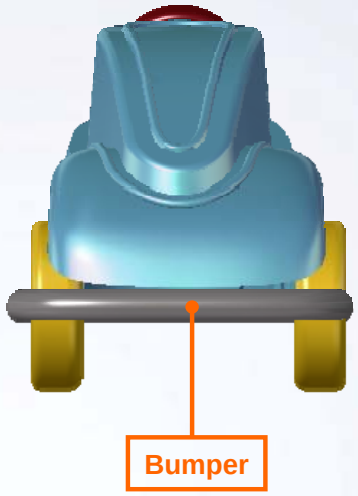
By the end of this project you will be able to:

- **Use the tools of wireframe and surface creation**
- **Use the analysis tools to analyze the wireframe and surface data**
- **Use the surface relimitation and connection tools to complete the model**

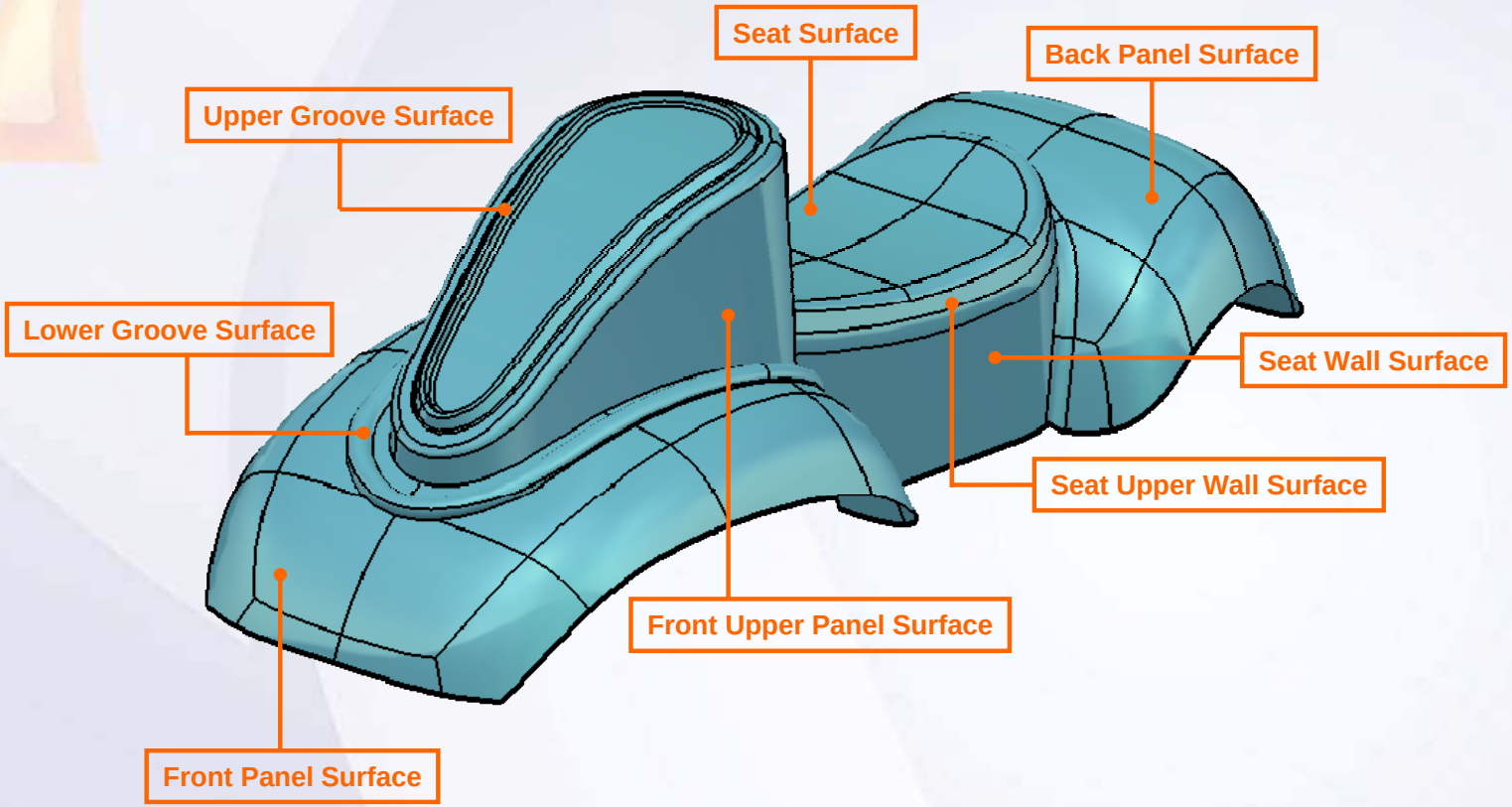
You will work in a multi-model Environment. This will ensure that whenever the stylist makes changes in the conceptual design the corresponding changes are propagated to your model.

Master Project: Toy Car





Master Project: Toy Car

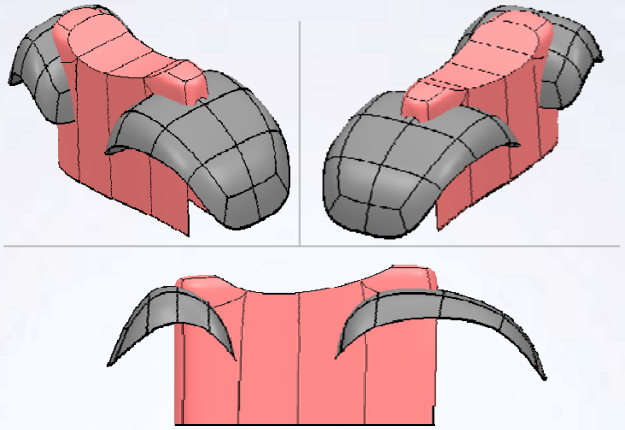


Master Project: Overview (1/6)

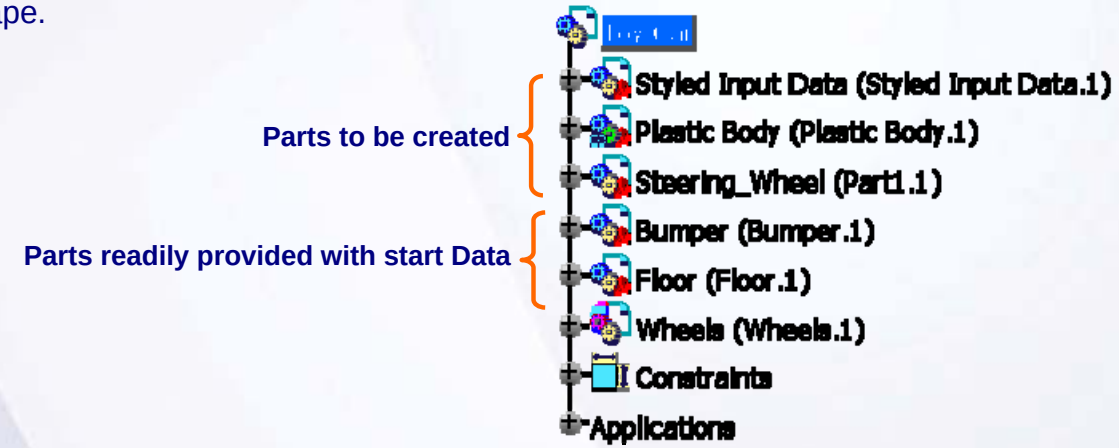
The following is a list of steps that are required to complete the master project:

1. **Study the given input data.**
   - Understand the use of surface and curve provided in the input styled data to create different parts of the toy.
   - Study the product tree structure. Understand the parts to be modeled and the parts readily available in Start data.

   - Design Intent:
     - ✓ To use the data provided by the stylist to create the surfaces in Generative Shape Design workbench.
     - ✓ To meet the requirements of the stylist and achieve the proposed shape.



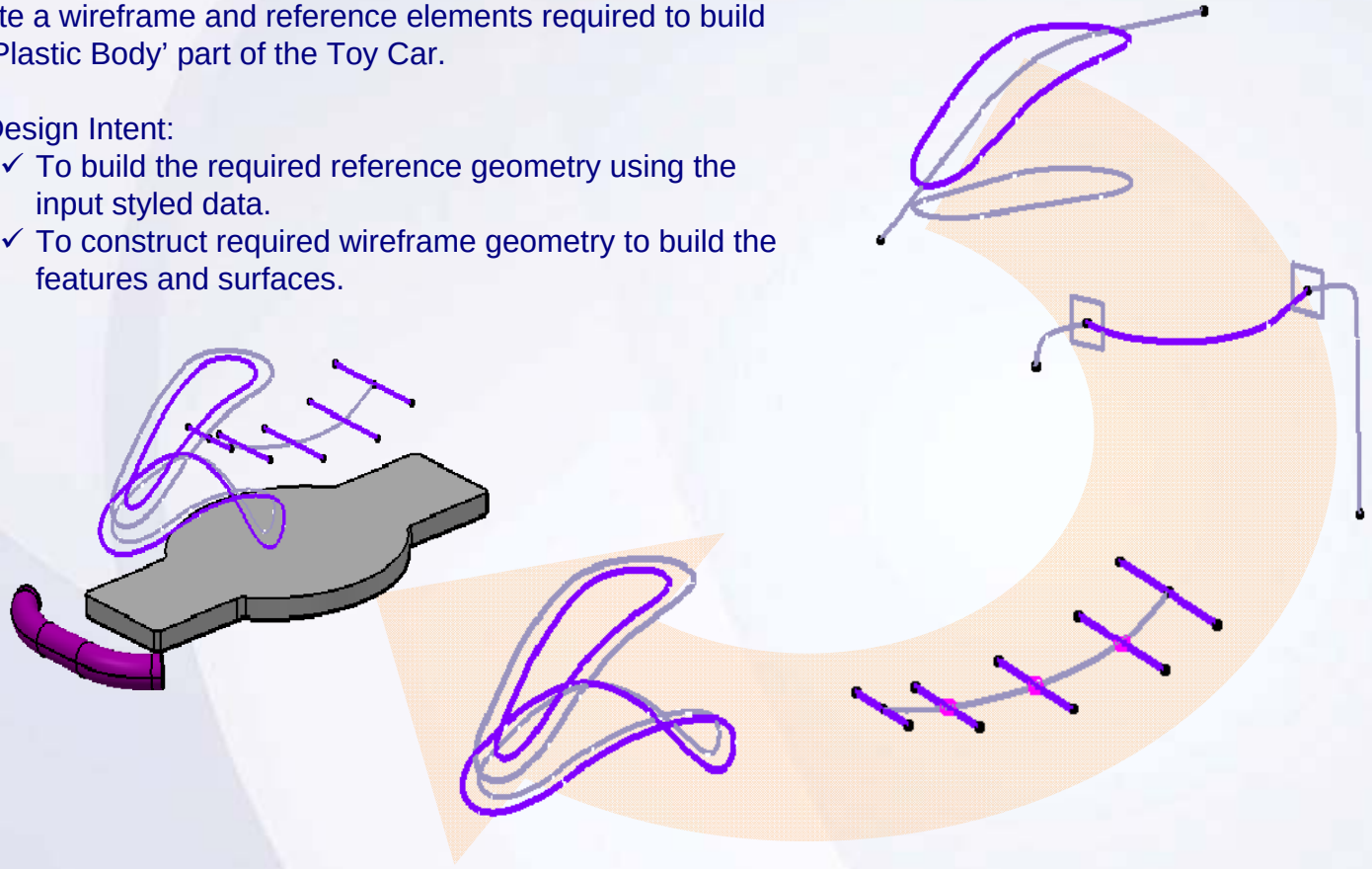
Styled Input Data



Master Project: Overview (2/6)

2. **Wireframe Creation - Design the wireframe to build Plastic Body part in Part context.**
   Create a wireframe and reference elements required to build the 'Plastic Body' part of the Toy Car.

   - Design Intent:
     - ✓ To build the required reference geometry using the input styled data.
     - ✓ To construct required wireframe geometry to build the features and surfaces.

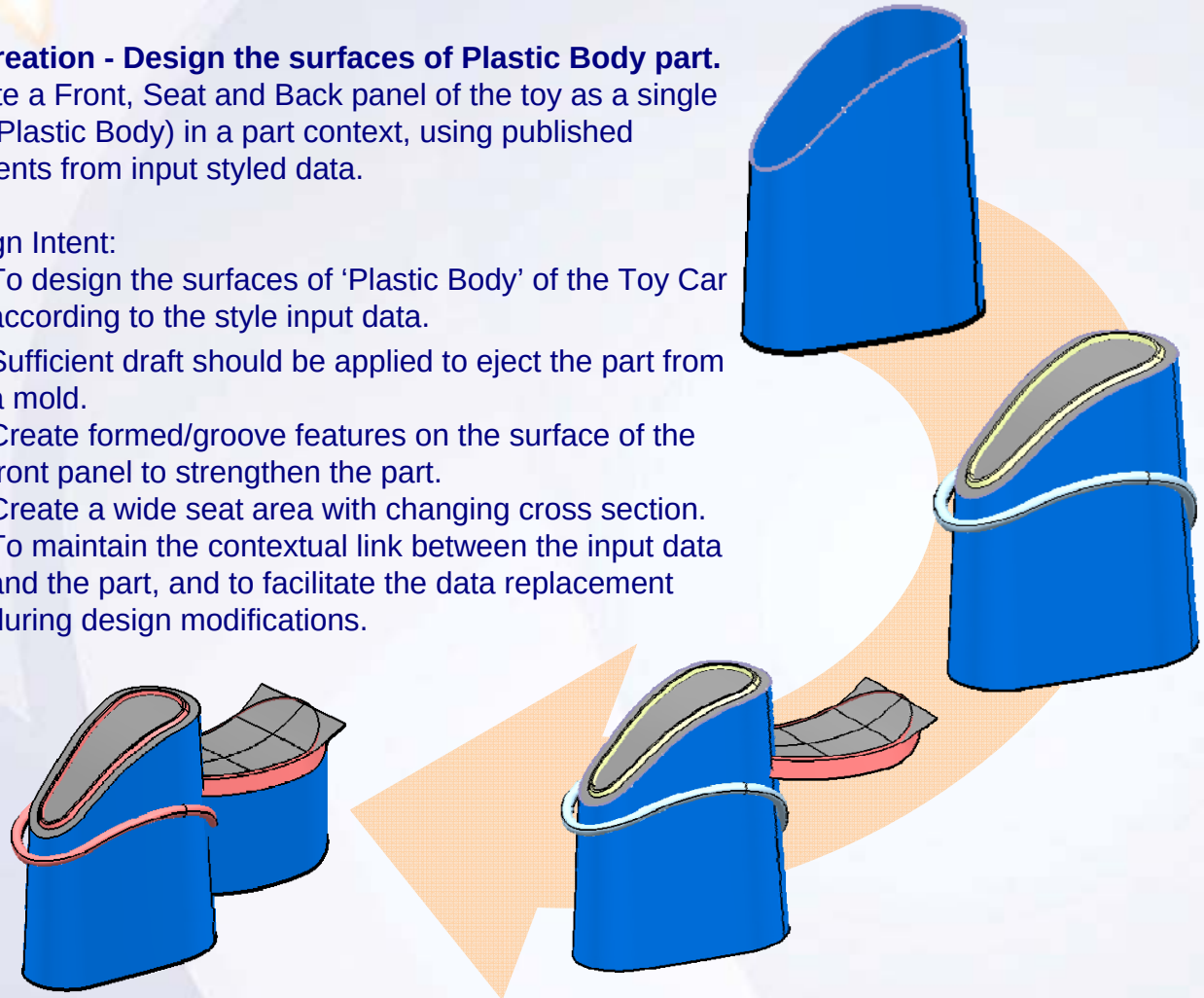


Master Project: Overview (3/6)

3. Surface Creation - Design the surfaces of Plastic Body part.

- Create a Front, Seat and Back panel of the toy as a single unit (Plastic Body) in a part context, using published elements from input styled data.

- Design Intent:
  - ✓ To design the surfaces of 'Plastic Body' of the Toy Car according to the style input data.
  - ✓ Sufficient draft should be applied to eject the part from a mold.
  - ✓ Create formed/groove features on the surface of the front panel to strengthen the part.
  - ✓ Create a wide seat area with changing cross section.
  - ✓ To maintain the contextual link between the input data and the part, and to facilitate the data replacement during design modifications.

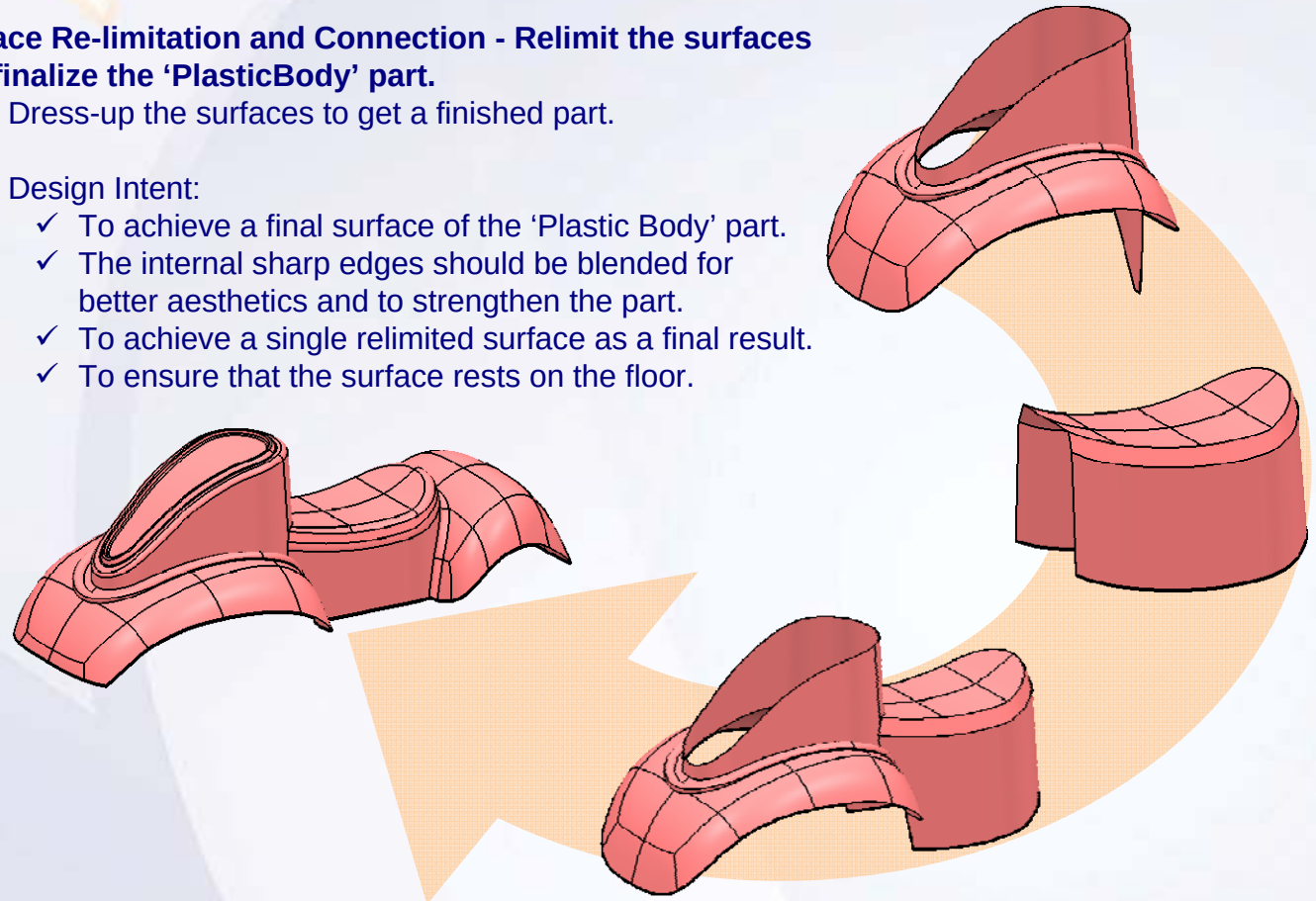


Master Project: Overview (4/6)

4. Surface Re-limitation and Connection - Relimit the surfaces and finalize the 'PlasticBody' part.

- Dress-up the surfaces to get a finished part.

- Design Intent:
  - ✓ To achieve a final surface of the 'Plastic Body' part.
  - ✓ The internal sharp edges should be blended for better aesthetics and to strengthen the part.
  - ✓ To achieve a single relimited surface as a final result.
  - ✓ To ensure that the surface rests on the floor.

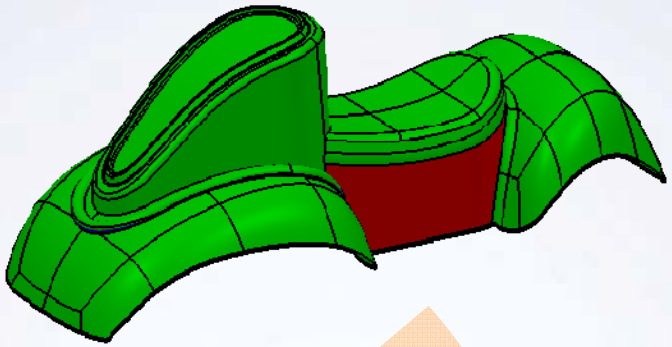


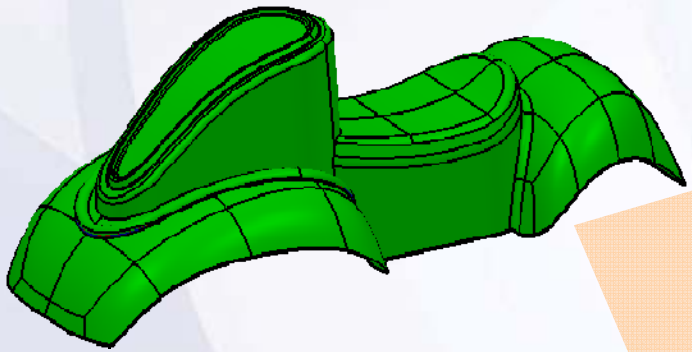
Master Project: Overview (5/6)

5. Surface Analysis - Analyze the Plastic Body part for moldability.

- Create a draft analysis on the panel to detect the undercuts and resolve them.

- Design Intent:
  - ✓ To check the surface for less and negative drafts.
  - ✓ Ensure moldability of the part.
  - ✓ Edit or replace the surface to achieve the moldability of the part.



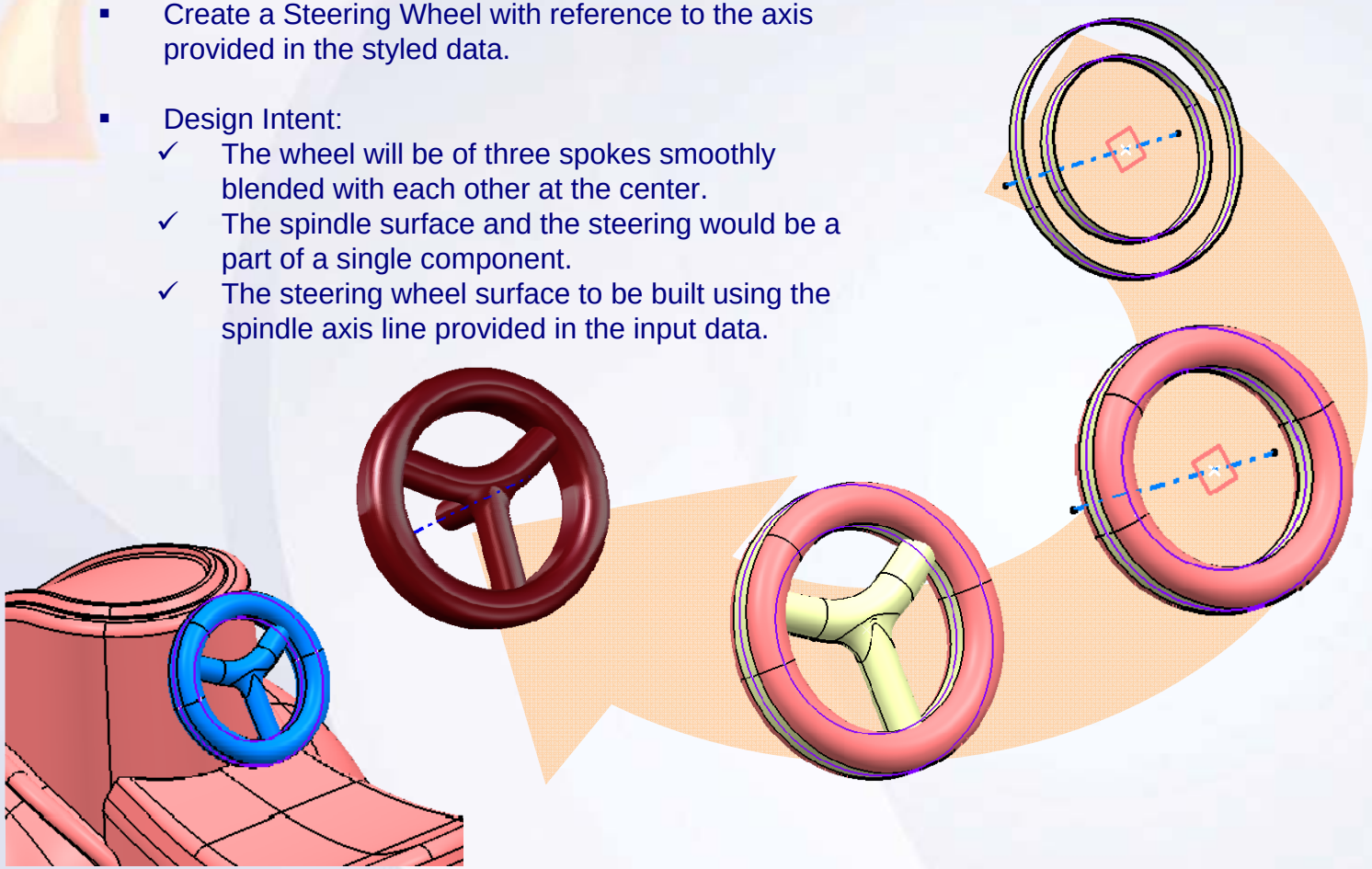


Master Project: Overview (6/6)

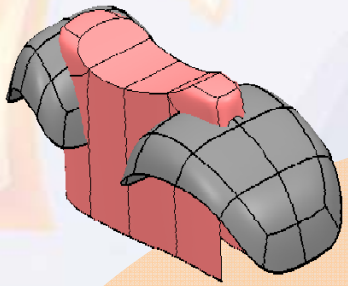
6. Design the Steering Wheel .

- Create a Steering Wheel with reference to the axis provided in the styled data.

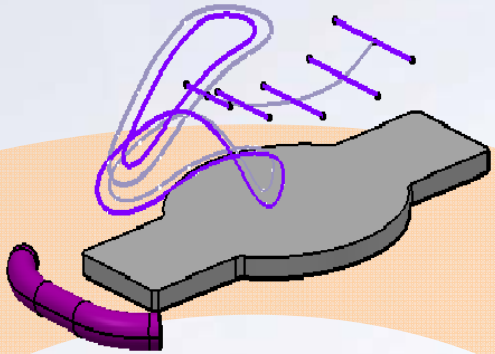
- Design Intent:
  - ✓ The wheel will be of three spokes smoothly blended with each other at the center.
  - ✓ The spindle surface and the steering would be a part of a single component.
  - ✓ The steering wheel surface to be built using the spindle axis line provided in the input data.



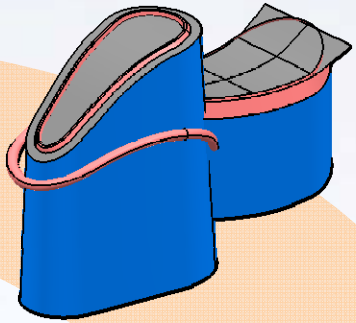
Master Project: Overview Recap



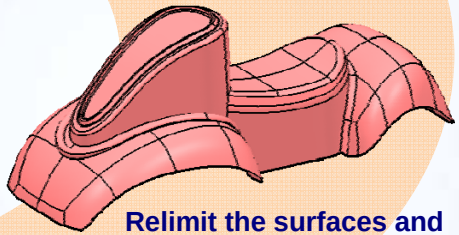
Studying the styled input data and the given product structure.



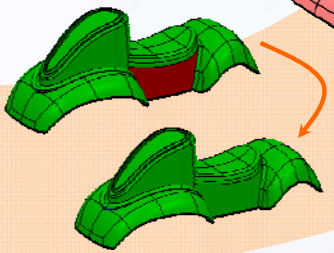
Design the wireframe to build Plastic Body part in Part context.



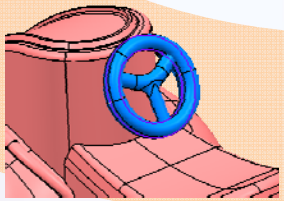
Design the Surfaces for Plastic Body part.



Relimit the surfaces and finalize the 'Plastic Body' part.



Analyze the Plastic Body part for moldability.



Design the Steering Wheel.

Master Project: Study the Input Data

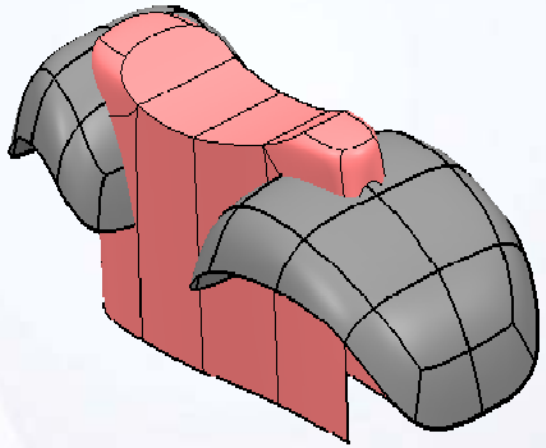
Toy Car

20 Min

The objective of this step is to study and understand the input styled data and the start data provided to you to create the Toy Car model.

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

- **Study and understand the input styled data.**
- **Study and understand the product structure of the start data that is provided.**

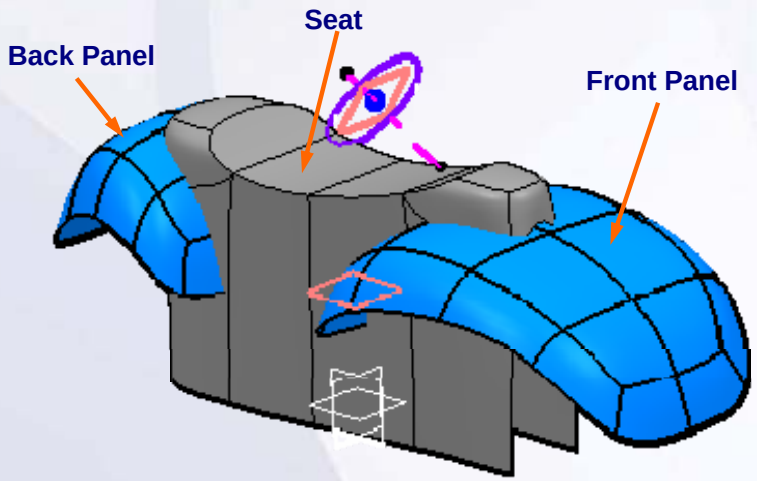


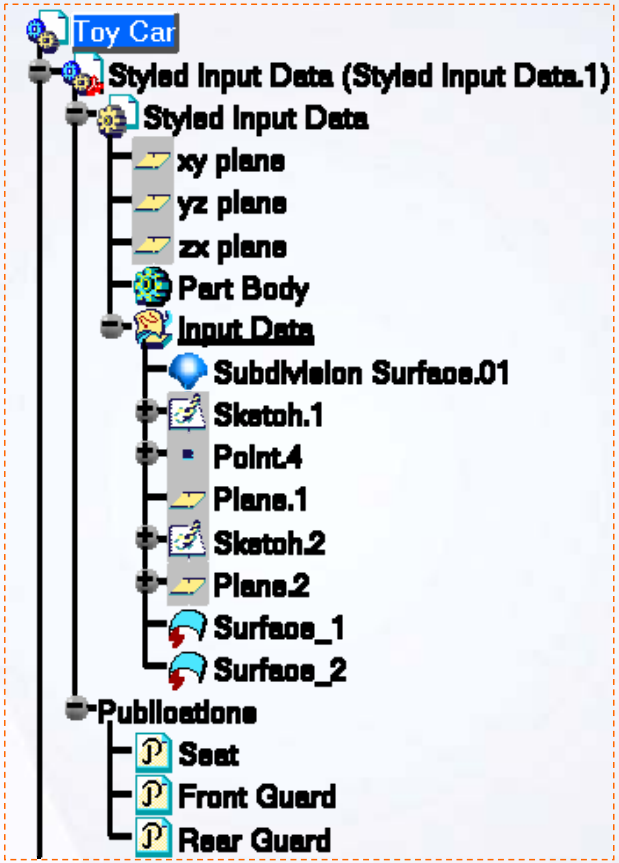
Master Project: Study the Input Data (1/2)

Here is the list of tasks to guide you:

1. **Open 'Toycar_StartData.CATProduct'.**

2. **Study the contents of 'Input Styled Data' Part.**
   - You will observe that the part contains styled surfaces of different parts of the Toycar designed by stylist.
   - Surfaces of Front Panel, Seat and Back Panel have been published.
   - The styled input data consists of reference elements and curves to build the steering surfaces.

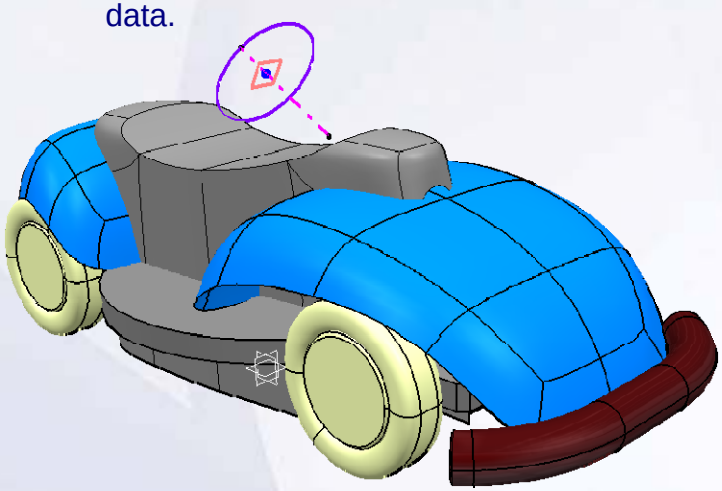


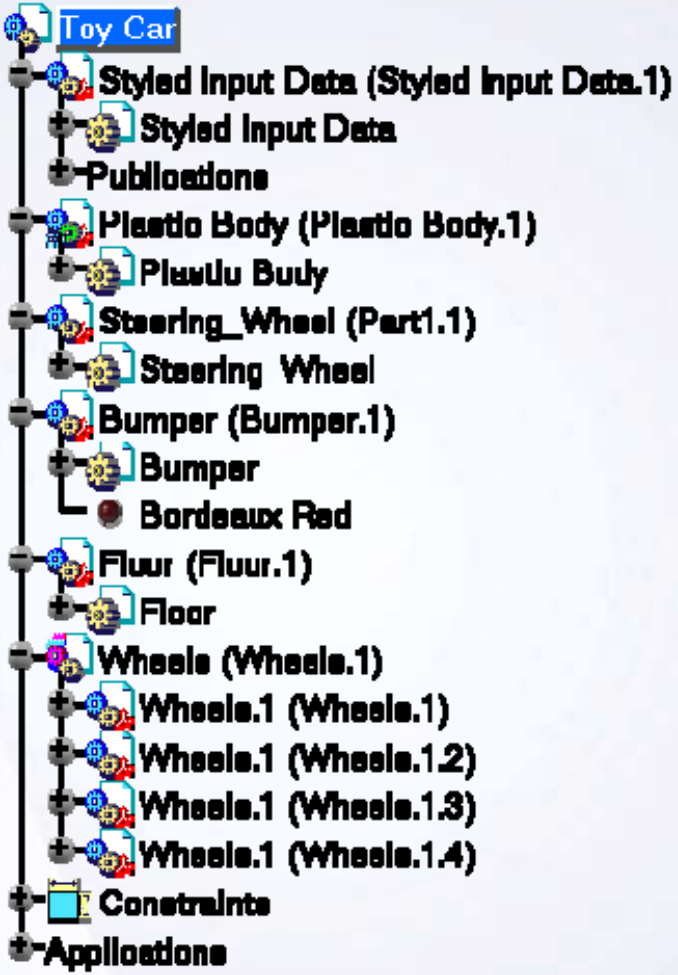


Master Project: Study the Input Data (2/2)

3. Study the contents of Toy Car. CATProduct

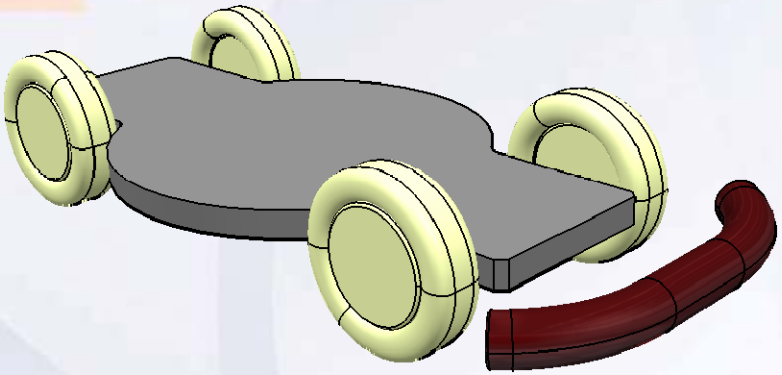
- You will observe that the parts like Bumper, Floor and Wheel are created for you.
- You will create the 'Plastic Body' part using the published elements of the styled input data.
- In a later step you will create a steering wheel surface using reference curves from styled input data.



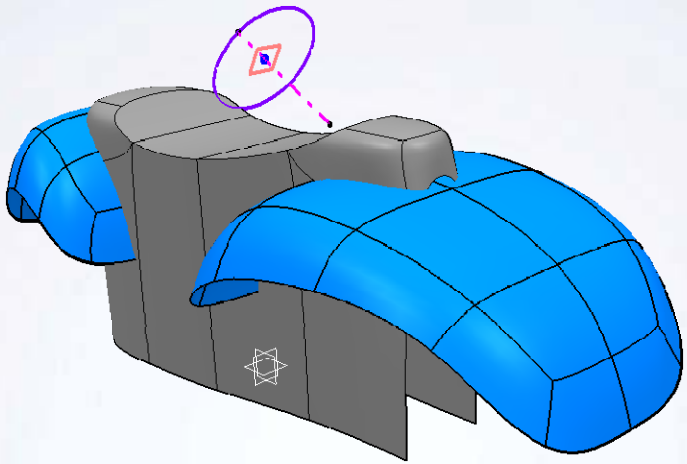


Master Project: Study the Input Data Recap

- ✓ Study and understand the input styled data.
- ✓ Study and understand the product structure of the start data provided.



Wheel, Floor and Bumper



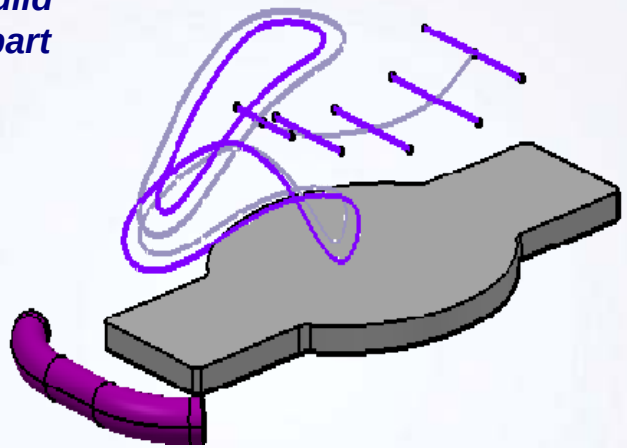
Input Styled Data

Master Project: Wireframe Creation

Toy Car

60 min

The objective of this step is to create a wireframe to build the surfaces of a 'Plastic Body' part. You will work in part context of 'Plastic Body' Part.



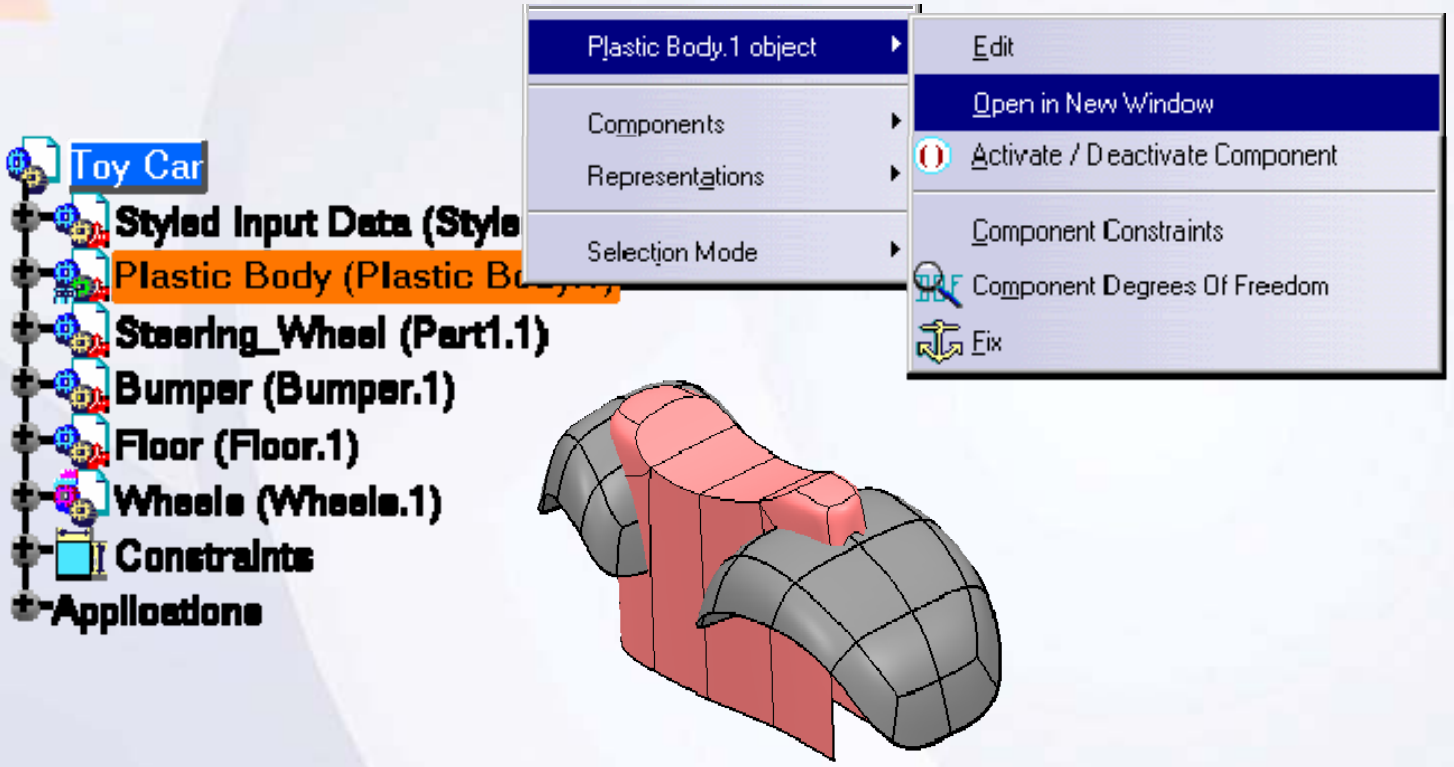
By the end of this exercise you will be able to:

- **Create a 3D curve using two planar curves.**
- **Create curves based on support such as Project curve and Parallel curve.**
- **Extract an intersection curve between two surfaces and two planar elements.**
- **Create the reference elements such as Points, Planes and Lines.**
- **Heal the discontinuities of the curve using the smoothening tools.**

Master Project: Wireframe Creation (1/4)

Here is the list of tasks to guide you:

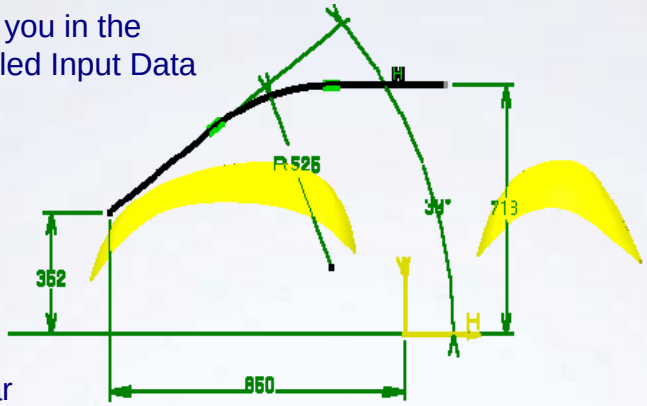
1. **Open the product 'Toycar_StartData.CATProduct'**
2. **Open 'Plastic Body.CATPart' in a New Window**

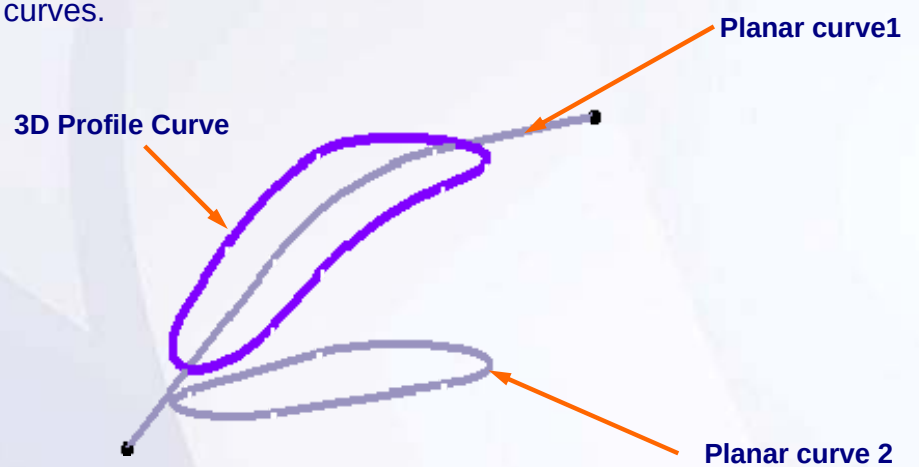


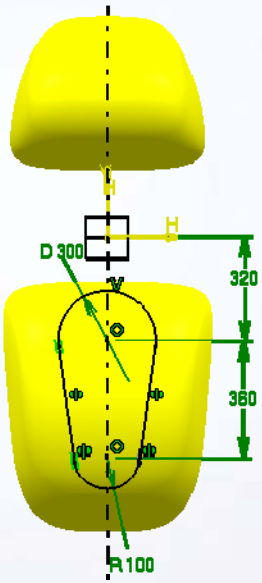
Master Project: Wireframe Creation (2/4)

The published elements are readily made available for you in the Plastic Body part. These elements are copied from Styled Input Data and pasted with a link in the Plastic Body part

1. **Create a 'Profile' curve for the Front upper panel surface.**
   1. Create two planar curves (Sketches), on two perpendicular planes.
   2. defining the outline Of the 3D curve.
   3. Create a 3D Profile curve using these two planar curves.



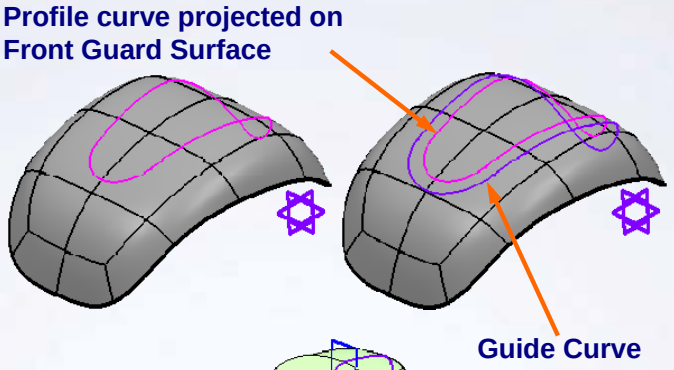




Master Project: Wireframe Creation (3/4)

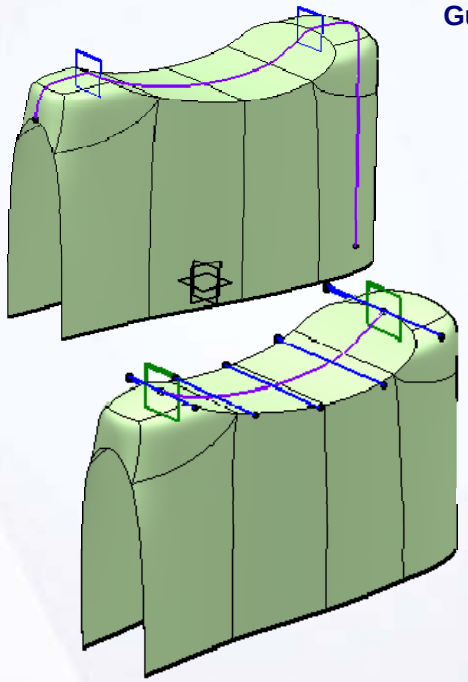
2. Create a guide curve for lower groove surface.

2. The guide curve of lower groove surface should be parallel to the profile curve created in the previous step and lie on the Front panel surface.
3. The guide curve should be at a distance of 40mm from the profile curve projected on the Front Guard surface.



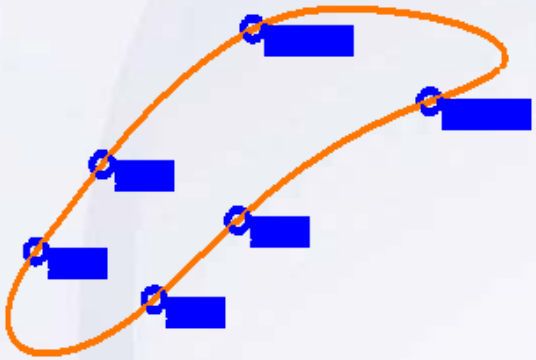
3. Create a wireframe required to build a seat surface.

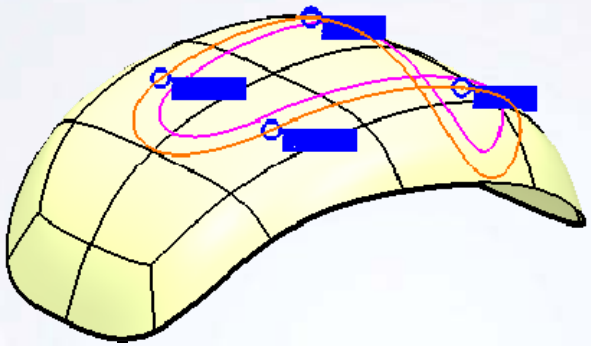
- Create a guide curve with reference to the seat surface from the styled input data as shown.
- Relimit the guide curve to the length required.
- Create a set of points on the relimited curve.
- Create the line sections at a point created on the guide curve (as shown) to achieve the changing section and a wider seat area.
- The length of the straight line section should vary as follows-
  100mm,120mm,140mm,160mm,170mm per side.



Master Project: Wireframe Creation (4/4)

4. **Create a continuity analysis of the Profile and Guide curves created in the second and third operation.**
   - Check the Curvature discontinuities on the curves.



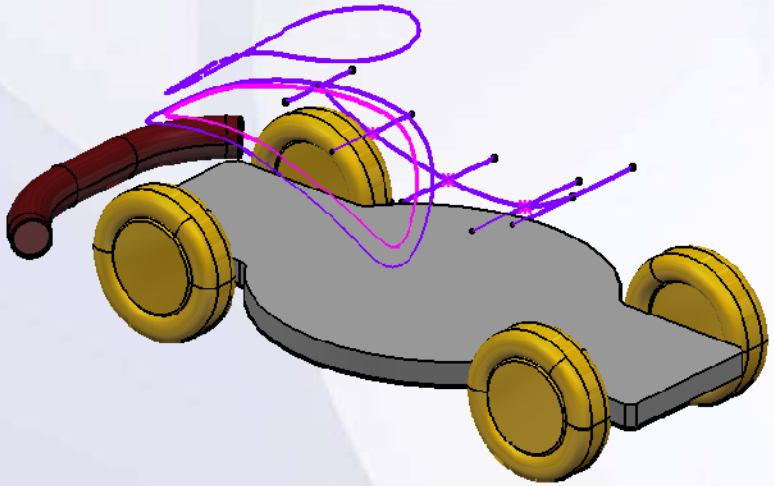


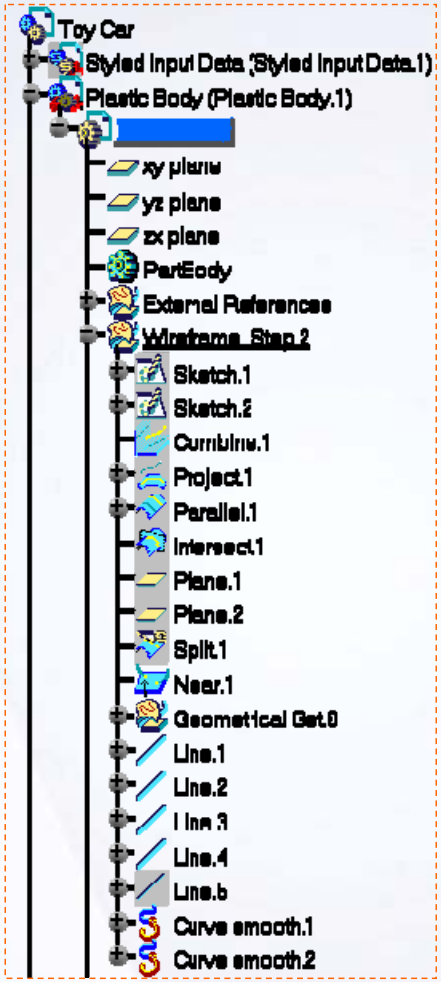
5. **Heal the discontinuities of curve analyzed in the previous operation.**
   - Based on the results of the analysis, smoothen the curves to further achieve a smooth and continuous surface from them.

6. **Save the Part Plastic.CATPart using Save Management.**

Master Project: Wireframe Creation Recap

- ✓ Create a 3D curve using two planar curves.
- ✓ Create curves based on support such as Project curve and Parallel curve.
- ✓ Extract an intersection curve between two surfaces and two planar elements.
- ✓ Create the reference elements such as Points, Planes and Lines.
- ✓ Heal the discontinuities of the curve using the smoothening tool.





Master Project: Surface Creation

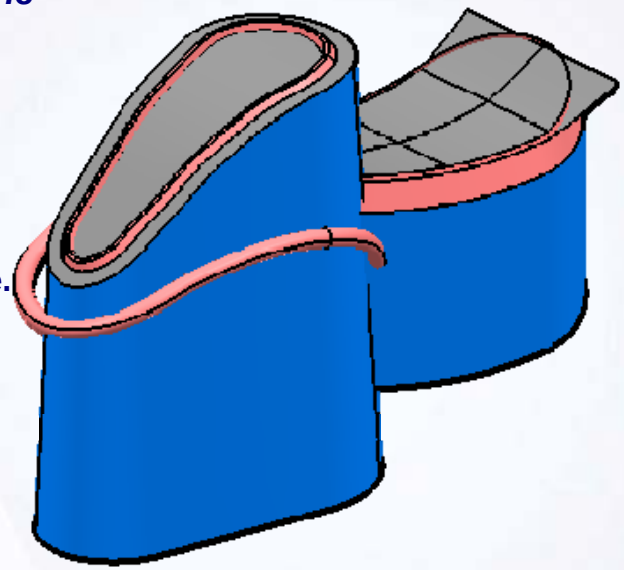
Toy car

45 min

The objective of this step is to create surfaces of 'Plastic Body' part. You will work in part context of the 'Plastic Body' part.

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

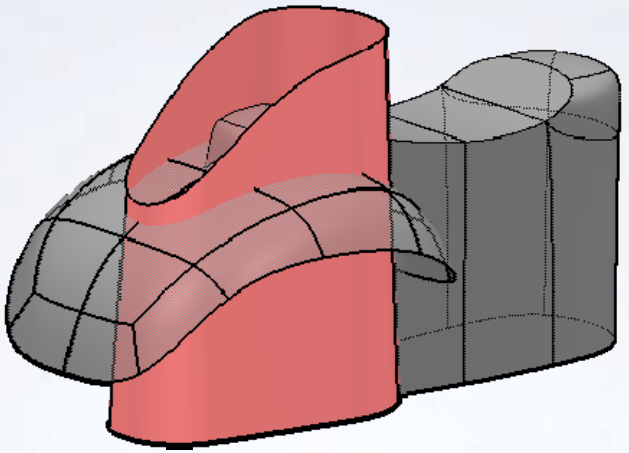
- **Create a simple surface.**
- **Create a drafted surface using a 3D curve.**
- **Create a circular surface using a single guide curve.**
- **Fill the closed contour with a surface.**
- **Extract a curve from a surface.**
- **Create a 3D curve using a support surface.**
- **Create a surface with variable sections.**
- **Create a surface which can adapt to the changing dimensions of the parent profile.**



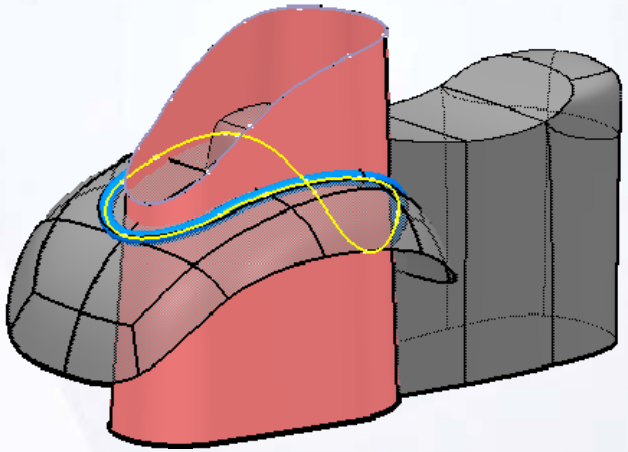
Master Project: Surface Creation (1/4)

Here is the list of tasks to guide you:

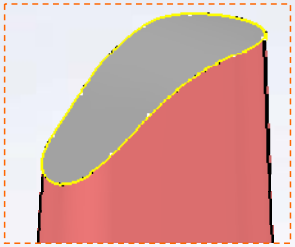
1. **Create a Front Upper Panel surface using the 'Profile' curve created in the previous step.**
   - The surface should have a draft of 3deg with reference to Z axis.
   - The surface has to be relimited using XY plane at the time of creation.



2. **Create a Lower Groove surface using the 'Guide' curve created in the previous step.**
   - The surface should be cylindrical with the radius of 15mm running along the guide curve.

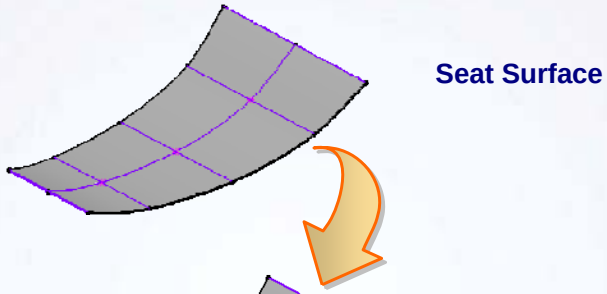


Master Project: Surface Creation (2/4)



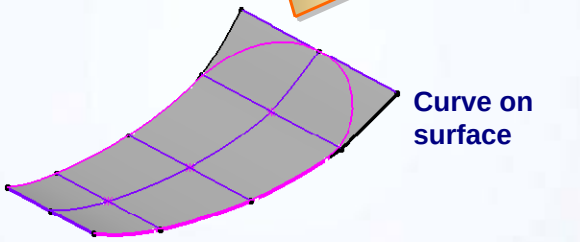
3. Close the top side of the Front Upper Panel.

- Close the 3D profile with the surface.
- The surface should relimit at 3D profile curve.



4. Create a seat surface.

- Use the wireframe created in the previous step.
- The surface should pass through the changing sections along the guide curve.



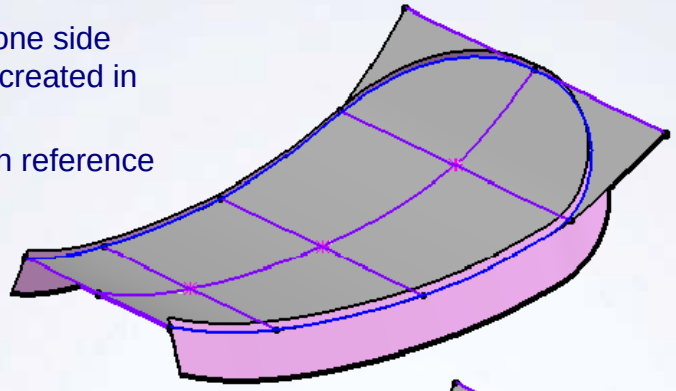
5. Create rounded corner on the rear side of the seat surface.

- Extract a curve from the side edges of multi-section surface as shown.
- Create a curve on surface to form a rounded corner of 160mm radius on both the sides, as shown.
- Smoothen the resultant curve from its discontinuities.

Master Project: Surface Creation (3/4)

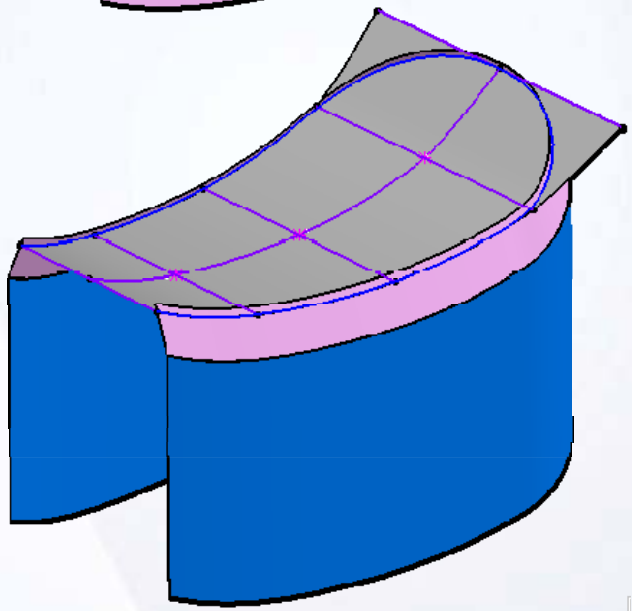
6. Create a Seat Upper Wall surface.

- Create a drafted surface of length 60mm on one side and 10mm on the other side along the curve created in the previous operation.
- The surface should have a draft of 20deg with reference to Z axis.



7. Create a seat wall surface.

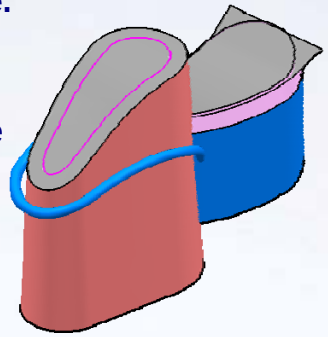
- Create a wall surface along the Seat Upper surface.
- Create a simple straight surface. Use the lower edge boundary of the 'Seat Upper wall' surface to create a vertical seat wall.
- Ensure that the length of the surface is 50mm.



Master Project: Surface Creation (4/4)

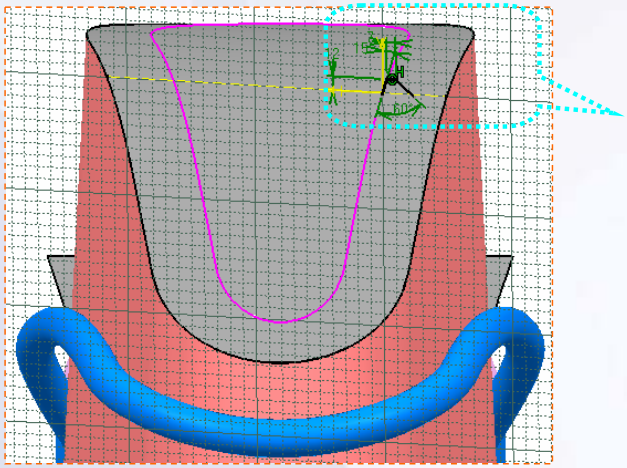
8. Create a guide curve on support to build Upper Groove surface.

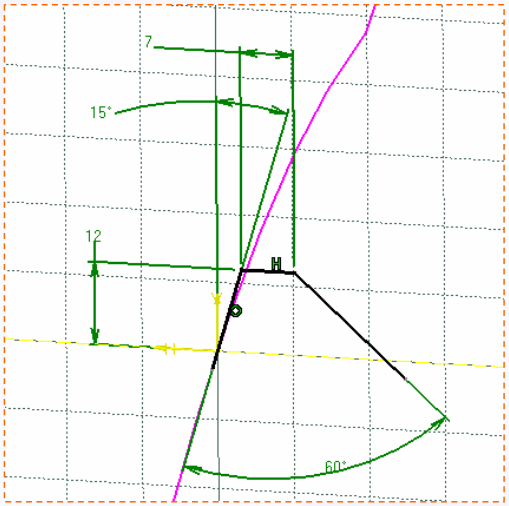
- Create a curve on the top face of the Front Upper Panel surface.
- The surface should be parallel at a distance of 50mm from the 3D Profile curve created in the 'wireframe creation' step.

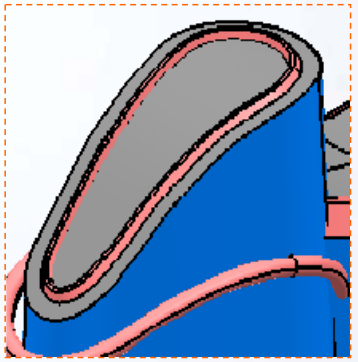


9. Create an Upper Groove surface.

- The surface should adapt to the changing dimensions of the profile along the guide curve.
- The profile should be free from negative drafts.



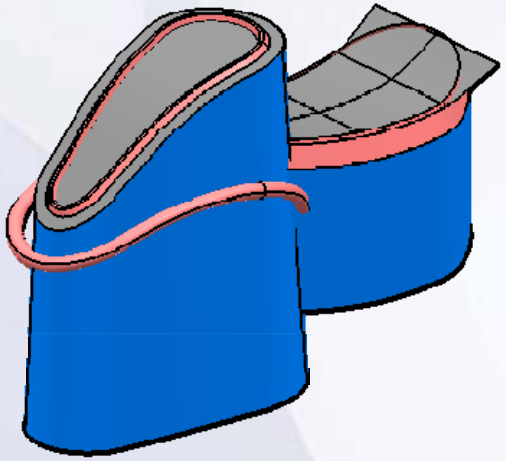


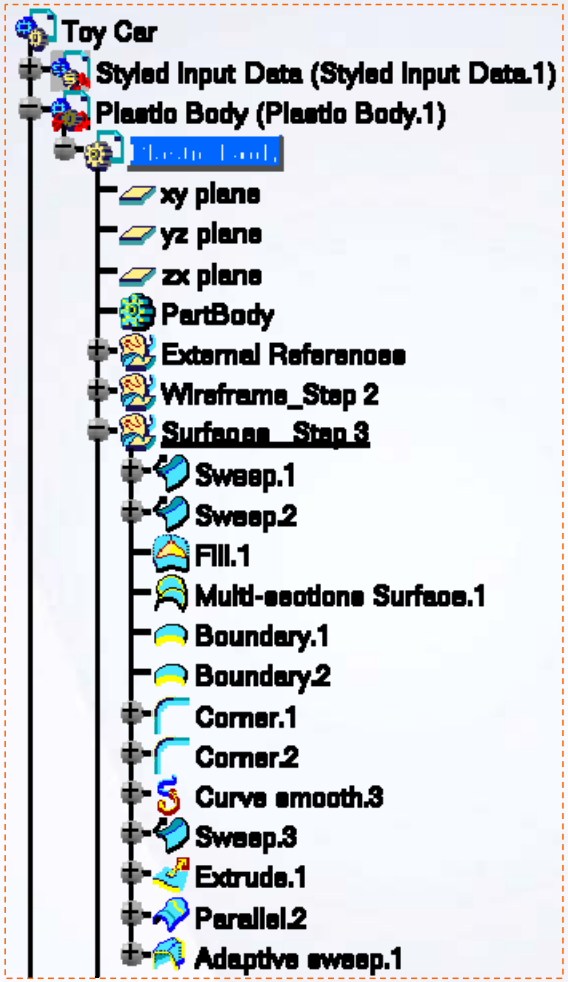


10. Save the Part Plastic_Body.CATPart using Save Management.

Master Project: Surface Creation Recap

- ✓ Create a simple surface.
- ✓ Create a drafted surface using a 3D curve.
- ✓ Create a circular surface using a single guide curve .
- ✓ Fill the closed contour with a surface.
- ✓ Extract a curve from a surface.
- ✓ Create a 3D curve using support surface.
- ✓ Create a surface with variable sections.
- ✓ Create a surface which can adapt to the changing dimensions of the parent profile.





Master Project: Re-limitation and Connection

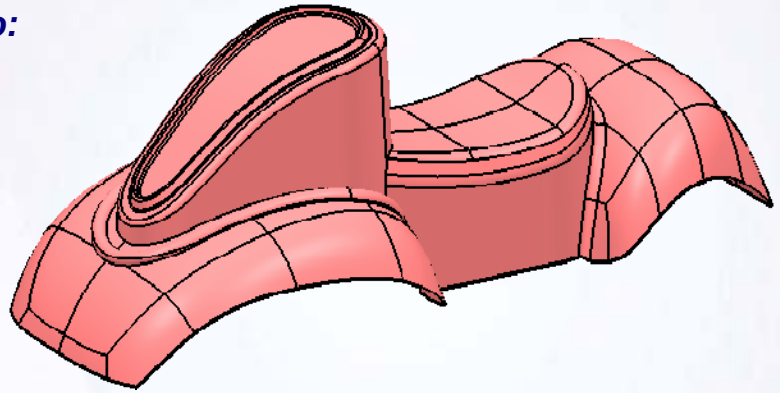
Toy Car

30 min

The objective of this step is to relimit and connect the surfaces created in the previous steps. You will work in the 'Plastic Body' Part.

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

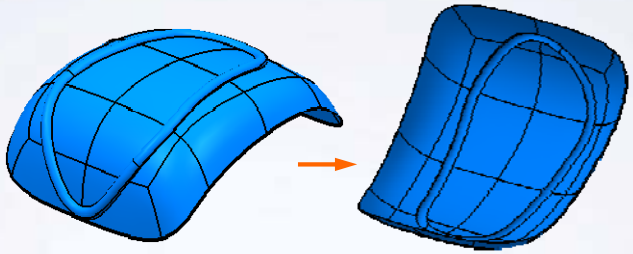
- **Trim the surfaces**
- **Split the surfaces into separate geometries**
- **Create a blend between two surfaces**
- **Join the surfaces**



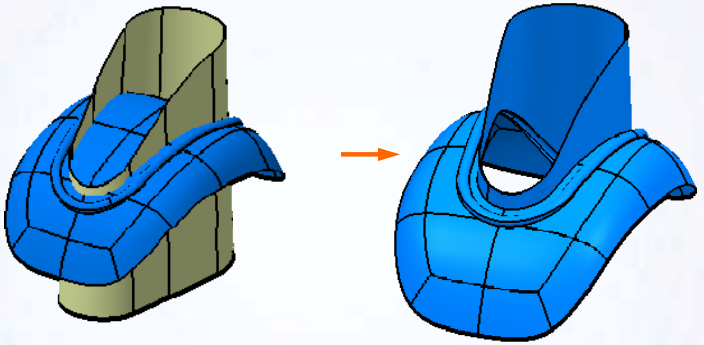
Master Project: Surface Re-limitation and Connection (1/4)

In the previous step you have created the surfaces required to build a Plastic Body part. Now you will finalize the surfaces by relimiting and connecting them with each other.

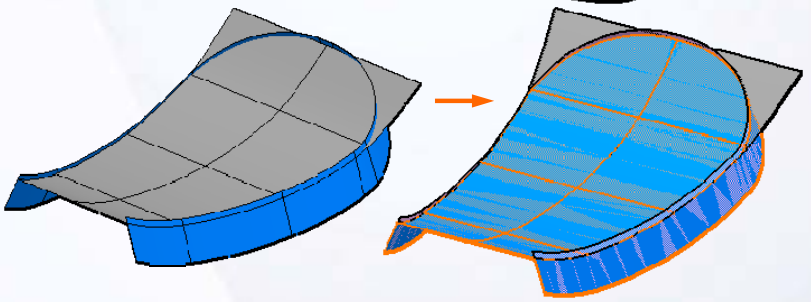
1. **Create a Trim between Front Panel surface and Lower Groove surface.**



2. **Create a Trim between the resultant surface and Front Upper Panel surface.**

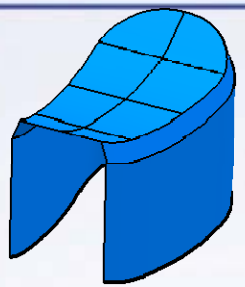


3. **Create a Trim between Seat surface and Seat Upper wall surface.**

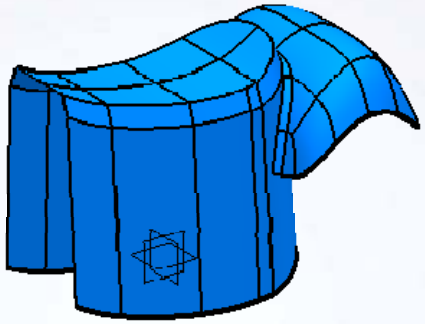


Master Project: Surface Re-limitation and Connection (2/4)

4. **Join the resultant surface of the previous operation, Upper wall surface and Seat wall surface.**

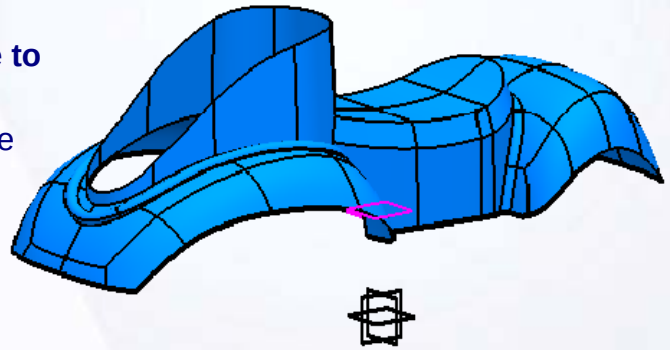


5. **Create a Fillet of radius 15mm between the resultant surface and the rear panel surface of the Toy car.**



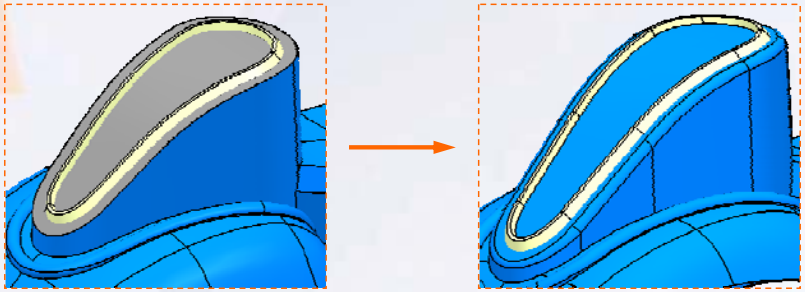
6. **Create a Trim between the resultant surface of the second and fifth operation.**

7. **Split the resultant surface with the Floor Upper face to rest the surface on the floor.**
   - Create a reference plane at 240mm from XY plane towards Z+ direction.
   - Relimit the resultant surface using this plane.

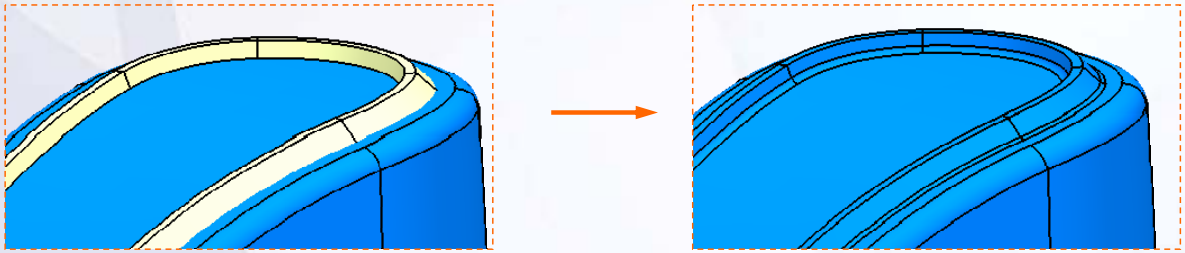


Master Project: Surface Re-limitation and Connection (3/4)

8. Create a Fillet of radius 15mm between the resultant surface and the top surface of the Front Upper Panel.

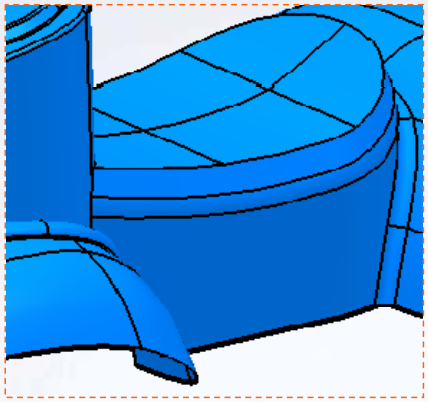


9. Create a Fillet of radius 5mm between the resultant surface and the Top Groove surface.

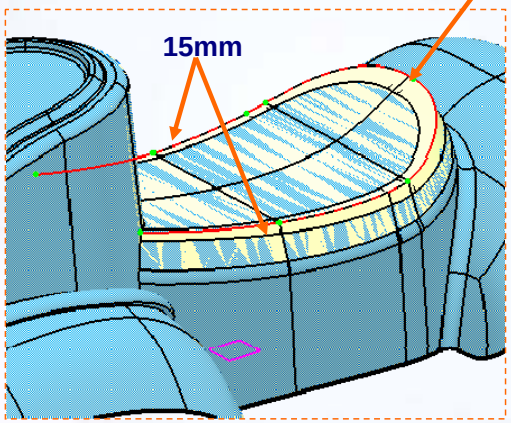


Master Project: Surface Re-limitation and Connection (4/4)

10. Create a Fillet of radius 50mm on the sharp edge of the Seat surface.



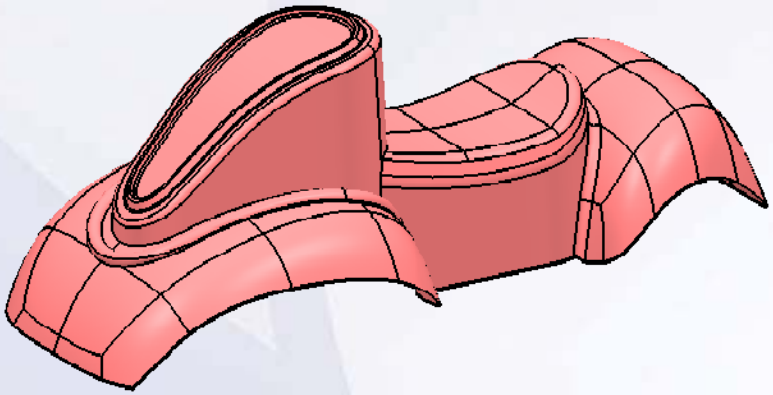
11. Create a Variable Fillet of radius 15mm to 25mm on the upper edge of the seat.

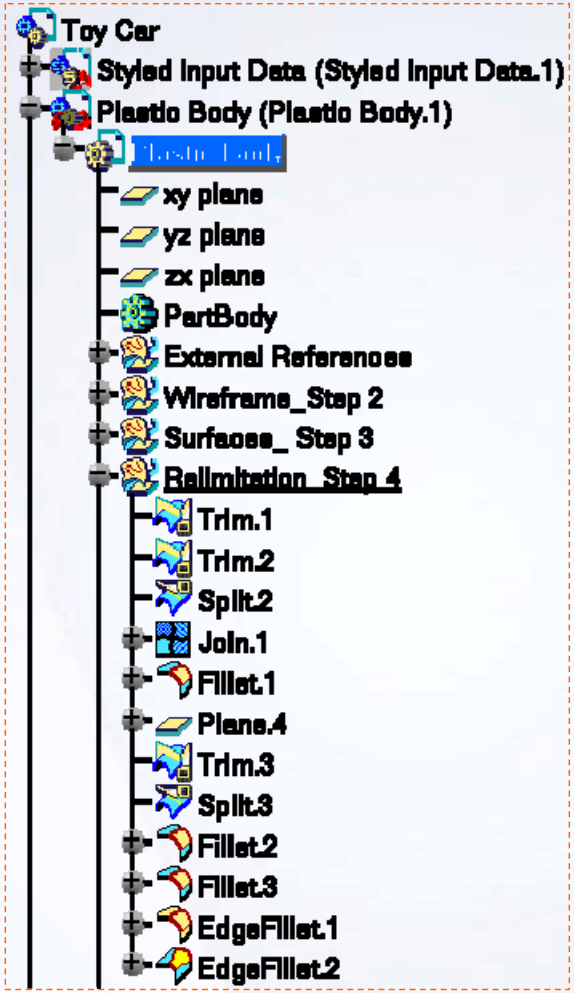


12. Save the Part Plastic_Body.CATPart using Save Management.

Master Project: Surface Re-limitation and Connection Recap

- ✓ Trim the surfaces.
- ✓ Split the surfaces into separate geometries.
- ✓ Create a blend between two surfaces.
- ✓ Join the surfaces.





Master Project: Surface Analysis

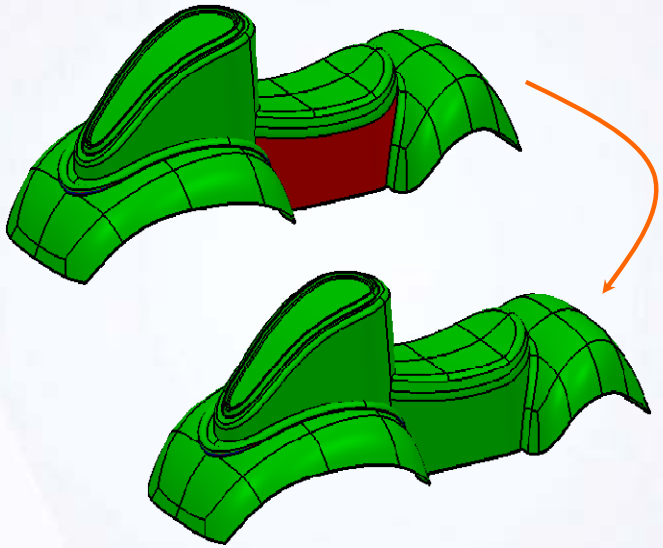
Toy Car

15 min

The objective of this step is to analyze the surfaces for its moldability. You will work in part context of 'Plastic Body' Part.

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

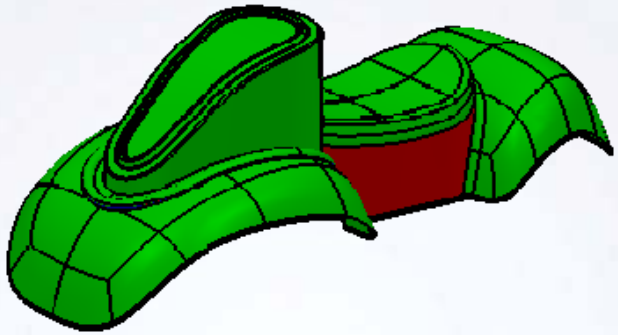
- **Perform Draft analysis.**
- **Identify erroneous surfaces and repair them.**
- **Replace an existing surface with a new surface.**



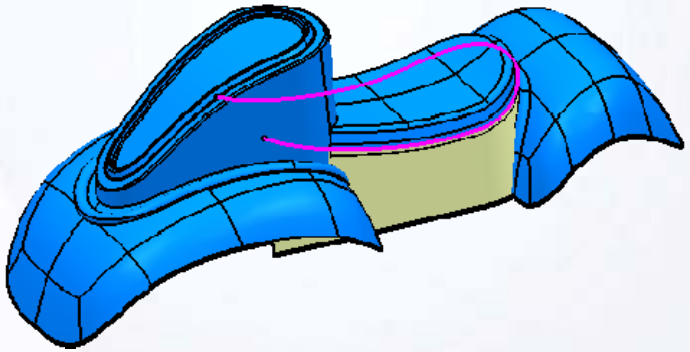
Master Project: Surface Analysis (1/2)

In the previous step you have created the final surface of a Plastic Body part. You have relimited and connected the surfaces to create a single surface for one piece moldability. Now you will analyze the surface for its draft and ensure that the part has sufficient draft to be removed from the mold.

1. **Analyze the surface to detect the negative and zero draft.**
   - You will detect the surface with zero draft in red.
   - You need to replace this surface with a new drafted surface.

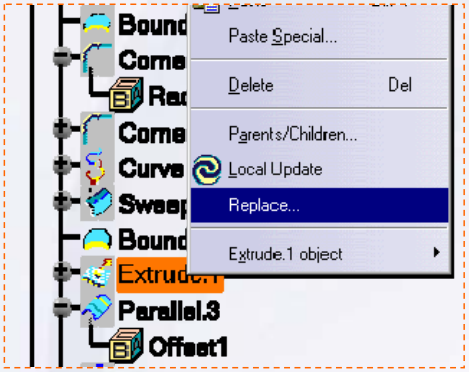


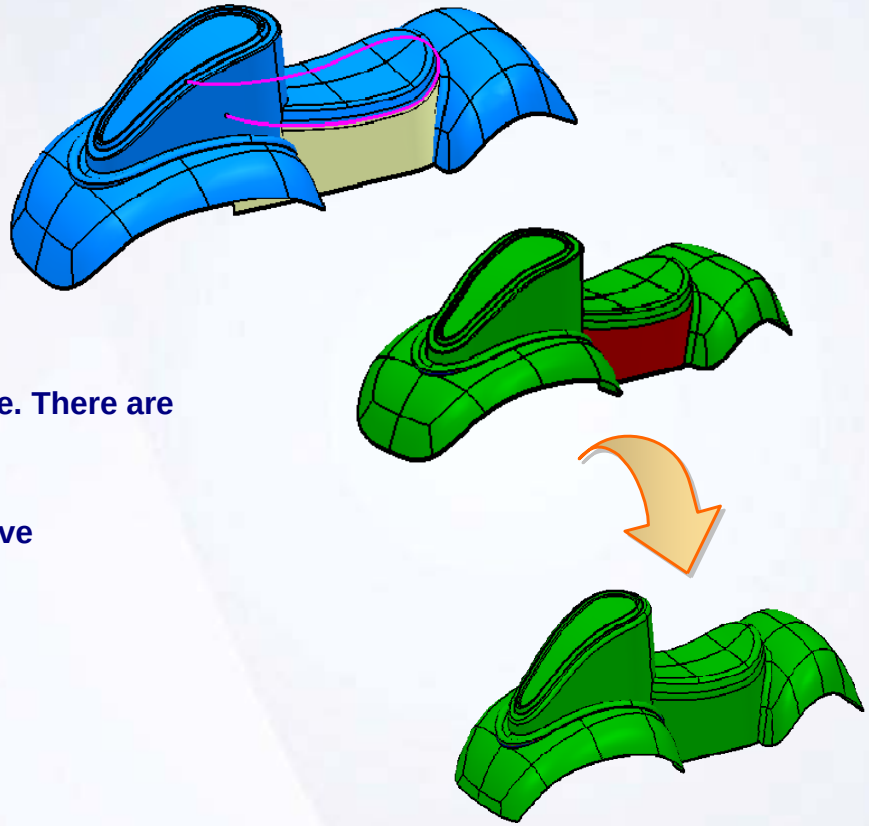
2. **Create a surface of 3deg draft with reference to Z axis using inputs of seat side wall surface.**



Master Project: Surface Analysis (2/2)

3. Replace the defective surface with the new drafted surface.



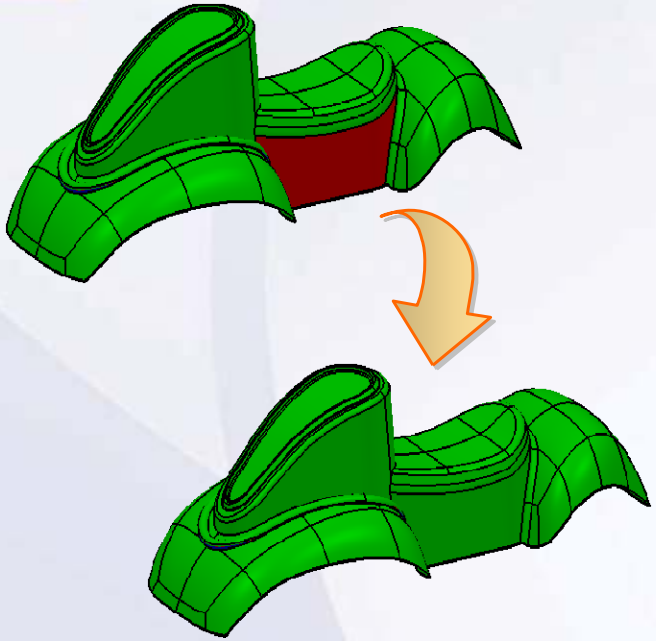


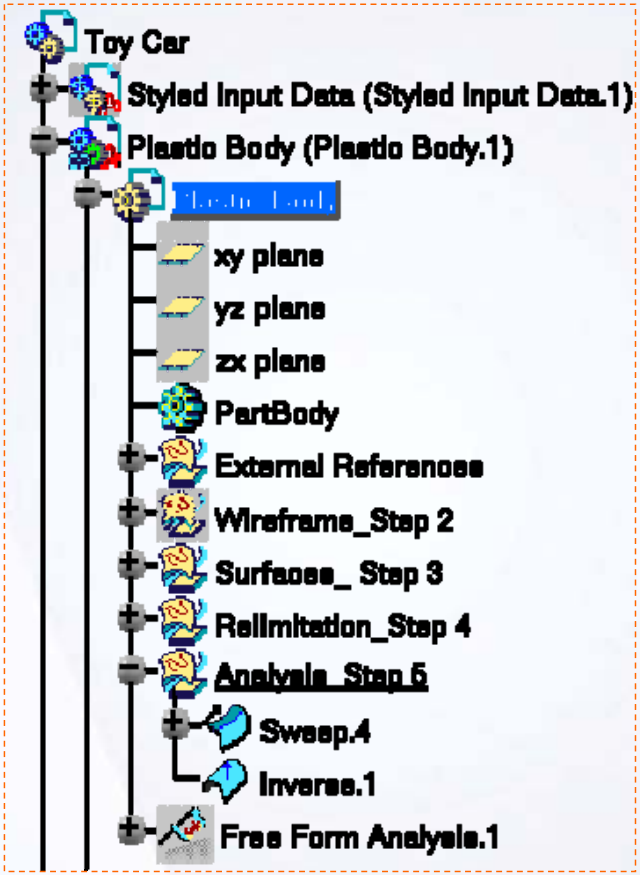
4. Repeat the Draft analysis on the surface. There are no defects on the surface.

5. Save the Part Plastic.CATPart using Save Management.

Master Project: Surface Analysis Recap

- ✓ Perform Draft analysis
- ✓ Identify erroneous surfaces and repair them.
- ✓ Replace an existing surface with a new surface.





Master Project: Create a Steering Wheel Surface

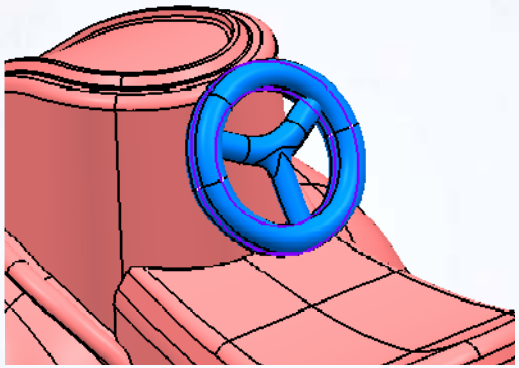
Toy Car

15 min

The objective of this step is to create wireframe and surfaces for Steering Wheel. You will work in product context of 'Toycar.CATProduct'.

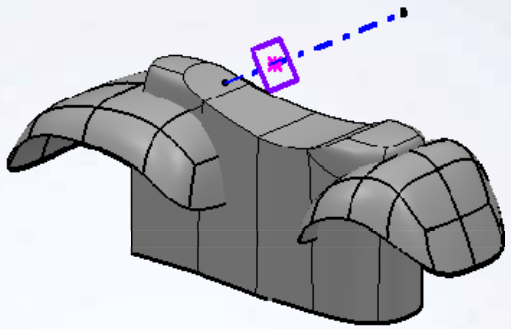
By the end of this exercise you will be able to:

- **Create simple surfaces and wireframe.**
- **Use pre-defined features of surface design workbench.**
- **Create a blend between two surface.**
- **Relimit between two surfaces.**
- **Publish and reuse the elements in a part.**



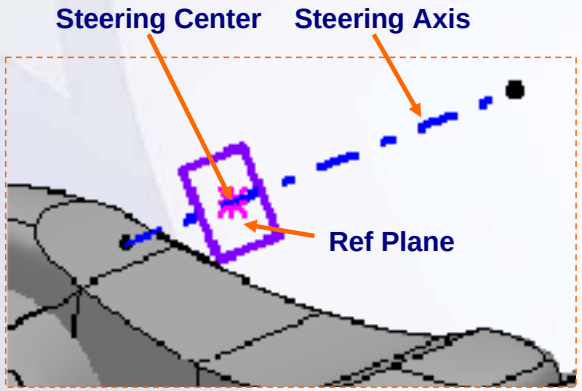
Master Project: Create a Steering Wheel Surface (1/4)

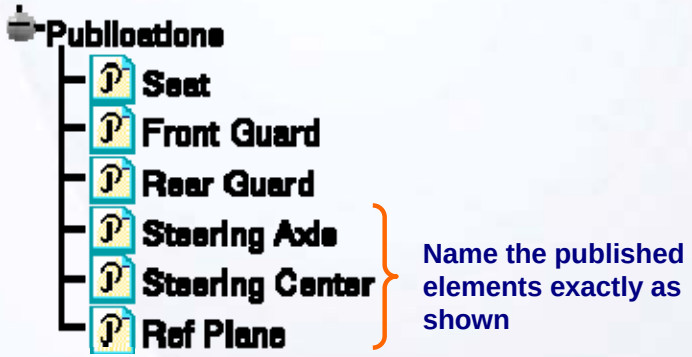
Here is the list of tasks to guide you:
Now refer to the Toycar.CATProduct you have opened at the start of the exercise.
Reopen it if it is closed.



1. **Study the Input styled data to understand the inputs provided by the stylist to create steering wheel surfaces.**
   - You will find that the input data consists of the following data required to create a steering wheel:
     a. Steering Axis
     b. Steering Centre
     c. Ref plane

2. **Publish these geometries in the input styled data for further reference to design the steering surfaces.**



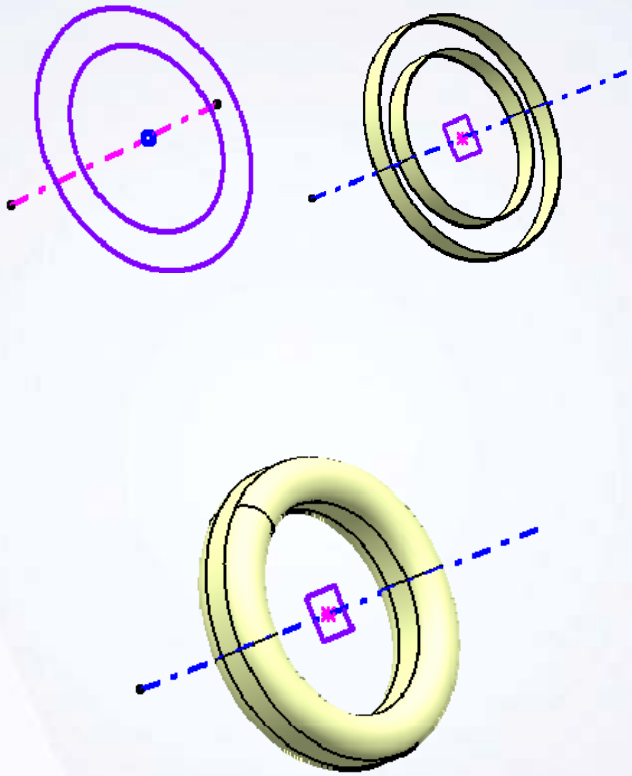


Master Project: Create a Steering Wheel Surface (2/4)

Create the steering wheel wireframe and surface into Steering_Wheel.CATPart in product context.

1. **Create the inner and outer wall surface of a wheel.**
   - Create the inner and outer wall curves of radius 82mm and 115mm respectively on the published plane of the input data.
   - When a published plane is selected, the published plane gets copied into the Steering wheel.CATPart in a Geometrical Set called 'External References'. This will create a contextual link between the plane and steering wheel geometry.
   - Create a straight wall surface of 10mm length on both the sides of the curve.

2. **Connect these surfaces.**
   - The connecting surface should be tangent continuous with the wall surfaces.
   - The surface should be flexible to modify the shape for better gripping of the steering wheel.
   - The surface should be aesthetically good.



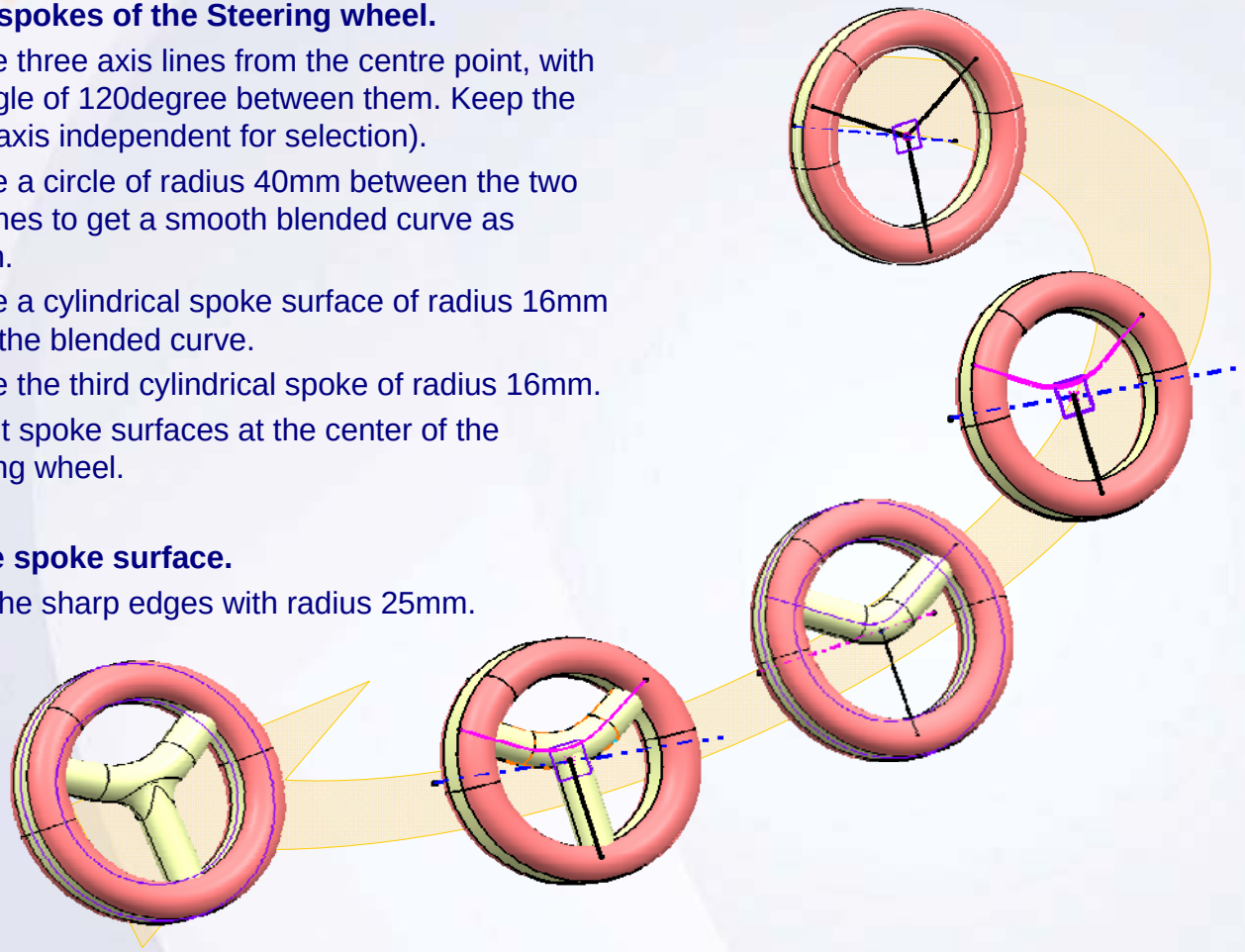
Master Project: Create a Steering Wheel Surface (3/4)

3. Create the spokes of the Steering wheel.

- Create three axis lines from the centre point, with an angle of 120degree between them. Keep the three axis independent for selection).
- Create a circle of radius 40mm between the two axis lines to get a smooth blended curve as shown.
- Create a cylindrical spoke surface of radius 16mm using the blended curve.
- Create the third cylindrical spoke of radius 16mm.
- Relimit spoke surfaces at the center of the steering wheel.

4. Finalize the spoke surface.

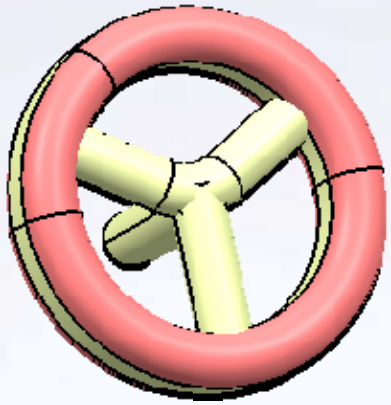
- Fillet the sharp edges with radius 25mm.



Master Project: Create Steering Wheel Surface (4/4)

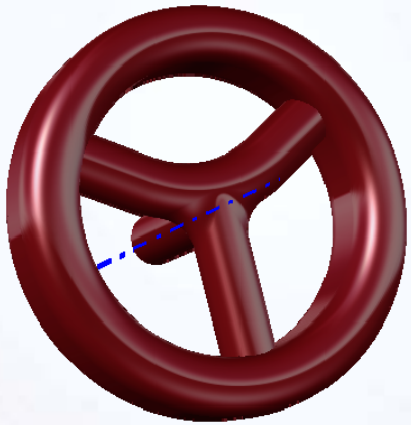
5. Create the spindle surface.

- The diameter of the spindle should be 15mm.
- The upper end of the spindle should merge into the spoke surface.



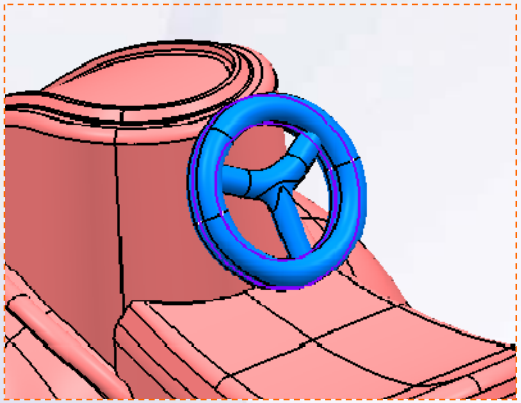
6. Finalize the steering surface.

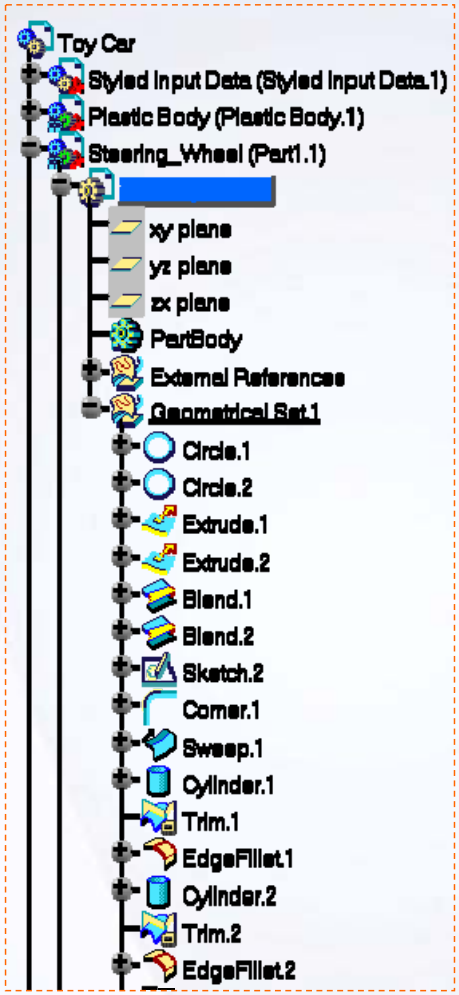
- Relimit the spindle surface and the spoke surface.
- Blend the sharp edges between the spindle and spoke surfaces with radius 25mm.



Master Project: Create Steering Wheel Surface Recap

- ✓ Create simple surfaces and wireframe.
- ✓ Use pre-defined features of surface design workbench.
- ✓ Create a blend between two surface.
- ✓ Relimit between two surfaces.
- ✓ Publish and reuse the elements in a part.





Master Project: Multi-Model Links

Toy car

15 min

The objective of this step is to replace the styled input data with its new version provided by a stylist. You will work in the 'Toycar.CATProduct'.

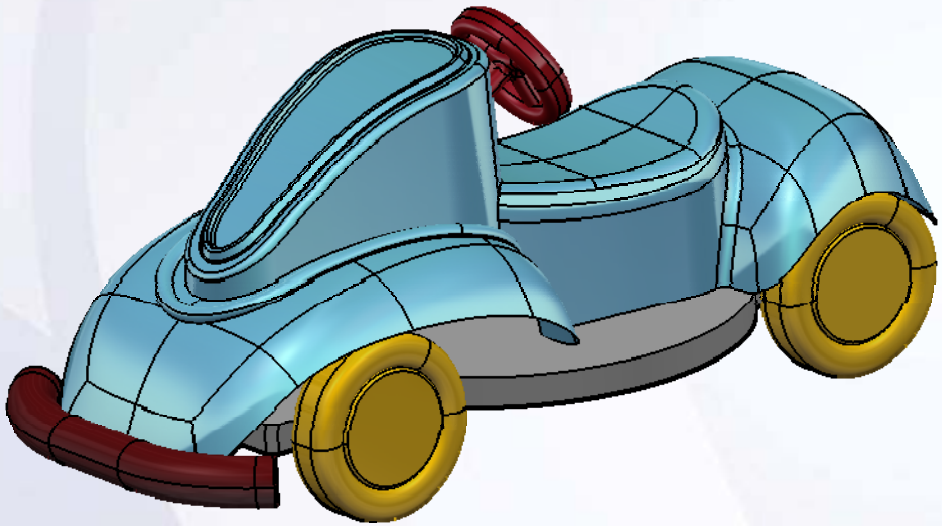
By the end of this exercise you will be able to:

- Replace the data in a multi-model environment
- Update the model with new inputs

Master Project: Multi- Model Links

You have successfully completed the modeling of different parts of the toy car. You have used the input styled data provided by the stylist to generate the mechanical surface in Generative Shape Design workbench.

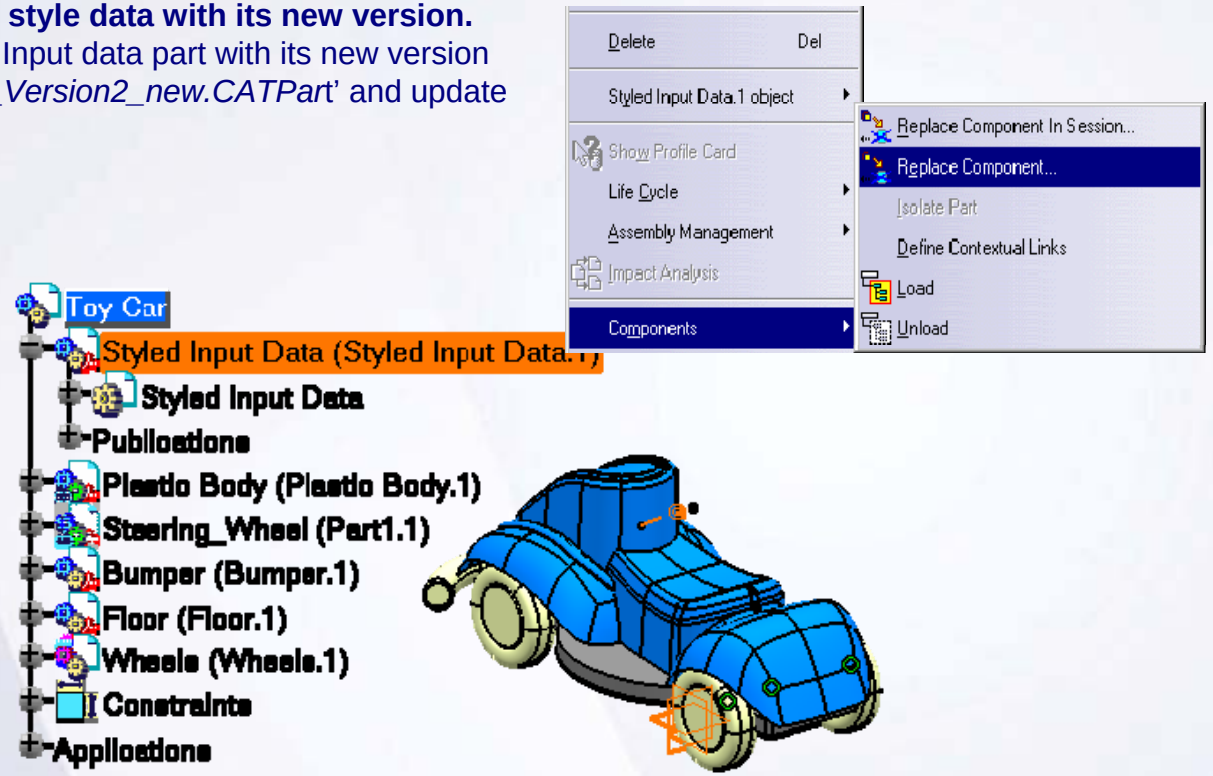
At the same time, the surfaces and the wireframes created by you are flexible to adapt the style changes if any, made by the stylist in the conceptual design. You have taken care of the multi-model links by using the published elements to design your parts. In the next step you will update your model with new versions of the styled data.



Master Project: Multi-Model Links (1/2)

The Stylist updates the styled data based on the feedback and market requirements.
Here the stylist has changed the front panel and the rear panel. The inclination angle of the steering axis has changed.
Let us update our design with the new version of styled data.

1. **Replace the Input style data with its new version.**
   - Replace the Input data part with its new version '*Input_Data_Version2_new.CATPart*' and update the part.



Master Project: Multi-Model Links (2/2)

1. **Update the product 'Toy Car.CATProduct' after replacing the input styled data.**
   - Update the main assembly to propagate the changes in all the parts.

Master Project: Multi-Model Links Recap

- ✓ Replace the data in a multi-model environment
- ✓ Update the model with new inputs

Shortcuts

F1	Link to on-line documentation
Shift F1	Contextual help for an icon
Shift F2	Overview of the specification tree
F3	Hide/Show the specification tree
Ctrl + Tab	Change CATIA V5 window
Ctrl N	New file
Ctrl O	Open file
Ctrl S	Save file
Ctrl P	Print
Ctrl Z	Undo
Ctrl Y	Redo
Ctrl C	Copy
Ctrl V	Paste
Ctrl X	Cut
Ctrl U	Update
Ctrl F	Find

Ctrl + several selections	Multiple selection
Shift + 2 selections	Selection of all elements between and including the 2 selected elements
Alt F8	Macros
Alt F11	Visual Basic editor
Alt + Enter	Properties
Alt + MB1	Pre-selection Navigator
Ctrl F11	Pre-selection Navigator
Up/Down or Left/Right arrow	Pre-selection Navigator
Shift + MB2	Local zoom and change of viewpoint
Shift + manipulation with compass	Displacement respecting constraints

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Glossary (1/2)

Angular Threshold is maximum angle below which two elements are considered as only one element.

Boundary Mode Connect Checker Analysis The connection analysis is performed between the boundary of elements.

Closing Point is the endpoint of a closed section.

Coupling Point are the connecting points used to compute the segmentation on the surface.

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

G0 Analysis The distance between the vertices of the surfaces is measured. If the distance is less than 1 micron, CATIA surfaces are considered continuous in Point.

G1 Analysis The angle between the surfaces at the connection is measured. If the angle is less than 0.5 deg, CATIA surfaces are considered continuous in Tangency.

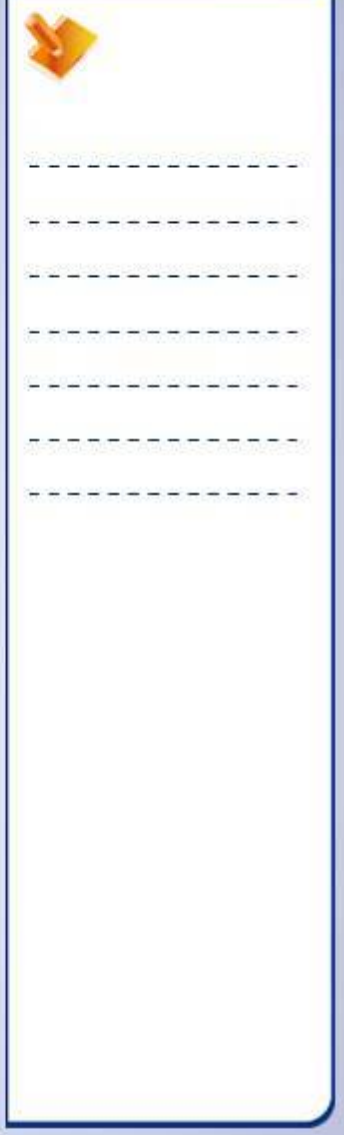
G2 Analysis The Curvature (1/R) difference at the connecting edges of the surfaces is measured. The curvature difference is measured using a formula and is expressed in percentage. The discontinuity range is between 0 – 200 %. Lesser the percentage, better the surface connection.

Merging Distance is the maximum distance below which two elements are considered as only one element.

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 used to define the pulling direction.

Pulling Direction defines the direction from which the draft angle is measured. It derives its name from the direction in which the mold is pulled to extract the molded part.

Projection Mode Connect Checker Analysis The connection analysis is performed between the boundary of one element and the projection of that boundary on another element.



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Glossary (2/2)

Property is an attribute such as color or a name that can be assigned to any feature. All features can be customized in both appearance and function.

Topology is the representation of geometry as seen by CAD system.

Work Object is the current state of the part during its design phase. This can be any feature in the part specification tree and is used to be able to insert a feature at a given point in the part history. It can be identified in the specification tree by its name, which is underlined.

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