

CATIA V5 Foundations for Body Designers



See what you mean

CATIA V5 Foundations for Body Designers

Student Handbook

Version 5 Release 19



56 Hours

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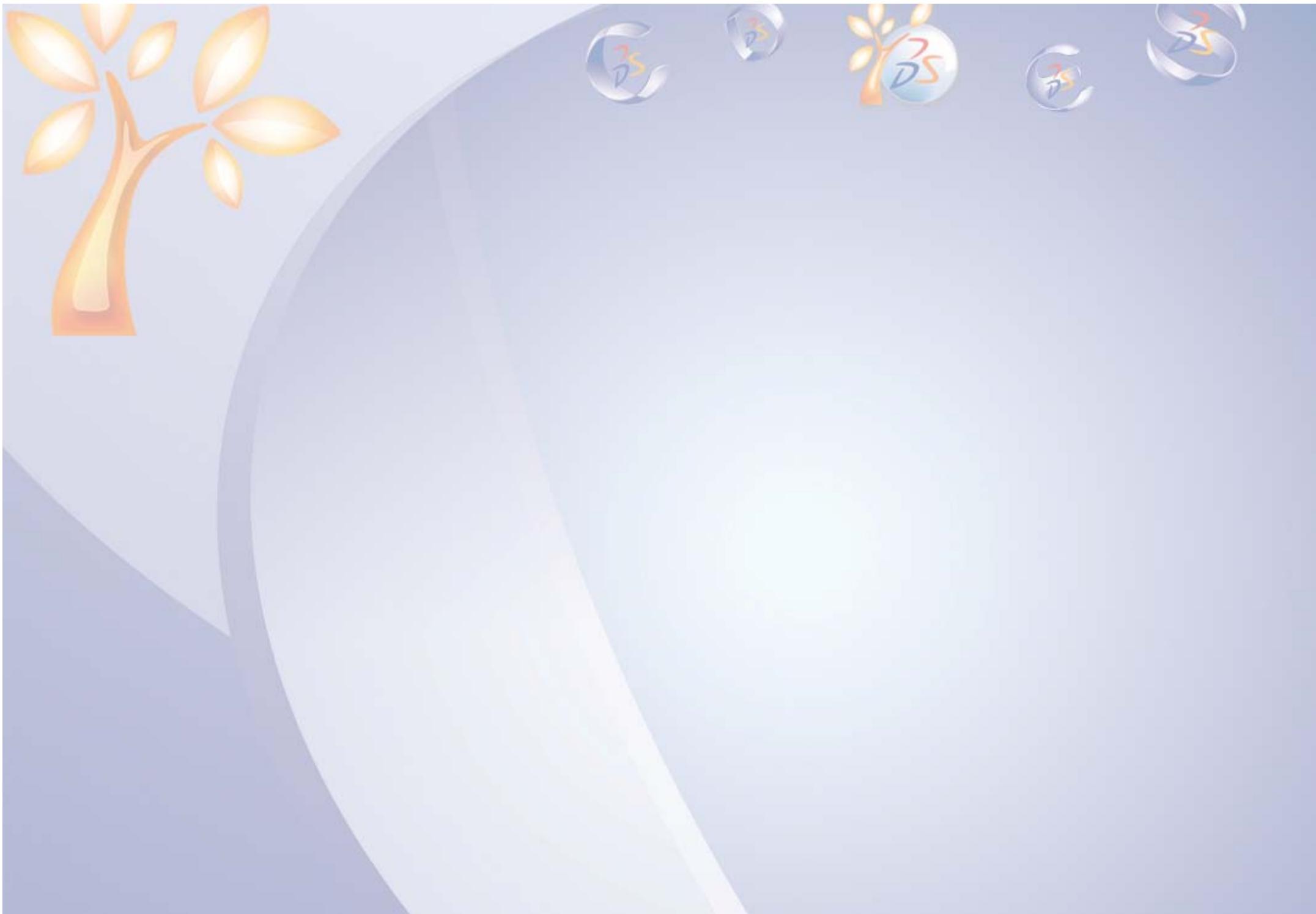
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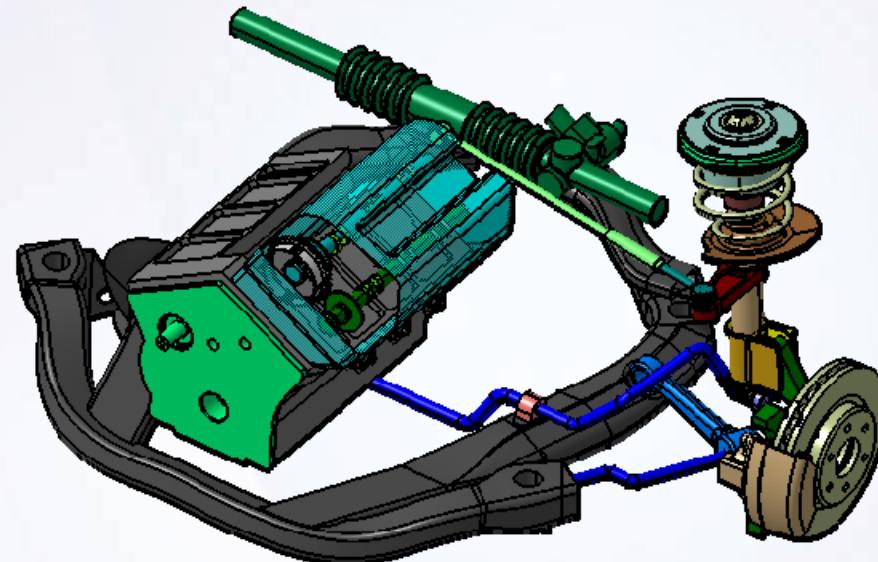
Introduction to CATIA

1

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Understand the CATIA software
- ✓ Open CATIA
- ✓ Understand the CATIA Interface



4 Hours

Case Study

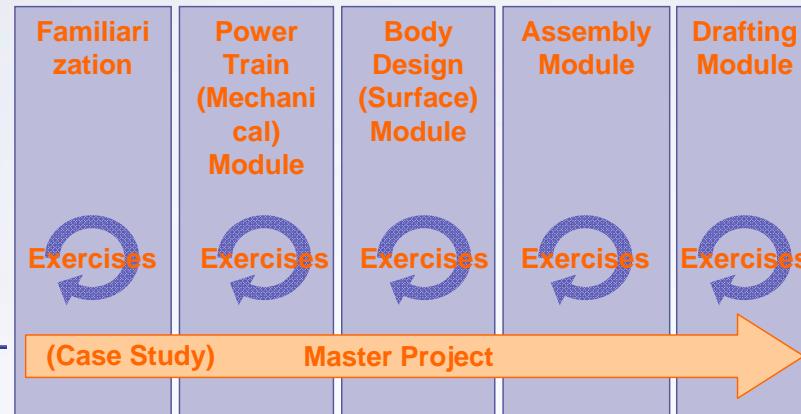
Each lesson in this course contains a case study, which explains the skills and concepts covered in the lesson. The case study will be described at the beginning of each lesson, and the student will be able to do the case study exercise once the theory for that lesson has been covered.

All models used in the case studies come from the Front Suspension and Engine Assembly.

Design Intent

Each case study contains a set of model requirements, known as the design intent. The first case study does not contain a design intent because you are not going to design anything. However, by the end of this lesson you should be able to:

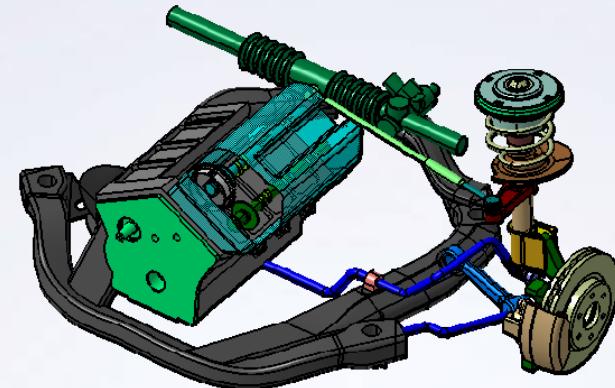
- ✓ Define key terms in CATIA.
- ✓ Identify and describe the design intent.
- ✓ Change the orientation of a model.
- ✓ Change the visualization properties of a model.
- ✓ Manipulate the specification tree.
- ✓ Access the Help system of CATIA.



Stages in the Process

Each lesson consists of steps. You will go through the following steps to introduce yourself to CATIA:

1. Understand the CATIA software.
2. Open CATIA.
3. Understand the CATIA interface.

A yellow puzzle piece icon in the top left corner of a large empty box. The box contains six horizontal dashed lines for notes.

Understand the CATIA software

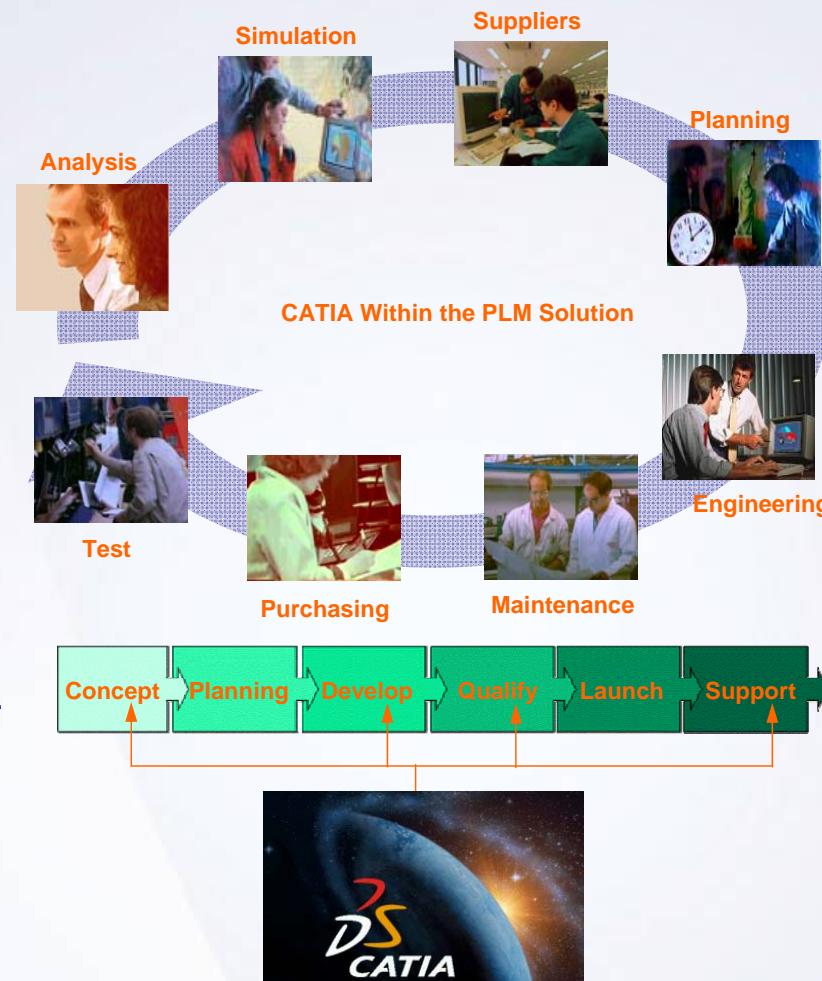
CATIA is a mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations to capture the design intent.

- ✓ CATIA acts as the backbone for concept, product definition, manufacturing, simulation, and after-market information found within various lifecycle stages of a product.
- ✓ It provides the specifications and geometrical data related to a product across several lifecycle phases.

Open CATIA.

In a Windows environment, you can start the CATIA application in several ways:

- A. Select CATIA from the Start > Programs > CATIA menu.
- B. Double-click the CATIA icon on your Windows desktop.
- C. Double-click on an existing CATIA document.

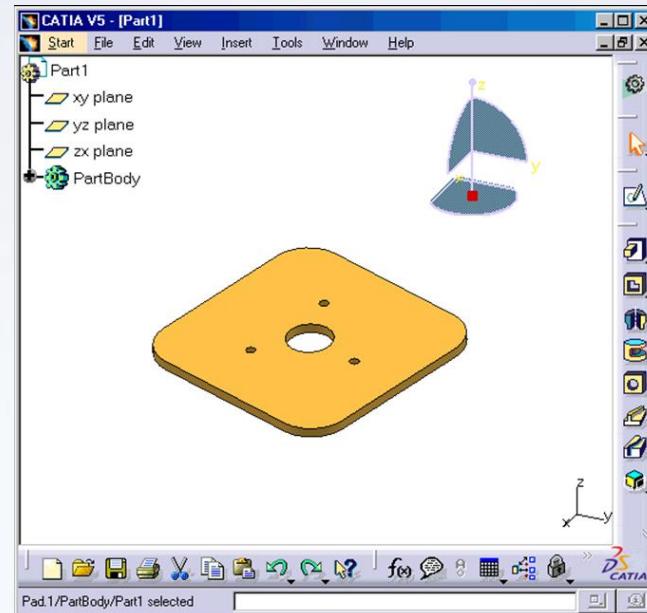


Understand the CATIA interface

CATIA V5 is specifically designed for the Windows operating environment, and it behaves in the same manner as other Windows applications. Traditional menus provide access to all the CATIA commands. Toolbars contain icons for quick access to the most frequently used commands.

CATIA's user interface adopts the Windows interface, and contains the following key features:

- A. Separate workbenches and their respective toolbars.
- B. Easy navigation from one workbench to another.
- C. Standard and specific menus & toolbars (File, Edit, Insert...).
- D. Standard manipulations (Copy-Paste, Drag-and-Drop, Edit in Place...).
- E. Intuitive (highlighting, copilot, pointer shapes...).
- F. Multi-document support.
- G. Contextual menu (MB3) support.
- H. Specification tree, which includes technological features, constraints, and relationships.

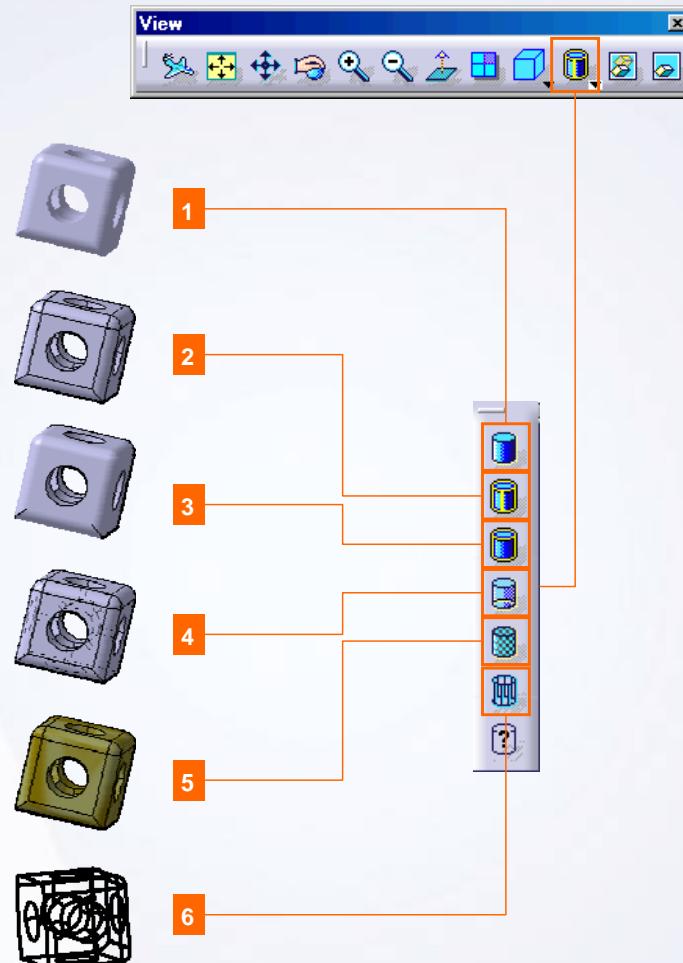


View Tools

Rendering Styles

CATIA has the ability to apply different styles of rendering to visualize the geometry and provide more clarity to the model.

- 1 Shading (SHD)
- 2 Shading with Edges
- 3 Shading with Edges without smooth Edges
- 4 Shading with Edges with Hidden edges
- 5 Shading with Material
- 6 Wireframe (NHR)



Main Tools

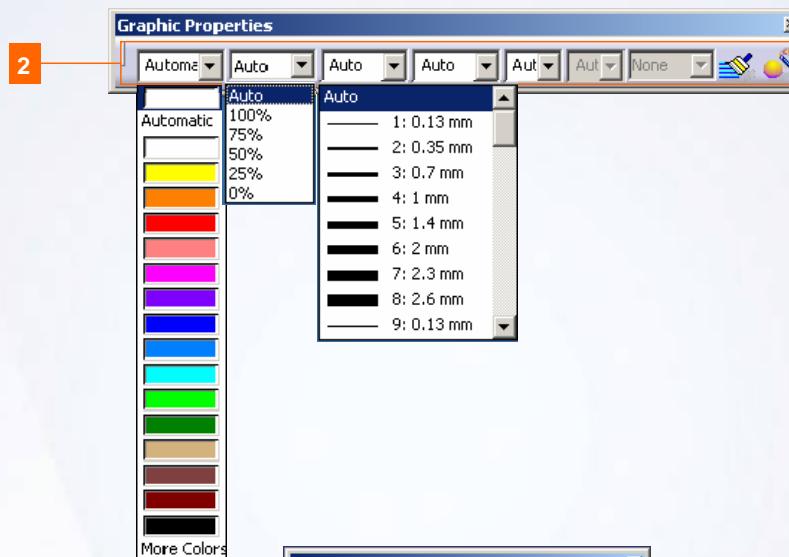
Standard

- 1 **New:** To create a new document with a particular file type.



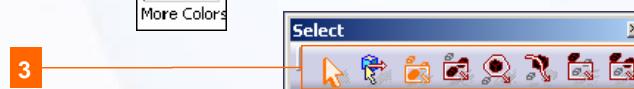
Graphic Properties

- 2 **Graphic Properties:** To change the various graphic properties (Fill color, Transparency, Line thickness, Line type, Point symbol) of the elements as they are displayed on the screen.



Select

- 3 **Select:** To select objects.



View

- 4 **Hide/Show:** To change the hide/show state of the components.



Case Study: Introduction to CATIA

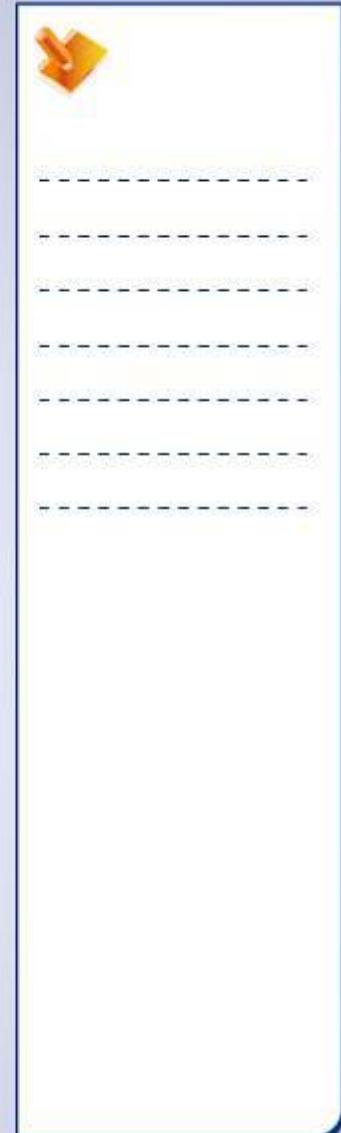
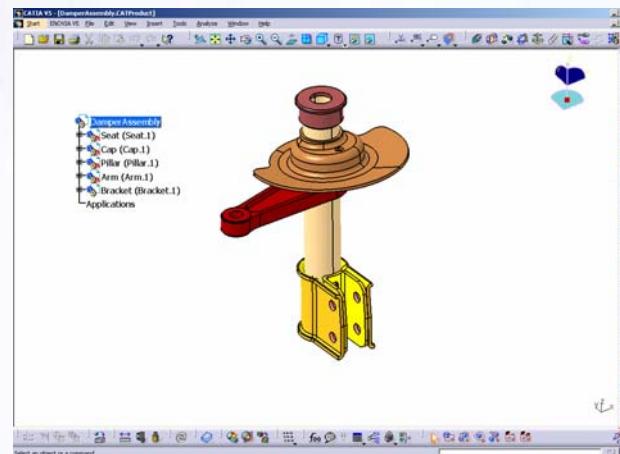
Recap Exercise



20 min

In this exercise you will review the Damper assembly. With the knowledge you have gained in this lesson, you should be able to:

- ✓ Change the orientation of the model
- ✓ Change the visualization properties of the model
- ✓ Manipulate the specification tree
- ✓ Access the CATIA help system



Case Study (1/6)

1. Load the file.

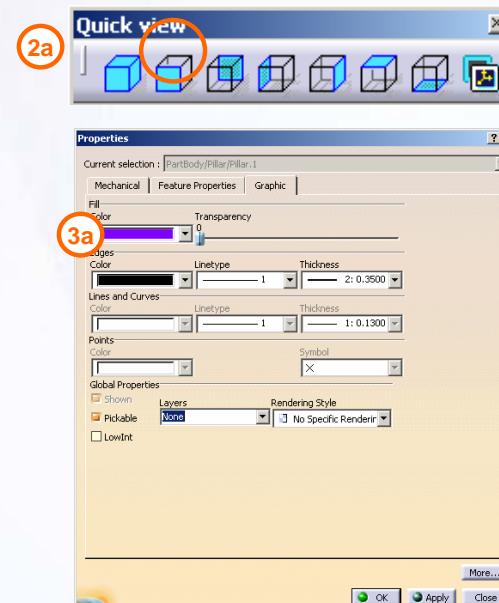
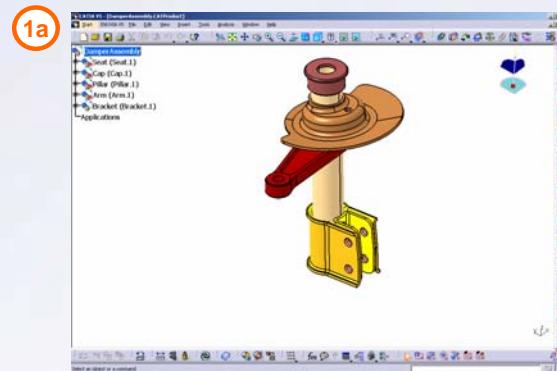
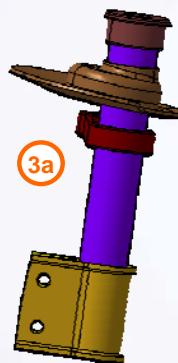
- Load DamperAssembly.CATProduct in CATIA.

2. Change the orientation of the assembly.

- Change the model orientation to Front.
- Zoom in on the area as shown.

3. Change the visualization properties.

- Select the front face of the Pillar part (as shown) and change its color to magenta.
- It is recommended not to use red, orange and green colors for geometries. By default, these colors are used by various CATIA diagnoses.



Case Study (2/6)

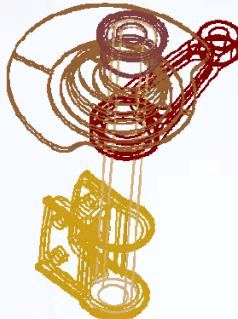
4. Zoom out on the model.

5.



5. Change to the Isometric View.

6.



6. Change the rendering style to Wireframe.

7.



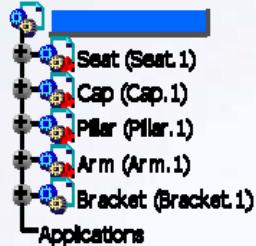
7. Change the rendering style to Shading with Edges.

Case Study (3/6)

8. Change the display of the specification tree.

- a. Press the <F3> key to toggle the specification tree on and off.
- b. Click on one of the branches of the specification tree and notice that the model darkens.
- c. Try zooming out; notice that the specification tree is being manipulated and not the model.
- d. Press <Shift> + <F3> to re-activate the model.

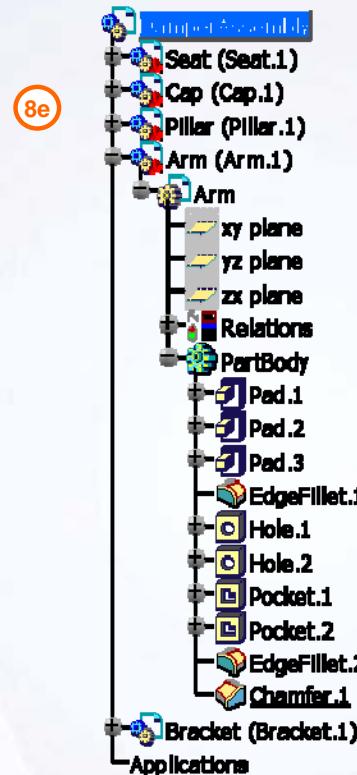
8b



Case Study (4/6)

8. Change the display of the specification tree (continued)

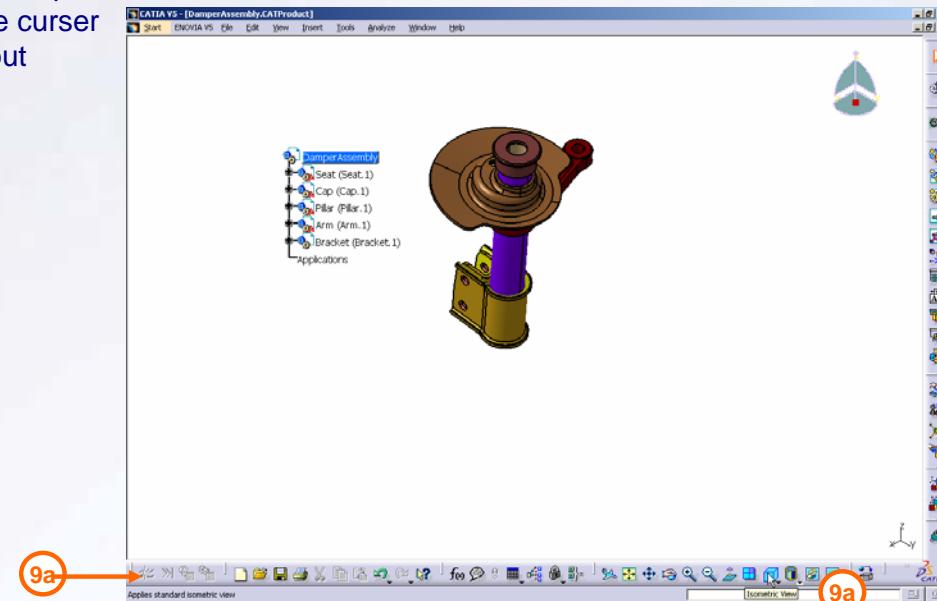
- e. Expand the Arm node of the tree.
Notice that the features of the part are now displayed in the tree.
- f. Collapse all the nodes to show only the top level of the tree.



Case Study (5/6)

9. Review the areas of information.

- a. Review the ToolTip and Short Help messages when you place the cursor over a command's icon (without selecting it).

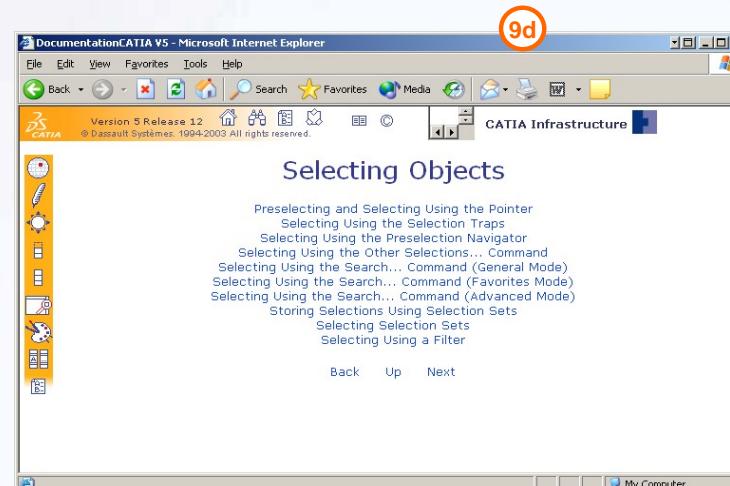


Case Study (6/6)

9. Review the areas of information (continued).

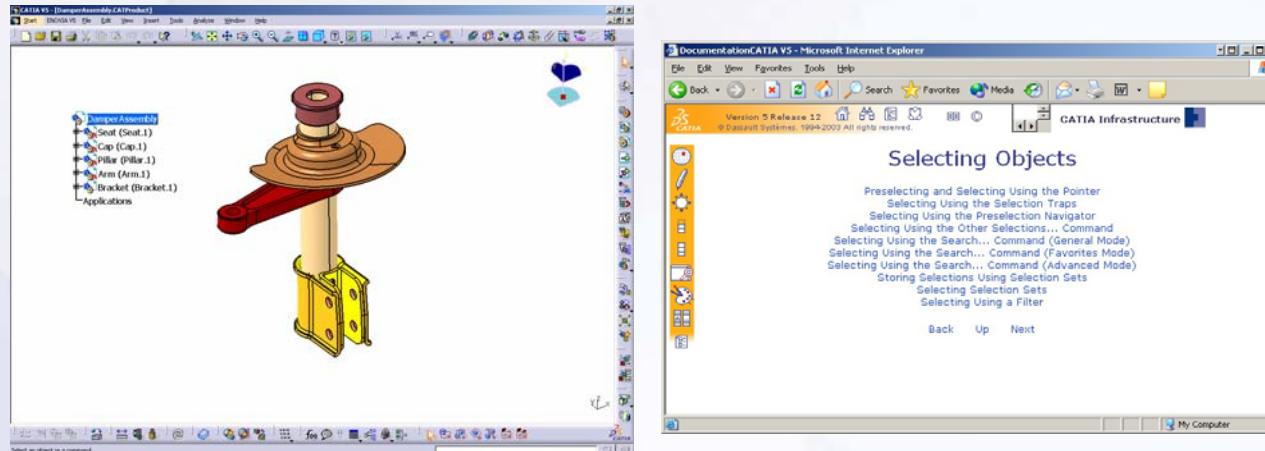
- b. Select Help > CATIA V5 Help.
- c. A web browser window opens, and the CATIA Help start page is displayed.
- d. Spend a few minutes browsing the various links of the help system.

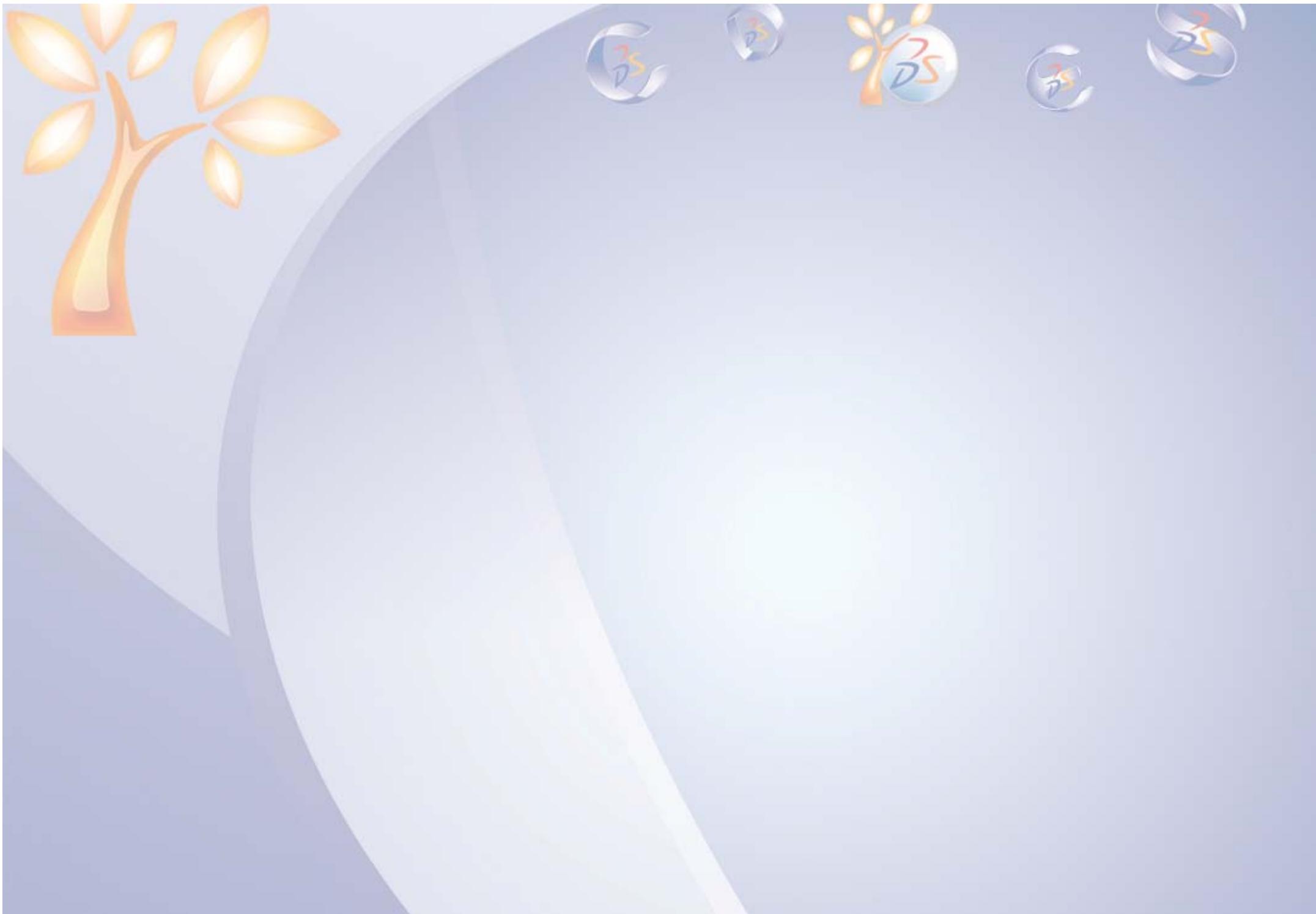
10. Close the assembly without saving the changes.



Case Study: Introduction to CATIA Recap

- ✓ Change the orientation of the model
- ✓ Change the visualization properties of the model
- ✓ Manipulate the specification tree
- ✓ Access the CATIA help system





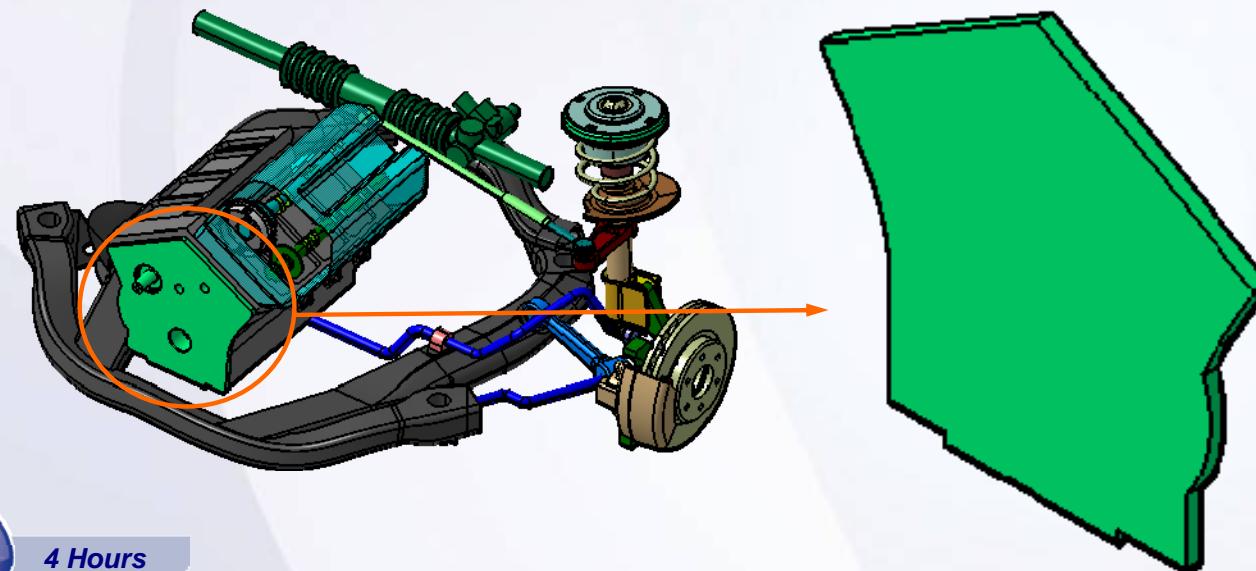
2

Profile Creation

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Create a New Part
- ✓ Select an Appropriate Sketch Support
- ✓ Create Sketched Geometry
- ✓ Constrain the Sketch
- ✓ Create the Pad Feature
- ✓ Save and close the document



4 Hours

Case Study

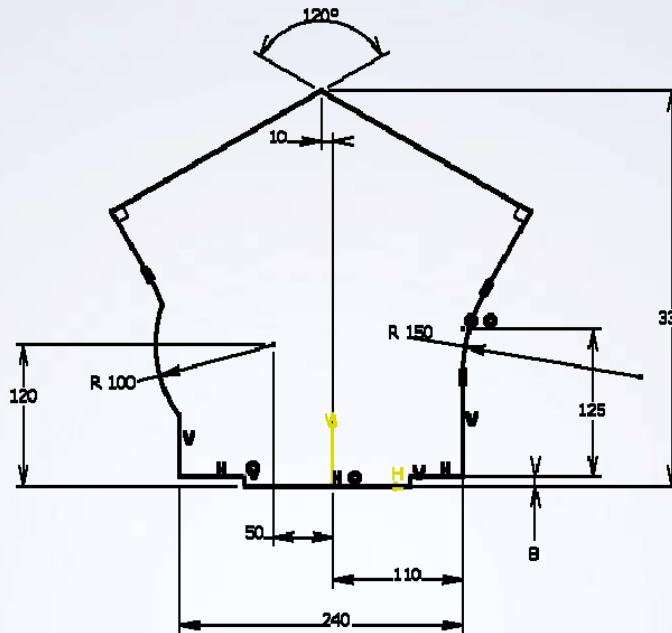
The case study for this lesson is the Timing Chain Cover used in the Front Suspension and Engine assembly.

Design Intent

- ✓ The model must be created in one feature.
- ✓ The top angle must be 120 degrees.
- ✓ The overall height must be 335 mm.
- ✓ The angular side walls must be perpendicular to the top walls.
- ✓ The centre of the convex circular arc must be 50 mm from the vertical reference and 120 mm from the horizontal reference.
- ✓ The thickness must be 12 mm.

Stages in the Process

1. Create a New Part.
2. Select an appropriate sketch support.
3. Create sketched geometry.
4. Constrain the sketch.
5. Create the Pad feature.
6. Save and close the document



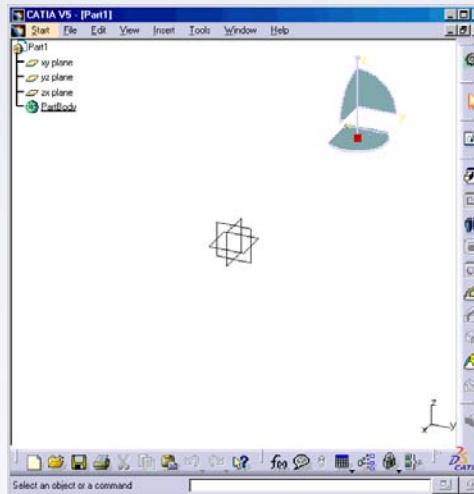
Create a New Part

When creating a new model, the Part Design workbench is activated. When a part is saved, it is saved with a .CATPart extension to distinguish it from other CATIA documents.

Use the following method to create a new part file:

- ✓ Click Start > Mechanical Design > Part design.
- ✓ Click File > New and select Part from the New dialog box.
- ✓ Select the New icon from the Standard toolbar and select Part from the New dialog box.

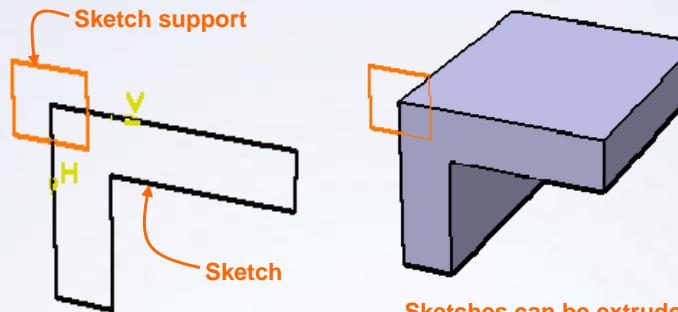
A new part contains only three default reference planes. These default reference planes are always the first elements in the specification tree and are used as a basis for feature creation.



Select an Appropriate Sketch Support

Every new part begins with a 2D profile. This profile can be created using the Sketcher workbench. The elements created within Sketcher are exclusively 2D WIREFRAME elements. In the Part Design workbench, the geometry created in Sketcher is seen as a single sketch. This sketch is used to create 3D features inside the Part Design workbench.

A sketch support is the plane on which the sketch is created. The sketch support must be planar. You can create a sketch on a reference plane or on a planar face of any existing geometry. The default orientation of the model depends on which reference plane is selected for the sketch support.

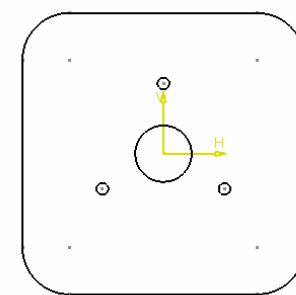
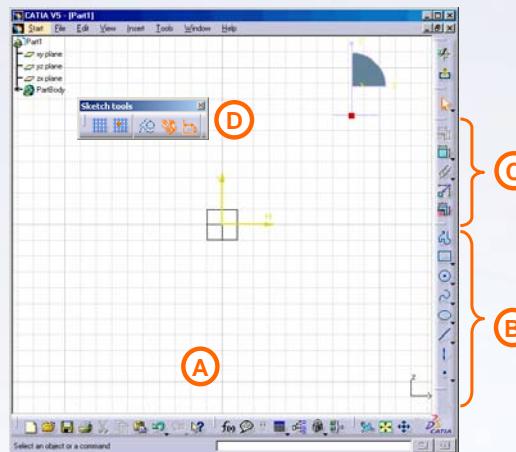


Create Sketched Geometry

The Sketcher workbench is an environment built to facilitate the creation of the 2D Profiles. The Sketcher workbench includes:

- A. The Grid, which guides you while you create the profiles.
- B. The Profile toolbar, which is used to create geometry.
- C. The Constraint toolbar, which is used to constrain your sketch.
- D. The Sketch Tools toolbar, that displays options available during geometry creation.

It is recommended to use a Positioned Sketch while creating a sketched profile. Do not to use Fillets, Chamfers and Drafts when creating sketched profile, because some of the manufacturing processes need to remove the Dress-Up features.

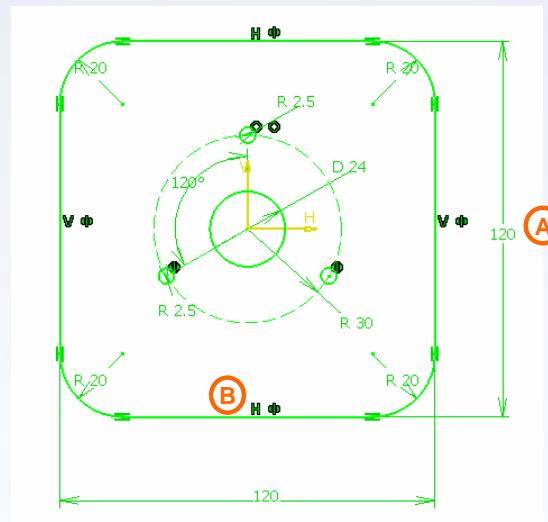


Constrain the Sketch

Constraints serve to mathematically fix geometry in space. Two types of constraints can be added to sketched geometry:

- A. Geometric constraints, which specify how sketched elements are positioned with respect to each other and existing 3D geometry.
- B. Dimensional constraints, which specify the distance between two elements. This distance can be linear, angular, or radial.

Ideally, a completed sketch must be fully constrained. As you begin to create geometry, try to create it reasonably close in shape and size to the final constrained sketch.



Create the Pad Feature

Once the sketched profile has been created, solid 3D geometry can be generated from it. A pad is a sketch-based feature that adds material to a model.

Save and Close the Document

Documents need to be saved so that work is not lost. There are different ways to save CATIA documents:

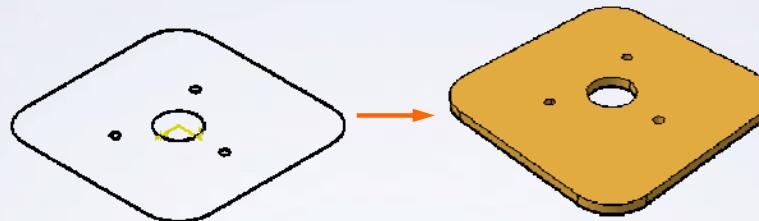
- ✓ Save
- ✓ Save As
- ✓ Save All
- ✓ Save Management

Documents are saved:

- ✓ After modifying them.
- ✓ After creating new ones.

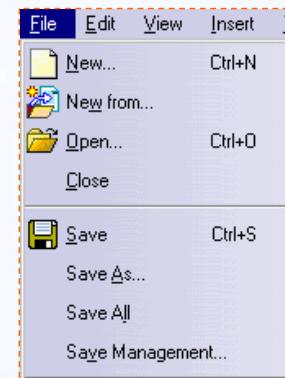
Documents can be saved:

- ✓ With the same name (to replace the initial document).
- ✓ With a new name (to create a new document).



2D Profile (sketch)

Extruded Pad



Sketcher Tools

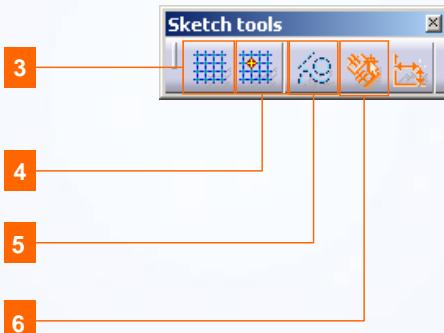
Sketcher

- 1 **Sketch:** creates a new sketch and opens sketcher workbench
- 2 **Positioned Sketch:** creates a new sketch and you can specify various parameter for sketch support



Sketch Tools

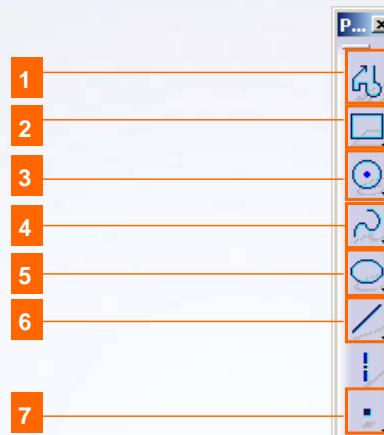
- 3 **Grid:** a grid is applied to the background of the Sketcher workbench
- 4 **Snap to Point:** the mouse pointer snaps to the points of the grid
- 5 **User-Defined Profile:** creates a sketched element as a construction element
- 6 **Geometrical Constraints:** controls whether geometric constraints are automatically created or not, during the development of the initial sketch



Geometry Creation Tools

Profile

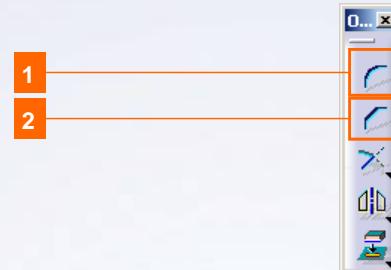
- 1 **User-Defined Profile:** creates complex profiles consisting of straight line and circular arcs
- 2 **Pre-defined Profiles:** creates predefined profiles such as rectangle, parallelogram, hexagon etc.
- 3 **Circles:** creates circles and circular arcs
- 4 **Splines:** creates splines and connecting curves
- 5 **Ellipses and Parabolas:** creates conic curves such as ellipse, parabola, hyperbola etc.
- 6 **Lines:** creates predefined profiles such as rectangle, parallelogram, hexagon etc.
- 7 **Points:** creates splines and connecting curves



Additional Tools

Operation

- 1 **Corner:** creates a corner shape between the two selected lines.
- 2 **Chamfer:** creates a chamfer between the two selected lines.



Constraint

- 3 **Constraint Defined in Dialog Box:** creates geometrical constraints on selected elements.
- 4 **Constraint:** creates geometrical and dimensional constraints.



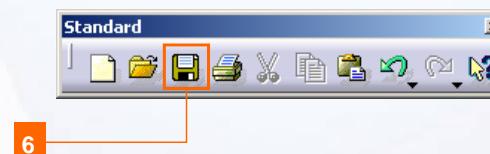
Sketch-Based Features

- 5 **Pad:** extrudes a profile sketched in the Sketcher workbench. This command is available in Part Design workbench.



Standard

- 6 **Save:** saves recent changes done in existing files and saves newly created files.



Exercise: Profile Creation

Recap Exercise

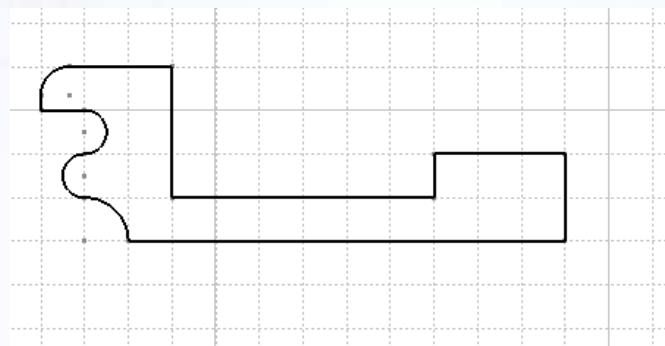


15 min

In this exercise you will create a sketched profile. High-level instruction for this exercise is provided.

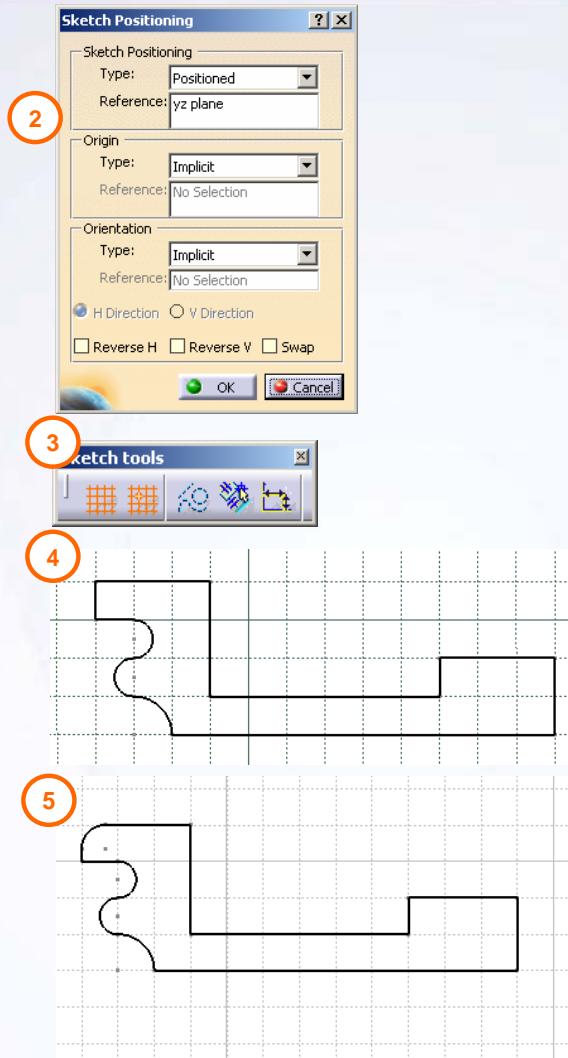
By the end of this exercise you will be able to:

- Create a new part
- Access the Sketcher workbench
- Create geometry using the Profile tool
- Create a corner
- Close the document without saving it



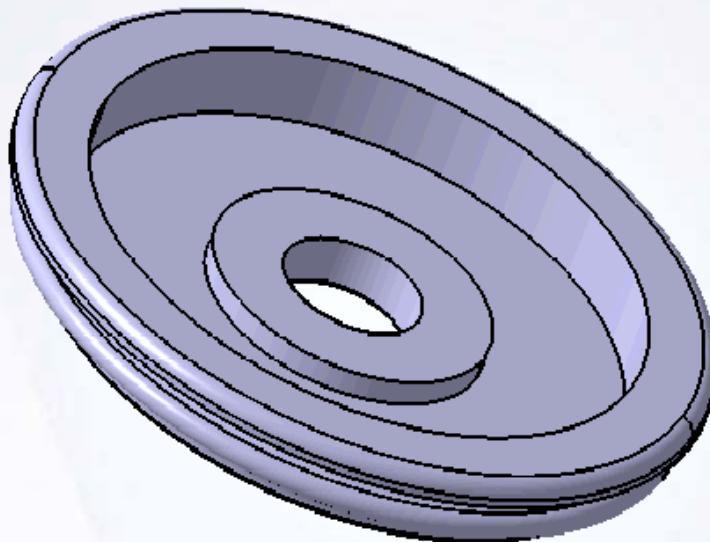
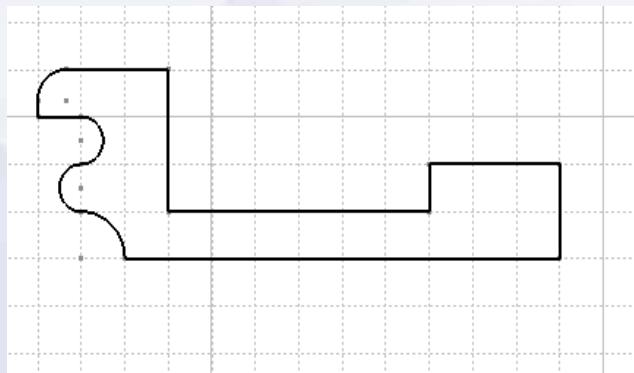
Do it Yourself

1. Create a new part.
2. Create a **Positioned Sketch** using YZ plane as the Reference plane and access the Sketcher workbench.
3. Ensure that the automatic constraints (located in the Sketch Tools toolbar) are deactivated.
4. Create the profile (as shown) using the **Profile** icon.
5. Create the corner (this is a Style type of radius).
6. Close the document without saving it.



Exercise Recap: Profile Creation

- ✓ Create a new part file
- ✓ Select the YZ plane as the sketch support
- ✓ Create sketched geometry
- ✓ Close the document without saving it



The sketch created in this exercise could be used to create a revolved feature, as shown above. You will learn how to create revolved features in Lesson 4.

Exercise: Profile Creation

Recap Exercise



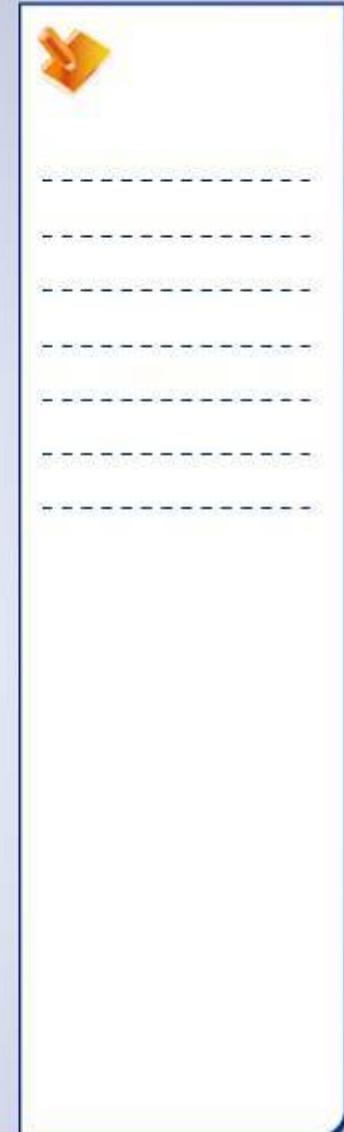
10 min

In this exercise you will create five profiles. You will use the tools learned in the previous exercises to complete this exercise.

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

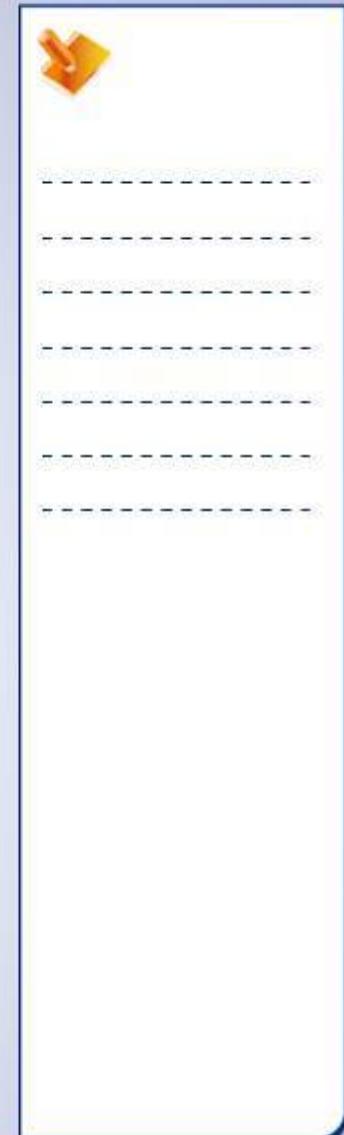
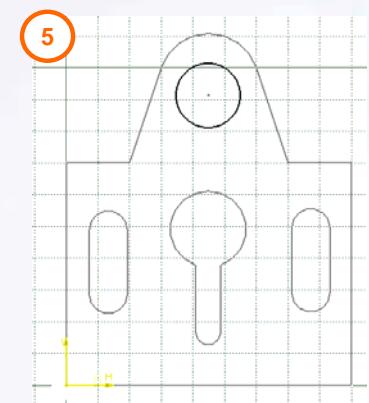
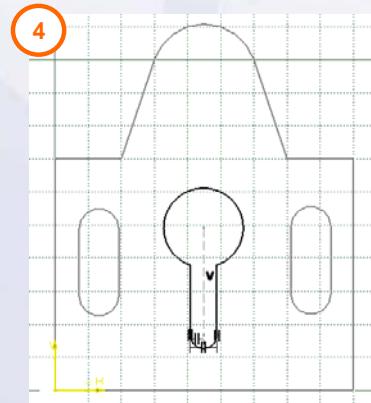
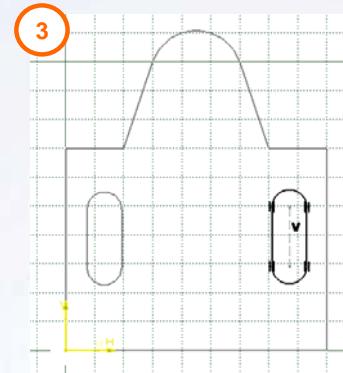
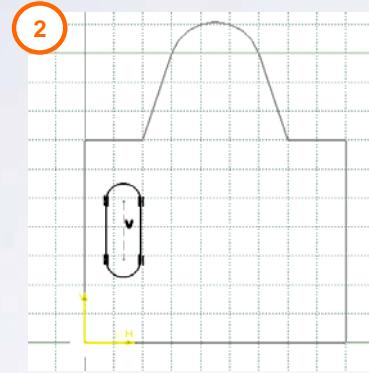
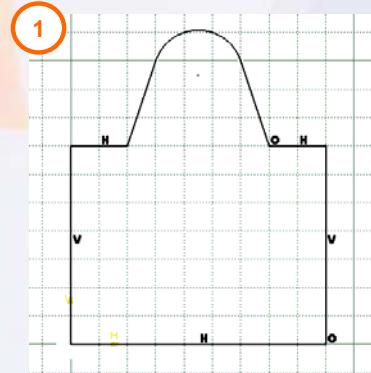
- Create a new part
- Create positioned sketch profiles using ZX plane as sketch support
- Close the document without saving it

CAUTION: Generally, for complex parts it is recommended that you simplify the sketches using dedicated 3D features like fillets, chamfers, holes, drafts, etc. to better fit the design and manufacturing intents.



Do it Yourself

1. Create a new part and create five different profiles as shown, using Positioned Sketch and ZX plane as the sketch support.

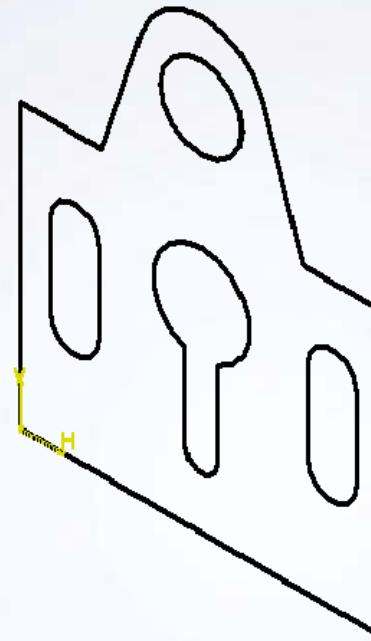


Exercise Recap: Profile Creation

- ✓ Create a new part file
- ✓ Access Sketcher workbench
- ✓ Create positioned sketch geometry using ZX plane as sketch support
- ✓ Close the document without saving it

You can create the outside profile using a number of ways:

- Create the profile using a series of lines and arcs.
- Create the whole profile using the Profile tool.



Exercise: Sketch Constraints

Recap Exercise



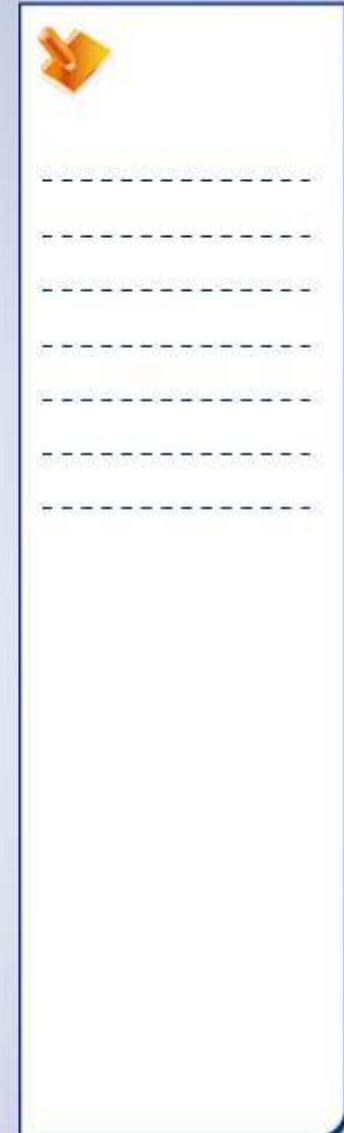
15 min

In this exercise, you will fully constrain an existing sketch using the tools from the previous exercise. This exercise will help you understand how to constrain and dimension sketched entities. High-level instructions for this exercise are provided.

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

- Load an existing document
- Constrain a sketch
- Dimension a sketch
- Use problem-solving skills
- Save and close a model

CAUTION: Generally, for complex parts it is recommended that you simplify the sketches using dedicated 3D features like fillets, chamfers, holes, drafts, etc. to better fit the design and manufacturing intents.

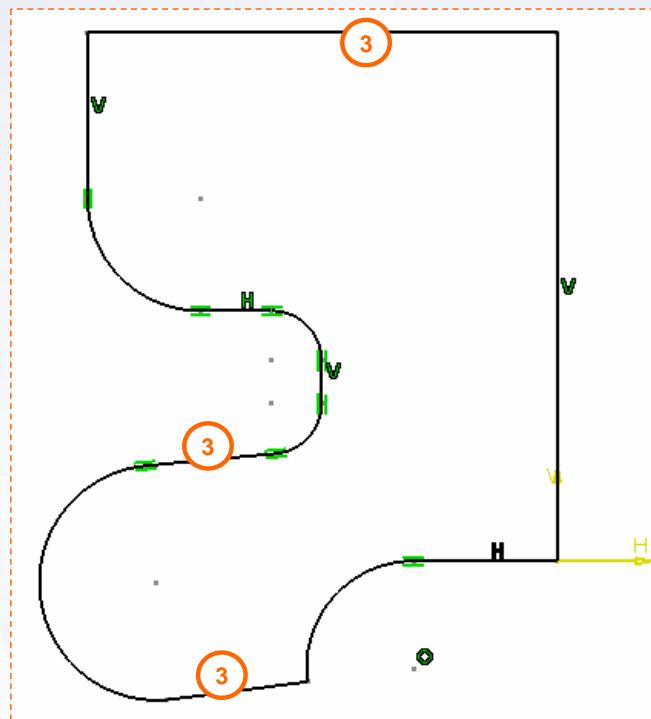


Do it Yourself (1/5)

- 1. Load the file.**
 - Load Ex2E_1.CATPart. Once loaded, notice that sketches have already been created for you.

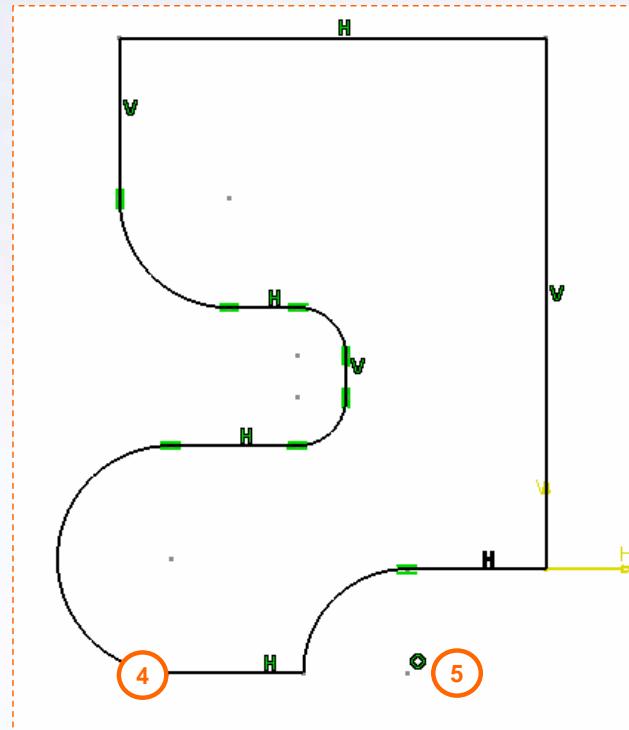
- 2. Edit the sketch.**
 - Modify the sketch.1 in the Sketcher workbench by double-clicking on the sketch directly in the model or in the specification tree.

- 3. Add Horizontal constraints.**



Do it Yourself (2/5)

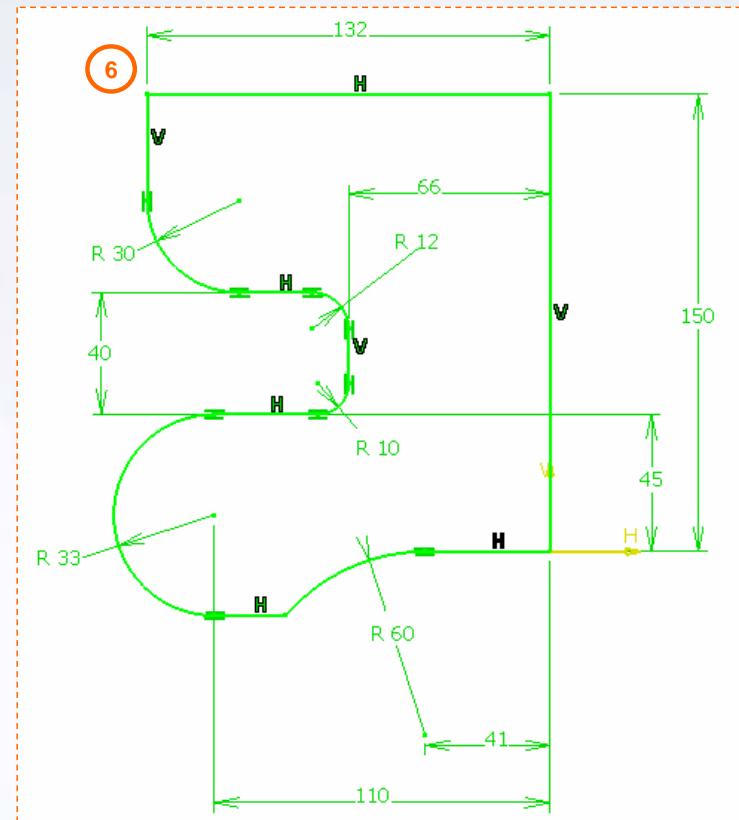
4. Add Tangency constraint.
 - Apply a tangency constraint between the bottom horizontal line and the arc.
5. Remove coincidence constraints which are not required.



Do it Yourself (3/5)

6. Add dimensional constraints.

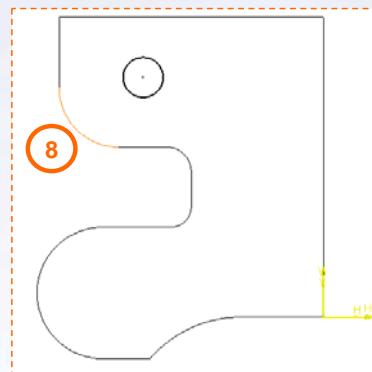
- Using proper techniques, add dimensional constraints to the sketch. Once all the dimensional constraints have been applied, the sketch will turn green, indicating that the sketch is fully-constrained.



Do it Yourself (4/5)

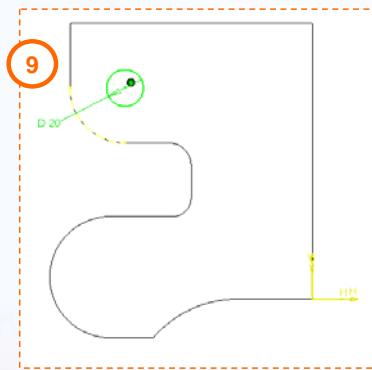
7. Edit the sketch.

- Modify the sketch.2 in the Sketcher workbench by double-clicking on the sketch directly in the model or in the specification tree.



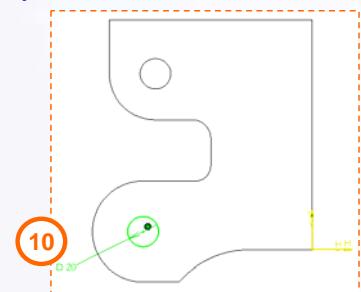
8. Sketch in Context.

- Project the edge of sketch.1 and keep it as a construction element. Add Concentricity constraint.



9. Add dimensional constraint.

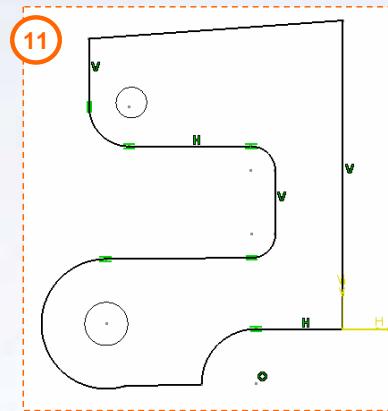
- Add 20mm diameter dimension to make sketch.2 fully-constrained.



Do it Yourself (5/5)

11. Search and Load Ex2E_2.CATPart.

- Load an existing part file. Once loaded, notice that sketches have already been created for you.

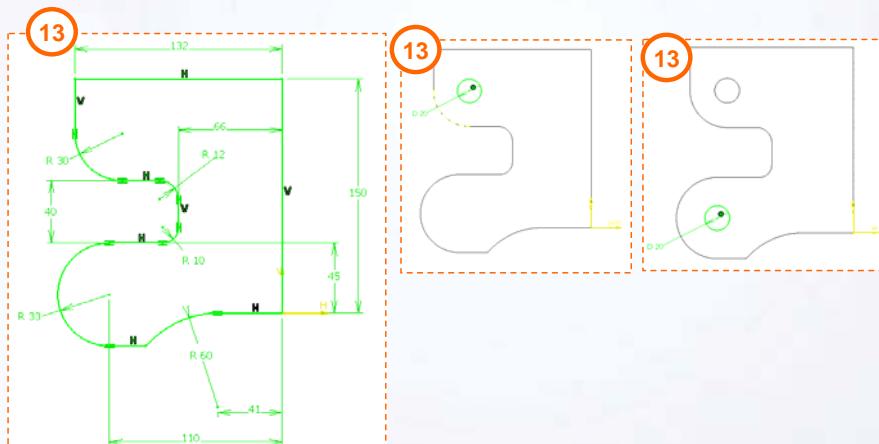


12. Edit the sketch.

- Modify the sketches in the Sketcher workbench by double-clicking on the sketch directly in the model or in the specification tree.

13. Geometrically and dimensionally constrain the sketches.

- Fully constrain the sketched circles with 20mm diameter dimensions as in the earlier instance.

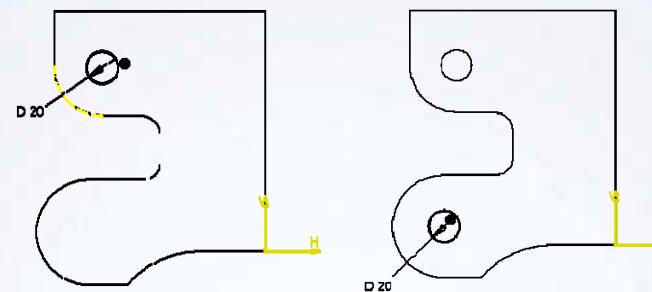
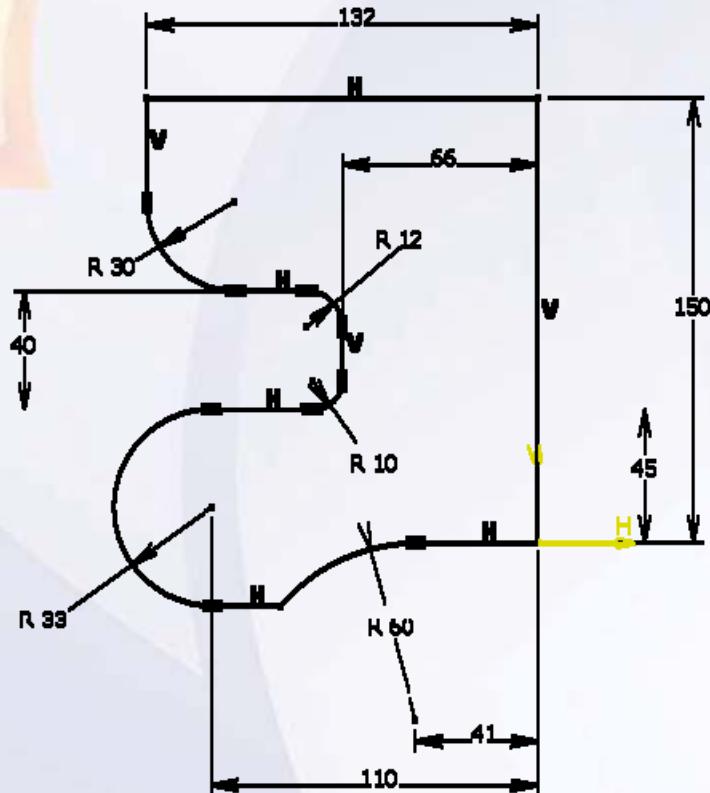


14. Compare sketches.

- Were the sketches easier to constrain compared to the earlier instance? Why?

15. Save and close both the documents.

Exercise Recap: Sketch Constraints



- ✓ Constrain the sketches
- ✓ Dimension the sketches
- ✓ Understand proper sketching techniques

Exercise: Sketch Constraints

Recap Exercise



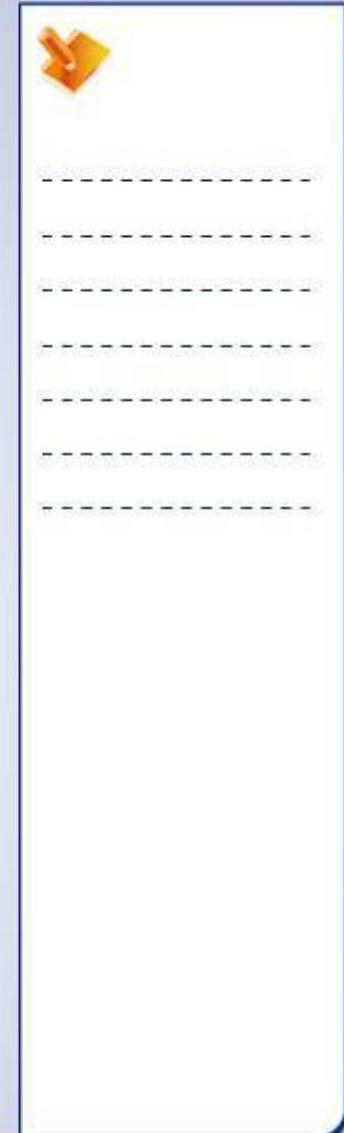
15 min

In this exercise you will fully constrain an existing sketch. You will use the tools learned in this lesson to complete the exercise with no detailed instructions.

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

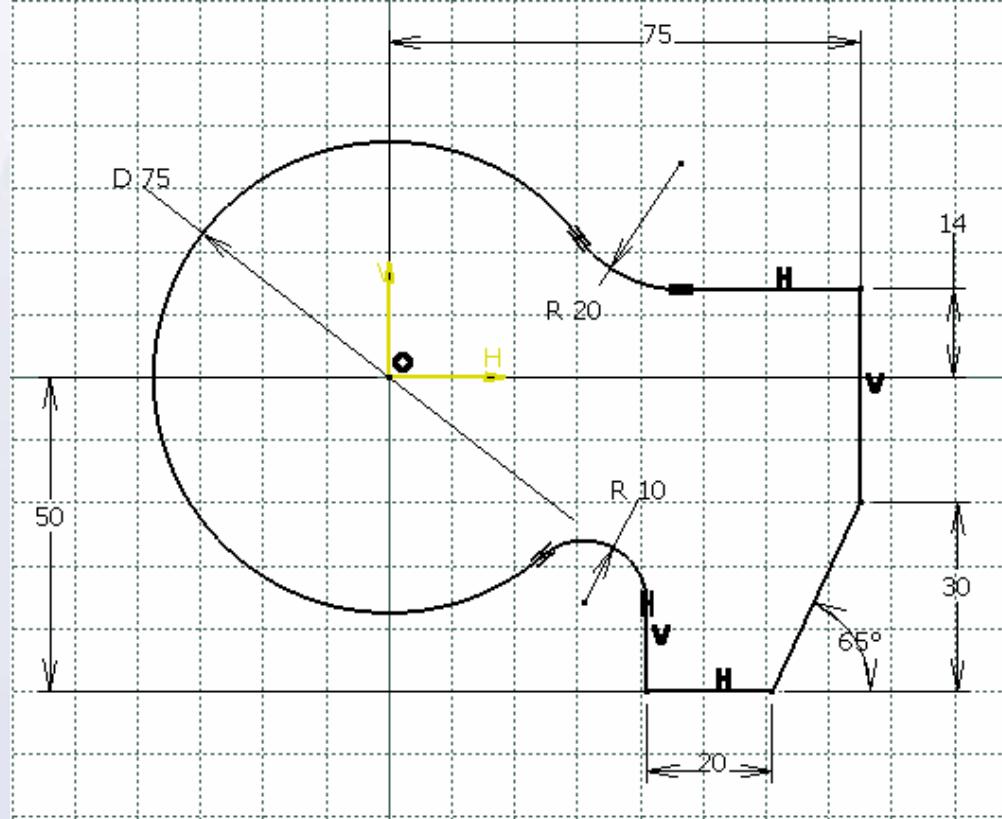
- Open an existing model
- Edit a sketch
- Constrain an existing sketched geometry
- Save and close the document

CAUTION: Generally, for complex parts it is recommended that you simplify the sketches using dedicated 3D features like fillets, chamfers, holes, drafts, etc. to better fit the design and manufacturing intents.



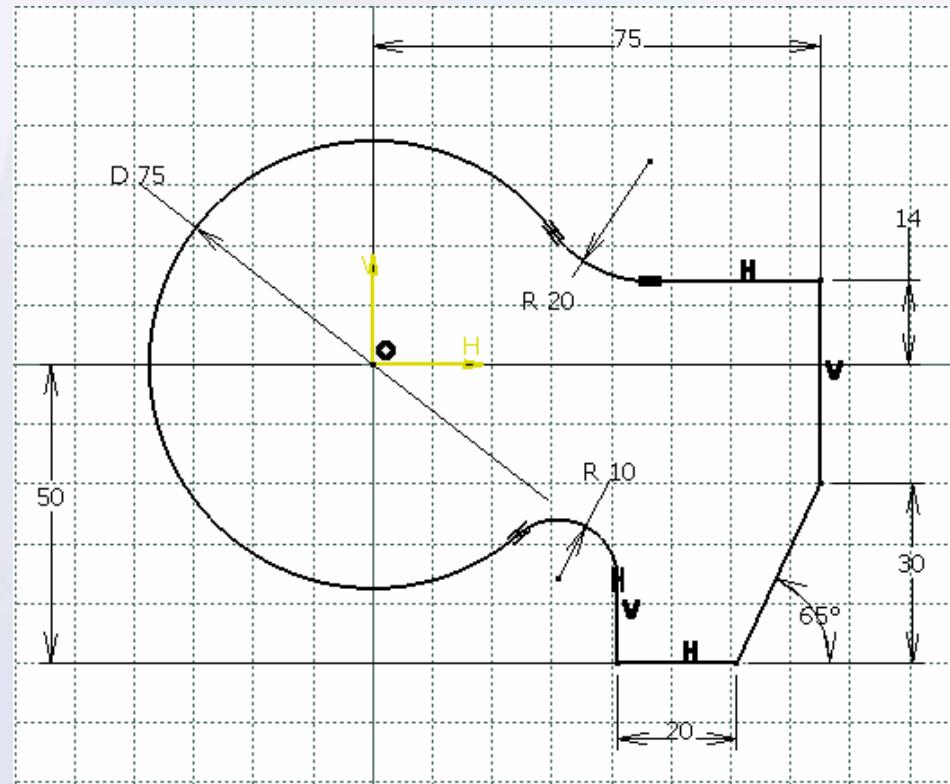
Do it Yourself

1. Load Ex_2F.CATPart and fully constrain the sketch.



Exercise Recap: Sketch Constraints

- ✓ Constrain a sketch
- ✓ Dimension a sketch



Case Study: Profile Creation

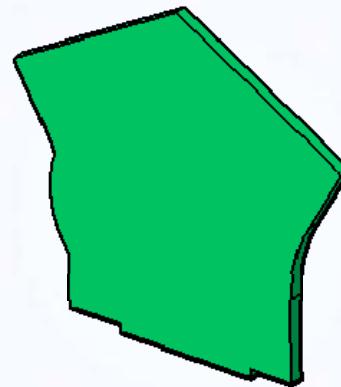
Recap Exercise



10 min

In this exercise you will create the case study model. Let us recall the design intent of this model:

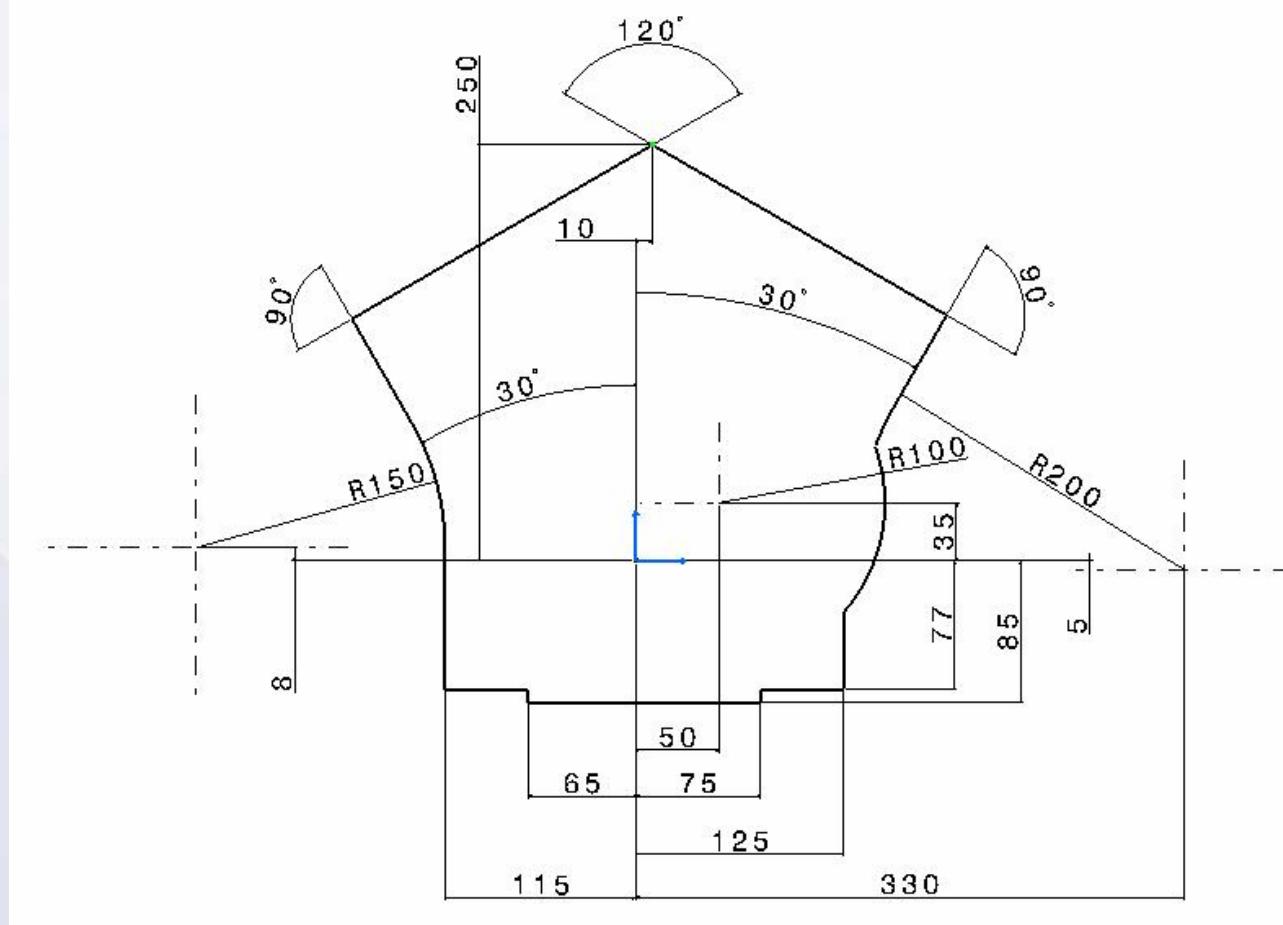
- ✓ The model must be created in one feature.
- ✓ The top angle must be 120 degree.
- ✓ The overall height must be 335mm.
- ✓ The angular side walls must be perpendicular to the top walls.
- ✓ The thickness must be 12mm.
- ✓ The center of convex circular arc must be 50mm from vertical reference and 120mm from horizontal reference.



Using the techniques you have learned so far, create the model without detailed instructions.

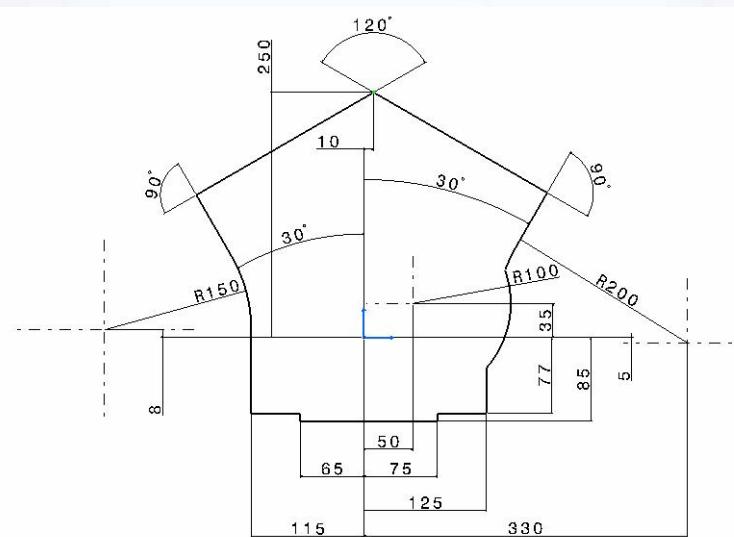
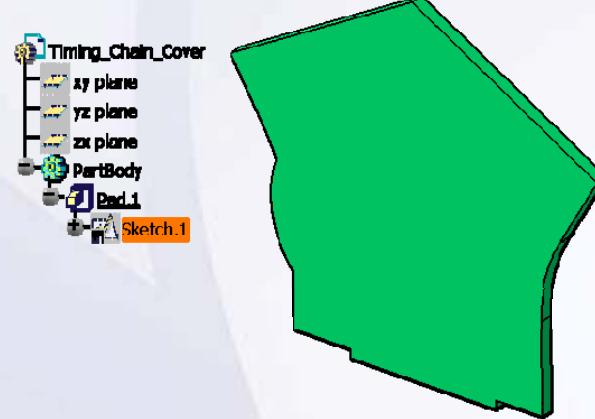
Do It Yourself: Drawing of the Timing Chain Cover

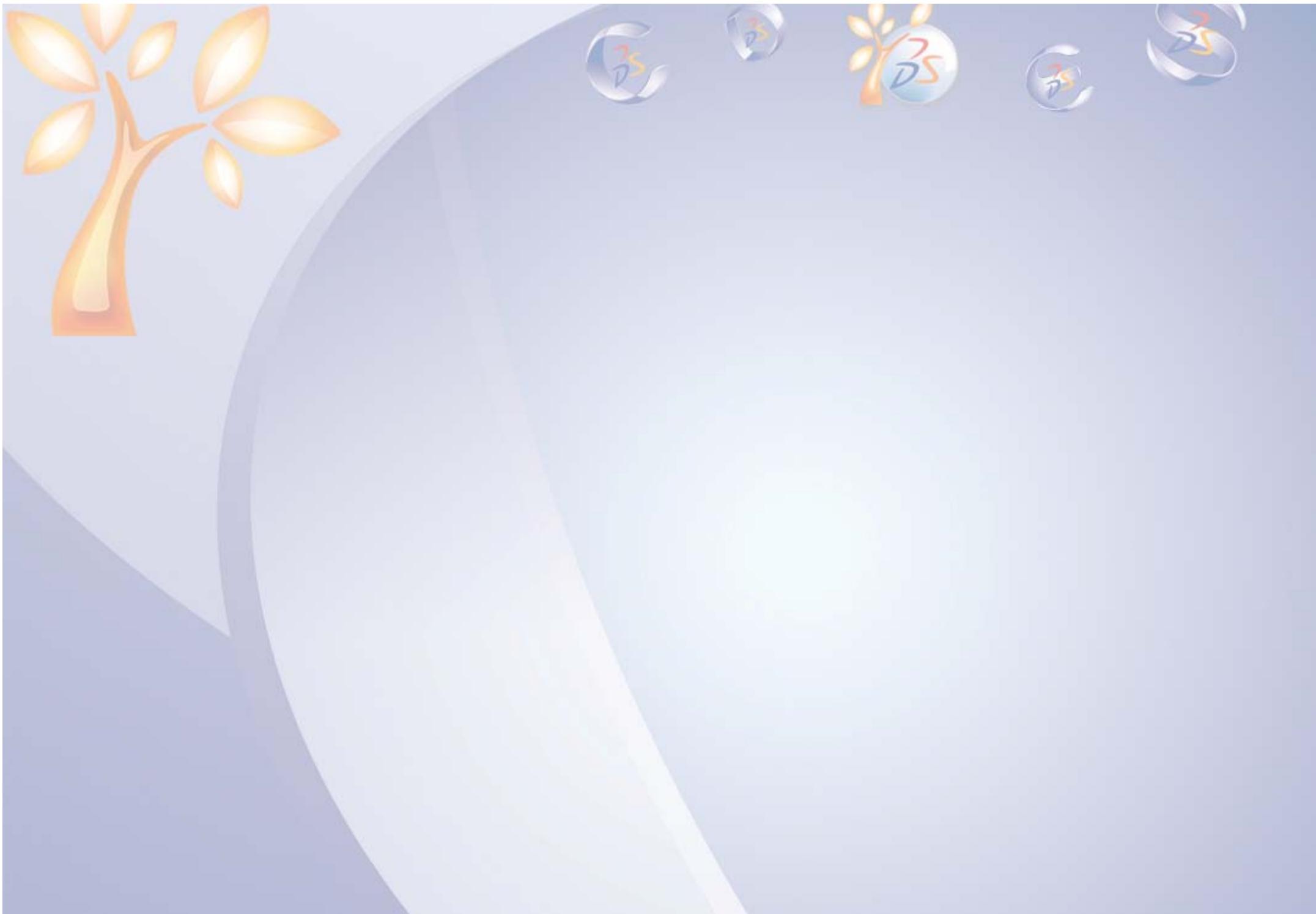
Create the model using the drawing provided here.



Case Study Recap: Timing Chain Cover

- ✓ Create a new part file
- ✓ Select the YZ plane as sketch support
- ✓ Create a sketch geometry
- ✓ Constrain the sketch according to design intent
- ✓ Create a pad feature
- ✓ Save and close the document



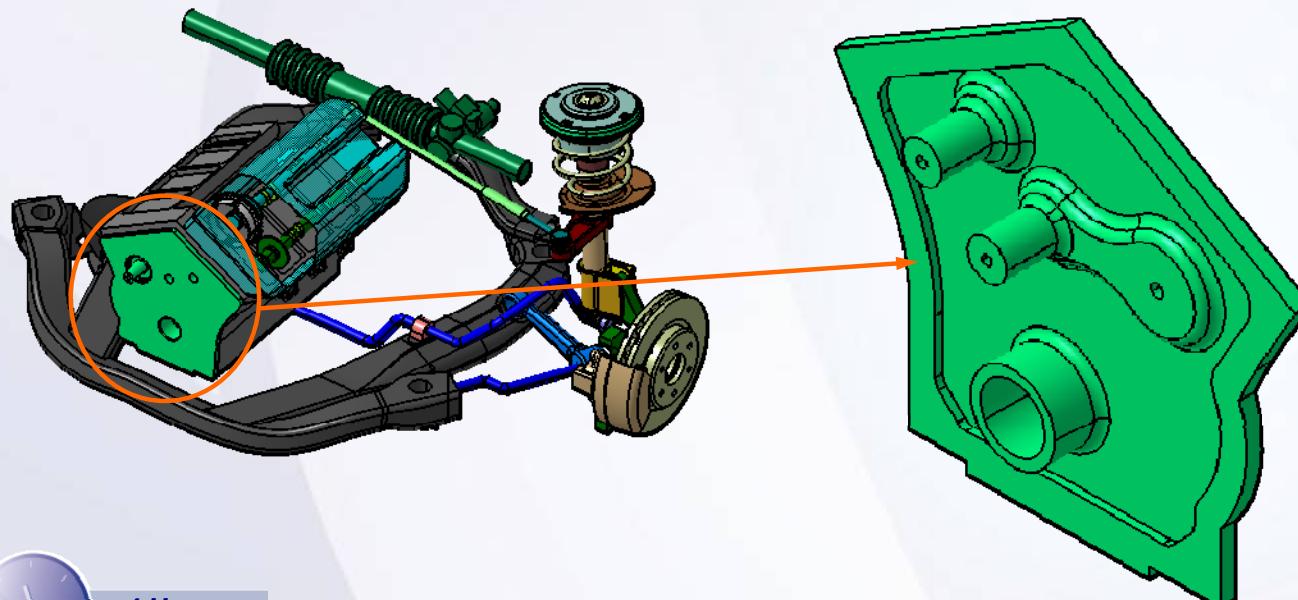


Basic Features

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Determine a Suitable Base Feature
- ✓ Create Pad and Pocket Features
- ✓ Create Holes
- ✓ Create Fillets and Chamfers



4 Hours

Case Study

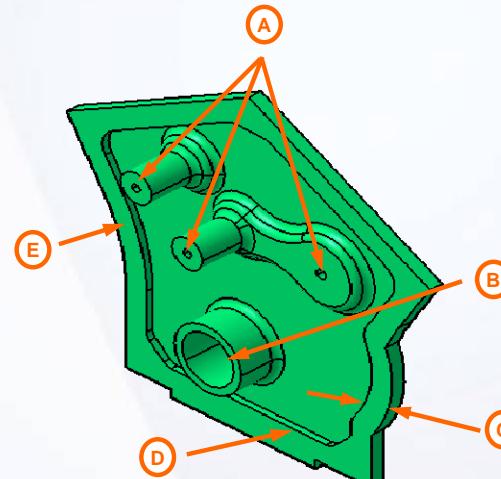
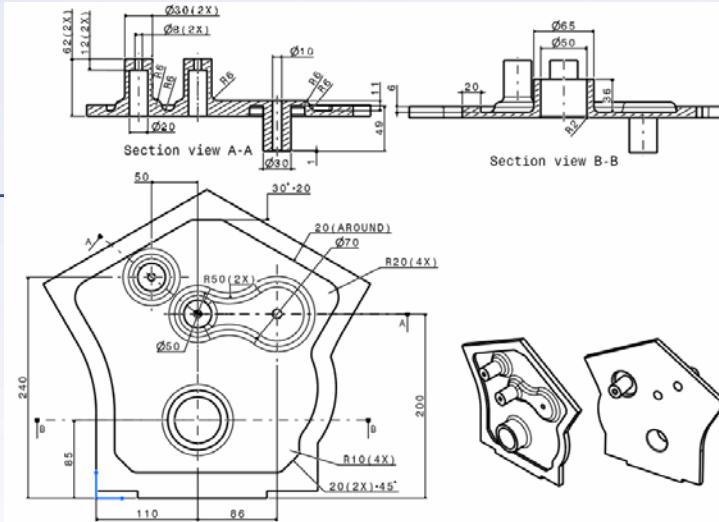
The case study for this lesson is the Timing Chain Cover used in the Front Suspension and Engine assembly.

Design Intent

- A. Three mounting holes must be created for mounting of timing chain cover.
- B. One large hole of diameter 50 mm must be created for shaft clearance.
- C. Rim width must be 20 mm.
- D. Rim height must be 6 mm.
- E. Concave radius 150 mm of the outer profile is style radius.

Stages in the Process

1. Determine a Suitable Base Feature
2. Create Pad and Pocket Features
3. Create Holes
4. Create Fillets and Chamfers



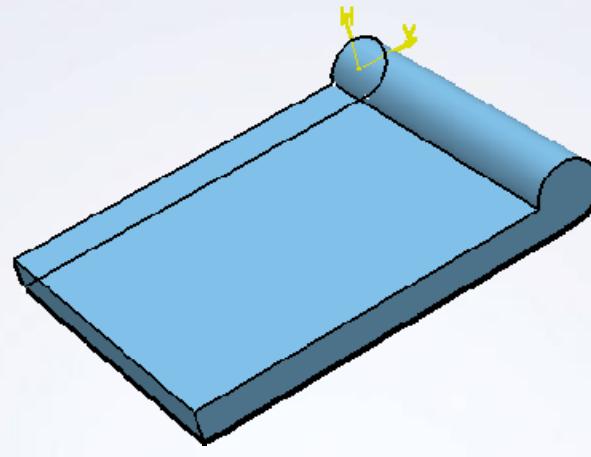
Determine a Suitable Base Feature

When selecting a base feature, it is recommended to select the basic elements that convey the primary shape or function of the part. This does not mean the level of detail for a base feature must be completely defined. For example, fillets, holes, pockets, or other features need not be created as a part of the base feature sketch; these can be created later as separate features.

Use the following steps to create a base feature:

- ✓ Identify the part features.
- ✓ Select one feature to represent the base element.
- ✓ Identify the CATIA tools (features) needed to create it.
- ✓ Create the feature.

The base feature usually starts from a sketch or a surface element.

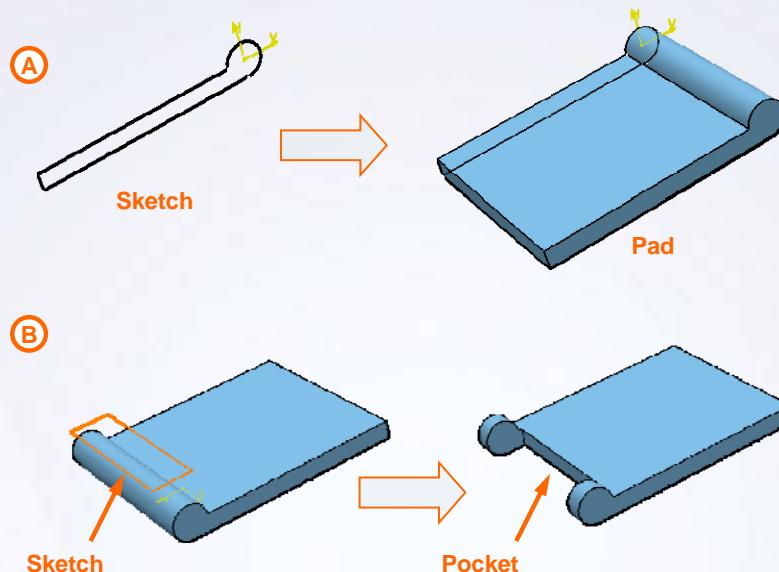


Create the Pad and Pocket Features

- A. A pad is a sketched-based feature that adds material to a model.
- B. A pocket is a sketched-based feature that removes material from a model.

The profile sketch should consist of connecting entities that form a closed loop. Open loop profile sketches can be used only with the Thick option.

The length of a pad or pocket can be defined by dimensions or with respect to existing 3D limiting elements. If the pad/pocket feature is defined by a limiting element, it becomes associative to that element.



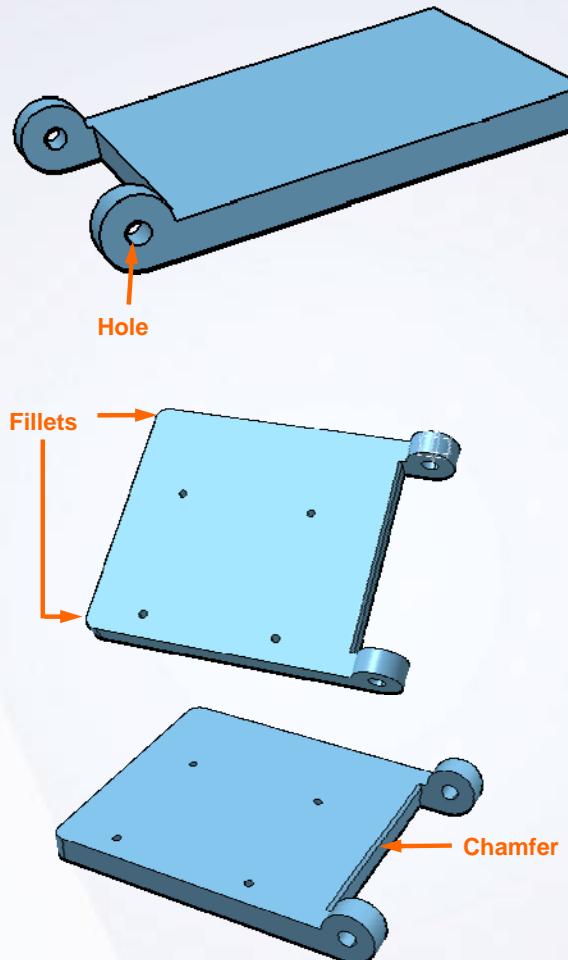
Create Holes

A hole removes circular material from an existing solid feature. A hole does not require a profile sketch. Like a pocket, its length can be defined using dimensions or with respect to the existing 3D elements.

A hole can be created using the Pocket or Hole tool. The advantage of creating a hole using a Hole tool is that a sketch gets created automatically.

The Hole tool also allows you to include technological information, such as thread, angle bottom, and counter bore.

If there is a possibility that the profile for the cutout may change from circular to another shape then consider using a pocket instead of a hole.



Create Fillets and Chamfers

A fillet is a curved face of a constant or variable radius that is tangent to, and that joins, two surfaces.

A chamfer replaces a selected edge by a flat section to create a beveled surface between the two original faces, which are common to that edge.

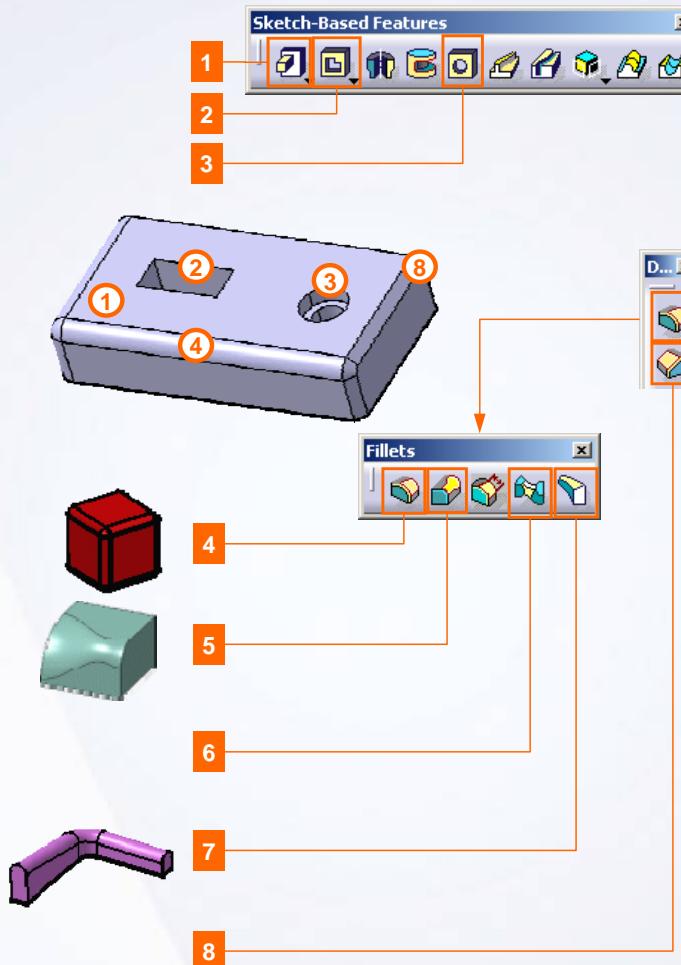
Basic Features Tools

Sketch-Based Features

- 1 **Pad:** adds material to a model by extruding a sketched profile
- 2 **Pocket:** removes material from a model by extruding a sketched profile
- 3 **Hole:** removes circular material from an existing solid model

Dress-Up Features

- 4 **Edge Fillet:** creates smooth transitional surfaces between two adjacent faces
- 5 **Variable Radius Fillet:** creates curved surfaces defined according to a variable radius
- 6 **Face-Face Fillet:** used when there is no intersection between the faces or when there are more than two sharp edges between the faces
- 7 **Tritangent Fillet:** removes one of the three faces which are selected
- 8 **Chamfer:** replaces a selected edge by a flat section to create a beveled surface



Exercise: Basic Features Creation

Recap Exercise

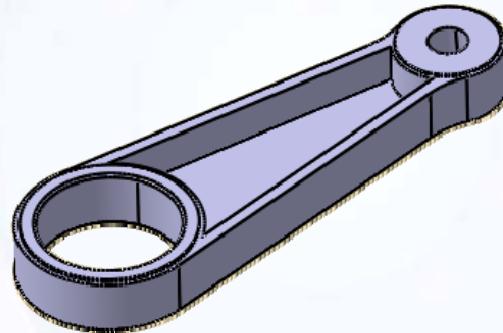


15 min

In this exercise, you will load an existing part that contains a base pad feature and a boss pad feature. In the base feature, you will create sketch-based features, simple and tapered holes, and simple dress-up features. High-level instructions for this exercise are provided.

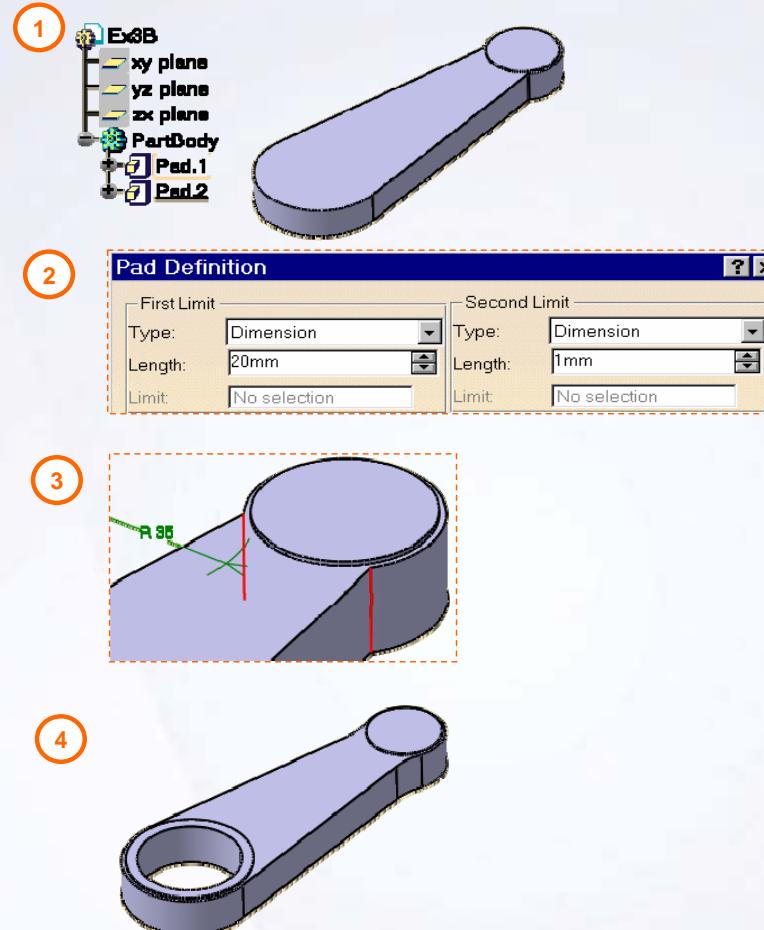
By the end of this exercise you will be able to:

- Create a concentric pad
- Create simple and tapered holes
- Create a pocket
- Create an edge fillet
- Create a chamfer



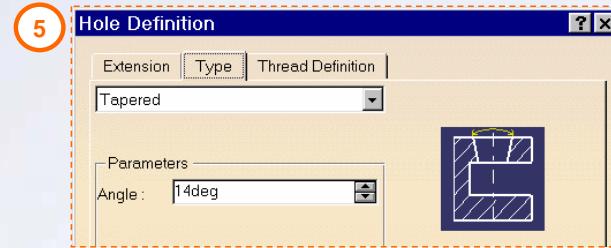
Do it Yourself (1/3)

1. Load Ex3B.CATPart.
2. Create a circular pad concentric with the larger end of the base feature. The diameter is 66mm and the first and second limits are 20mm and 1mm respectively.
3. Create a fillet of 35mm on 2 edges.
4. Create a hole of 54mm diameter, concentric to larger end of base feature.

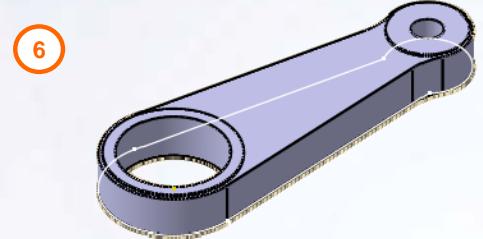


Do it Yourself (2/3)

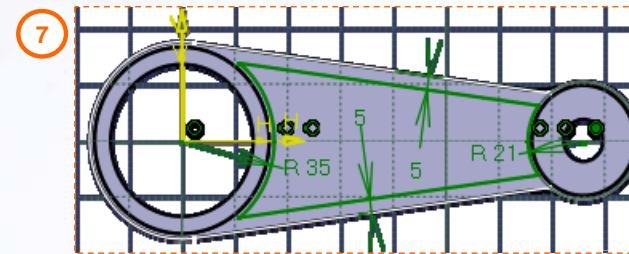
5. Create a tapered hole of diameter 15mm and tapered angle 14deg, concentric to small end of base feature.



6. Display the sketch for the base pad feature.



7. Create a pocket whose sketch is positioned on XY plane and depth is 14mm.

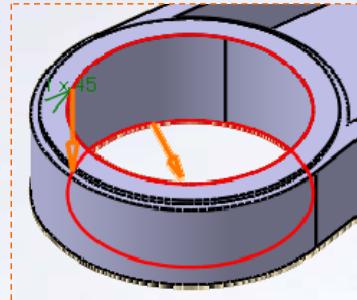


8. Hide sketch of base pad feature.

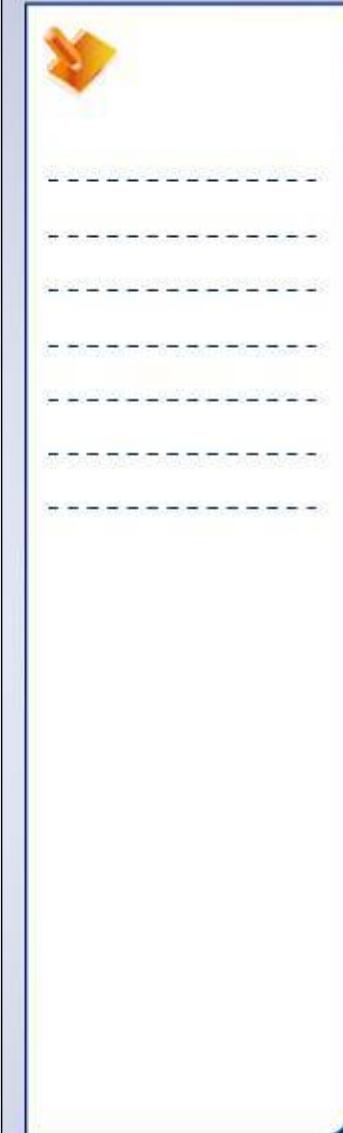
Do it Yourself (3/3)

9. Create a chamfer of 1mm X 45 degree on 2 edges.

9

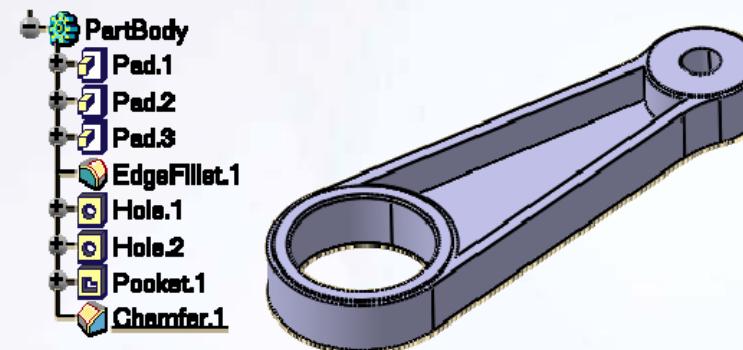


10. Save and close the part.



Exercise Recap: Basic Features Creation

- ✓ Create a concentric pad
- ✓ Create simple and tapered holes
- ✓ Create a pocket
- ✓ Create an edge fillet
- ✓ Create a chamfer



Exercise: Basic Features Creation

Recap Exercise

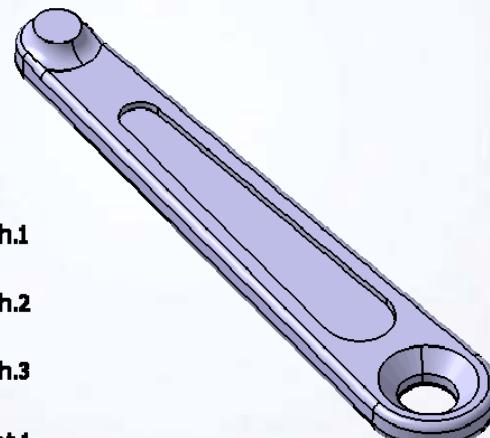


10 min

In this exercise, you will create a part that contains features taught in this and the previous lessons. You will use the tools you have learned to complete the exercise with no detailed instructions.

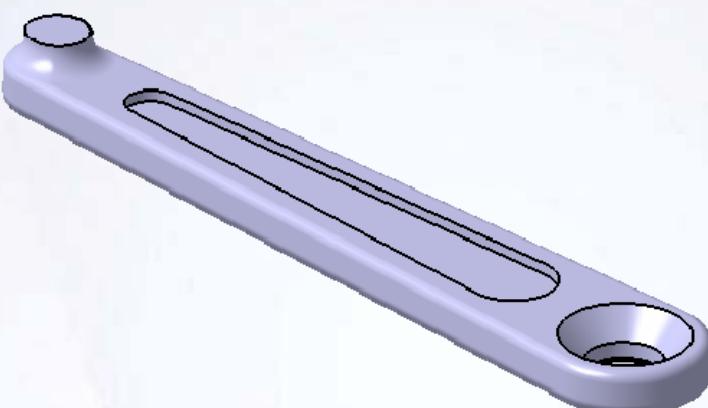
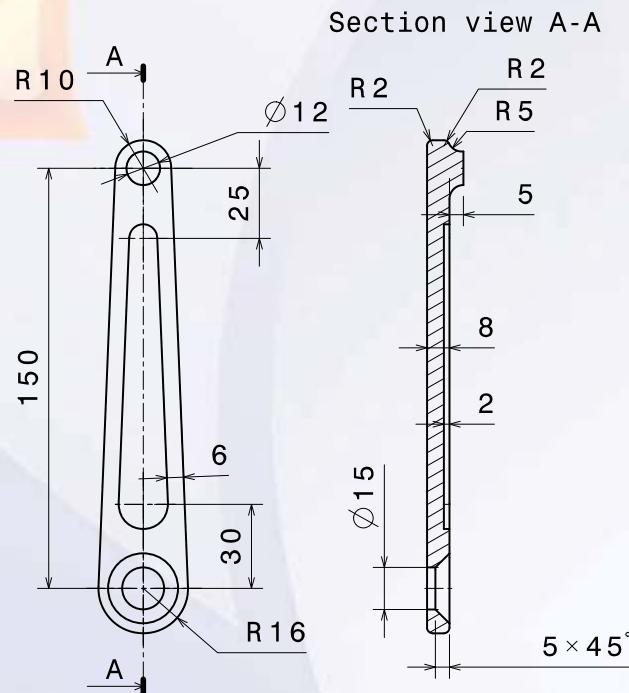
By the end of this exercise you will be able to:

- Create a pad
- Create a pocket
- Create a countersunk hole
- Create an edge fillet



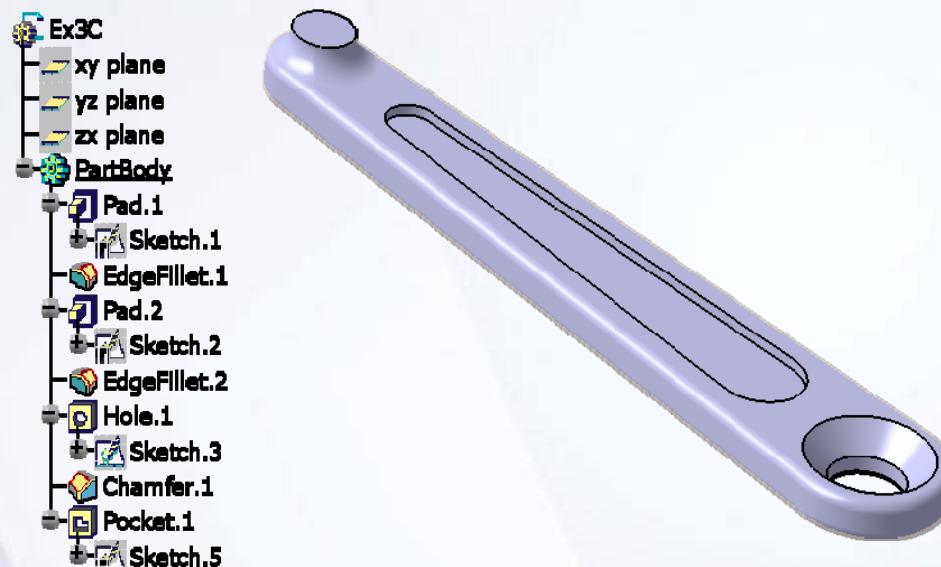
Do it Yourself

1. Create the following part.



Exercise Recap: Basic Features Creation

- ✓ Create a pad
- ✓ Create a pocket
- ✓ Create a countersunk hole
- ✓ Create an edge fillet



Case Study: Basic Features

Recap Exercise

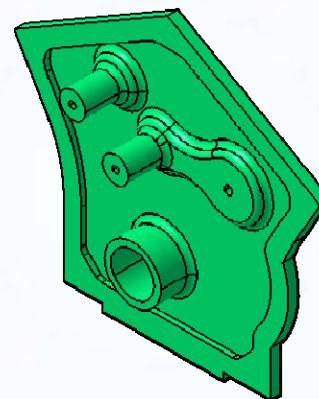


20 min

In this exercise you will modify the case study model. Let us recall the design intent of this model:

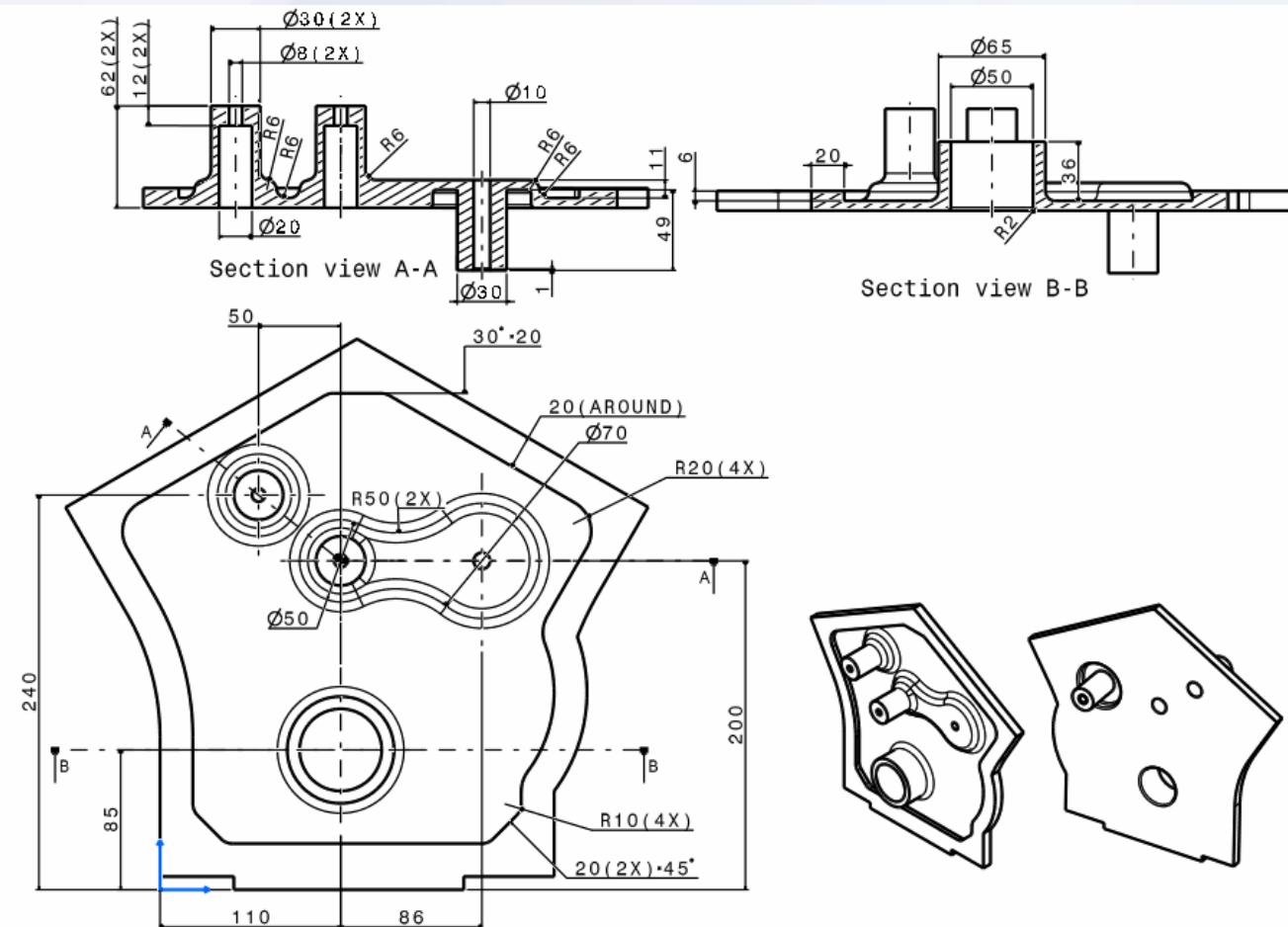
- ✓ Three mounting holes must be created for mounting of Timing Chain Cover.
- ✓ One large hole of diameter 50 mm must be created for shaft clearance.
- ✓ Rim width must be 20mm.
- ✓ Rim height must be 6mm.
- ✓ Concave radius 150mm of the outer profile is style radius.

Using the techniques you have learned so far, modify the model without detailed instructions.



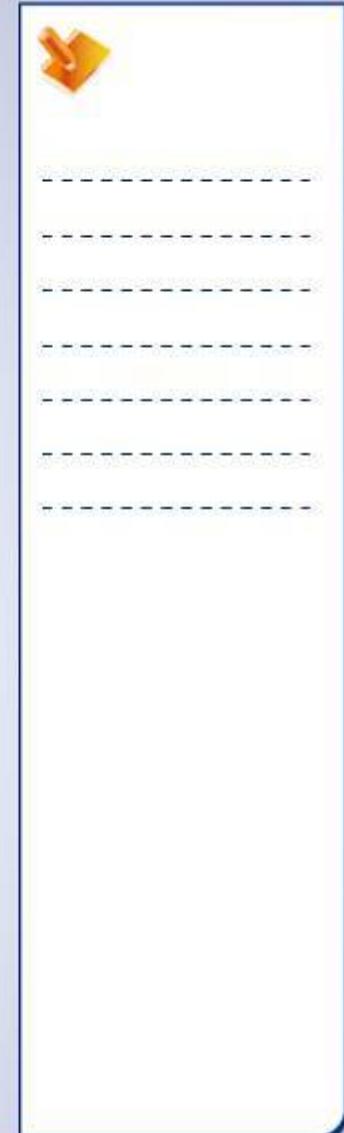
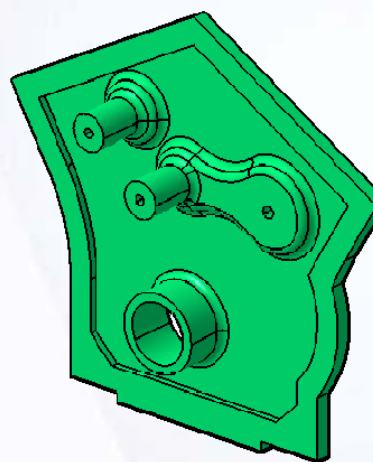
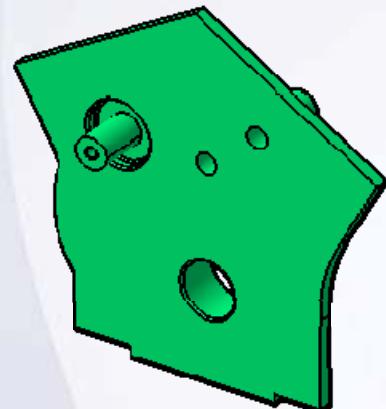
Do It Yourself: Drawing of the Timing Chain Cover

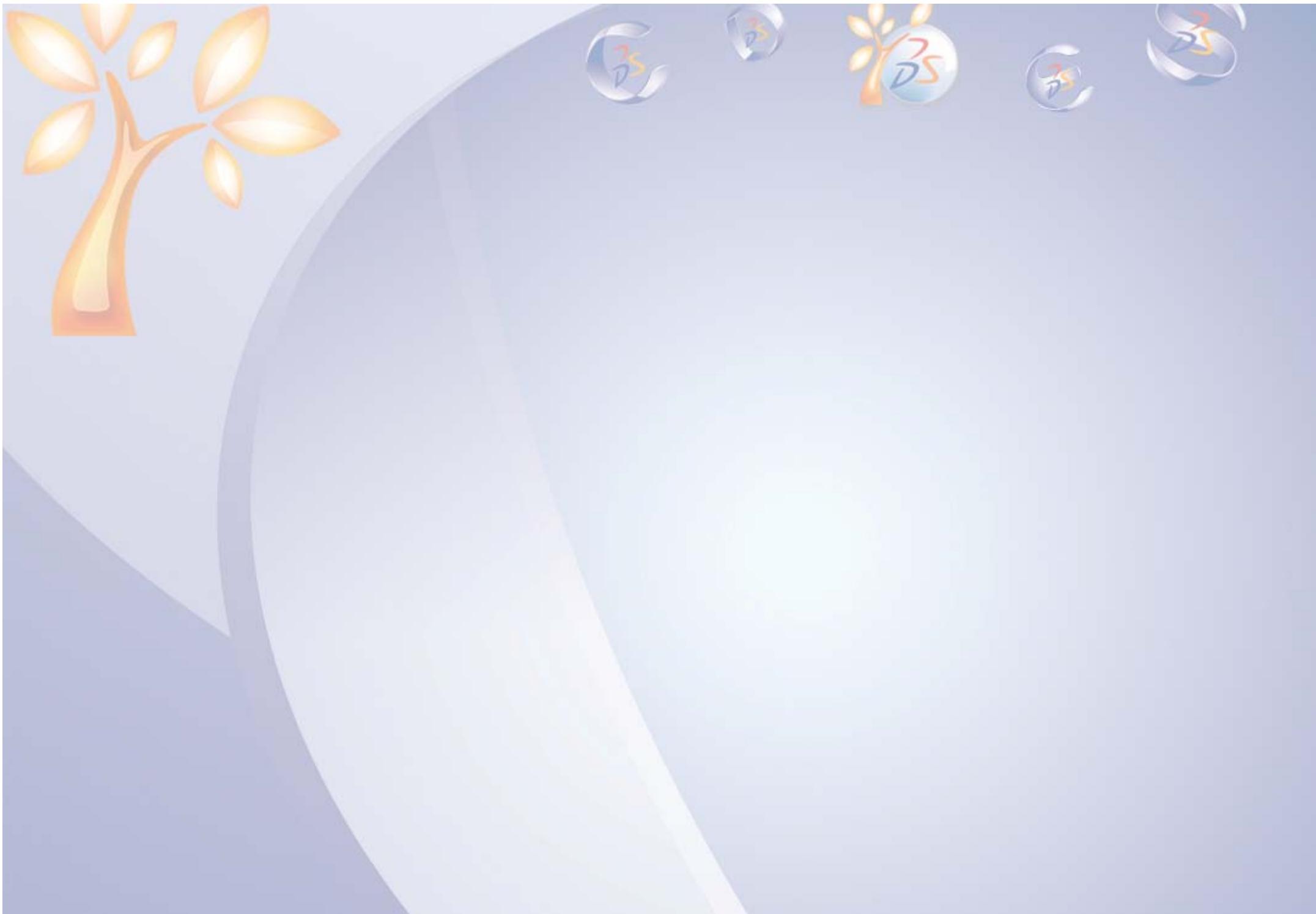
Search and load Exercise3_CaseStudy_Start.CATPart and add features to the part using the drawing provided here.



Case Study Recap: Timing Chain Cover

- ✓ Determine a suitable base feature
- ✓ Create pad features
- ✓ Create pockets and holes
- ✓ Create fillets and chamfers
- ✓ Save and close document





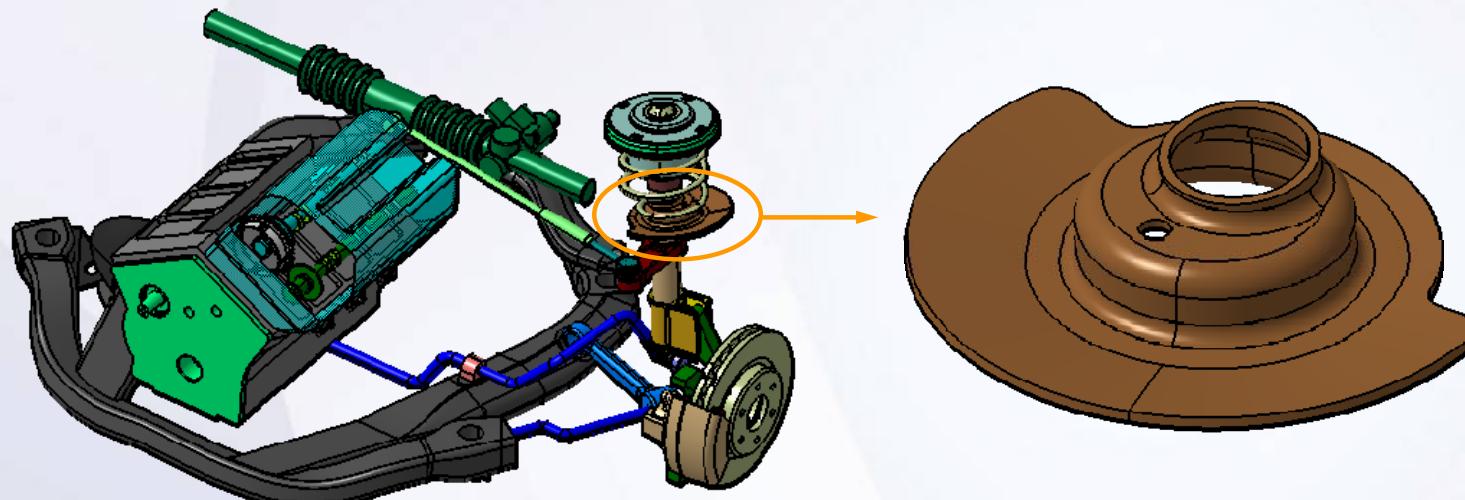
4

Additional Features

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Create Feature Profiles and Axis Systems
- ✓ Create Shaft and Groove Features
- ✓ Create Basic Wireframe Geometry
- ✓ Shell the Model



4 Hours

Case Study

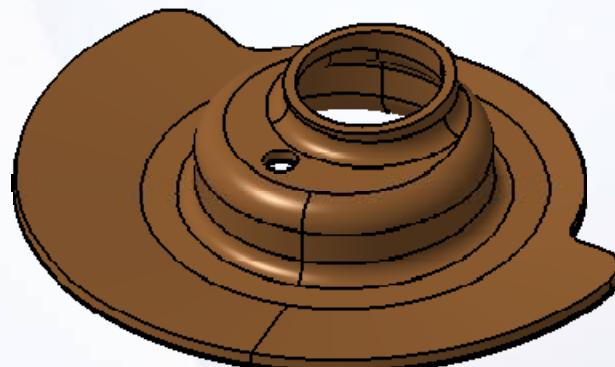
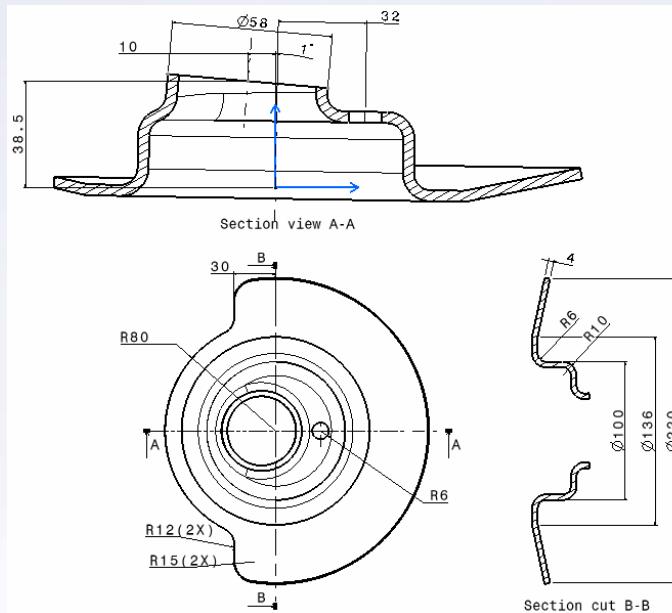
The case study for this lesson is the Suspension Seat used in the Front Suspension and Engine assembly.

Design Intent

- ✓ The axis main flange must be at 5 degrees from Z axis.
- ✓ The main flange must be at 12 degrees from horizontal plane.
- ✓ One large hole of diameter 50 mm must be created for Pillar clearance.
- ✓ The sketch of seat must be 4 mm.
- ✓ There must not be any sharp corners.

Stages in the Process

1. Create a reference geometry.
2. Create an axis system.
3. Create a sketched geometry.
4. Create shafts.
5. Create fillets.
6. Create a pocket.
7. Shell the model.
8. Create a hole.

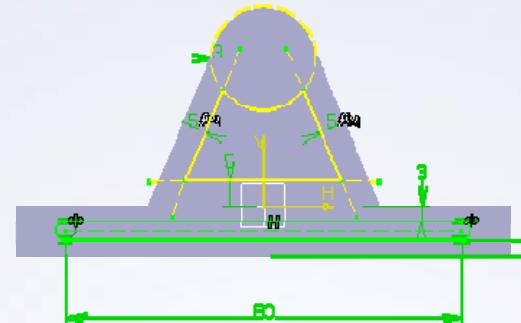


Create Feature Profiles and Axis System

Lesson 2 introduced you to the basic Sketcher tools and the Sketcher environment. This lesson will introduce you to the advanced Sketcher tools.

Sketcher includes the following additional tools:

- ✓ Re-limitation tools.
- ✓ Transformation tools.
- ✓ Project 3D element tools.
- ✓ Analyze a sketch using the Sketch Analysis tool.

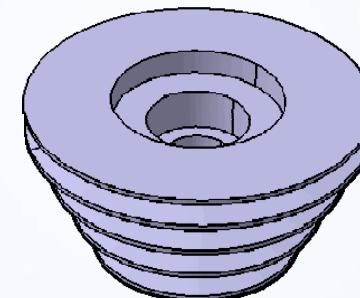


Create Shaft and Groove Features

A revolved feature is created by revolving a 2D profile around an axis of revolution. In the Part Design workbench, you can create two types of revolved features.

The axis of revolution for a revolved feature can be created inside the sketch containing the profile, using the Axis tool.

If you did not create an axis in the sketch you can define it from the Shaft/Groove definition window in the Axis selection field. Any linear element in the model can be used.



Create Basic Wireframe Geometry

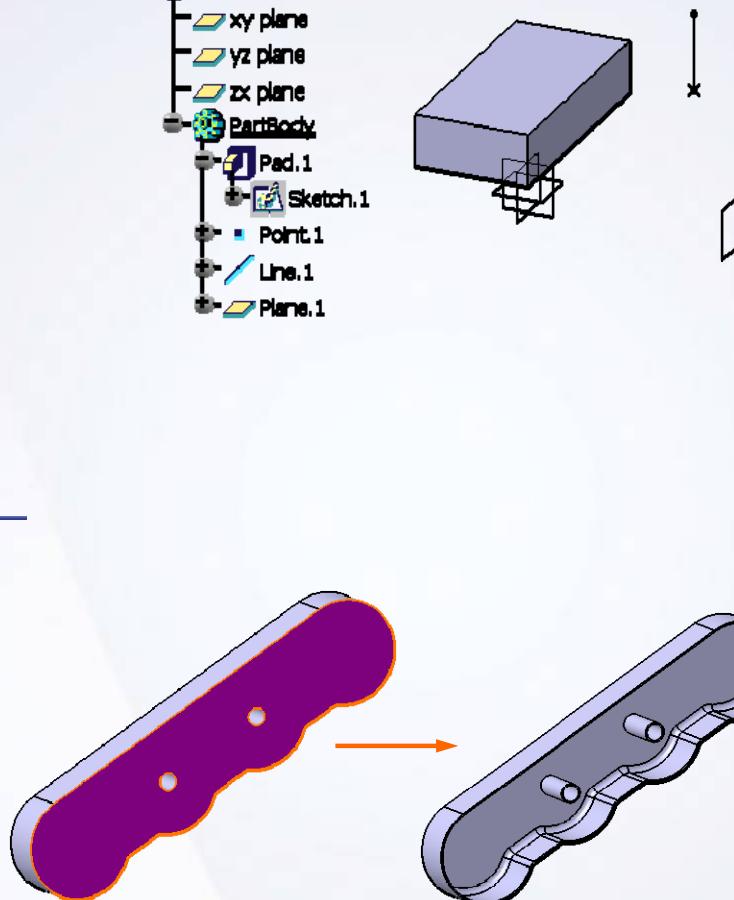
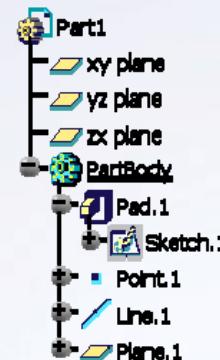
In the Part Design workbench, you have the ability to create points, lines, and planes outside of the Sketcher environment. These elements are called reference or 3D wireframe geometry.

Depending on how the part was initially created, these elements can be represented in the specification tree in two ways. If the Enable hybrid design option is selected, CATIA will place these features within the main PartBody. If the Enable hybrid design option is cleared, wireframe elements are inserted under a group called a Geometrical set. Geometrical sets contain only 3D wireframe and surface elements and not solid geometry.

Shell the Model

Shelling a feature hollows out solid geometry. The shelling operation removes one or more faces from the solid and applies a constant thickness to the remaining faces. You can also apply a different thickness to the selected faces.

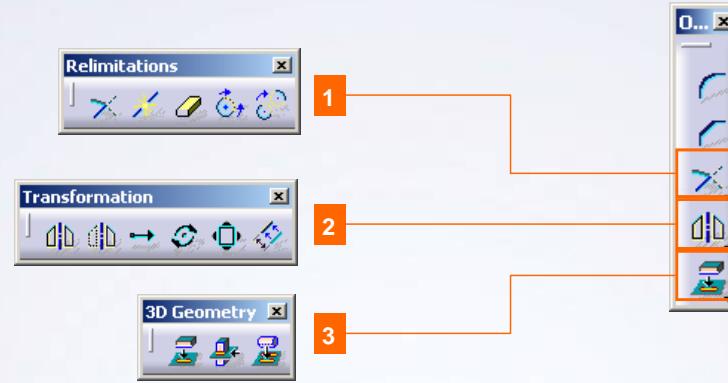
While shelling a model, it is important to consider the feature order. The Shell operation hollows all solid features in a model. If you do not want a feature to be shelled, it must be created after the shell operation.



Additional Sketcher Tools

Operation

- 1 **Relimitations:** trim or extend the existing sketched geometry
- 2 **Transformation:** modify existing sketcher geometry
- 3 **3D Geometry:** project the existing 3D elements onto the sketch plane



Tools

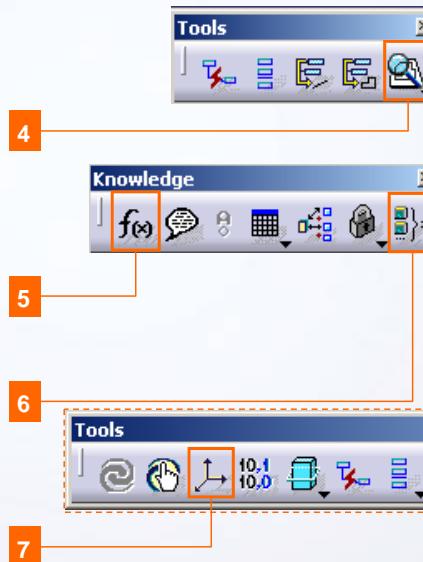
- 4 **Sketch Analysis:** help to resolve problems with a sketch

Knowledge

- 5 **Formula:** create Relationships between Dimensions
- 6 **Equivalent dimensions:** equates all selected parameters to a value

Tools

- 7 **Axis System:** used to define local coordinates



Additional Part Design Tools

Reference Elements

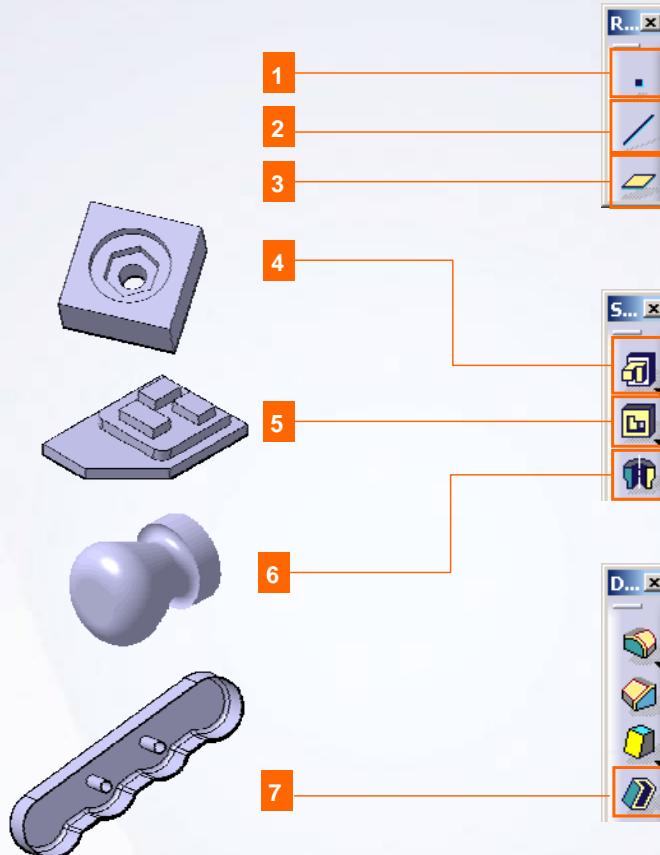
- 1 **Point:** creates a point in 3D space
- 2 **Line:** creates a line in 3D space
- 3 **Plane:** creates a plane in 3D space

Sketch-Based Features

- 4 **Multi-pad:** creates several pads in one operation
- 5 **Multi-pocket:** creates several pockets in one operation
- 6 **Shaft:** helps to resolve problems with the sketch

Dress-Up Features

- 7 **Shell:** removes one or more faces from the solid and applies a constant thickness to the remaining faces



Exercise: Additional Sketcher Tools

Recap Exercise

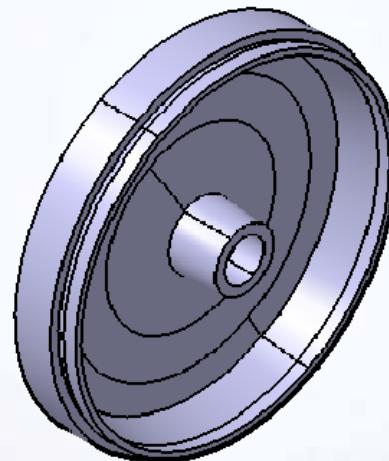


15 min

In this exercise you will open an existing part that contains a positioned sketch. You will add a flange to this sketch and constrain it fully. High-level instructions for this exercise are provided.

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

- Create a positioned sketch
- Use transformation tools in sketcher
- Use re-limitation tools in sketcher
- Use Equivalent Dimensions
- Create a Formula

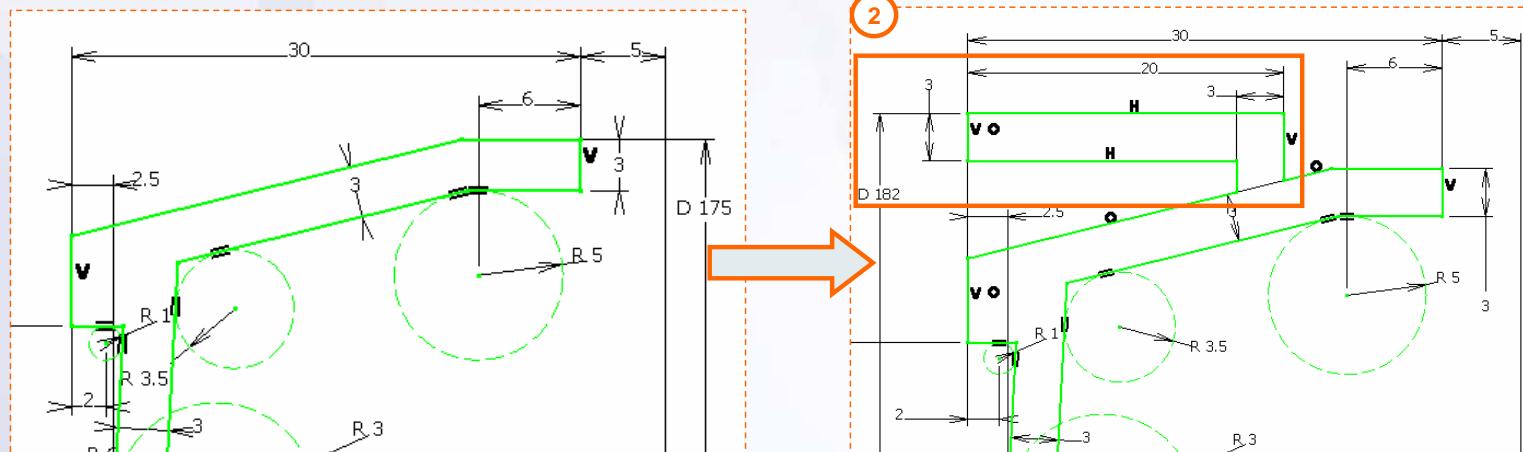
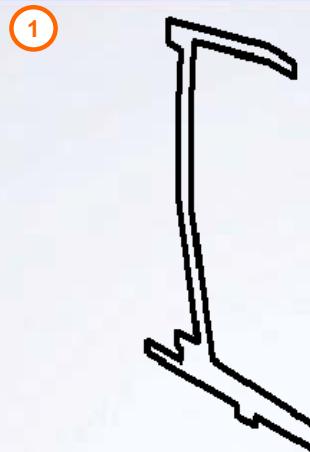


Do it Yourself (1/5)

1. Load Ex4B.CATPart.

2. Edit the sketch.

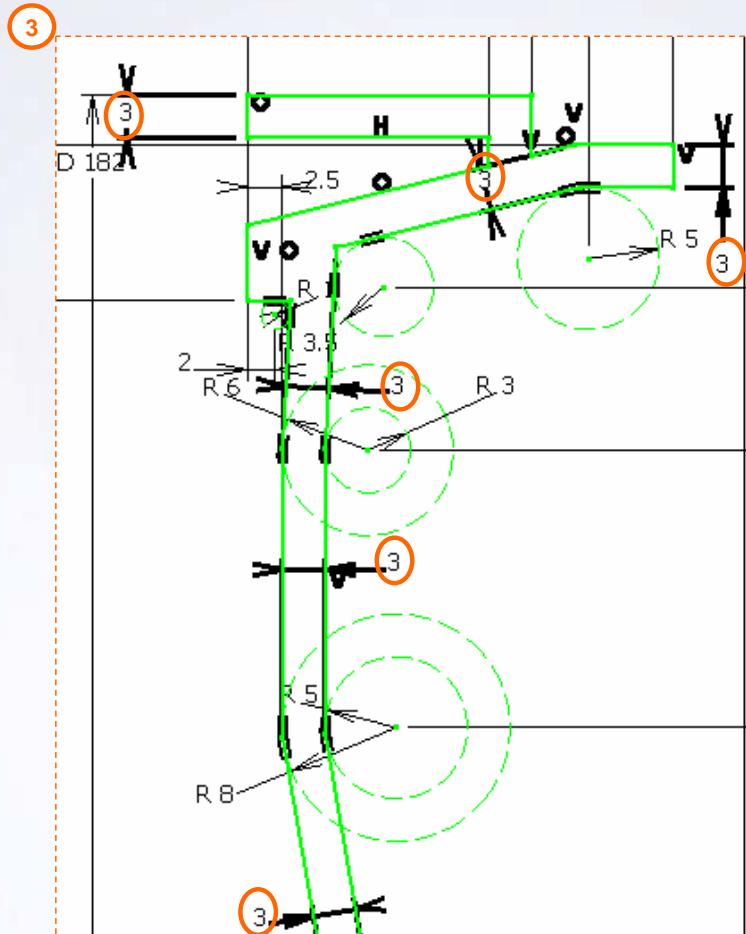
- Access the Sketcher workbench for Sketch.1 and add a flange geometry as shown. Use transformation tools (Offset) and re-limitation tools (Break, Trim) to create geometry.



Do it Yourself (2/5)

3. Create Equivalent Dimensions for thickness parameters.

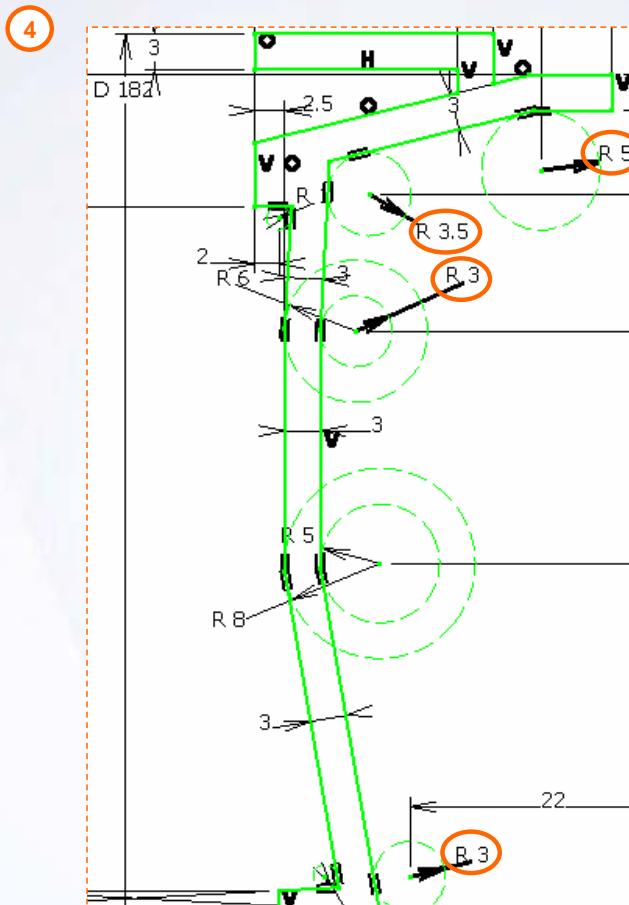
- Multiselect all thickness dimensions as shown, and create Equivalent Dimensions of [3mm]. By this, you can have better control on the thickness constraint.



Do it Yourself (3/5)

4. Create Equivalent Dimensions for inner radius.

- Multiselect the inner radius dimensions as shown, and create Equivalent Dimensions of [3mm]. By this, you can have better control on the inner radius constraint.

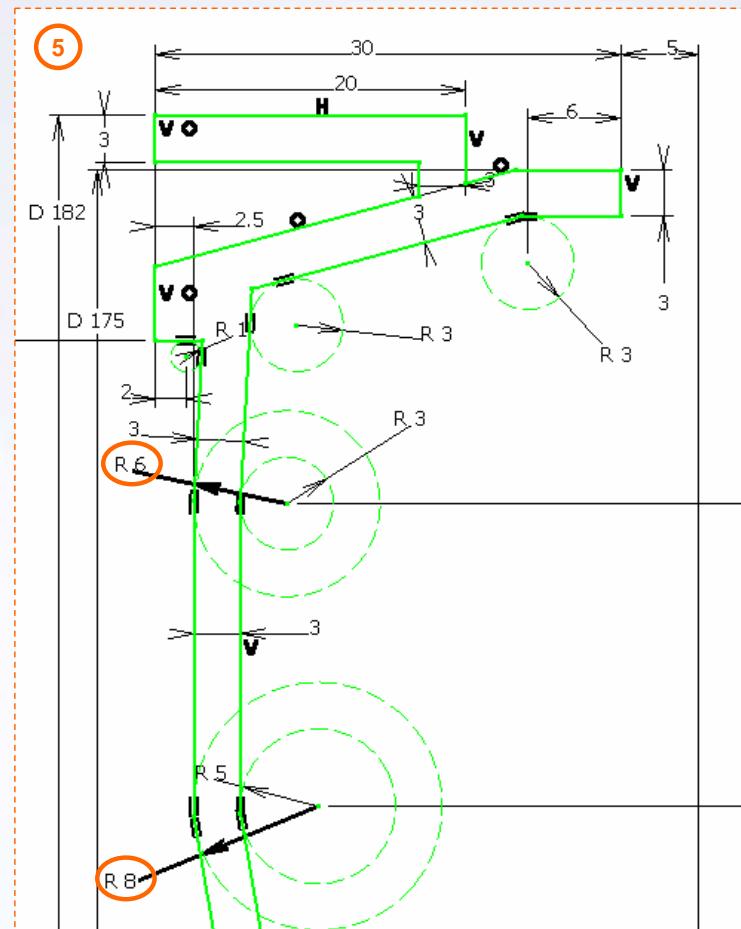


Do it Yourself (4/5)

5. Create Formula for outer radius.

- Create a formula for each of the highlighted outer radius,
 $\text{Outer radius} = \text{Inner radius} + \text{Thickness.}$

By this you can have better control on outer radius. A change in the value of inner radius and/or thickness will be reflected in outer radius.



Do it Yourself (5/5)

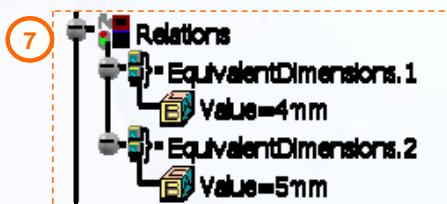
6. Create a shaft.

- Revolve the sketch to get a solid.



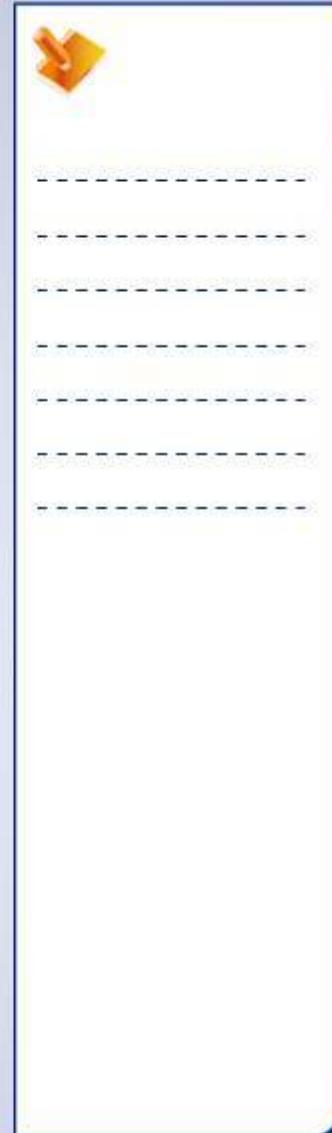
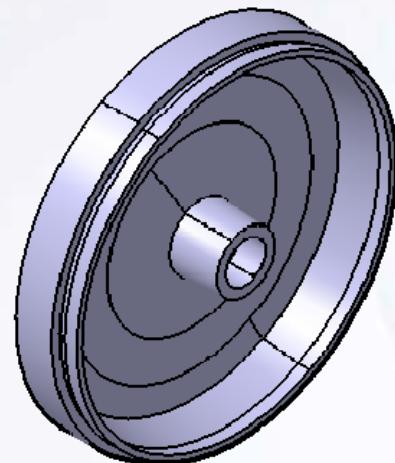
7. Modify value of equivalent dimensions.

- Modify thickness to [4] and inner radius to [5] from Relations node of feature tree.



Exercise Recap: Additional Sketcher Tools

- ✓ Create a position sketch
- ✓ Use transformation tools in sketcher
- ✓ Use re-limitation tools in sketcher
- ✓ Use equivalent dimensions
- ✓ Create a formula



Exercise: Additional Sketcher Tools

Recap Exercise

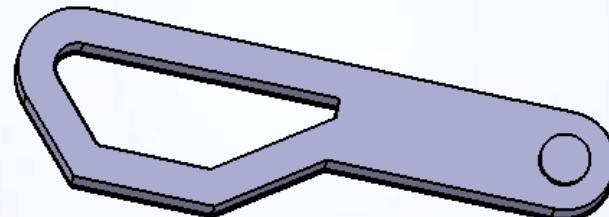


15 min

In this exercise, you will perform the sketch analysis, and create pads and pocket. You will use the tools you have learned so far, to complete the exercise with no detailed instruction.

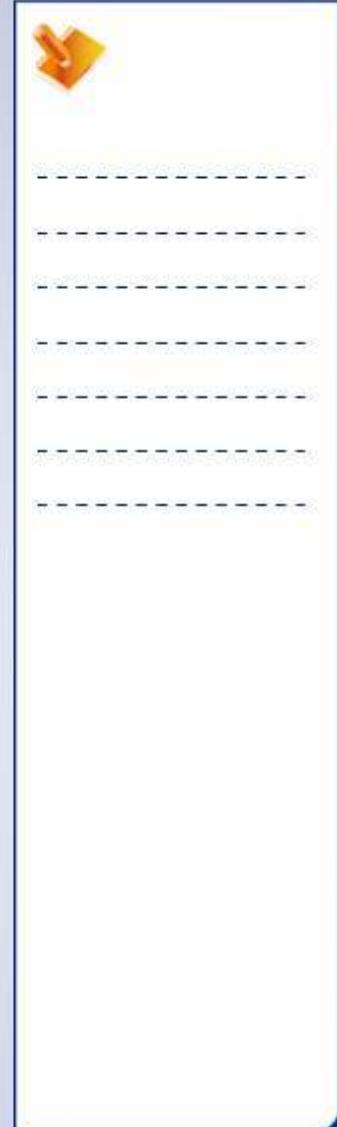
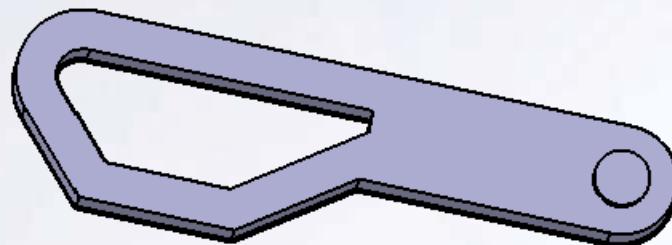
By the end of this exercise you will be able to:

- Solve a sketch related problem
- Use the sketch analysis tool and take corrective actions
- Create a pad
- Create a pocket



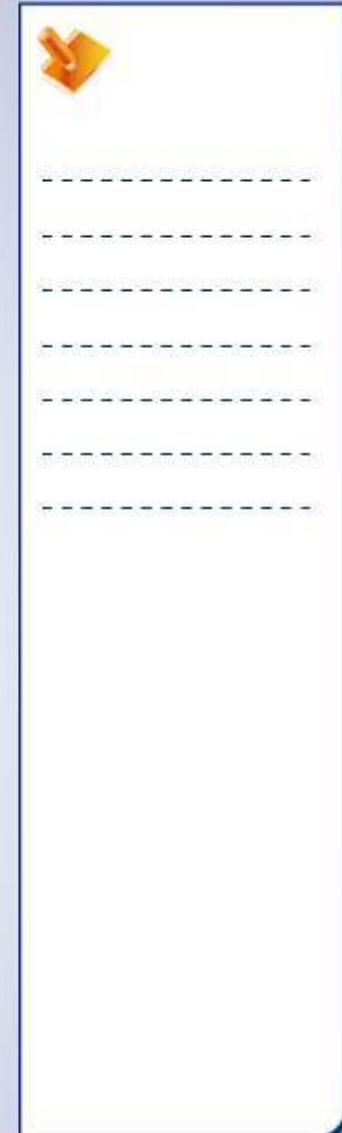
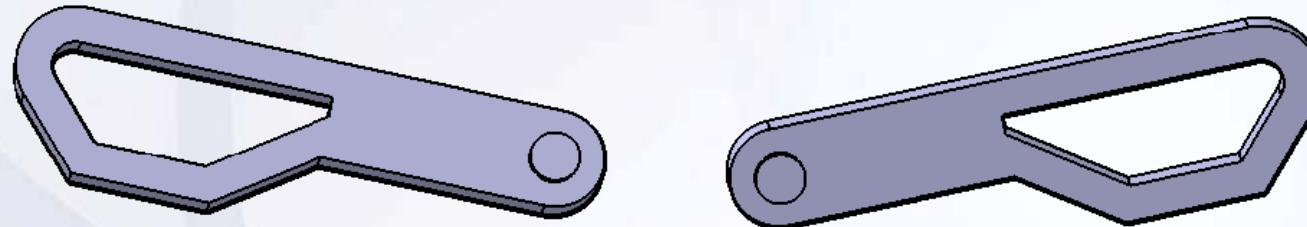
Do it Yourself

1. Load Ex4C.CATPart and create pads and pocket. Perform sketch analysis and take corrective actions.



Exercise Recap: Additional Sketcher Tools

- ✓ Solve a sketch related problem
- ✓ Use the sketch analysis tool and take corrective actions
- ✓ Create a pad
- ✓ Create a pocket



Exercise: Fillet and Chamfer

Recap Exercise

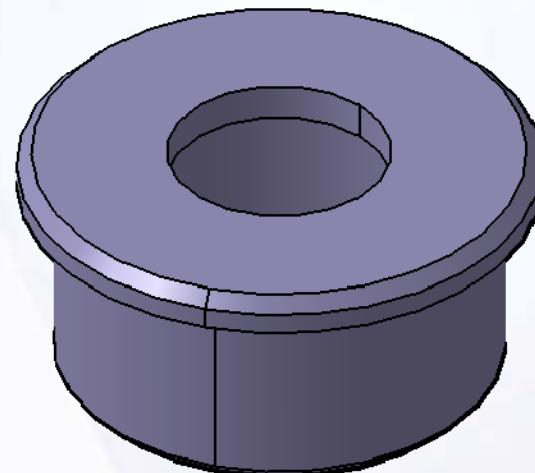


15 min

In this exercise, you will create a new part. Using shaft, pockets, fillets and chamfer, you will construct a damper assembly cap. High-level instructions for this exercise are provided.

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

- Create a shaft
- Create a pocket
- Create an edge fillet
- Create a chamfer



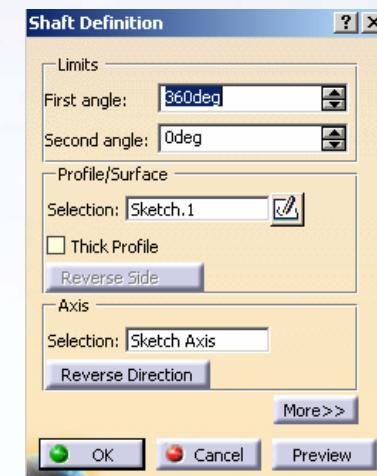
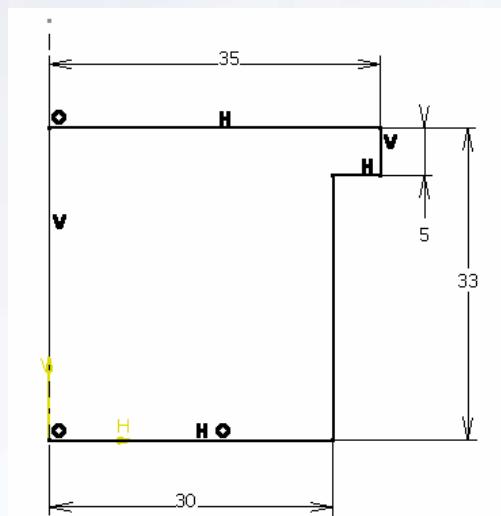
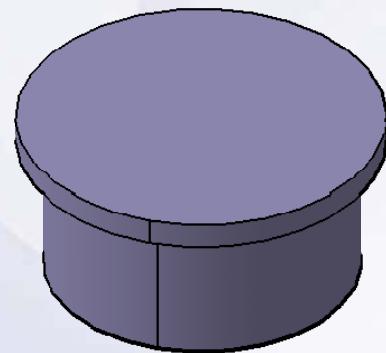
Do it Yourself (1/5)

1. Create a new part file.

- Create a new part file called [Ex4E.CATPart].

2. Create a shaft feature.

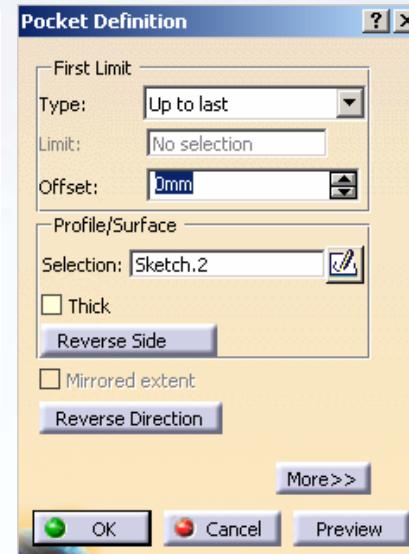
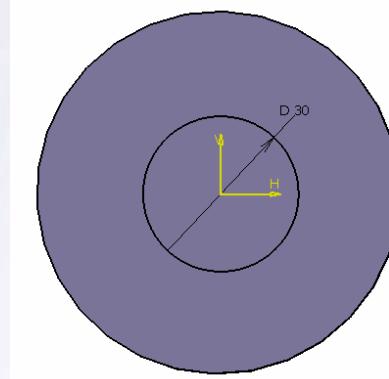
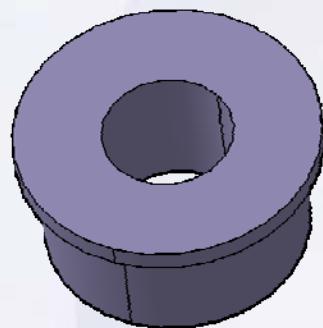
- Create the profile shown to construct a shaft feature.



Do it Yourself (2/5)

3. Create a pocket feature.

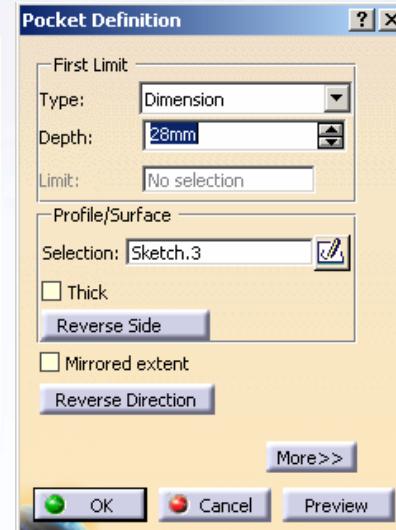
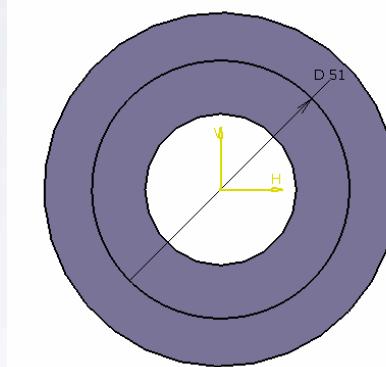
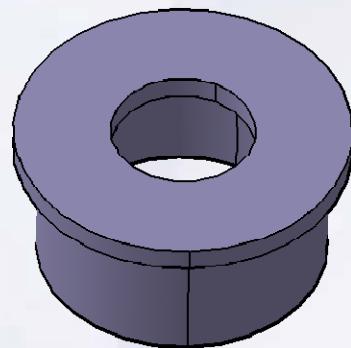
- Create the profile shown to construct a pocket feature.



Do it Yourself (3/5)

4. Create a pocket feature.

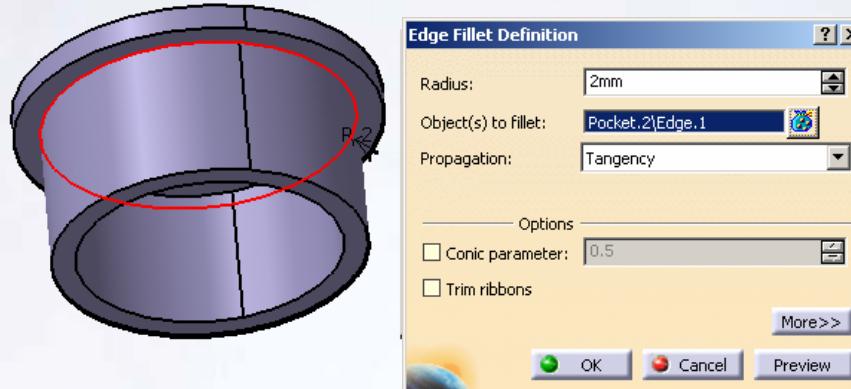
- Create the profile shown to construct a pocket feature. Enter depth = [28mm].



Do it Yourself (4/5)

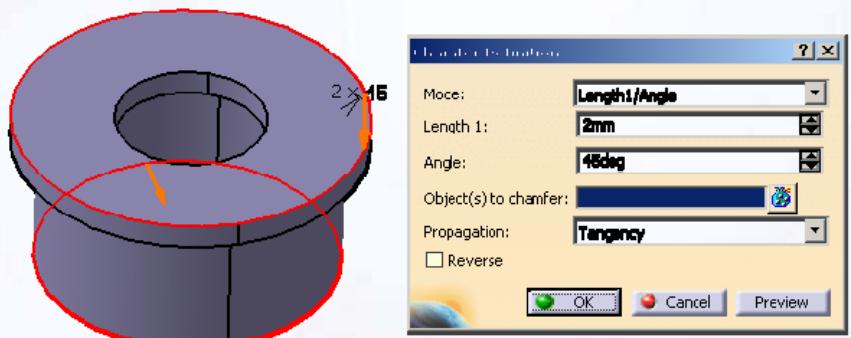
5. Create an edge fillet.

- Create an edge fillet on one edge as shown (radius = 2mm).



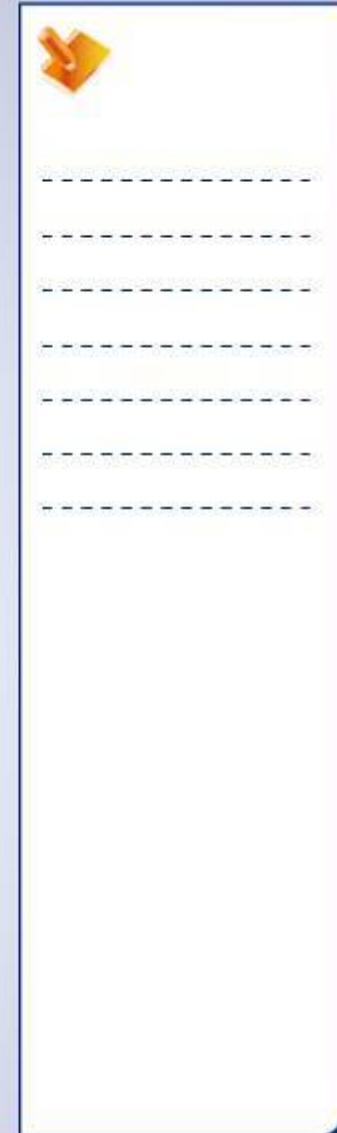
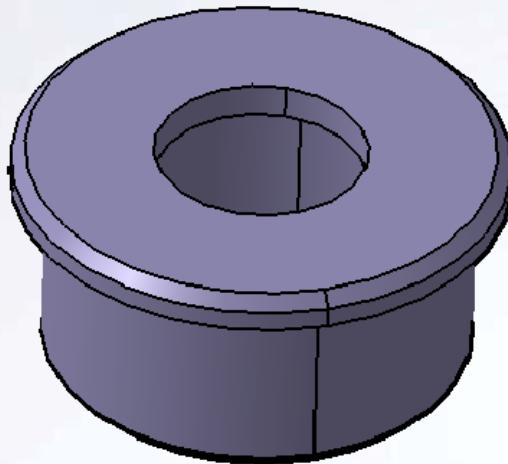
6. Create a chamfer.

- Create a chamfer on 2 edges as shown (length = 2mm and angle = 45deg).



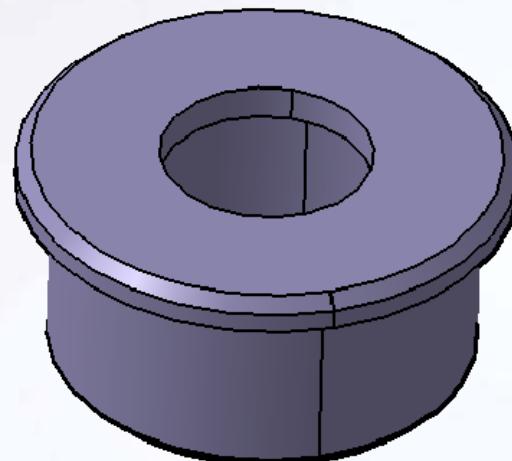
Do it Yourself (5/5)

7. Save and close the part.



Exercise Recap: Fillet and Chamfer

- ✓ Create a shaft feature
- ✓ Create a pocket feature
- ✓ Create a fillet feature
- ✓ Create a chamfer feature



Exercise: Shaft and Groove

Recap Exercise

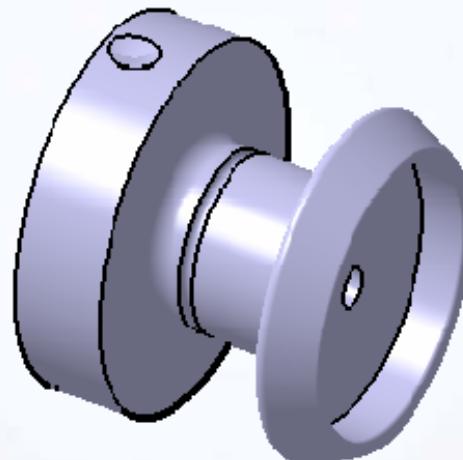


15 min

In this exercise, you will create a part that contains features taught in this and the previous lessons. You will use the tools learned in this lesson to complete the exercise with no detailed instructions.

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

- Create a shaft feature
- Create edge fillets
- Create internal and external groove features
- Create a pocket feature
- Create a reference point and line
- Create a cone-shaped groove feature



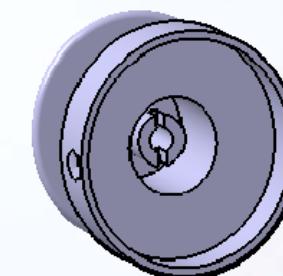
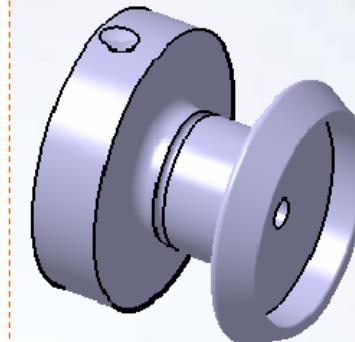
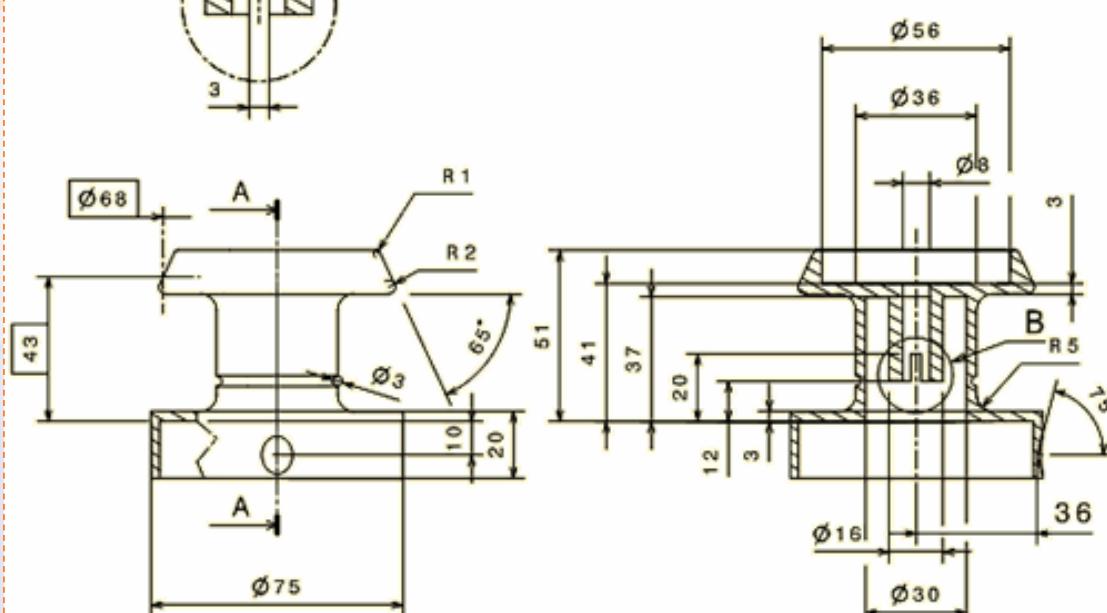
Do it Yourself

1. Create the following spool part.

Detail B

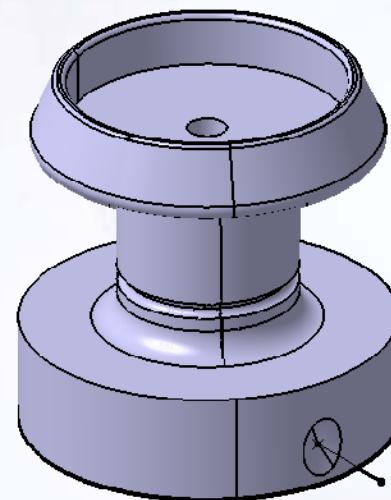
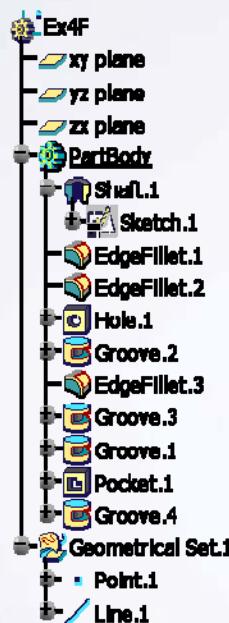


Section view A-A



Exercise Recap: Shaft and Groove

- ✓ Create a shaft feature
- ✓ Create edge fillets
- ✓ Create internal and external groove features
- ✓ Create a pocket feature
- ✓ Create a reference point and line
- ✓ Create a cone-shaped groove feature



Exercise: Features from Wireframe

Recap Exercise

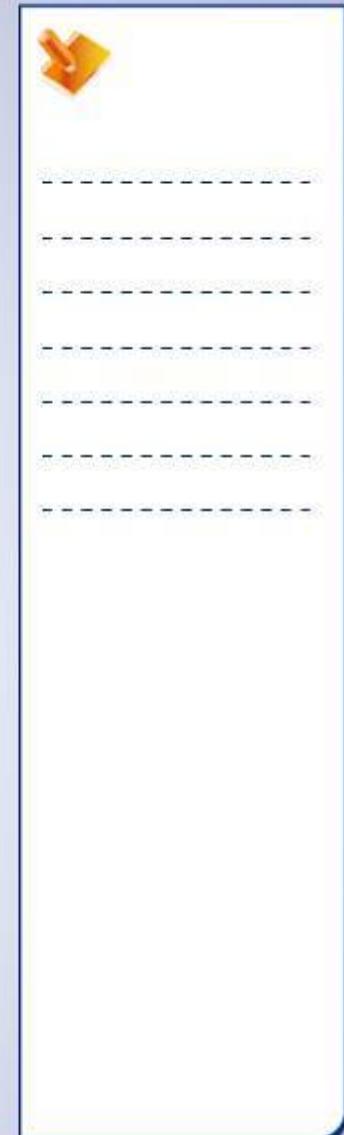
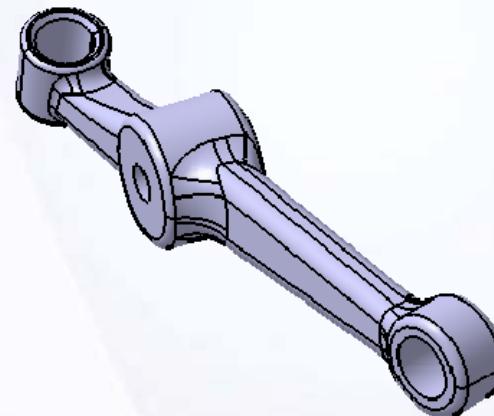


20 min

In this exercise you will open an existing part that contains a sketch. You will use this sketch to create pads, fillets, and holes feature. High-level instructions for this exercise are provided.

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

- Create a pad
- Create a fillet
- Create a hole

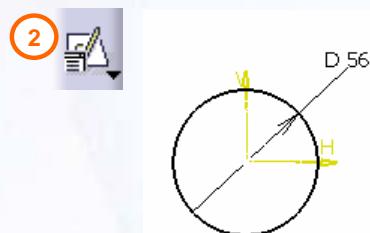
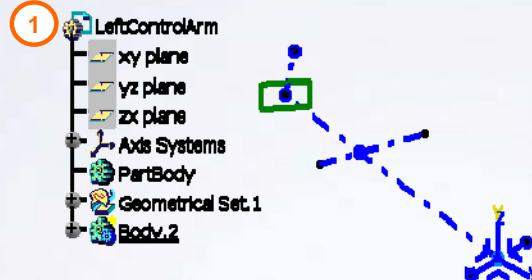


Do it Yourself (1/13)

1. Load Ex4H.CATPart from database.

2. Create a pad.

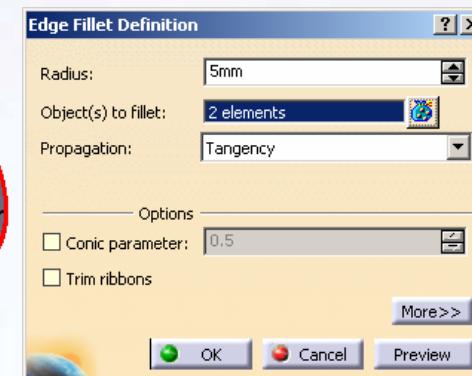
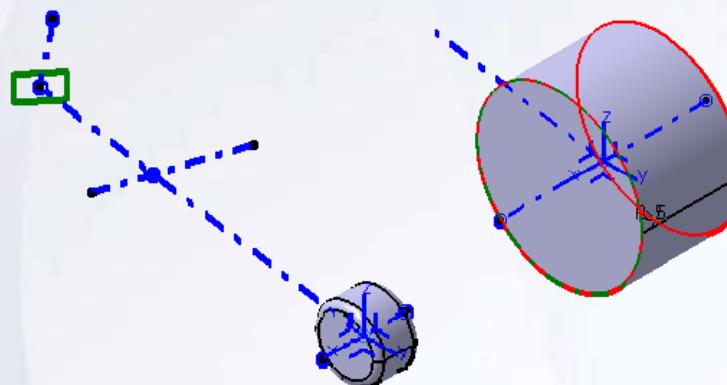
- Create a positioned sketch on YZ plane of AxisSystem.1 and create a pad with length 1 of [15mm] and length 2 of [19mm].



Do it Yourself (2/13)

3. Create edge fillets.

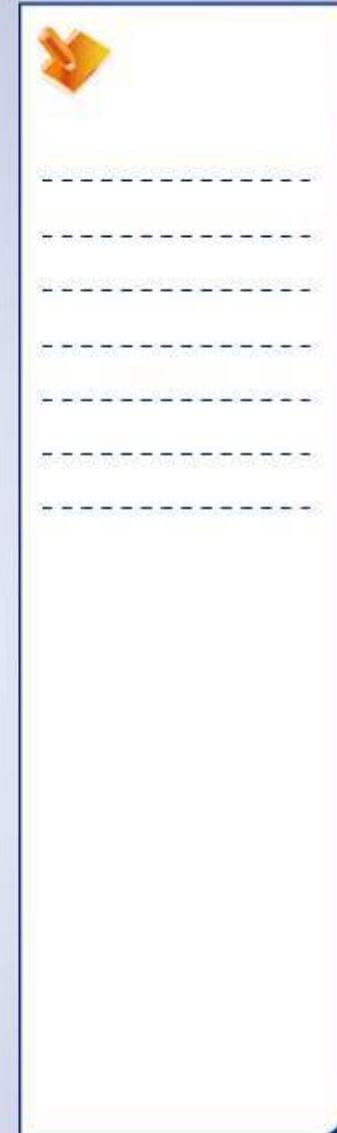
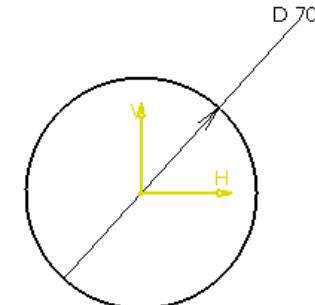
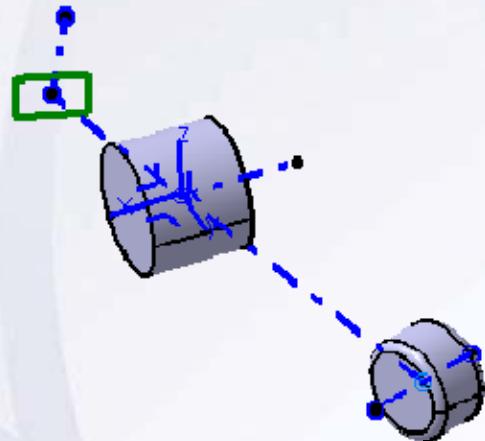
- Create an edge fillet feature. Select the two edges of pad and apply fillet of [5mm].



Do it Yourself (3/13)

4. Create a pad.

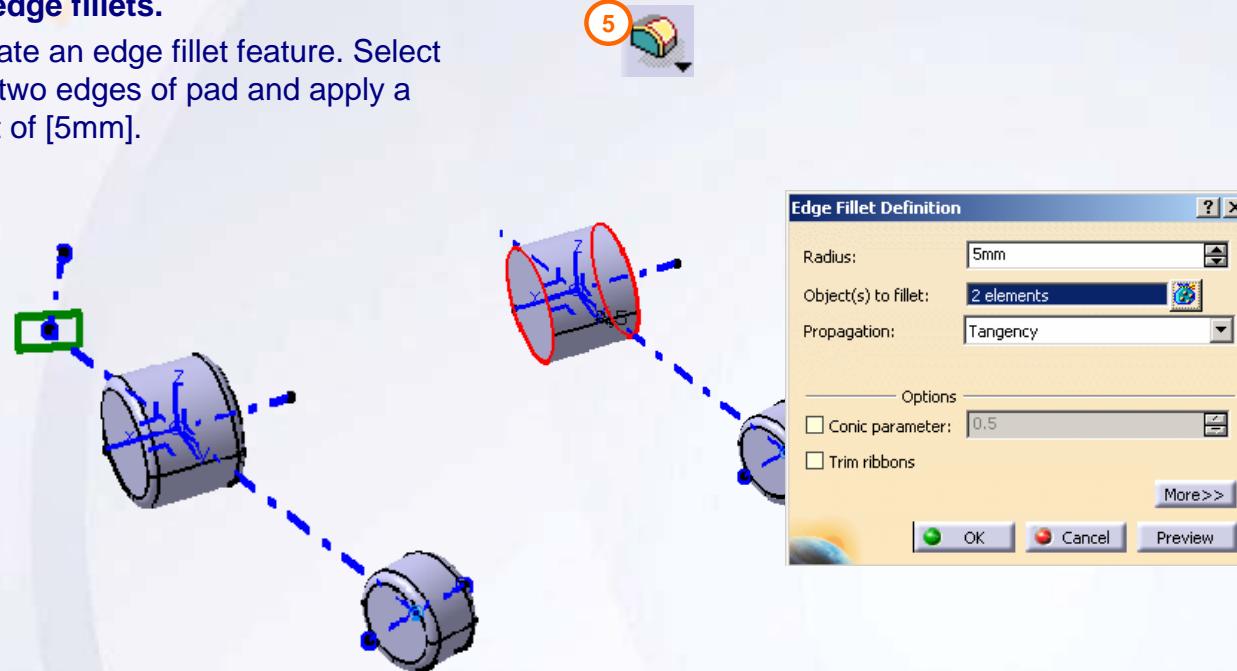
- Create a positioned sketch on YZ plane of AxisSystem.2 and create a pad with length 1 of [30mm] and length 2 of [25mm].



Do it Yourself (4/13)

5. Create edge fillets.

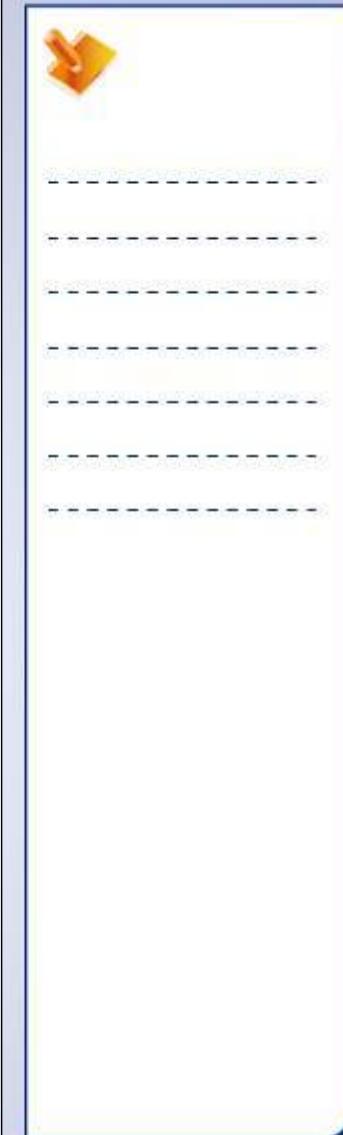
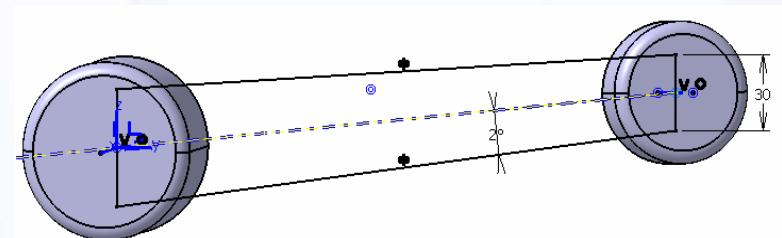
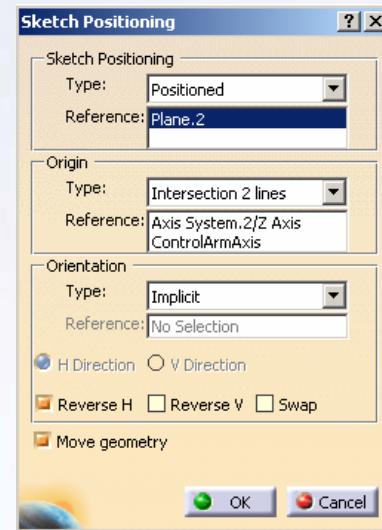
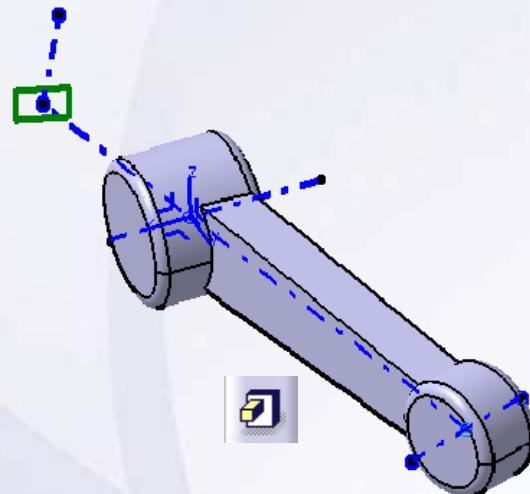
- Create an edge fillet feature. Select the two edges of pad and apply a fillet of [5mm].



Do it Yourself (5/13)

6. Create a pad.

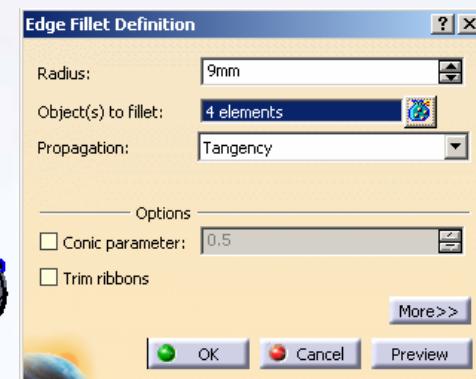
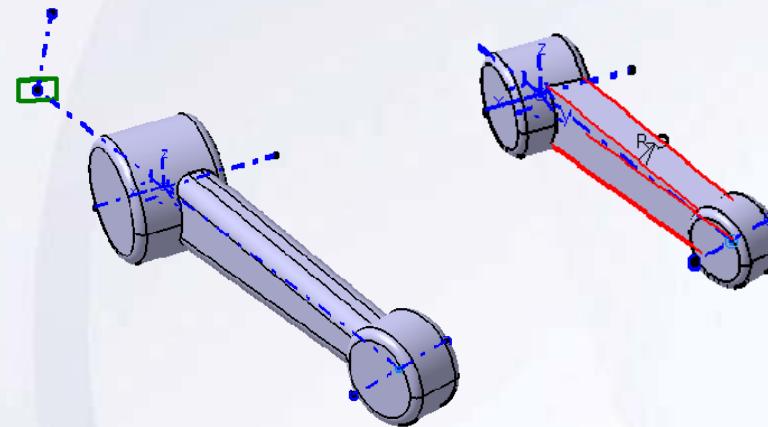
- Create a positioned sketch on Plane.2 and create a pad with length 1 of [15mm] and length 2 of [10mm].



Do it Yourself (6/13)

7. Create edge fillets.

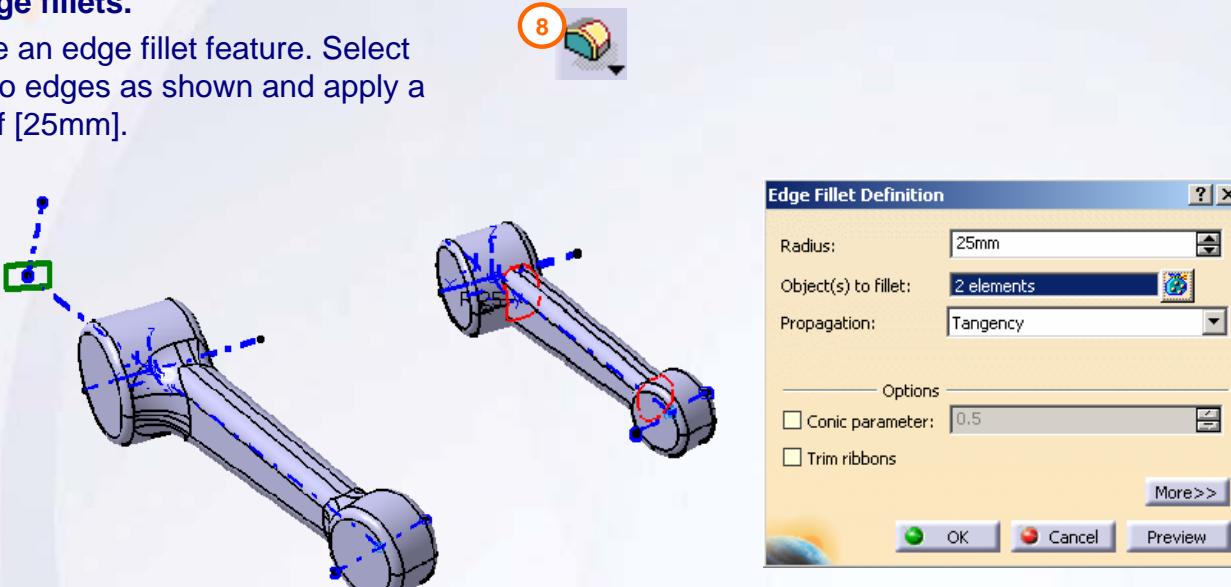
- Create an edge fillet feature. Select the four edges of the pad and apply a fillet of [9mm].



Do it Yourself (7/13)

8. Create edge fillets.

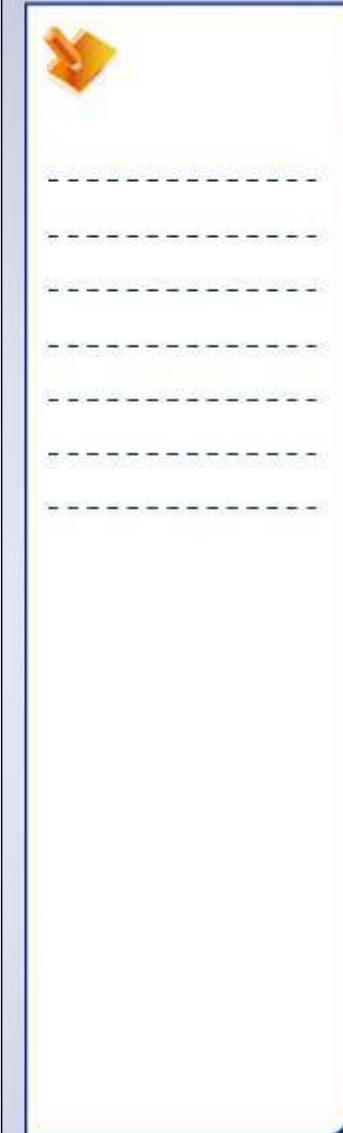
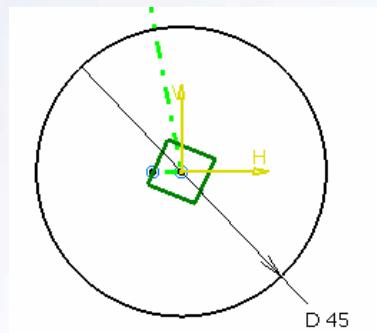
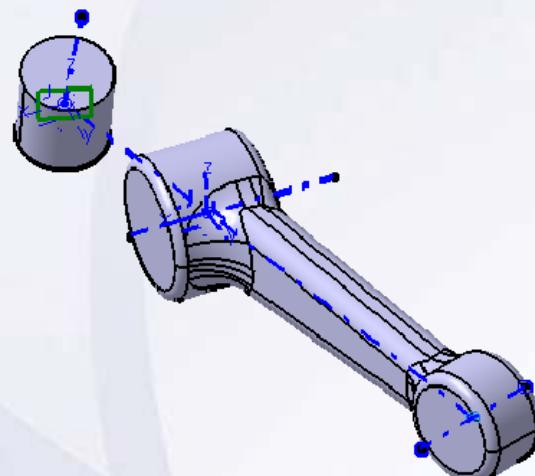
- Create an edge fillet feature. Select the two edges as shown and apply a fillet of [25mm].



Do it Yourself (8/13)

9. Create a pad.

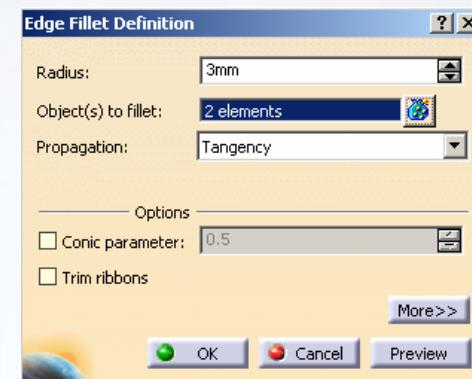
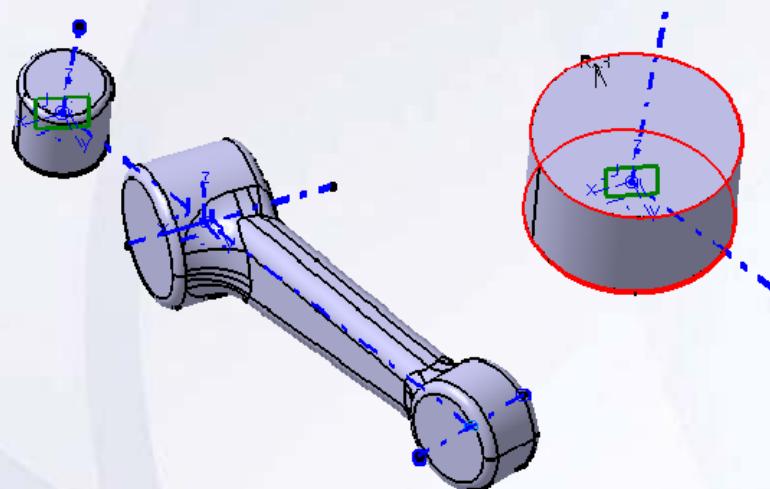
- Create a positioned sketch on XY plane of Axis System.3 and create a pad with length 1 of [22mm] and Mirrored Extent.



Do it Yourself (9/13)

10. Create edge fillets.

- Create an edge fillet feature. Select the two edges of pad and apply a fillet of [3mm].

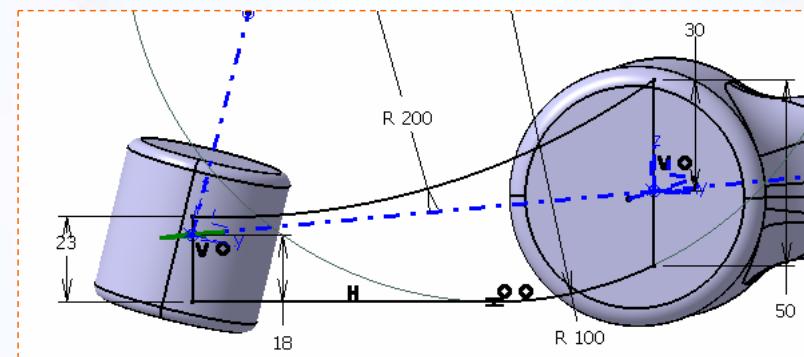
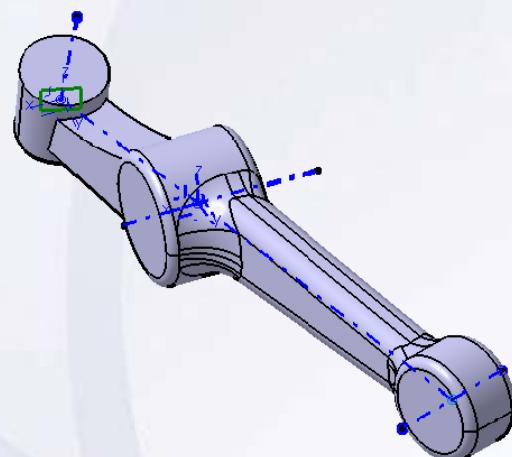
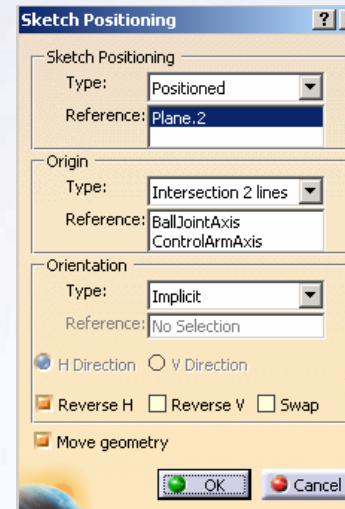


Do it Yourself (10/13)

11. Create a pad.

- Create a positioned sketch on Plane.2 and create a pad with length 1 of [15mm] and mirrored extent.

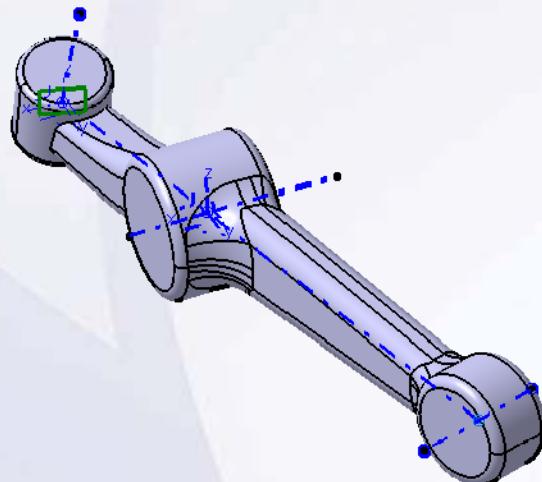
(11)



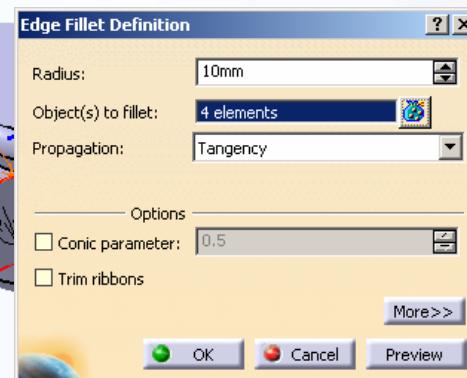
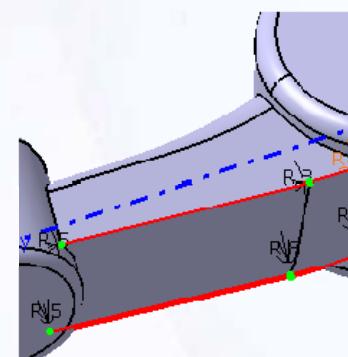
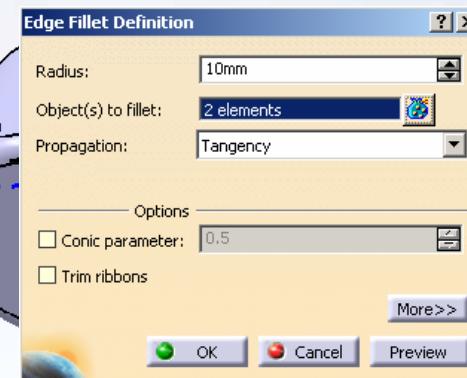
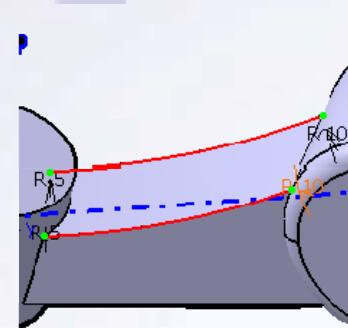
Do it Yourself (11/13)

12. Create variable edge fillets.

- Create a variable edge fillet feature. Select the two edges of pad. Radius values are 5mm and 10mm.
- Create a variable edge fillet feature. Select the four edges as shown. Radius values are 5mm, 8mm, 10mm.



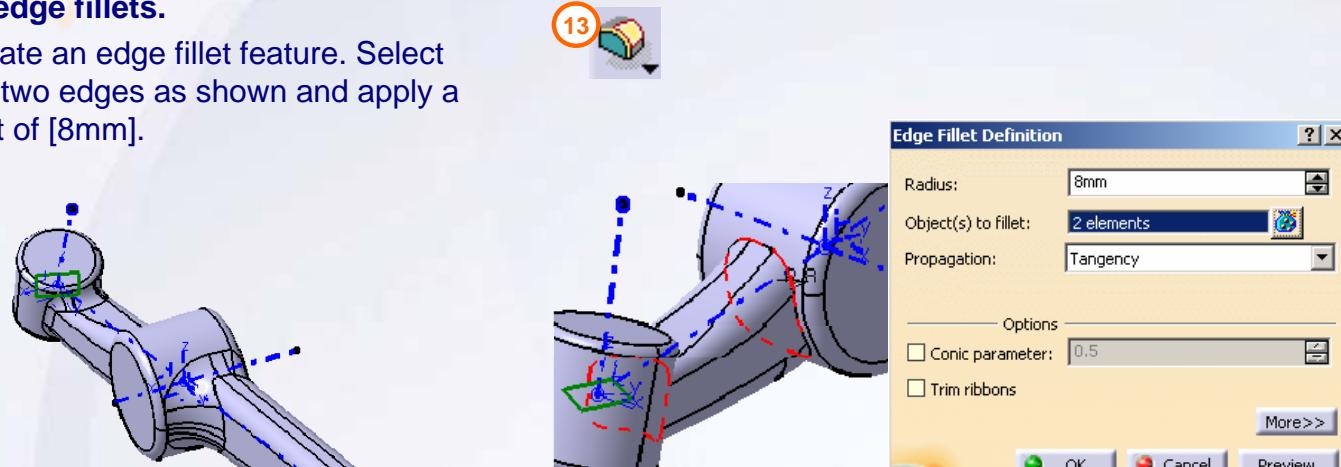
12



Do it Yourself (12/13)

13. Create edge fillets.

- Create an edge fillet feature. Select the two edges as shown and apply a fillet of [8mm].



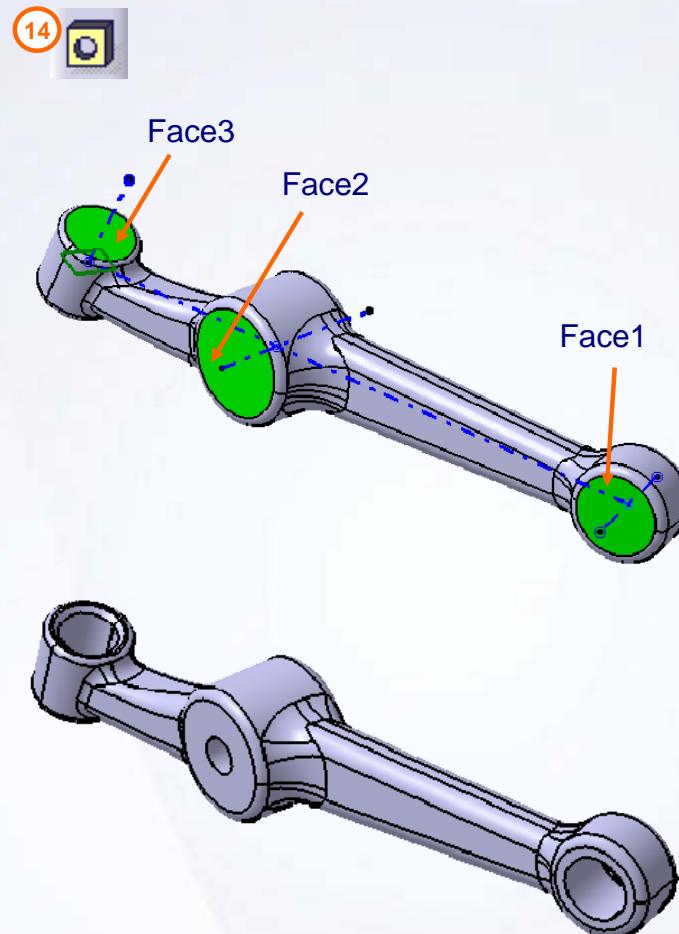
Do it Yourself (13/13)

14. Create holes.

- Create a simple hole of diameter [35mm] on face1.
- Create a simple hole of diameter [20mm] on face2.
- Create a countersunk hole of diameter [30mm] on face3 and countersunk depth of [3mm].

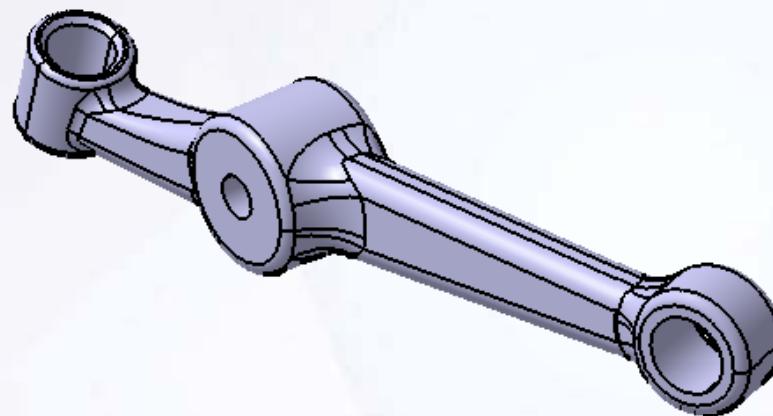
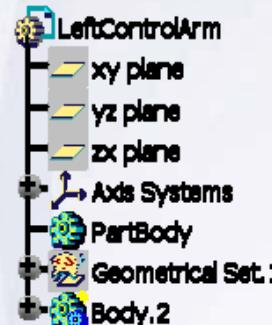
15. Close the part without saving it.

- Hide Axis system and Geometric Set.1.
- Close the part.



Exercise Recap: Features from Wireframe

- ✓ Create a pad
- ✓ Create a fillet
- ✓ Create a hole



Exercise: Shell and Holes

Recap Exercise

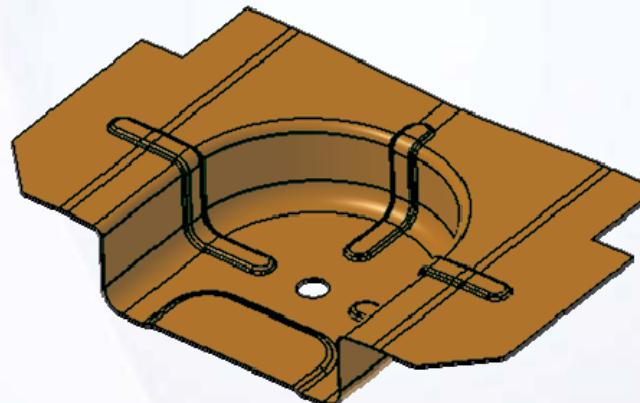


20 min

In this exercise, you will create a Spare wheel mount, that contains features learned in this and previous lessons. You will use the tools learned in this lesson to complete the exercise with no detailed instruction.

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

- Create a pad
- Create a pocket
- Create a chamfer
- Create a fillet
- Create a shell
- Create a hole



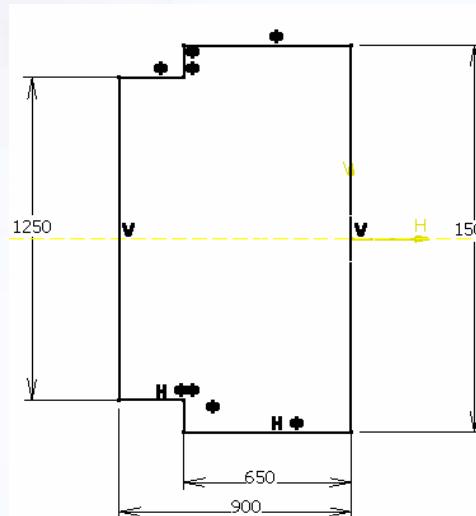
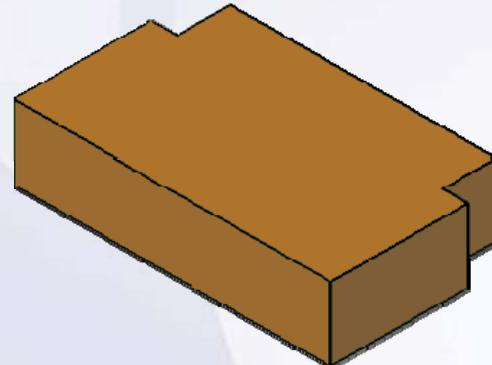
Do it Yourself (1/9)

1. Create a new part file.

- Create a new part file called [Ex4I.CATPart].

2. Create a pad.

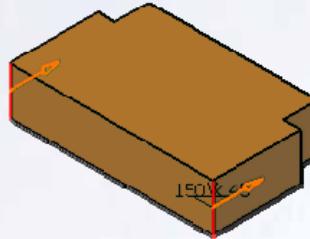
- Create a positioned sketch on XY plane and create a pad with length 1 of [350mm].



Do it Yourself (2/9)

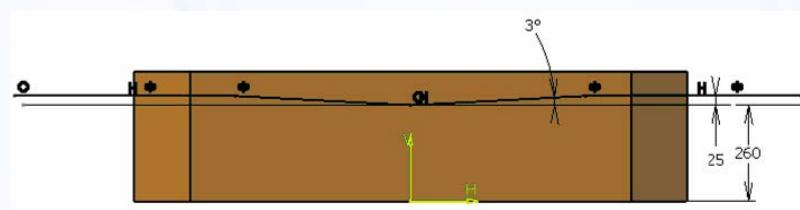
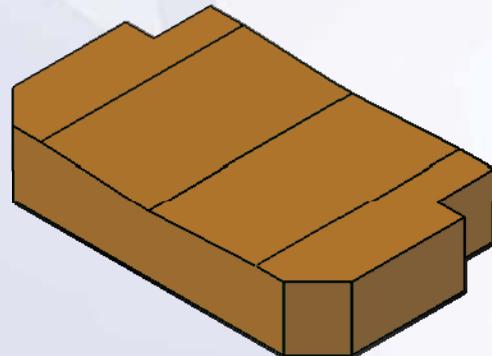
3. Create a chamfer.

- Create a chamfer on two edges. Enter Length 1 as [150mm] and Angle as [45deg].



4. Create a pocket.

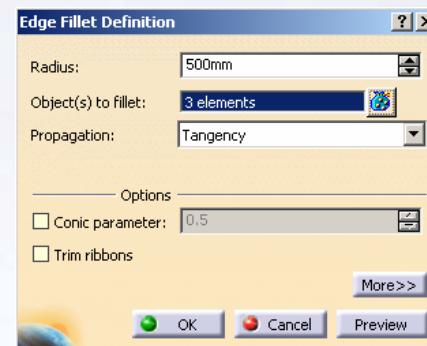
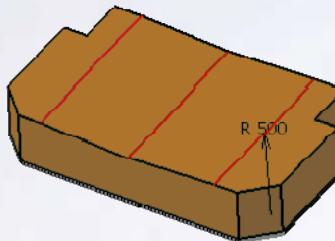
- Create a positioned sketch on YZ plane and create a pocket of depth Upto next.



Do it Yourself (3/9)

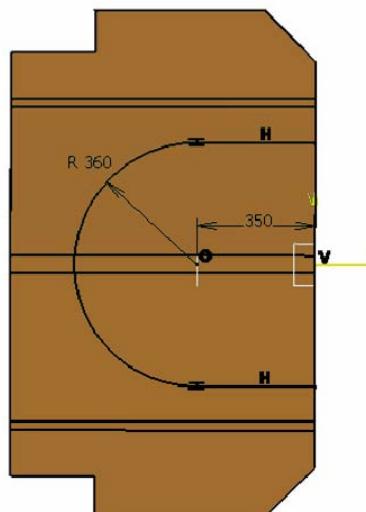
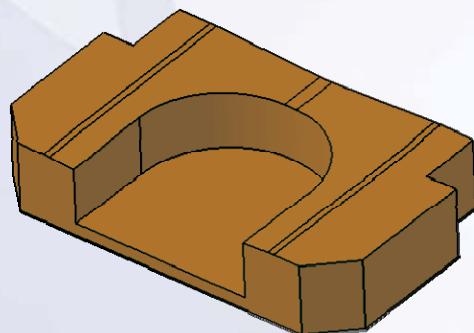
5. Create a fillet.

- Create an edge fillet on three edges. Select the edges as shown and apply a fillet of [500mm].



6. Create a pocket.

- Create a positioned sketch on a plane 350mm offset from XY plane and create a pocket of depth [300mm].



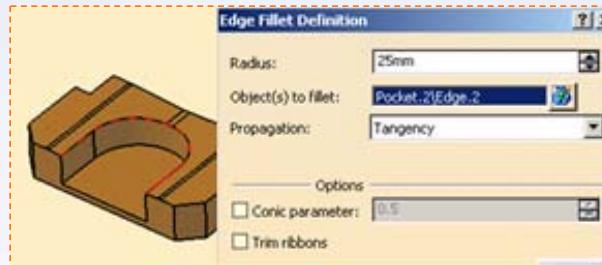
Do it Yourself (4/9)

7. Create fillets.

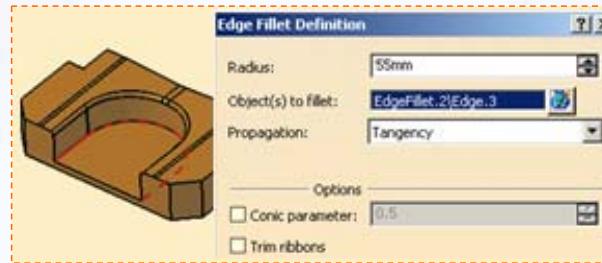
- Create edge fillets as shown.
 - a. Create an edge fillet of [25mm] on one edge.
 - b. Create an edge fillet of [55mm] on one edge.



7a

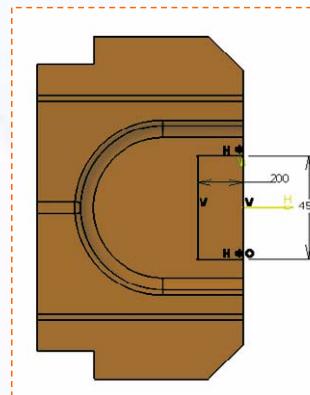
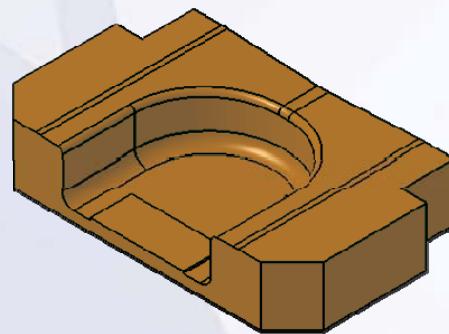


7b



8. Create a pad.

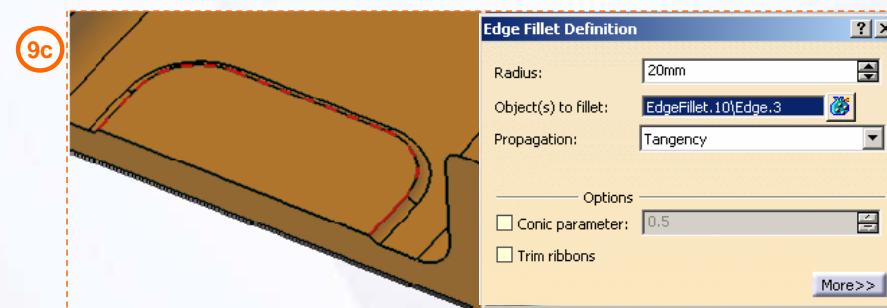
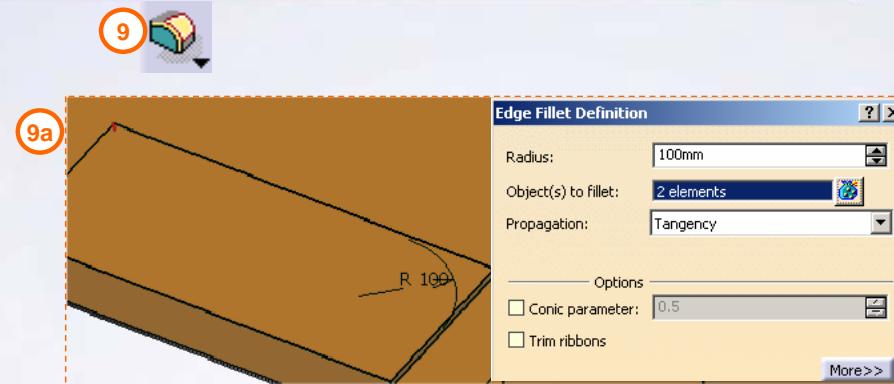
- Create a positioned sketch on a plane 350mm from XY plane and create a pad of length Upto next and offset [10mm].



Do it Yourself (5/9)

9. Create fillets.

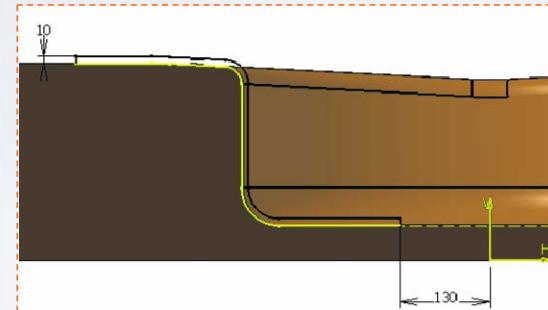
- Create edge fillets as shown.
 - a. Create an edge fillet of [100mm]on two edges.
 - b. Create an edge fillet of [20mm]on one edge. Select edge to keep as shown.
 - c. Create an edge fillet of [20mm]on one edge.



Do it Yourself (6/9)

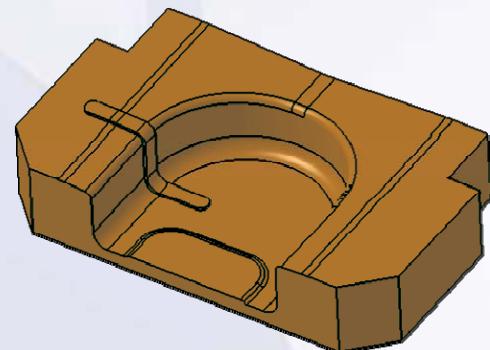
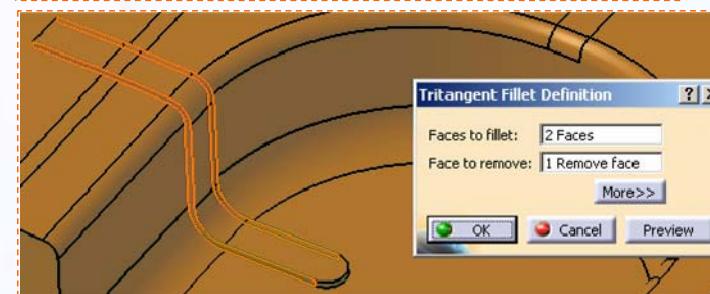
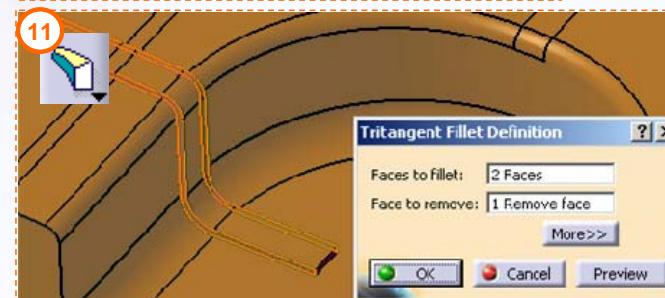
10. Create a pad.

- Create a positioned sketch on a plane 350mm from YZ plane and create a pad of length [30mm] and Mirrored extent.



11. Create Tritangent fillets.

- Create two tritangent fillets as shown.

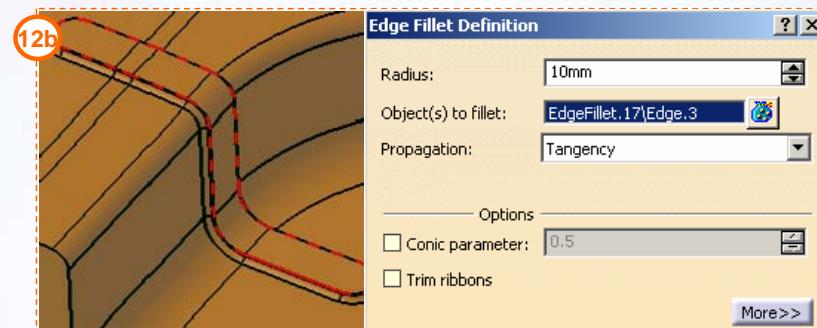
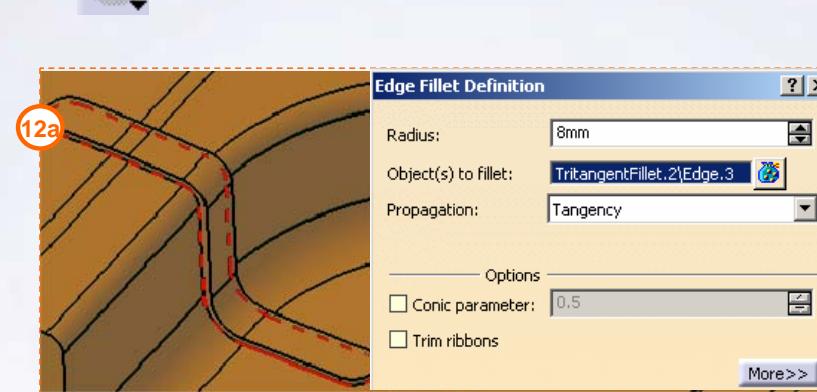


Do it Yourself (7/9)

12. Create fillets.

- Create edge fillets as shown.
 - a. Create an edge fillet of [8mm]on one edge.
 - b. Create an edge fillet of [10mm]on one edge.

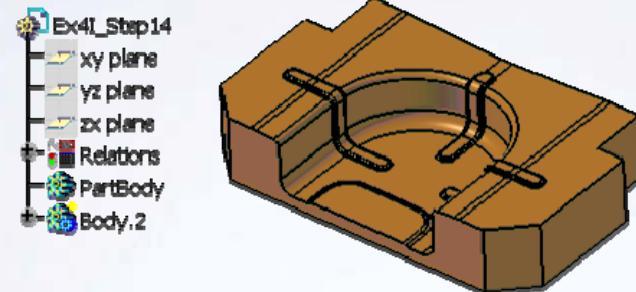
12



Do it Yourself (8/9)

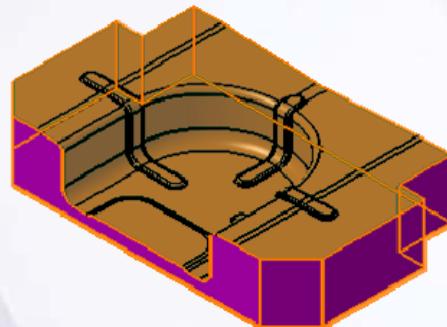
13. Load Ex4I_Step14.CATPart from database.

- Close the existing part without saving it and load Ex4I_Step14.CATPart.



14. Create a shell.

- Create a shell of 1 mm thickness.

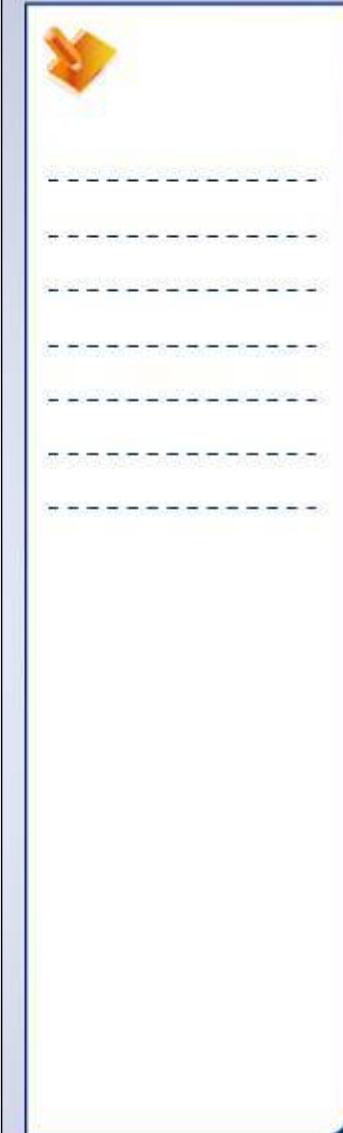
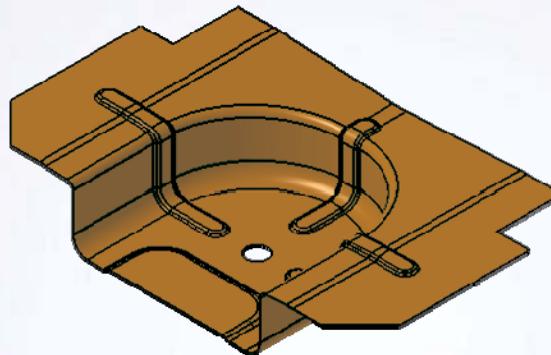


Do it Yourself (9/9)

15. Create a simple hole of diameter 80mm.

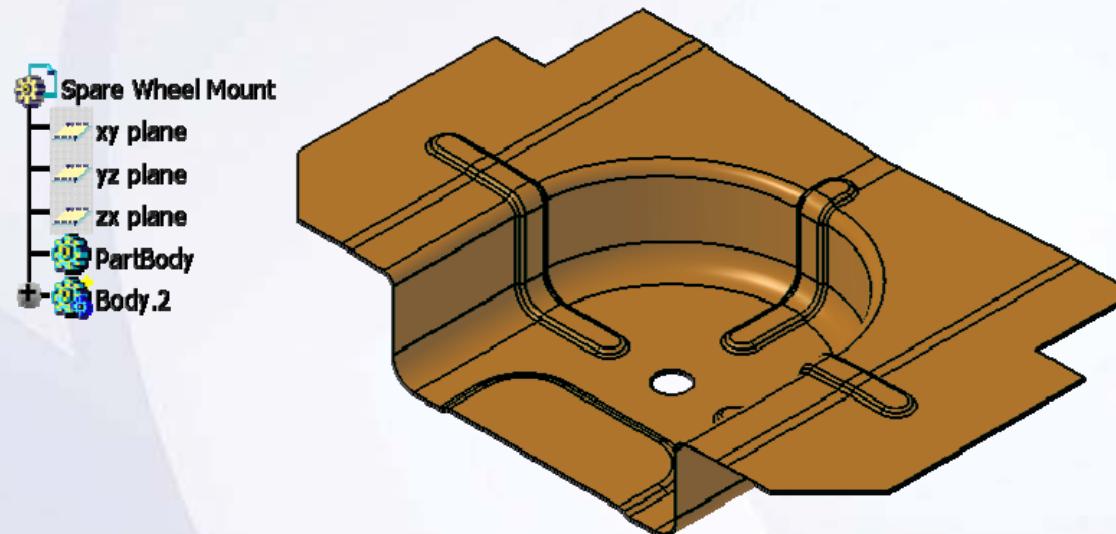


16. Close the part without saving it.



Exercise Recap: Shell and Holes

- ✓ Create a pad
- ✓ Create a pocket
- ✓ Create a chamfer
- ✓ Create a fillet
- ✓ Create a shell
- ✓ Create a hole



Case Study: Additional Features

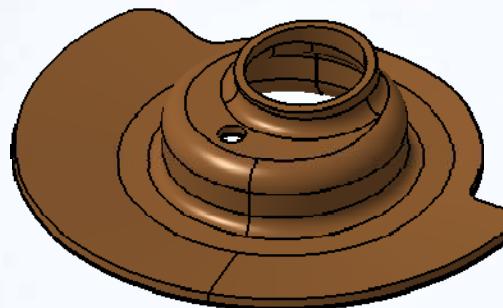
Recap Exercise



20 min

In this exercise you will create the case study model. Let us recall the design intent of this model:

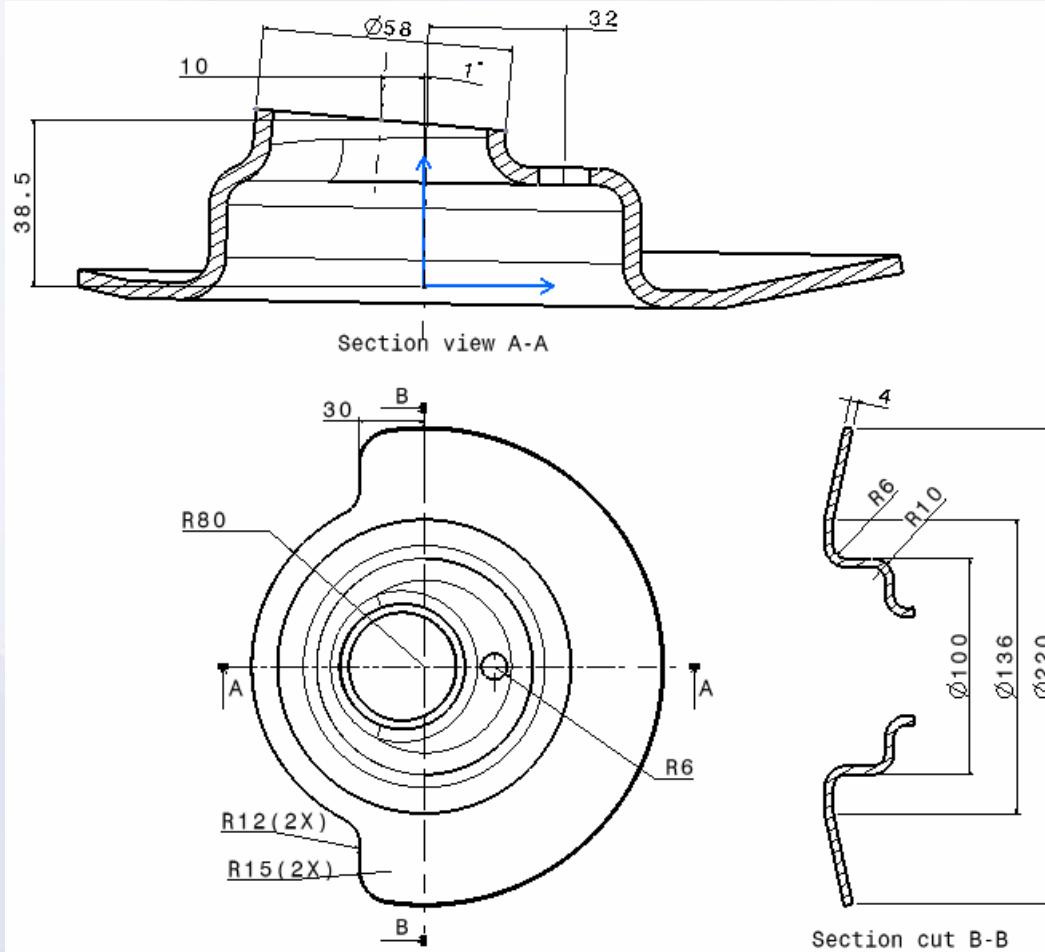
- ✓ The axis of main flange must be at 5 degree from Z axis.
- ✓ The main flange must be at 12 degree from horizontal plane.
- ✓ One large hole of diameter 50mm must be created for Pillar clearance.
- ✓ The thickness of Seat must be 4 mm.
- ✓ There must not be any sharp corners.



Using the techniques you have learned so far, create the model without detailed instructions.

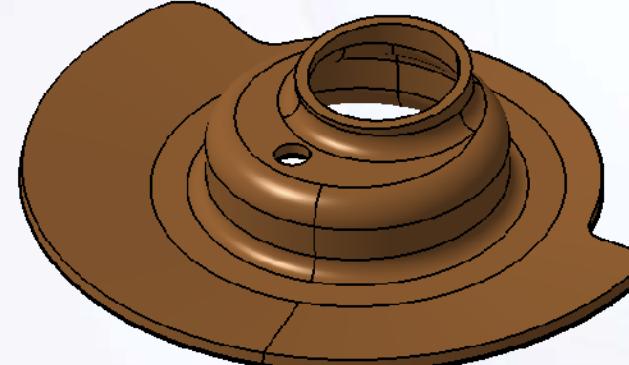
Do It Yourself: Create the Suspension Seat

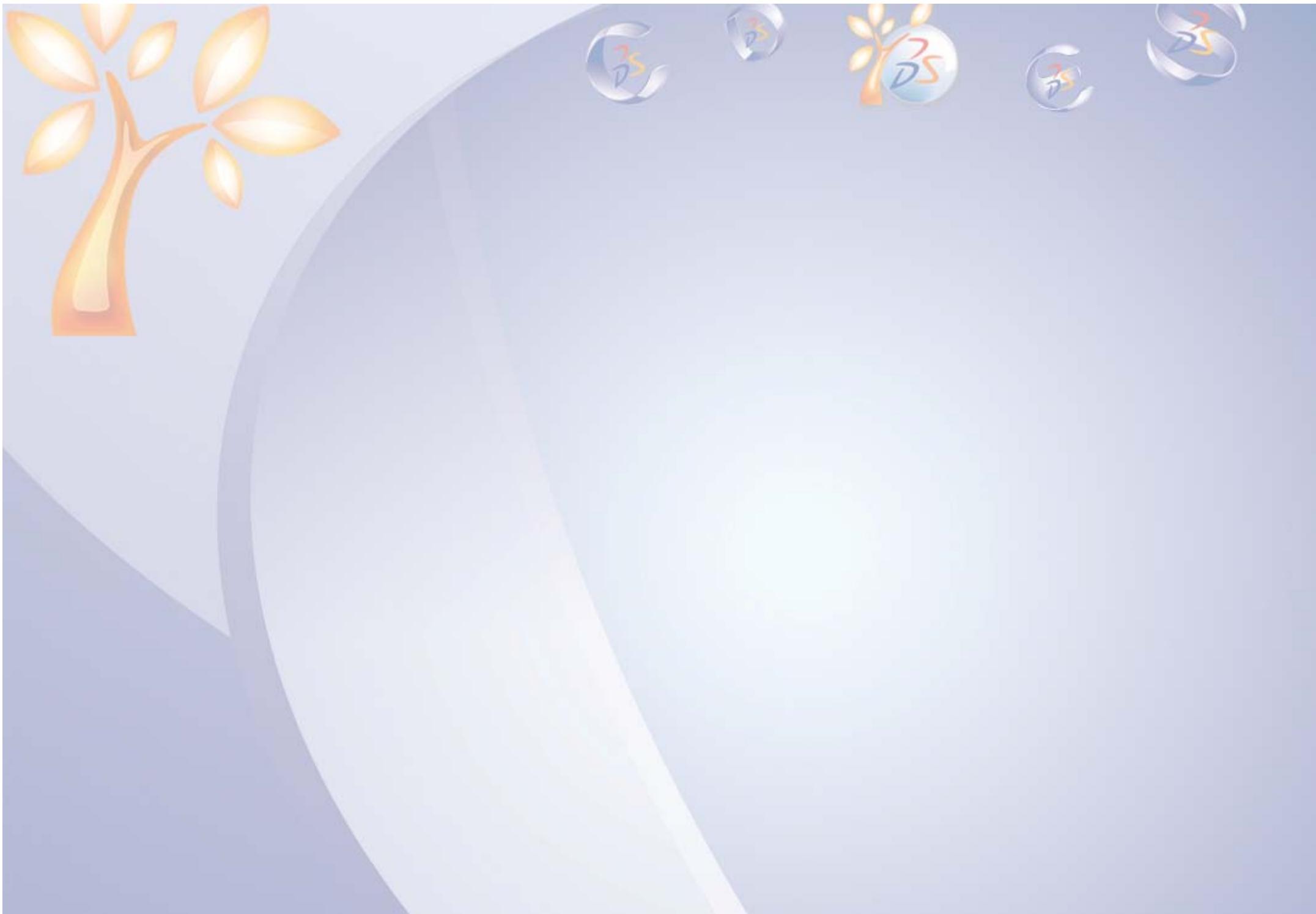
Create the model using the drawing provided here.



Case Study Recap: Suspension Seat

- ✓ Create a reference geometry.
- ✓ Create an axis system.
- ✓ Create a sketched geometry.
- ✓ Create shafts.
- ✓ Create fillets.
- ✓ Create a pocket.
- ✓ Shell the model.
- ✓ Create a hole.





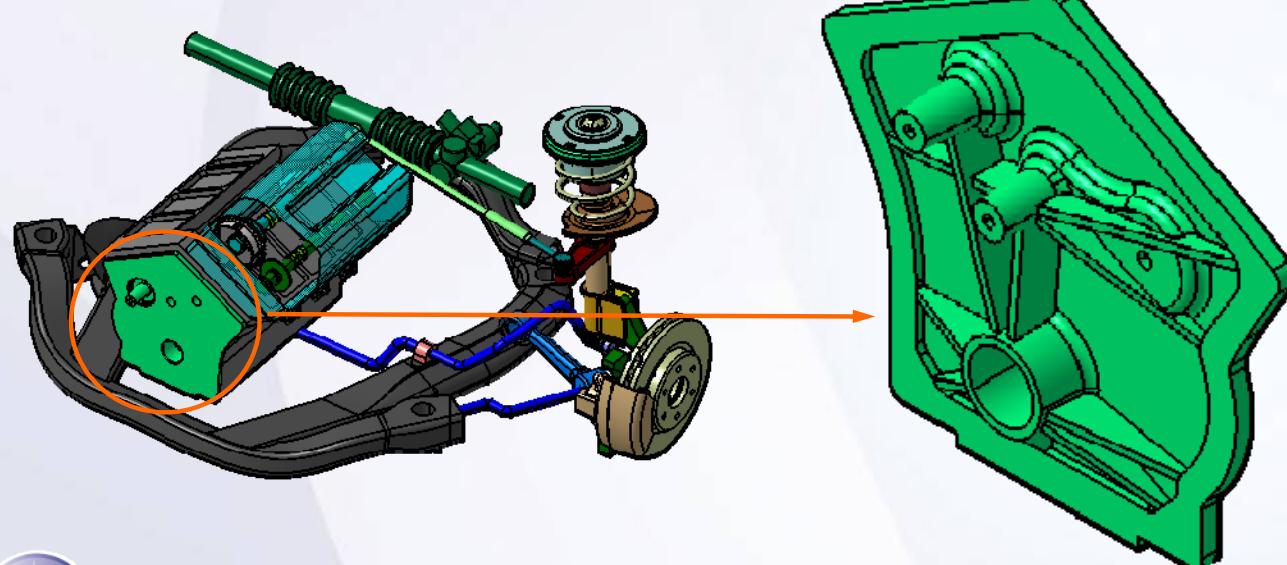
5

Dress-Up Features

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Apply a Draft
- ✓ Create a stiffener
- ✓ Create Threads and Taps
- ✓ Edit Features



4 Hours

Case Study

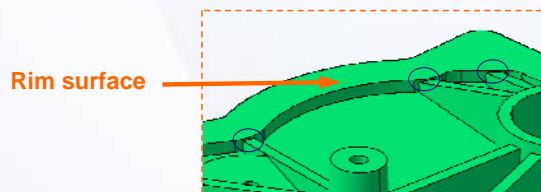
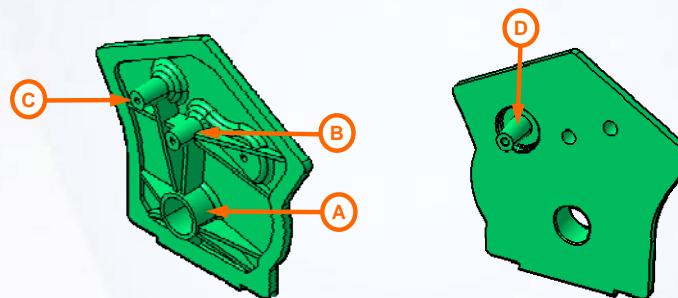
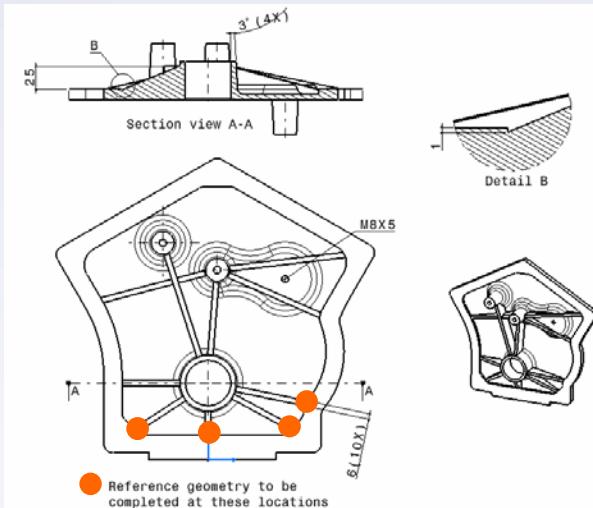
The case study for this lesson is the Timing Chain Cover used in the Front Suspension and Engine assembly.

Design Intent

- ✓ This part is most likely to be manufactured through a casting process, which requires drafts. Bosses of all holes (A, B, C, D) must be drafted.
- ✓ External extremity of ribs must be 1 mm below the rim surface.
- ✓ Rib thickness must be 6 mm.
- ✓ Large boss A must be supported by 5 ribs, boss B must be supported by 4 ribs, and boss C must be supported by 1 rib.
- ✓ Define tap on boss D.

Stages in the Process

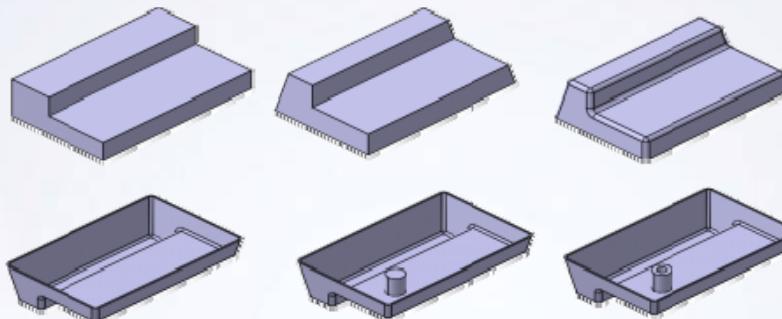
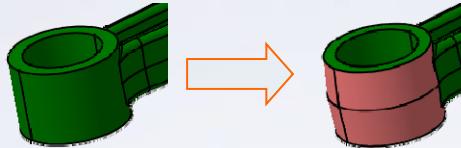
1. Edit feature.
2. Apply a draft.
3. Create reference geometry.
4. Create profile for stiffener.
5. Create threads and taps.



Apply a Draft

Draft features are used to apply an angle to a part surface relative to some reference. Material is added or removed depending on the draft angle and the pull direction applied during the operation.

- ✓ Whenever possible, use the same reference for the parting and neutral elements. Doing so can often avoid unexpected geometry.
- ✓ Whenever possible, create parts in the following general order:
 1. Main part features
 2. Drafts
 3. Fillets
 4. Shells
 5. Minor part features

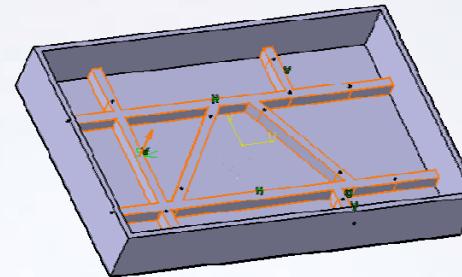
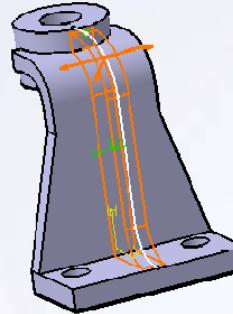


Create a Stiffener

In CATIA, stiffeners are created by extruding and thickening an open-sketched profile.

A. From Side

The sketch is extruded in the profile plane and thickened normal to it.



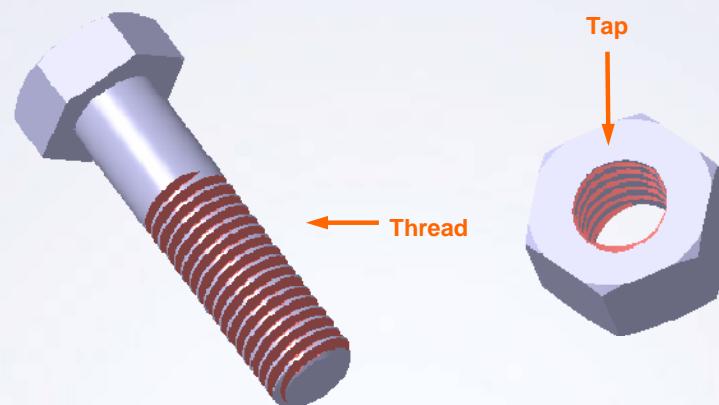
B. From Top

The sketch is extruded normal to the profile plane and thickened in the profile plane.

Create Threads and Taps

A thread is a helical groove outside of a cylindrical shaft, while a tap is a helical groove inside a cylindrical hole.

- ✓ In CATIA, the actual geometry of threads and taps is not displayed. It is represented on the part cosmetically.
- ✓ The features contain parameters that define the intended thread and tap geometry, such as diameter, pitch, and depth.
- ✓ It can also be displayed in a drawing view.



Edit Features

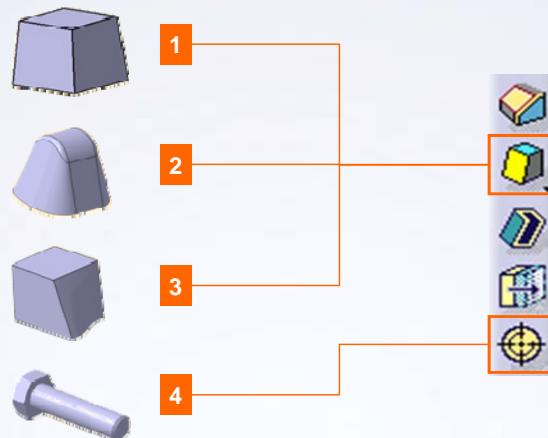
Feature editing and manipulation, beyond dimension changes, is often required as design intent changes or modeling strategies evolve. CATIA has several functionalities that enable you to edit features.

- ✓ Define in work object
- ✓ Reorder features
- ✓ Properties
- ✓ Filters (Search)
- ✓ Parent-child relationships
- ✓ Resolve feature failures

Main Tools

Dress-Up Features

- 1 **Draft Angle:** Creates a basic draft.
- 2 **Draft Reflect Line:** Creates drafts on non-planar surfaces, such as a cylinder.
- 3 **Variable Angle Draft:** Creates a draft that has different angles at transition edges.
- 4 **Thread/Tap:** Applies threads or taps on shafts or holes.



Sketch-Based Features

- 5 **Stiffener:** Creates a stiffener by extruding and thickening an open-sketched profile.



Exercise: Drafts

Recap Exercise

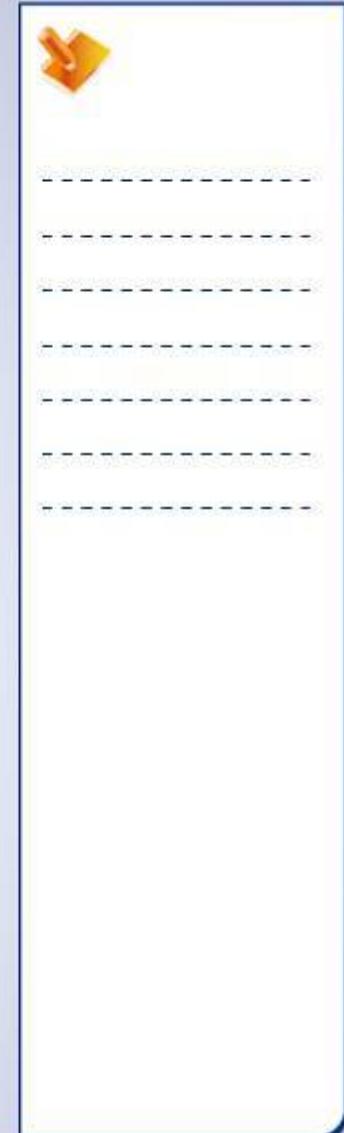
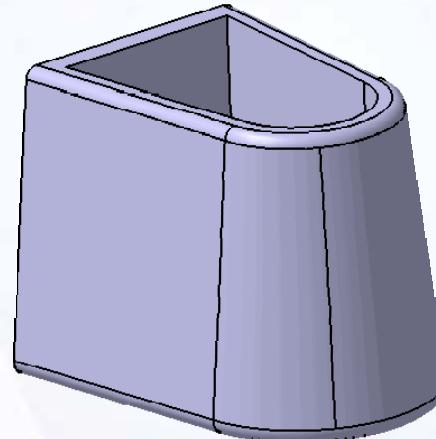


20 min

In this exercise you will practice creating drafts. High-level instructions for this exercise are provided.

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

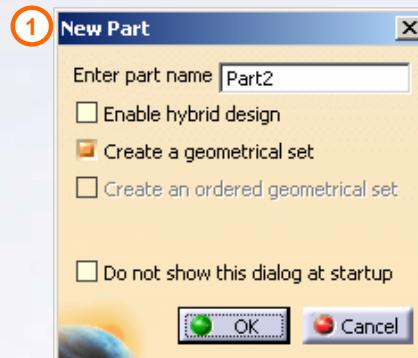
- Create a basic draft
- Create a reflect draft



Do it Yourself (1/5)

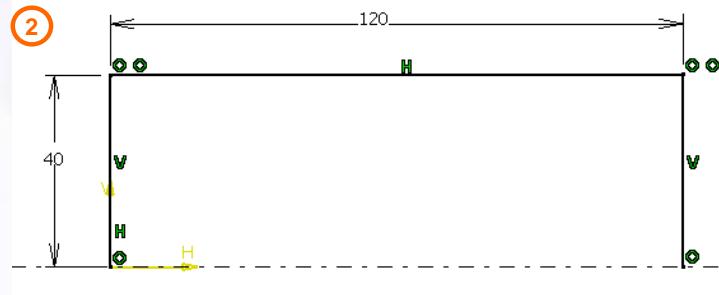
1. Create a new part.

- Create a new part with the geometrical set.



2. Create a shaft.

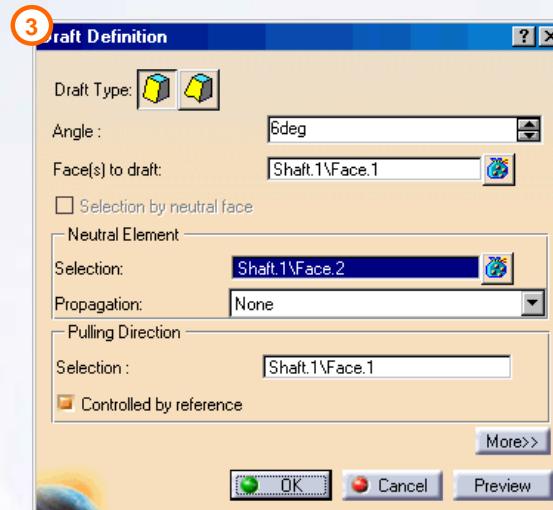
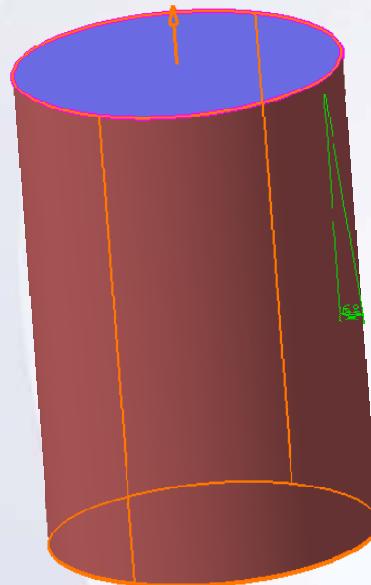
- You will create a sketch of the shown profile and use that to create a pad feature.
 - Create the sketch on the YZ plane.
 - Create a 360° shaft feature.



Do it Yourself (2/5)

3. Create a basic draft.

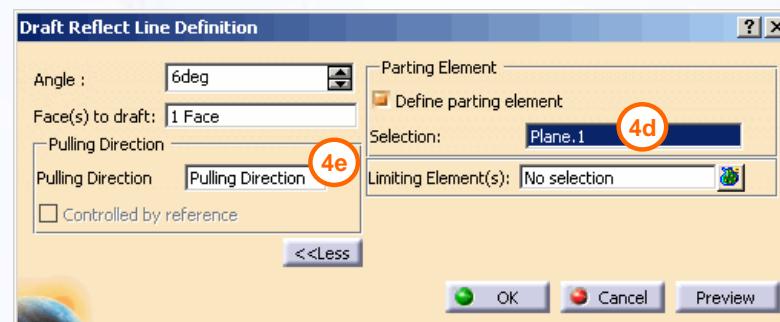
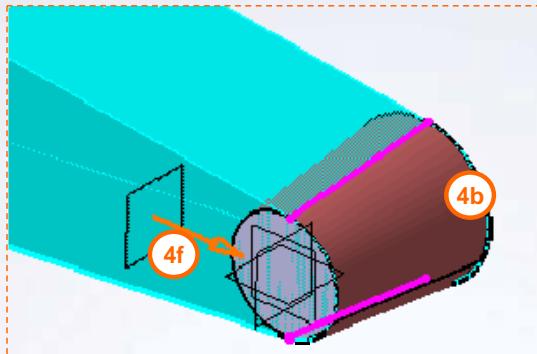
- Select the walls of the cylinder as the faces to draft and the top surface as the neutral and pulling direction.
- Enter a [6deg] draft angle.



Do it Yourself (3/5)

4. Create a Reflect draft.

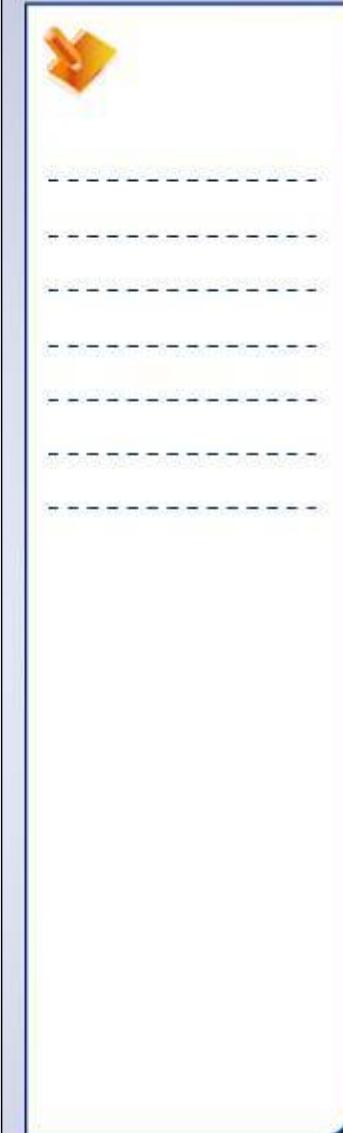
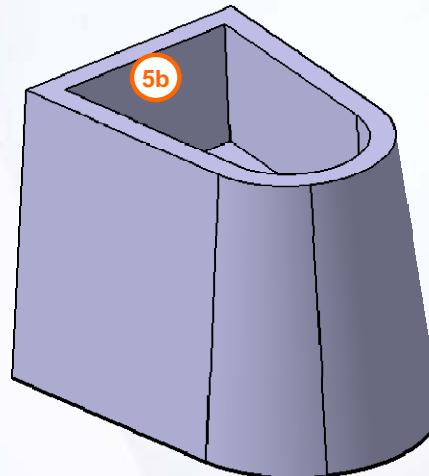
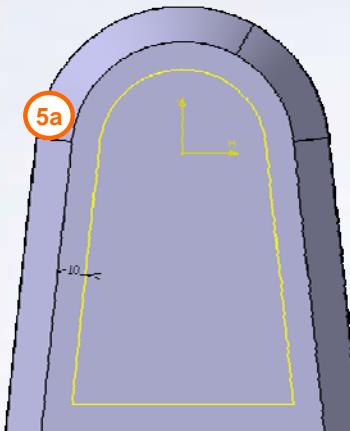
- a. Create an offset datum plane that is [100 mm] from the XY plane in the negative direction.
- b. Select the face of the cylinder to apply the reflect draft.
- c. Click **OK** to the Feature Definition Error.
- d. Define the parting element as the offset plane created earlier.
- e. Define the pulling direction as the offset plane created earlier.
- f. Ensure the pull direction is correct.



Do it Yourself (4/5)

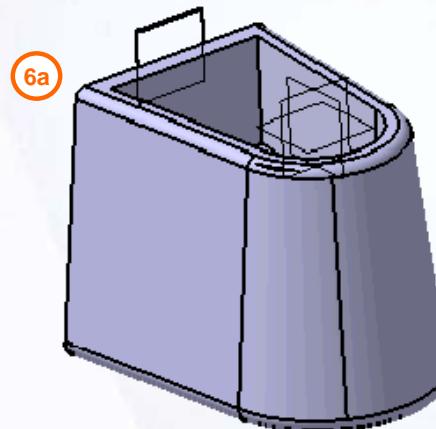
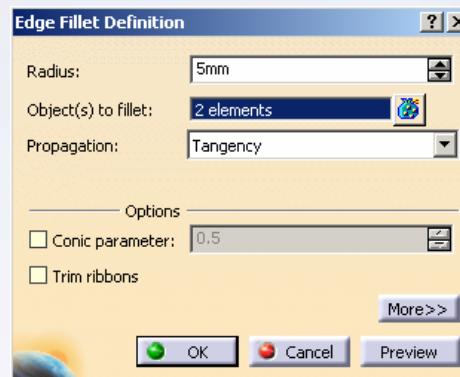
5. Create a pocket.

- a. Select the top surface of the pad and sketch the following profile. Use the existing edge of the pad to create a [10mm] offset.
- b. Create a pocket that is [50mm] deep.



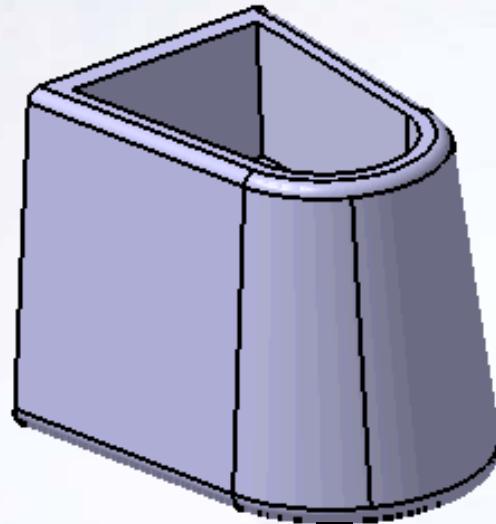
Do it Yourself (5/5)

6. Create an edge fillet.
 - a. Select the edges around the entire top and bottom profiles and specify a [5mm] radius value.
7. Hide all the references plane.
8. Close the file without saving it.



Exercise Recap: Drafts

- ✓ Create a basic draft
- ✓ Create a reflect draft



Exercise: Stiffeners and Draft

Recap Exercise

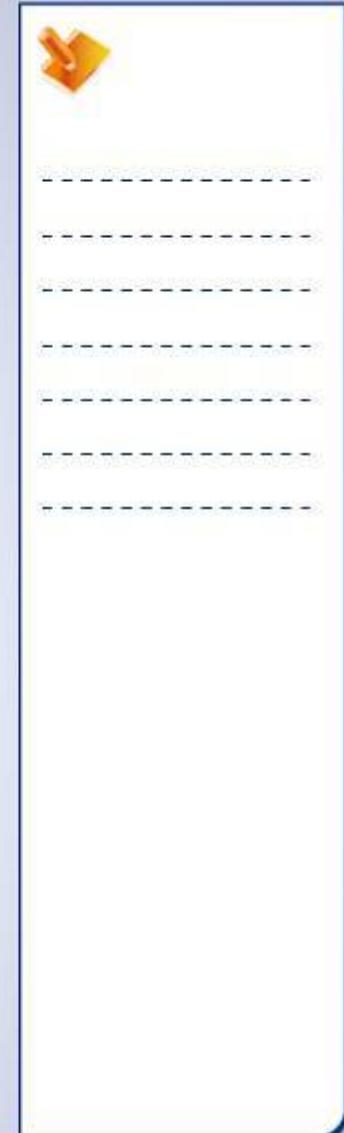
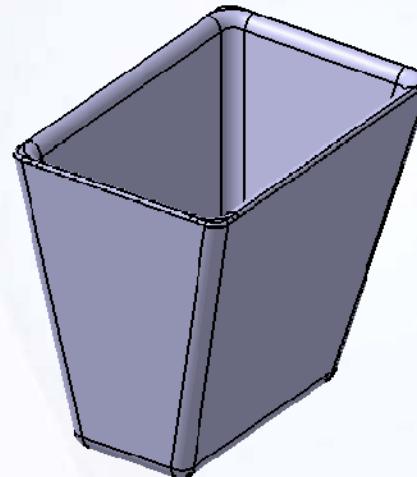


10 min

In this exercise you will use the new skills you have acquired to create a part that contains a draft and four stiffeners. You will use the tools used in the previous exercises to complete this exercise with no detailed instructions.

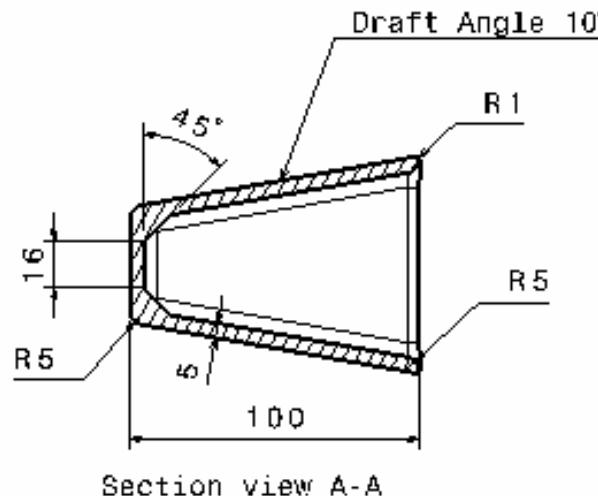
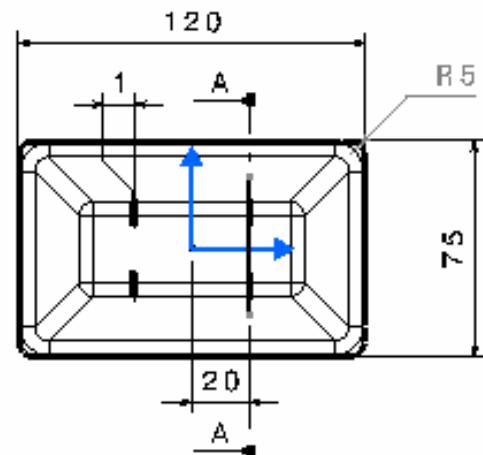
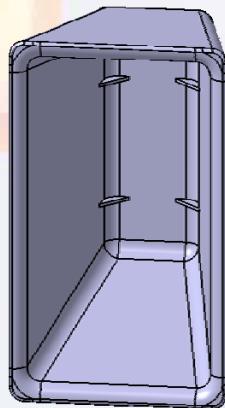
By the end of this exercise you will be able to:

- Create a new part
- Apply draft to a part
- Create stiffeners



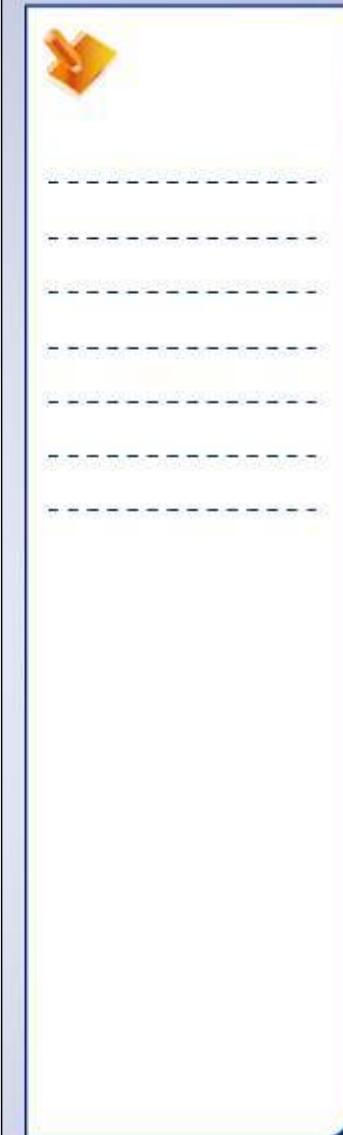
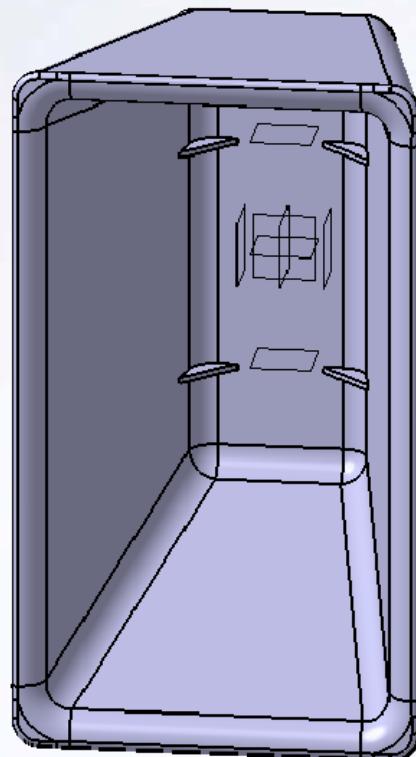
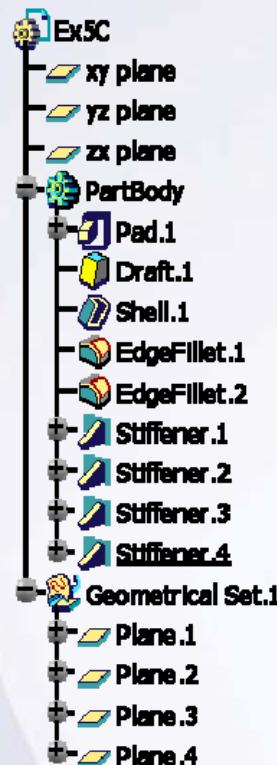
Do it Yourself

1. Create the part shown below.



Exercise Recap: Stiffeners and Draft

- ✓ Create a new part
- ✓ Apply draft to a part
- ✓ Create stiffeners



Exercise: Features Activation

Recap Exercise

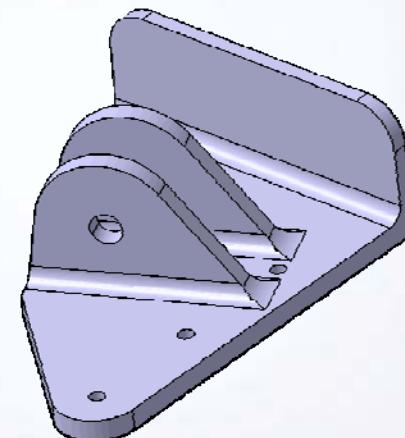
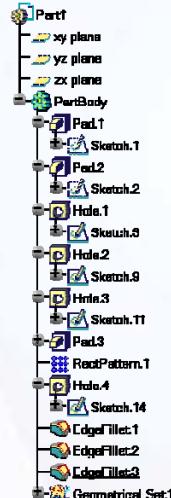


10 min

In this exercise you will open an existing part and investigate how it was modeled. High-level instructions for this exercise are provided.

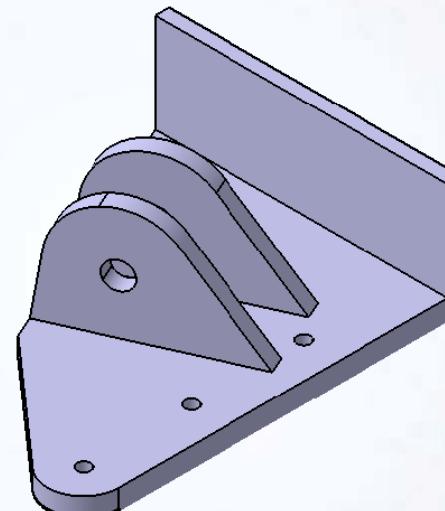
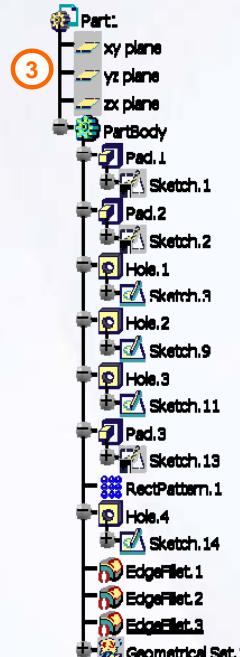
By the end of this exercise you will be able to:

- Review the specification tree
- Hide features
- Activate features



Do it Yourself (1/2)

1. Load Ex3E.CATPart.
2. Review the specification tree.
 - Review the specification tree and note the hidden and deactivated features.
3. Hide the default reference planes.
 - The reference planes are no longer required to simplify the display. Hide them from visible space.

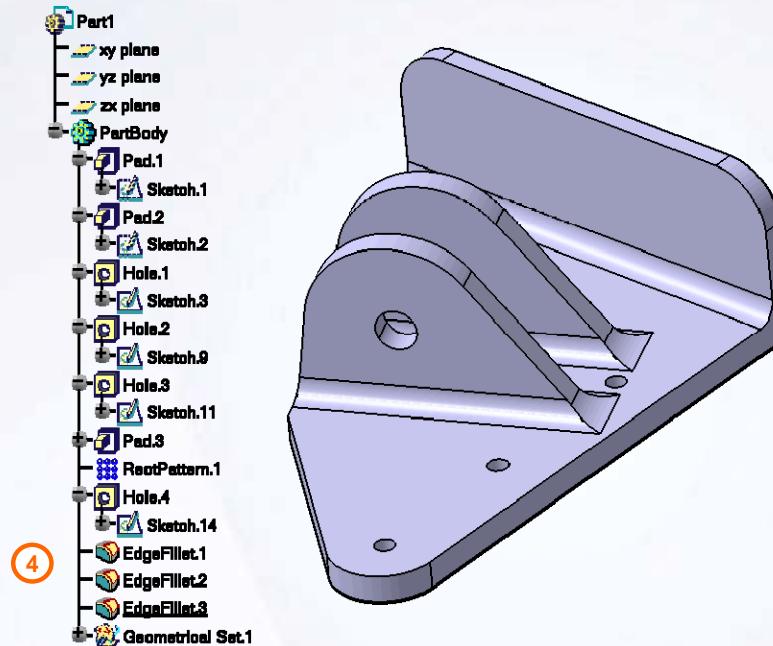


Do it Yourself (2/2)

4. Activate the edge fillets.

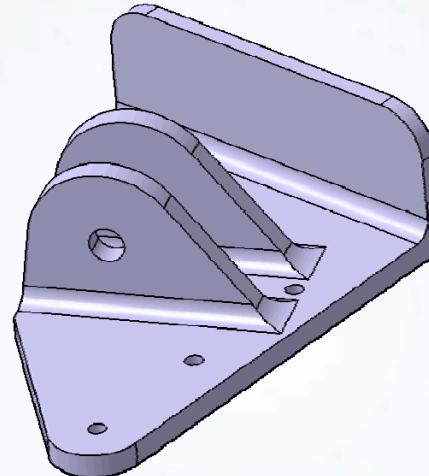
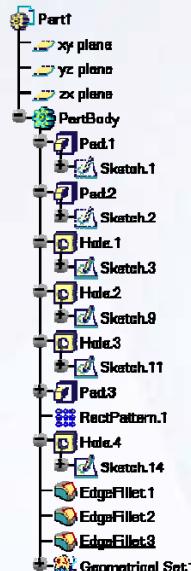
- The last three features in the specification tree have been deactivated.
Activate these features.

5. Close the model.



Exercise Recap: Features Activation

- ✓ Review a specification tree
- ✓ Hide features
- ✓ Activate features



Exercise: Feature Failures

Recap Exercise

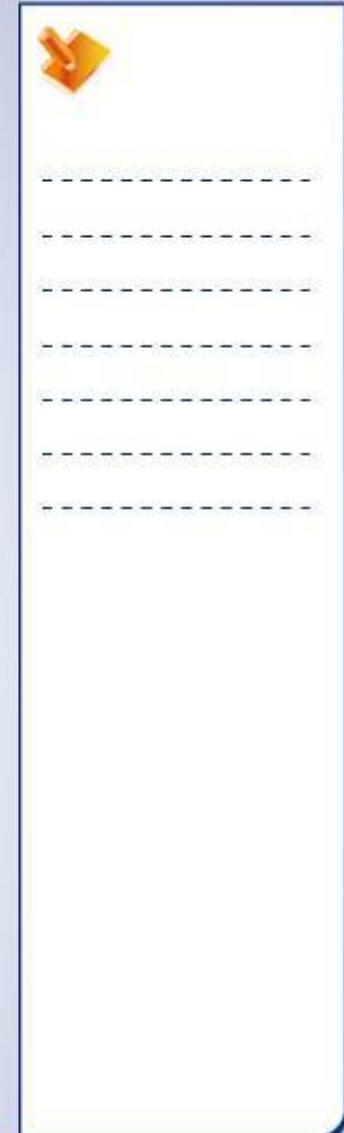
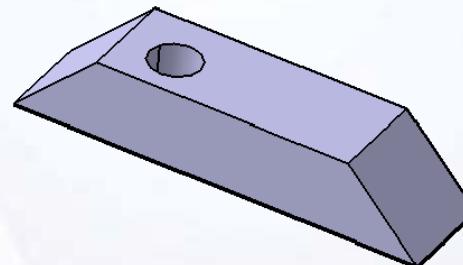
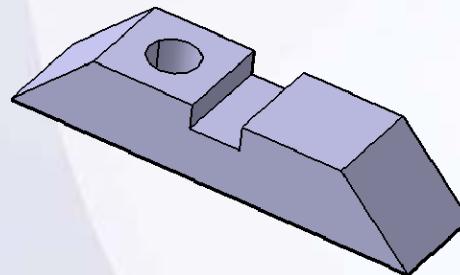


10 min

In this exercise you will open an existing part that contains a pad and hole. You will change the profile of the pad, update it, and resolve any feature failures that may occur. High-level instructions for this exercise are provided.

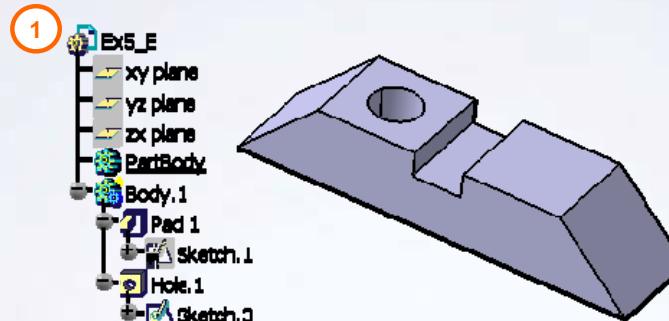
By the end of this exercise you will be able to:

- Troubleshoot a part that contains features that fail.



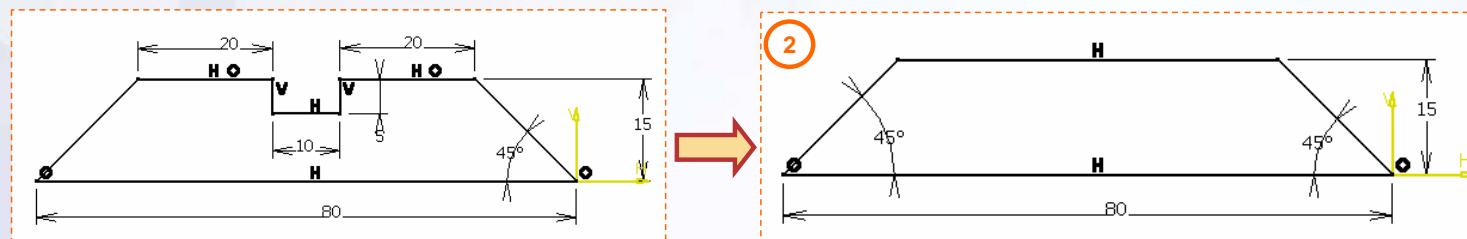
Do it Yourself (1/3)

1. Load Ex5E.CATPart.



2. Change the profile of the pad.

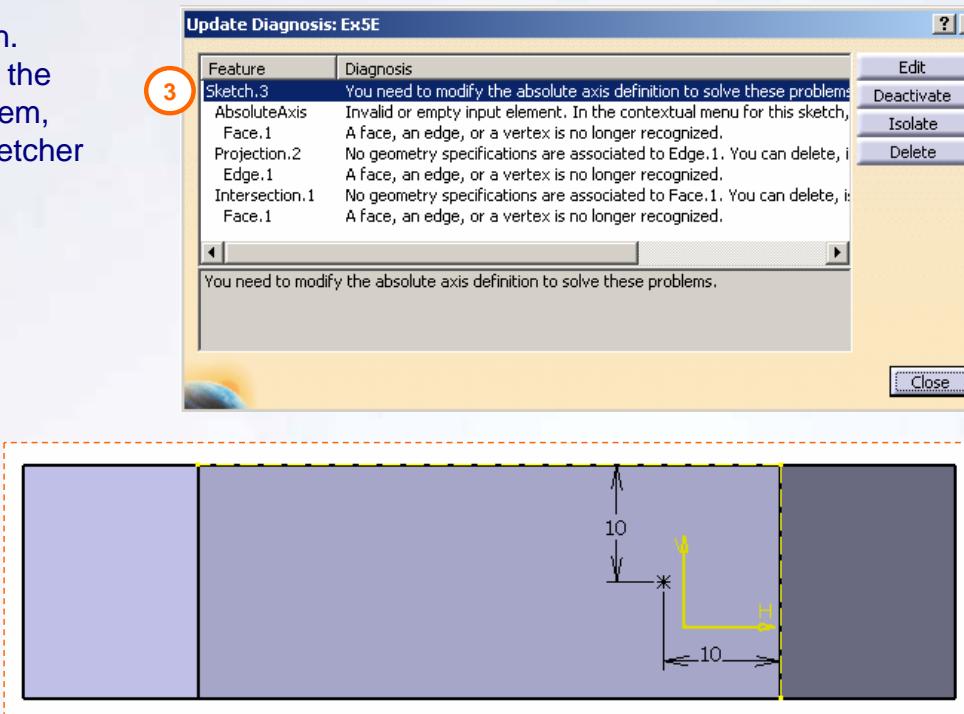
- Edit the sketch.1 of Pad and change it as shown:



Do it Yourself (2/3)

3. Resolve feature failures.

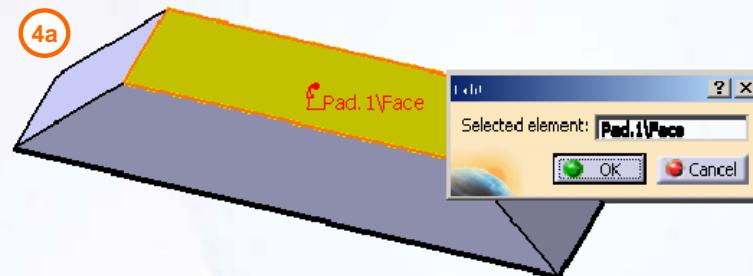
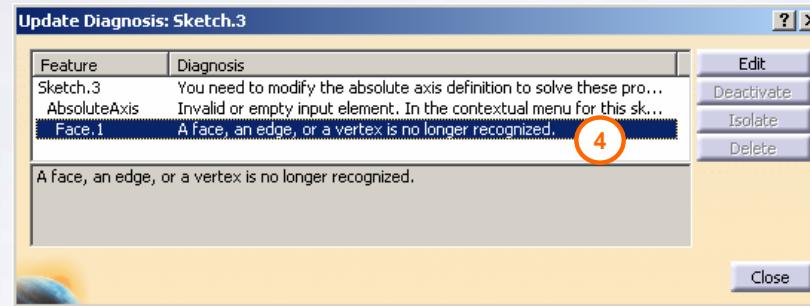
- Once CATIA tries to regenerate Hole.1, sketch.3 fails. CATIA prompts you to edit the sketch. Review the sketch and notice the missing references. Delete them, recreate them and exit the sketcher workbench.



Do it Yourself (3/3)

4. Resolve feature failures (continued).

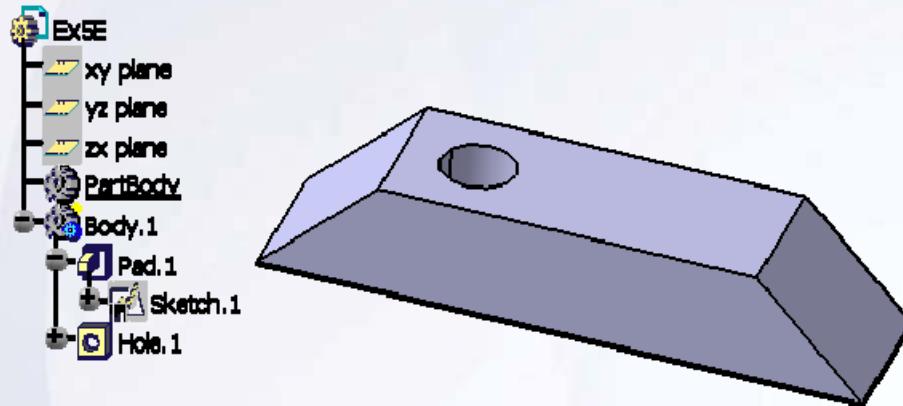
- You still have a failure on Hole.1. Review the feature and notice the support of hole (face.1) is missing.
 - Edit Face.1 and select the correct support.



5. Close the file without saving it.

Exercise Recap: Feature Failures

- ✓ Troubleshoot a part that contains features that fail.



Exercise: Feature Failures

Recap Exercise

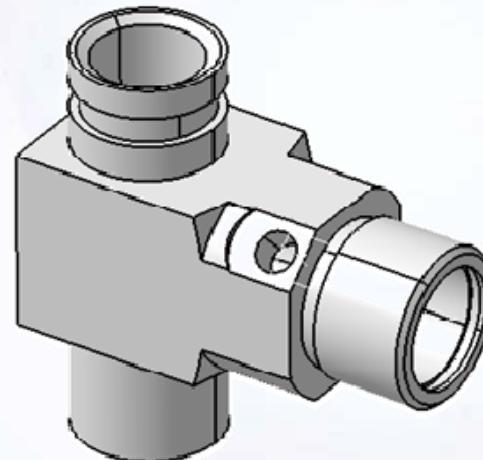


10 min

In this exercise you will open an existing part. You will delete a dress-up feature, verify the impact and correct the update errors. You will use the tools you have learned in this lesson to complete the exercise with no detailed instruction.

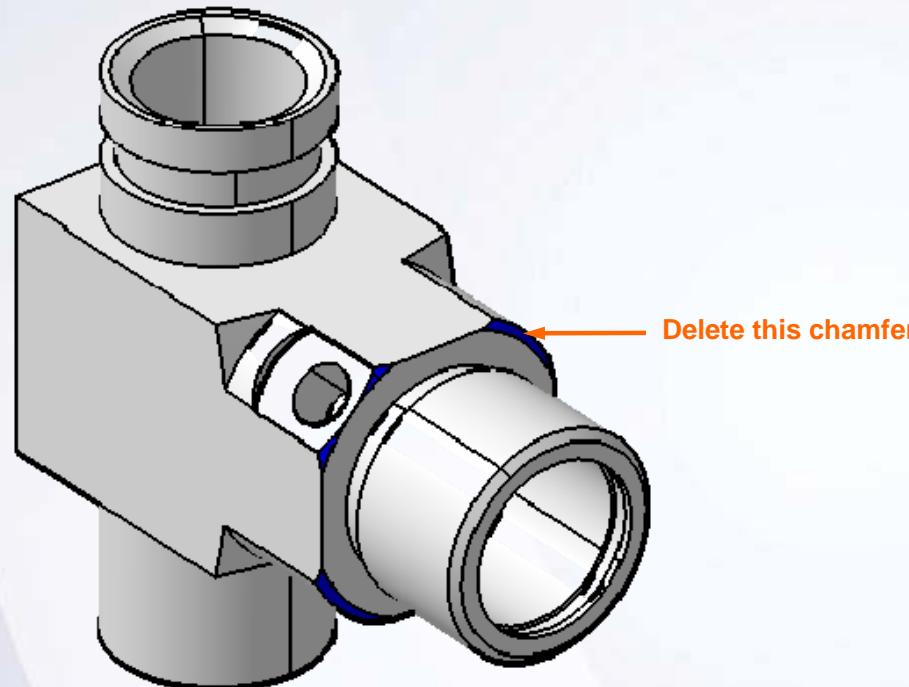
By the end of this exercise you will be able to:

- Use Parents/Children relationship.
- Troubleshoot a part that contains features that fail.



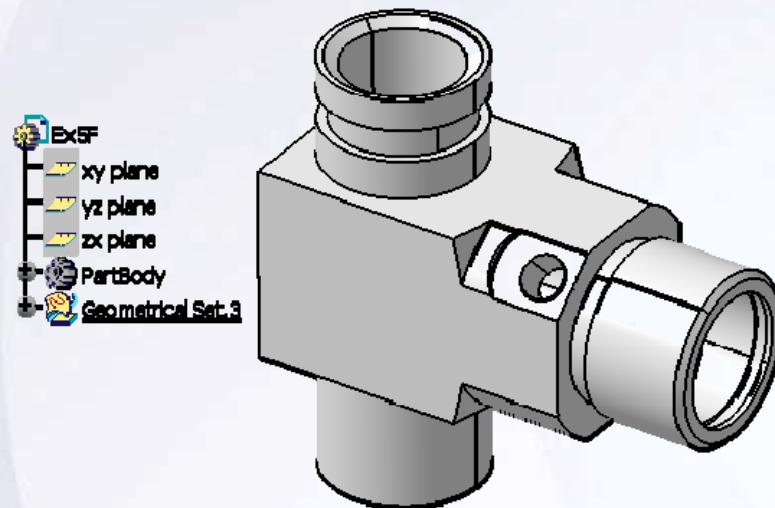
Do it Yourself

1. Load Ex5F.CATPart from database.
2. Study the impact of deletion of Chamfer.6 by Parents/Children.
3. Delete Chamfer.6 and correct the update errors.



Exercise Recap: Feature Failures

- ✓ Use Parents/Children relationship
- ✓ Troubleshoot a part that contains features that fail



Case Study: Dress-up Features

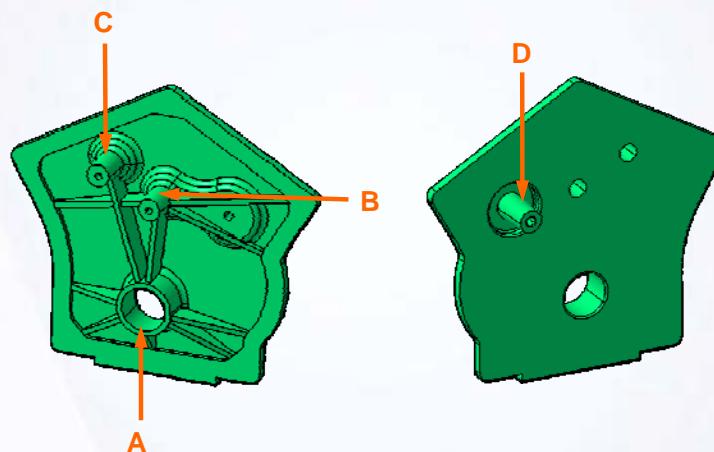
Recap Exercise



25 min

In this exercise you will modify the case study model. Let us recall the design intent of this model:

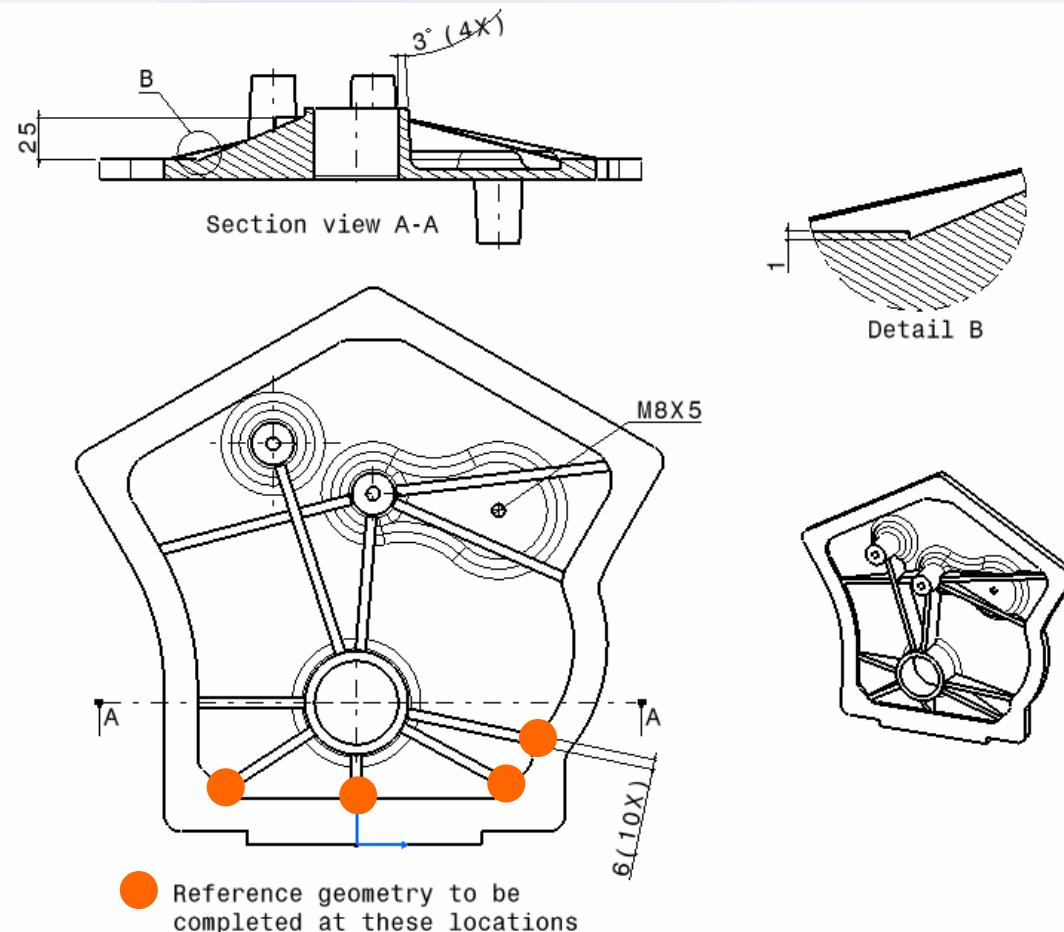
- ✓ Bosses of all holes (A, B, C, D) must be drafted.
- ✓ External extremity of ribs must be 1mm below the rim surface.
- ✓ Rib thickness must be 6mm.
- ✓ Large hole A must be supported by five ribs, hole B must be supported by four ribs, and hole C must be supported by one rib.
- ✓ Define tap on hole D.



Using the techniques you have learned so far, create the model without detailed instructions.

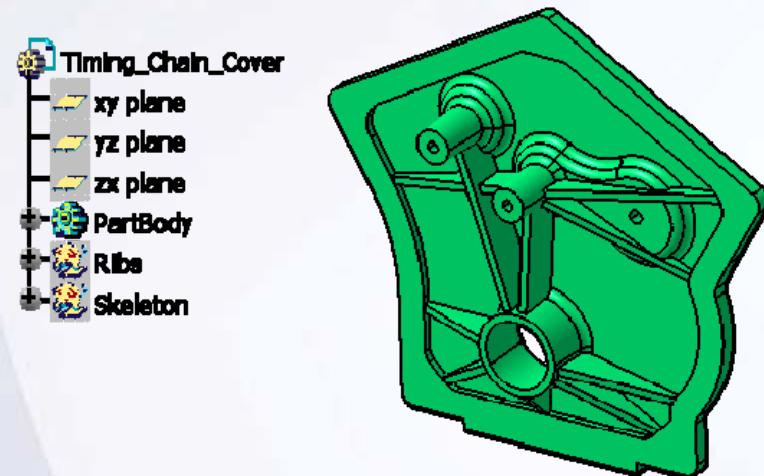
Do It Yourself: Create the Timing Chain Cover

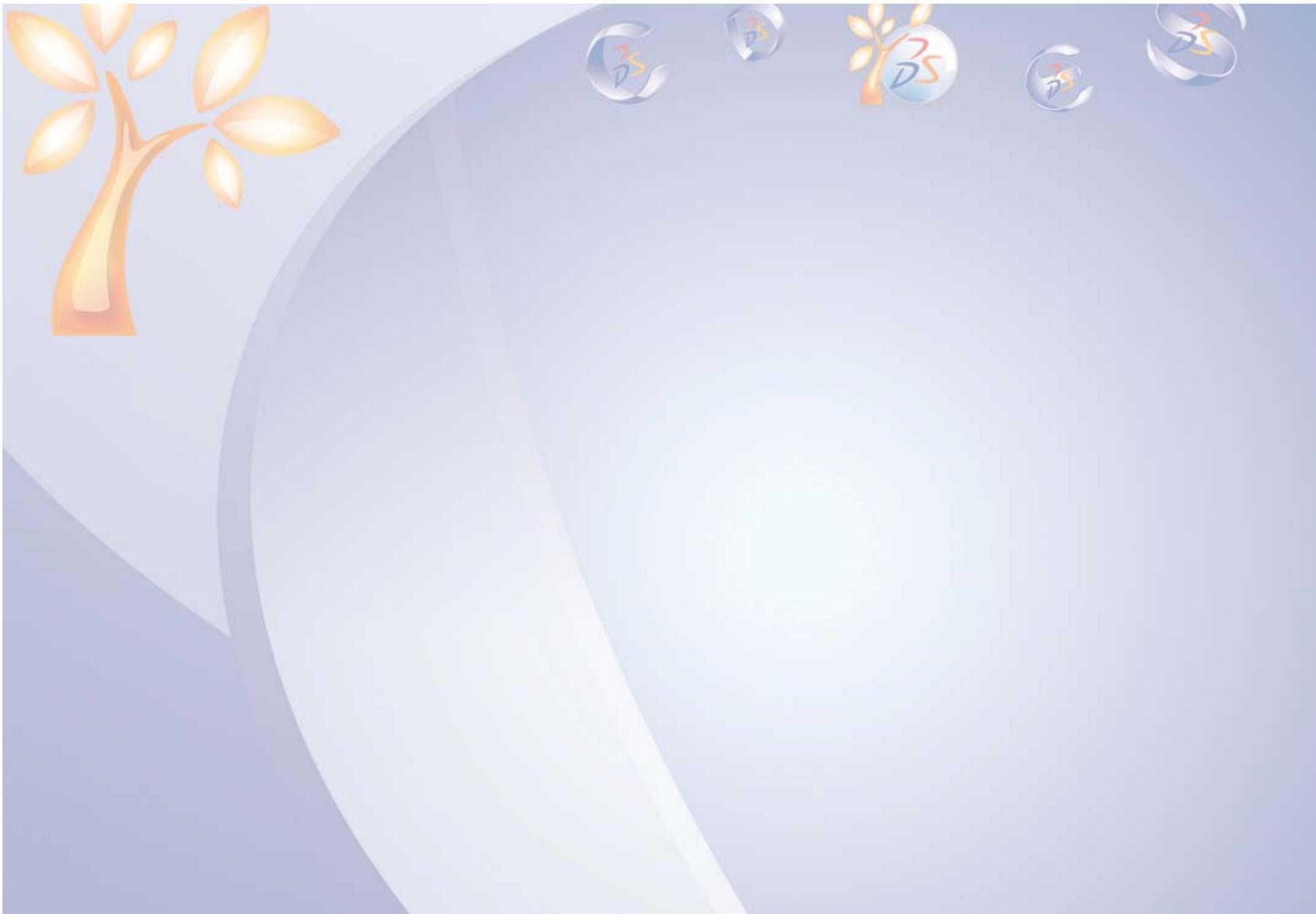
Search and load Exercise5-CaseStudy_Start.CATPart and add the features to the part using the drawing provided here.



Case Study Recap: Timing Chain Cover

- ✓ Create reference geometry
- ✓ Create sketch profiles
- ✓ Create draft features
- ✓ Create stiffener features
- ✓ Create an edge fillet
- ✓ Create a tap feature



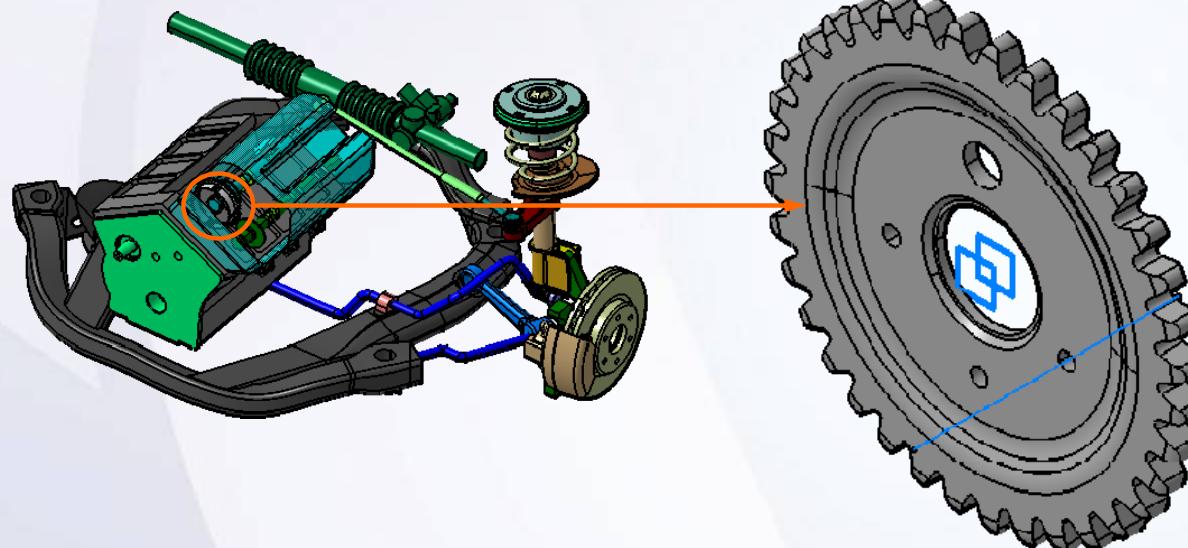


Reusing Data

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Duplicate Features
- ✓ Copy and Paste Data
- ✓ Create the Published Elements



4 Hours

Case Study

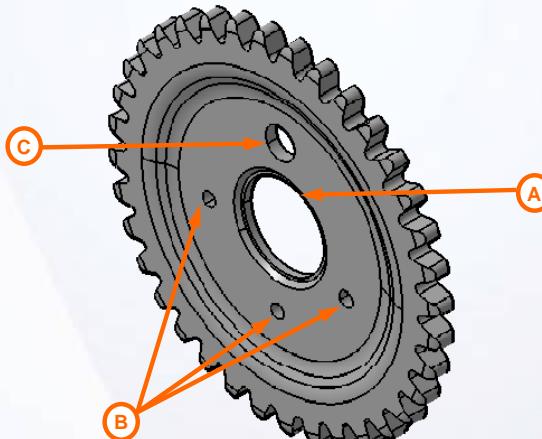
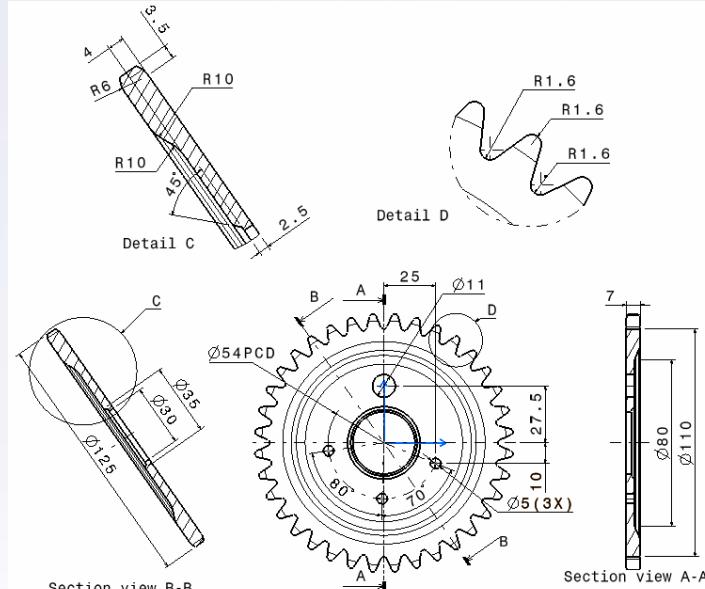
The case study for this lesson is the Sprocket used in the Front Suspension and Engine assembly.

Design Intent

- ✓ The outer diameter must be 125 mm.
- ✓ The inner diameter must be 110 mm.
- ✓ The number of teeth of sprocket must be 36.
- ✓ The mounting hole (A) must have a diameter of 30 mm.
- ✓ The three mounting holes (B) must have a diameter of 5 mm and be spaced at pre-defined angles around the central axis.
- ✓ The mounting hole (C) must have a diameter of 11 mm.
- ✓ Publish axis of hole A, hole B and rear face of the sprocket.

Stages in the Process

1. Create shaft.
2. Create pocket.
3. Create circular pattern.
4. Create groove.
5. Create fillet.
6. Create hole.
7. Publish geometry.



Duplicate Features

In order to avoid creation of each feature individually, duplication tools are used. Two types of duplication features:

A. **Mirror:** While designing parts, it is better to identify areas of symmetry before you start making the model. This enables you to plan and reduce the amount of work needed by only building half of the part, then using the Mirror tool to build the other side.

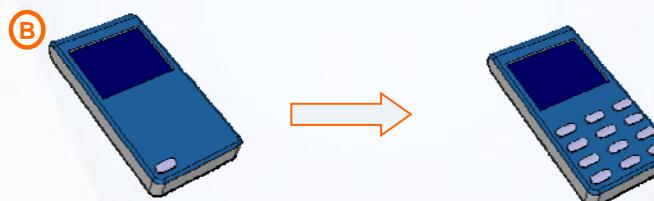
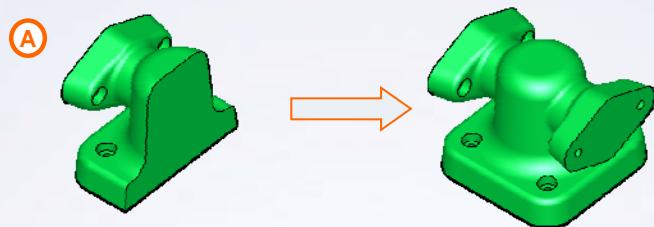
B. **Pattern:** Using Patterns you can create several identical features from an existing one and simultaneously position them on a part.

Three different types of patterns are:

i. **Rectangular patterns** are linear and can be created in two directions.

ii. **Circular patterns** are radial and defined about an axis.

iii. **User patterns** use an existing sketch of points to define the location of the instances.

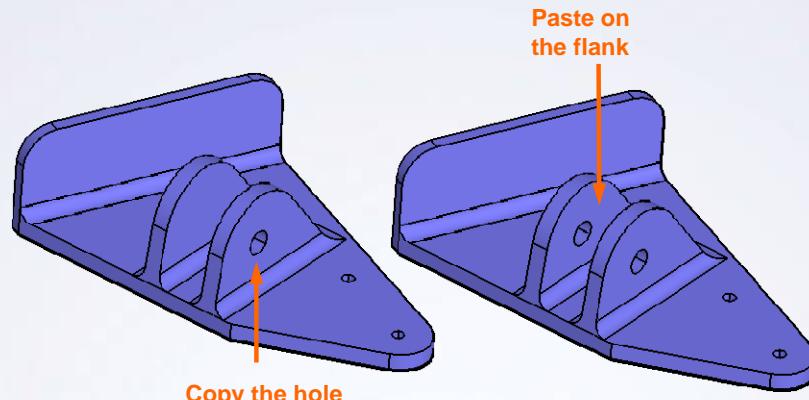


Copy and Paste the Data

Features can be duplicated by copying and pasting them within a part. The pasted feature is identical and completely independent of the original feature.

To copy and paste, you can use any one of the following techniques:

- ✓ Click Copy then Paste in the Standard toolbar
- ✓ Select Edit > Copy then Edit > Paste
- ✓ Press Ctrl+C and then Ctrl+V
- ✓ Right-click then select Copy and Paste, or
- ✓ In the geometry area or the specification tree, press and hold down the Ctrl key and drag the selection and drop in the geometry area or the specification tree.

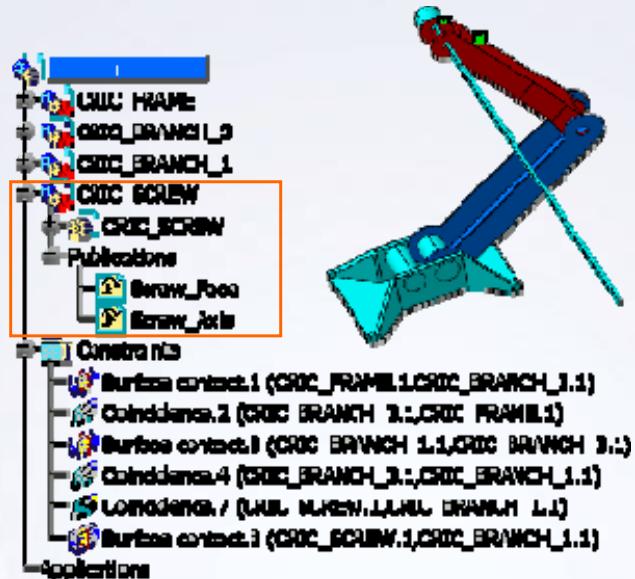


Create the Published Elements

Publishing geometrical elements is the process of making geometrical features available to different users. Publishing geometry and parameters in a skeleton model is suggested to help control the external references created.

The benefits of using publishing geometry are as given below:

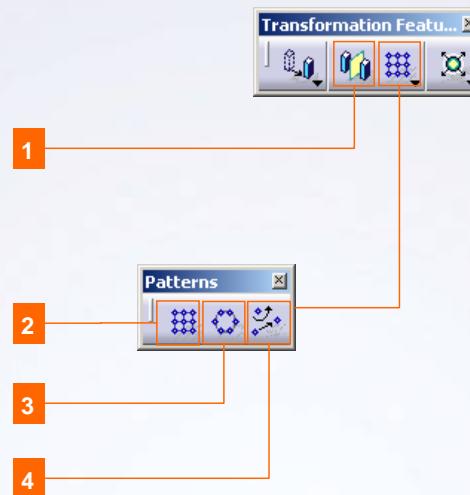
- ✓ Label geometry to give it a name that can be easily recognized (particularly in the case of publishing edges, faces, etc.).
- ✓ To make particular geometry easier to access from the specification tree.
- ✓ Control external references. An option is available that lets you only select as external reference only the published elements.
- ✓ Easy replacement of one feature of the part with another.



Main Tools

Transformation Features

- 1 **Mirror:** duplicates one side of a symmetrical part about a given reference plane.
- 2 **Rectangular Pattern:** creates a linear pattern in two directions.
- 3 **Circular Pattern:** creates a pattern in a circular manner about specified axis.
- 4 **Insert in New Body:** projects the existing 3D elements onto the sketch plane.



Exercise: Patterns

Recap Exercise

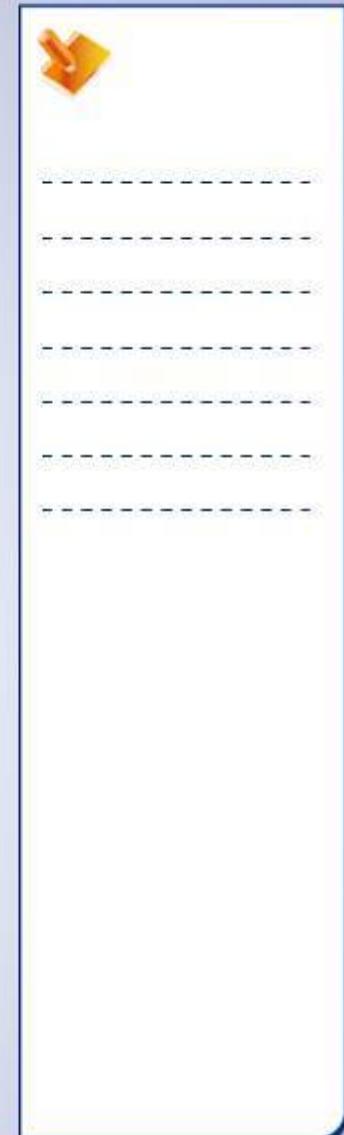
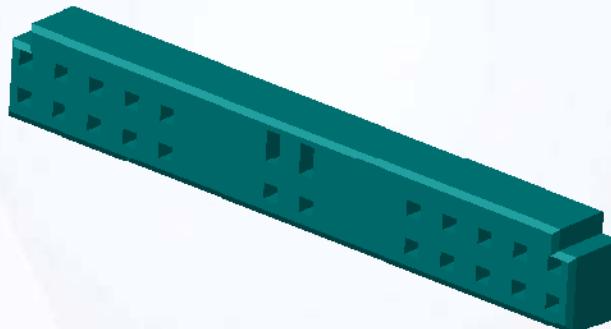


15 min

In this exercise you will practice creating and manipulating patterns. High-level instructions for this exercise are provided.

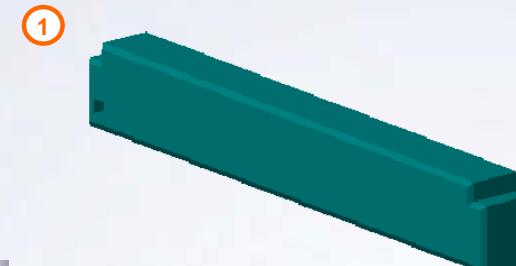
By the end of this exercise you will be able to:

- Create a rectangular pattern
- Remove instances from a pattern
- Explode a pattern
- Modify an instance of the pattern

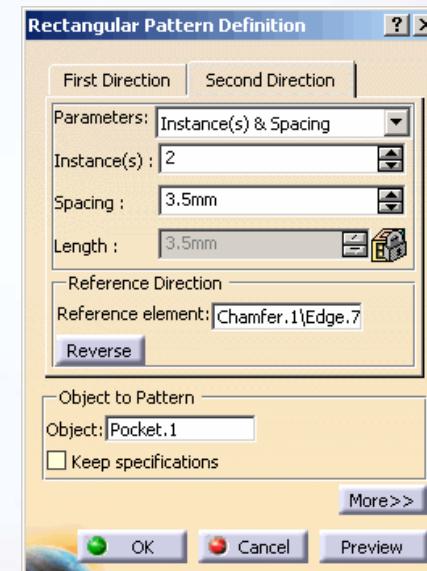
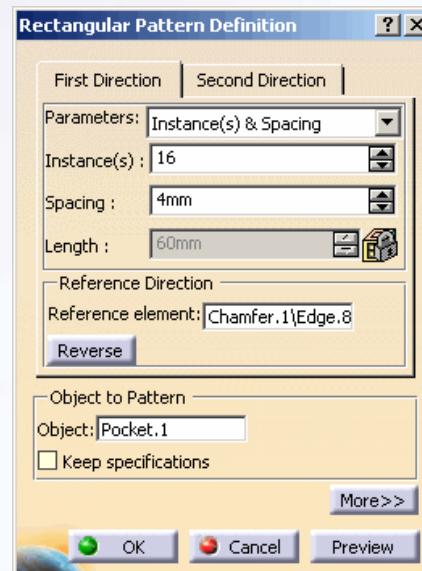
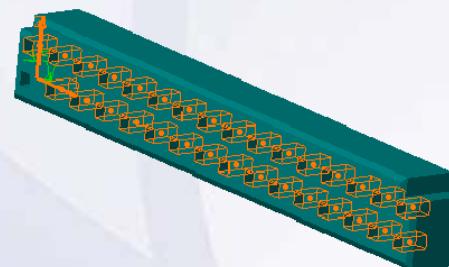


Do it Yourself (1/3)

1. Load Ex6B.CATPart from database.

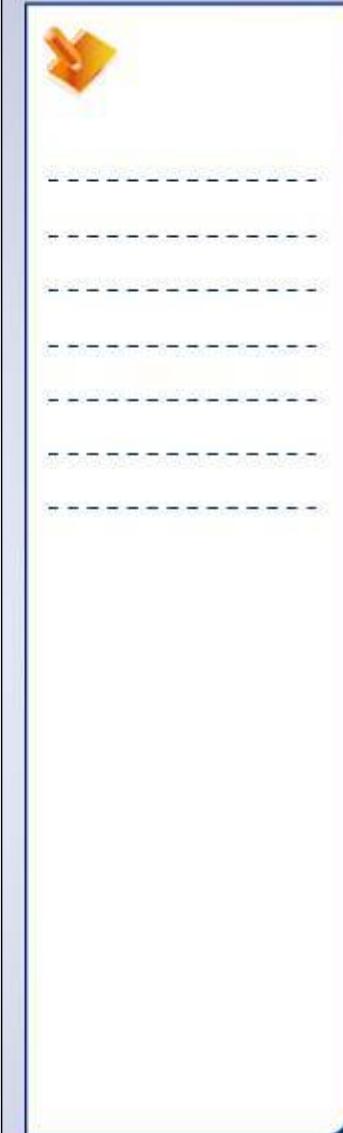
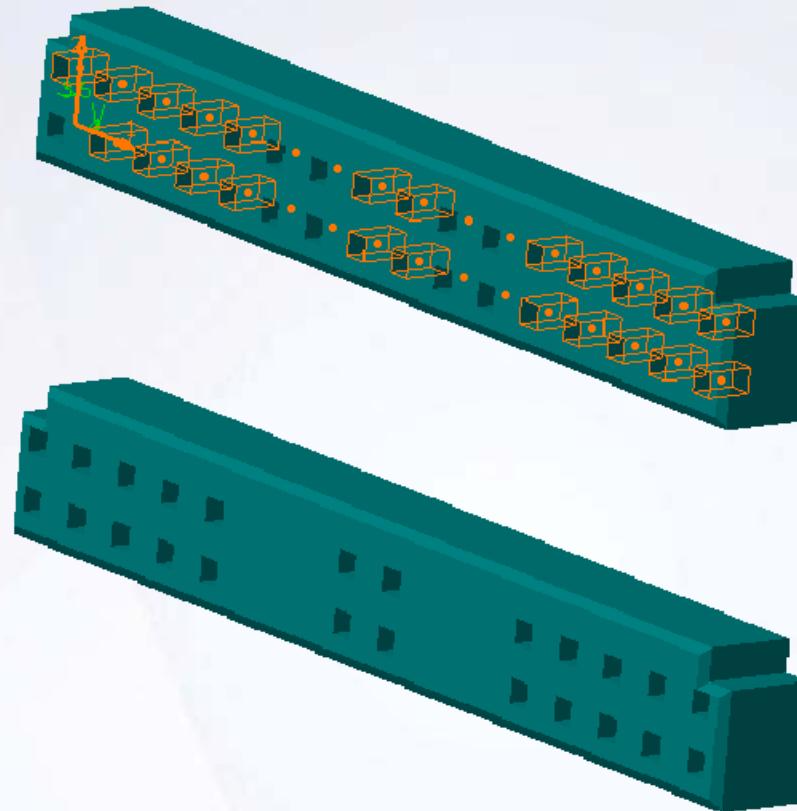


2. Create a rectangular pattern of Pocket.1.



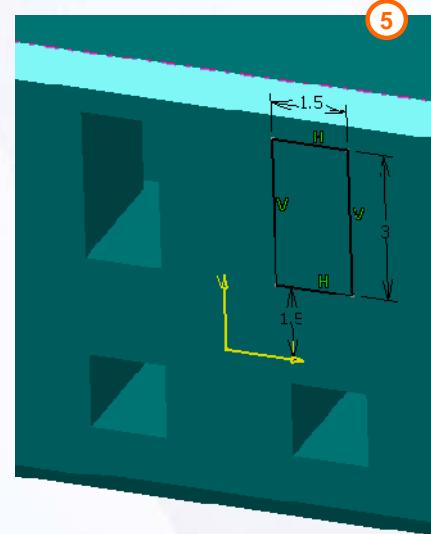
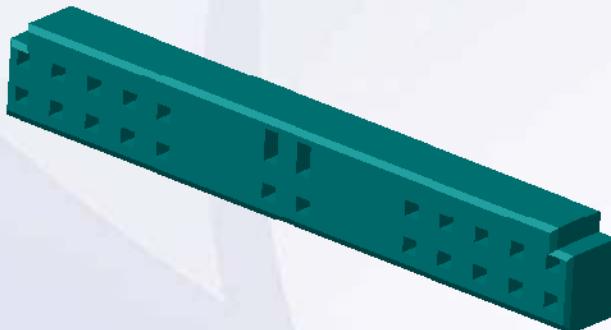
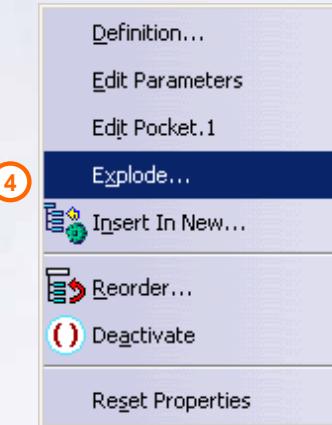
Do it Yourself (2/3)

3. Remove the following instances from the pattern.



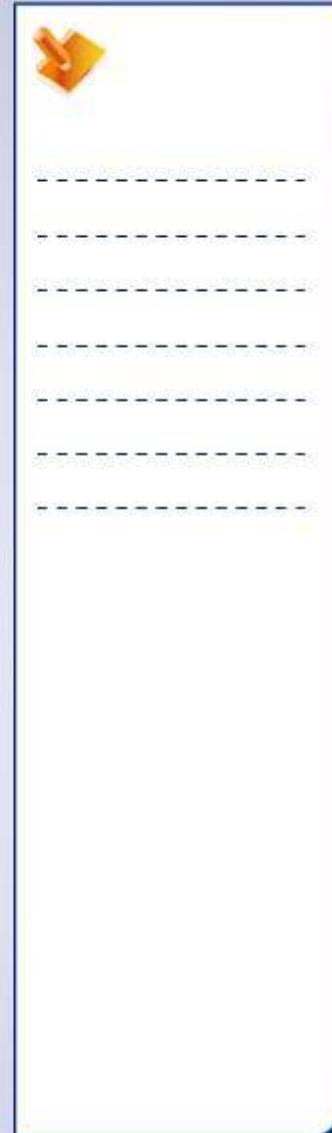
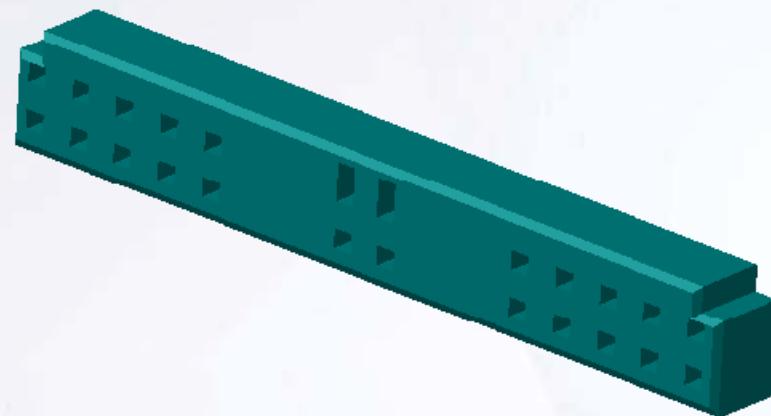
Do it Yourself (3/3)

4. Explode the pattern.
5. Modify the two pockets as per the following sketch.
6. Close the file without saving it.



Exercise Recap: Patterns

- ✓ Create a rectangular pattern
- ✓ Remove instances from a pattern
- ✓ Explode a pattern
- ✓ Modify an instance of the pattern



Exercise: Patterns

Recap Exercise

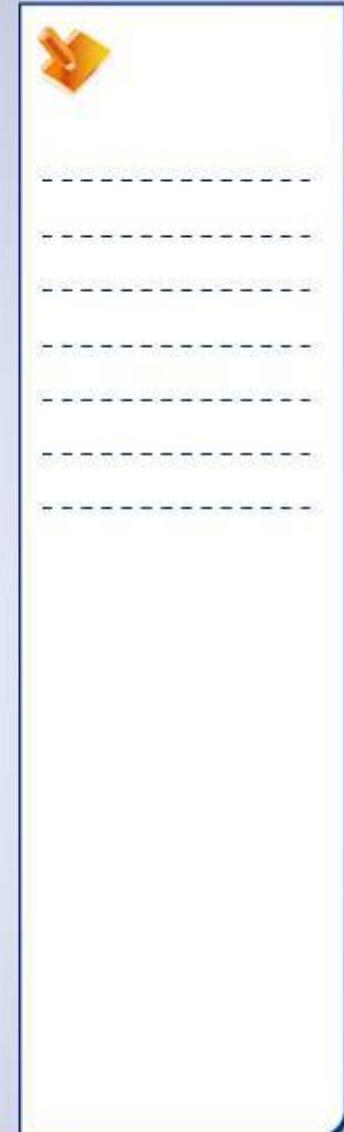
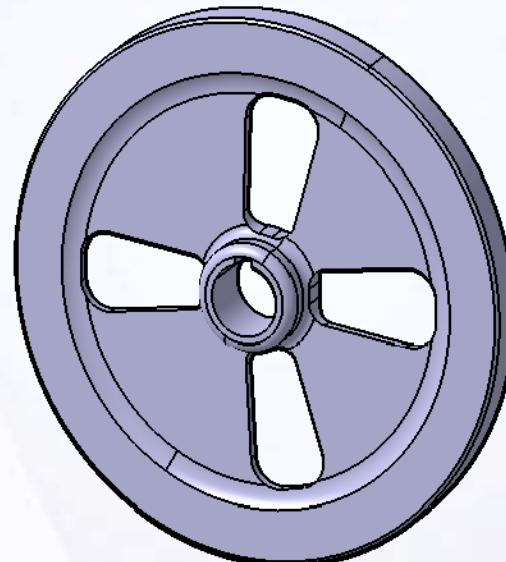


25 min

In this exercise you will use the newly acquired skills to create a part containing a circular pattern. You will use the tools used in the previous exercises to complete this exercise with no detailed instructions.

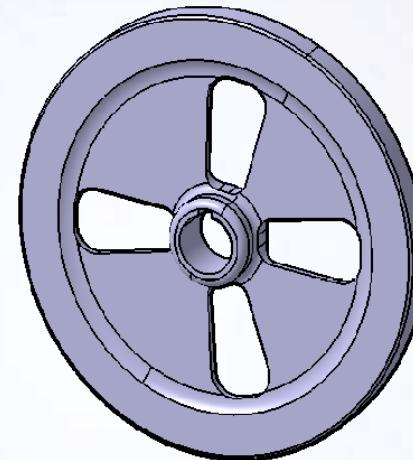
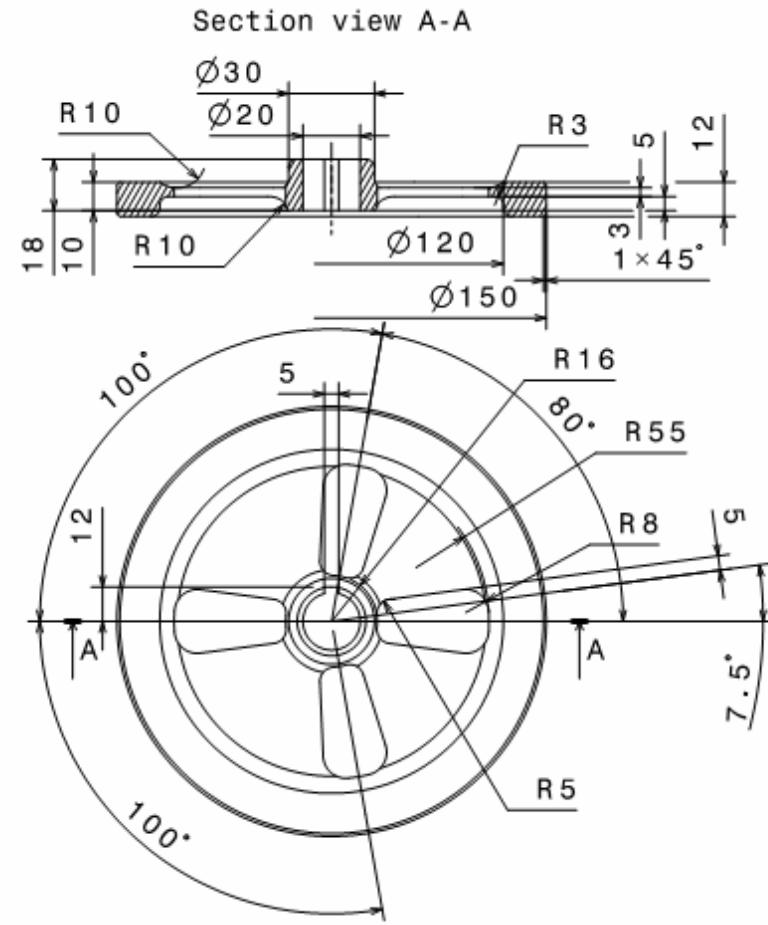
By the end of this exercise you will be able to:

- Create a new part
- Create a circular pattern



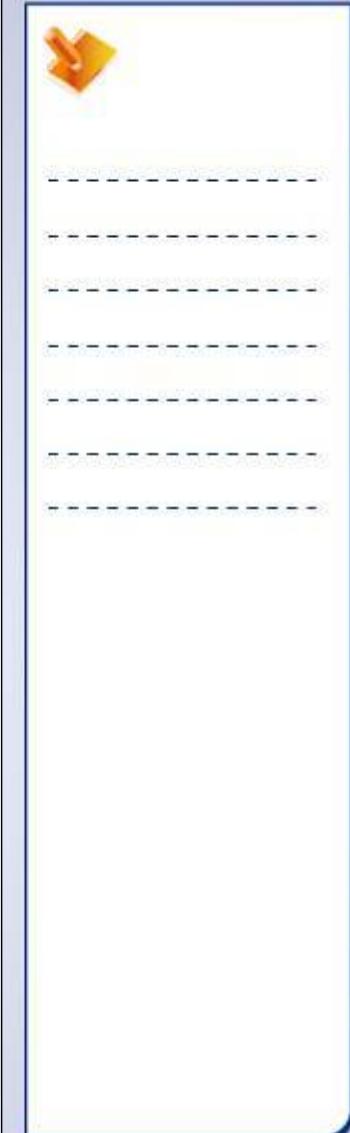
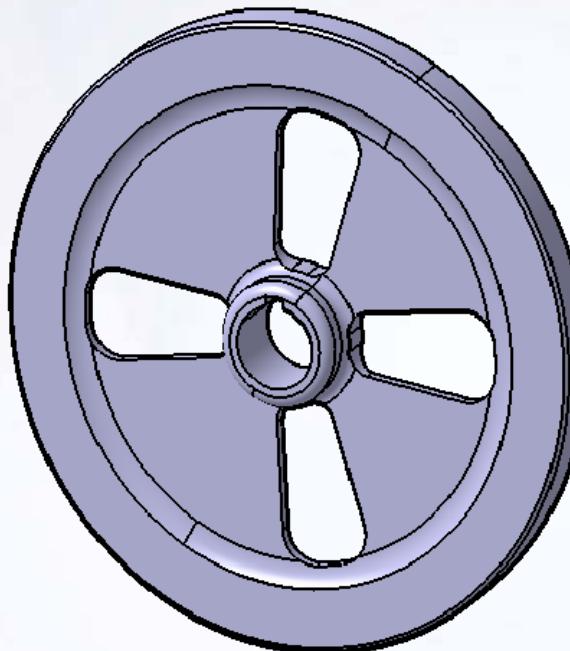
Do it Yourself

Create the part as shown below:



Exercise Recap: Patterns

- ✓ Create a new part
- ✓ Create a circular pattern



Exercise: Wireframe and Publication

Recap Exercise

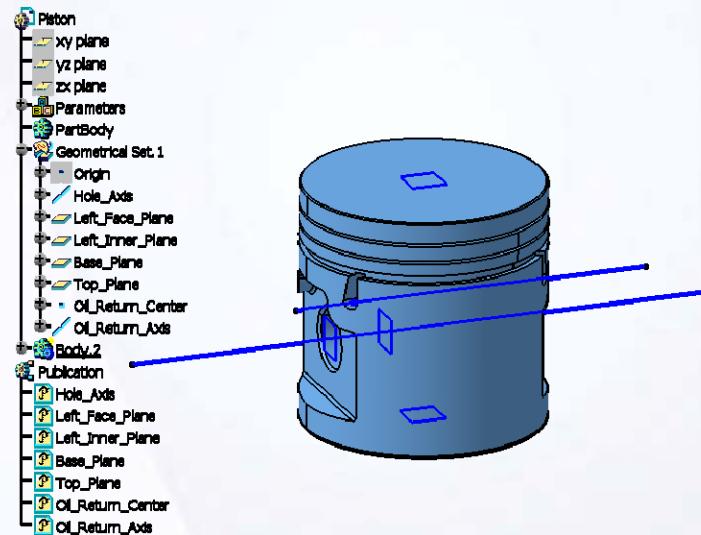


10 min

In this exercise you will create wireframe geometry for a piston and publish it. You will use the tools you have learned in this lesson to complete the exercise with no detailed instruction.

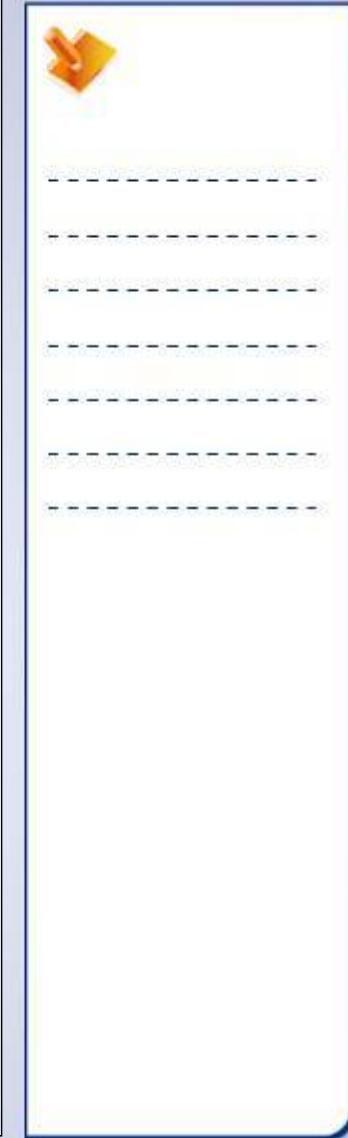
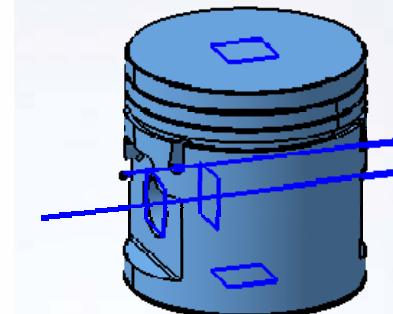
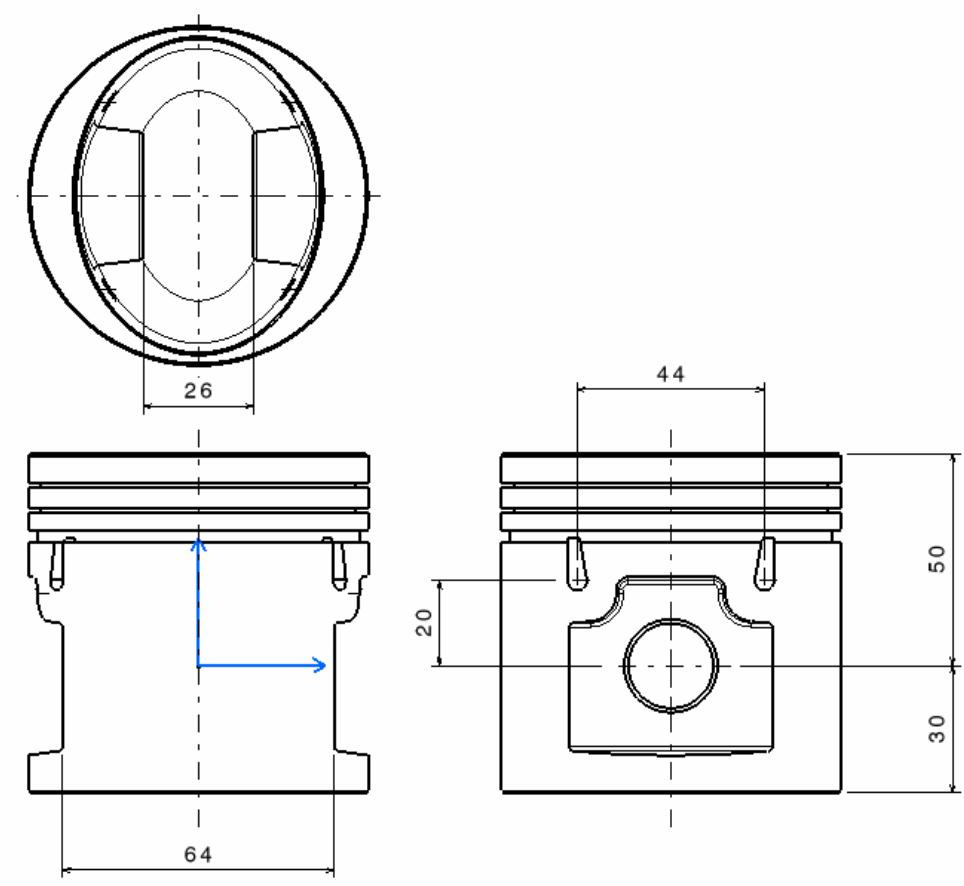
By the end of this exercise you will be able to:

- Create a wireframe geometry
- Change properties of elements
- Publish elements



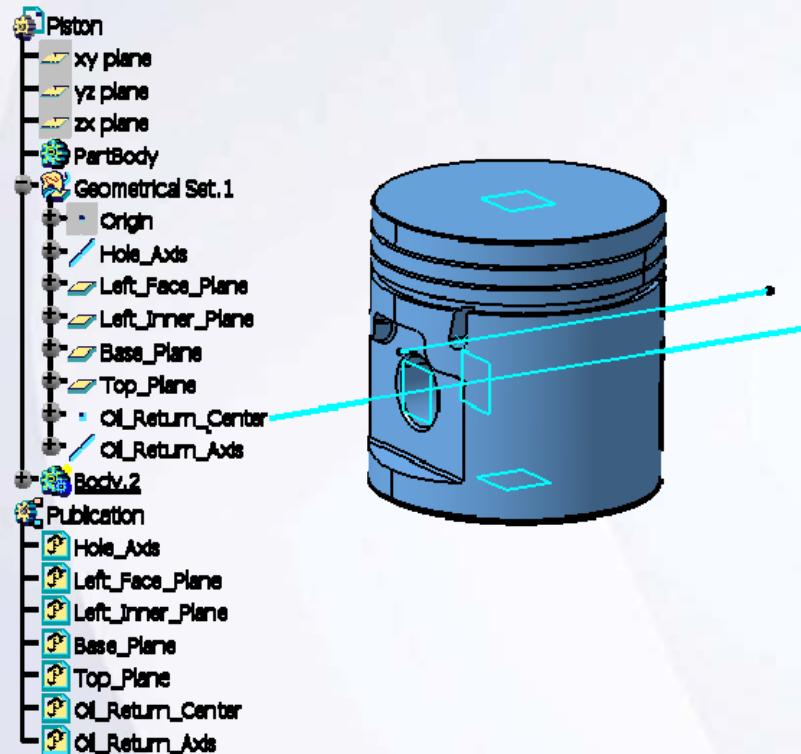
Do it Yourself

1. Load Ex6E.CATPart from database and create the wireframe geometry (shown in dark blue color) using this drawing. Rename and publish these elements.



Exercise Recap: Wireframe and Publication

- ✓ Create wireframe geometry
- ✓ Change properties of elements
- ✓ Publish elements



Case Study: Reusing Data

Recap Exercise

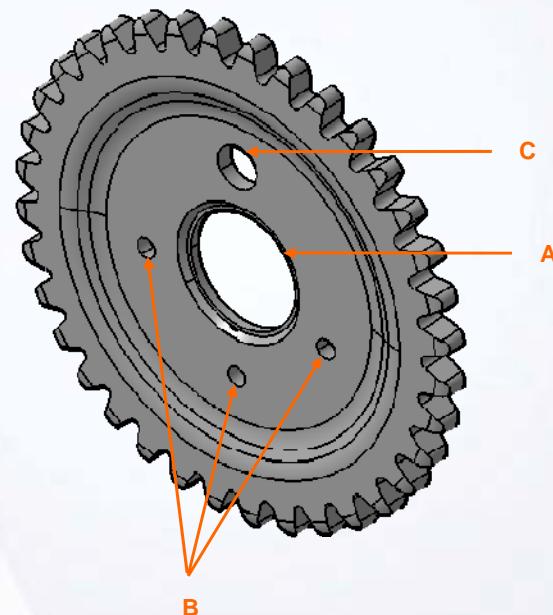


35 min

In this exercise you will create the case study model. Let us recall the design intent of this model:

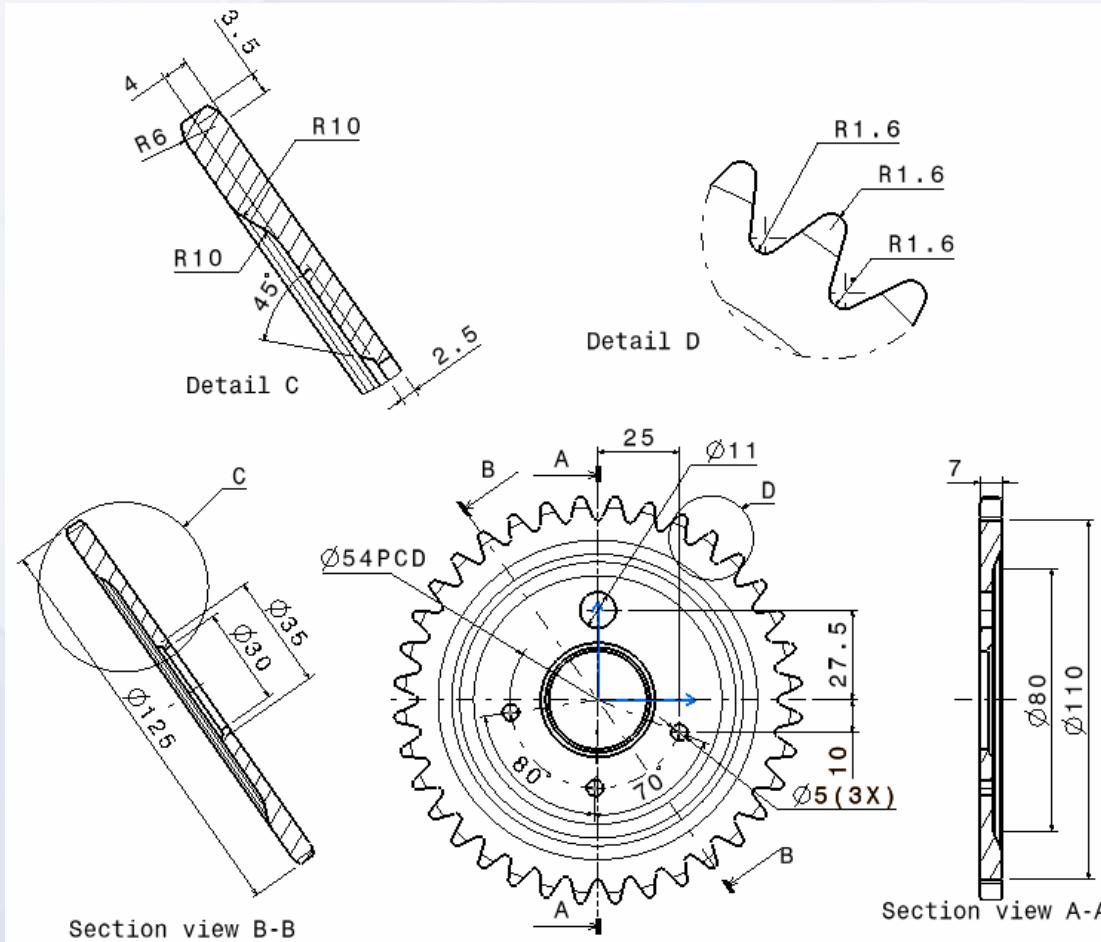
- ✓ The outer diameter must be 125mm.
- ✓ The inner diameter must be 110mm.
- ✓ The number of teeth of sprocket must be 36.
- ✓ The mounting hole (A) must have a diameter of 30mm.
- ✓ The three mounting holes (B) must have a diameter of 5mm and be spaced at pre-defined angles around the central axis.
- ✓ The mounting hole (C) must have a diameter of 11mm.
- ✓ Publish axis of hole (A), hole (B) and rear face of the sprocket.

Using the techniques you have learned so far, create the model without detailed instructions.



Do It Yourself: Create the Sprocket

Create a part using the drawing provided here and publish geometrical elements.



Case Study Recap: Sprocket

- ✓ Create a shaft
- ✓ Create a pocket
- ✓ Create a circular pattern
- ✓ Create a groove
- ✓ Create an edge fillet
- ✓ Create a hole
- ✓ Publish the geometry

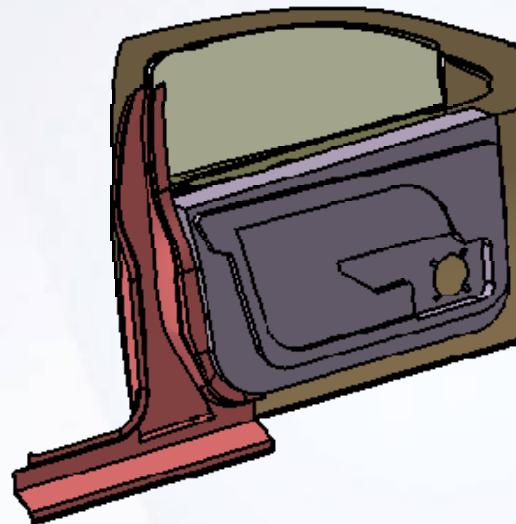


Create Simple Surfaces

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Organize a Surface Model
- ✓ Create the Reference Geometry
- ✓ Create the basic Surface Geometry
- ✓ Perform Trim operation
- ✓ Create Multi-Model Links



8 Hours

Case Study

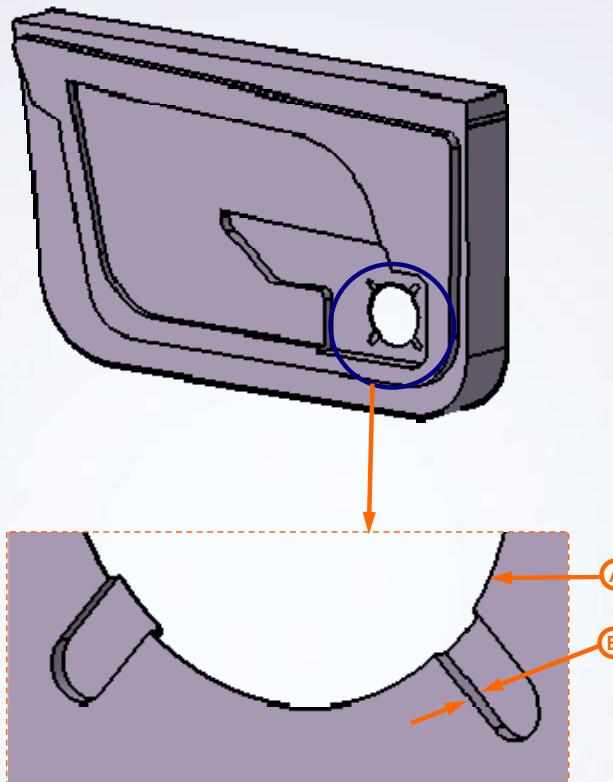
The case study for this lesson is the Inner Door used in the Door assembly.

Design Intent

- ✓ The model must be organized in geometrical sets.
- ✓ The speaker opening diameter (A) must be 100mm.
- ✓ The depth (B) of all four speaker mounting must be 3mm.
- ✓ The result must be single surface.

Stages in the Process

1. Organize the model in geometrical sets.
2. Create a speaker opening curve.
3. Extrude SpkrMountCrvs and trim them with SpkrMountSurs.
4. Extrude the speaker opening curve and use it to split SpkrMountSurs.
5. Create a speaker opening.
6. Trim the speaker opening and speaker mount.



Organizing a Surface Model

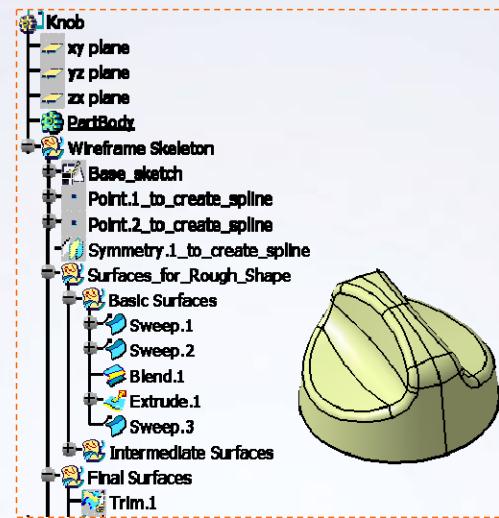
Manage complex model efficiently by logical grouping of geometries, groups or sets. Model organization will help to reduce the size of the specification tree and organize it. It is easier to reorder and replace features.

A geometrical set is the default container for wireframe and surface elements. An ordered geometrical Sets adds the functionality of a linear update sequence to the geometrical set.

Geometrical sets maintain better flexibility. A Group is a visualization element applied on the geometrical Sets

Model Organization can be performed by using following tools:

- ✓ Insert a Geometrical Sets
- ✓ Move elements of Geometrical Sets
- ✓ Ordered Geometrical Sets
- ✓ Create Groups

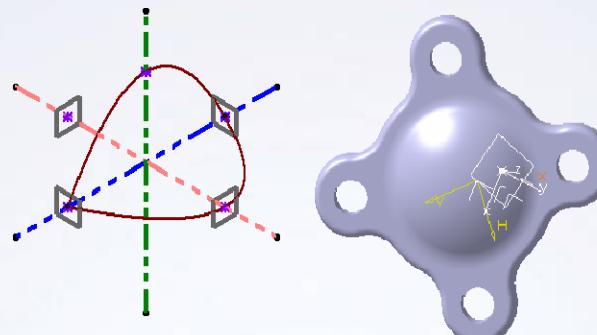


Create the Reference Geometry

Reference Geometry are the basic elements, which provide a stable support to your geometry. This reference geometry can be used to design more intricate wireframe and surface geometries.

A local axis is a user-defined axis system that can be used to define local coordinates. An axis system can automatically be generated when a new part is created. This axis system is defined at the origin of the model and uses the default reference planes for direction.

Curves can be used as guides, limits, or references to create other geometric elements. Surfaces inherits the flaws of the parent curve. Hence great care should be taken while constructing the wireframe.

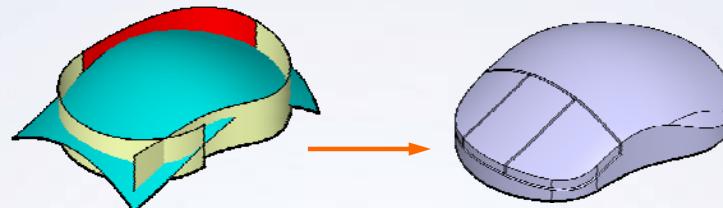


Create the Basic Surface Geometry

Complex 3D shapes need to be defined using surface geometry which is created based on explicit wireframe construction geometry. Surface geometry can then be integrated into the final solid part definition.

In this lesson you will learn the following operations:

- ✓ Creating an Extruded Surface: An extruded surface is created by extruding profiles in a specified direction.
- ✓ Creating a Surface of Revolution: A revolve feature is created by revolving a profile about an axis.
- ✓ Creating a Sphere: A sphere feature is a full or partial spherical surface.
- ✓ Creating a Cylinder Surface: A cylinder surface extrudes a circular profile in a specified direction.

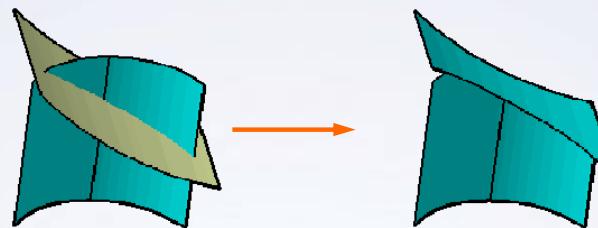


Perform Trim Operation

After the basic surface geometry is created, it may be composed of construction elements that do not describe the finished shape. Operations such as trim are then performed to produce the required geometry.

In this lesson you will learn following operations:

- ✓ Splitting Elements: Use the Split tool to remove unwanted portions of the wireframe and surface elements.
- ✓ Trimming Elements: Use the Trim tool to trim two or more intersecting elements and keep only a part of these elements.
- ✓ Replacing Elements: Use the Replace tool to substitute one element in the specification tree with another element using the replace command.



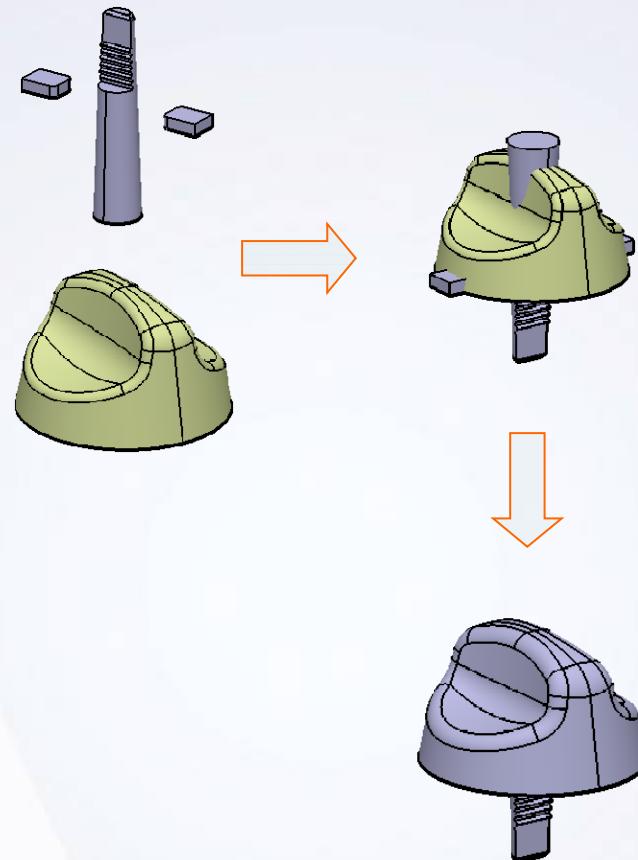
Creating Multi Model Links

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

In the context of the concurrent engineering, Multi Model links enable you to design a model using elements from another model. Many features can be reused in other documents. This reduces the time to recreate the features. You can achieve this by using available Paste Special options.

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

Paste Special: With paste special option many features can be reused in other documents. This reduces the need to recreate features that are commonly used in files and also aids in concurrent engineering.



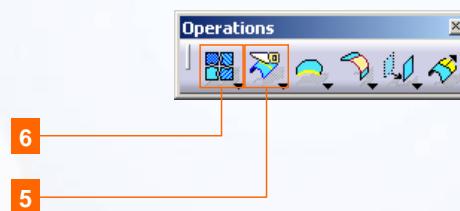
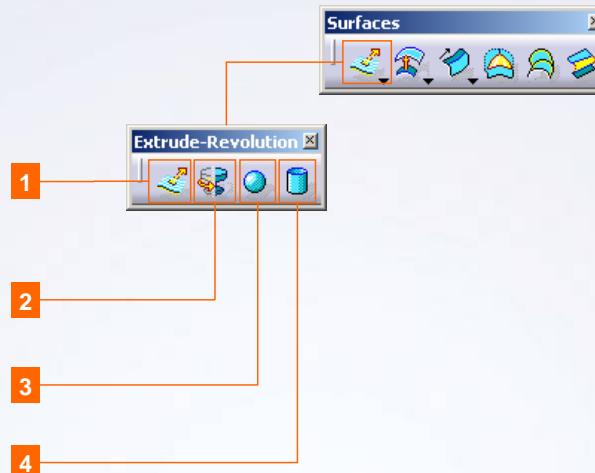
Generative Shape Design Tools

Surfaces

- 1 **Extrude:** creates surfaces by extruding a profile
- 2 **Revolve:** creates surfaces by revolving a profile about an axis
- 3 **Sphere:** creates a full or partial spherical surface.
- 4 **Cylinder:** creates a surface by extruding a circular profile in a specified direction

Operations

- 5 **Split:** removes unwanted portions of wireframe and surface elements
- 6 **Join:** logically fills the gap between two surfaces



Exercise: Complex Wireframe Creation

Recap Exercise

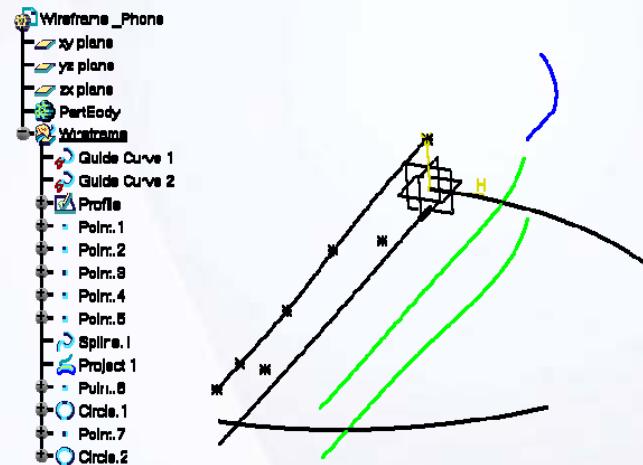


20 min

In this exercise, you will open an existing model and use the tools learned in this lesson to create the wireframe geometry for a mobile phone. Two guide curves and a sketch have already been created for you. You will use points and curves to complete the wireframe geometry. High-level instructions for this exercise are provided.

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

- Create Points
- Create Splines
- Create Projections
- Create Circles



Do it Yourself (1/4)

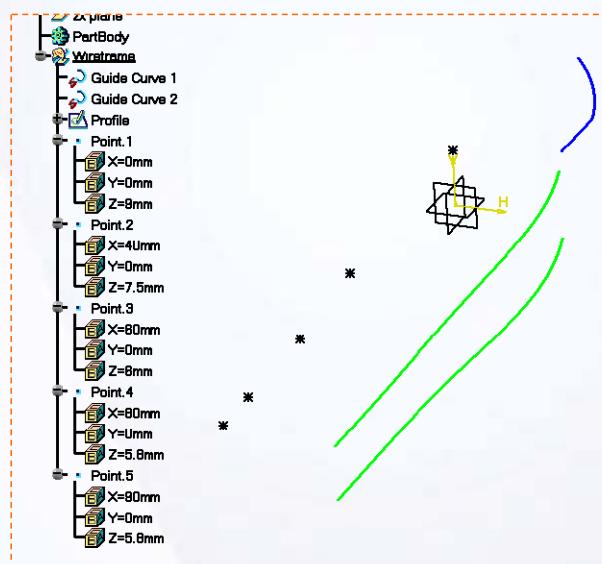
1. Load Ex7B.CATPart.

- Load Ex7B.CATPart. Observe that two curves and a sketch have already been created for you.
 - Ensure that the Wireframe Geometrical Set is active.



2. Create points.

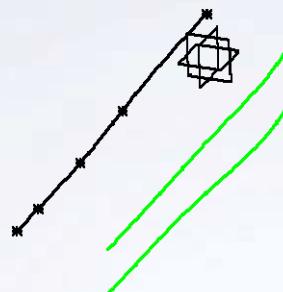
- Create points. These points are used to construct a spline.
 - Create points at the following given coordinates:
 - Point 1:
X = 0mm, Y = 0mm, Z = 9mm
 - Point 2:
X = 40mm, Y = 0mm, Z = 7.5mm
 - Point 3:
X = 60mm, Y = 0mm, Z = 6mm
 - Point 4:
X = 80mm, Y = 0mm, Z = 5.8mm
 - Point 5:
X = 90mm, Y = 0mm, Z = 5.8mm



Do it Yourself (2/4)

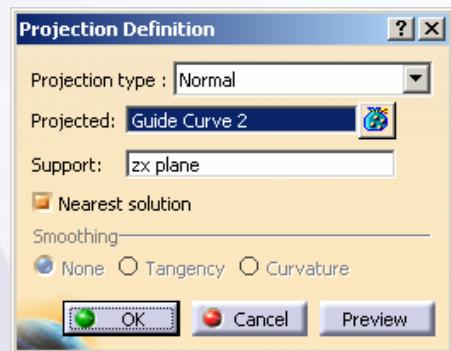
3. Create a spline.

- Create a spline through the points. Apply tangency with respect to YZ plane at both ends of the spline.

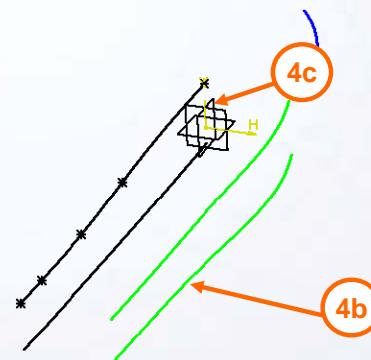
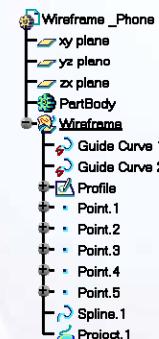


4. Create a projection.

- Create a projection of Guide Curve 2.
 - Click the **Projection** icon.
 - Select Guide Curve 2.
 - Select the ZX plane as the support.
 - Click **OK** to create the projection.



4d



Do it Yourself (3/4)

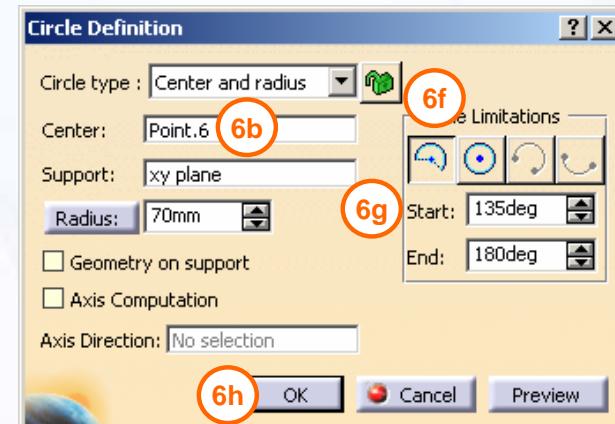
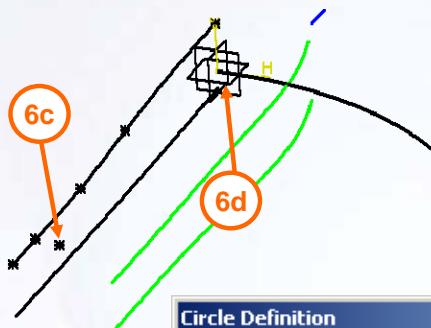
5. Create a point.

- Create a point to be used as a center point for a circle.
 - a. Create a point by coordinates at:
 - X = 70, Y = 0, Z = 0



6. Create a part arc.

- Using the Circle tool, create a part arc.
 - a. Click the **Circle** icon.
 - b. Select **Center and Radius** from the Circle type menu.
 - c. Select the point created in the last step as the circle center.
 - d. The XY plane as the circle support.
 - e. Enter a radius of [70mm].
 - f. Select **Part Arc**.
 - g. Start the arc at [135 deg] and end the arc at [180deg].
 - h. Click **OK** to create the circle.



Do it Yourself (4/4)

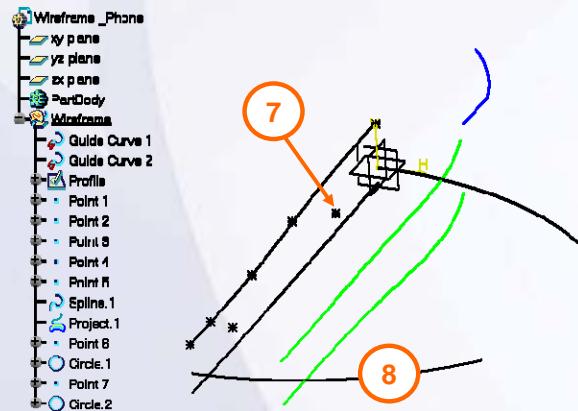
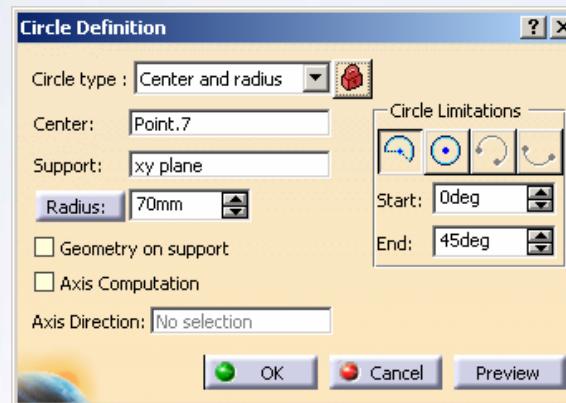
7. Create a point.

- Create another point to locate another circle center at the following co-ordinates
 - X = 20, Y = 0, Z = 0

8. Create a part arc.

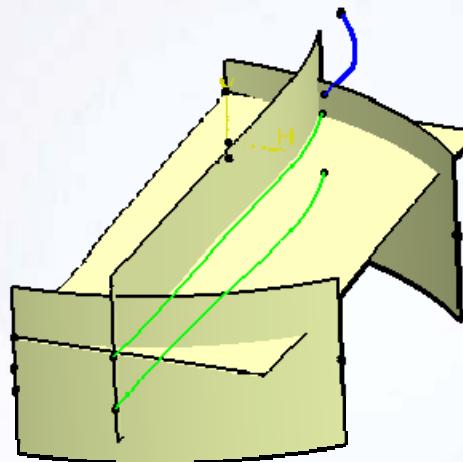
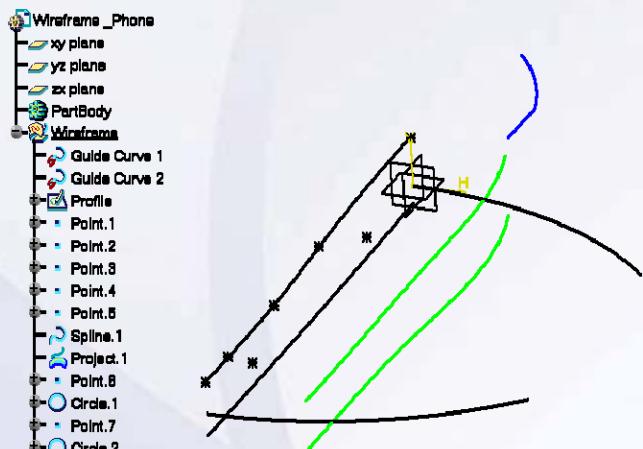
- Using the Circle tool, create a part arc as shown.

9. Close the file without saving it.



Exercise Recap: Complex Wireframe Creation

- ✓ Create a point
- ✓ Create a spline
- ✓ Create a projection
- ✓ Create a circle



Exercise: Complex Wireframe Creation

Recap Exercise

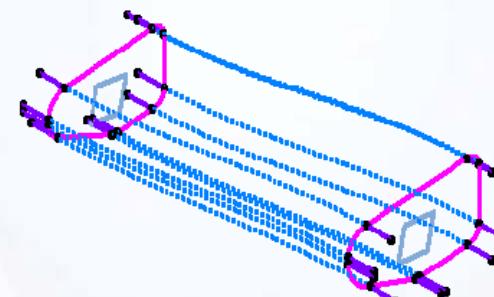


20 min

In this exercise, you will practice the tools learned in this lesson to create the wireframe geometry necessary for the shell of a Cradle. You will be creating the guide curves connecting the vertices of two given sections using the Spline tool. Later, you will create the same guide curves using the Connect curve tool. This is to understand the difference between the two tools. To save time, simple wireframe elements have already been created for you. Detailed instructions for this exercise are provided for all new topics.

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

- Create a Line
- Create a Spline
- Create a Connect curve



Do it Yourself (1/5)

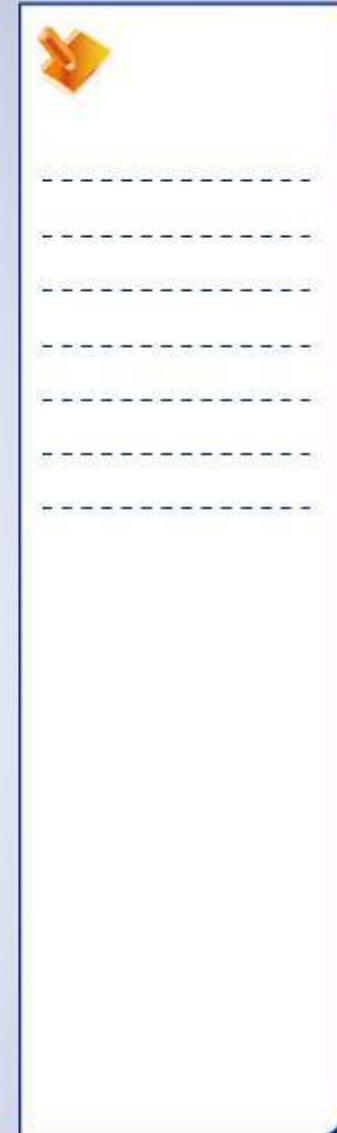
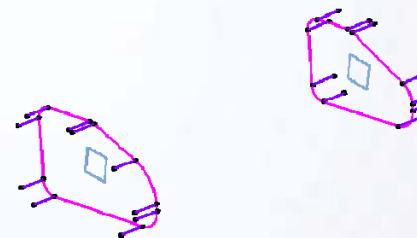
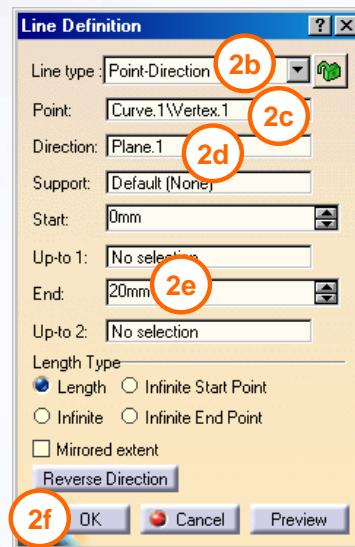
1. Load Ex7C.CATPart.

- Load Ex7C.CATPart. This part already has some wireframes and planes.
 - a. Observe that all the wireframe elements have been created in a separate Geometrical Set.
 - b. Ensure the 'Curves' Geometrical Set is active.



2. Create lines.

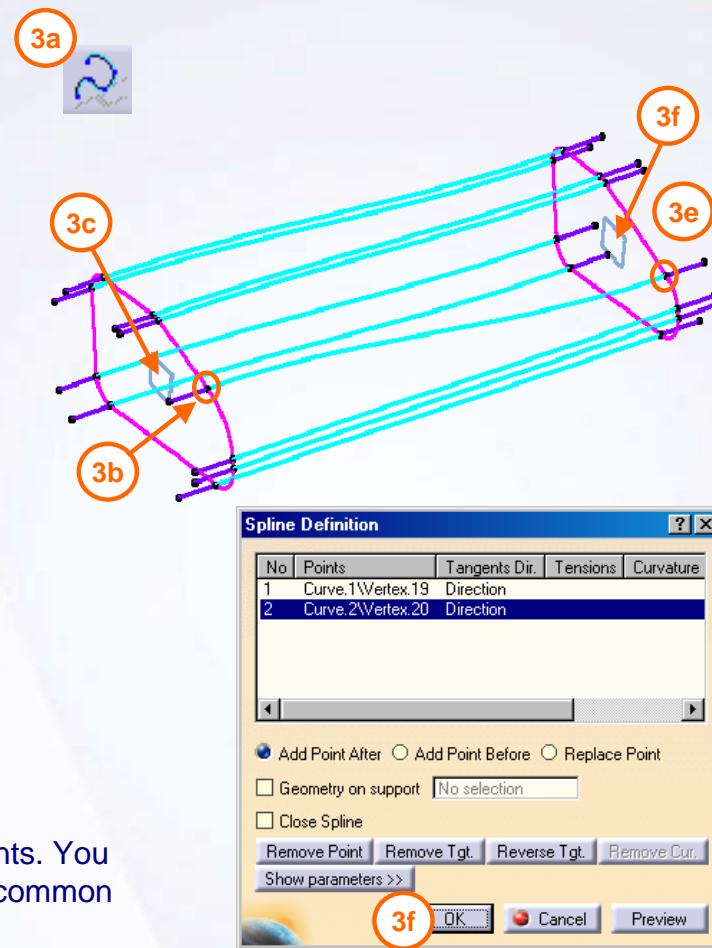
- Create lines of 20mm length normal to the sections at the vertices as shown.
 - a. Click the **Line** icon.
 - b. Select Line Type as **Point-Direction**.
 - c. Select point as Vertex of a Curve.1.
 - d. Select direction as Plane.1.
 - e. Enter [20mm] as end.
 - f. Click **OK** to complete the line definition.
 - g. In a same way, create lines at all 20 vertices of Curve.1 and Curve2.



Do it Yourself (2/5)

3. Create Splines.

- Create splines to connect two corresponding vertices on the both the sections. The steps to create a spline is shown below. You will create other set of splines accordingly.
 - Click the **Spline** icon.
 - Select the start point of Line.1.
 - Select Plane.1 to make the spline tangent to it. Ensure that the arrow is pointing in the correct direction. If it is not, click on the arrow to change its direction.
 - Select the start point of the corresponding line on the other section.
 - Select plane.2 to make the spline tangent to it.
 - Click **OK** to create the spline.
 - Similarly connect all the vertices of the first section with the corresponding vertices of the second section.



Observation:

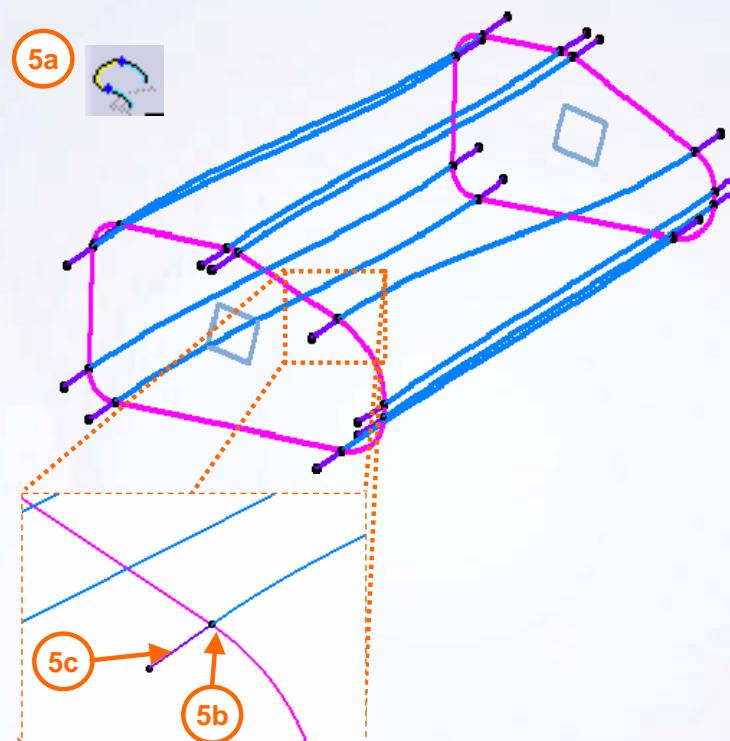
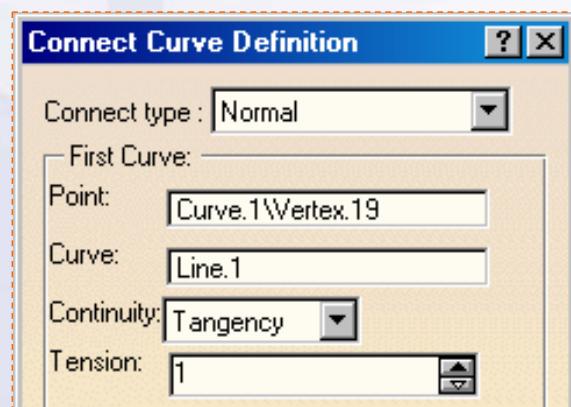
Spline can be created between two or more points. You can imply the tangency at each point using the common vector direction at each point.

Do it Yourself (3/5)

4. Hide the splines created in the previous step.

5. Create Connect curves

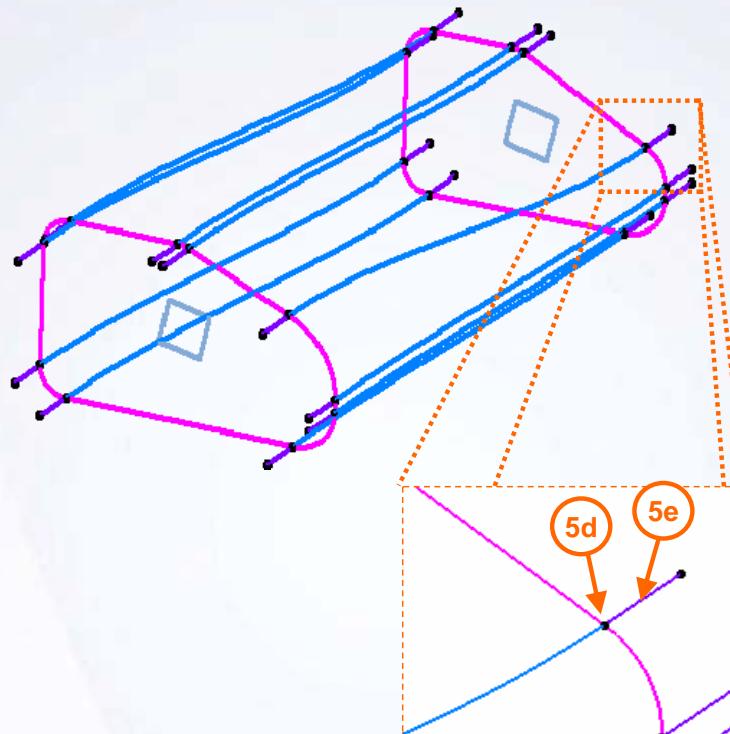
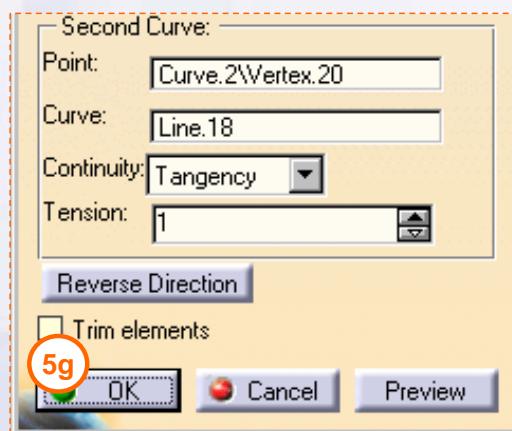
- Create a Connect curve between two corresponding vertices of the two sections. This is to compare the creation of the 3D curves using Spline and Connect curve tools.
 - Click the **Connect curve** icon.
 - Select the start point of the first line as point input.
 - Select Line .1 as a curve.



Do it Yourself (4/5)

5. Create a Connect curve (continued)

- d. Select the start point of the corresponding line on the second section.
- e. Select the same line as a curve.
- a. Ensure that the arrow is pointing in the correct direction. If it is not, click on the arrow to change its direction.
- g. Select **OK** to create a connect curve.

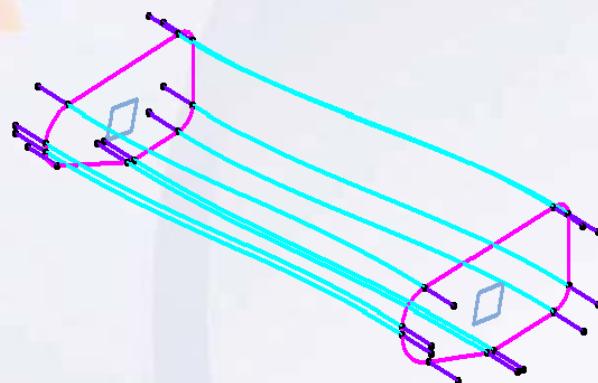


Observation:

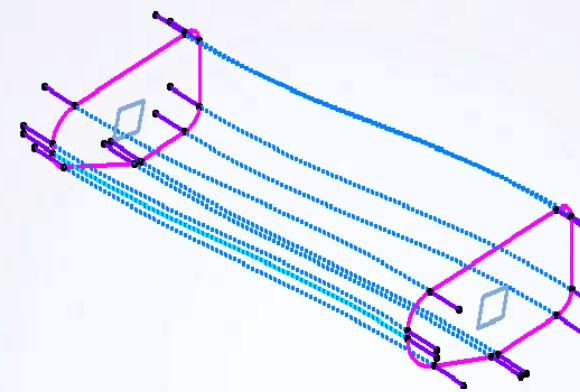
Connect curve can be created between two points only.
You can imply the tangency at each point using the corresponding curve on which the point lies.

Do it Yourself (5/5)

6. Study and note the difference between the Spline curve and Connect curve.



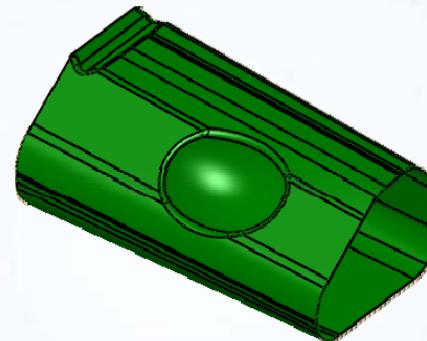
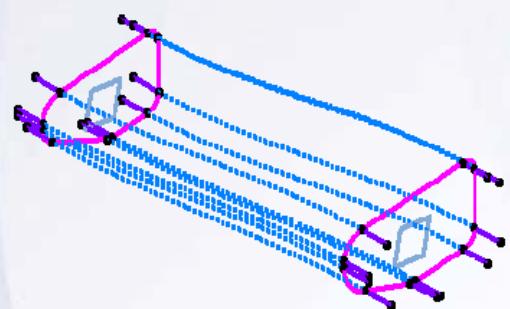
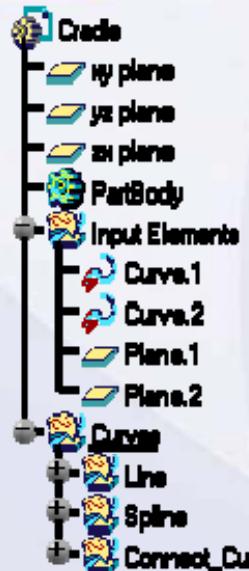
Vertices connected using Spline curves



Vertices connected using Connect curves

Exercise Recap: Complex Wireframe Creation

- ✓ Create a Line
- ✓ Create a Spline
- ✓ Create a Connect curve



The wireframe created in this exercise could be used to create a surface feature, as shown above. You will learn how to create surface features in the subsequent steps .

Exercise: Surface Creation

Recap Exercise



20 min

In this exercise, you will create a new model using pre-defined features. You will practice the stacking commands and the tools learnt in the previous step. High-level instructions for this exercise are provided.

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

- Create a Cylinder
- Create a Sphere
- Create a Point on a surface
- Create a Trim

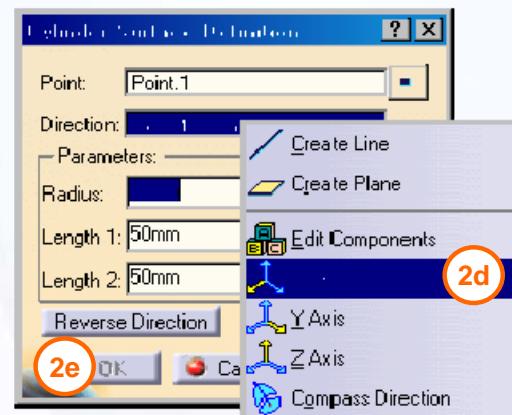
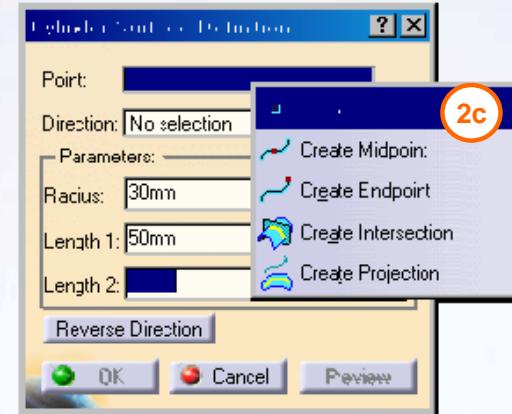
A large rectangular area with a thin blue border, containing a small yellow puzzle piece icon in the top-left corner. The interior is divided into five horizontal dashed-line sections for writing.

Do it Yourself (1/3)

1. Create a new part file (Ex7E.CATPart) in Generative Shape Design Workbench.

2. Create a Cylinder.

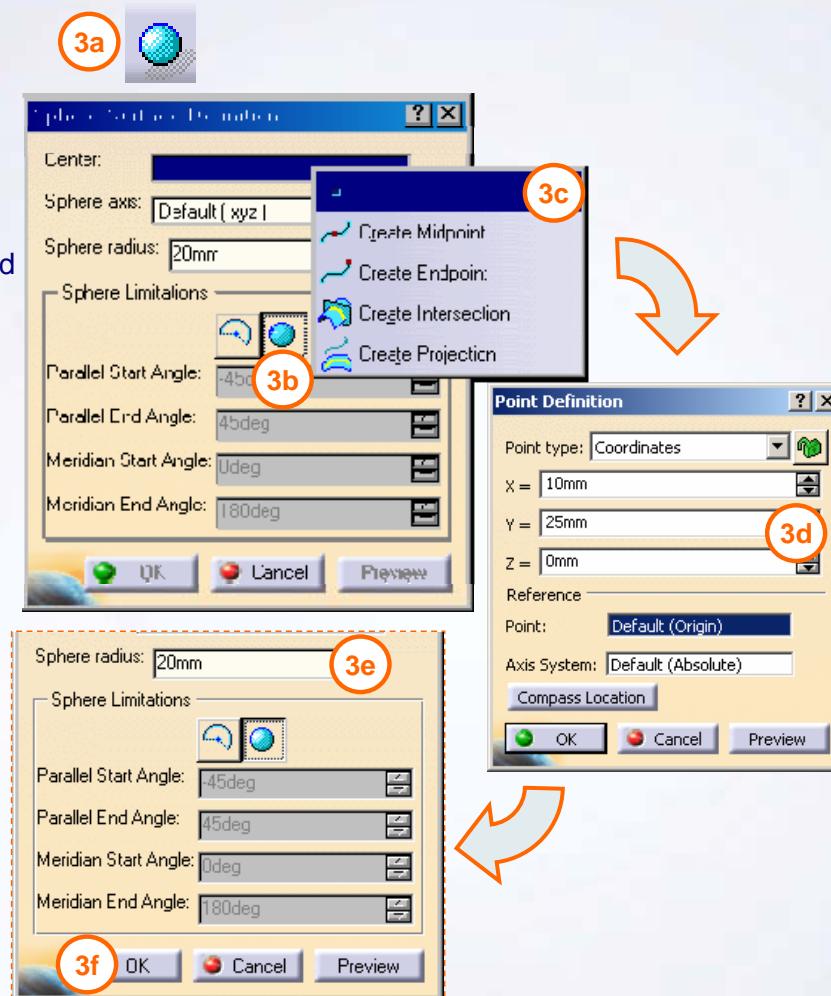
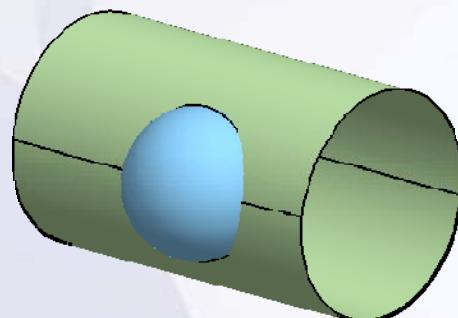
- You will contextually create the inputs required for a cylinder from **Cylinder Surface Definition** dialog box.
 - Select the **Cylinder** icon.
 - Specify the parameters as follows:
 - Radius = 30mm
 - Length 1 = 50mm
 - Length 2 = 50mm
 - Create a point contextually on absolute co-ordinate system.
 - Specify the Direction as X axis contextually.
 - Click **OK** to create a Cylinder.



Do it Yourself (2/3)

3. Create a Sphere.

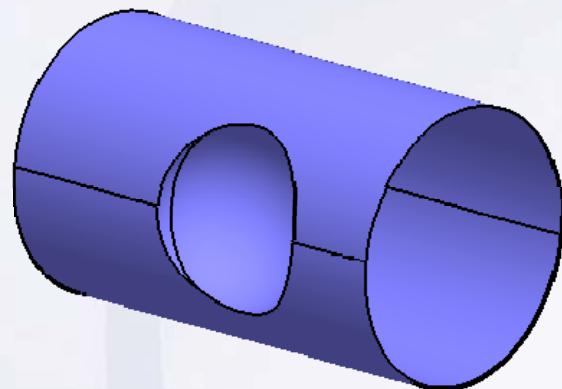
- You will contextually create the inputs required for a sphere from the **Sphere Surface Definition** dialog box.
 - Click the **Sphere** icon.
 - Specify the full sphere icon
 - Right-click on the center selection field and create a point contextually at the following given co-ordinates:
 - X =10mm, Y=25mm, Z=0mm
 - Specify the default Sphere axis.
 - Specify the Sphere radius as 20mm
 - Click **OK** to create the Sphere.



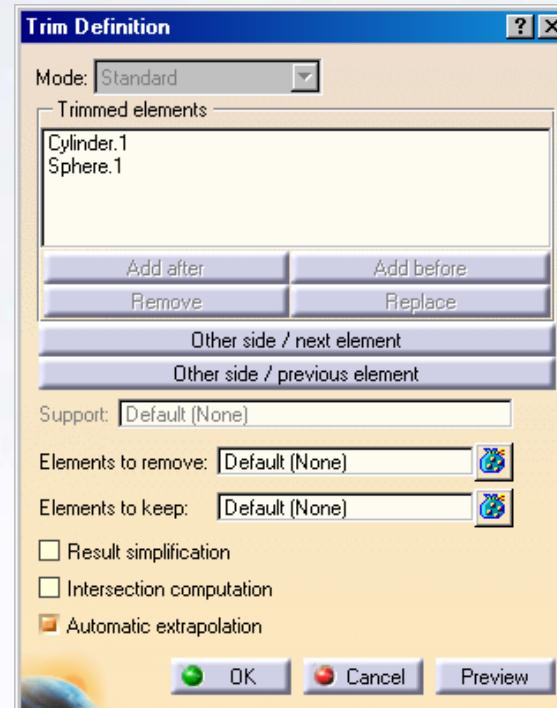
Do it Yourself (3/3)

4. Create a Trim

- Create a trim between a Cylinder and a Sphere to get the shown result.

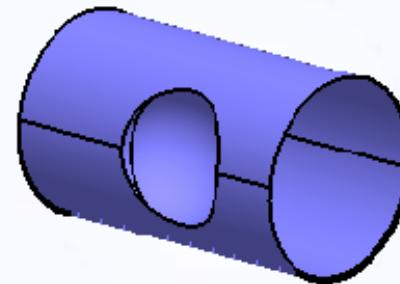


5. Close the file without saving it.



Exercise Recap: Surface Creation

- ✓ Create a Cylinder
- ✓ Create a Sphere
- ✓ Create a Point on a surface
- ✓ Create a Trim



Case Study: Create Simple Surfaces

Recap Exercise

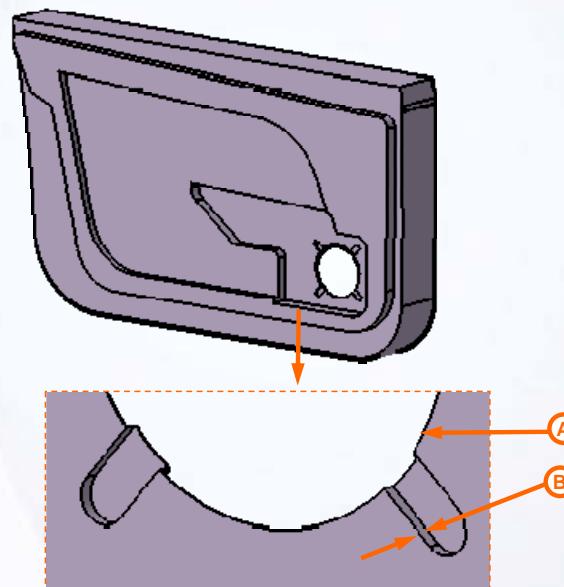


10 min

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

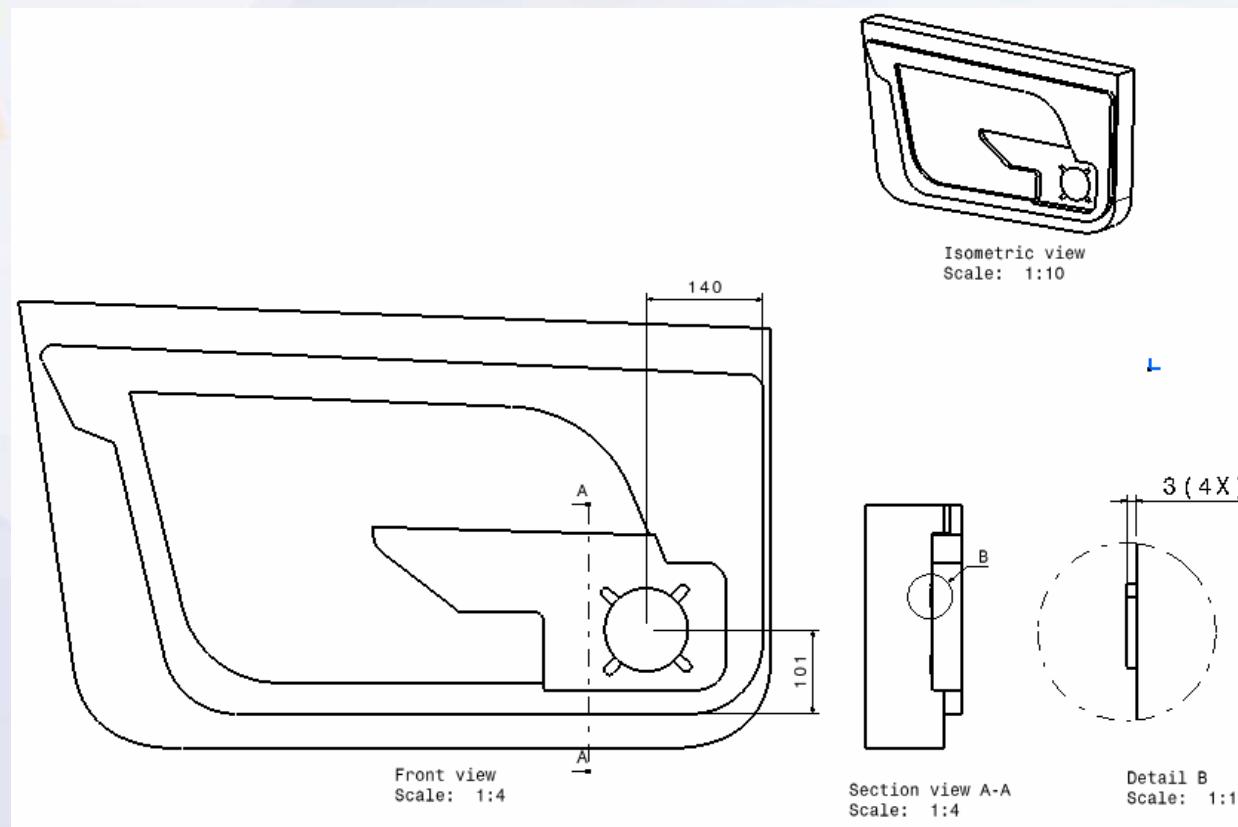
- ✓ The model must be organized in geometrical sets.
- ✓ The speaker opening diameter (A) must be 100mm.
- ✓ The depth (B) of all the four speaker mountings must be 3mm.
- ✓ The resulting surface must be in a separate geometrical set.

Using the techniques you have learnt so far, create the model without detailed instruction.



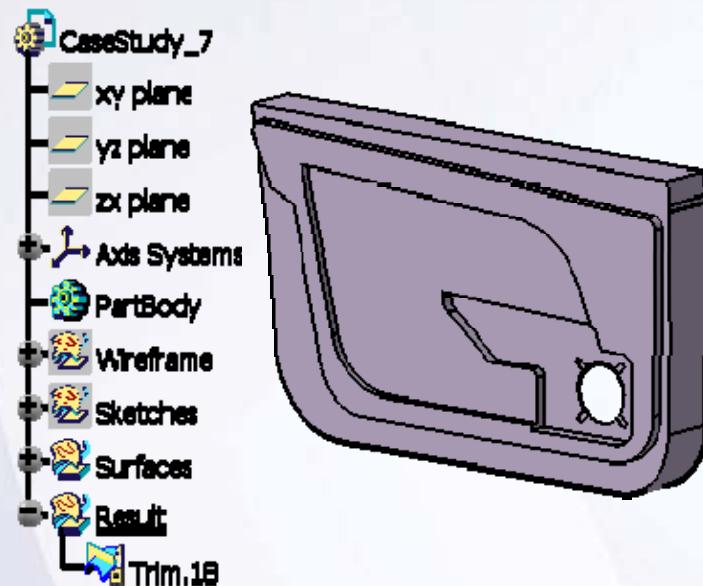
Do It Yourself: Create the Simplified Inner Door

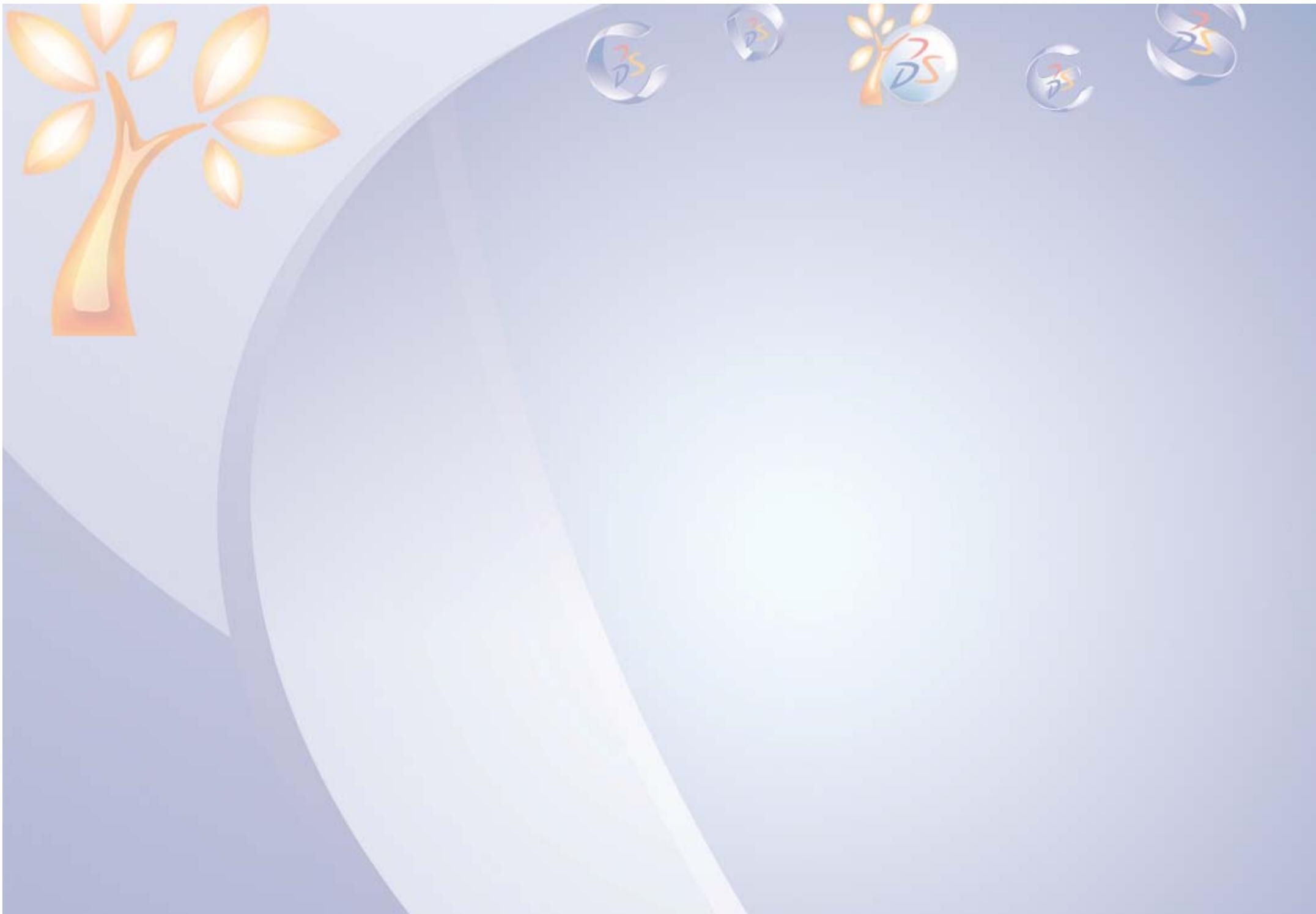
Modify the CaseStudy7.CATPart using the drawing provided here.



Case Study Recap: Simplified Inner Door

- ✓ Organize the model
- ✓ Create an extrude surface
- ✓ Split the surfaces
- ✓ Create reference geometry
- ✓ Trim the surface





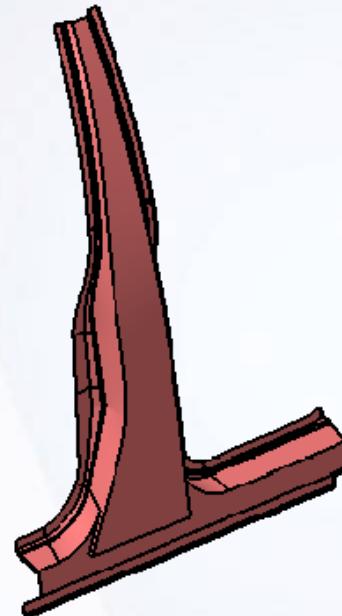
Create Complex Surfaces

8

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Creating the complex surface geometry
- ✓ Perform fillet operation



4 Hours

Case Study

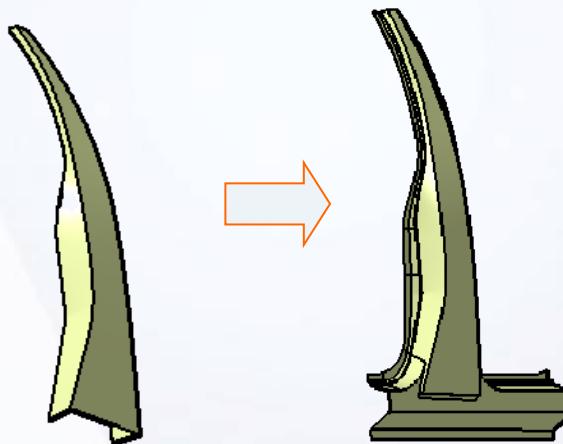
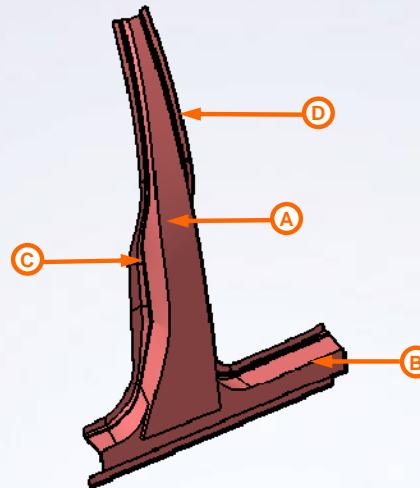
The case study for this lesson is a simplified B pillar, it focuses on creation of surface features and performing surface operation that incorporate the design intent of the part.

Design Intent

- ✓ The main junction surface (A) must pass through the four section curves.
- ✓ Length of the base junction surface (B) must be 700mm
- ✓ Sharp edges must not be present between the main junction surface and base junction surface.
- ✓ The front pillar surface (C) and the back pillar surface (D) must pass through the respective guide curves.
- ✓ The result must be a single surface.

Stages in the Process

1. Create Multi Section surface.
2. Fill the surface and create a join..
3. Create an extruded surface.
4. Apply fillet to the sharp edges.
5. Create a sweep.
6. Join the surface to get a single result.

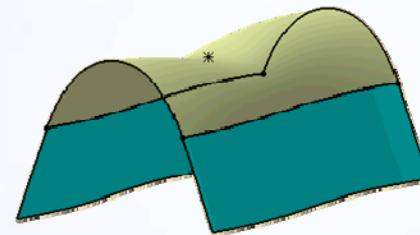
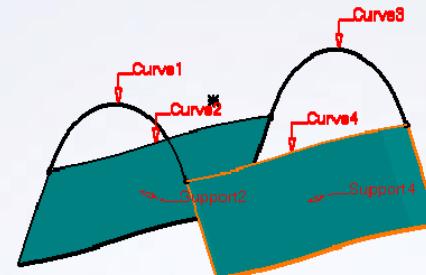


Creating the Complex Surface Geometry

Sweep is a surface generated by sweeping a profile along a guide curve with respect to a spine. The profile can be a user-defined or pre-defined profile. A Spine can control the orientation of the profile as it sweeps along the guide curve's. By default the first guide is used as the spine for the Swept feature.

Complex surfaces can be created by following commands:

- ✓ Creating an Offset Surface: The new instances will be created in a new open body.
- ✓ Creating a Fill Surface: Surface can be filled inside closed boundary.
- ✓ Creating a Blend Surface: A blend surface is used to create a surface between two wireframe elements.
- ✓ Creating a Multi-Section Surface: A surface is computed by passing through two or more consecutive sections along a spine is called Multi-Section surface.

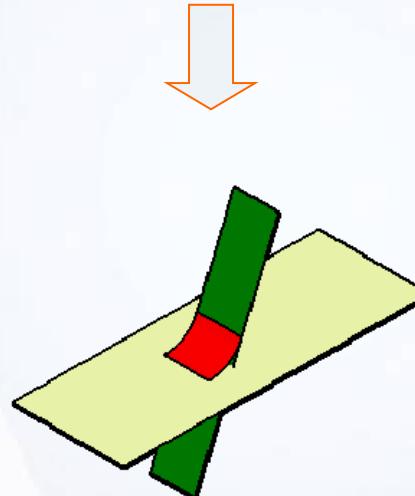
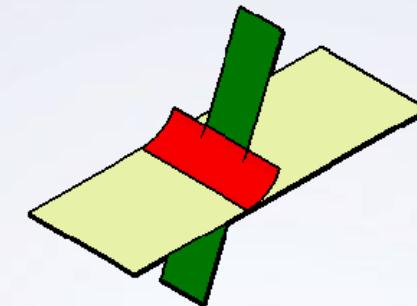


Perform Fillet Operations

After the basic surface geometry is created, it may be composed of construction elements that do not describe the finished shape. Operations such as trim, join, extrapolate, and transform are then performed to produce the required finished geometry.

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

Fillets are used to remove sharp edges on the parts. The Edge Fillet tool is used to provide a transitional surface along a sharp edge of a surface.



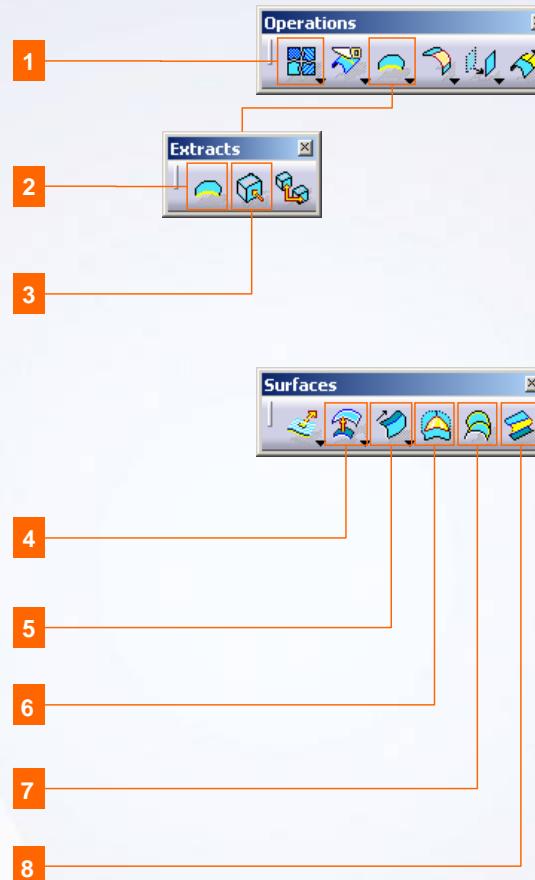
Main Tools

Operations

- 1 **Join:** logically fills the gap between two surfaces.
- 2 **Boundary:** creates boundary curves of internal or external surface edges.
- 3 **Extract:** creates an extract sub elements from a surface.

Surfaces

- 4 **Offset:** creates a surface offset from an existing surface.
- 5 **Sweep:** creates a surface by sweeping a profile along a spine.
- 6 **Fill:** creates a surface inside a close boundary.
- 7 **Multi-section surface:** creates a surface passing through two or more consecutive sections along spine.
- 8 **Blend:** creates a surface between two wireframe elements.



Exercise: Multi-Sections Surface

Recap Exercise

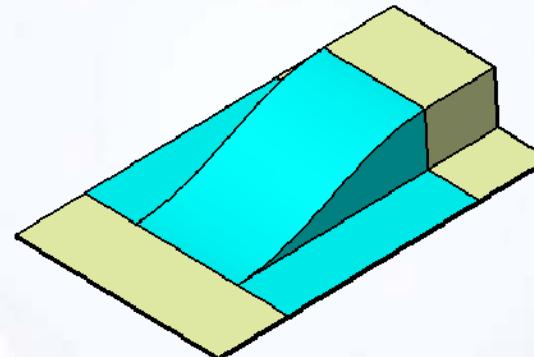


10 min

In this exercise, you will open an existing model consisting of wireframe and surface inputs. You will practice how to use the Multi-Sections surface tool. High-level instructions for this exercise are provided.

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

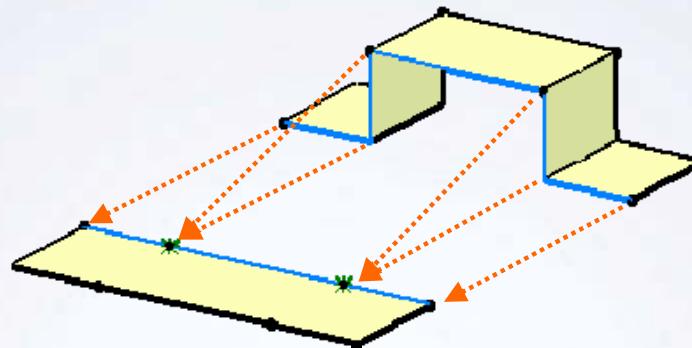
- Create a Multi-Sections surface.



Do it Yourself (1/2)

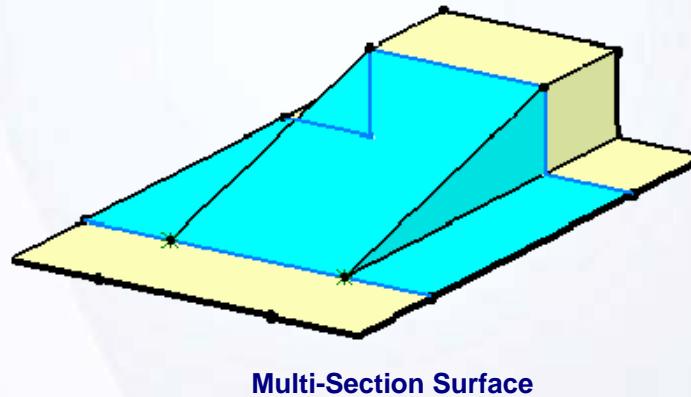
1. Load Ex8B.CATPart.

- Load Ex8B.CATPart. This part already has some curves and surfaces created for you.
 - Observe that the two sections (flat and formed sections) are in different geometrical sets.
 - Create a geometrical set called 'Multi-Section'.



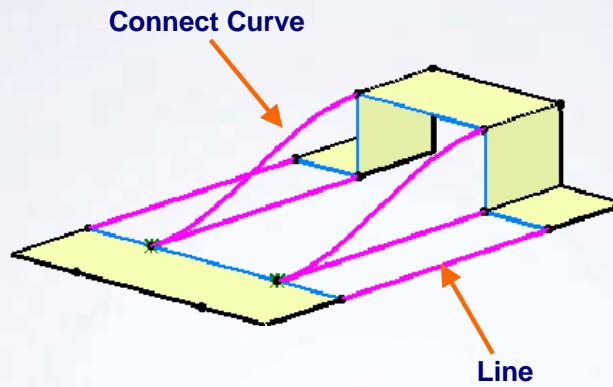
2. Create a Multi-Sections surface.

- Create a Multi-Sections surface from a given profile. Use manual coupling to guide the surface.



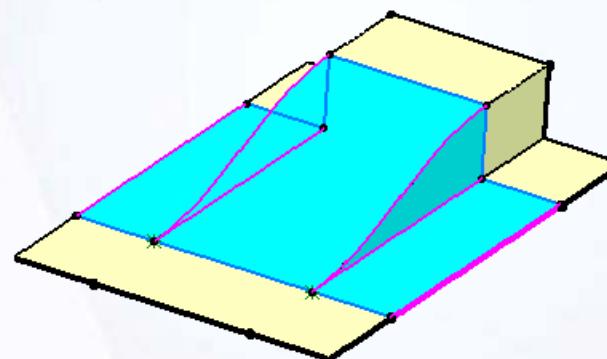
Do it Yourself (2/2)

3. Hide the Multi-Sections surface created in the previous step.



4. Create the Guide curves.

- Create the guide curves between the two sections as shown.



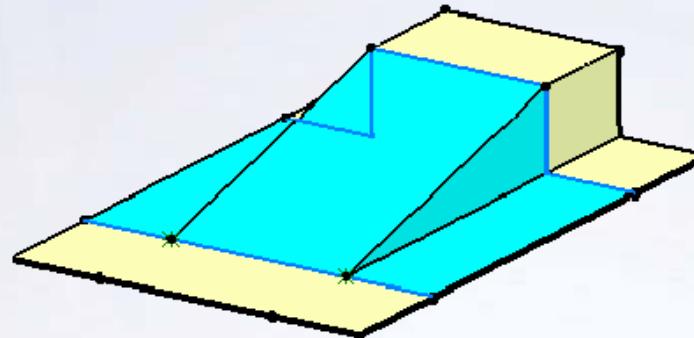
5. Create a Multi-Sections surface.

- Create a Multi-Section surface from a given profile. Use the guide curves created in the previous step.

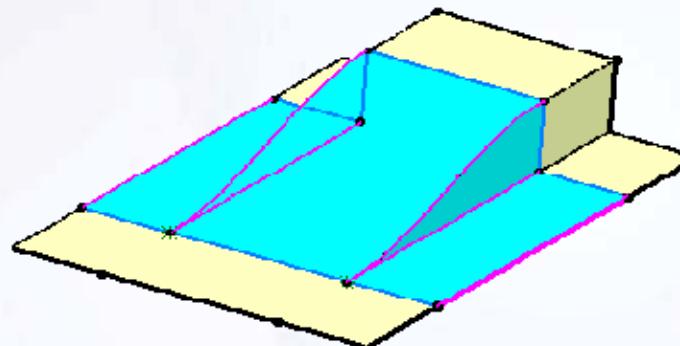
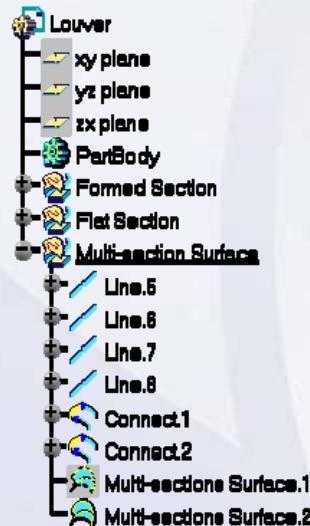
6. Close the part without saving it.

Exercise Recap: Multi-Sections Surface

- ✓ Create a Multi-Section Sweep using coupling points
- ✓ Create Multi-Section surface using guide curves



Create a Multi-Section Sweep using coupling points



Create a Multi-Section Sweep using guide curves

Exercise: Complex Surfaces Creation

Recap Exercise

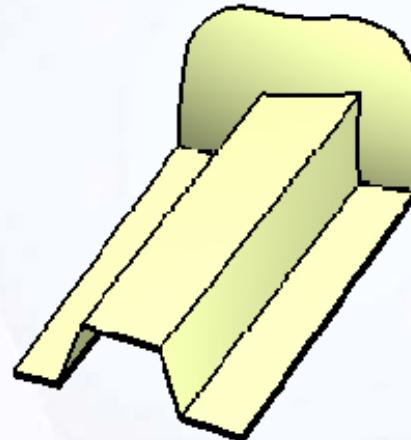


20 min

In this exercise, you will create a thermal insulation cover using simple surfaces.
High-level instructions for this exercise are provided.

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

- Create a Multi-Sections surface
- Create a Sweep surface
- Create an Extrude surface
- Create a surface Trim



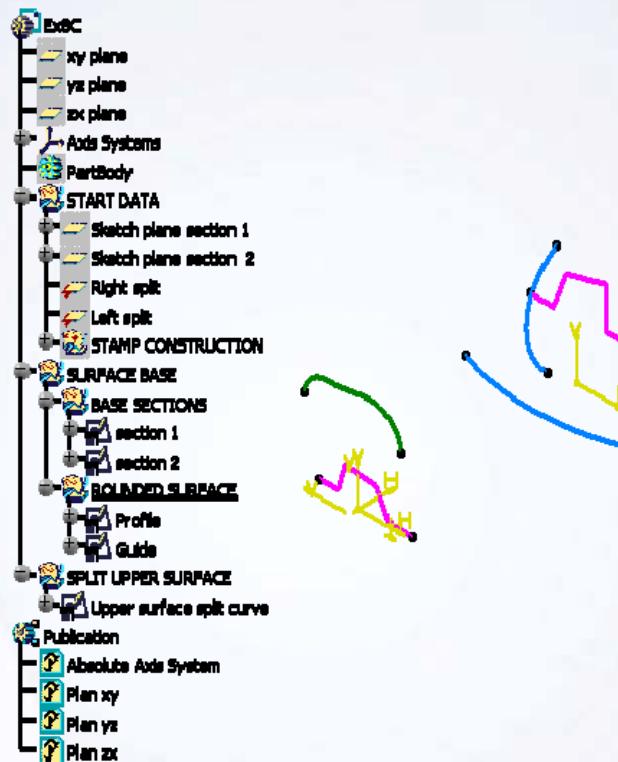
Do it Yourself (1/4)

1. Load Ex8C.CATPart.

- Load Ex8C.CATPart. This part already has some curves created for you.

2. Analyze the model

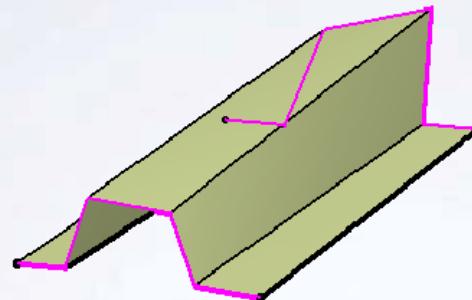
- Study the model along with the specification tree to understand the sequence of the model construction.



Do it Yourself (2/4)

3. Create a main surface

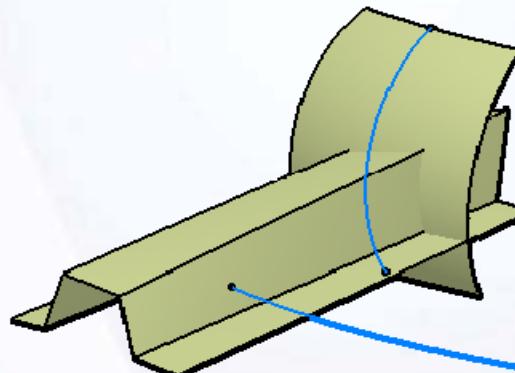
- In Geometrical Set 'BASE SECTIONS', create Multi-Sections surface using section 1 and 2 as Profiles and 'Vertices' as the coupling option.



Multi-Section using 'Vertices' option

4. Create rounded front wall

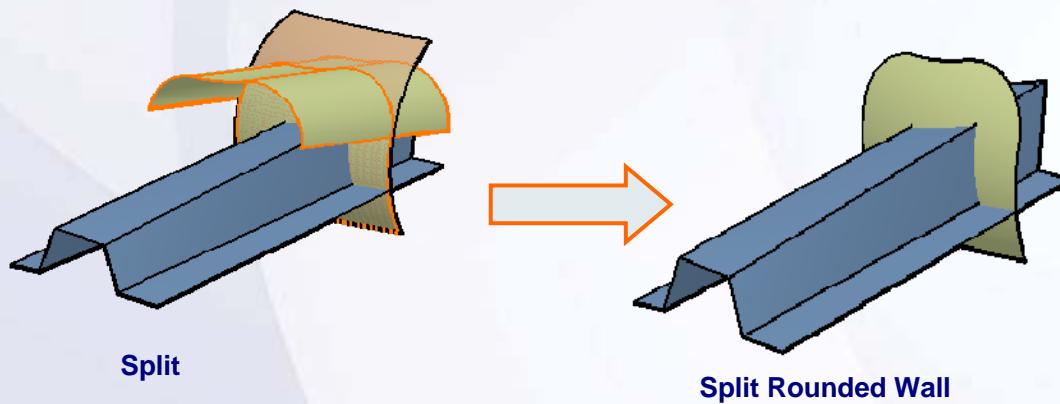
- In Geometric Set 'Rounded Surface', create a sweep surface as shown using Profile and Guide.
- Display the Split planes in the 'Start data' Geometrical Set.
- Specify these planes as Relimiter 1 and 2 in the Sweep dialog box.



Do it Yourself (3/4)

5. Create upper Split surface

- In Geometric Set 'Split upper Surface' create an extruded surface as shown.
- In Geometric Set 'Rounded Surface' create the split as shown.

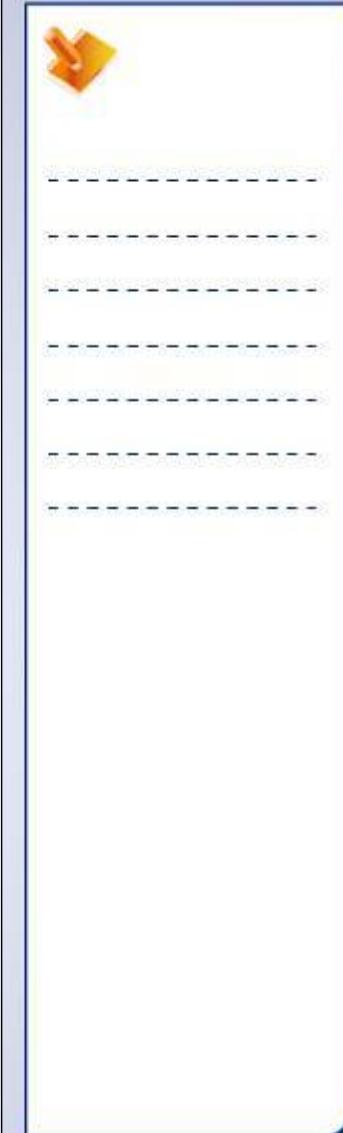
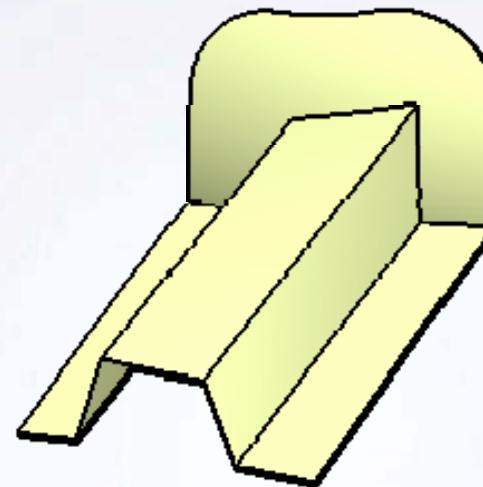


Do it Yourself (4/4)

6. Create Final surface

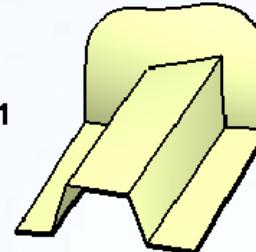
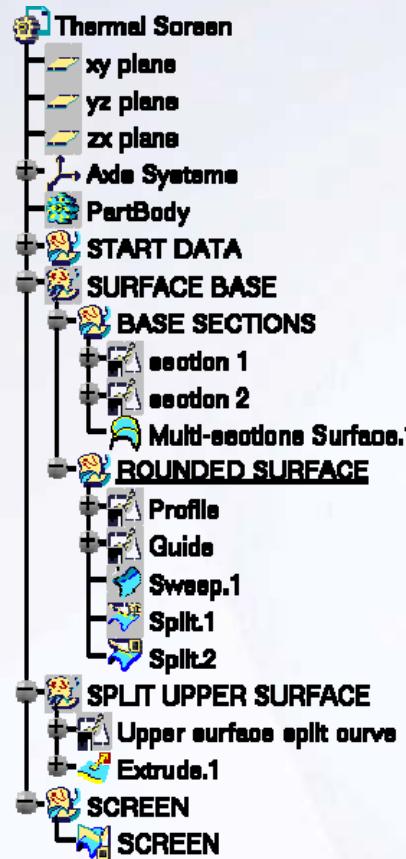
- Create a Geometrical Set called 'Screen' under the father node of the thermal screen.
- Create a trim between the main surface and the front wall surface
- Rename the trim surface as 'Screen'.

7. Close the part without saving it.



Exercise Recap: Complex Surfaces Creation

- ✓ Create a Multi-Sections surface
- ✓ Create a Sweep surface
- ✓ Create an Extrude surface
- ✓ Create a surface Trim



Exercise: Boss and Flange Creation

Recap Exercise

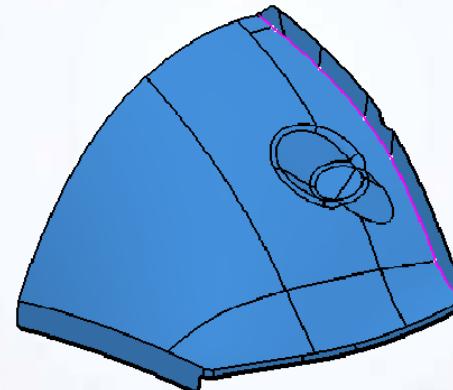


30 min

In this exercise, you will create a flange surface of the wheel arch and add the boss. High-level instructions for this exercise are provided.

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

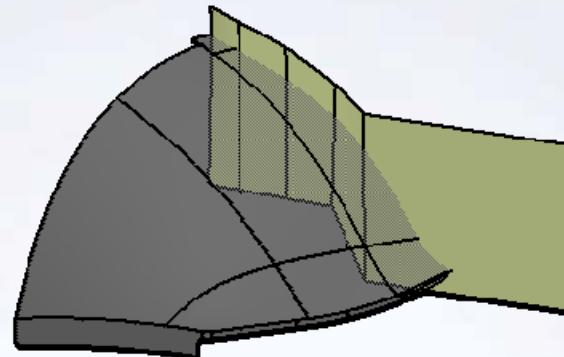
- Create the flange surface
- Create the boss surface
- Relimit the surfaces using the main surface



Do it Yourself (1/6)

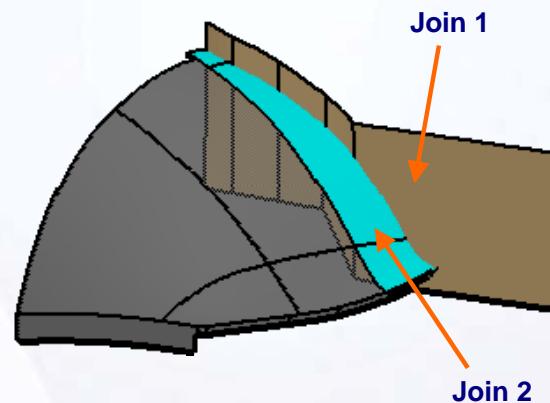
1. Load Ex8F.CATPart.

- Load Ex8F.CATPart. This part already has some curves and surfaces created for you.



2. Create construction elements for a flange surface

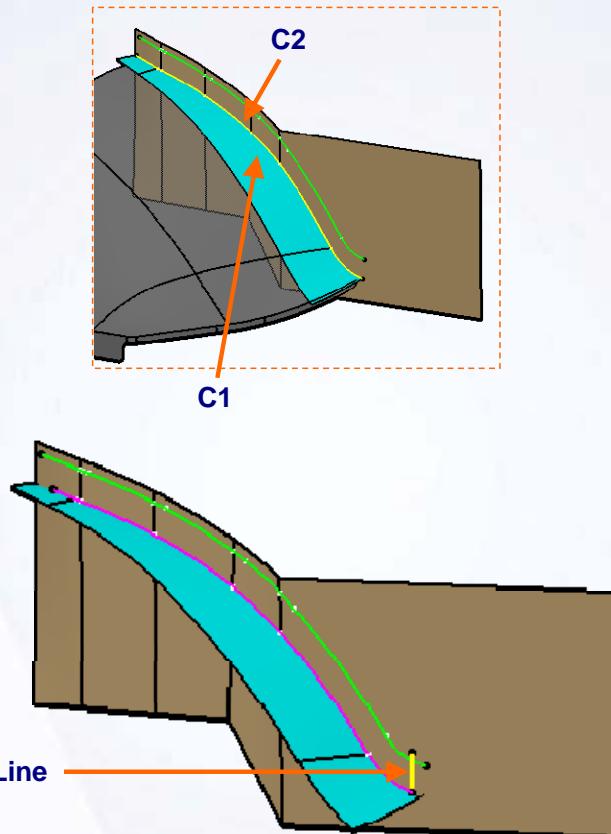
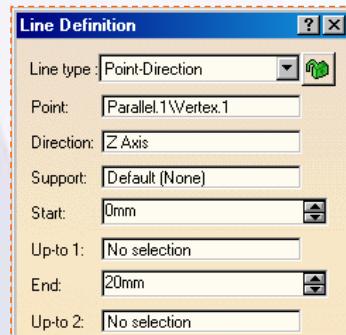
- Define in work object the 'Reference Elements' Geometrical Set.
- Join all the surfaces available in the 'WING_SURFACE' Geometrical Set. Specify the Merging distance as 0.04mm
- Join the surface 8,9 and 10 available in the 'WHEEL_ARC_STUDY_AREA' Geometrical Set. Specify Merging distance as 0.04mm



Do it Yourself (2/6)

3. Create a flange surface

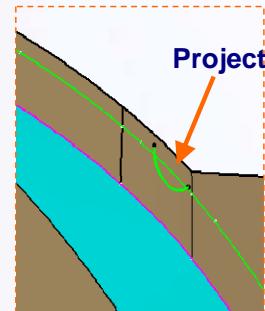
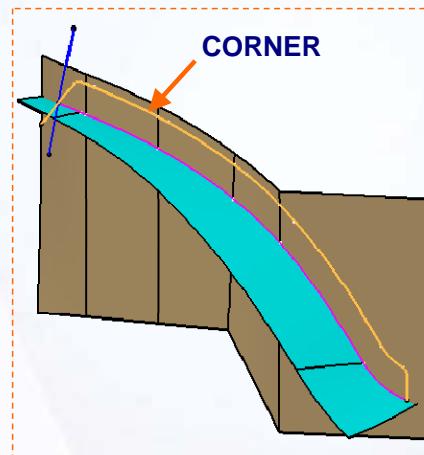
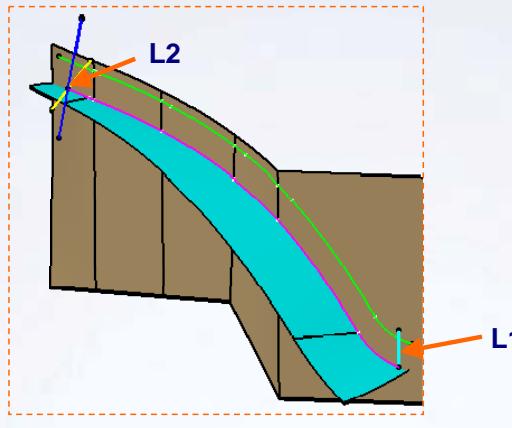
- Create a Geometrical Set called 'Flange_Surface' under the father node of the WHEEL_ARC.H.
- Create the intersection of the two join surfaces created in the previous step.
- Rename the intersection as C1
- Create a parallel curve from C1 at a distance of 15mm using the first join surface as the support element.
- Rename the parallel curve as C2
- Split C2 between two planes 'Plane X=-310' and 'Plane 2'.
- Rename the Split as C3
- Create a vertical line of length 20mm from one extremity of 'C3'.



Do it Yourself (3/6)

3. Create a flange surface (continued)

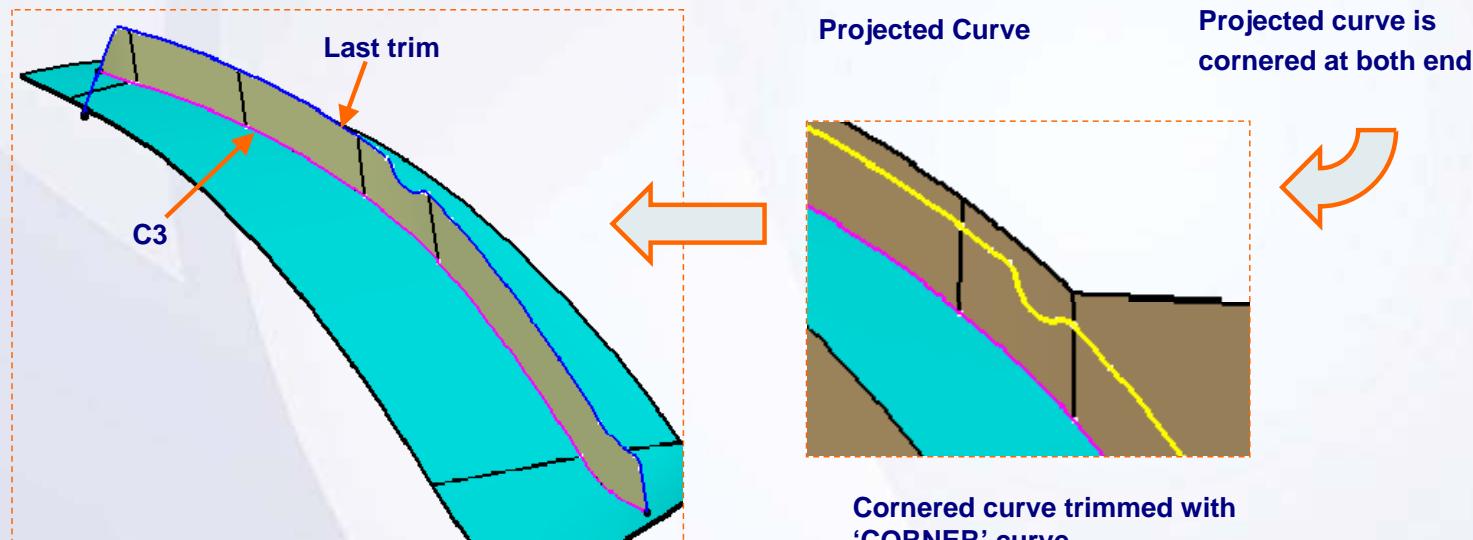
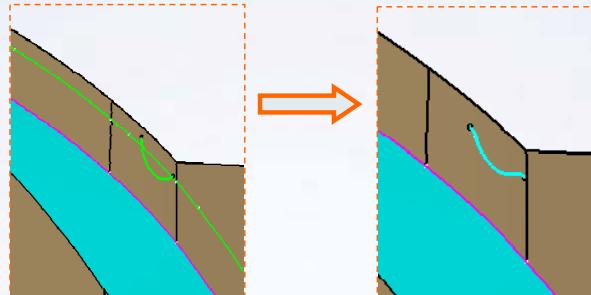
- Create a line normal to surface 'Join.2' from the other extremity of 'C3' (of default length, later used for reference only)
- Create an angle line from the same point with reference to the previous line at 30deg angle on the XZ plane
- Project the vertical line and the angle line on the Join.1 surface
- Rename these projection curves L1 and L2 respectively
- Create a Corner curve of radius 4mm between L1 and C2
- Create a Corner curve of 4mm radius between the resulting curve and L2
- Rename the resulting curve as 'CORNER'.
- Project the curve "CONTOUR" on Join.1



Do it Yourself (4/6)

3. Create a flange surface (continued)

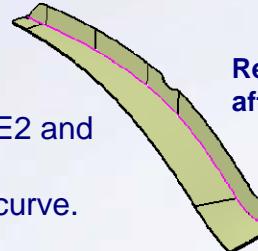
- Create a Corner of radius 4mm between this projection and the offset curve C2. Do not trim the C2 curve while creating this corner. Repeat the corner operation twice to get the corner on both the sides of the projected curve.
- Hide the C2 curve.
- Trim the previous corner with 'CORNER' curve.
- Split 'Join.1' using the last trim and 'C3'.



Do it Yourself (5/6)

4. Relimit the surfaces

- Trim Join.2 and the last created split
- Split the resultant surface between PLANE2 and Surface.7
- Trim the previous corner with 'CORNER' curve.
- Split 'Join.1' using the last trim and 'C3'.

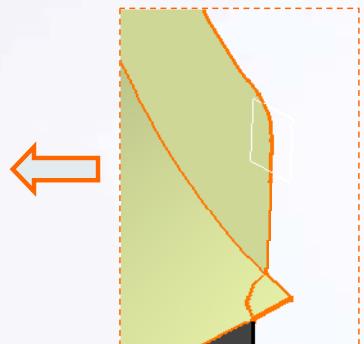
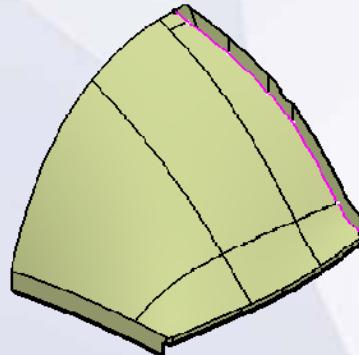


Relimited surface
after split operation

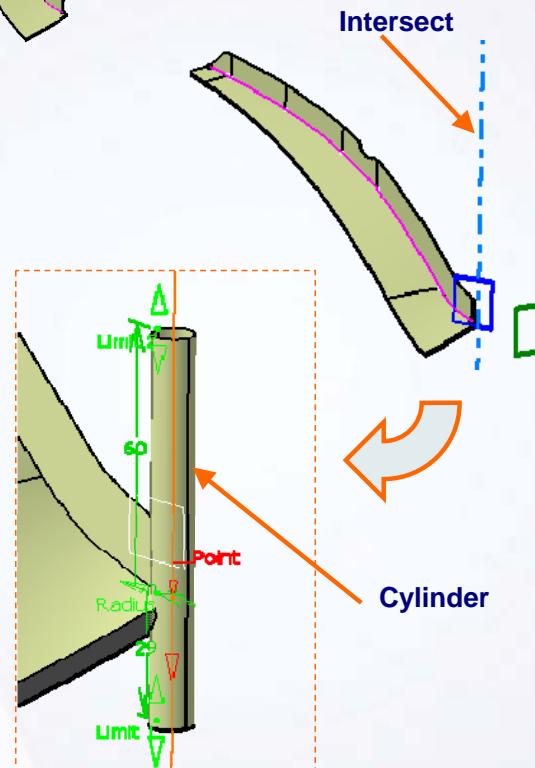
5. Create a Blend cut

- Intersect 'PLANE X=-314' and 'PLANE.3'.
- With intersection as an axis, create a cylinder (default length) and a radius of 4mm.
- Split the main surface with this cylinder
- Join this split with the remaining surfaces of 'WHEEL_ARC_STUDY_AREA'.

Specify the Merging distance as 0.04mm



Cylinder split with main
surface

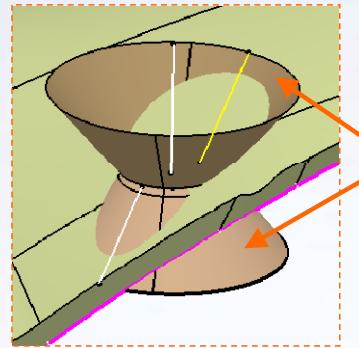
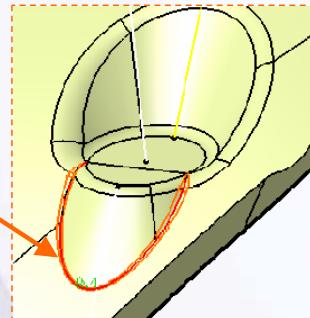


Do it Yourself (6/6)

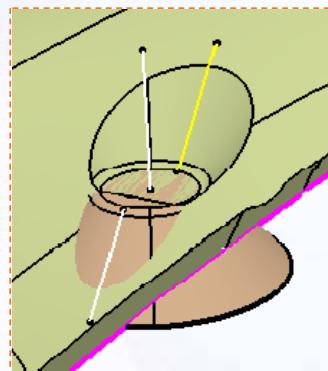
6. Create a revolved shape of the boss

- Create a Geometrical Set called 'Bosses' under the father node of the WHEEL_ARC.
- Create revolution surfaces using 'L3 stamp axis' as axis and the 2 lines 'L1 40° top' and 'L2 40° down' around a full circle.
- Create a 5 mm shape fillet using the two revolution surfaces and the blue surface called 'Boss surface'.
- Create a trim between the upper fillet surface and the main join.
- Create a fillet of 10mm on the upper edge of the surface.
- Similarly, create a trim between lower filleted surface and the main surface.
- Create a fillet of 4mm on the edge that is formed (shown).

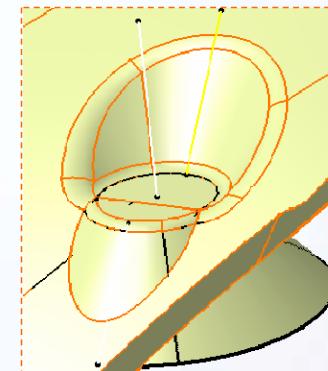
Apply 4mm fillet on this edge



Revolved surface filleted with 'BOSS SURFACE'



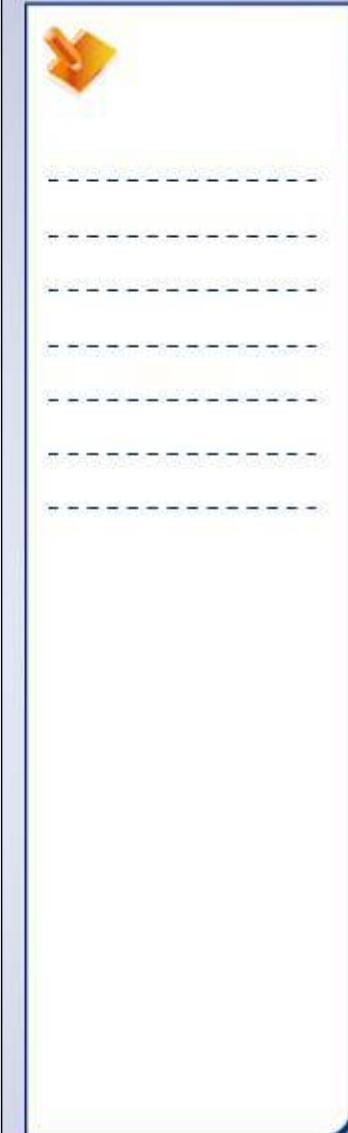
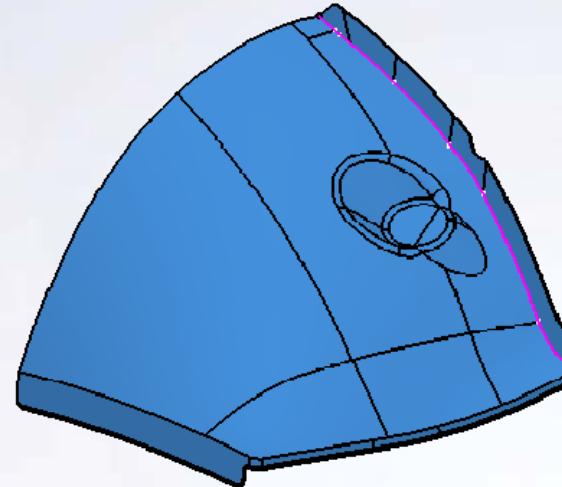
Upper revolve surface trimmed with main surface



Fillet the upper edge of the trimmed cutout

Exercise Recap: Boss and Flange Creation

- ✓ Create the flange surface
- ✓ Create the boss surface
- ✓ Relimit the surfaces using the main surface



Case Study: Create Complex Surfaces

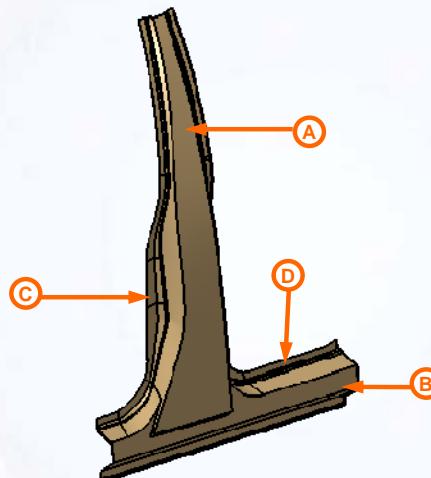
Recap Exercise



30 min

In this exercise, you will create a case study model. Let us recall the design intent of this model:

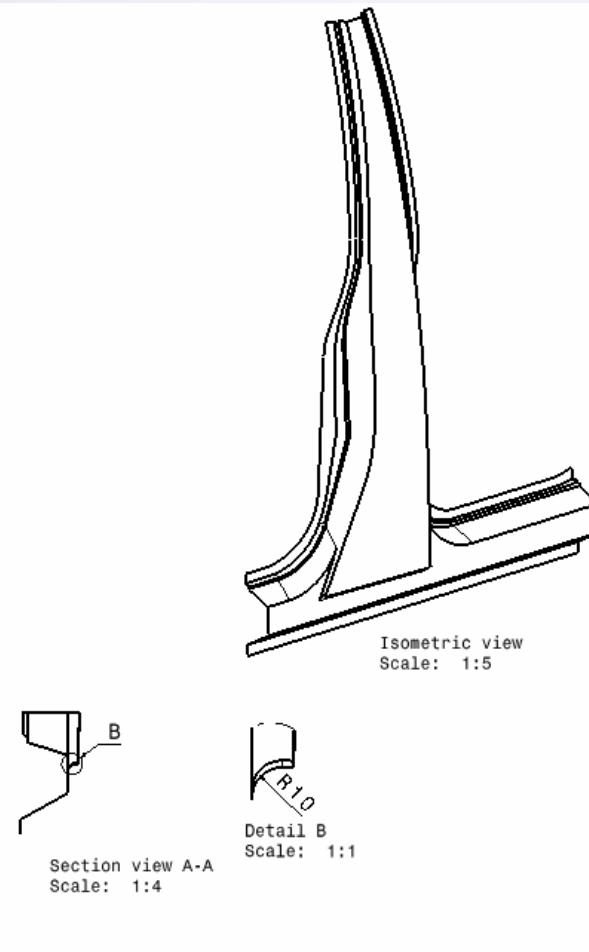
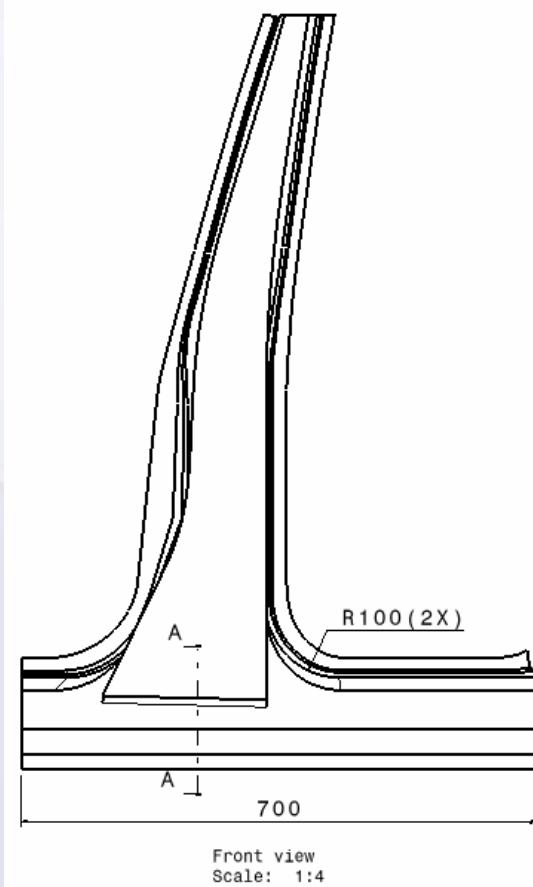
- ✓ The main junction surface (A) must pass through four section curves.
- ✓ The length of the base junction surface (B) must be 700mm.
- ✓ Sharp edges must not be present between the main junction surface and base junction surface.
- ✓ The front pillar surface (C) and the back pillar surface (D) must pass through the respective guide curves.
- ✓ The result must be a single surface



Using the techniques you have learned so far, create the model without detailed instructions.

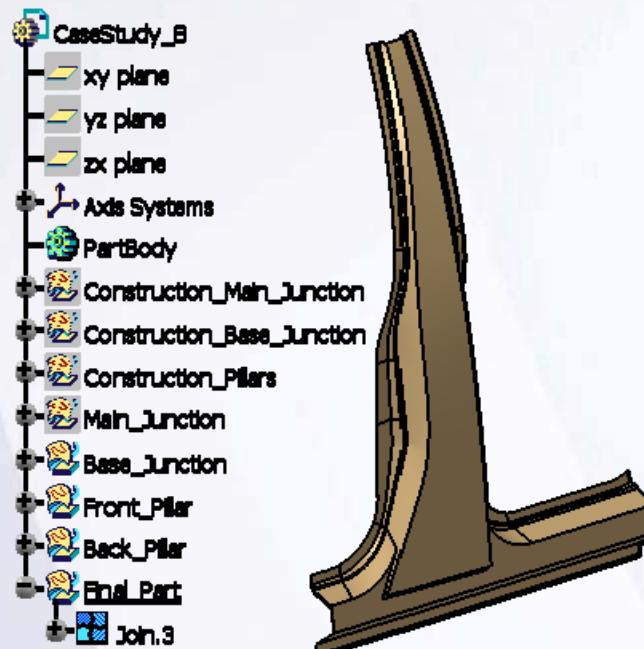
Do It Yourself: Create the Simplified B Pillar

Load Start_CaseStudy8.CATPart and modify the model using the information below.



Case Study Recap: Simplified B Pillar

- ✓ Create a Multi-Section surface.
- ✓ Fill the surface and create a join.
- ✓ Create an extruded surface.
- ✓ Apply fillets to sharp edges.
- ✓ Create a sweep.
- ✓ Join the surfaces to get a single result.



Surface Modification Tools

9

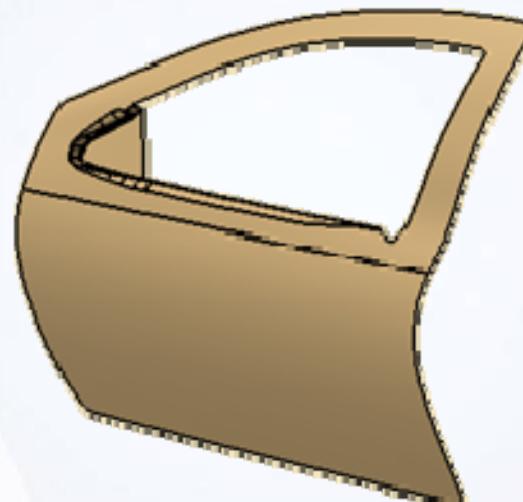
Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Transform Surfaces
- ✓ Analysis and Correction



4 Hours



Case Study

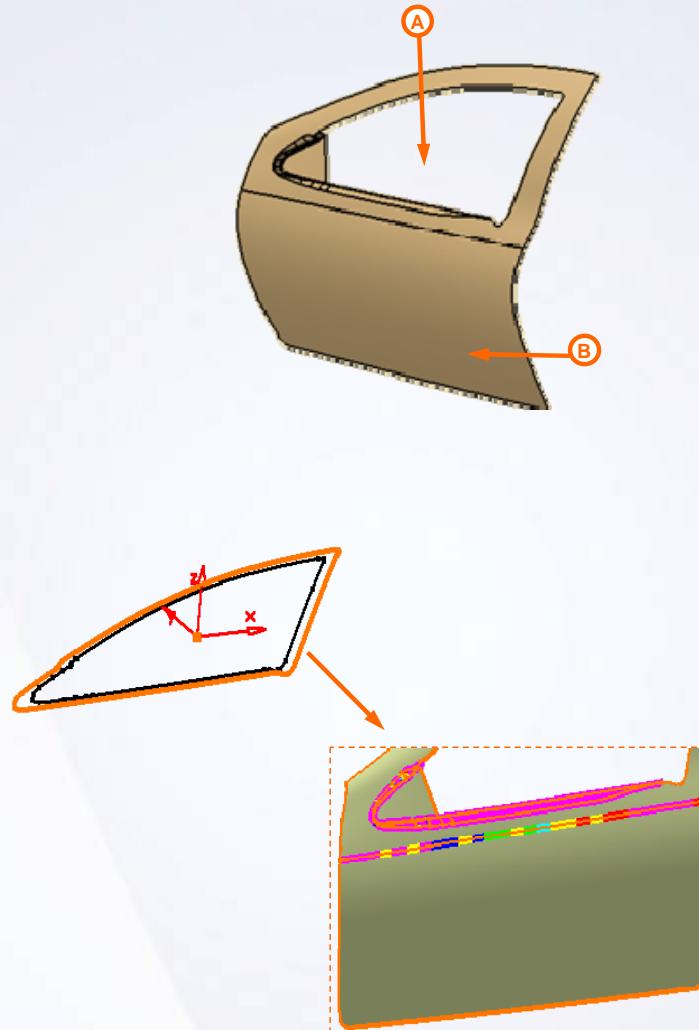
The case study for this lesson is a simplified outer door used in the door assembly.

Design Intent

- ✓ Window opening (A) of the inner door must be reduced by 10% in X and Z direction.
 - Affine the window opening curve using the centre of gravity as the origin.
- ✓ The lower door surface (B) must be curvature continuous.
 - Smooth the parent curves of lower door surface.
- ✓ Simplify the model structure
 - Delete the redundant elements in the model.

Stages in the Process

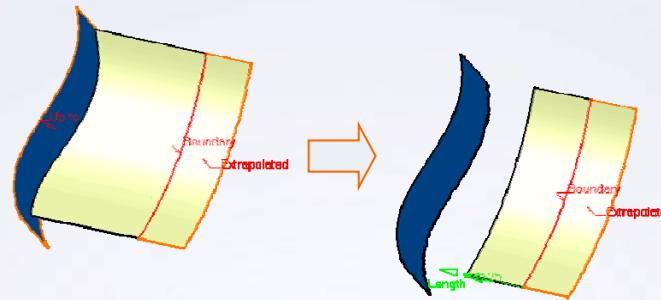
1. Affine a window curve using the centre of gravity as the origin.
2. Replace the window curve by the new affined curve.
3. Check the surface for curvature continuity.
4. Smoothen the section curves of lower door surface.
5. Replace the section curves of the lower door surface with new smooth curves.
6. Delete unused elements.



Transform Surfaces

Transformation operations such as scaling and affinity, help to resize the part. Transformation operations such as translate and rotate, help to change the positioning of the part in the co-ordinate axis system.

- ✓ Extrapolating Elements: Extrapolates help to extend the curve or surface.
- ✓ Transformations: Transformations are used to modify the size, location, and orientation of a wireframe or surface element.
- ✓ Restoring Elements: Restores the limits of the surface or a curve which has been split once or several times.

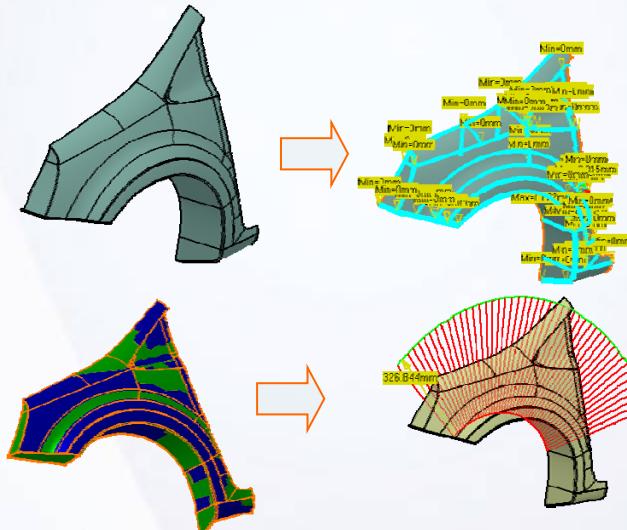


Analysis and Correction

In this lesson you will learn how to analyze curve connection, surface connections and to achieve the required continuities.

In this lesson you will learn following operations:

- ✓ Curve Connect Checker
- ✓ Surface Connect checker
- ✓ Curve Smoothing
- ✓ Surface Quality
- ✓ Deleting the Redundant Elements



Main Tools

Operations

- 1 Healing:** correct the defects found after performing analysis.
- 2 Curve Smoothing:** correct the discontinuities in a curve.
- 3 Untrim:** restore the original surface from a trimmed surface.

Transformations

- 4 Translate:** moves selected elements in a given direction and distance.
- 5 Rotate:** moves selected elements around an axis.
- 6 Symmetry:** creates mirror image of a selected elements about a plane.
- 7 Scaling:** resizes a selected element with respect to a point or a plane.
- 8 Affinity:** scales a selected element with respect to given direction.
- 9 Axis to Axis:** duplicates and moves selected geometry from one axis system to other.



Exercise: Update Errors Management

Recap Exercise

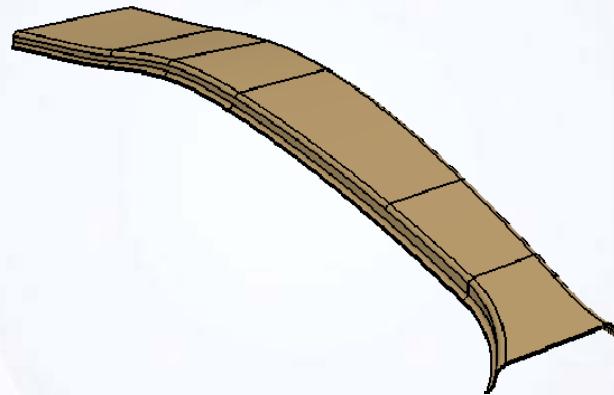


30 min

In this exercise you will practice update error management on B-pillar. High-level instructions for this exercise are provided.

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

- Modify the input of the B-pillar and manage the errors during the update.
- Create necessary corrections.



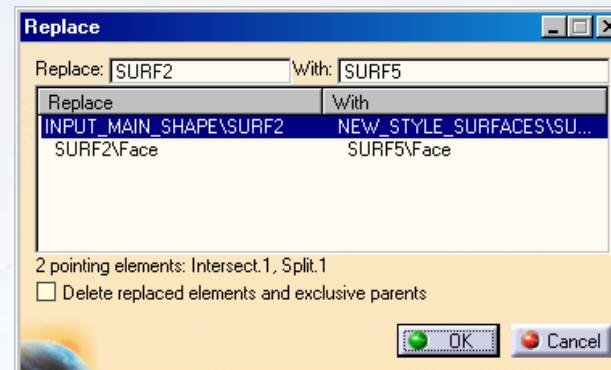
Do it Yourself (1/6)

1. Load Ex9A.CATPart.

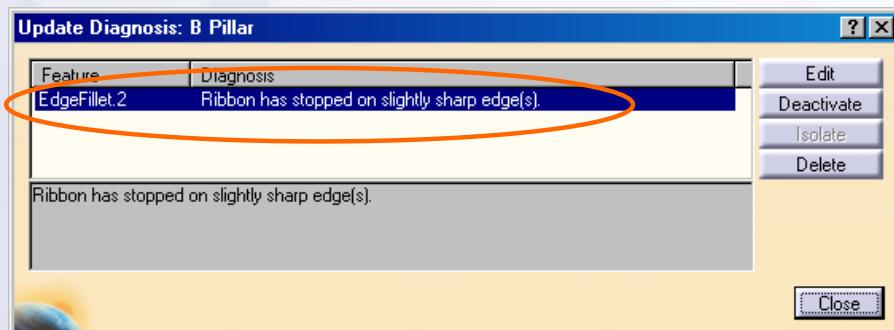
- Load Ex9A.CATPart. This part already has some curves and surfaces created for you.

2. Replace the cutting surface in the completed model

- Replace 'SURF2' available in the geometrical set 'INPUT_MAIN_SHAPE' by 'SURF5' available in the geometrical set 'NEW_STYLE_SURFACES'.
- During the update of the part, you get this error message



Replace SURF2 with SURF5



Do it Yourself (2/6)

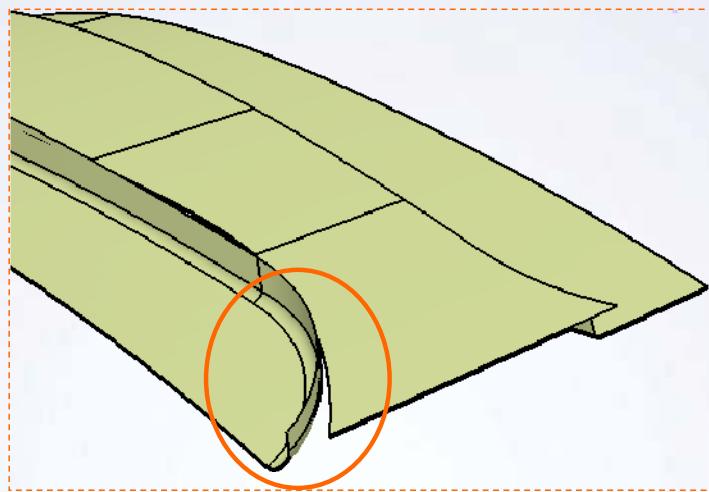
3. Analyze and find the problem area in the model

- Hide 'MAIN_SHAPE' feature available in the 'Main_Shape' geometrical set.
- Show 'EdgeFillet.1' to visualize the feature.
- Hide 'SURF2' and 'SURF5' surfaces for better visualization of the problem area.

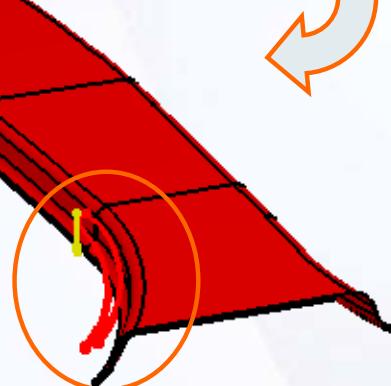
Observation:

You can see that the problem comes from an interruption in the filleted edge.

So we have to scan the parents of EdgeFillet.1 to find the source of the problem.



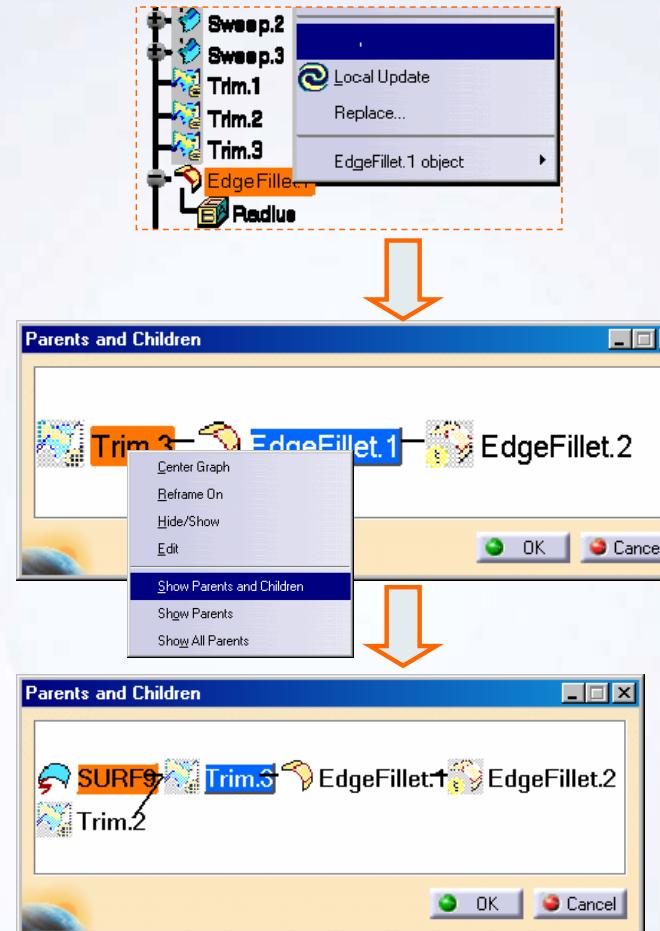
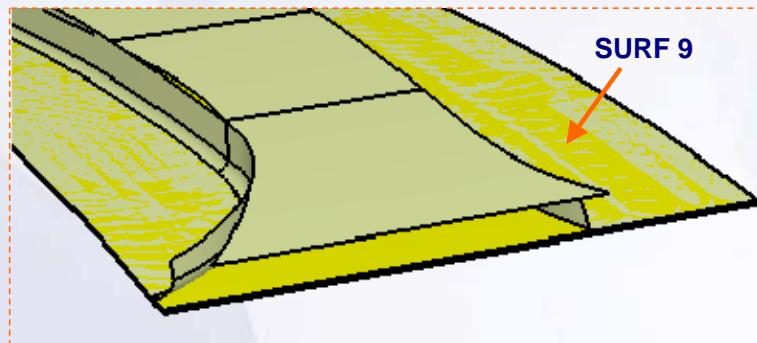
"Ribbon has stopped on slightly sharp edge"



Do it Yourself (3/6)

4. Analyze the Parents/Children relations of the feature

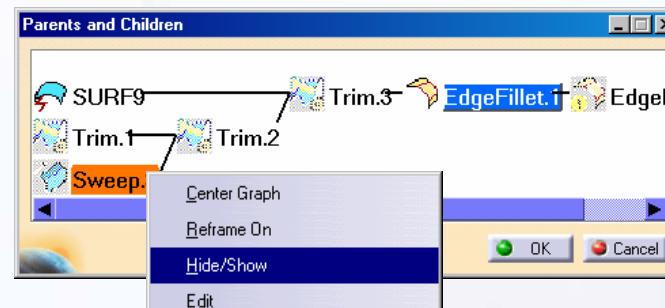
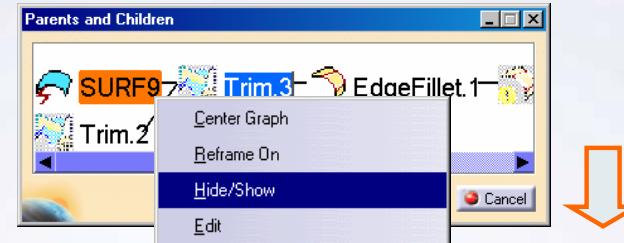
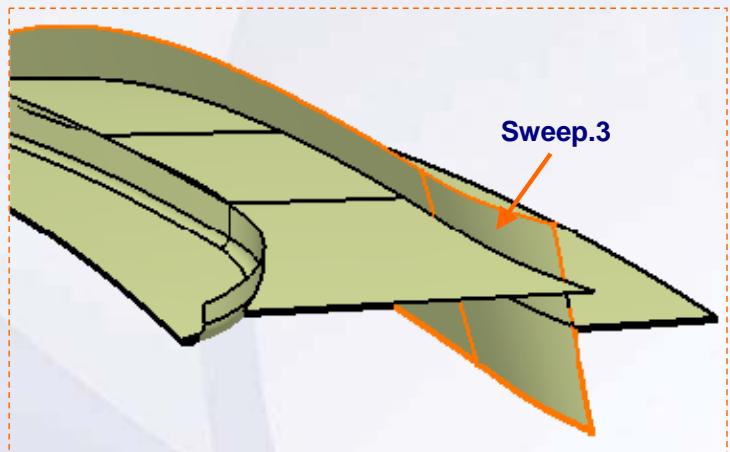
- Find the parents of Edge fillet.1 through Parents/Children dialog box. Show Trim.3. You would not find any problem in Trim.3 surface affecting the edge fillet.
- Find the parents of Trim.3 through the Parents/Children dialog box. Show SURF.9. You will see that SURF9 cannot be the source of this corrupted edge.



Do it Yourself (4/6)

4. Analyze the Parent/Children relations of the feature (continued)

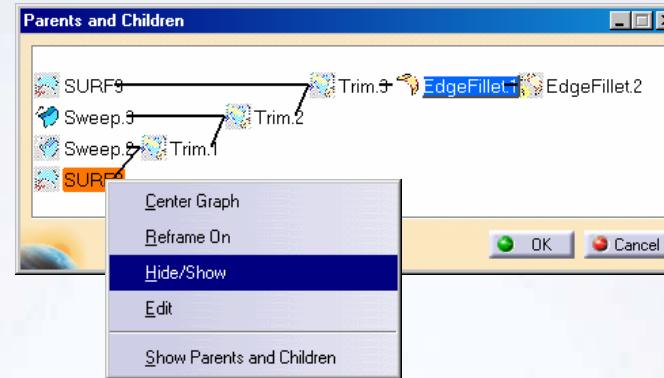
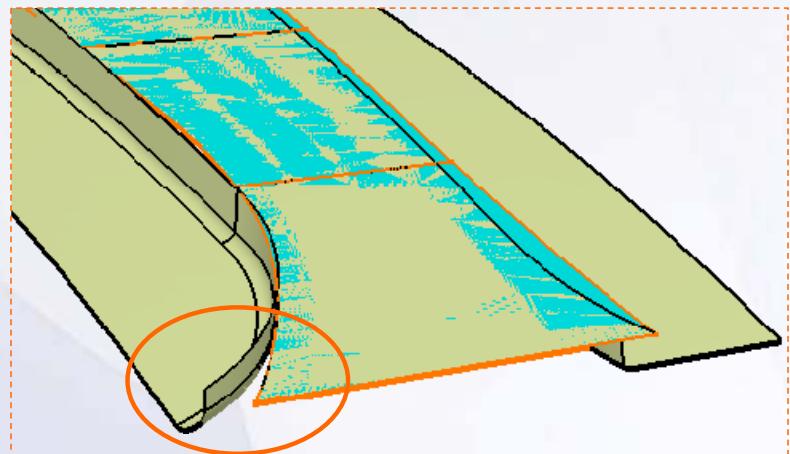
- Hide SURF9 and look for the parents of 'Trim2'. Show 'Sweep.3' and see that it is not the source of the corrupted edge.



Do it Yourself (5/6)

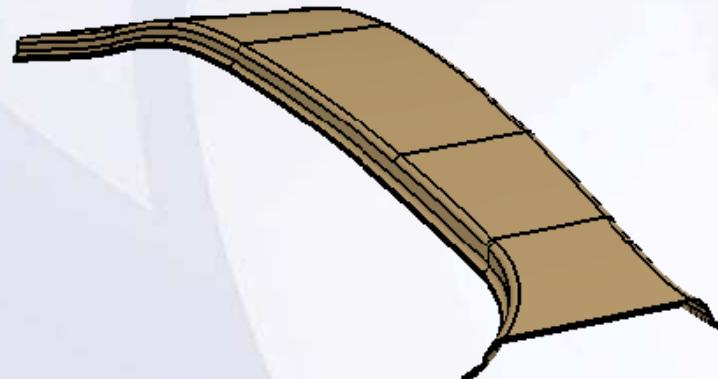
4. Analyze the Parent/Children relations of the feature (continued)

- Hide 'Sweep.3' and look for the parents of 'Trim.1'.
- Show SURF8, it is the source of the problem, this surface is not large enough to create a good intersection, it needs to be extrapolated.

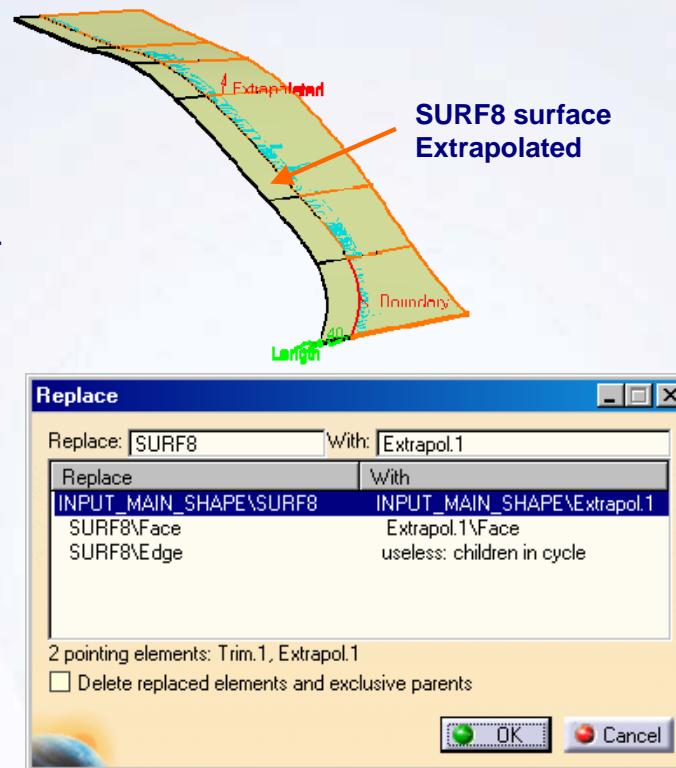


Do it Yourself (6/6)

5. Extend the SURF8 to solve the corrupt edge problem.
- Define in work object the 'INPUT_MAIN_SHAPE' geometrical set.
 - In this geometrical set, extrapolate the surface SURF8.
 - Replace SURF8 with a new extrapolated surface.
 - Update the part. The part now gets updated correctly.
 - Hide the intermediate surfaces and show the MAIN_SHAPE



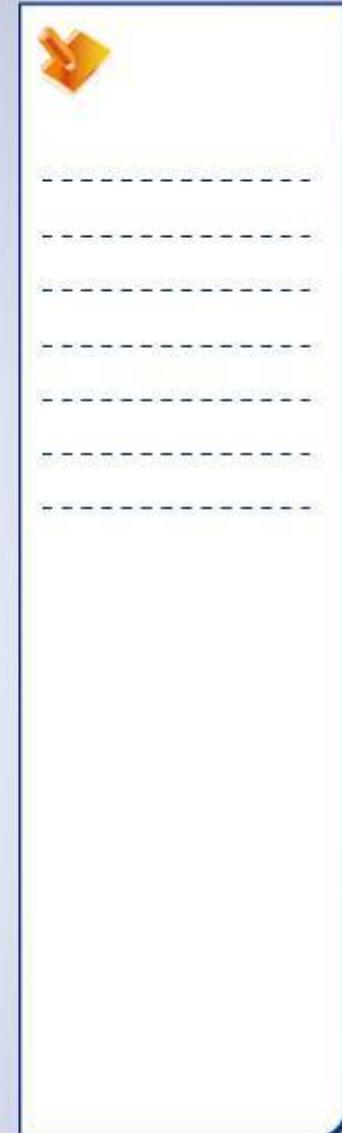
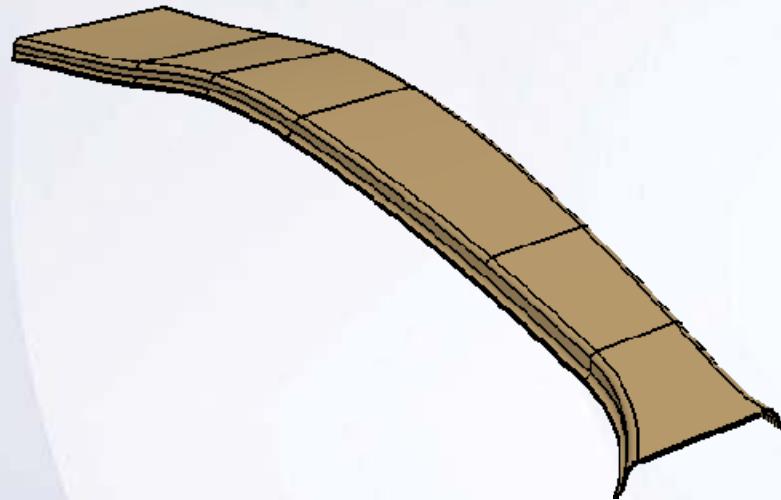
Surface gets updated with a new styled surface



Replace SURF8 with a new Extrapolated surface

Exercise Recap: Update Errors Management

- ✓ Modify the input of the B-pillar and manage the errors during the update.
- ✓ Create necessary corrections.



Exercise: Surface Symmetry

Recap Exercise

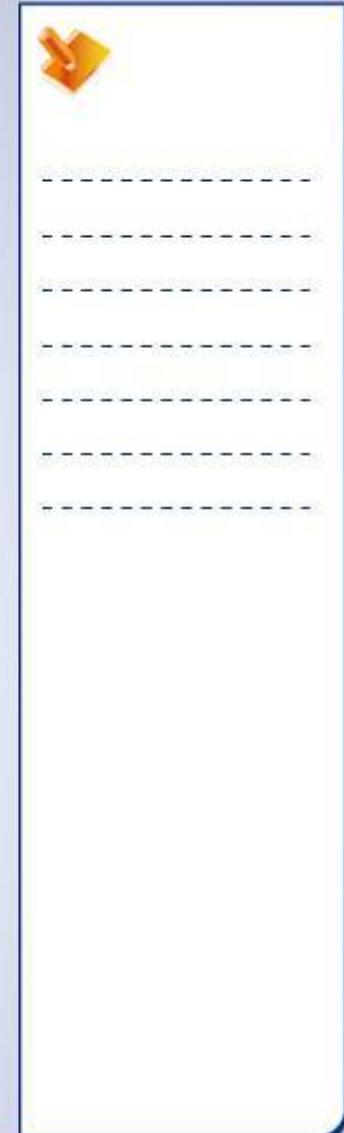
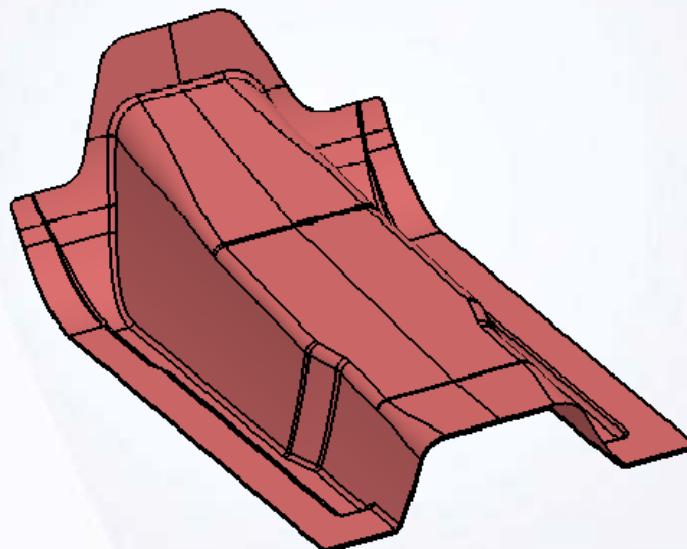


15 min

**In this exercise you will finalize the tunnel model using the tools learnt in this step.
High-level instructions for this exercise are provided.**

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

- Create a Symmetry of the surfaces



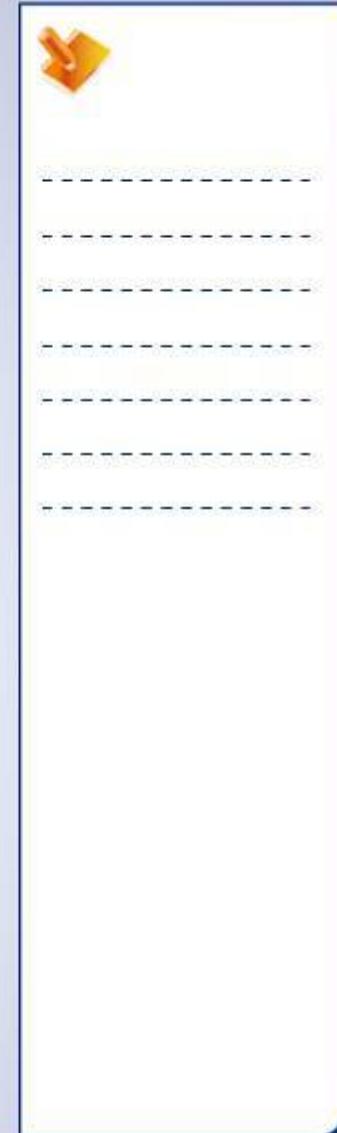
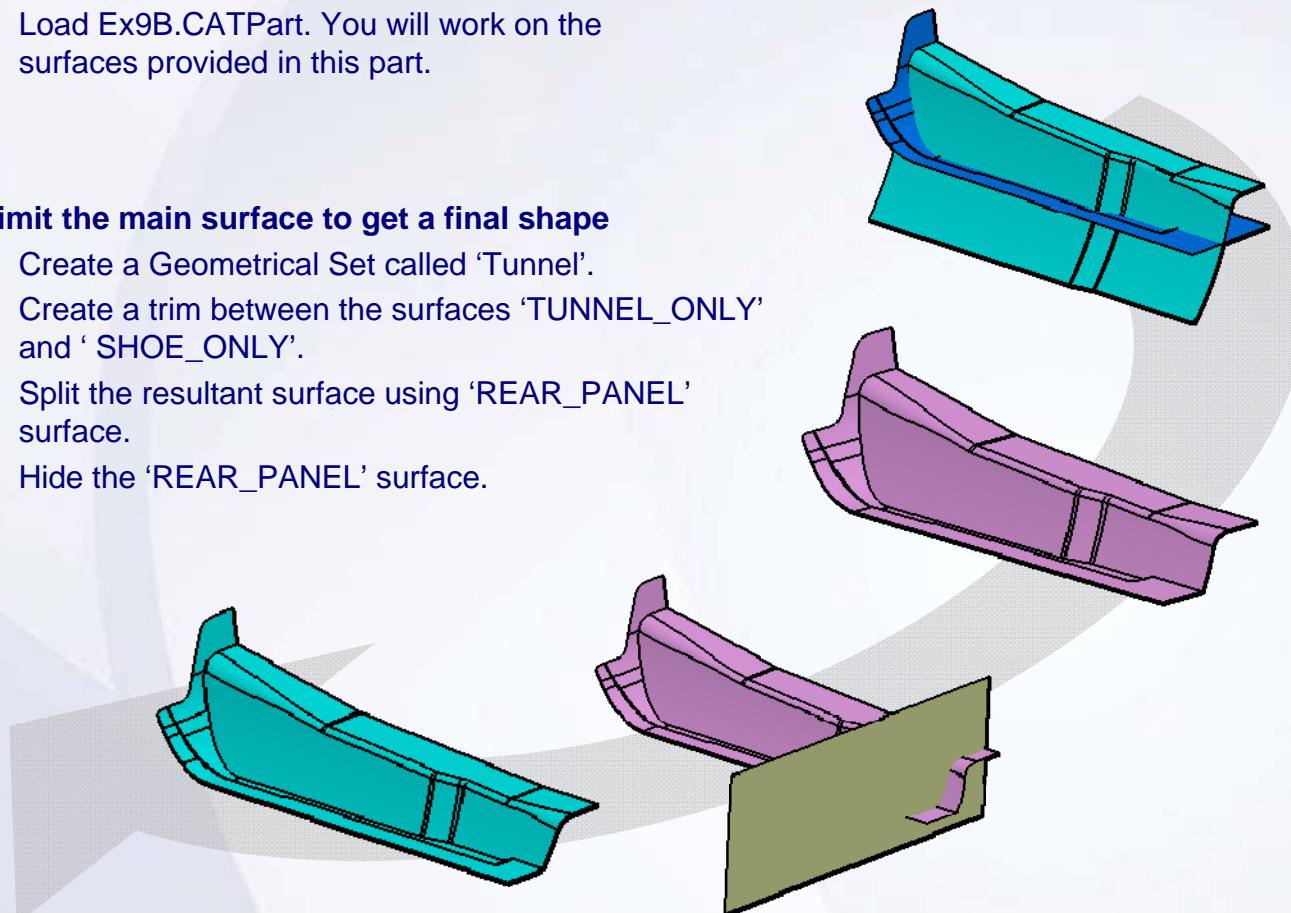
Do it Yourself (1/2)

1. Load Ex9B.CATPart.

- Load Ex9B.CATPart. You will work on the surfaces provided in this part.

2. Relimit the main surface to get a final shape

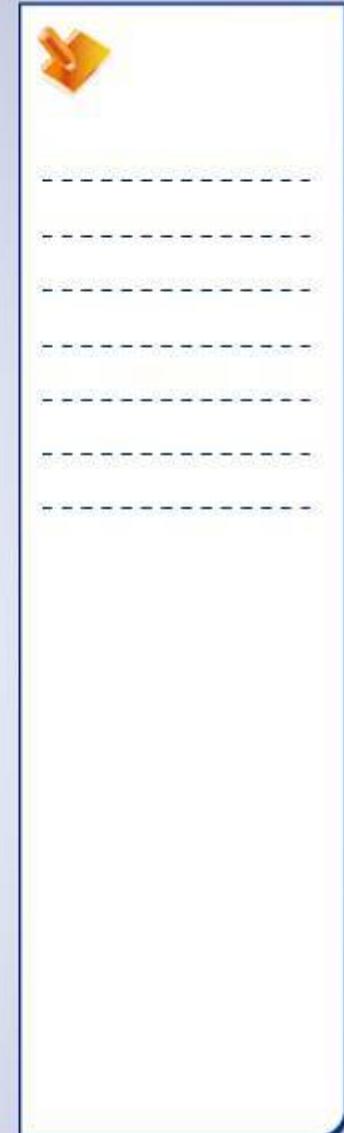
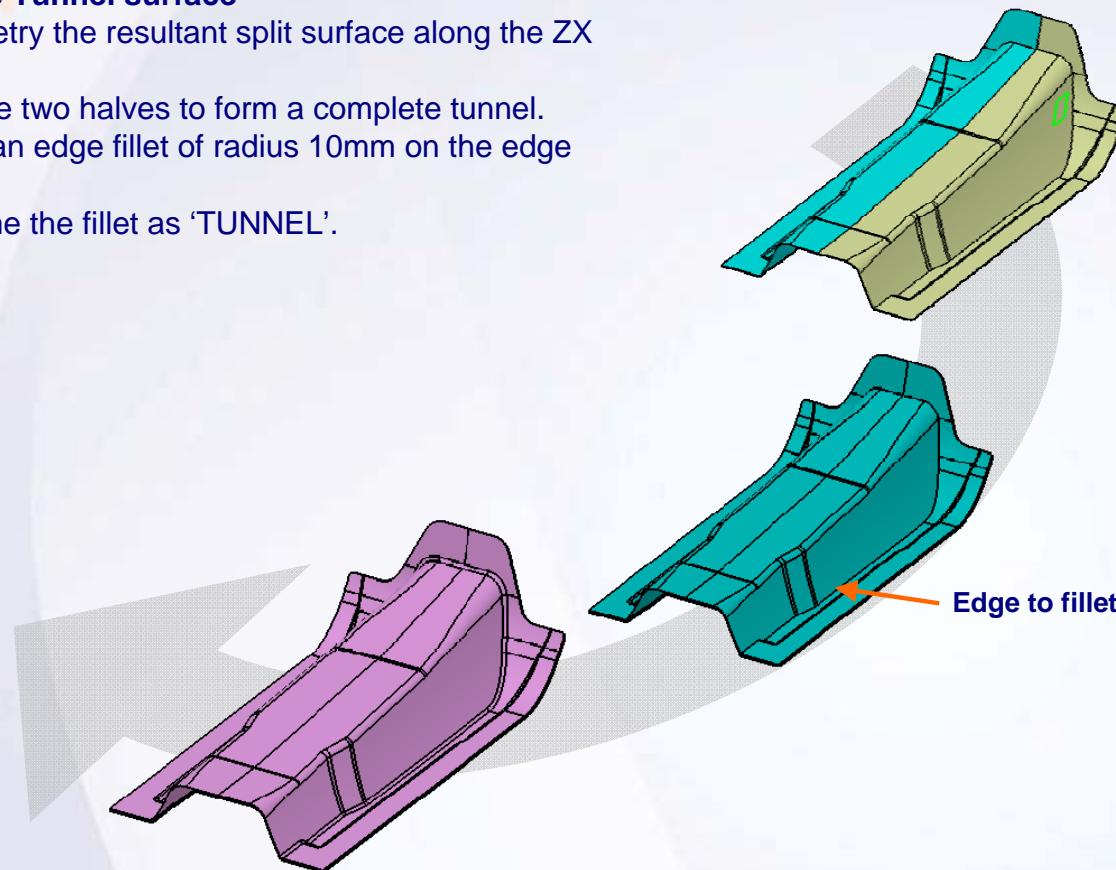
- Create a Geometrical Set called 'Tunnel'.
- Create a trim between the surfaces 'TUNNEL_ONLY' and 'SHOE_ONLY'.
- Split the resultant surface using 'REAR_PANEL' surface.
- Hide the 'REAR_PANEL' surface.



Do it Yourself (2/2)

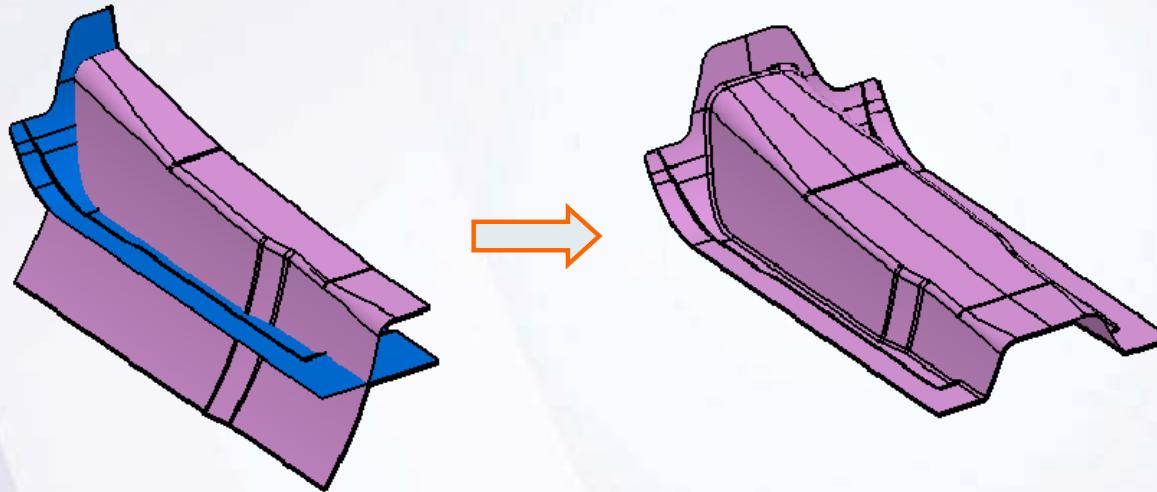
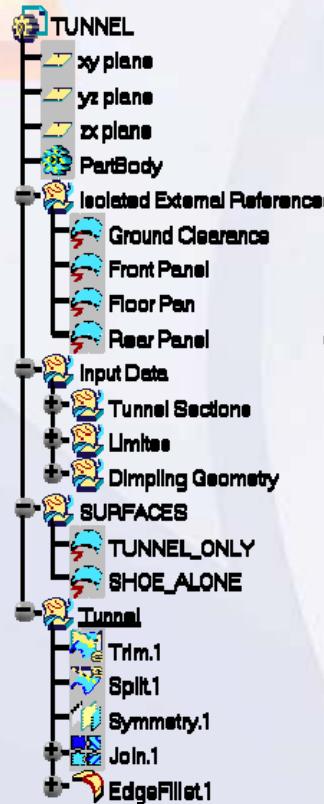
3. Finalize the Tunnel surface

- Symmetry the resultant split surface along the ZX plane.
- Join the two halves to form a complete tunnel.
- Apply an edge fillet of radius 10mm on the edge shown.
- Rename the fillet as 'TUNNEL'.



Exercise Recap: Surface Symmetry

- ✓ Create a Symmetry of the surfaces



Exercise: Surface Analysis

Recap Exercise

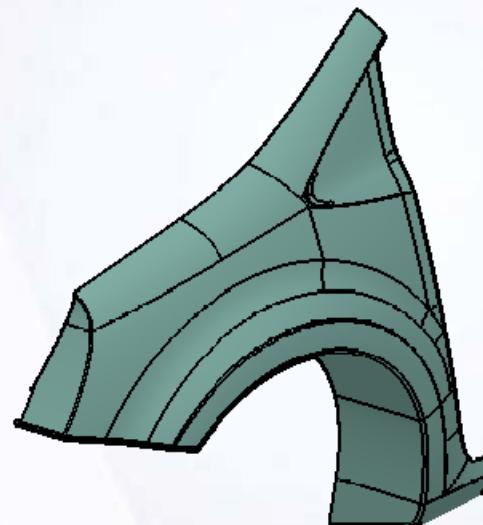


15 min

In this exercise you will perform surface analysis on the front wing surface using the analysis tools learnt in the previous steps. High-level instructions for this exercise are provided.

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

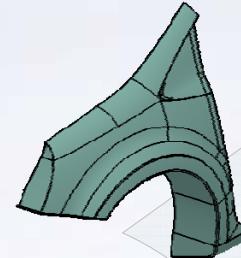
- Analyze surface connections
- Heal the discontinuities of the surfaces
- Analyze the radius and curvature on the surface



Do it Yourself (1/4)

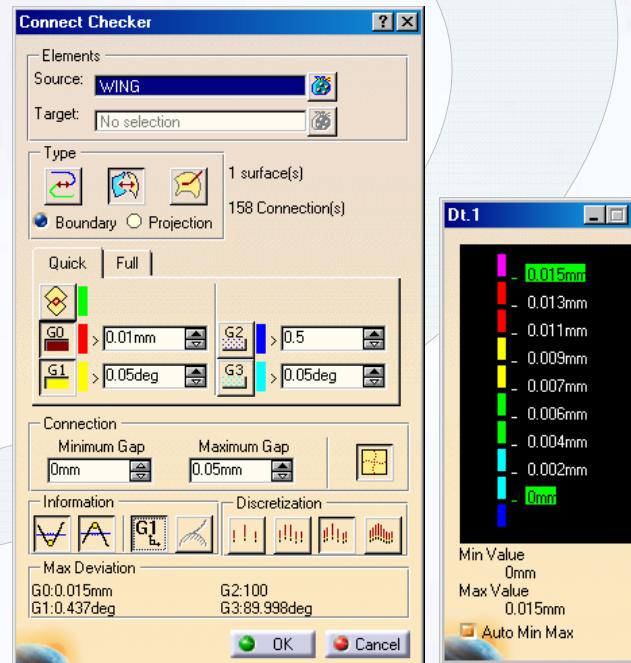
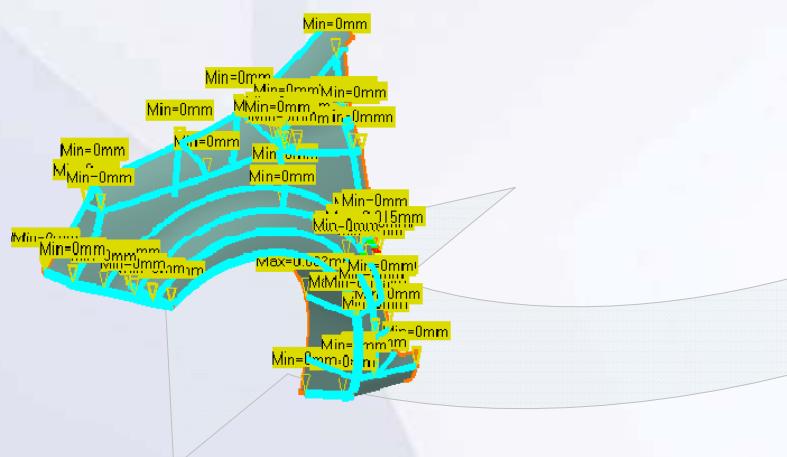
1. Load Ex9C.CATPart.

- Load Ex9C.CATPart. You will work on the surfaces provided in this part.



2. Perform the Connect Checker analysis

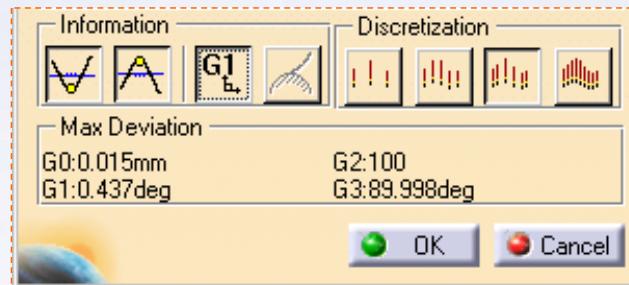
- Select the surface WING and click the connect checker icon.
- Select the **Quick** analysis mode .
- Choose the **G0** mode.
- Study the gaps and severity on the surface.



Do it Yourself (2/4)

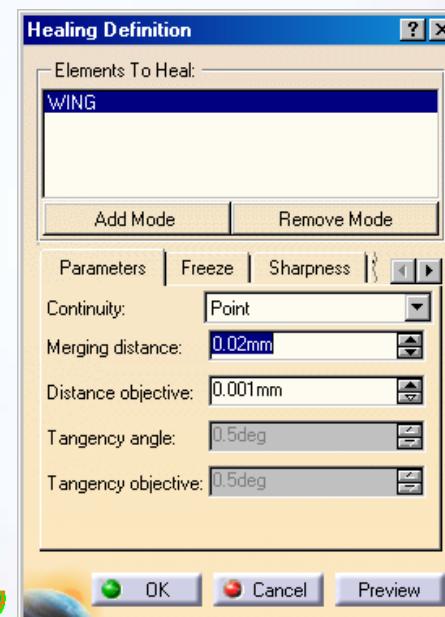
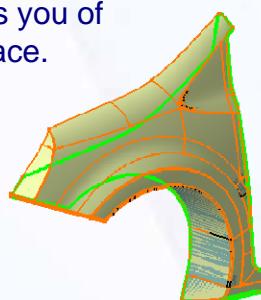
2. Perform the Connect Checker analysis (continued)

- Activate the Tangency analysis
- Notice the maximum discontinuities values
 - a. 0.015mm for distance
 - b. 0.437deg for tangency



3. Heal the surface

- In the Healing Definition dialog box, specify the following parameters.
 - a. Click the **Heal** icon
 - b. In the dialog box specify the following parameters. Merging Distance: 0.02mm (> maximum distance continuity), Distance Objective: 0.001mm
 - c. Choose Point as Continuity.
 - d. Click the **Preview** button. CATIA informs you of the necessary deviation to heal the surface.
 - e. Click **OK** to heal the surface.

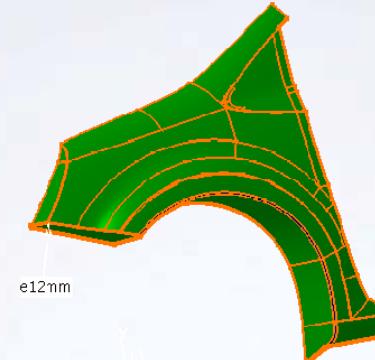


Do it Yourself (3/4)

4. Repeat the Connect Checker analysis on the surface

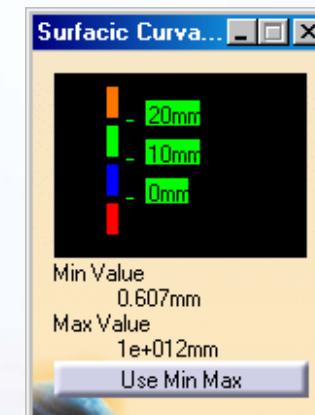
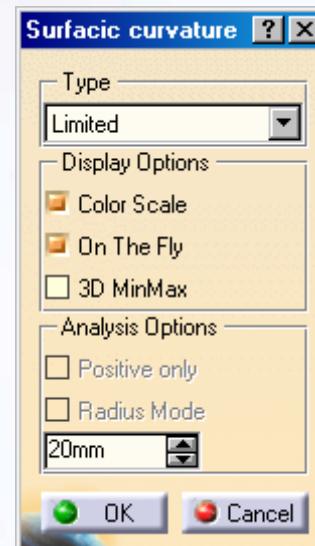
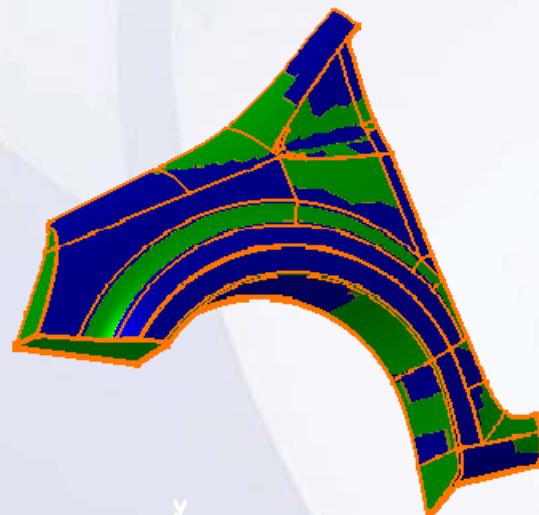
Observation

You will observe that the distance discontinuities on the surface are healed.



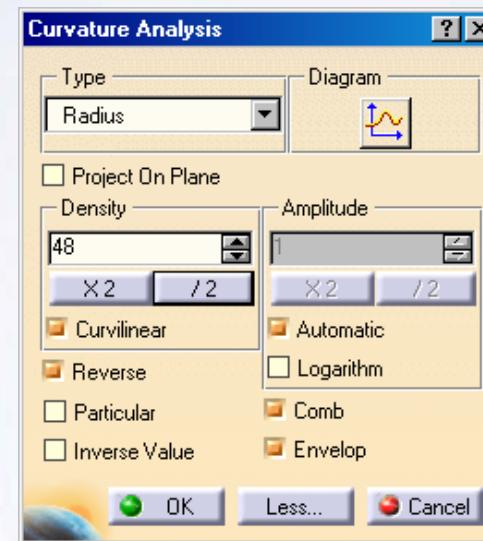
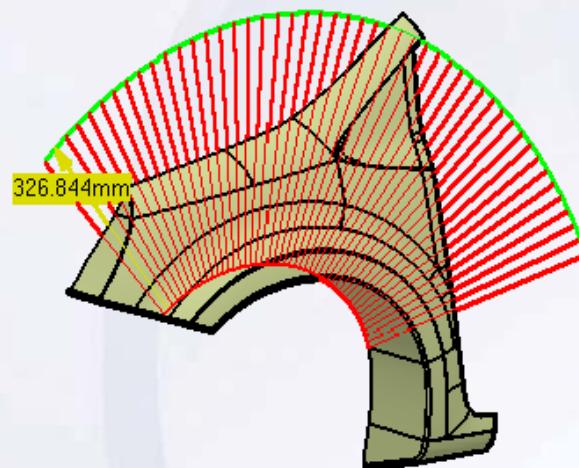
5. Perform Curvature analysis on the surface

- Analyze the Limited radius of the healed surface
- Notice the blue zone for which the radius is less than 10mm



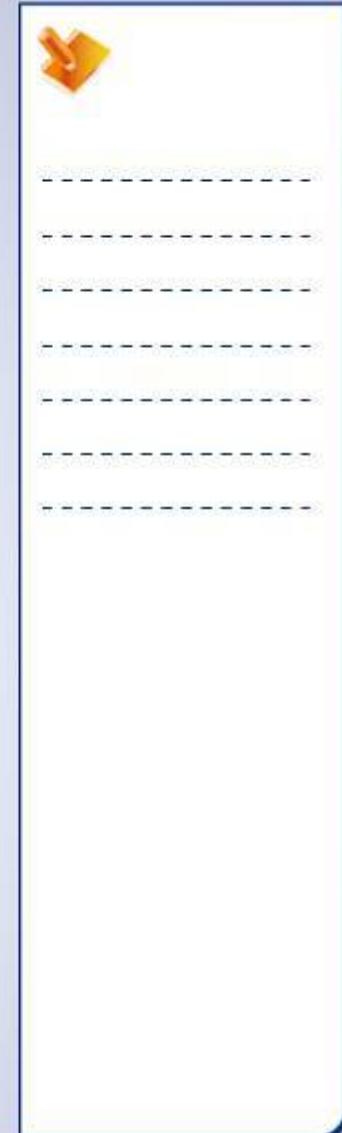
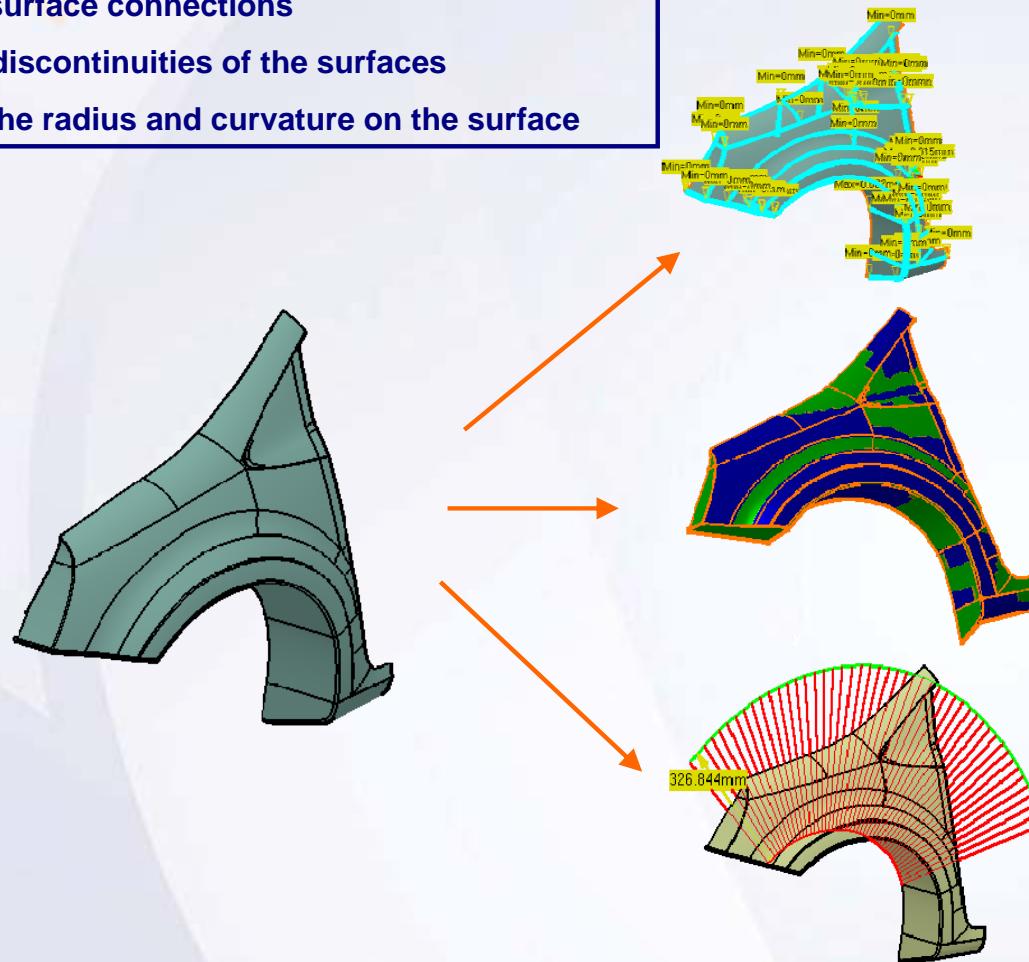
Do it Yourself (4/4)

6. Using the porcupine analysis (as shown), analyze the curvature radius of the wheel arch



Exercise Recap: Surface Analysis

- ✓ Analyze surface connections
- ✓ Heal the discontinuities of the surfaces
- ✓ Analyze the radius and curvature on the surface



Case Study: Surface Modification Tools

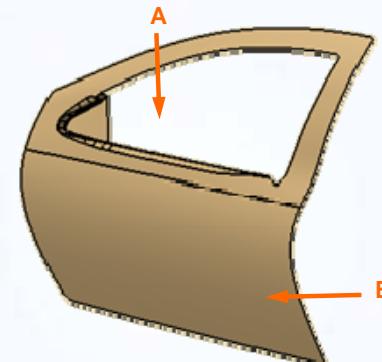
Recap Exercise



10 min

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

- ✓ Window opening (A) of the inner door must be reduced by 10% in X and Z directions.
- ✓ The lower door surface (B) must be curvature continuous.
- ✓ Simplify the model structure.

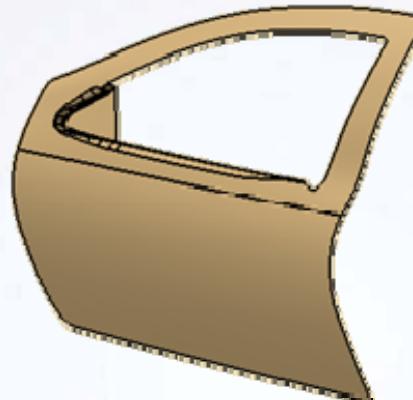


Using the techniques you have acquired in this and previous lessons, create the model without detailed instructions.

Do It Yourself: Create the Simplified Outer Door

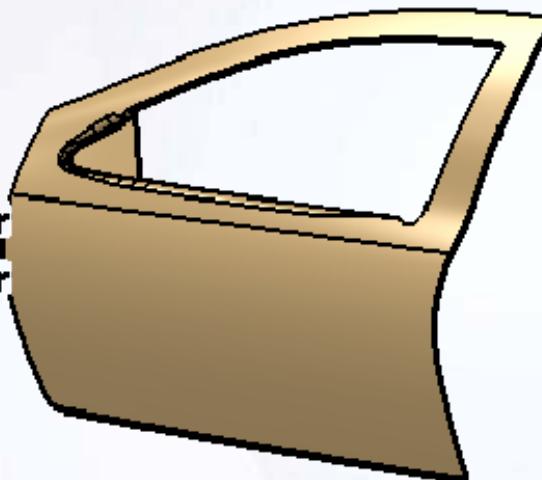
Load Start_CaseStudy9.CATPart and modify the model using the information below.

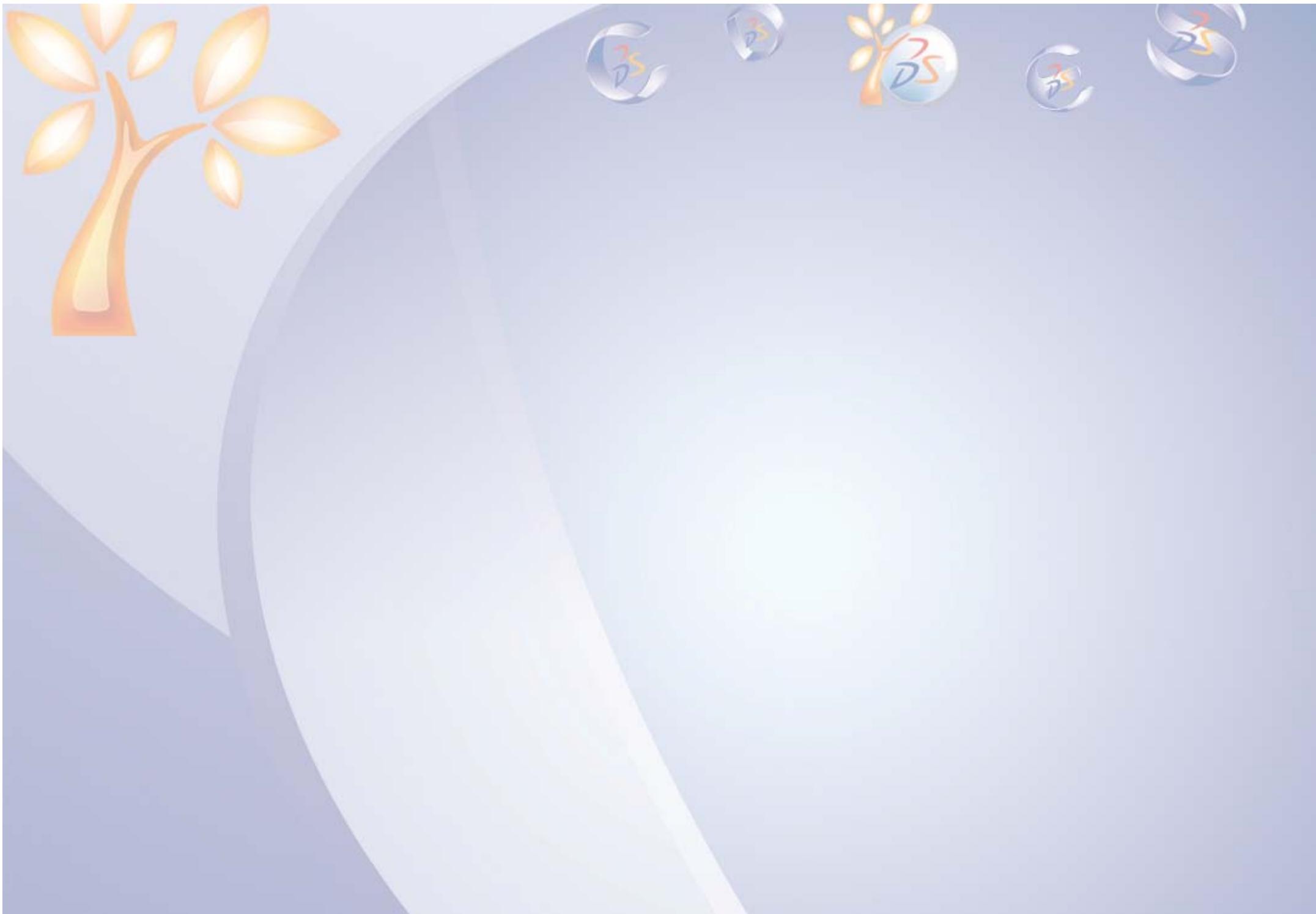
- ✓ Affine the Window_Curve in X and Z direction using Center_Of_Gravity as the origin and use this curve to replace the previous Window_Curve.
- ✓ Perform the curvature analysis to find any discontinuities on the lower door surface.
- ✓ Smoothen the section curves of the lower door surface and use these curves to replace the previous section curves.
- ✓ Delete the useless elements to simplify the model.



Case Study Recap: Simplified Outer Door

- ✓ Window opening of the inner door must be reduced by 10% in X and Z directions.
- ✓ The lower door surface must be curvature continuous.
- ✓ Simplify the model structure.





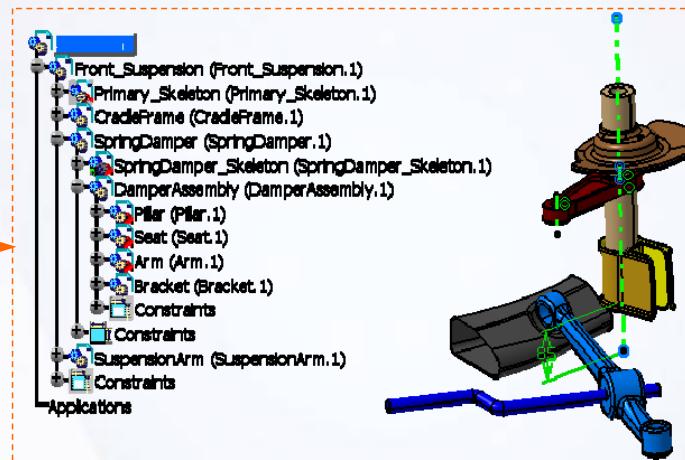
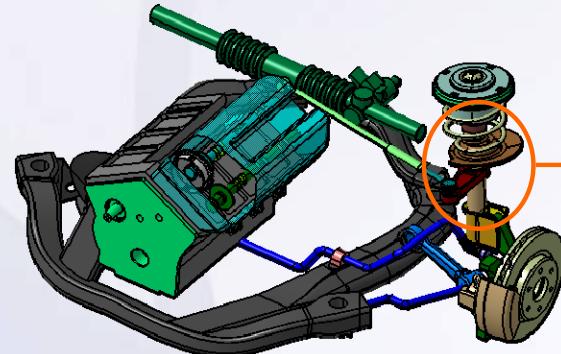
10

Assembly Design

Learning Objectives

Upon completion of this lesson you will able to

- ✓ Create a new CATProduct
- ✓ Assemble the Base component
- ✓ Manipulate the position of the component
- ✓ Assemble and fully constrain the components



4 Hours

Case Study

The case study for this lesson is the Damper Assembly used in the Front Suspension and Engine assembly.

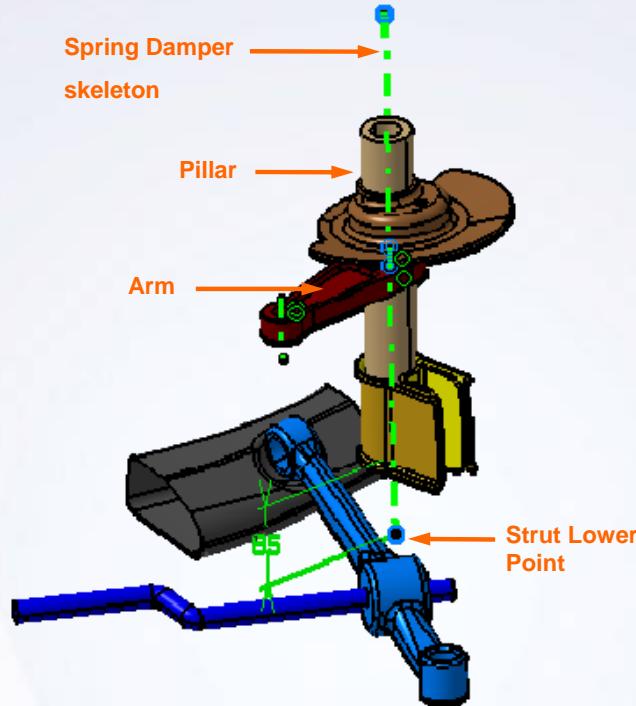
Design Intent

The Damper sub-assembly must be positioned with reference to Spring Damper skeleton.

- Pillar axis must coincide with the axis of Spring Damper Skeleton.
- Pillar lower face must be 85 mm from Strut Lower point of Spring Damper skeleton.
- Arm axis must coincide with the axis of Spring Damper Skeleton.

Stages in the Process

1. Insert Damper Assembly in Spring Damper sub-assembly.
2. Move the Spring Damper sub-assembly to its approximate final position using the compass.
3. Position the Spring Damper sub-assembly by creating constraints with reference to Spring Damper skeleton.



Create a New CATProduct

A product is a assembly document that stores a collection of components (parts or other assemblies). It uses the.CATProduct extension.

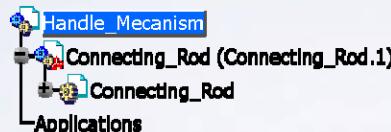
The components used in an assembly can be pre-existing components or components created within the assembly. Like a part, a product contains a specification tree. The tree shows the inserted components, and the constraints used to fix the components. Products are created in the Assembly Design workbench.



Assemble in the Base Component

You can add a component to an assembly in one of the three ways:

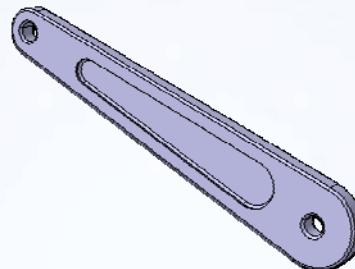
A. Contextual menu: Right-click the assembly that will receive the component and use the contextual menu to insert the component.



B. Product Structure: Tools Select the assembly in the specification tree and use the icons in the Product Structure Tools.

C. Insert menu: Select the assembly in the specification tree and use the Insert menu.

When you add existing parts or assemblies as components, their corresponding files are not copied into the assembly; they are only referenced by the assembly. Once components are inserted into a product you can customize their display and their properties.



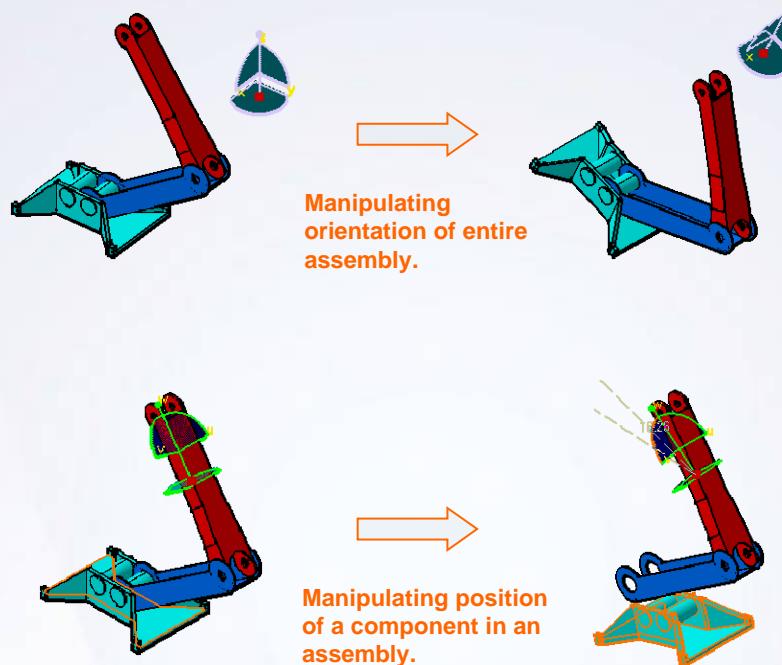
Manipulate the Position of the Component

After inserting the base component, it can be left to “float” in space (without constraints), but it is a good practice to fix this component. Fixing it will serve as a reference for placing all other components that are assembled later. Once you have fixed a component, you can still temporarily manipulate its location using the compass.

Besides using the compass, components can be moved using the Snap tool. These changes are temporary and after updating the assembly, the constraint will be re-evaluated.

In addition to manipulating the position of components in geometry area you can reorder components in the specification tree to match your design requirements.

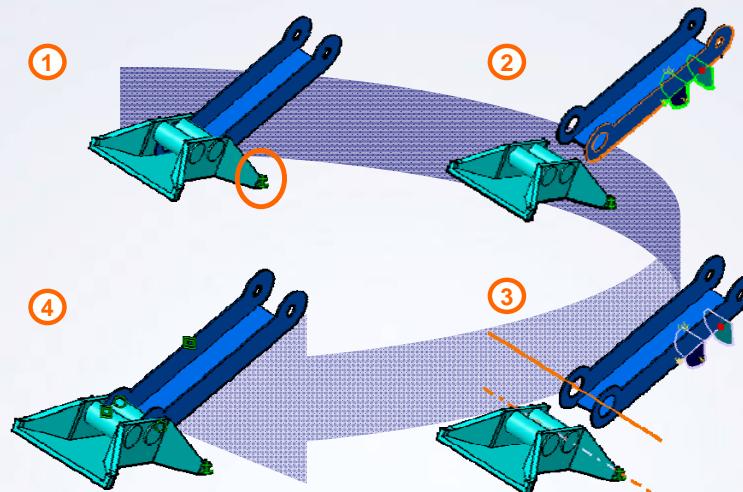
An assembly may require more than one instance of a component. The Copy and Paste options provide an easy way to duplicate a component.



Assemble & Fully Constrain Components

When components are first inserted into assembly, they can be translated and rotated in any direction. As constraints are applied to the component, the degrees of freedom decrease. Similar to sketching constraints, the assembly constraints also locate a geometry that is relative to the existing features. Use the following general steps to add the assembly constraints:

1. Fix one component in space in the assembly.
2. With the compass, drag and rotate components to their approximate positions.
3. Position each component precisely by selecting and applying the appropriate constraints.
4. To control the result, update the assembly.



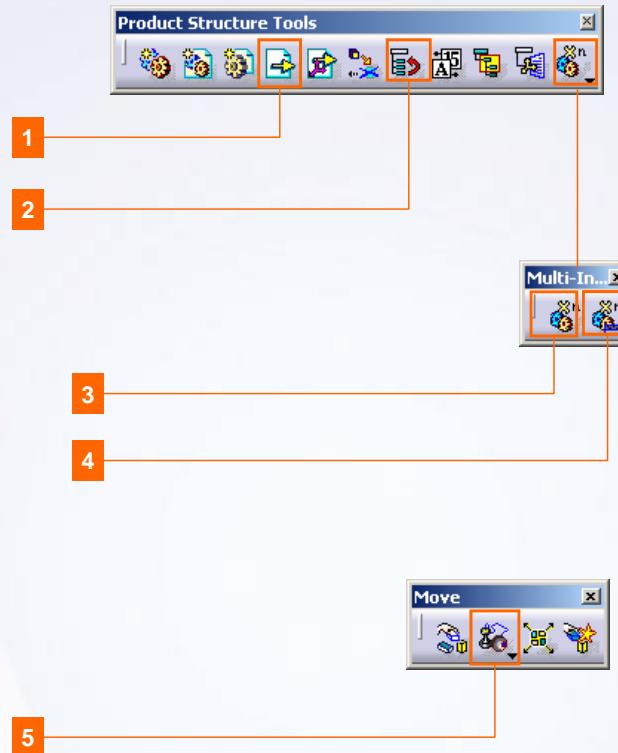
Assembly Design Tools

Product Structure tools

- 1 **Add components:** adds existing components such as parts, subassemblies etc. to a Product
- 2 **Graph tree Reordering:** reorders components in the specification tree to meet your needs
- 3 **Fast Multi Instantiation:** repeats components using the parameters previously set in the Define Multi Instantiation command
- 4 **Define Multi Instantiation:** repeats components as many times as you want in the direction of your choice

Move

- 5 **Snap:** snaps one component over another with reference to selected faces



Assembly Design Tools

Constraints

- 1 **Coincidence Constraint:** creates an alignment between two components that can be coaxial, coplanar, or merged points
- 2 **Contact Constraint:** connects two planes or faces
- 3 **Offset Constraint:** defines the distance between two elements
- 4 **Angle Constraint:** defines the angular distance between two elements
- 5 **Fix Component:** prevents the component from moving from its position during the update operation
- 6 **Fix Together:** enables you to constrain components so that, they move as a single entity
- 7 **Reuse Pattern:** duplicates a component reusing a pattern created in the Part Design workbench



Exercise: Reuse Components

Recap Exercise

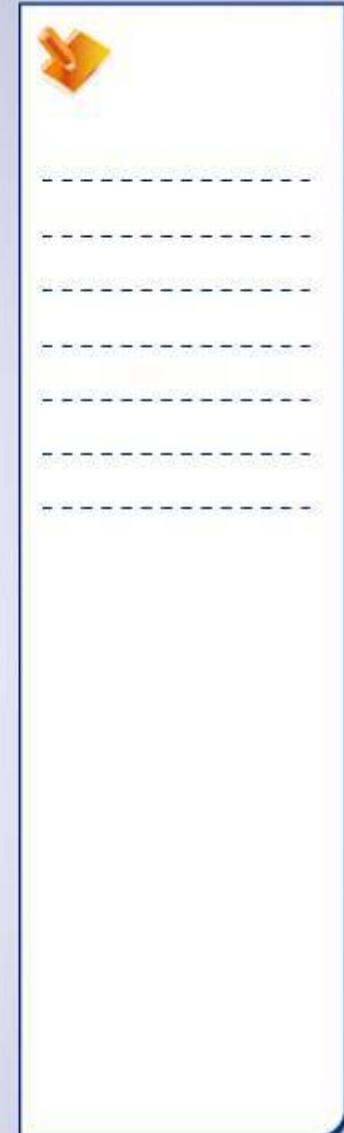
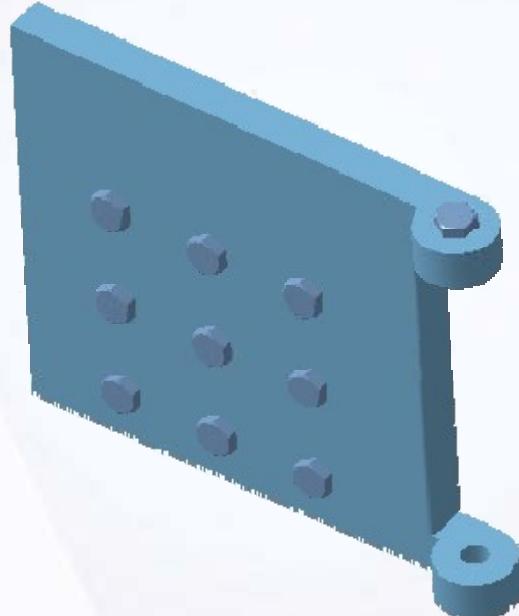


20 min

In this exercise you will practice reusing the patterns. High-level instructions for this exercise are provided.

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

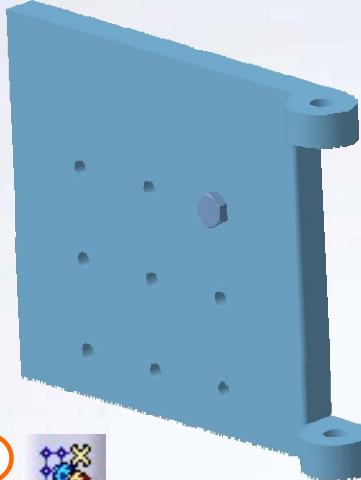
- Use the Reuse Pattern tool
- Copy and paste a component



Do it Yourself (1/3)

1. Load Exercise10B.CATProduct.

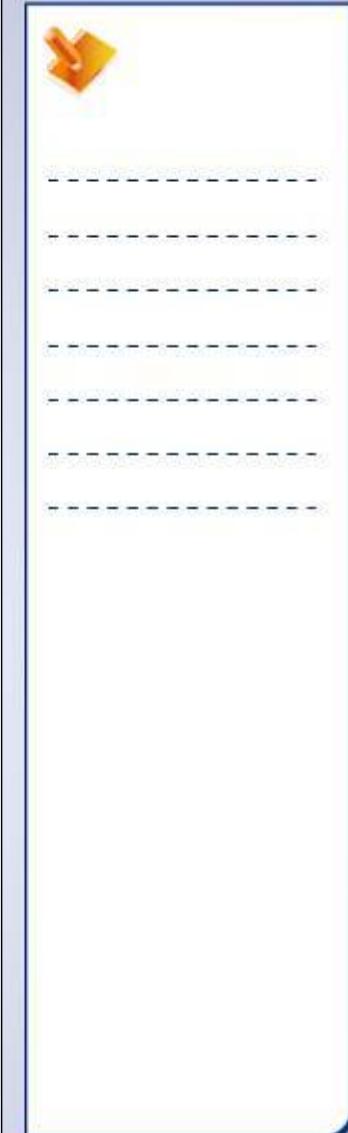
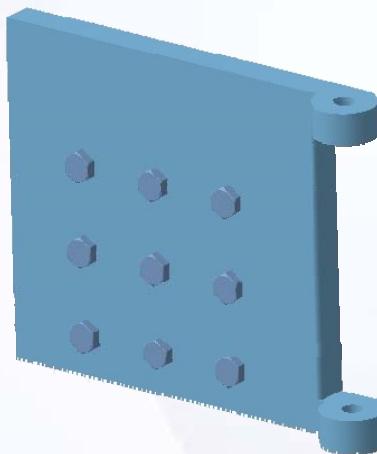
①



2. Instantiate VISD8.CATPart using Reuse Pattern.

- Using Rectangular pattern defined in the **Engine_support_pattern.catpart**, instantiate **VISD8.CATPart**.

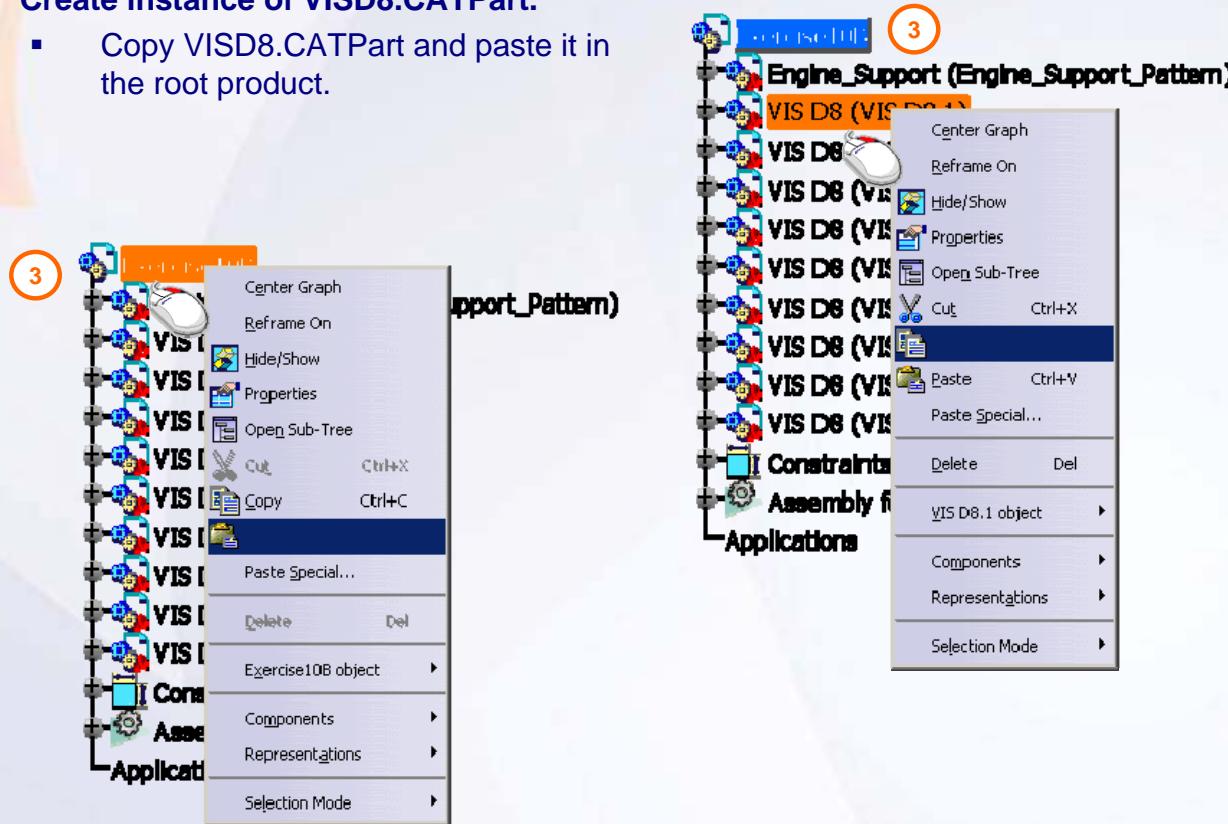
②



Do it Yourself (2/3)

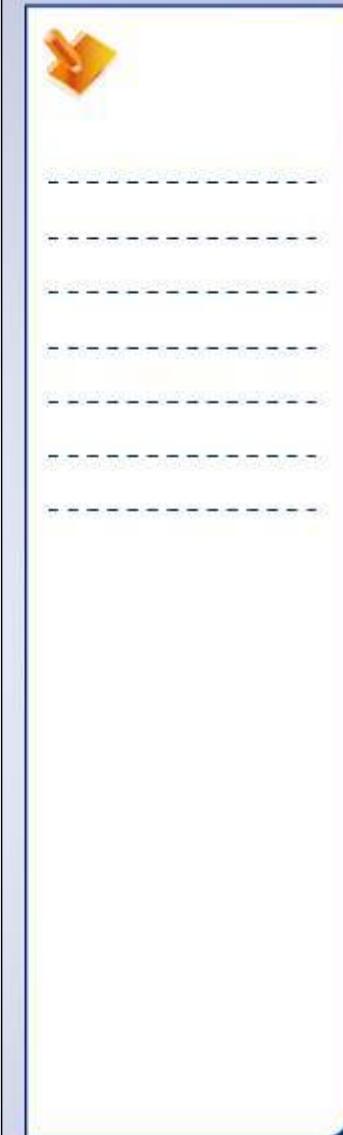
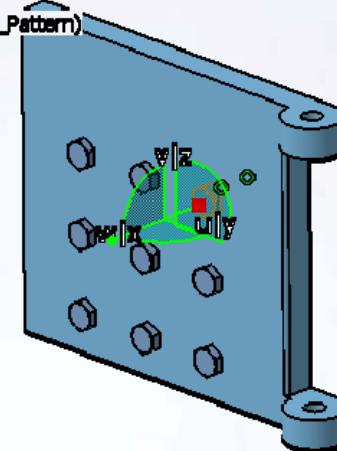
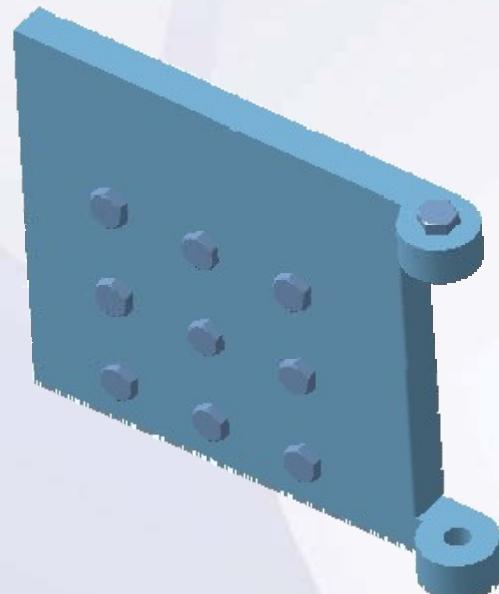
3. Create instance of VISD8.CATPart.

- Copy VISD8.CATPart and paste it in the root product.



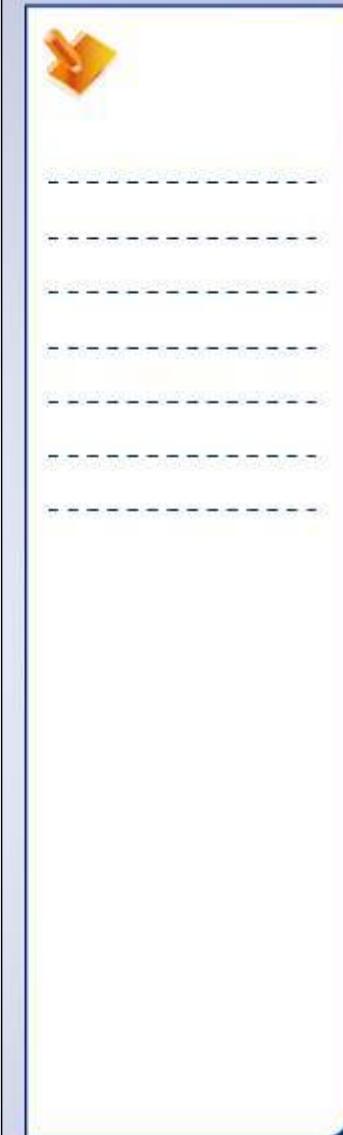
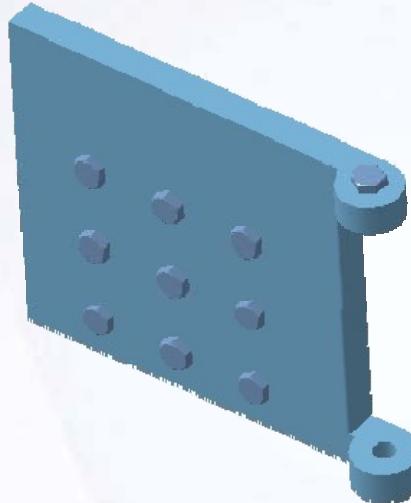
Do it Yourself (3/3)

4. Manipulate the position of new instance using the compass.
 - Drag and drop the compass on the pasted instance and manipulate the position.
5. Close the assembly without saving it.



Exercise Recap: Reuse Components

- ✓ Use the Reuse Pattern tool
- ✓ Copy and paste a component



Exercise: Components Manipulation

Recap Exercise

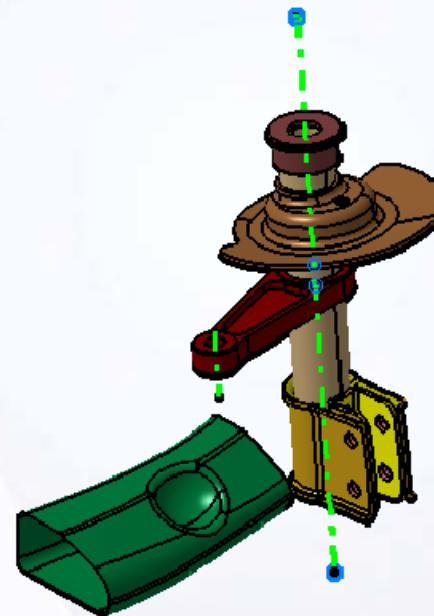


10 min

In this exercise you will manipulate the Damper Assembly using the tools learnt in this lesson. You will complete this exercise with no detailed instructions.

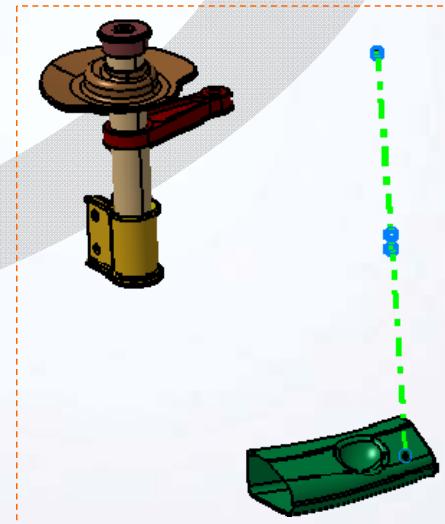
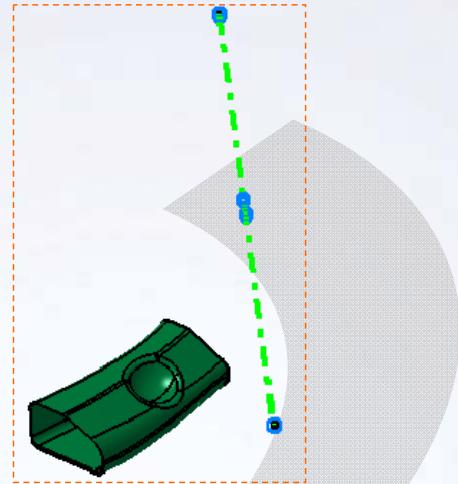
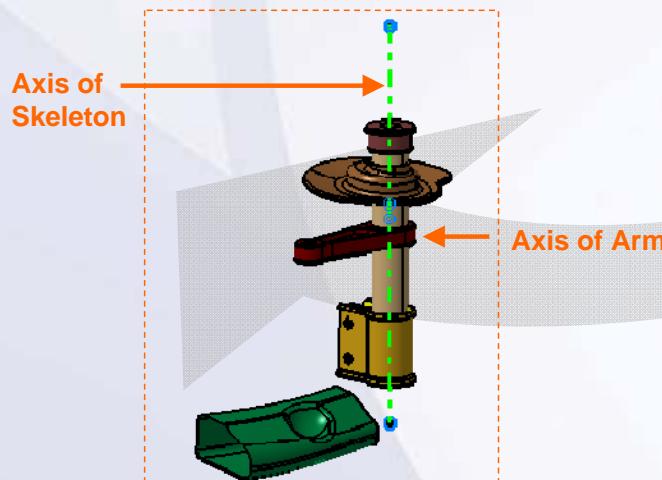
By the end of this exercise you will be able to:

- Apply a fix constraint
- Insert an existing component
- Manipulate the assembly using compass
- Manipulate the assembly using snap
- Use Edit compass to translate a component



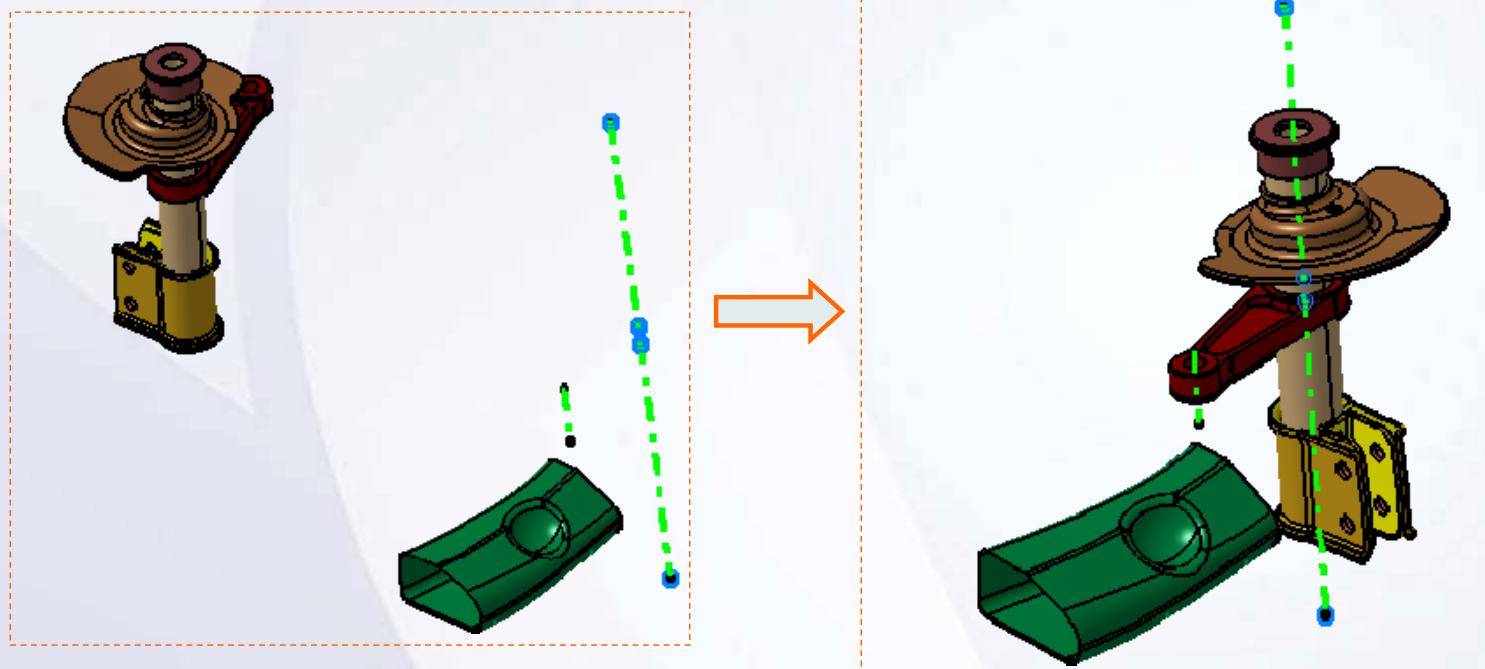
Do it Yourself

1. Load Exercise10C.CATProduct and apply fix constraint to skeleton part.
 - If you do not see the model as shown, then use the contextual menu **Representation > Design Mode** on the Product root Exercise 10C.
2. Insert an existing component.
 - Insert DamperAssembly.CATProduct to root assembly.
3. Manipulate the assembly using compass.
 - Manually position DamperAssembly.CATProduct and Arm.CATPart using a compass.
 - Snap the axis of arm and axis of skeleton part.
 - Use edit compass to translate the DamperAssembly.



Exercise Recap: Components Manipulation

- ✓ Apply a fix constraint
- ✓ Insert an existing component
- ✓ Manipulate the assembly using compass
- ✓ Manipulate the assembly using snap
- ✓ Use Edit compass to translate a component



Exercise: Constrain Components

Recap Exercise

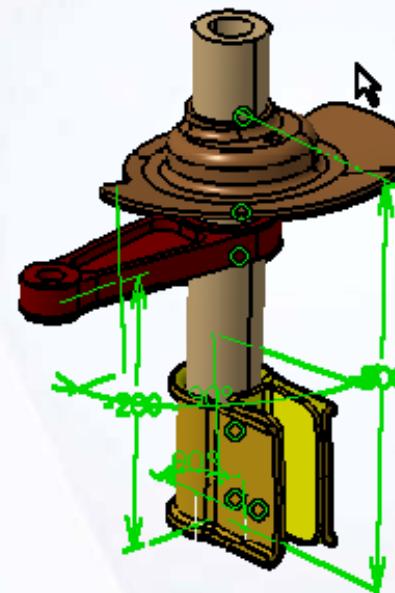


20 min

In this exercise you will practice adding components to the assembly and constraining them. High-level instructions are provided for this exercise.

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

- Add components to the assembly
- Constrain the components



Do it Yourself (1/5)

1. Create a new product.

- Create a new product and name it Exercise10E.CATProduct.
- If the Cache is ON by default, the user will have to switch to Design Mode.

2. Add existing components to the assembly.

- Add Pillar.CATPart, Seat.CATPart, Arm.CATPart and Bracket.CATProduct to the root assembly.



3. Fix Pillar.CATPart.

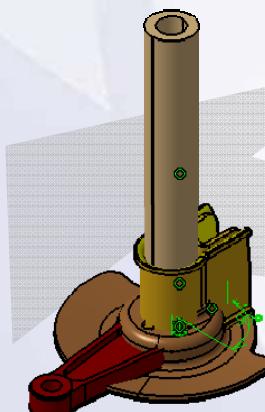
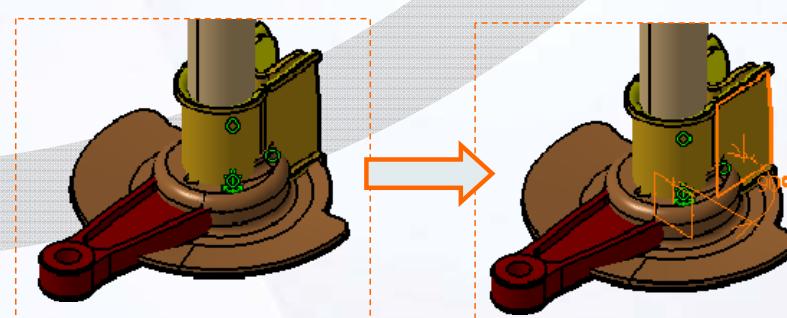
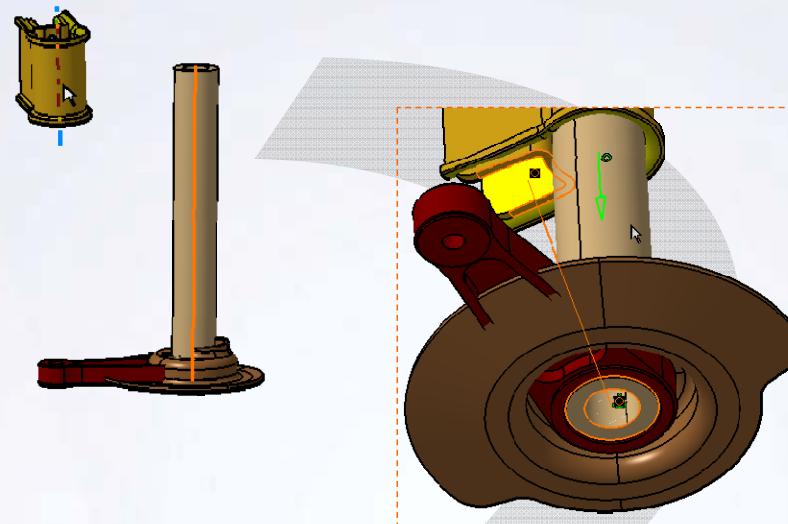
- Select Pillar.CATPart and click **Fix** constraint icon.



Do it Yourself (2/5)

4. Assemble Bracket with reference to Pillar.

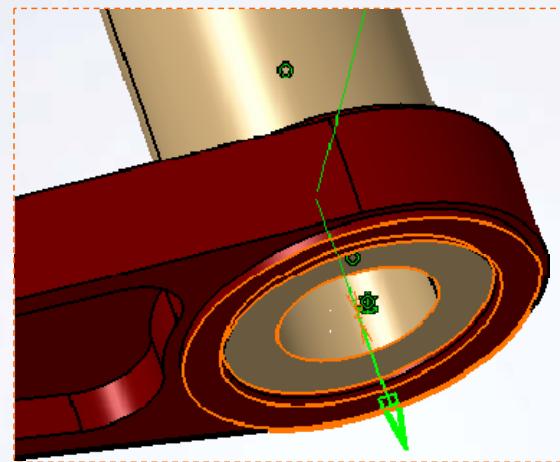
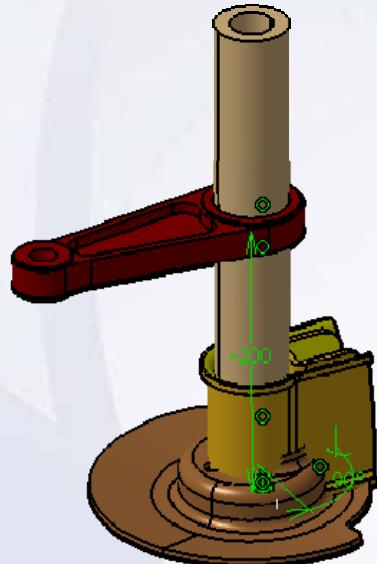
- Create a coincidence constraint between axis of Pillar and axis of Bracket.
- Create a coincidence constraint between the bottom face of Pillar and bottom face of Bracket.
- Manipulate the position of the Bracket using a mouse. Then create an angle constraint of [90 degrees] between YZ plane of Pillar and the vertical wall of the Outer Bracket.



Do it Yourself (3/5)

5. Assemble Arm with reference to Pillar.

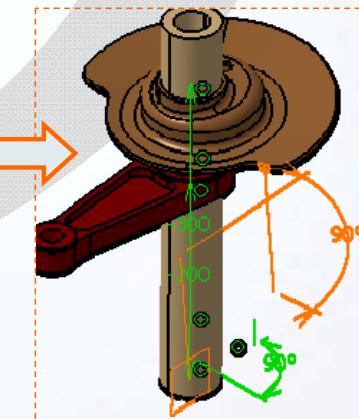
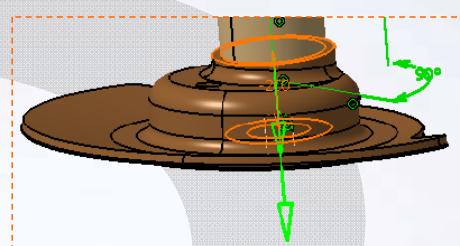
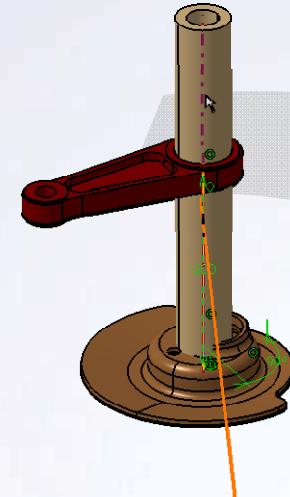
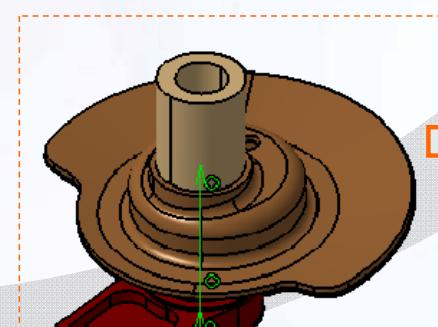
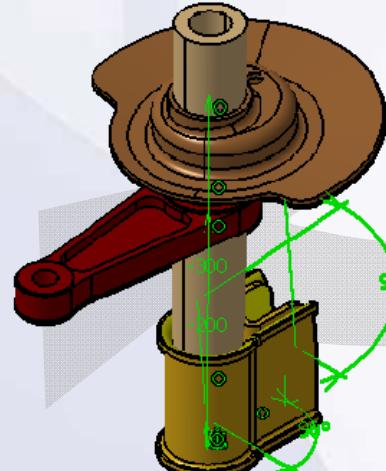
- Create a coincidence constraint between axis of Pillar and axis of Arm.
- Create an offset constraint of [-200mm] between the bottom surface of Pillar and bottom surface of Arm.



Do it Yourself (4/5)

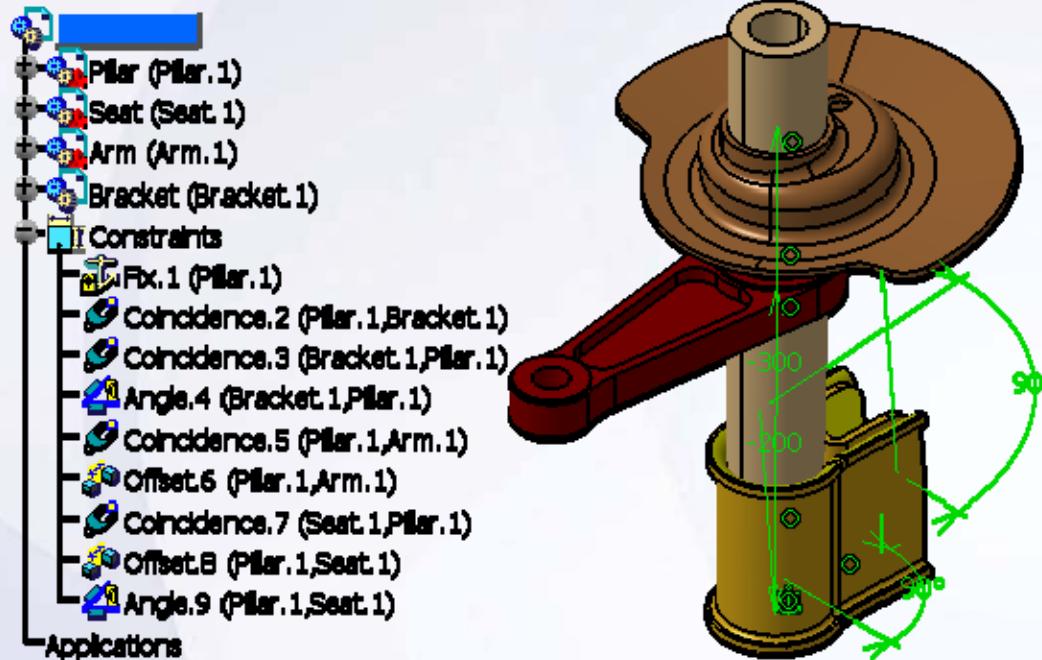
6. Assemble Seat with reference to Pillar.

- Create coincidence constraint between the axis of Pillar and axis of Seat.
- Create an offset constraint of [-300mm] between the bottom surface of Pillar and top surface of Seat.
- Manipulate the position of Bracket using a mouse. Then create an angle constraint of [90 degrees] between the ZX plane of Pillar and the vertical wall of Seat.



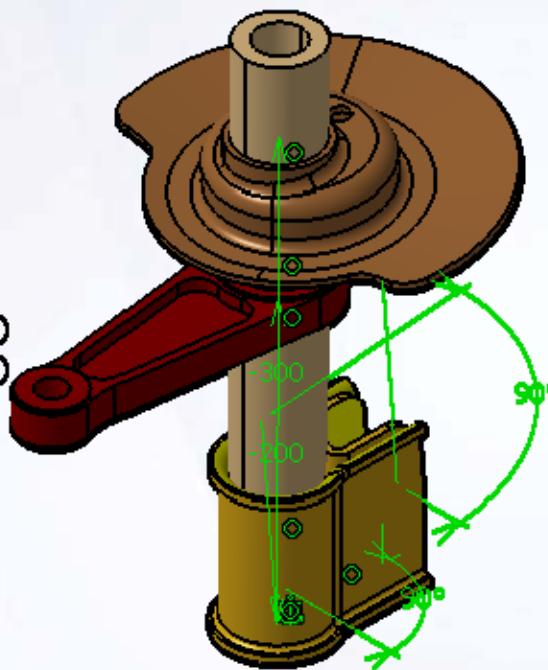
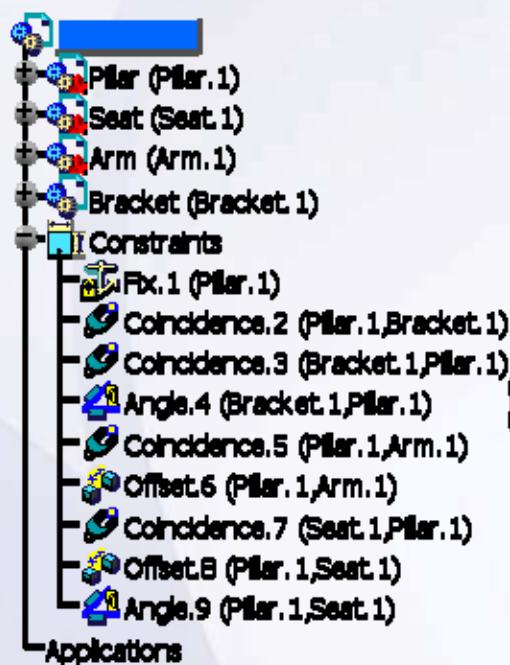
Do it Yourself (5/5)

7. Close the assembly without saving it.



Exercise Recap: Constrain Components

- ✓ Adding existing components to the assembly
- ✓ Constraining the components



Exercise: Constrain Components

Recap Exercise

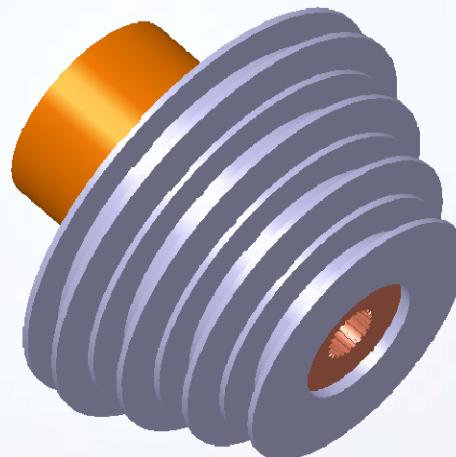


20 min

In this exercise you will troubleshoot an existing assembly and determine which of its components are not fully constrained. You will use the tools used in the previous exercises to complete this exercise.

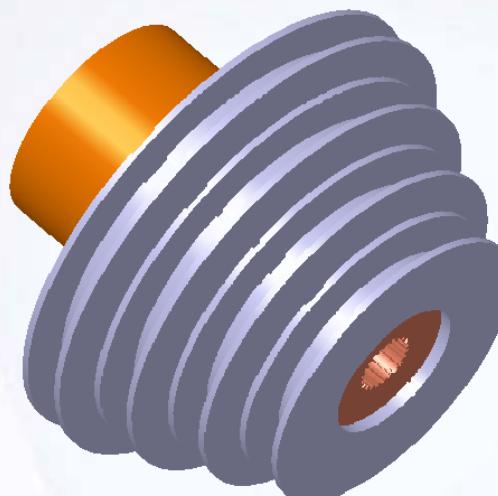
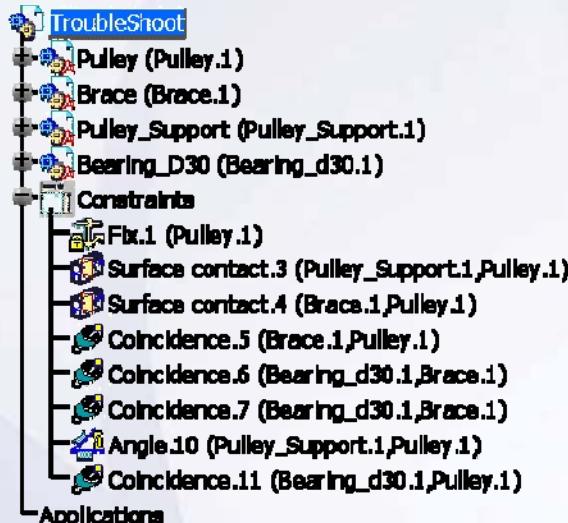
By the end of this exercise you will be able to:

- Determine the degrees of freedom of a component
- Fully constrain a component



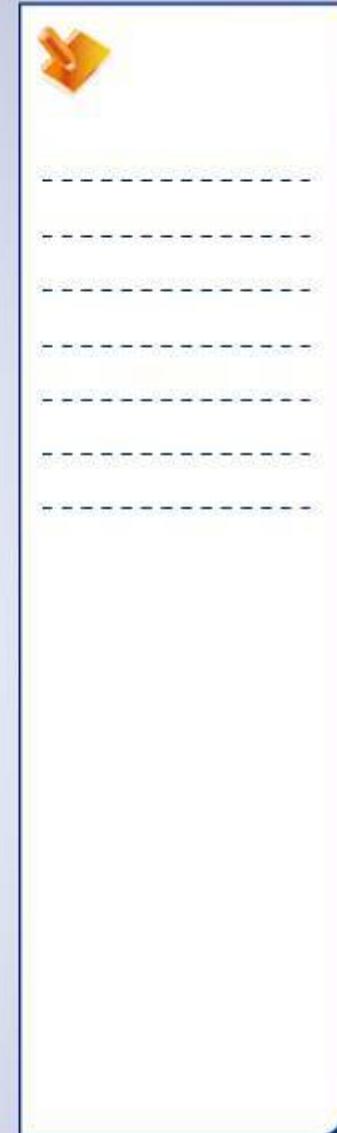
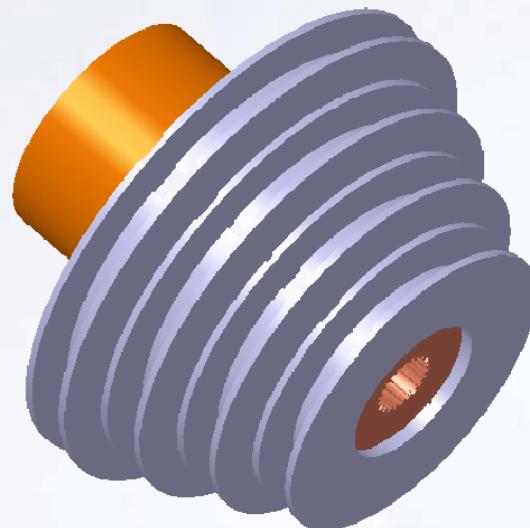
Do it Yourself

Open Troubleshoot.CATProduct and determine which components in the assembly are not fully constrained. Create the constraints necessary to fully constrain the components of the assembly.



Exercise Recap: Constrain Components

- ✓ Determine the degrees of freedom of a component
- ✓ Fully constrain a component



Case Study: Assembly Design

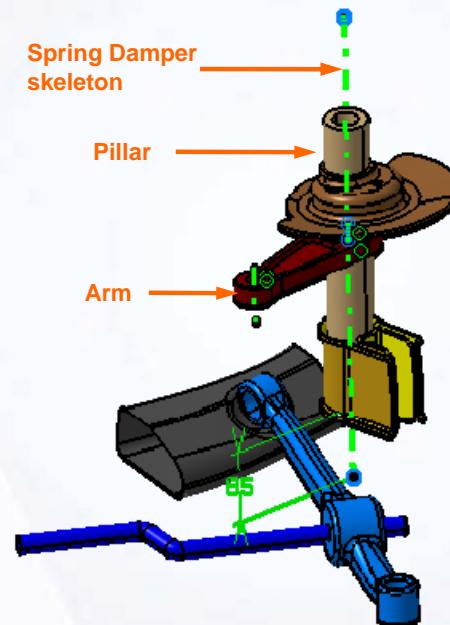
Recap Exercise



10 min

In this exercise you will create the case study model. Let us recall the design intent of this model:

- ✓ Position the Spring Damper sub-assembly with reference to the Spring Damper skeleton.
 - The Pillar axis must coincide with the axis of Spring Damper skeleton.
 - The Pillar lower face must be 85mm from the Strut Lower point of Spring Damper skeleton.
 - The Arm axis must coincide with the axis of Spring Damper skeleton.

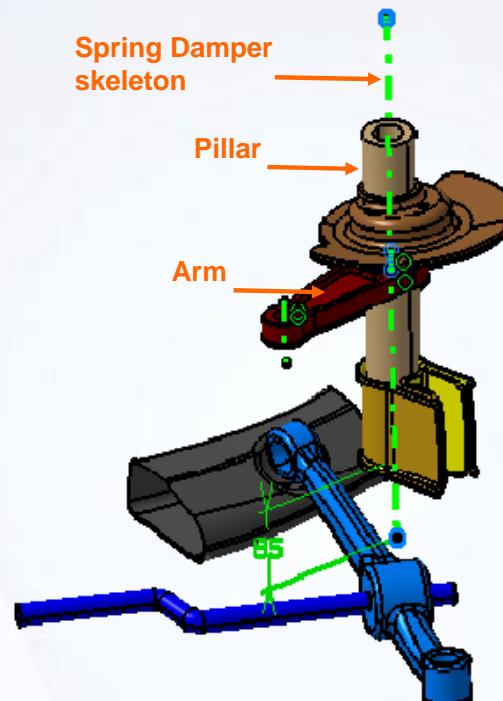


Using the techniques you have learned so far, create the model without detailed instructions.

Do It Yourself: Assemble the Damper Assembly

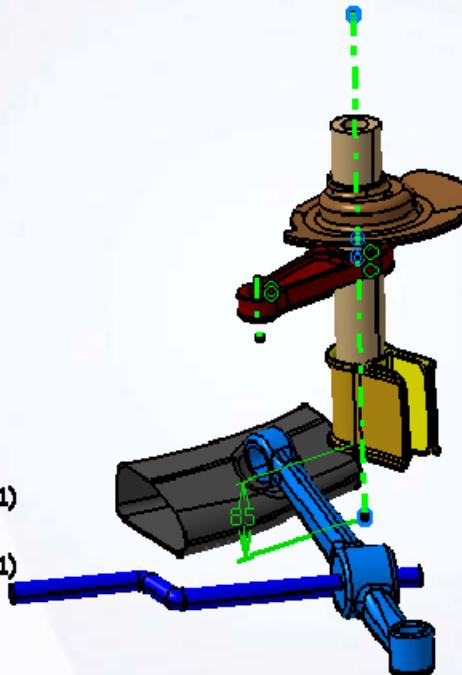
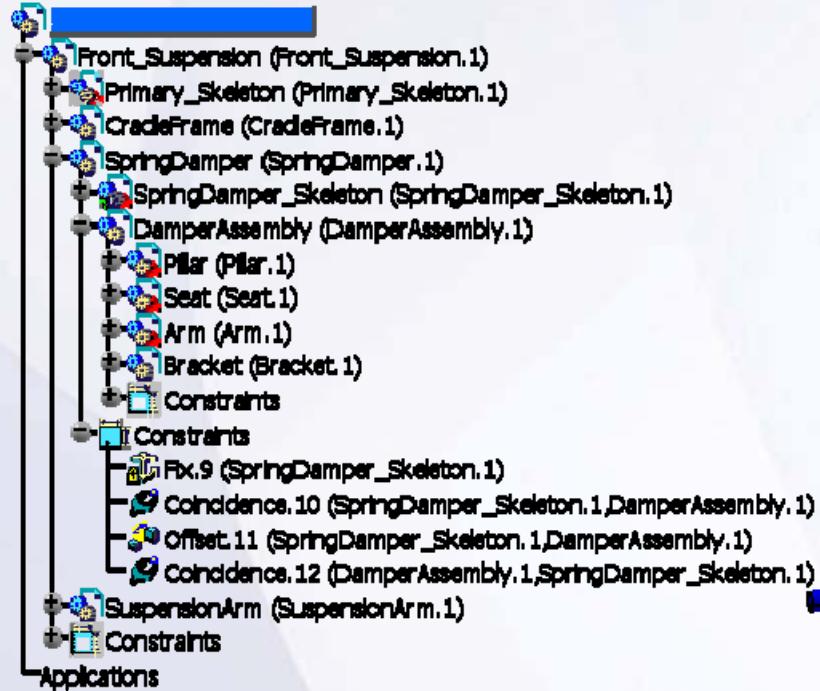
Load Automobile_CaseStudy10.CATProduct and add constraint using the information below.

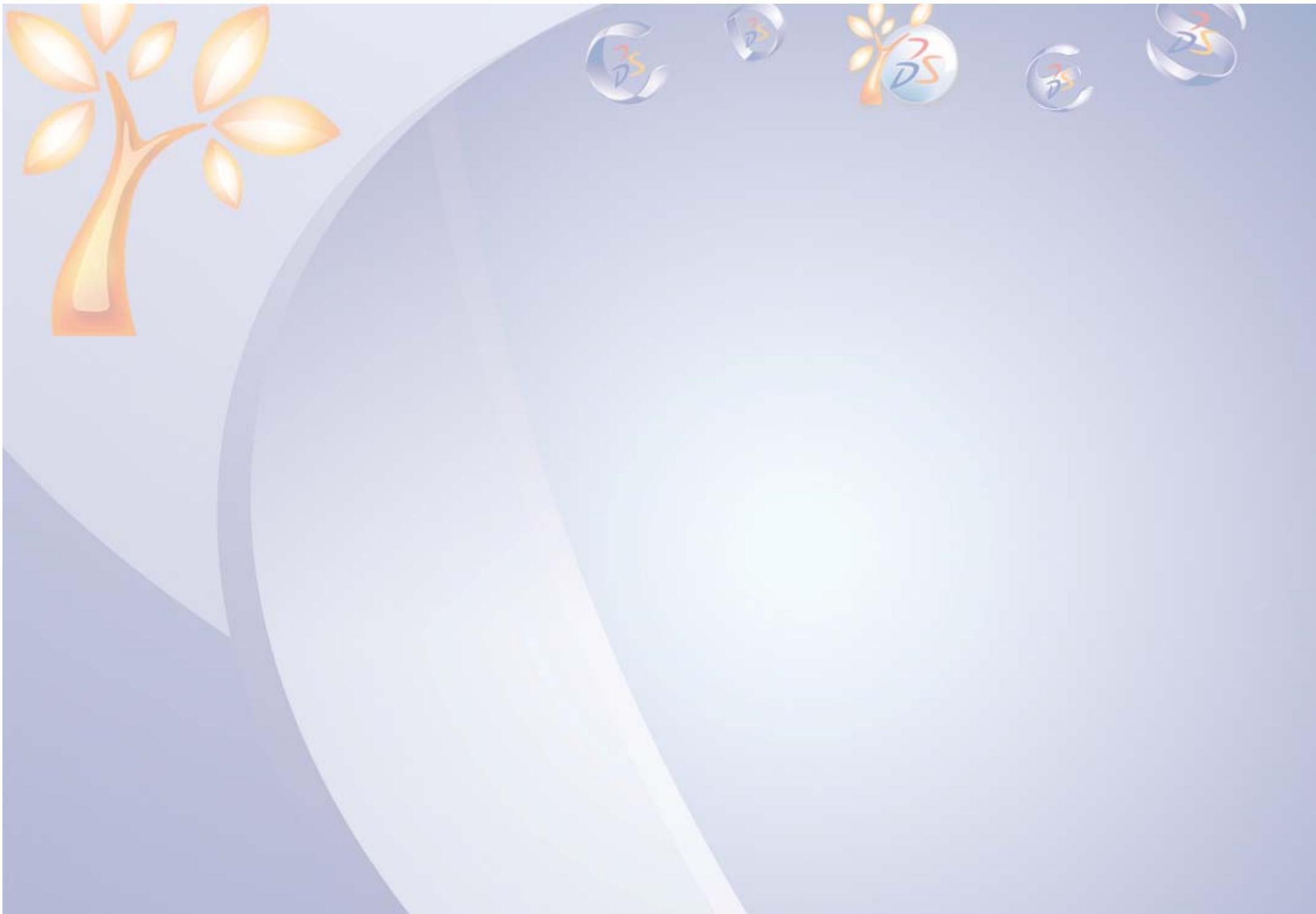
- ✓ Insert the Damper Assembly in Spring Damper sub-assembly.
- ✓ Move the Spring Damper sub-assembly with a compass.
- ✓ Position the Spring Damper sub-assembly with reference to Spring Damper skeleton.
 - The Pillar axis must coincide with the axis of Spring Damper skeleton.
 - The Pillar lower face must be 85mm from the Strut Lower point of Spring Damper skeleton.
 - The Arm axis must coincide with the axis of Spring Damper skeleton.



Case Study Recap: Damper Assembly

- ✓ Create an existing component
- ✓ Manipulate the position of the component using a compass
- ✓ Constrain the component





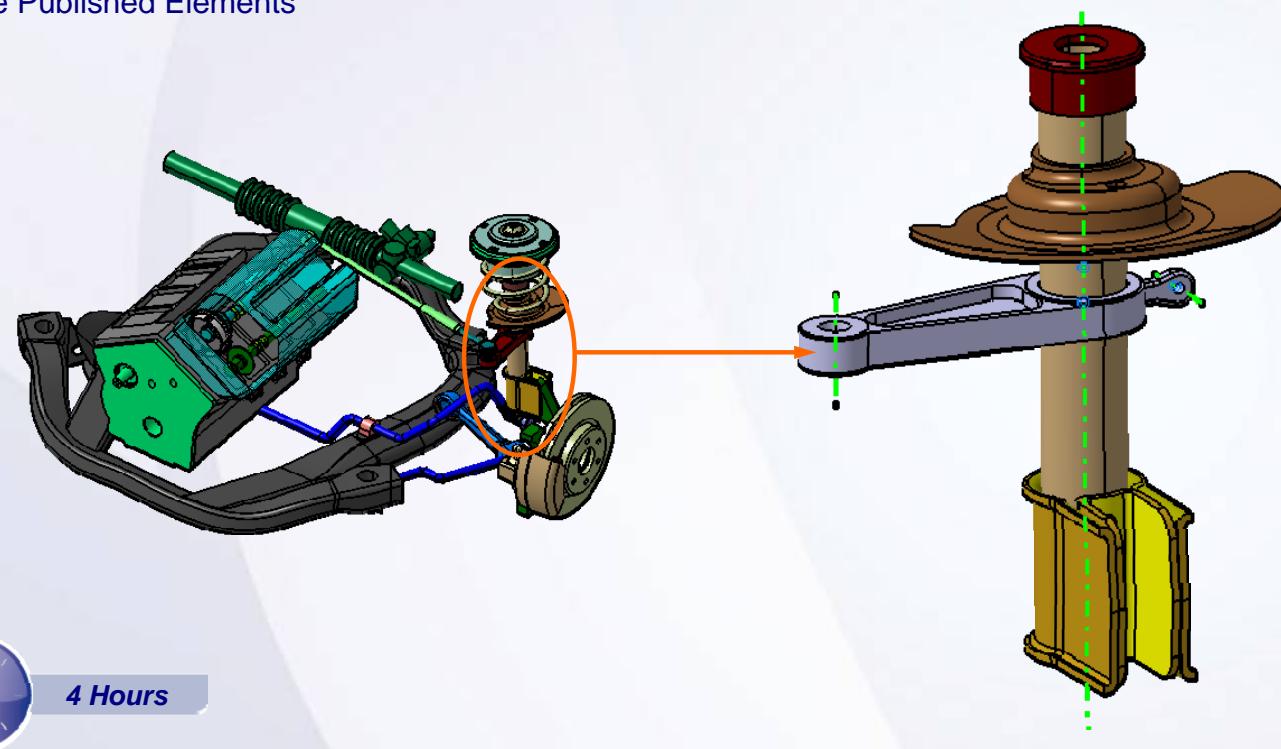
Designing in Context

11

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Clarify the Display
- ✓ Create the Skeleton Model
- ✓ Create Contextual Parts
- ✓ Use Published Elements



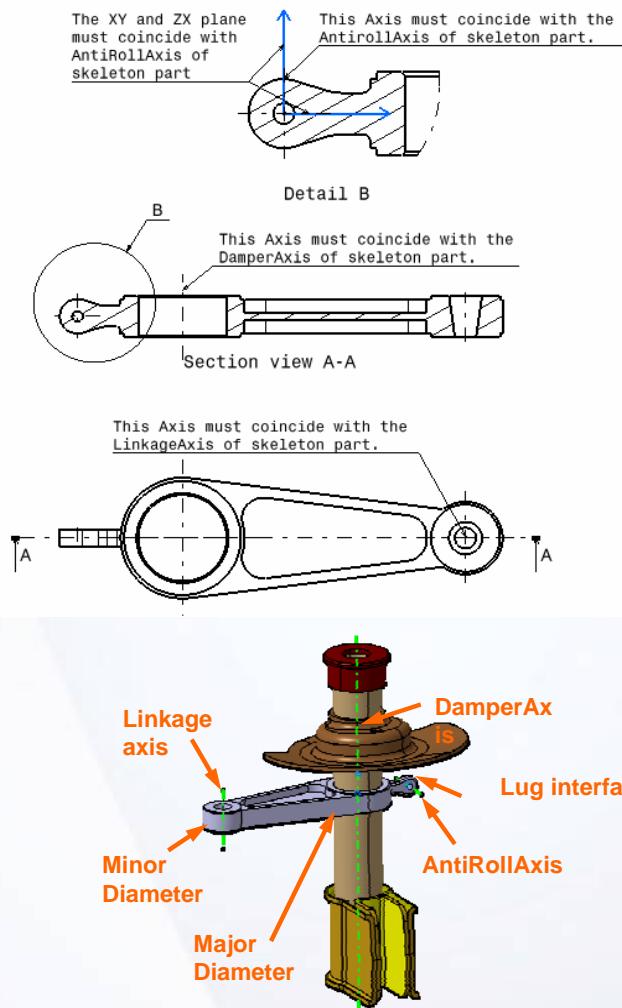
4 Hours

Case Study

The case study for this lesson is the Arm used in the Front Suspension and Engine assembly.

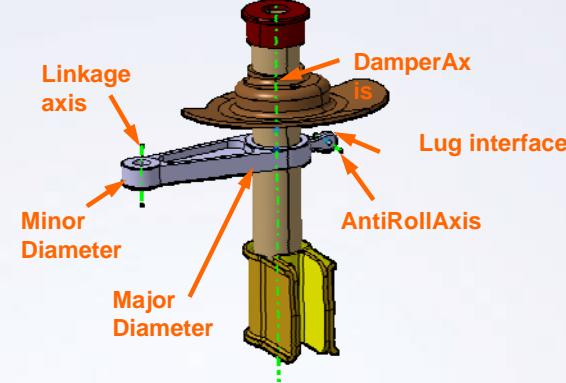
Design Intent

- ✓ Design Arm contextually using published elements of the Skeleton Model
(DamperAssembly_Skeleton.CATPart)
 - A. The axis of the major diameter of Arm must coincide with the DamperAxis of skeleton part.
 - B. The axis of the minor diameter of Arm must coincide with the LinkageAxis of skeleton part.
 - C. The lug interface of the Arm must coincide with the AntiRollAxis of skeleton part.
- ✓ Position the Arm with reference to the skeleton part.
 - D. The bottom face of the Arm must be more than 200 mm from the PillarSupportPlane of the skeleton part.
- ✓ Modify the axis spacing between the Pillar and AntiRollAxis to > 61mm.



Stages in the Process

1. Insert a new part (Arm.CATPart) in DamperAssembly.CATProduct.
2. Position the XY and ZX planes of the arm with coincidence constraints on the AntiRollAxis.
3. Using publications of the (DamperAxis, LinkageAxis and AntiRollAxis), create the Arm contextually.
4. Position the Arm with reference to the skeleton part.



Clarify the Display

For large assemblies, the performance of CATIA can be improved while panning, zooming, updating and saving the assemblies. Following tools help to improve the performance of CATIA while working with large assemblies:

1. Using Visualization mode: In this mode only a light CGR representation of the model is loaded.
2. Hiding components: You can hide components to clarify the display and see only desired components.
3. Deactivating representations: Deactivating representations improves performance by hiding the components and excluding them from Mass Property analysis.
4. Deactivating components: It will remove the component from show and no show space, bill of Material.



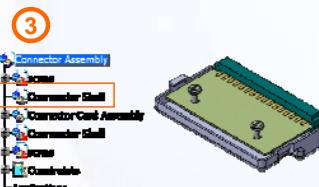
Powertrain Assembly



Visualization Mode



Hiding Components



Deactivating Representations

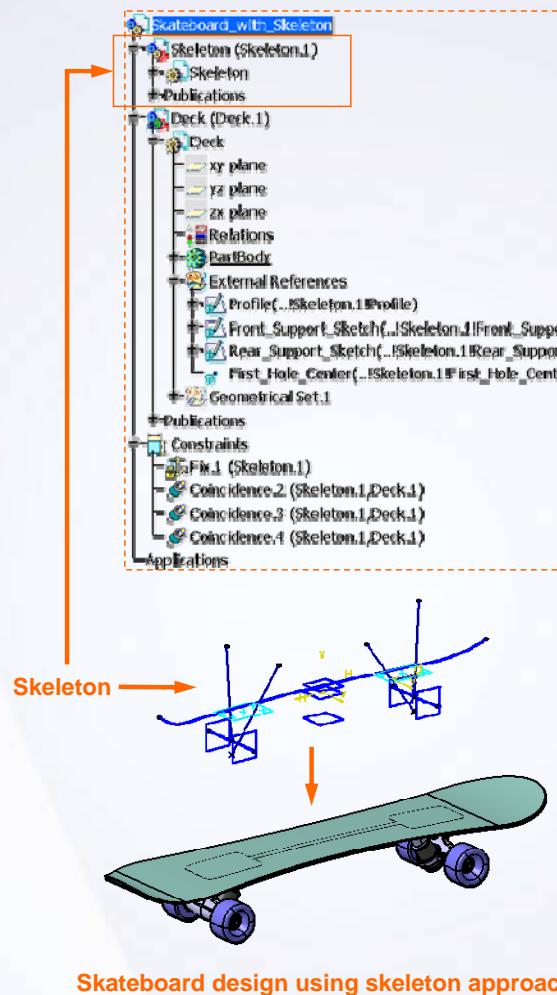


Deactivating Components

Create the Skeleton Model

The skeleton method is a top down design approach. Using the skeleton method you can create and reuse the information stored in a single part, called the skeleton, to define the underlying design framework of individual components and assemblies. It offers the following key advantages.

- ✓ Specification-driven design: All important information related to the design and space requirements are defined within the skeleton.
- ✓ Design changes: It helps manage high-level design changes and propagate them throughout the assembly.
- ✓ Collaborative design: Key information from skeleton model can be associatively copied into the components. Changes to the design can be made in the skeleton and all models will update to reflect these modifications. As the components are not linked to each other, the deletion of a component within an assembly will not impact the others.
- ✓ Avoid update loops: As all are external references point to the skeleton part, update loops are avoided.



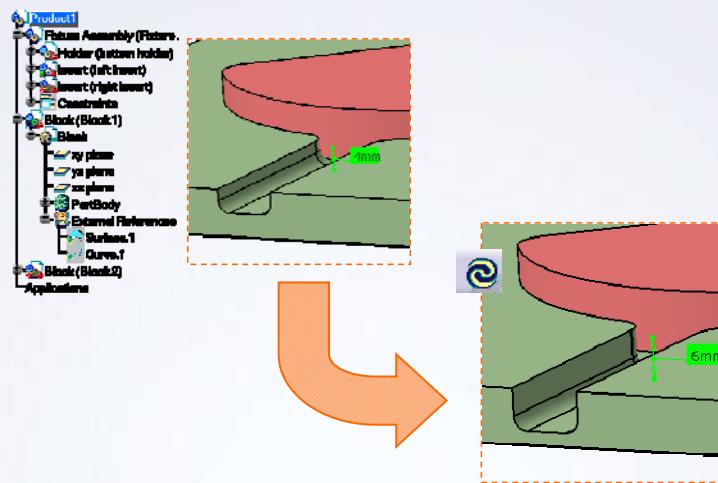
Create Contextual Parts

Contextual parts are parts that have their geometry driven by another component. A change in the driving geometry of the referenced part will result in changes in the contextual part.

- ✓ It is important to fully constrain the contextual parts to avoid unintentional distortion of the geometry.
- ✓ The contextual part can be edited within or outside the context of assembly in which the contextual elements were defined.
- ✓ When the component that is used as a reference for the contextual part is replaced, the driven component needs to be reconnected to the new driving geometry.

The benefits of using Design in Context:

1. Reuses existing geometry in one part.
2. Reuses parameters defined in one part.
3. Automatic update of an assembly and its contextual parts.

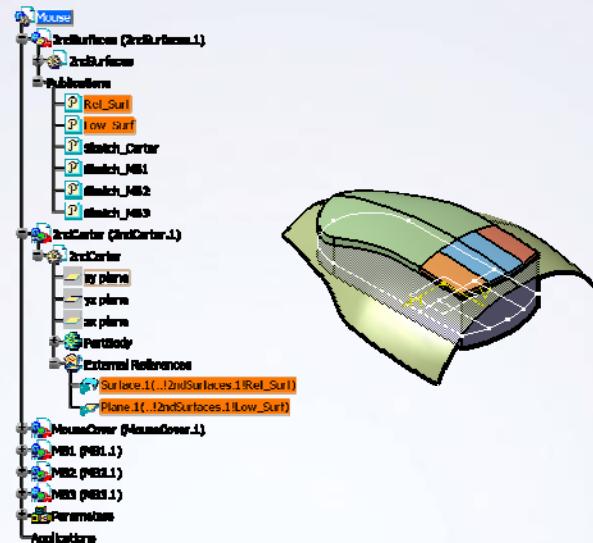


Use Published Elements

Published geometry can be used to control external references when:

- A. Constraining an assembly.
- B. Designing in context.

It is particularly useful when replacing components with assembly constraints or that have been designed in context.



Main Tools

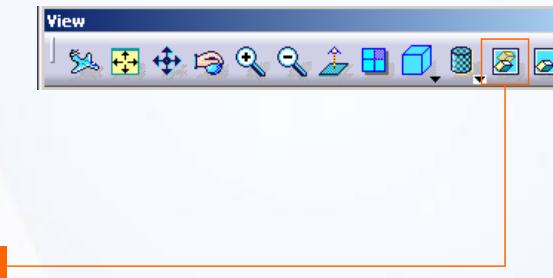
Representations

- 1 **Design Mode:** To load exact geometry in the memory and make the part history available.
- 2 **Visualization Mode:** To display a representation of the geometry. The exact geometry is not loaded in the memory, part history is not available.
- 3 **Active Node:** To activate the geometrical representation of the selected component.
- 4 **Deactivate Node:** To deactivate the geometrical representation of the selected component.



View

- 5 **Hide/Show:** To change hide / show state of a component.



Exercise: Visualization Mode

Recap Exercise

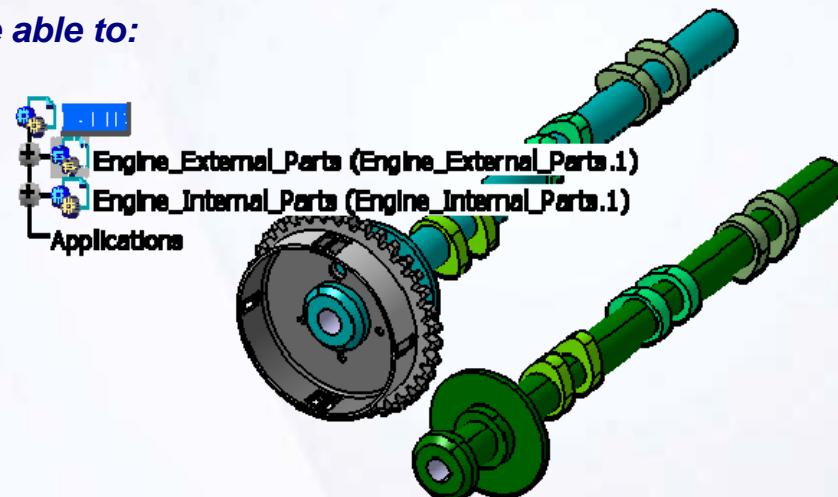


10 min

In this exercise you will practice working in the visualization mode and modifying a component in the assembly context. High-level instructions for this exercise are provided.

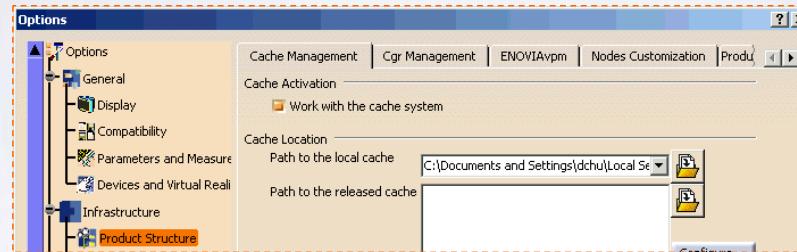
By the end of this exercise you will be able to:

- Use the visualization mode
- Switch from Visualization mode to Design mode
- Modify a component



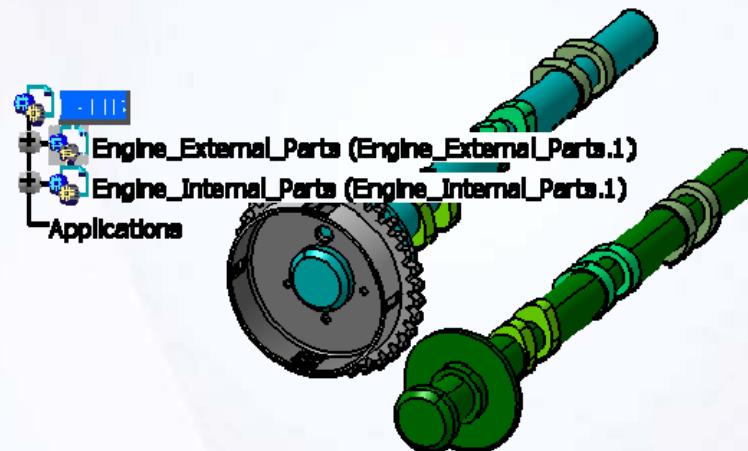
Do it Yourself (1/4)

1. Verify whether the Visualization mode is active.



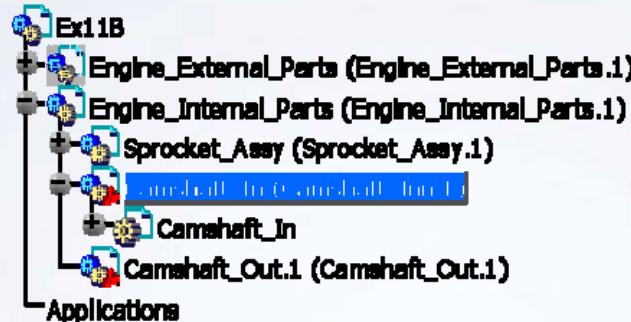
2. Load Ex11B.CATProduct.

3. Hide the Engine_External_Parts.CATProduct.



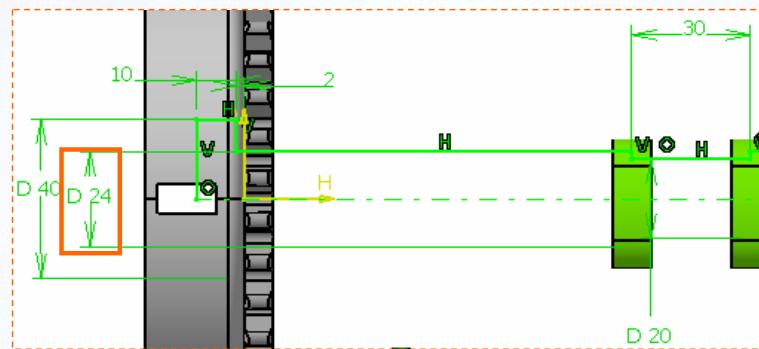
Do it Yourself (2/4)

4. Switch Camshaft_In.CATPart to Design mode.



5. Modify the shaft diameter.

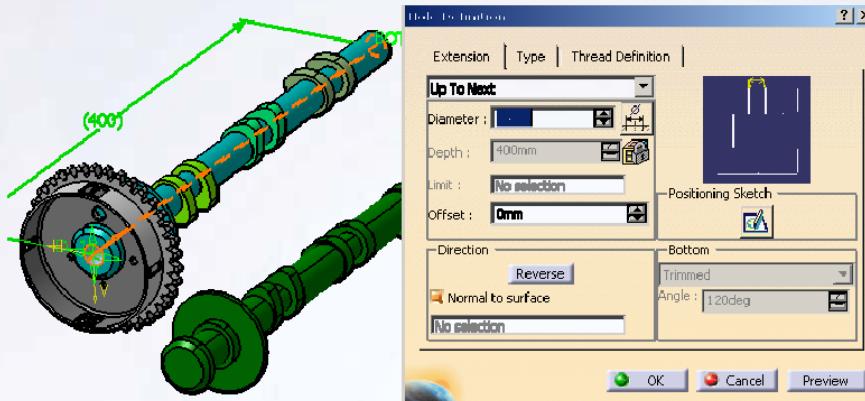
- Modify the major diameter to [24mm].



Do it Yourself (3/4)

6. Create a hole.

- Create a simple hole of diameter [16mm].



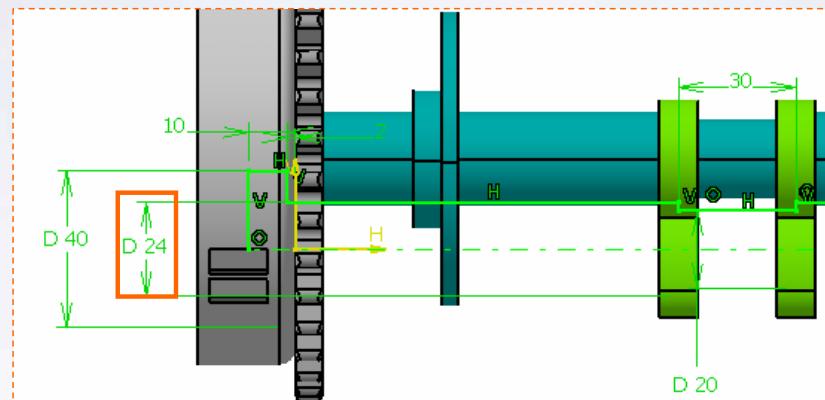
7. Switch Camshaft_Out.CATPart to Design mode.



Do it Yourself (4/4)

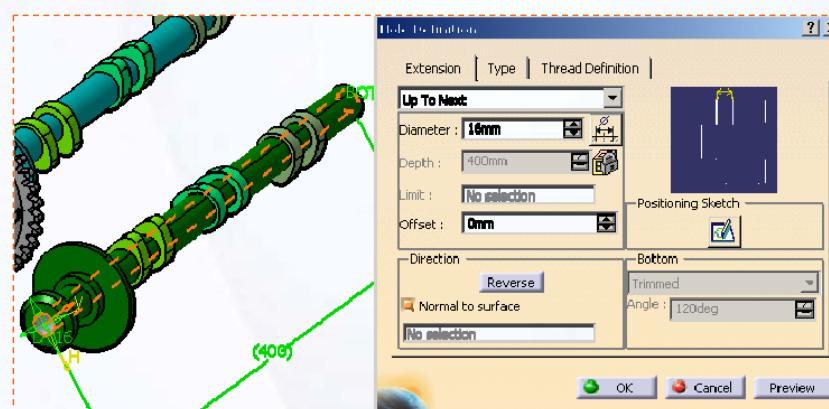
8. Modify the shaft diameter.

- Modify the major diameter to [24mm].



9. Create a hole.

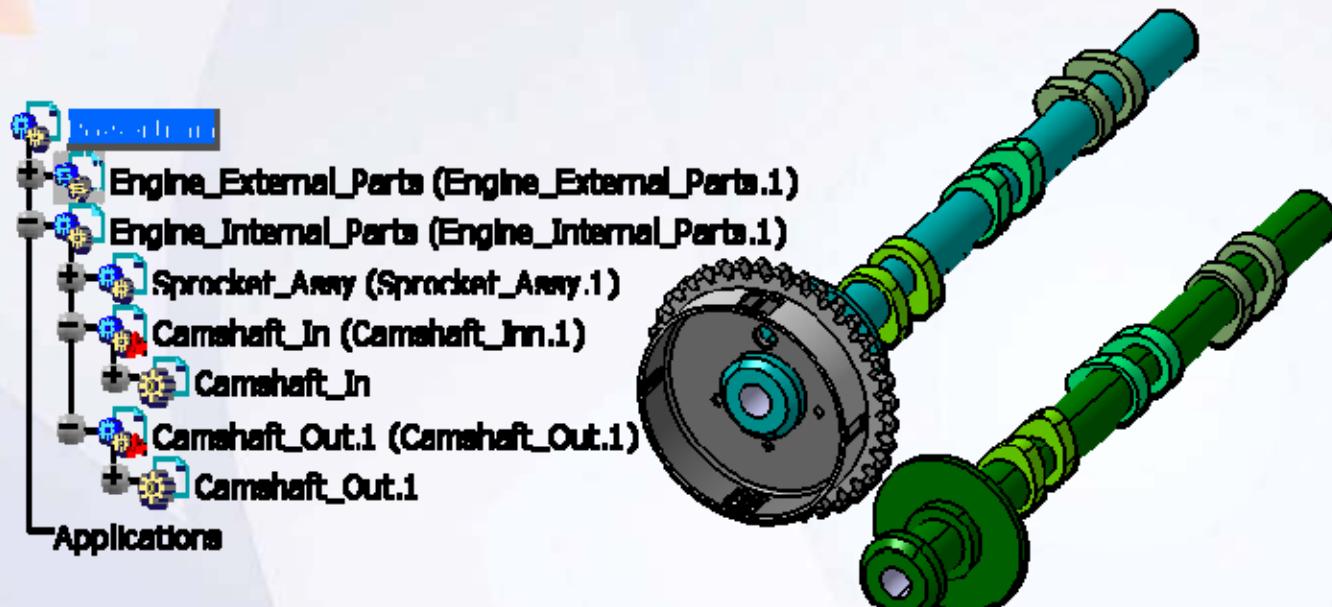
- Create a simple hole of diameter [16mm].



10. Close the file without saving it.

Exercise Recap: Visualization Mode

- ✓ Use the visualization mode
- ✓ Switch from visualization mode to design mode
- ✓ Modify a component



Exercise: Contextual Part

Recap Exercise

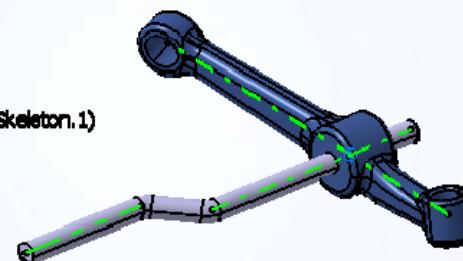
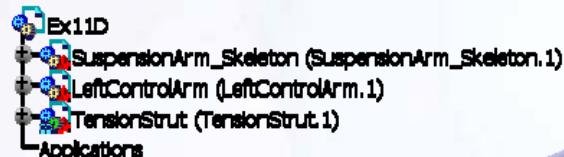


10 min

In this exercise, you use the skills learned in this lesson to create a component in context using published external references and skeleton. High-level instructions for this exercise are provided.

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

- Insert a new part
- Create a contextual part using published elements

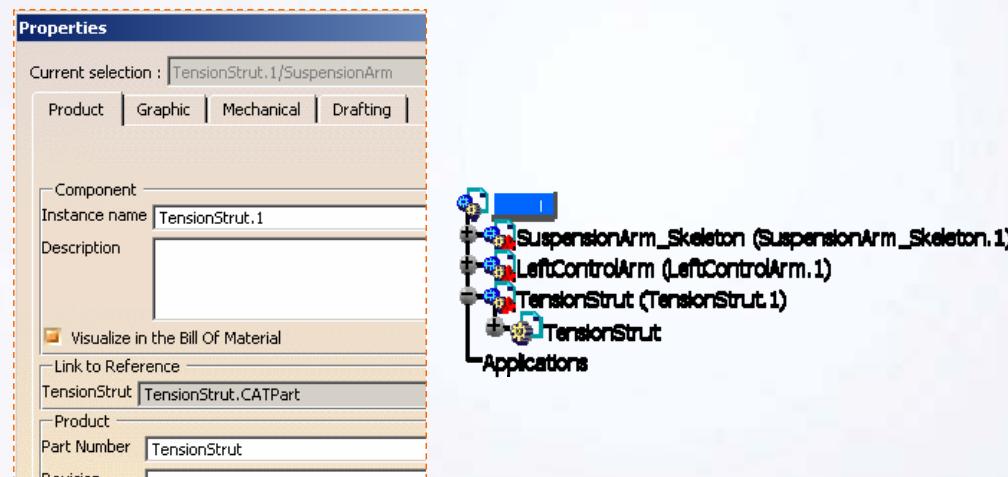


Do it Yourself (1/4)

1. Load the CATProduct.
 - Load Ex11D.CATProduct.



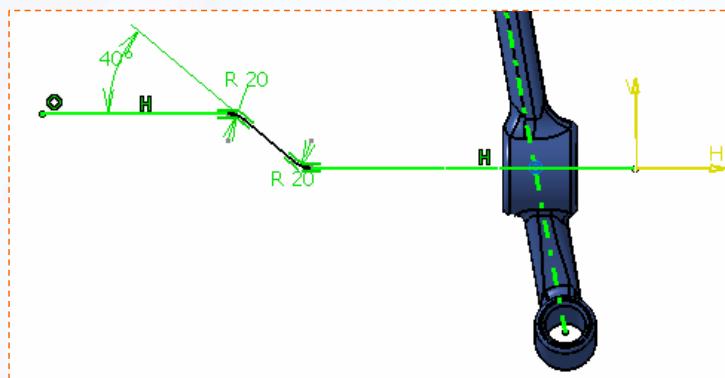
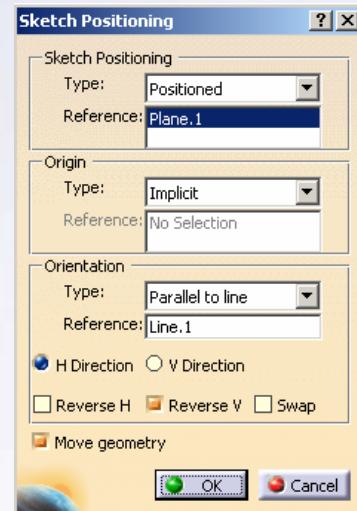
2. Insert a new part.
 - Insert a new part in root assembly and rename it as TensionStrut.CATPart.



Do it Yourself (2/4)

3. Create a positioned sketch.

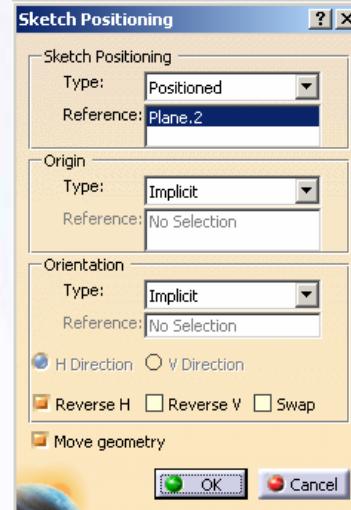
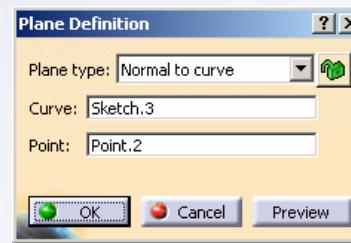
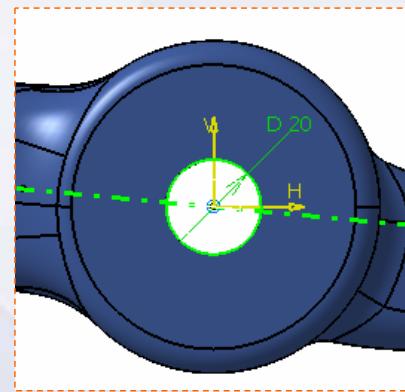
- Create a positioned sketch on a plane, which is passing through published elements ChassisMount and ControlArmMount of SuspensionArm_Skeleton.CATPart.



Do it Yourself (3/4)

4. Create a positioned sketch.

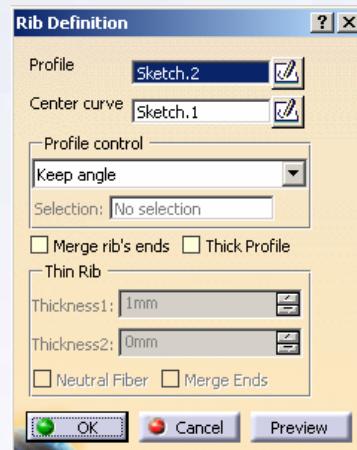
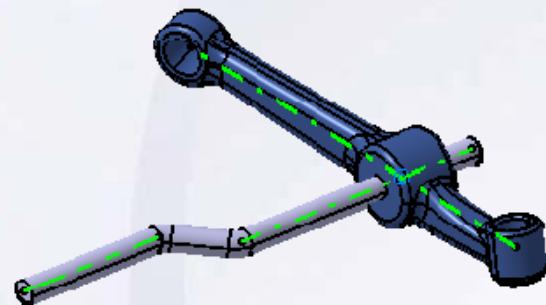
- Create a positioned sketch on a plane, which is normal to the sketch created in step 3 and passing through the ControlArmPt of SuspensionArm_Skeleton.CATPart.



Do it Yourself (4/4)

5. Create a rib.

- Create a rib using the sketch created in step 4 as the profile and the sketch created in step 3 as the guide.

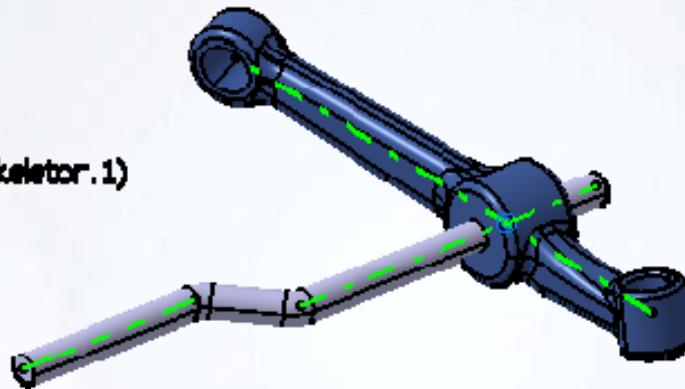


6. Close the assembly without saving it.

Exercise Recap: Contextual Part

- ✓ Insert a new part
- ✓ Create a contextual part using published elements

Bx11D
└ SuspensionArm_Skeleton (SuspensionArm_Skeleton.1)
└ LeftControlArm (LeftControlArm.1)
└ TensionStrut (TensionStrut.1)
Applications



Case Study: Designing in Context

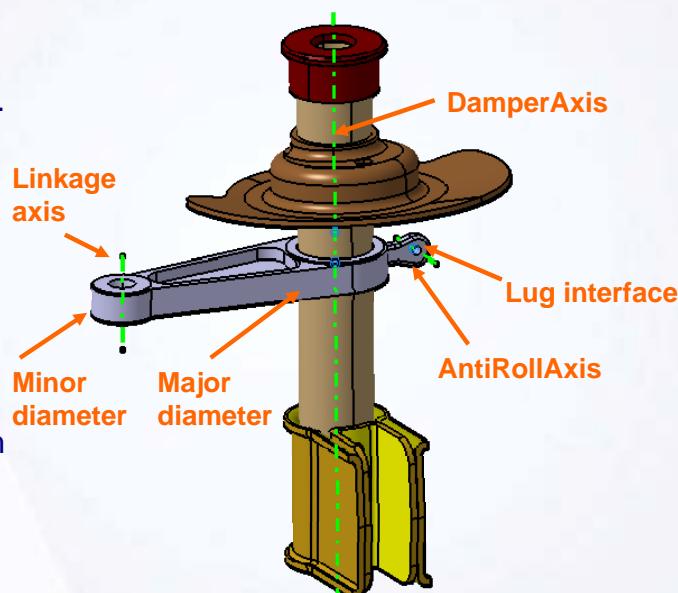
Recap Exercise



20 min

In this exercise you will contextually create the case study model. Let us recall the design intent of this model:

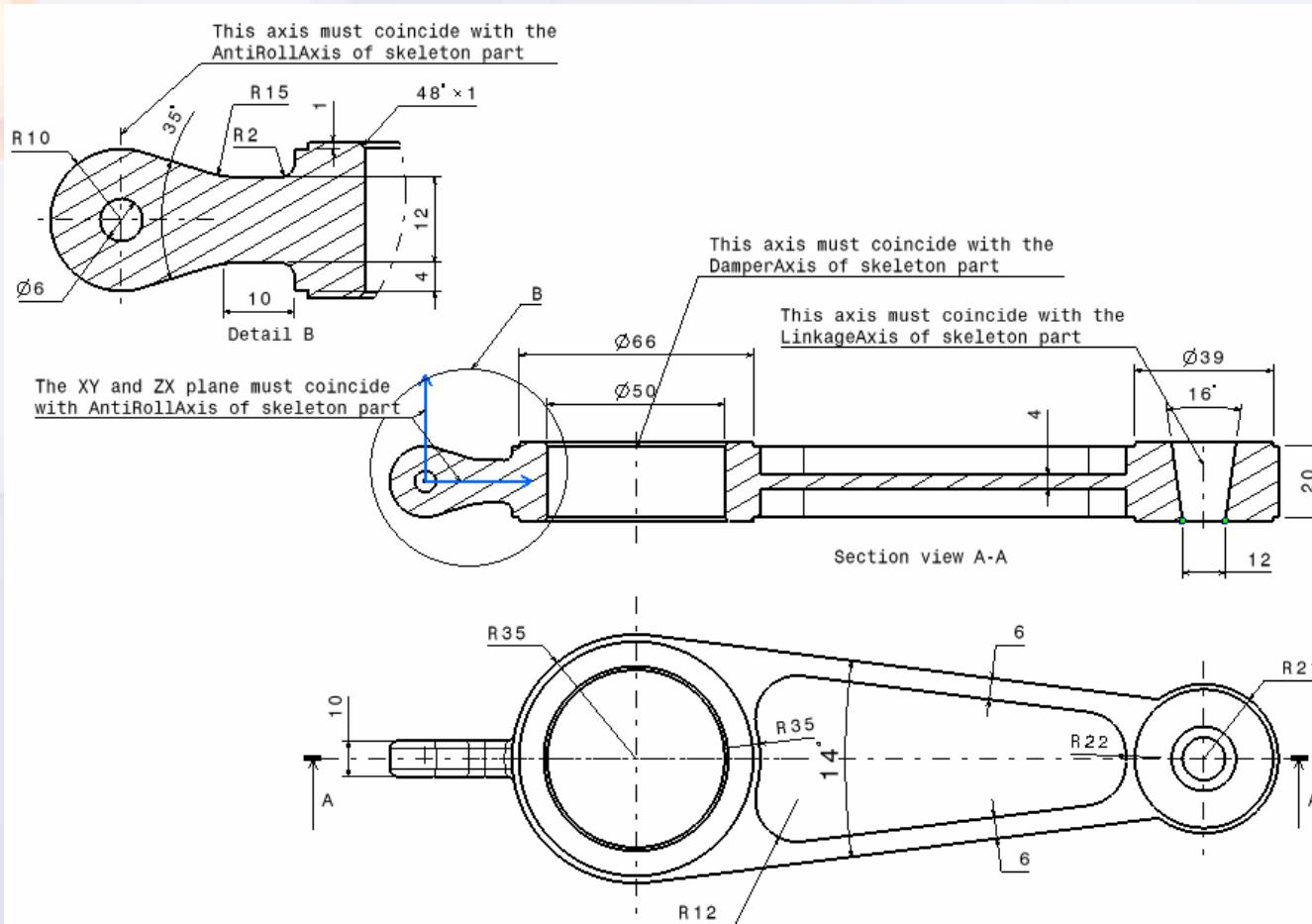
- ✓ Design the Arm contextually using published elements of the skeleton model. (DamperAssembly_Skeleton.CATPart).
 - The axis of major diameter of the Arm must coincide with DamperAxis of the skeleton part.
 - The axis of minor diameter of the Arm must coincide with LinkageAxis of the skeleton part.
 - The lug interface of the Arm must coincide with AntiRollAxis of the skeleton part.
- ✓ Position the Arm with reference to the skeleton part.
 - The bottom face of the Arm must be more than 200mm from the PillarSupportPlane of skeleton part.
- ✓ Modify the axis spacing between the Pillar and AntiRollAxis to > 61mm.



Using the techniques you have learned so far, create the model without detailed instructions.

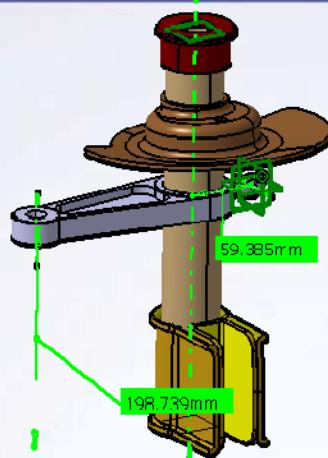
Do It Yourself: Create the Arm Part contextually (1/2)

Load the Automobile_CaseStudy11.CATProduct. Create a new contextual part (Arm.CATPart) in DamperAssembly using the drawing provided here.

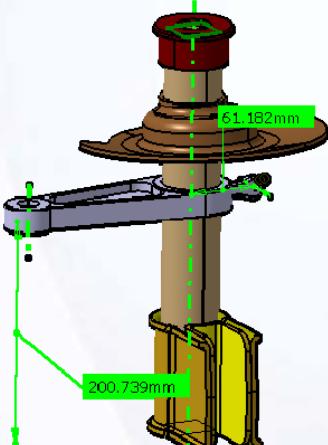


Do It Yourself: Create the Arm Part contextually (2/2)

A measured length will show that the distance between the Pillar bottom plane and the Arm lower surface is approx. 198mm (<200mm).

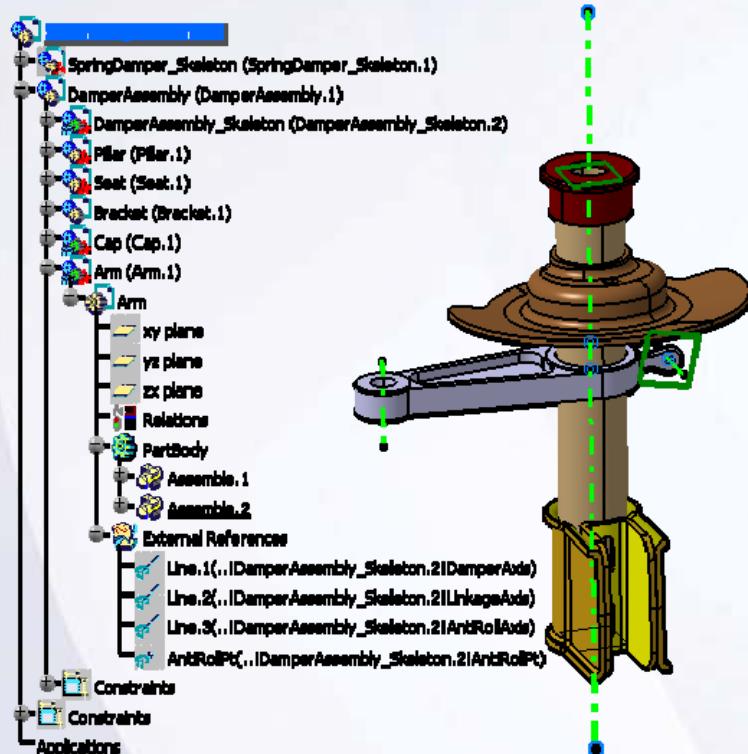


The modification can be an adjustment to the Z coordinate (+2/3mm) of the AntiRollPt to increase the measured dimension to >200mm, and an increase in X coordinate to 92mm to increase spacing between the Pillar and AntiRollAxis.



Case Study Recap: Arm Part

- ✓ Insertion a new part to the assembly
- ✓ Create a component in context using the published external references
- ✓ Position the part with reference to the skeleton part
- ✓ Observe the impact of modification of the skeleton part



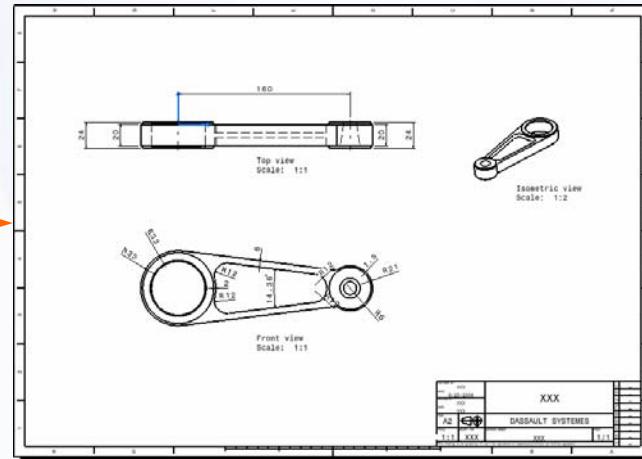
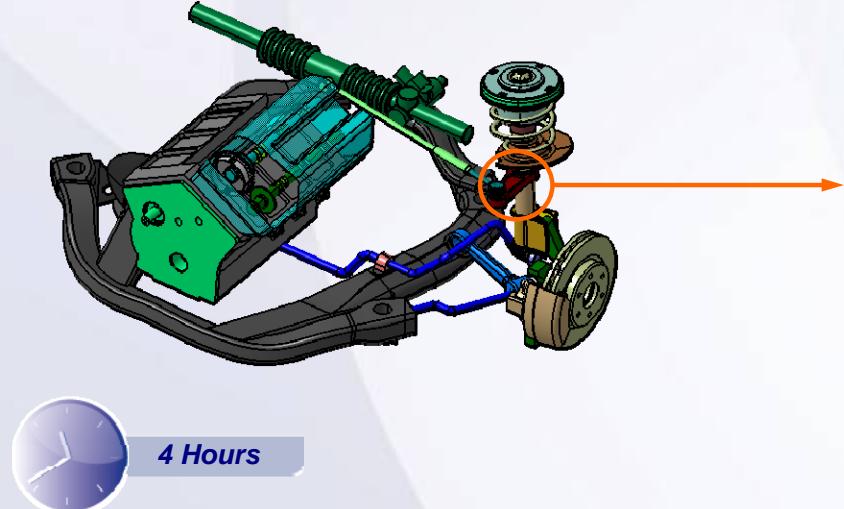
Drafting

12

Learning Objectives

Upon completion of this lesson you will able to:

- ✓ Start a New Drawing
 - ✓ Create Views
 - ✓ Create Dimensions and Annotations
 - ✓ Create Additional Views
 - ✓ View Modifications
 - ✓ Save the Drawing
 - ✓ Print the Drawing



Case Study

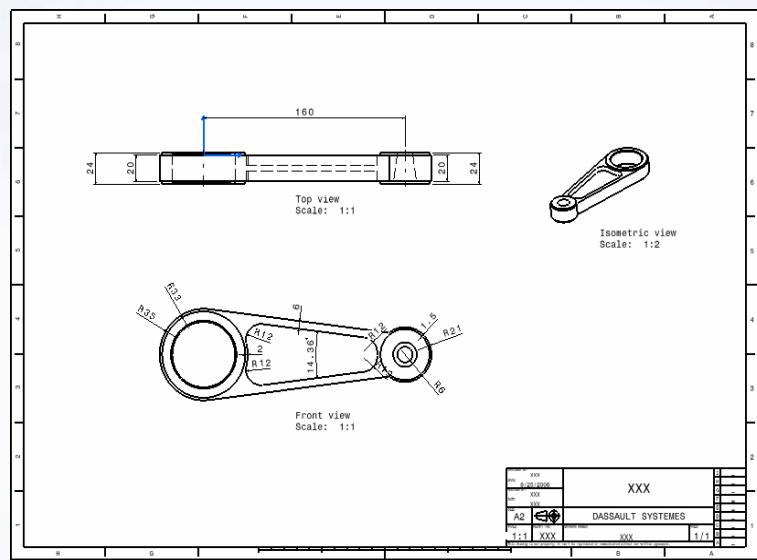
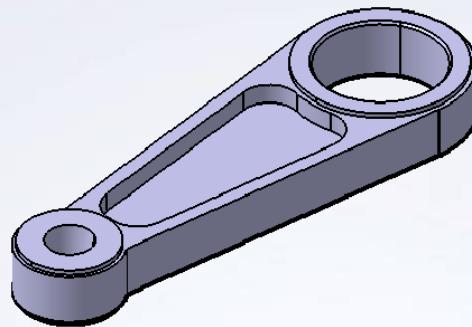
The case study for this lesson is the Arm used in the Front Suspension and Engine assembly.

Design Intent

- ✓ The drawing should be created using an ISO standard.
- ✓ The drawing should contain one view that shows hidden lines and axis.
- ✓ The drawing should contain a title block.

Stages in the Process

1. Start a new drawing.
2. Apply a title block.
3. Create views.
4. Create dimensions and annotations.



Start a New Drawing

The 3D environment gives designers a very efficient and flexible tool to create parts and assemblies. However, it is often necessary to communicate the manufacturing information with 2D drawings. Once a new drawing is started you are prompted to define the properties of the drawing.

Sample CATDrawing files corresponding to the organization standards can be stored at the central location. These files contain Title Blocks of the organization and drafting standards

To use these files to start new drawings, select “File > New from” command.



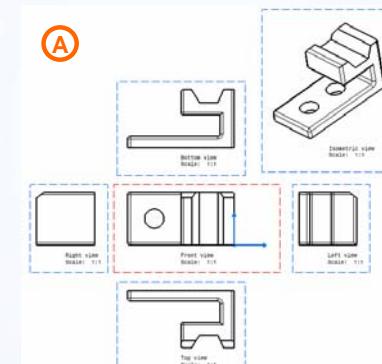
Create Views

Views represent different orientations of a part, which help to convey its design intent. Two types of views can be created in CATIA:

A. Associative: (linked to 3D models), which are called Generated Views.

B. Non-associative: (not linked to 3D models), which are called Draw Views.

The View Wizard enables you to quickly create different views in one operation.



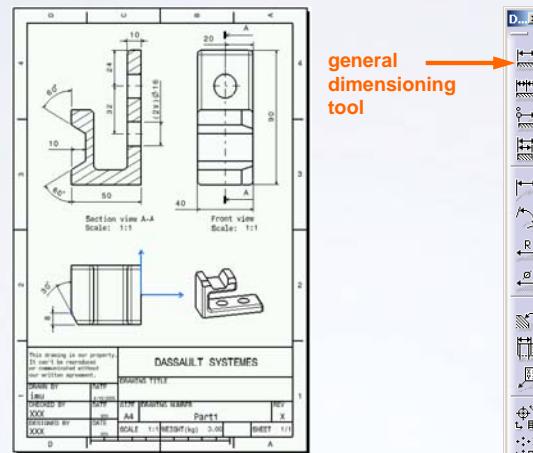
Create Dimensions and Annotations

Dimensions define the size and functional intent of a part and are often used to create a fabrication drawing for a manufacturer.

You can create dimensions either using tools dedicated to the type of dimension you want to create; length/distance, angular, radius, diameter, etc. or you can use general dimensioning tool, CATIA interprets the elements you select, and creates a Length/ Distance, Angular, Radius, or Diameter dimension "automatically" for you.

You can control the display of dimensions using the Dimension Properties such as dimension line style, tolerance formats, etc. and Numerical Properties such as numerical display, precision.

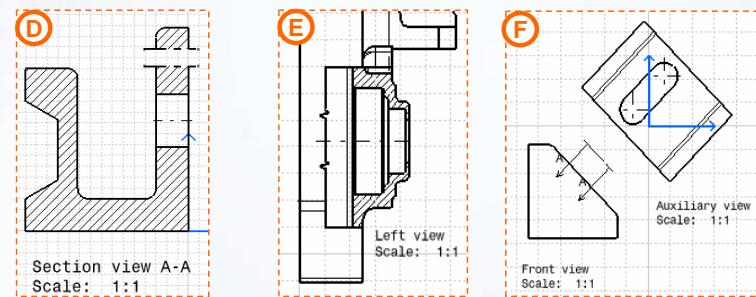
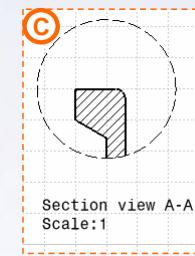
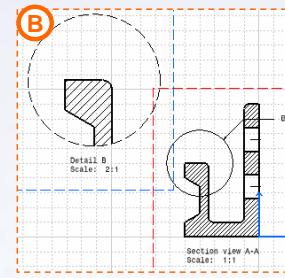
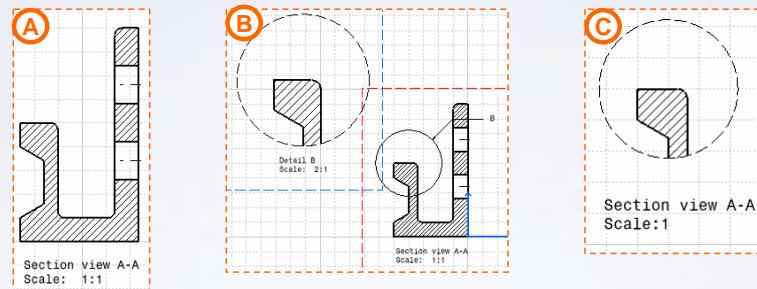
In addition to creating dimensions in a drawing, you can add notes and annotations to it using Text Toolbar.



Create Additional Views

Secondary Views are added to improve the clarity of the description of a part through better visualization and/or to aid in dimensioning.

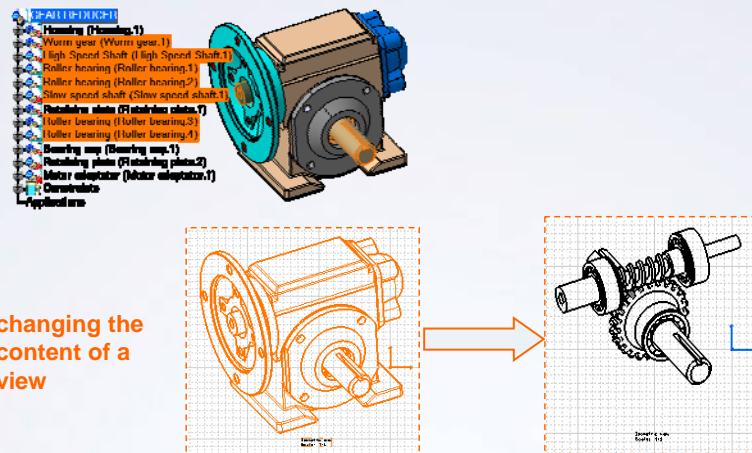
- A. Section View:** created by cutting the solid by section plane
- B. Detail View:** created by defining a "callout" on an existing view around the area to be enlarged, creates new view
- C. Clipping View:** created by defining a "callout" on an existing view around the area to be enlarged, modifies existing view
- D. Broken View:** defined by adding break lines to determine an area of the view that will be removed
- E. Breakout View:** created by cutting the solid locally order to see the inside of a part
- F. Auxiliary View:** created in a given direction that cannot be obtained with a standard view



View Modifications

To modify a view use the following steps:

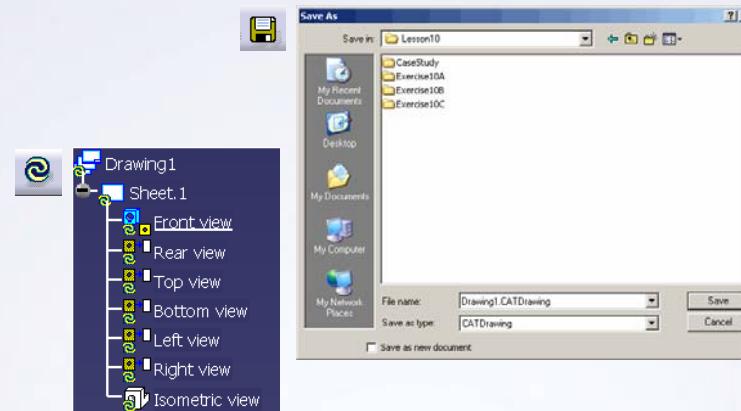
- A. To modify the position of a view: Select the view frame and drag to move it to another location.
- B. To delete the unnecessary views: Select the view frame and select Edit > Delete.
- C. To modify the properties of a view: Select the view frame and select Properties from the contextual menu.
- D. To change the content of a view: Select the view frame and select Modify Links from contextual menu.
- E. To change the definition of the projection plane of a view: select Modify Projection Plane from the contextual menu.
- F. To change the section profile definition use the Edit /Replace toolbar.



Save the Drawing

Before saving any drawing make sure that it's updated. If the Update icon is highlighted, it implies that the drawing needs to be updated to reflect the changes made to the corresponding 3D model.

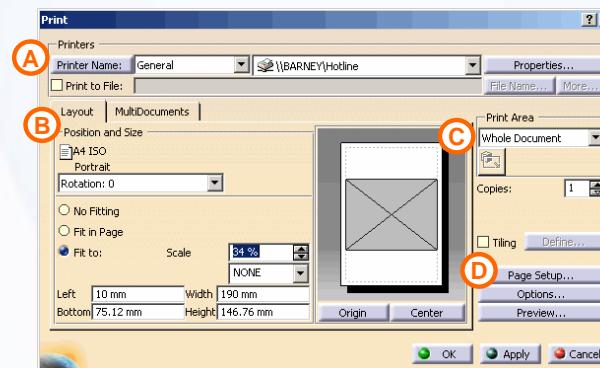
It is possible that a drawing may be opened without loading its referenced documents in the session. This could be caused by a missing file or a global CATIA setting, the tree identifies this with broken icons. In order to update the drawing correctly the links of the drawing have to be verified.



Print the Drawing

The Print window enables you to customize the layout, page setup, and options. The Print window contains the following information, which you can modify:

- A. Printer: Select the printer or key in the file name of printer.
- B. Position and Size: Define the position and size of the geometry on the page.
- C. Print Area: Define the area to print.
- D. Page Setup: Define the page size and characteristics.



Drafting Tools

Drafting Toolbars

- 1 **Views:** create different kinds of views
- 2 **Drawing:** create sheets, views, 2D components and frame title blocks
- 3 **Dimensioning:** create all types of dimensions needed to complete drawing
- 4 **Dimension Generation:** generate dimensions and balloons
- 5 **Annotations:** add annotations to existing views
- 6 **Dress-Up:** add dress-up elements on the drawing
- 7 **Geometry Creation:** create 2D geometry elements such as points, lines, planes, circles etc.
- 8 **Geometry Modifications:** transform existing 2D elements and add constraints to elements on the drawing



Exercise: Drawing Creation

Recap Exercise

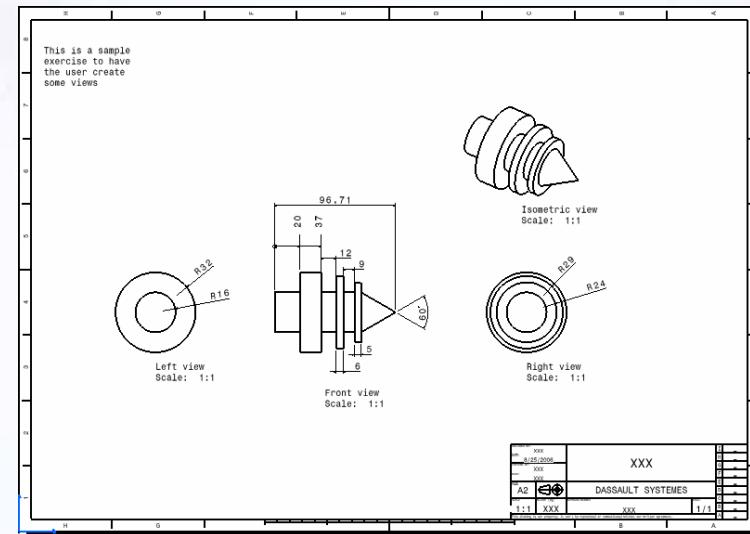


20 min

In this exercise you will create a drawing using ISO standard. High-level instructions for this exercise are provided.

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

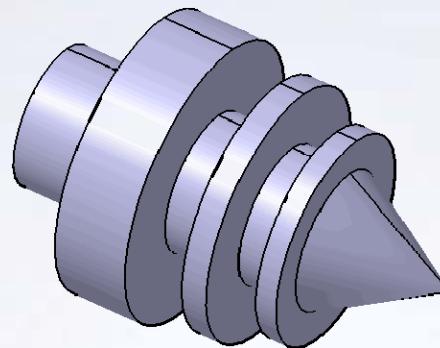
- Create a drawing
- Insert a title block using catalog
- Create views using the view wizard
- Move and delete views
- Dimension geometry



Do it Yourself (1/3)

1. Load Ex12B.CATPart .

①

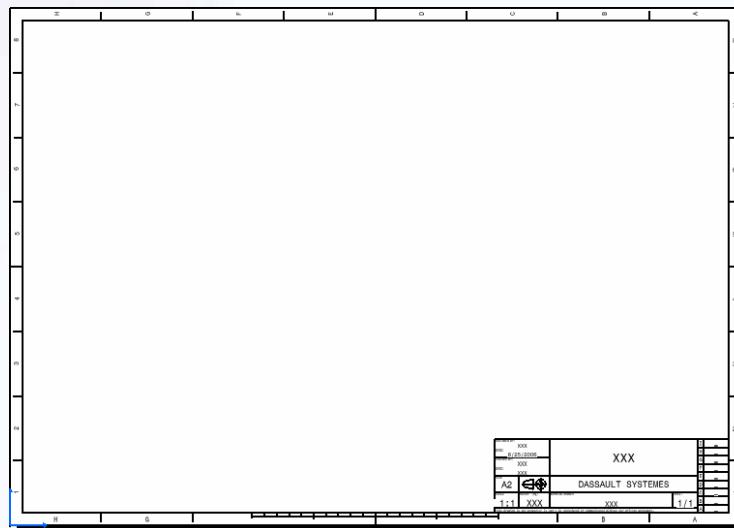


2. Create a new drawing.

- Use the A2 and standard ISO drawing size.

3. Insert a title block using catalog.

- Insert ISO_A2 from Catalog_Title_Blocks.catalog.

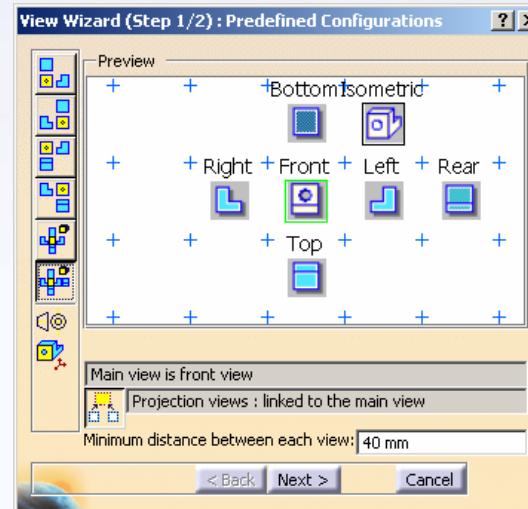


Do it Yourself (2/3)

4. Use the view wizard to create views.

- Place the pre-defined layout of Configuration 6 with a third angle projection.

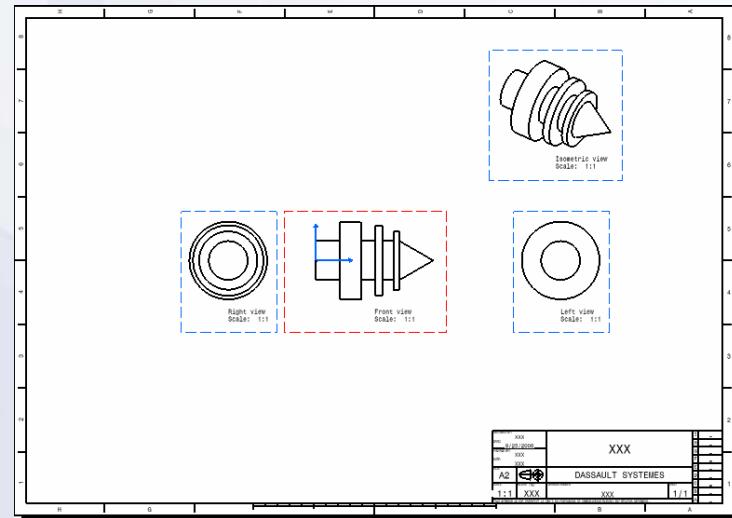
4



5. Move and delete some views.

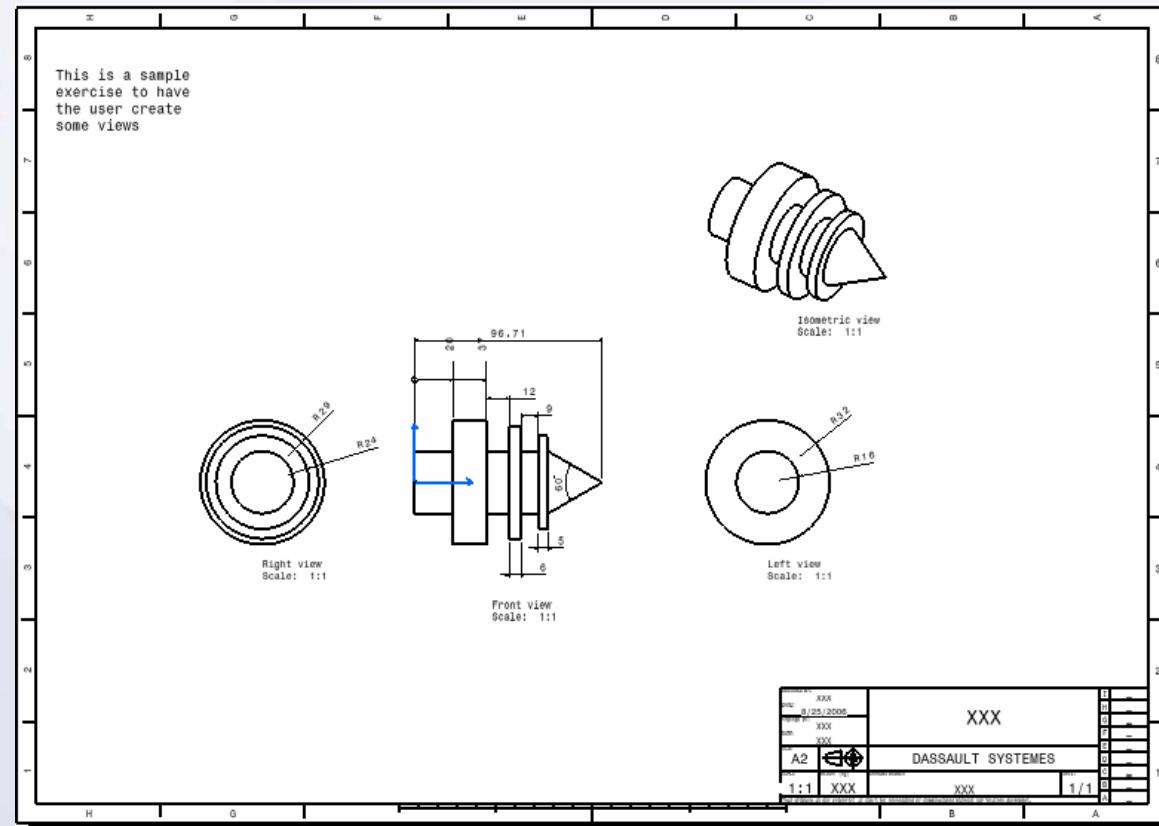
- Delete the top, bottom, and rear views.
- Position the views so that they appear evenly spaced out in the drawing.

5



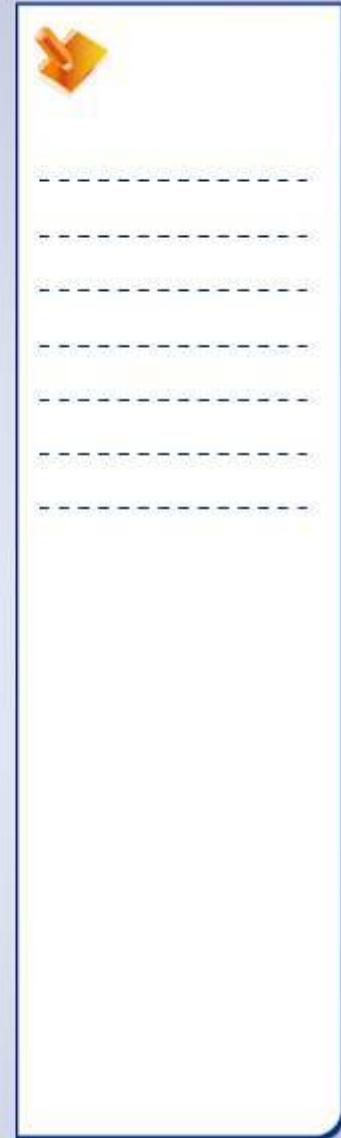
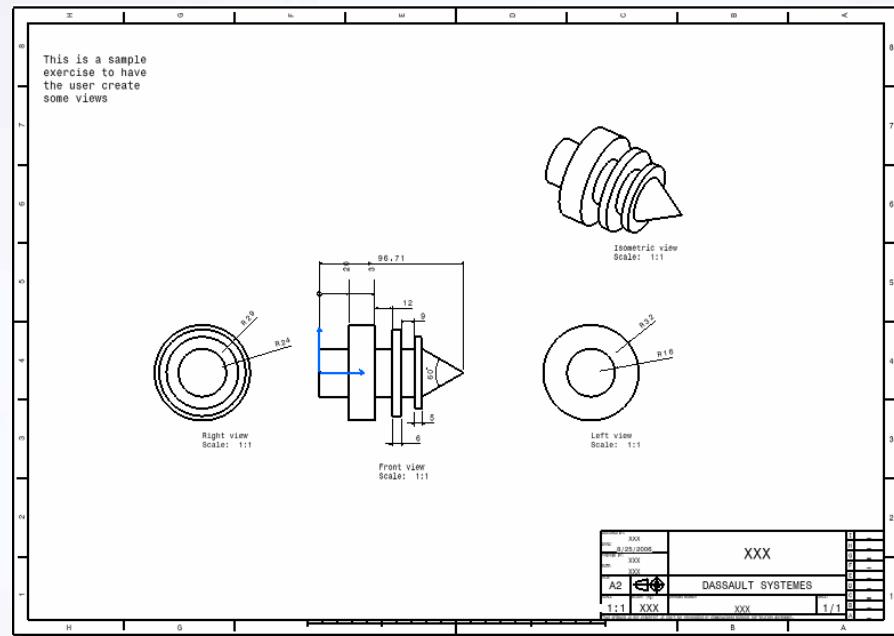
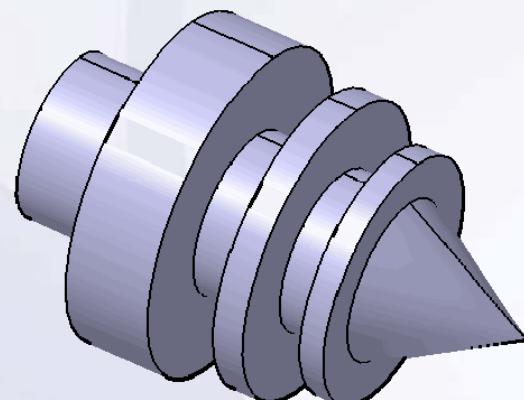
Do it Yourself (3/3)

6. Dimension and annotate the drawing as shown.
7. Close the drawing without saving.



Exercise Recap: Drawing Creation

- ✓ Create a drawing
- ✓ Insert a title block
- ✓ Create views using the view wizard
- ✓ Move and delete views
- ✓ Dimension geometry



Exercise: Additional Views and Dimensions

Recap Exercise

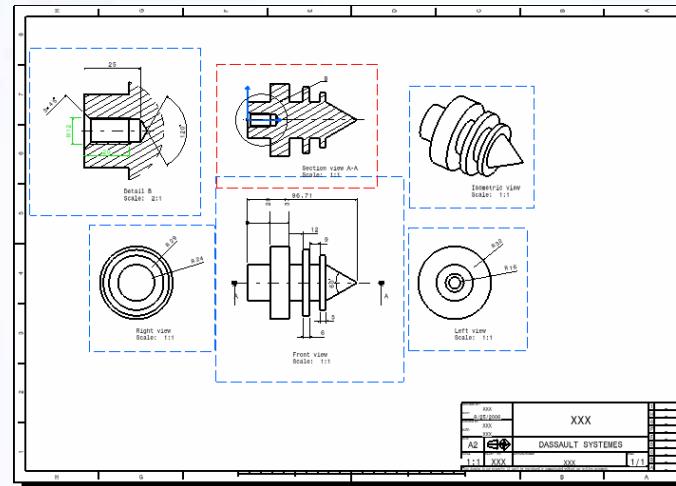


15 min

In this exercise you will modify the existing drawing. High-level instructions for this exercise are provided.

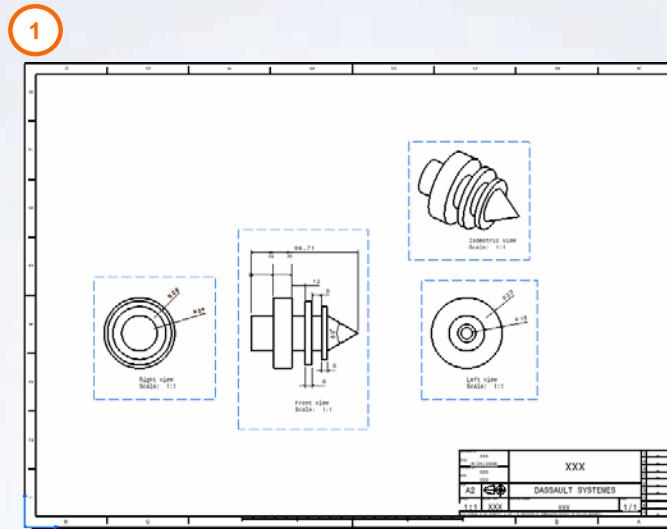
By the end of this exercise you will be able to:

- Create a section view
 - Create a detail view
 - Create a thread dimension
 - Create a chamfer dimension



Do it Yourself (1/2)

1. Load Exercise12D.CATDrawing.

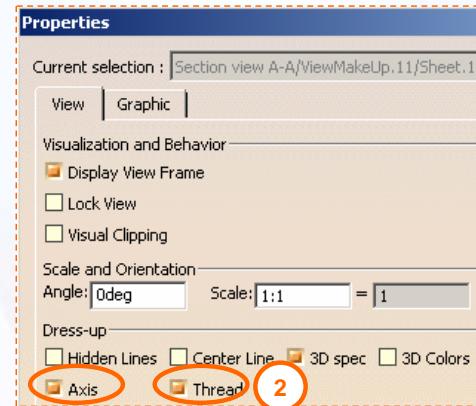
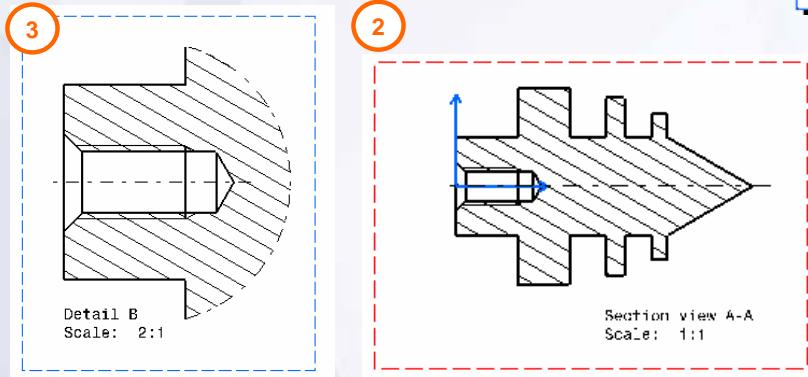


2. Create a Section view.

- Create section view from front view.
- Change Properties of the view.

3. Create a detail view.

- Create a detail view of the threaded hole.



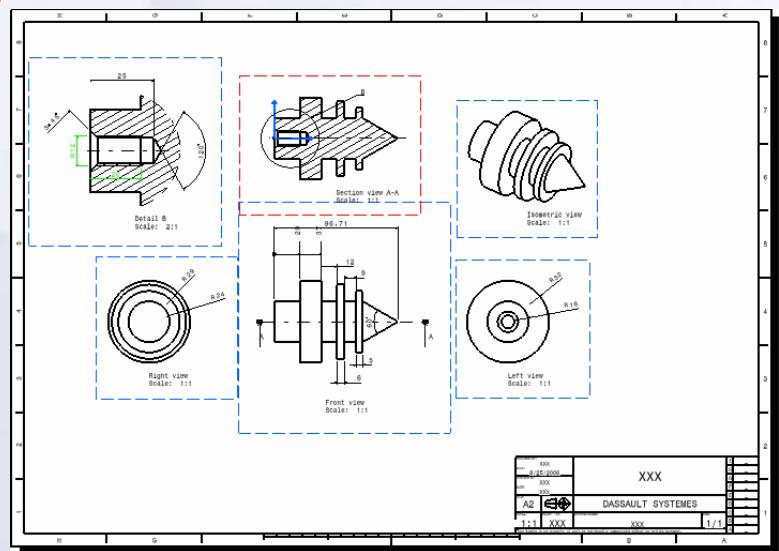
Do it Yourself (2/2)

4. Dimension the threaded hole.

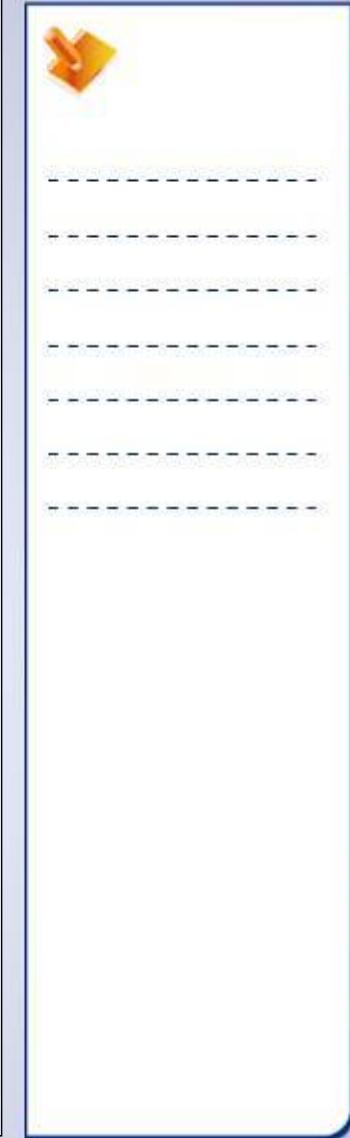
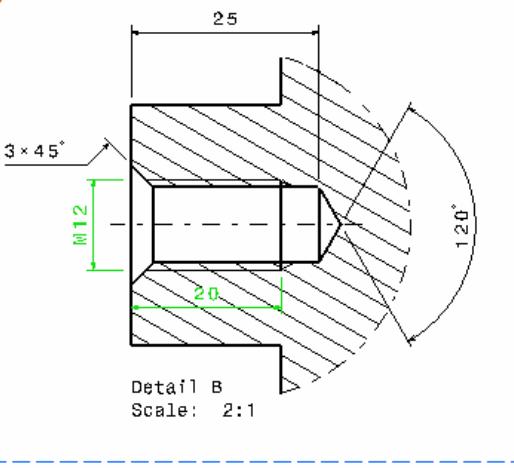
- Add thread chamfer angle and length dimensions to the detailed view.

5. Close the file without saving it.

5

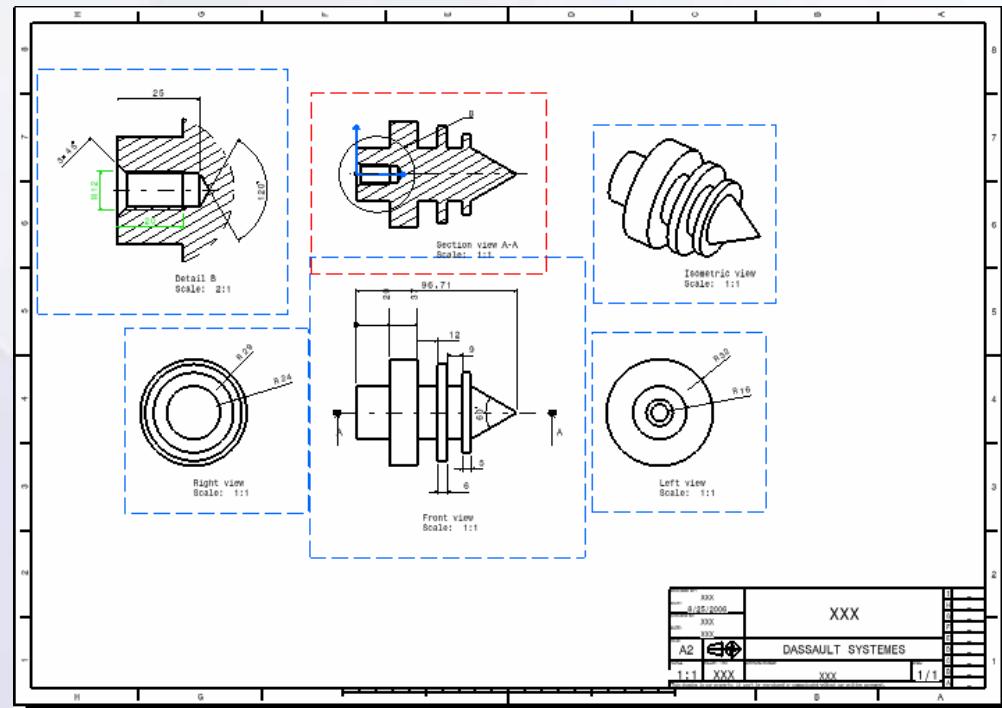


4



Exercise Recap: Additional Views and Dimensions

- ✓ Create a section view
- ✓ Create a detail view
- ✓ Create a thread dimension
- ✓ Create a chamfer dimension



Case Study: Drafting (ISO)

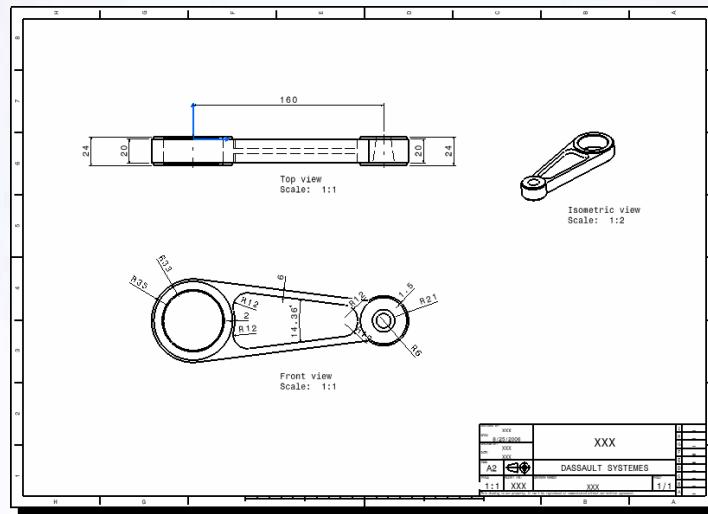
Recap Exercise



10 min

In this exercise you will create the case study drawing. Let us recall the drawing requirements:

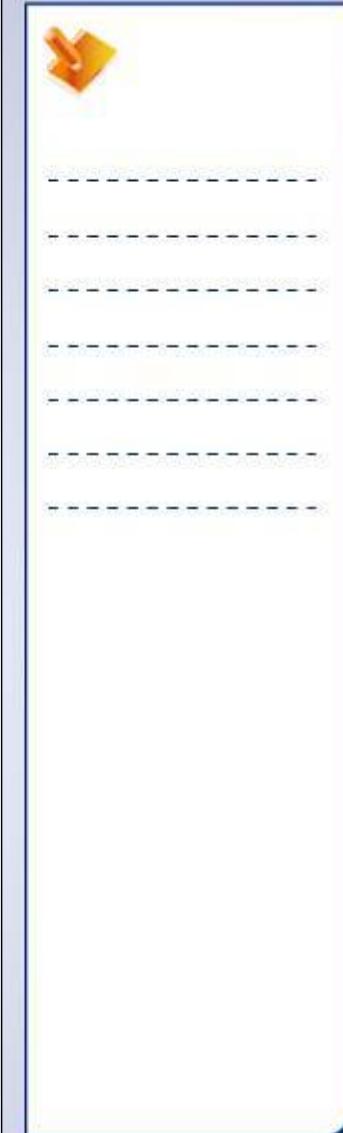
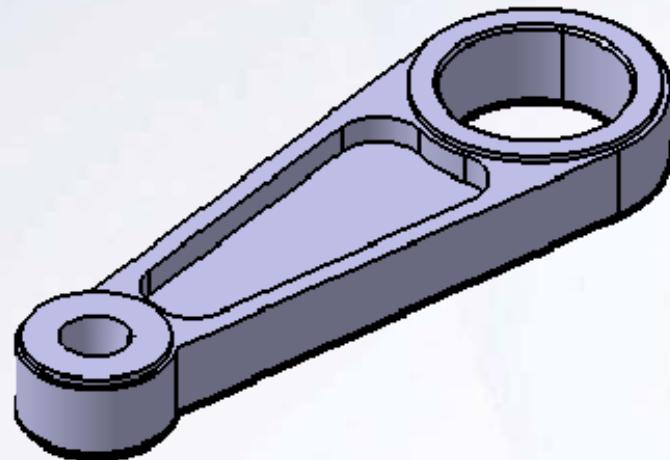
- ✓ The drawing should be created using an ISO standard.
 - Standards are pre-defined formats for dimensions, annotations, and views, which provide a consistent interpretation of information.
 - ✓ The drawing should contain one view that shows hidden lines and axis.
 - The display of these items in a single view enables a better understanding of the model by showing depth and internal features.
 - ✓ The drawing should contain a title block.
 - This is typically required with any drawing. Title block must be instantiated from catalog.



Using the techniques you have learned so far, create the drawing of the model without detailed instructions.

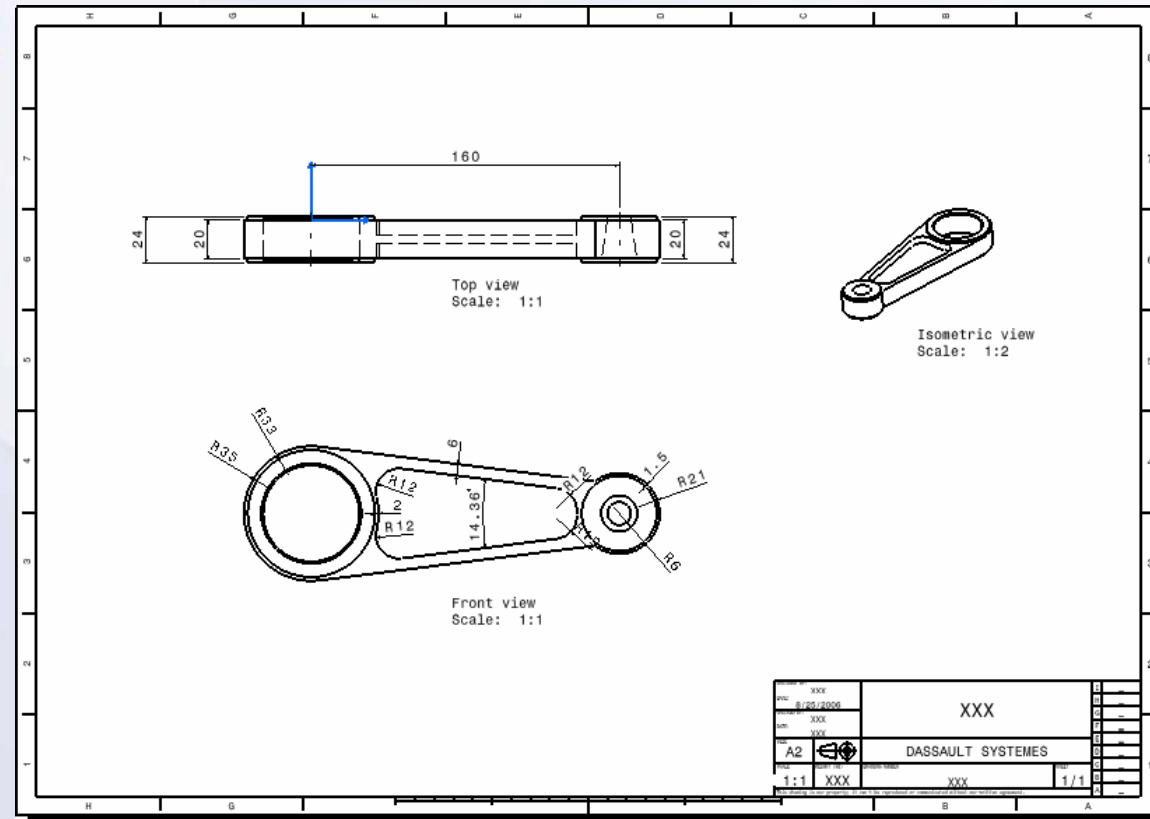
Do It Yourself: Create the Drawing of Arm

1. Load CaseStudy12_ISO.CATPart and create a 'A2' size drawing.
2. Instantiate a title block using the Catalog_Title_Blocks.catalog.



Case Study Recap: Drawing of Arm

- ✓ Create a drawing using ISO standard.
- ✓ One view must show the hidden lines and axis.
- ✓ Title block must be instantiated from a catalog.



Master Project

Door Assembly

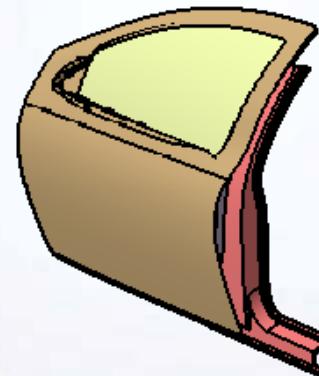


30 min

The objectives of this project are first to create a part, add it to an assembly and position it with constraints; then to create a new part in the context of an assembly. Finally, you will replace the input curves, manage update errors, and create an assembly drawing. The assembly used in this project is the Door assembly. You have created several components used in this assembly in different case studies throughout the course.

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

- Create a part using solid modeling
- Create an assembly and position parts with constraints
- Create a surface model in the context of the assembly
- Replace input curves of the part and update the assembly
- Create an assembly drawing

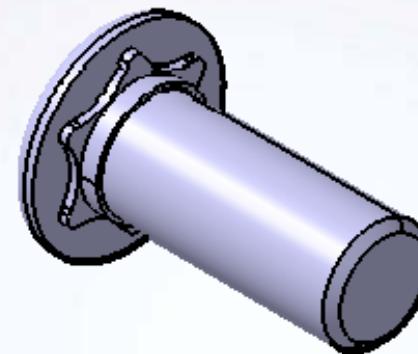


Master Project: Overview (1/4)

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

1. Create a part.

- Create the window support part (Window_Pin). This part uses the features learned in this course.
- Design Intent:
 - ✓ A stable base feature must be determined.
 - ✓ The model must be created symmetrically about the XY and YZ plane.
 - ✓ The model must be created using the multi-body method
 - ✓ Fillets and chamfers must be created as separate features (they cannot be created within the profile sketches).
 - ✓ Avoid complicated profiles.



Master Project: Overview (2/4)

2. Create the assembly.

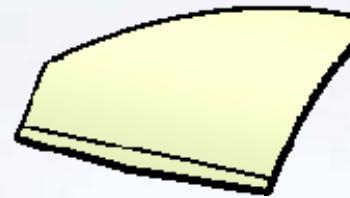
- Insert the existing components (Window_Clip and Window_Pin) in the Mechanism assembly and position them by constraining with reference to the skeleton part. Insert mechanism in the Window assembly and position it. Finally insert Window in Door assembly and position it.
- Design Intent:
 - ✓ Insert the existing components to Mechanism assembly.
 - ✓ Insert Mechanism in Window assembly.
 - ✓ Insert the Window in Door assembly.
 - ✓ All the components must be constrained with reference to the skeleton part.
 - ✓ Fully constrain all the components using axis system.



Master Project: Overview (3/4)

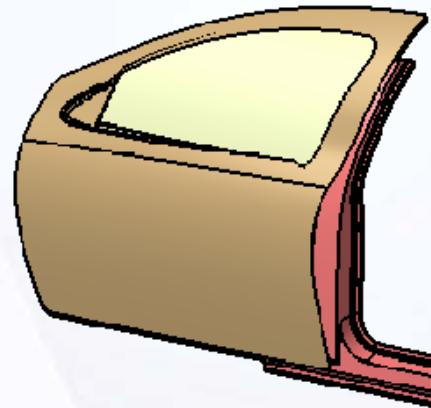
3. Create a contextual part.

- Create Window_Glass part in context to Door assembly .
- Design Intent:
 - ✓ The Window_Glass part must be in context of the skeleton part of the Door assembly.
 - ✓ The glass surface must be parallel to the edge of Front_Seal.
 - ✓ The profile of the Glass surface must match the Inner_Front_Seal.
 - ✓ The thickness must be 4mm.



4. Replace the input curves.

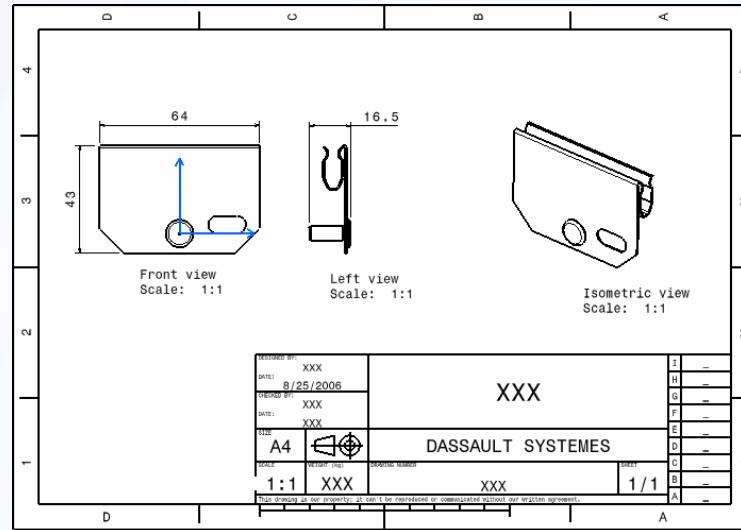
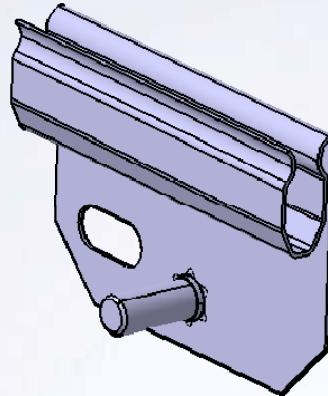
- Replace the input curves of Front_Seal and manage the update errors.
- Design Intent:
 - ✓ The input curve Front_Seal_Curve1 must be replaced by Front_Seal_CurveN1.
 - ✓ The input curve Front_Seal_Curve2 must be replaced by Front_Seal_CurveN2.
 - ✓ The Door assembly must be fully updated.



Master Project: Overview (4/4)

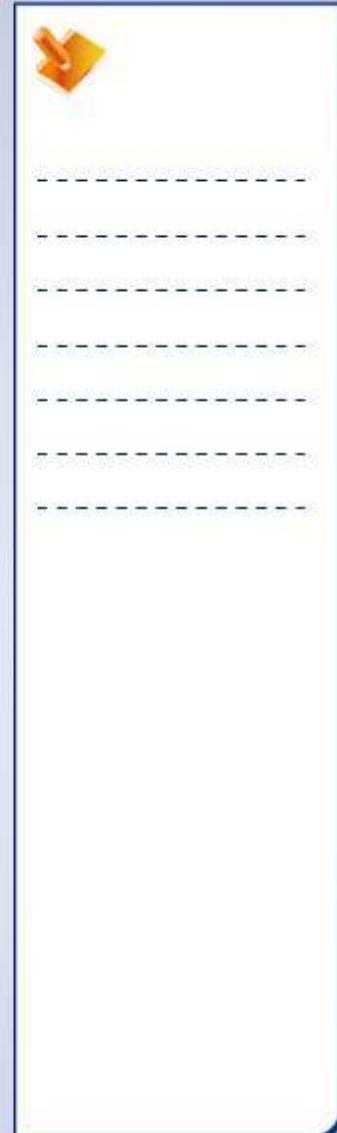
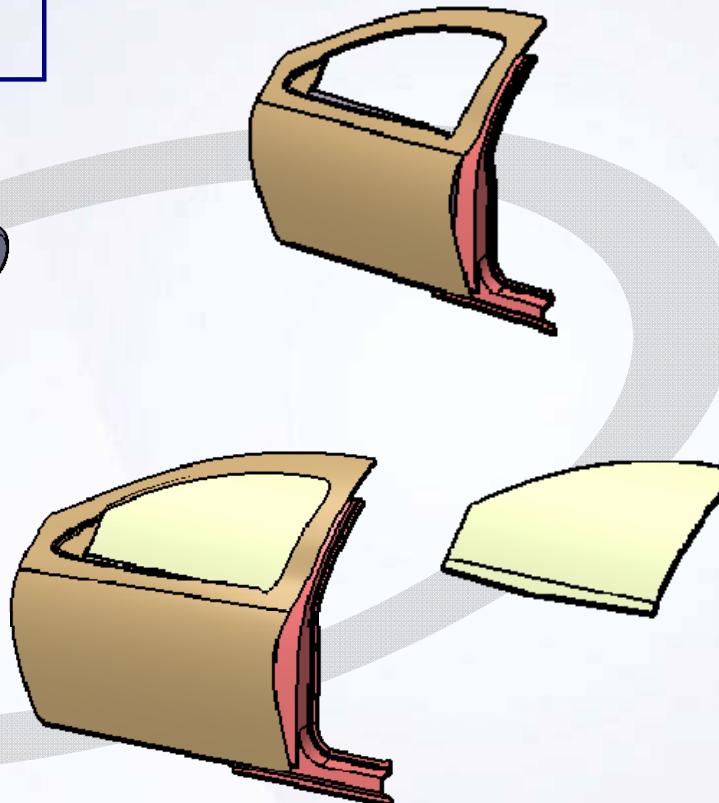
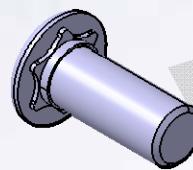
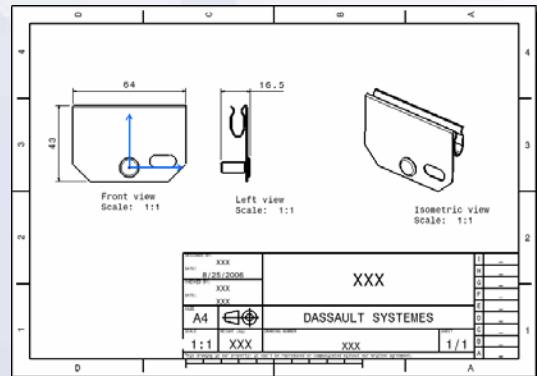
5. Create an assembly drawing.

- Create a simple assembly drawing of the Mechanism assembly.
- Design Intent:
 - ✓ Two main views of the drawing must be shown.
 - ✓ The overall dimensions for the Mechanism assembly must be displayed on the drawing.
 - ✓ The drawing must contain a title block.



Master Project: Overview Recap

- ✓ Create a part
- ✓ Create the assembly
- ✓ Create a contextual part
- ✓ Replace the input curves
- ✓ Create an assembly drawing



Master Project - Create New Part

Door Assembly

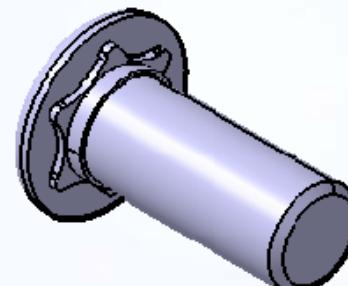


20 min

The objective of this step is to create the Window_Pin. This is a part of the Window Assembly. High-level instructions for this exercise are provided.

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

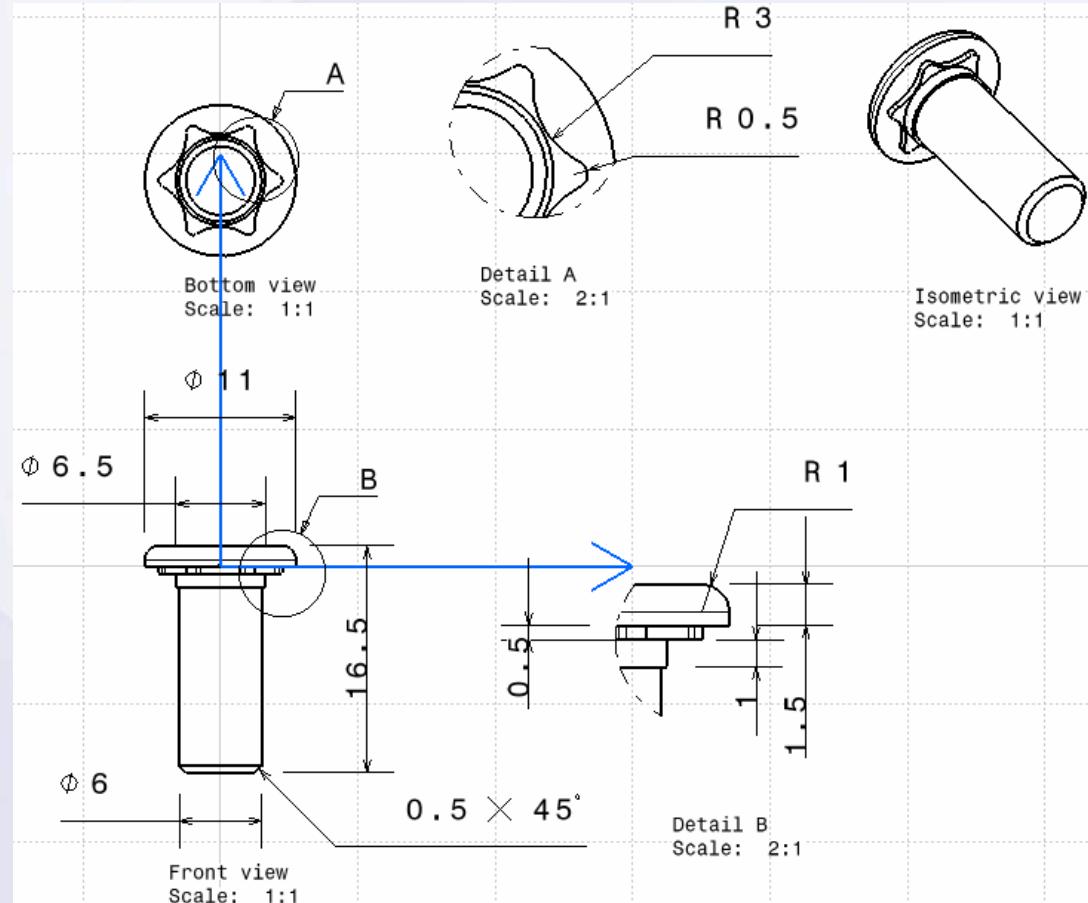
- Determine the best base feature
- Determine the best tool for each feature
- Save a file



Pin

Master Project: Create New Part (1/7)

Open the Window_Pin part and modify it using the drawing provided here.

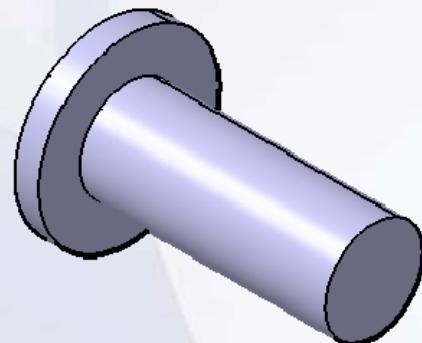
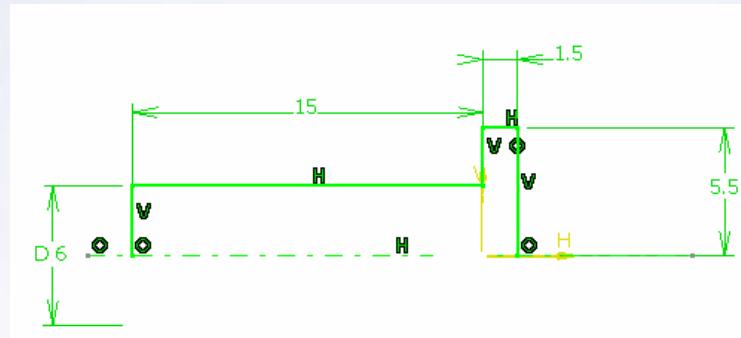


Refer to next slides for high-level instructions.

Master Project: Create New Part (2/7)

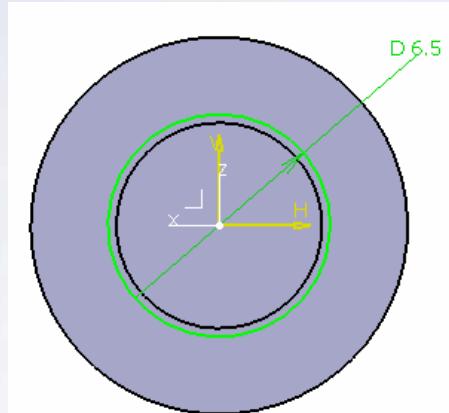
Here is a list of required tasks to create Window_Pin:

1. Create a base feature.
 - Open the window_Pin part.
 - Determine a stable base feature. For this part, a simple shaft shape is the best feature.
 - Create a default Axis System.
 - In the 'MainBody', create a shaft so that the pin head rests on the ZX plane of the Axis system. Use Y axis as the revolution axis.

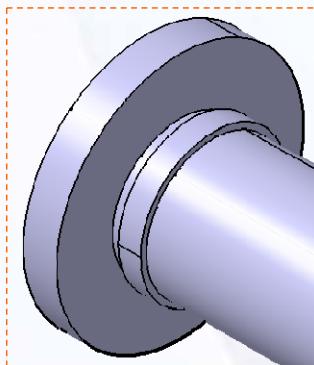
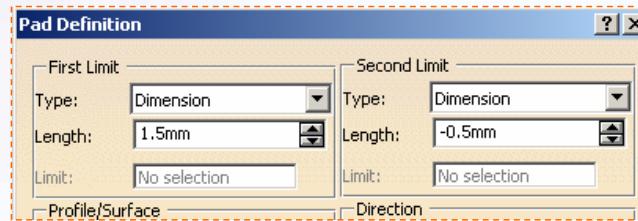


Master Project: Create New Part (3/7)

2. Create a sketch on the ZX plane of the Axis system.

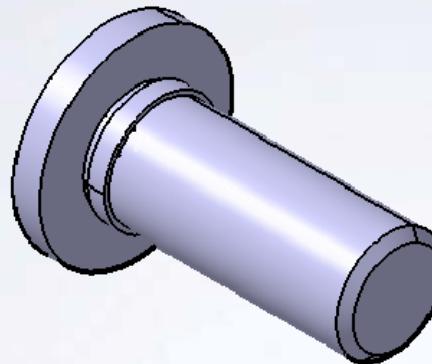


3. Create a pad as shown.

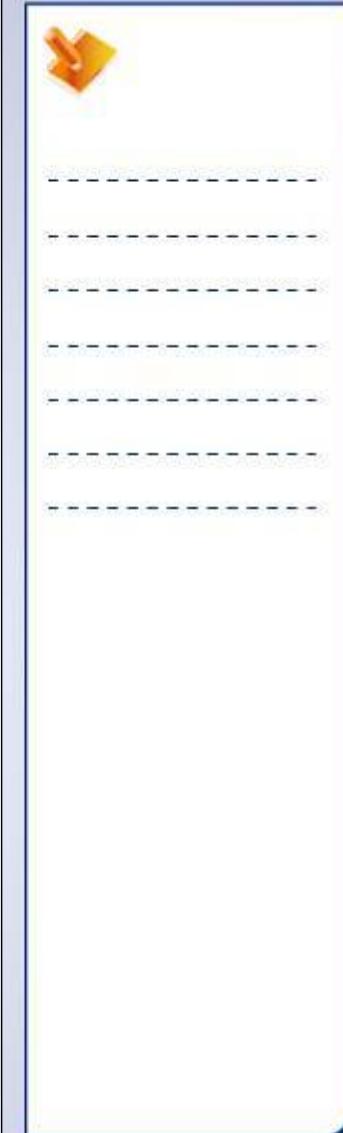
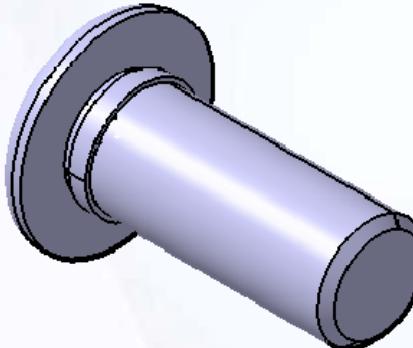


Master Project: Create New Part (4/7)

4. Create a chamfer of 0.5mm X 45 deg. as shown.

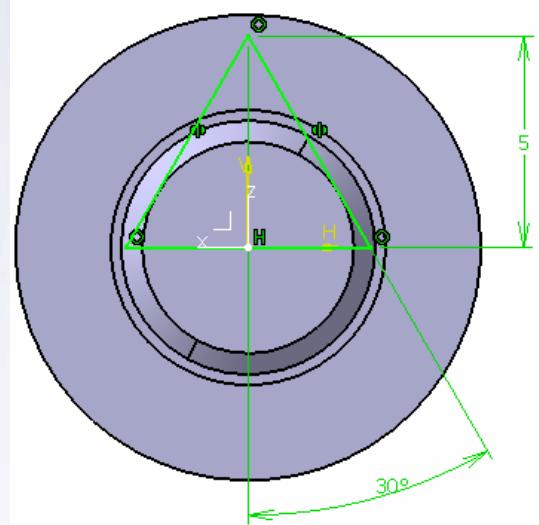


5. Create an edge fillet of 1mm as shown.

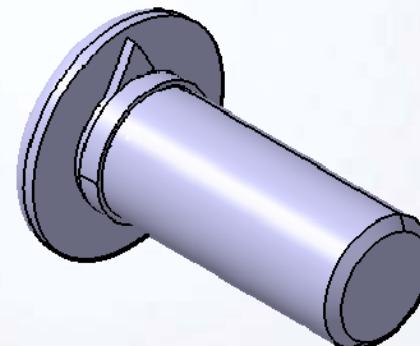
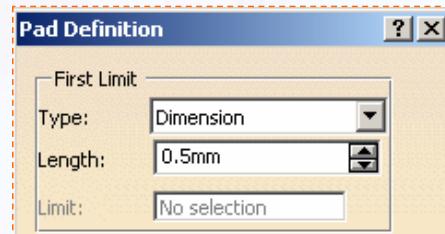


Master Project: Create New Part (5/7)

6. In the 'RibBody' create a sketch on the ZX plane of the Axis system.

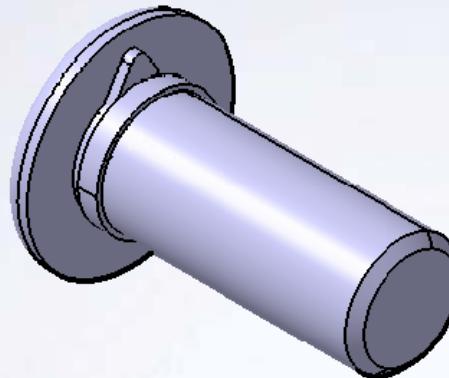


7. Create a pad as shown.

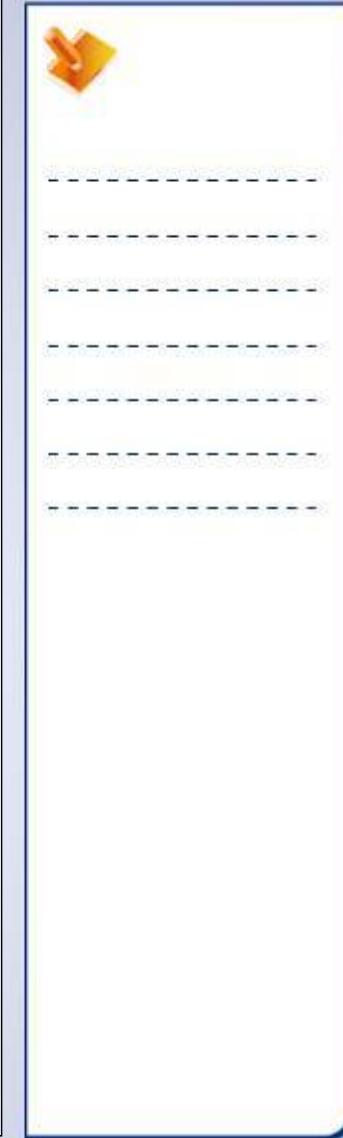
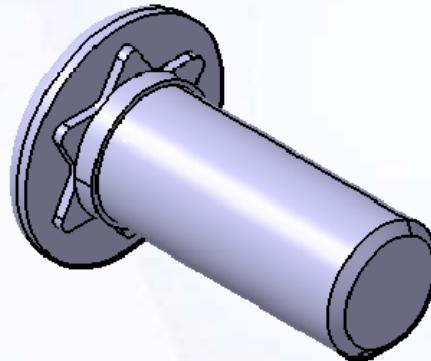


Master Project: Create New Part (6/7)

8. Create an edge fillet of 0.5mm as shown.

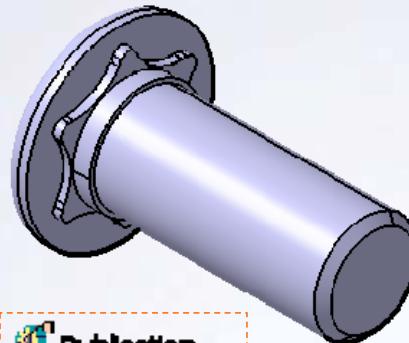


9. Create a circular pattern using the Y axis as the reference direction.

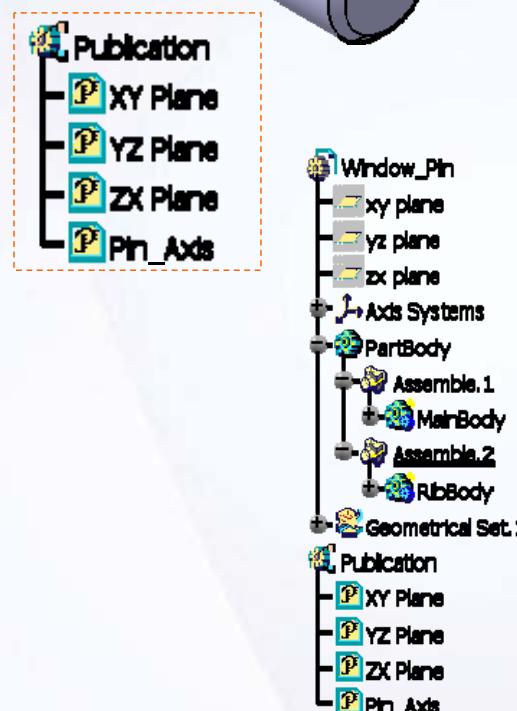


Master Project: Create New Part (7/7)

10. Create an edge fillet of 3mm on the six edges as shown.

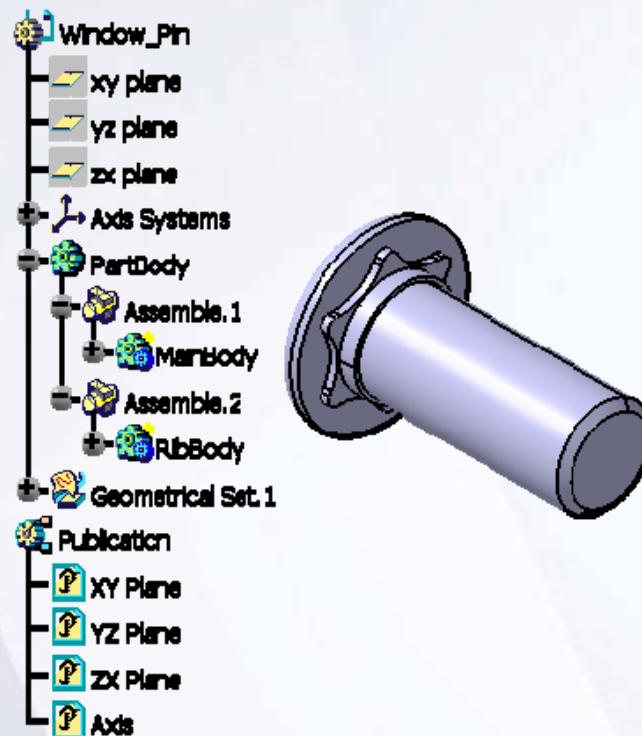


11. Publish the elements.
 - Publish the planes and axis of the part.
12. Save and close the part.



Master Project: Create New Part Recap

- ✓ Determine the best base feature
- ✓ Determine the best tool for each feature
- ✓ Save a file



Master Project - Create an Assembly

Door Assembly

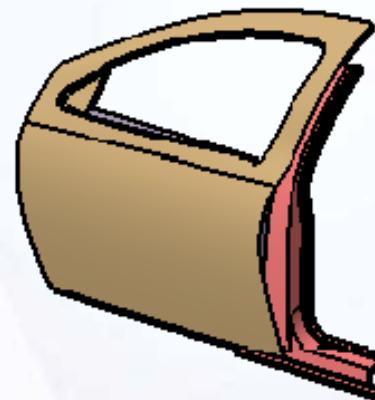


20 min

The objective of this step is to add parts to the Window assembly and Door assembly, and to position them with constraints. This assembly contains all the parts you have created in the case studies. High-level instructions are provided for this exercise.

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

- Add components to the assembly
- Create an instance of a component
- Fully constrain the assembly
- Modify display properties
- Save the file

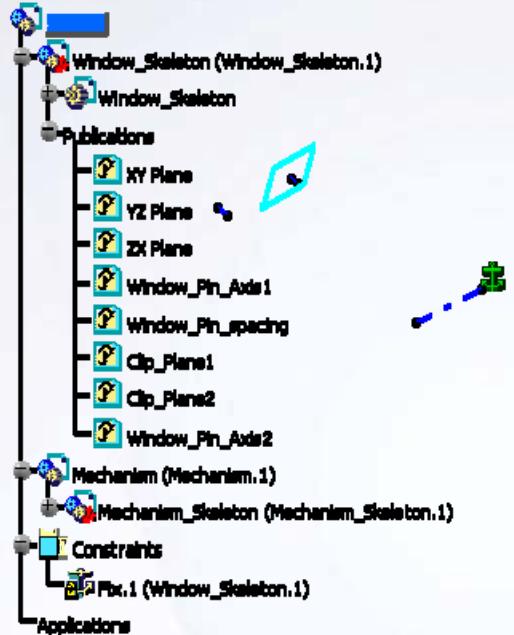


Master Project: Create an Assembly (1/6)

Here is a list of required tasks to guide you:

1. Load Window.CATProduct.
 - This assembly has skeleton part and Mechanism sub-assembly already created for you.
 - The Fix constraint is used for the skeleton part.

2. Edit Mechanism.CATProduct and insert the existing components.
 - Insert Window_Clip.CATpart.
 - Insert Window_Pin.CATPart created in step 1. If you did not complete step 1, insert Step1_end.CATPart.



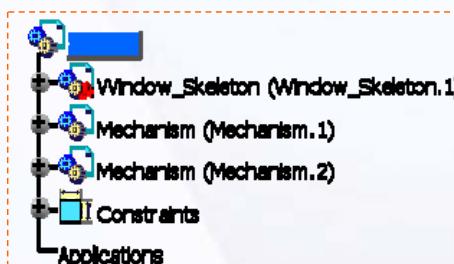
Master Project: Create an Assembly (2/6)

3. Assemble the component of Mechanism with respect to the skeleton part.

- Use published elements of Mechanism_Skeleton part for creating constraints.
- Fix the Mechanism_Skeleton part.
- Add two offset constraints of [0mm] between the ZX and XY planes of skeleton part and Window_Clip part.
- Add one coincidence constraint between Window_Pin_Axis of skeleton part and Window_Clip part.
- Add similar constraints (two offsets and one coincidence) for the Window_Pin part.

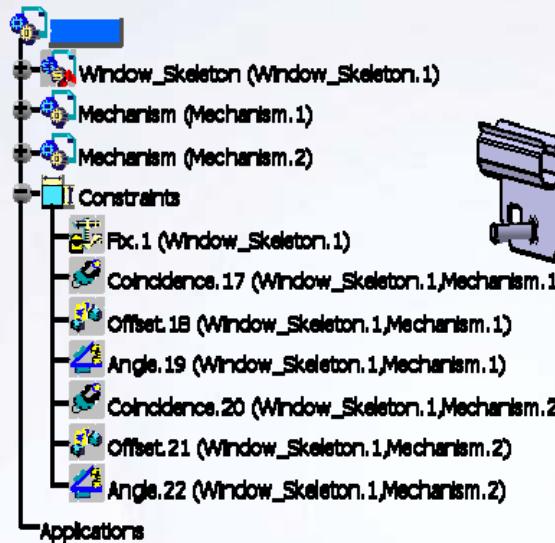


4. Create an instance of a Mechanism.CATProduct and assemble it with respect to the skeleton part.
- Edit Window product and copy-paste the Mechanism.CATProduct.



Master Project: Create an Assembly (3/6)

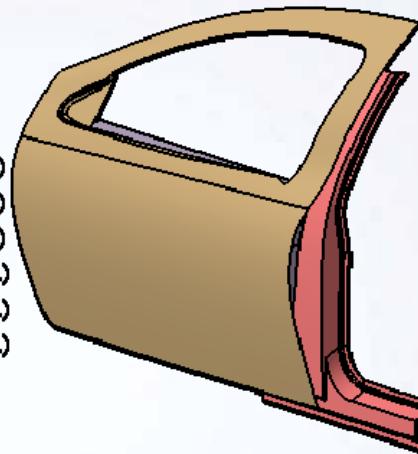
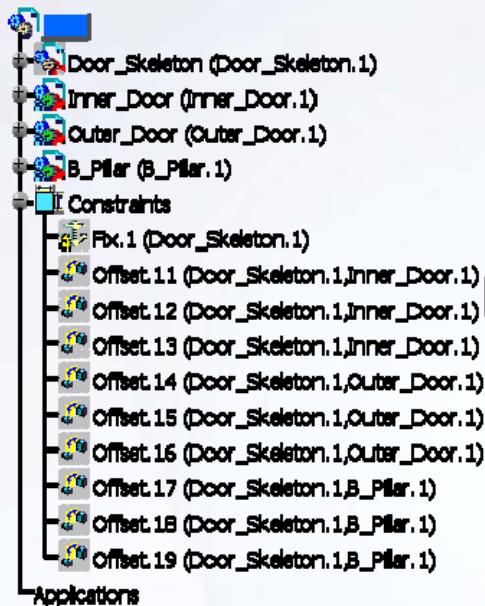
5. Assemble the components of Window with respect to the skeleton part.
 - Use the published elements of the Window_Skeleton part and Mechanism_Skeleton part for creating constraints.
 - Add one coincidence constraint between Window_Pin_Axis1 and Pin_Axis.
 - Add an offset constraint of [0mm] between Clip_Plane1 and ZX plane.
 - Add an angle constraint of [3deg] between YZ planes.
 - Add similar constraints (one coincidence, one offset and one angle) using the published elements of Mechanical_skeleton part of the Mechanism.2.
 - Save and close the Window.CATProduct.



Master Project: Create an Assembly (4/6)

6. Load Door.CATProduct.

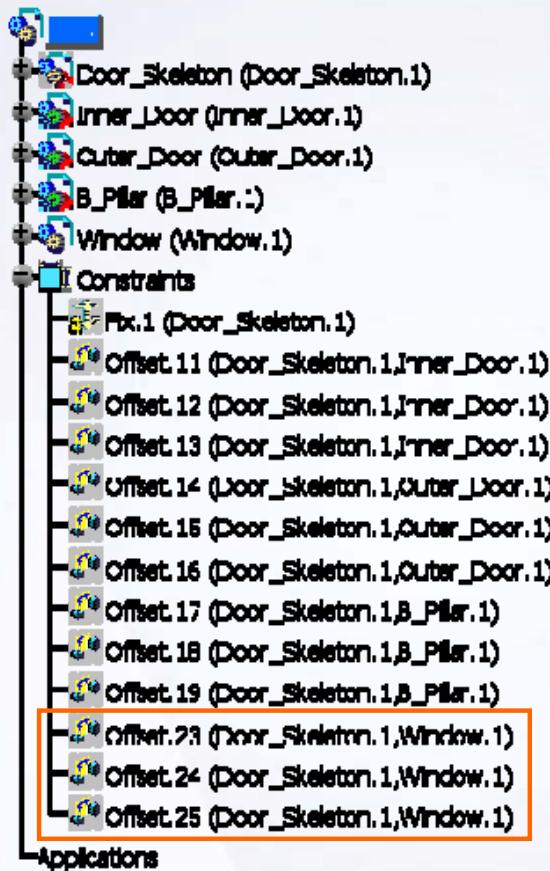
- This assembly has the skeleton part, Inner Door, Outer Door, and B-pillar part already created for you.
- The Fix constraint is used for the skeleton part.
- The Inner Door, Outer Door, and B-pillar part are already assembled and fully constrained.



Master Project: Create an Assembly (5/6)

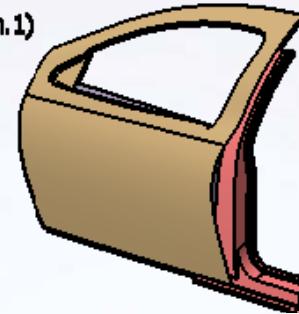
7. Insert an existing component to the Door assembly and assemble it with respect to the skeleton part.

- Insert Window.CATProduct created in slide 3/6. If you did not complete that step, insert Window_end.CATProduct.
- Use the published elements of Door_Skeleton part and Window_Skeleton part for creating constraints.
- Add three offset constraints of [0mm] between XY, YZ, and ZX planes of the Door_skeleton part and Window_Skeleton part.



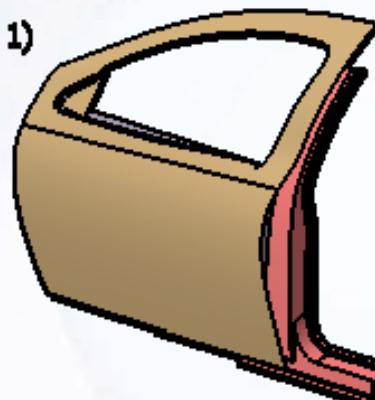
Master Project: Create an Assembly (6/6)

8. Modify the display.
 - Hide all the constraints
 - Hide any visible reference plane.
 - Hide any visible axis system.
9. Save the file.



Master Project: Create an Assembly Recap

- ✓ Add components to the assembly
- ✓ Create an instance of the component
- ✓ Fully constrain the assembly
- ✓ Modify the display properties
- ✓ Save the file



Master Project - Create a Contextual Part

Door Assembly

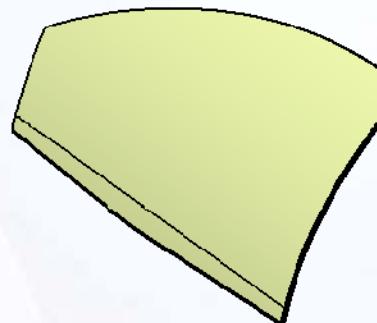


10 min

The objective of this step is to create a new part (WindowGlass) in the context of the skeleton part of door. High-level instructions are provided for this exercise.

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

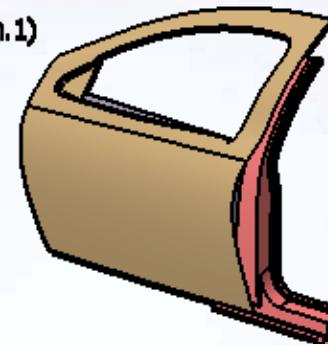
- Insert an existing component in the assembly
- Create a feature using the published elements from skeleton part
- Organize the model



Master Project: Create a Contextual Part (1/7)

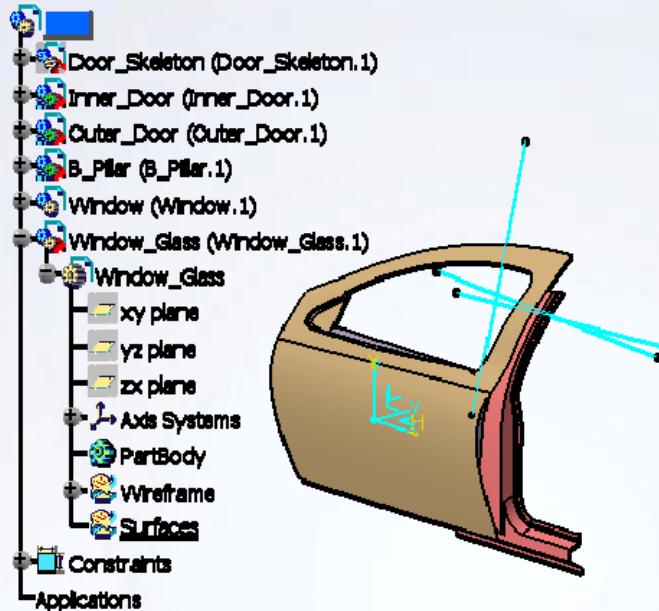
Here is a list of required tasks to guide you to create a WindowGlass:

1. Load Door.CATProduct created in step 2. If you did not complete step 2, open Step2_end.CATProduct.

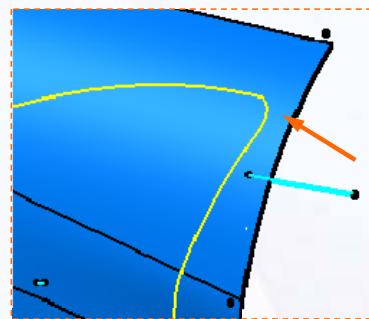


Master Project: Create a Contextual Part (2/7)

2. Insert an existing component in the assembly.
 - Insert Window_Glass.CATPart in the root assembly.

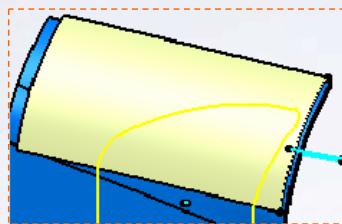


3. Create an extended glass surface.
 - Copy and paste (as result with link) Front_Seal from Window_skeleton part to the Window_Glass part. Front_Seal is a published element of the Window_Skeleton part.
 - Create a parallel curve at an offset distance of 45mm using an edge of the Front_Seal.

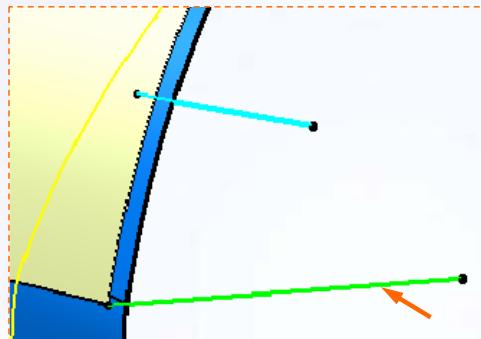


Master Project: Create a Contextual Part (3/7)

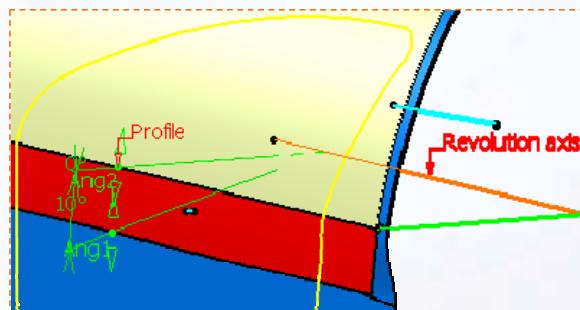
- Create an extruded surface in X direction and keep 1500mm as the length of extrude.



- Create a curve normal to the parallel curve and keep 900mm as the length.

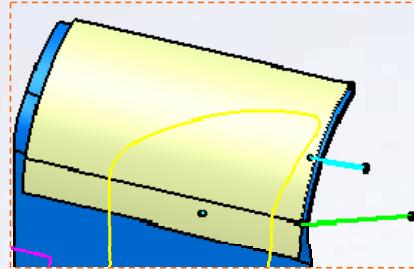


- Create a revolute surface using the boundary of extrude as the profile and keep 10deg as the angle limit. Use the revolution axis as shown.



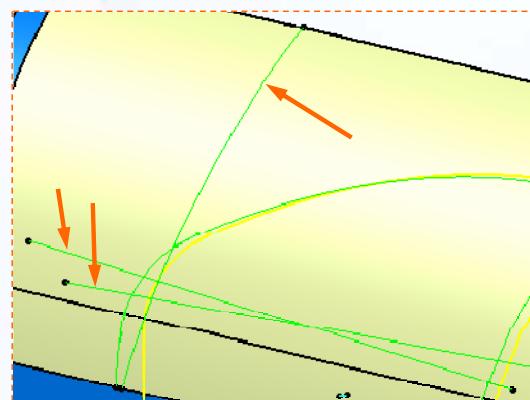
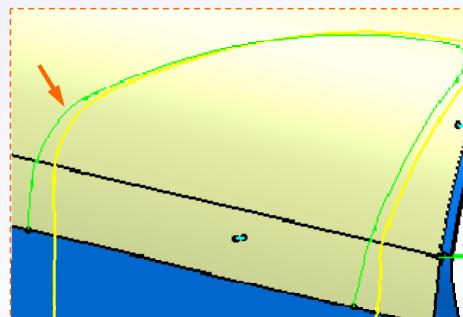
Master Project: Create a Contextual Part (4/7)

- Join the extrude surface and revolve surface.



4. Create the curves for glass limitation.

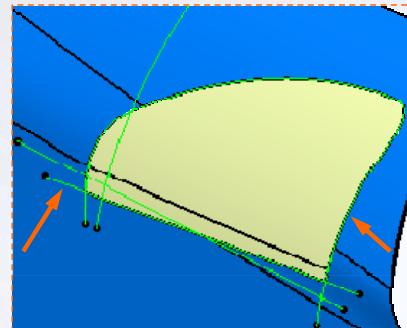
- Project the Inner Front Seal on this join surface. Use the Stamping plane as a projection direction. The Inner Front Seal and Stamping plane are the published elements of the skeleton part.
- In a similar way, project Sketch.1, Sketch.2 and Sketch.3. Use the Y axis as the projection direction.



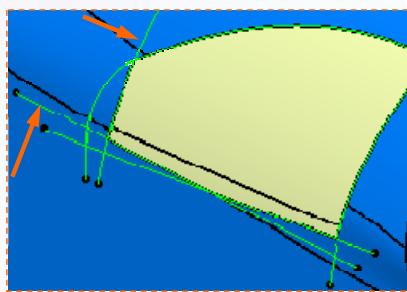
Master Project: Create a Contextual Part (5/7)

5. Split the glass.

- Split the extended glass surface as shown.



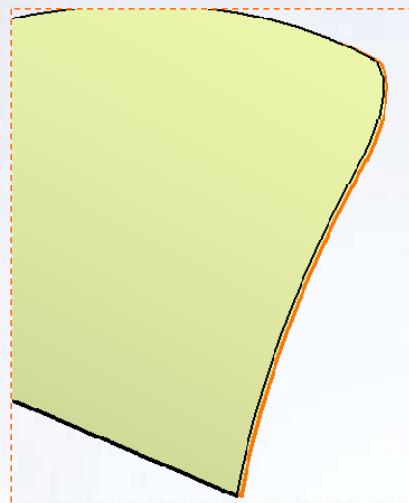
- Split the result as shown.



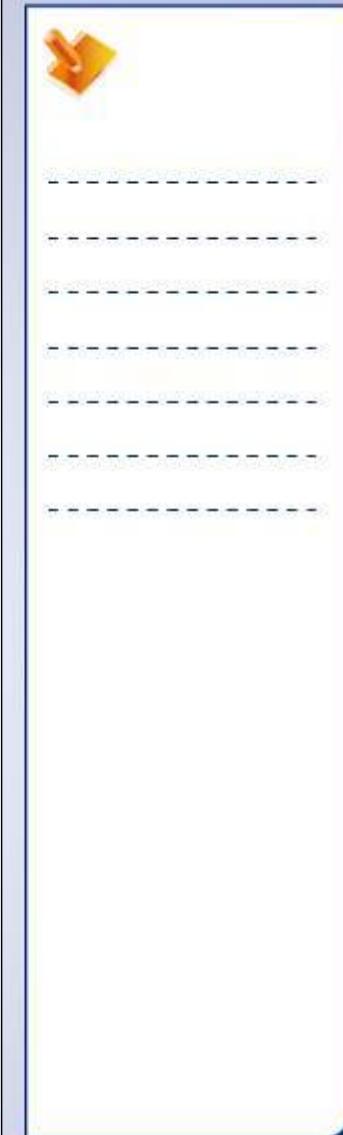
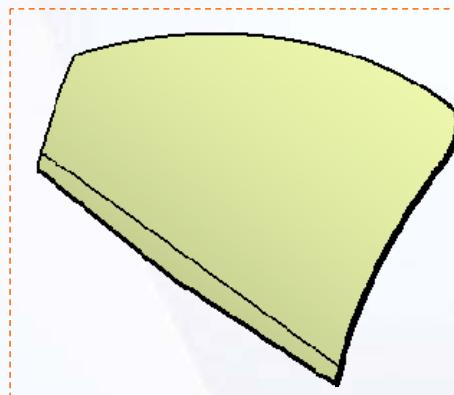
Master Project: Create a Contextual Part (6/7)

6. Create a thickness for the glass.

- Offset the split surface by 4mm.

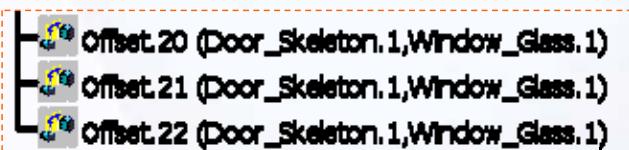
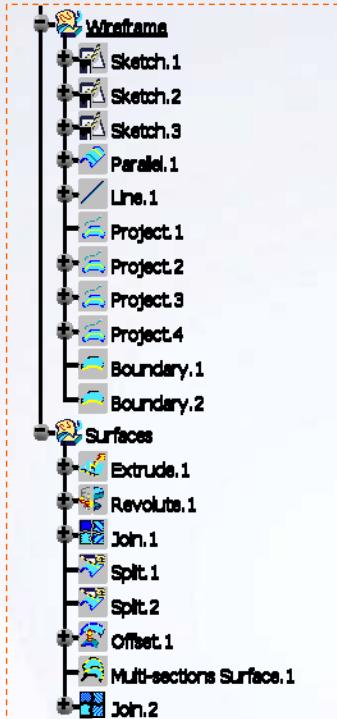


- Create boundaries of the split surface and offset surface.
- Create a multi-sections surface using both the boundaries as sections.
- Join the three surfaces to create the glass.



Master Project: Create a Contextual Part (7/7)

7. Organize the elements in the geometrical sets.
8. Publish the XY, YZ, and XZ planes of the axis system.
9. Assemble the Window_Glass with respect to the Door_Skeleton part.
 - Add three offset constraints of [0mm] between the XY, YZ, and ZX planes of the Door_skeleton part and Window_Glass part.



Master Project: Create a Contextual Part Recap

- ✓ Insert an existing component in the assembly
- ✓ Create a feature using the published elements from the skeleton part
- ✓ Organize the model
- ✓ Save a file



Master Project - Replace the Input Curves

Door Assembly

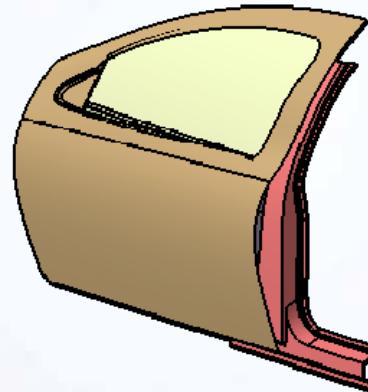


10 min

The objective of this step is to replace the input curves of the Front_Seal, update the Door assembly and manage the update errors. High-level instructions are provided for this exercise.

By the end of this step, you will be able to:

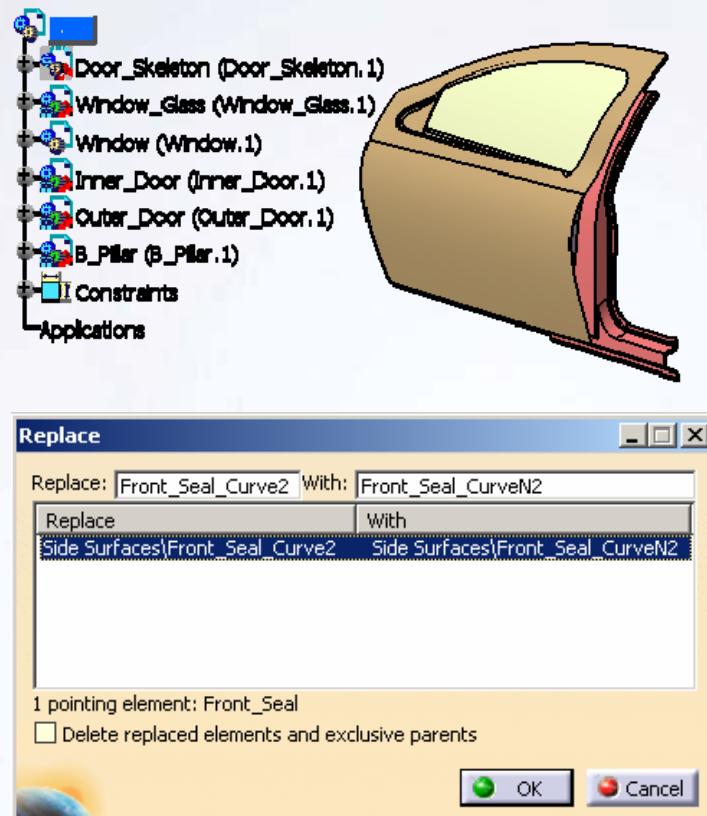
- Replace the input curves.
- Manage the errors during the update.
- Make necessary corrections.



Master Project: Replace the Input Curves (1/3)

Here is a list of required tasks to guide you:

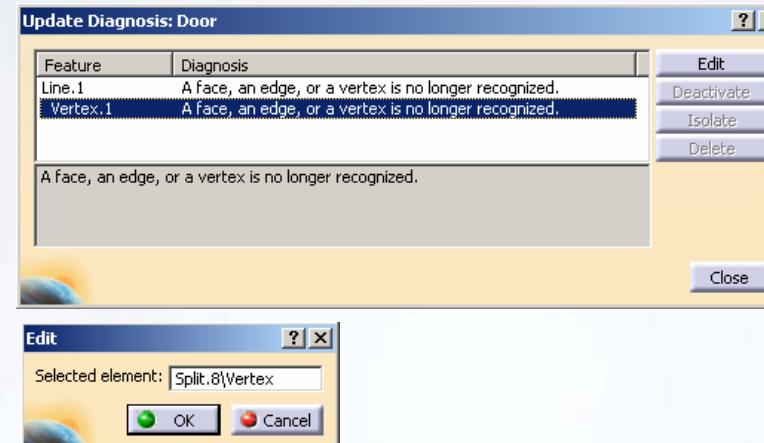
1. Load Door.CATProduct created in step 3. If you did not complete step 3, load Step3_end.CATProduct.
2. Replace the input curves of Front_Seal surface.
 - Edit the Door_Skeleton part.
 - Replace 'Front_Seal_Curve1' available in the geometrical set 'Side Surfaces' by 'Front_Seal_CurveN1' available in the geometrical set 'Side Surfaces'.
 - Similarly replace 'Front_Seal_Curve2' by 'Front_Seal_CurveN2'.



Master Project: Replace the Input Curves (2/3)

3. Analyze and resolve the problem in the model.

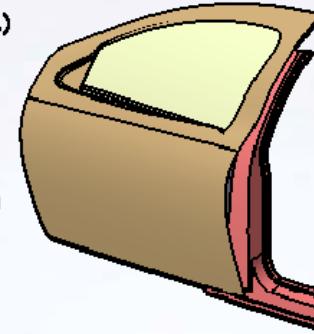
- Edit the Door.CATProduct and update it.
- Update diagnosis occurs on the Inner_Door. Edit the reference element (Vertex.1 in this case) and select new reference.



- Edit the Door.CATProduct, update it, and resolve the update errors on Outer_Door, B_Pillar and Window_Glass.

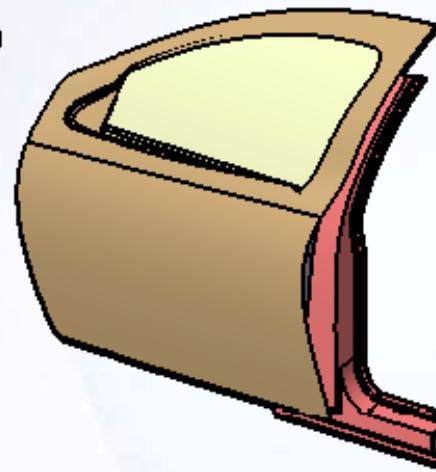
Master Project: Replace the Input Curves (3/3)

- Save and close the assembly.



Master Project: Replace the Input Curves Recap

- ✓ Replace the input curves.
- ✓ Manage the errors during the update.
- ✓ Make necessary corrections



Master Project - Create an Assembly Drawing

Door Assembly

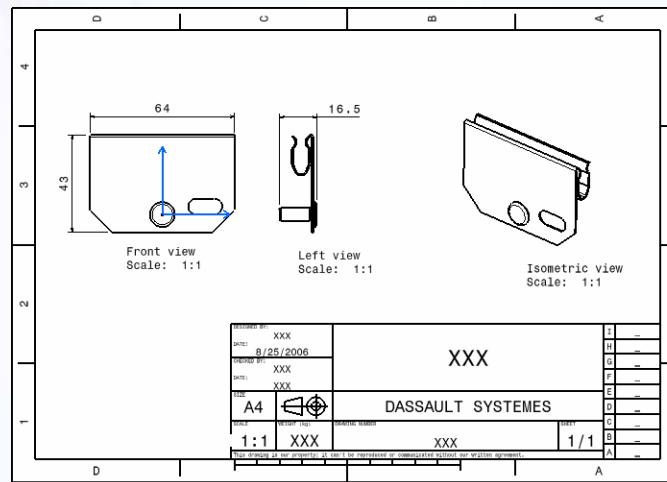


10 min

The objective of this step is to create a drawing of the Mechanism Assembly.
Include the overall dimensions of the model and a title block in the drawing.
High-level instructions for this exercise are provided.

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

- Create a new drawing
- Create two main views
- Add dimensions
- Apply title block
- Save a file



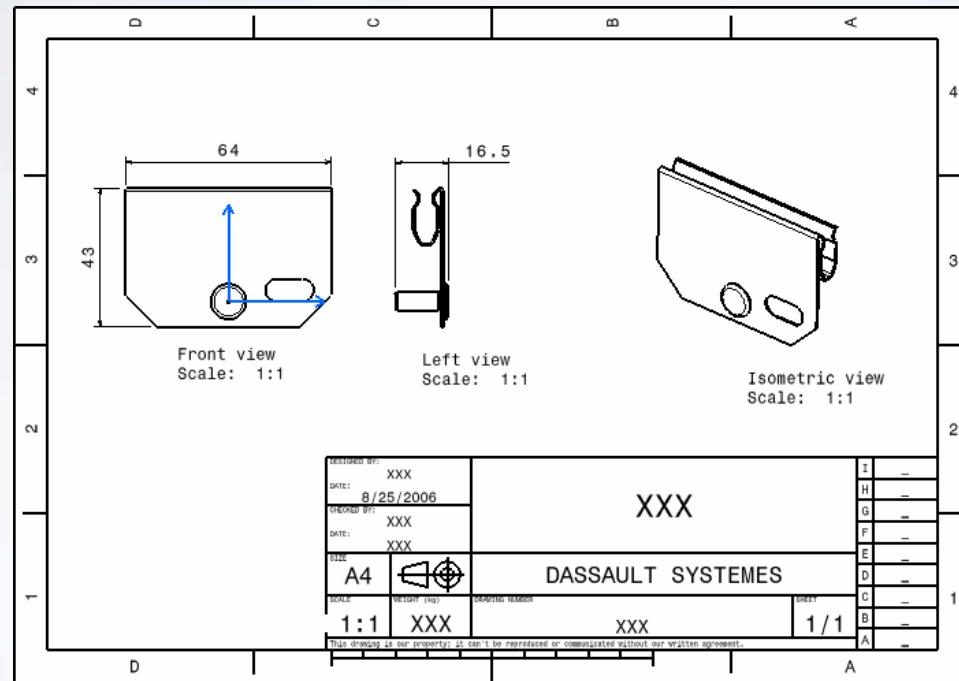
Master Project: Create an Assembly Drawing

Create a drawing of the Mechanism Assembly as shown. If you did not finish step 2, use Step2_1_end.CATProduct.

1. Use the following criteria:

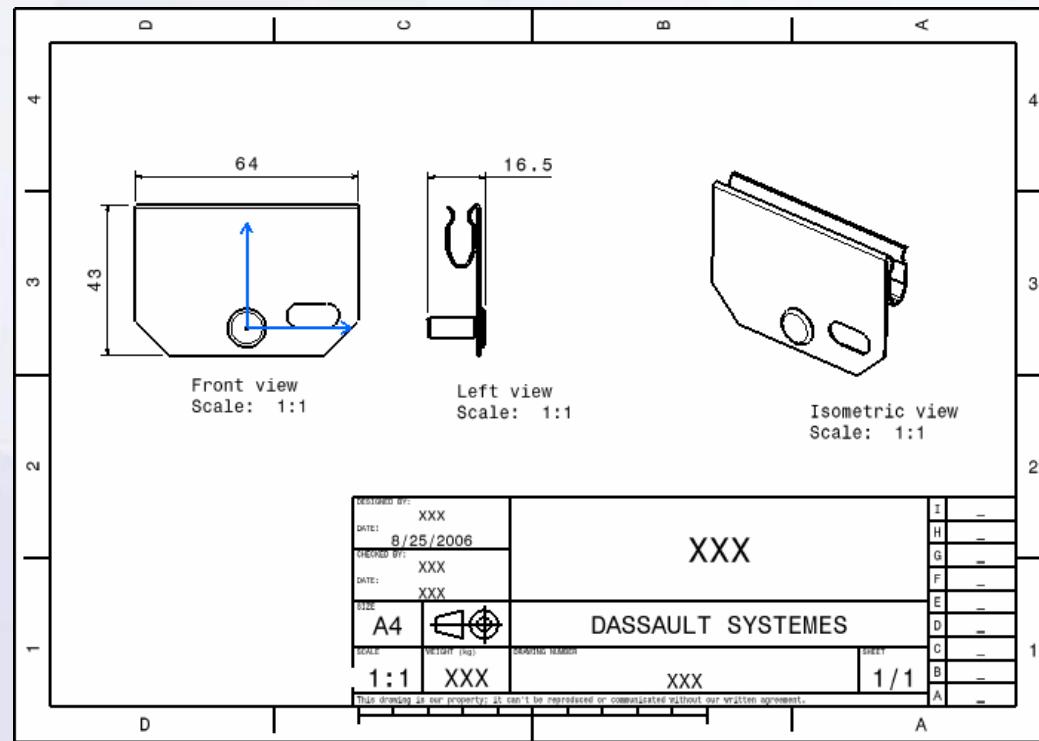
- Standard: ISO
- Format: A4 ISO
- Orientation: Landscape
- Sheet Scale: 1:1
- Isometric view's scale 1:1
- Title Block: ISO_A4
(Instantiate from the catalog already loaded and used during lesson 12)

2. Save the drawing.



Master Project: Create an Assembly Drawing Recap

- ✓ Create a new drawing
- ✓ Create two main views
- ✓ Add dimensions
- ✓ Apply title block
- ✓ Save a file



Shortcuts

F1	Link to on-line documentation	Ctrl + several selections	Multiple selection
Shift F1	Contextual help for an icon	Shift + 2 selections	Selection of all elements
Shift F2	Overview of the specification tree		between and including the 2 selected elements
F3	Hide/Show the specification tree		
Ctrl + Tab	Change CATIA V5 window	Alt F8	Macros
Ctrl N	New file	Alt F11	Visual Basic editor
Ctrl O	Open file	Alt + Enter	Properties
Ctrl S	Save file	Alt + MB1	Pre-selection Navigator
Ctrl P	Print	Ctrl F11	Pre-selection Navigator
Ctrl Z	Undo	Up/Down or Left/Right arrow	Pre-selection Navigator
Ctrl Y	Redo	Shift + MB2	Local zoom and change of viewpoint
Ctrl C	Copy		
Ctrl V	Paste	Shift + manipulation with compass	Displacement respecting constraints
Ctrl X	Cut		
Ctrl U	Update		
Ctrl F	Find		

Glossary

Active Item (Assembly Design): The component which is being edited is called the active item. To make an item active, double-click it. The active item will be highlighted.

Assembly Design: Creation of specifications for parts, constraints, and features in the context of an assembly.

Assembly Features: Features created inside an assembly which affects more than one component of the assembly.

Assembly: Assembly is a document that contains a collection of components. It has the file extension.CATProduct. An assembly is also called a product.

Associative: A CATIA model is fully associative with the drawings and parts or assemblies that reference it. Changes in the model are automatically reflected in the associated drawings, parts, and/or assemblies. Likewise, changes in the context of the drawing or assembly are reflected back in the model.

Catalogs: Catalogs are sets of frequently used features or components which are stored as a library of information.

Chain Mode: In this mode of Measurement Between, the second element for a measurement automatically becomes the first element for next measurement.

Compass: An orientation reference tool that helps while performing view rotations. The Compass is a powerful tool that can be used to physically move and manipulate objects. This is especially useful in Assembly Design, Freestyle, and Digital Mockup workbenches.

Component: A reference of a part or a subassembly integrated in an assembly. A component possesses characteristics related to how a referenced part or subassembly is integrated in an assembly.

Glossary

Constraints: Constraints establish geometrical or dimensional relationships between the features of a model by fixing their positions with respect to one another.

Construction Geometry: Construction Geometry is created within a sketch to aid in profile creation. Unlike standard sketched geometry, construction geometry does not appear outside the Sketcher workbench.

Design in Context: When a new part is created in an assembly, the new part features and sketches can be defined using geometry of existing components. This process is known as design in context.

Design Intent: Design Intent is the plan of how to construct the solid model of a part in order to properly convey its visual and functional aspects. In order to efficiently use a parametric modeler like CATIA, you must consider the design intent before and while modeling the part.

Design Mode: In this mode, the part definitions (exact geometry and parameters) of all the components in an assembly are loaded into memory. By default, the assemblies and their components are loaded into a CATIA session in the Design mode.

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

Drawing: A document which contains the geometrical information and specifications in form of 2D views. It has the extension as '.CATPart'.

Dress-up Features: Features created directly on the solid model. Fillets and chamfers are examples of this type of feature.

Explode: Command used to break down a pattern of features into number of individual features.

Glossary

Fan Mode: In this mode of Measurement Between, measurements are made between the first selected element and each element selected thereafter.

Feature-based: Like an assembly is made up of a number of individual parts, a CATIA document is made up of individual elements. These elements are called features and the approach is called Feature-based.

Features: Elements that make up a part. They can be based on sketches (sketch-based) or features that build on existing elements (dress-up and transformation). They can also be generated from surfaces (surface-based).

Grid: A network of horizontal and vertical lines applied to the background of the Sketcher workbench, that provides coordinates for locating points.

In Work Object: It is the current state of the part during its design phase. It can be identified in the specification tree by its name, which is underlined.

Instances (Assembly Design): Each component inserted into an assembly is a separate instance. For example, if the same part is inserted into an assembly twice, they will have the same part number but different instance numbers. No two components in an assembly can have the same instance number.

Layered Approach: The layered approach builds the part one piece at a time, adding a layer or feature onto the previous one until the desired solution is obtained.

Manufacturing Approach: The manufacturing approach to modeling mimics the way the part would be manufactured.

Neutral Element: It is a plane or face which defines the neutral curve. The drafted surfaces pivot about a neutral curve.

Part: A document which contains the geometrical information and specifications that define a 3D solid model. It has the extension as 'CATPart'.

Glossary

Part Body: Default container that contains the features that make up a part.

Part Design: The workbench dedicated for designing parts using the solid modeling approach.

Part Number: It identifies the part file used in the assembly. Normally, the part number is same as the file name for the components, at times, it may be different.

Pattern: array of several identical features created from an existing feature.

Potter's Wheel Approach: The potter's wheel approach builds the part as a single, revolved feature. A single sketch, representing the cross-section, includes all the information and dimensions necessary to make the part as one feature.

Positioned Sketch: Sketch for which you have specified the reference plane, origin, and the orientation of the absolute axis.

Property: It is an attribute such as color or a name that can be assigned to any feature.

Pulling Direction: 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.

Sketch-based features: Features created using a 2D sketch. Generally, the sketch is transformed into a 3D solid by extruding, rotating, sweeping, or lofting.

Sketcher: The workbench dedicated for creating 2D profiles with associated constraints, which can then be used to create 3D geometry.

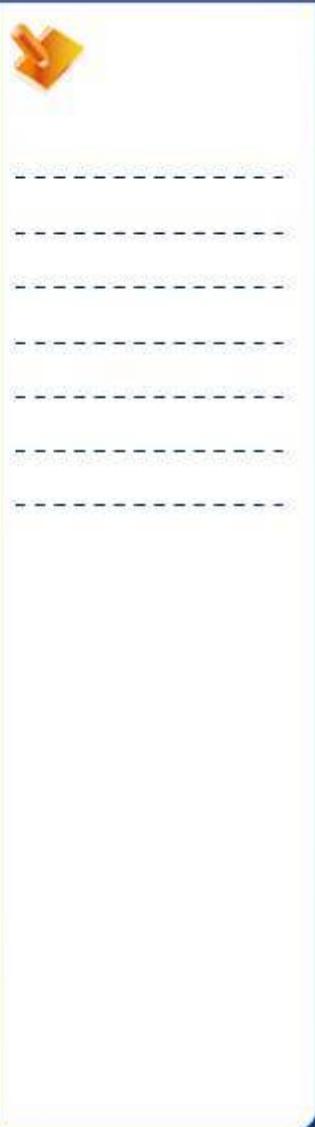
Specification Tree: It keeps the hierarchy of features, constraints, and processes, and the assembly information for a CATIA document. The specification tree provides a visual step-by-step record of the sequence followed while creating a solid model.

Standard Mode: In this mode of Measurement Between, two elements must be selected for each measurement.

Glossary

System Generated Parameter: A parameter that gets created automatically during creation of a feature or element. It has a unique identifier.

Visualization mode: In this mode, CGR representations of the geometry, of all the components in an assembly, are loaded instead of the actual geometry. CGR (.cgr) files contain no geometry or part information; they are only a tessellated visual representation of the model.





Handwriting practice lines for the left template. The lines consist of a solid blue top line, a dashed midline, and a solid blue bottom line, providing a guide for letter height and placement.

A blank white rectangular area enclosed in a dark blue border, serving as a template for handwriting practice or drawing.



Handwriting practice lines for the left template. The lines consist of a solid blue top line, a dashed midline, and a solid blue bottom line, providing a guide for letter height and placement.

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A blank white rectangular area enclosed in a dark blue border, serving as a template for handwriting practice or drawing.



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