CATIA V5 Fundamentals

DASSAULT SYSTEMES

See what you mean

CATIA V5 Fundamentals

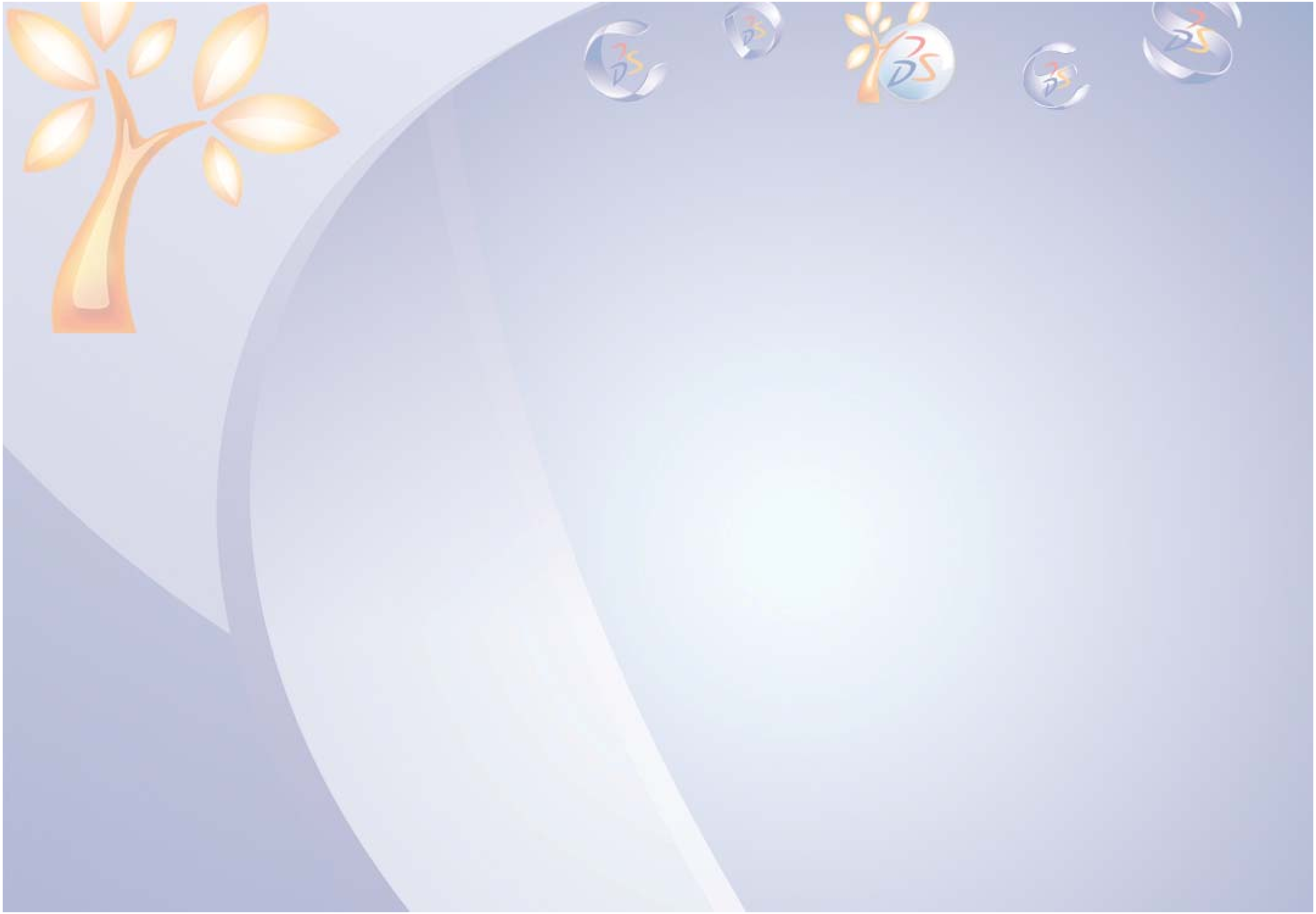
Student Handbook

Version 5 Release 19

40 Hours

Copyright DASSAULT SYSTEMES

Table of Contents



Introduction to CATIA 1

Learning Objectives

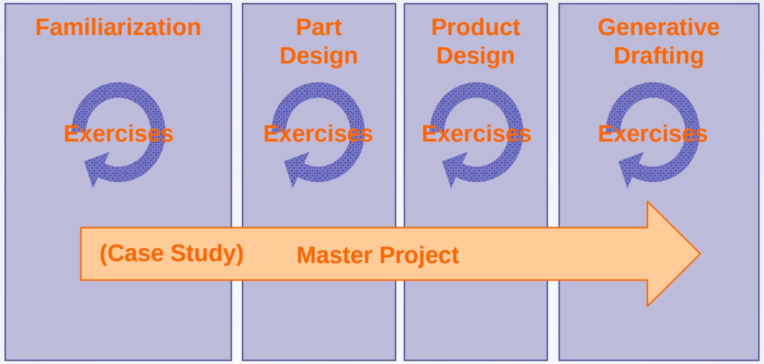
Upon completion of this lesson you will be able to:

- ✓ Understand the CATIA software.
- ✓ Open CATIA.
- ✓ Understand the CATIA interface.

4 Hours

Case Study

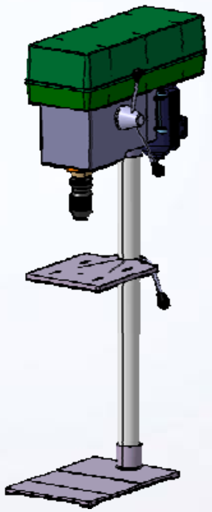
Each lesson in this course contains a case study, which explains the skills and concepts covered in the lesson. The case study will be described at the beginning of each lesson, and the student will be able to do the case study exercise once the theory for that lesson has been covered.



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:

- ✓ 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.

Understand the CATIA software

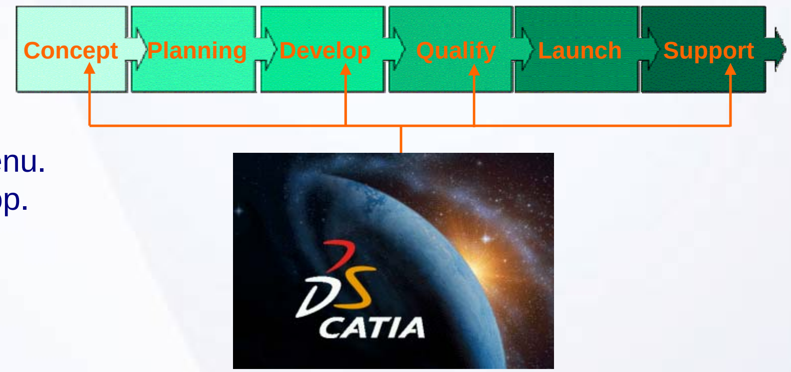
CATIA is a mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations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



CATIA's Scope From Concept To Realization

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...).

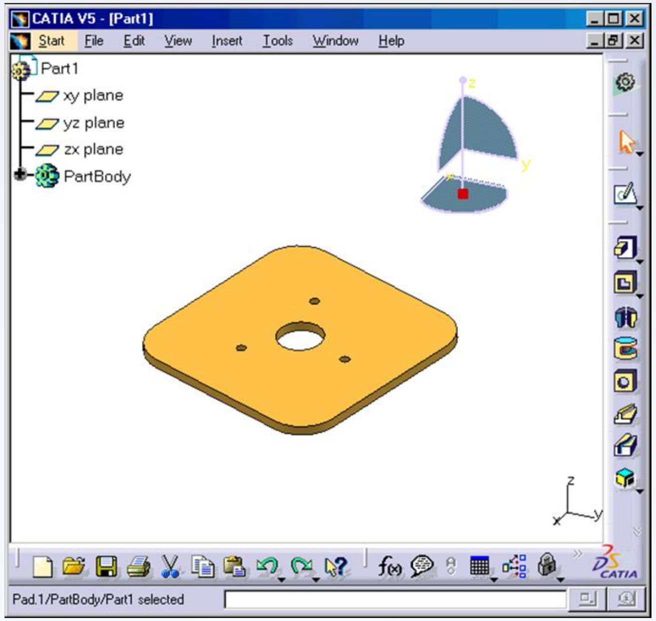
D. Standard manipulations (Copy-Paste, Drag-and-Drop, Edit in Place...).

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)**

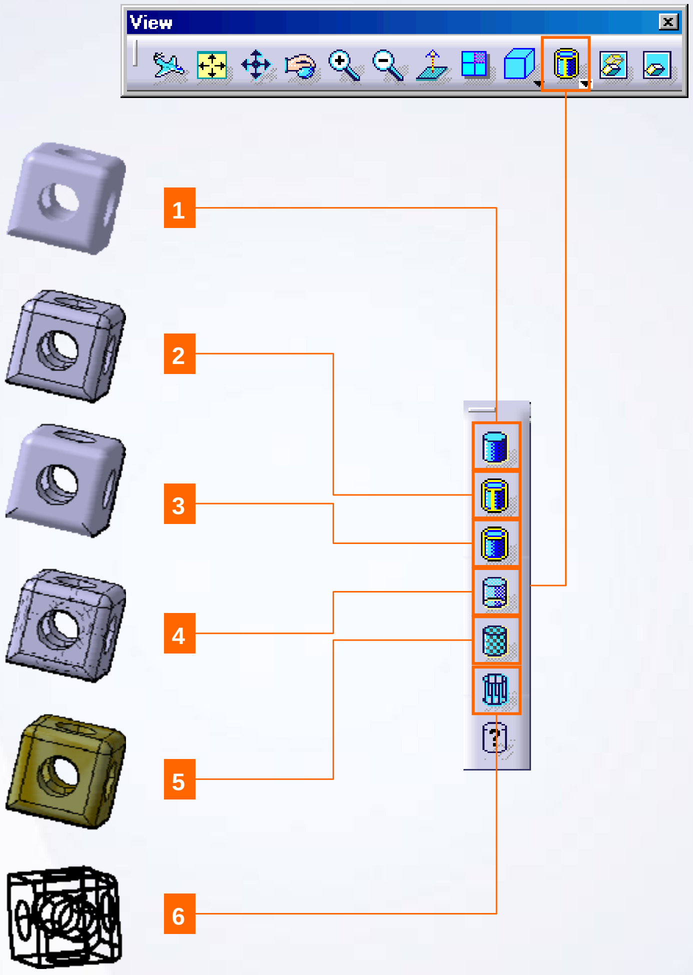
2 **Shading with Edges**

3 **Shading with Edges without smooth Edges**

4 **Shading with Edges with Hidden edges**

5 **Shading with Material**

6 **Wireframe (NHR)**



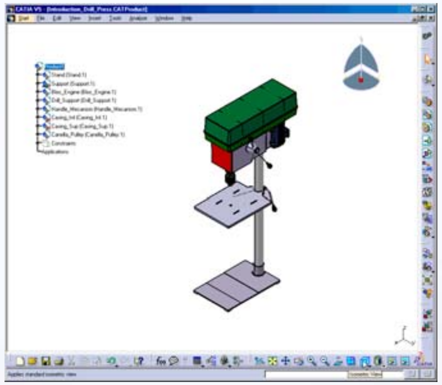
Case Study: Introduction to CATIA

Recap Exercise

20 min

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

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



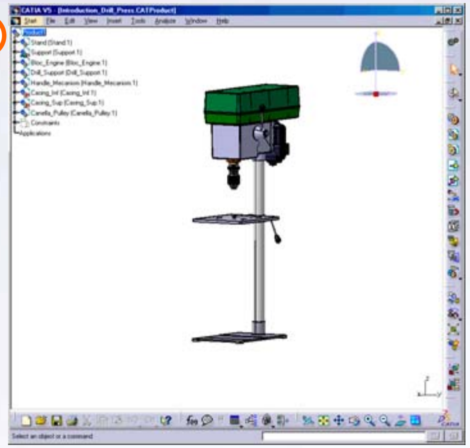
Case Study (1/6)

1. **Open Introduction_Drill_Press.CATProduct.**

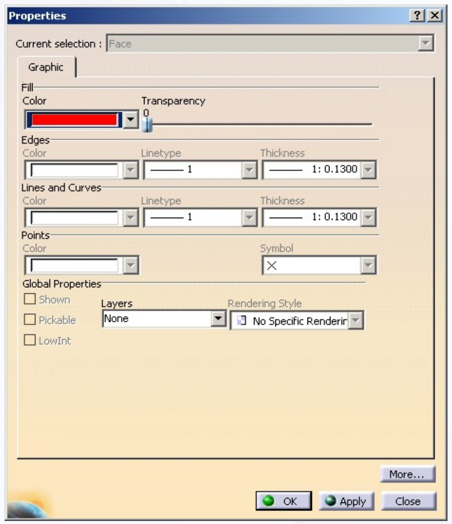
2. **Change the Orientation of the assembly.**
   a. Change the model orientation to Front.
   b. Zoom in on the area as shown.

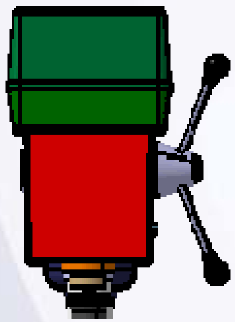
3. **Change the Visualization properties.**
   a. Select the front face of the support part as shown and change the color to red.

1



2a



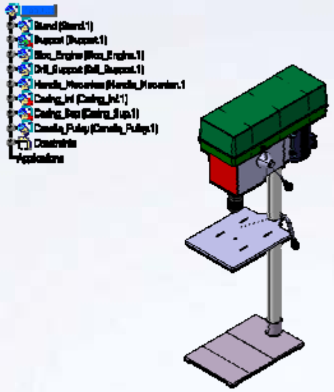


2b

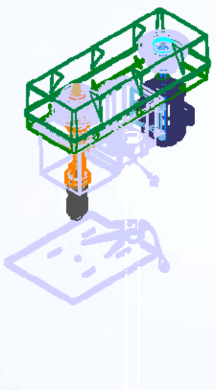
Case Study (2/6)

4. Zoom out on the model.
5. Change to the Isometric view.
6. Change the rendering style to Wireframe.
7. Change the rendering style to Shading with Edges.

5



6



7

Case Study (3/6)

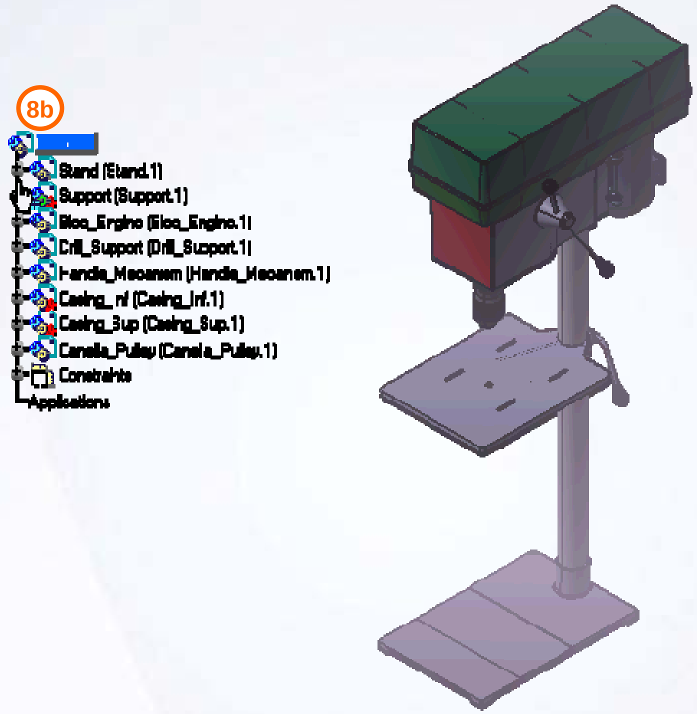
8. Change the display of the specification tree.

a. Press the <F3> key to toggle the specification tree on and off.

b. Select 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> and <F3> to re-activate the model.



Case Study (4/6)

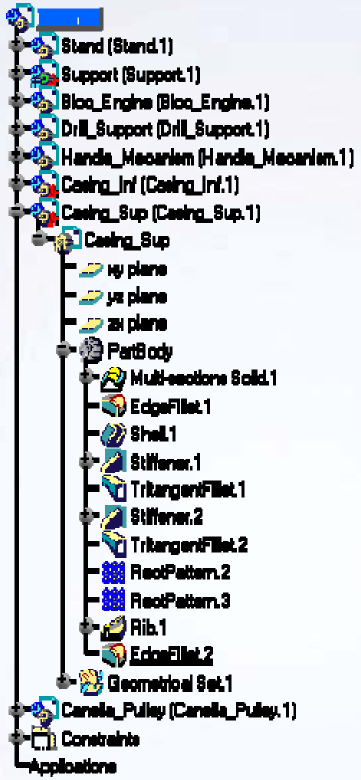
8. Change the display of the specification tree (continued)

e. Expand the Casing_sup 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.

8f

8e

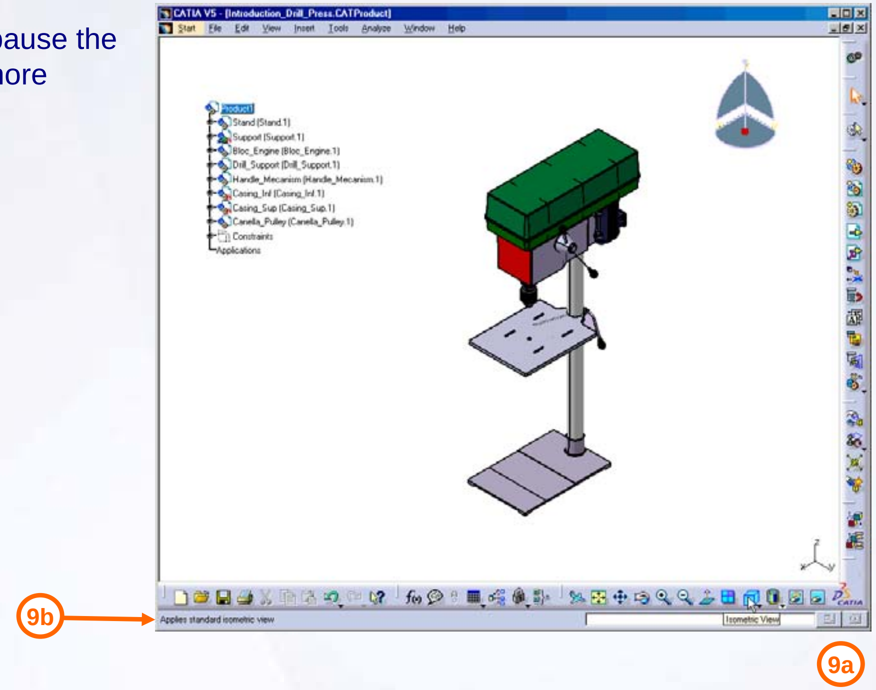


Case Study (5/6)

9. **Review the areas of information.**

   a. Hover the mouse over the various tools and areas of the toolbars and notice the tool tip comments.

   b. Review the message bar as you pause the mouse cursor over the icons for more information.



Case Study (6/6)

9. Review the areas of information (continued).

c. Click **Help > CATIA V5 Help**.

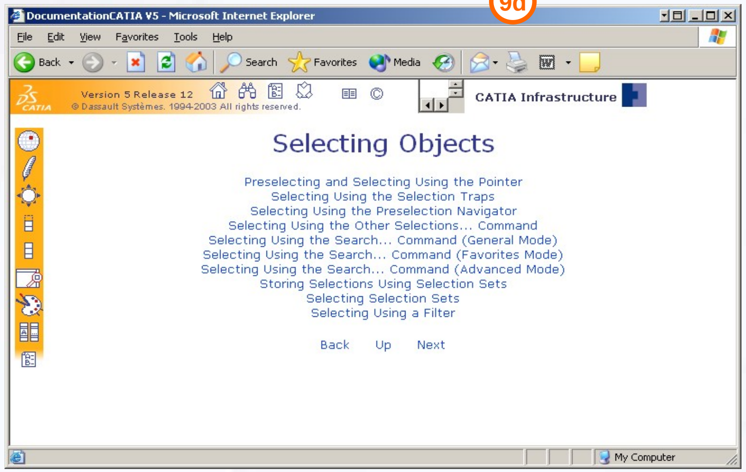
d. A web browser launches, with the CATIA Help start page loaded.

e. Spend few minutes browsing the various links within the system.

10. Close the assembly without saving changes.

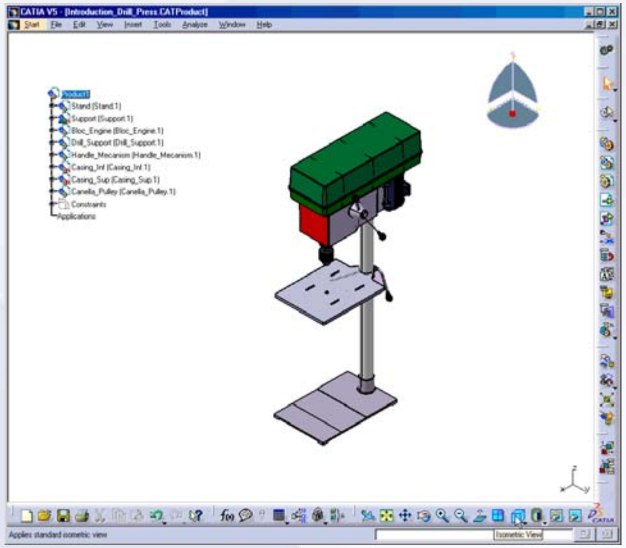
9c

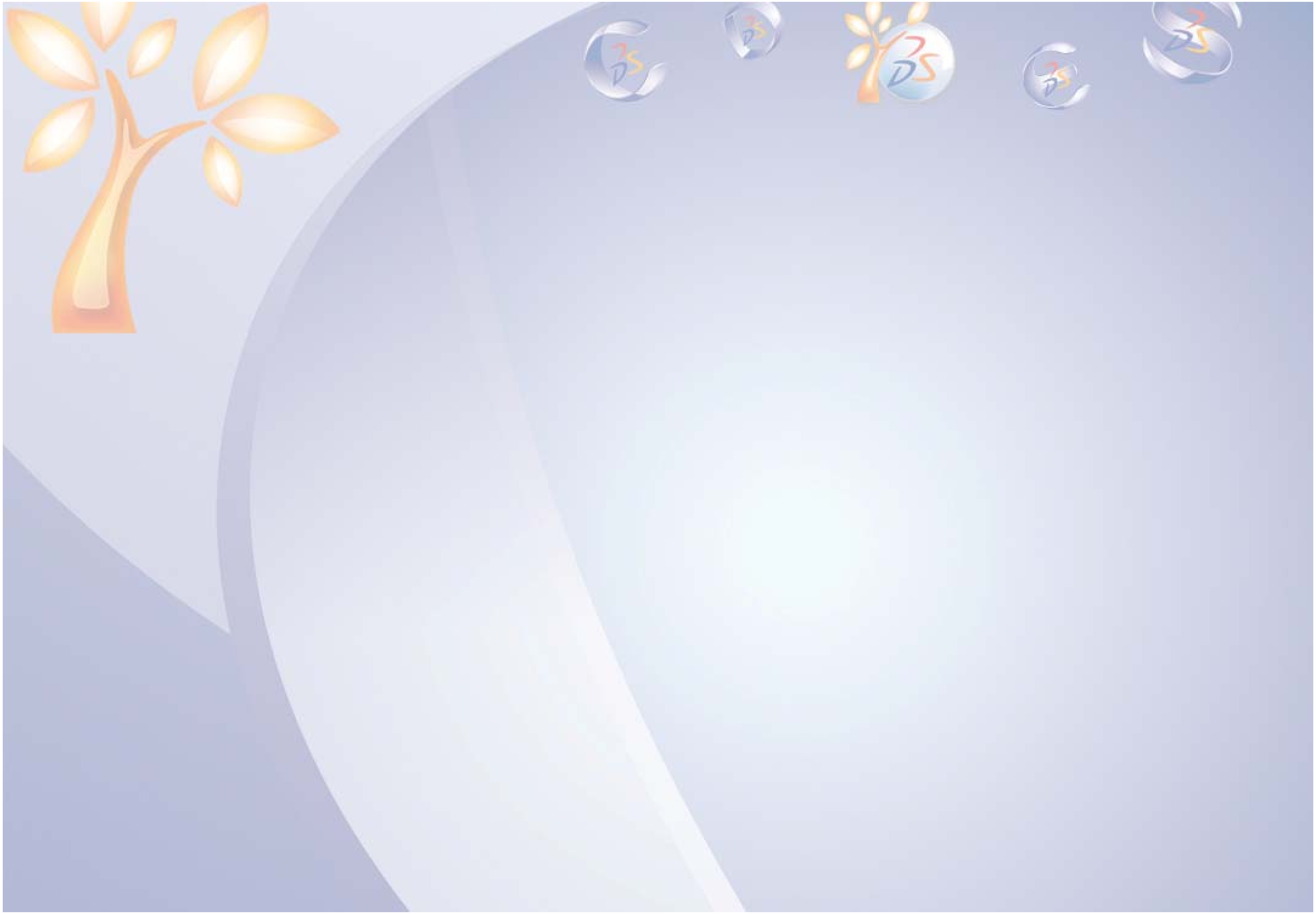
9d



Case Study: Introduction to CATIA Recap

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



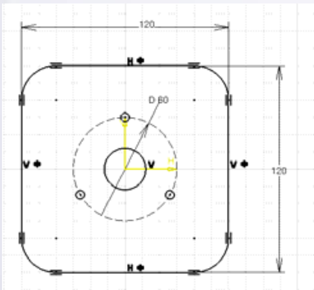


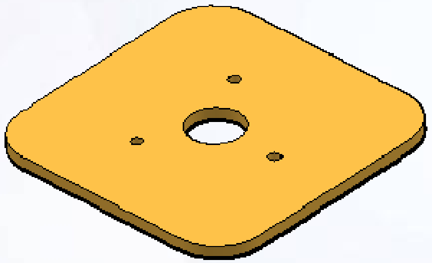
Profile Creation 2

Learning Objectives

Upon completion of this lesson you will be 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.





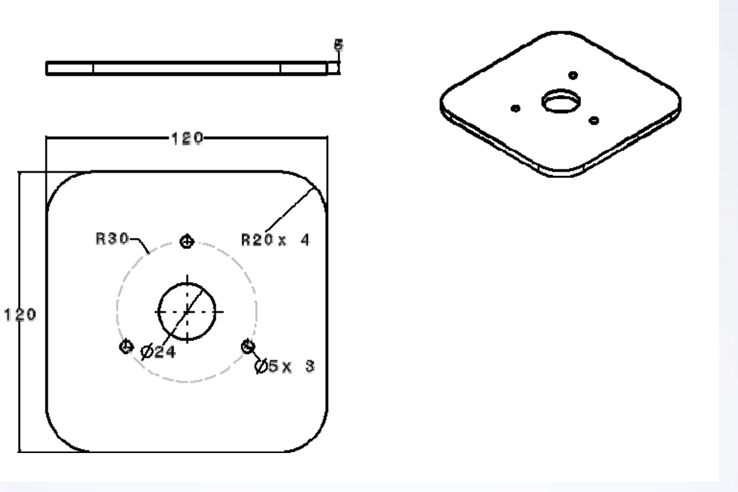
4 Hours

Case Study

The case study for this lesson is the support plaque used in the drill support assembly.

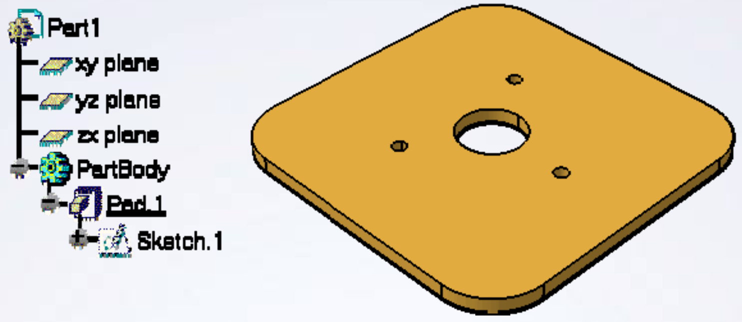
Design Intent

- ✓ The model must be created in one feature.
  - While this is not typical practice, in this case it is a requirement.
- ✓ The center hole must remain in the center of the support.
  - This requirement can be met by creating a rectangular profile symmetric about part origin and locating the center of the circle (representing the hole) at the origin.
- ✓ The smaller holes must be 30mm away from the center hole.
  - Constraining the three small holes on a construction circle that is 60mm in diameter will ensure this requirement is met.
- ✓ The model must be saved with the name Support_Plaque.



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.

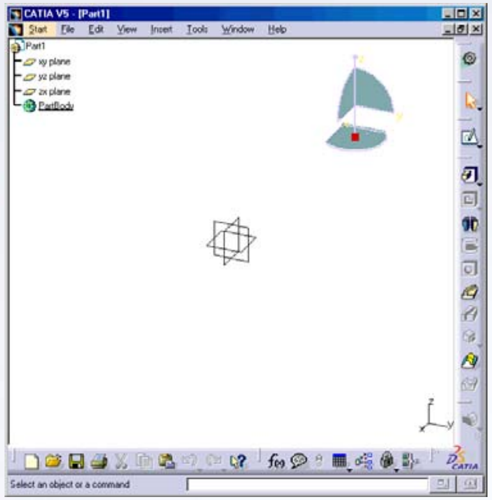


Create a New Part

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.

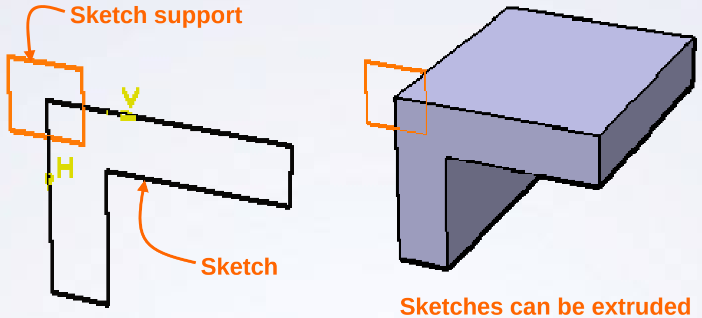


A new part contains only three default reference planes. These default reference planes are always the first elements in the specification tree and are used as a basis for feature creation.

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.

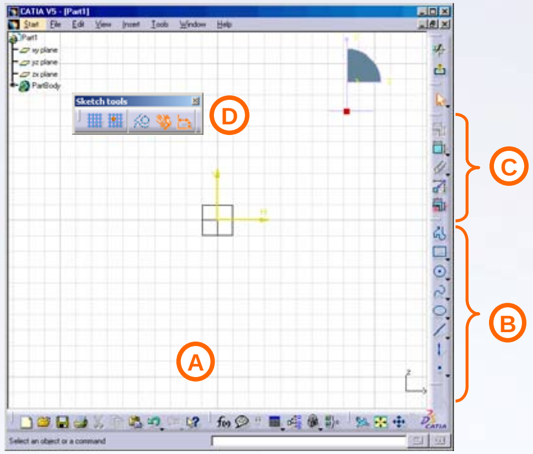


Sketches can be extruded to create solid geometry.

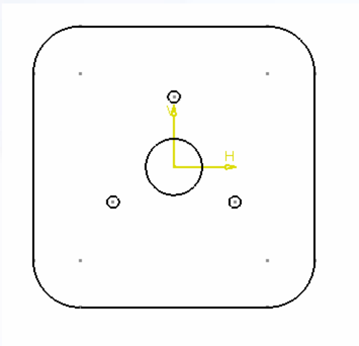
Create Sketched Geometry

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.



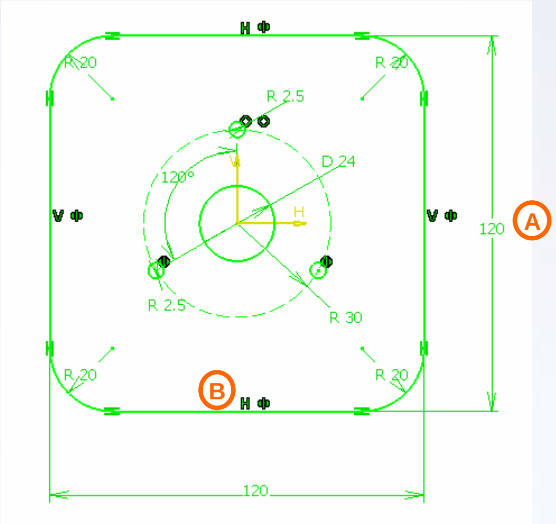
Constrain the Sketch

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.

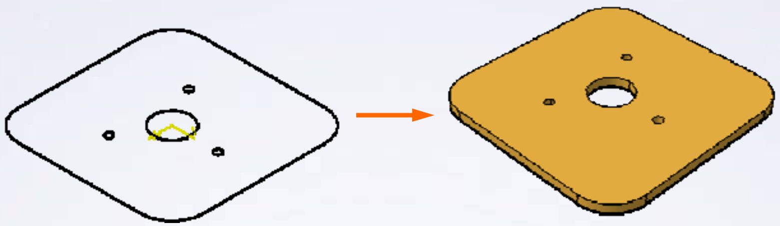
B. Dimensional constraints, which specify the distance between two elements. This distance can be linear, angular, or radial.

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.



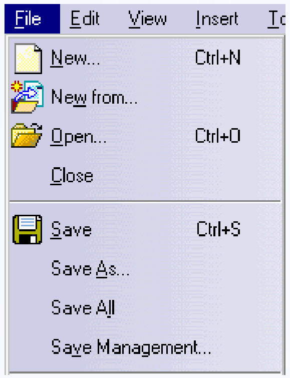
2D Profile (sketch)   Extruded Pad

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).
- ✓ Under a new name (to create a new document).

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

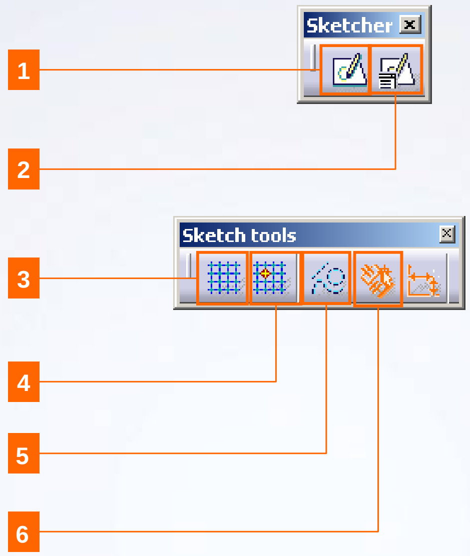
Sketch Tools

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.

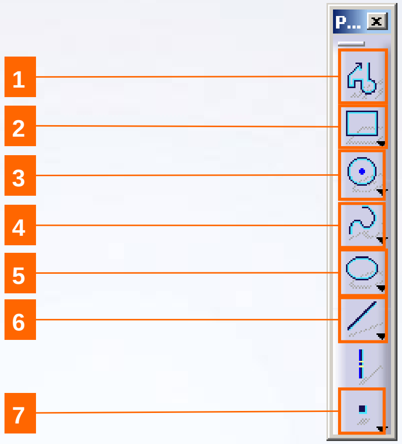
3 **Circles:** creates circles and circular arcs

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:** create a corner shape between the two selected lines

2 **Chamfer:** Create a chamfer between the two selected lines

Constraint

3 **Constraint Defined in Dialog Box:** creates geometrical constraints on selected elements

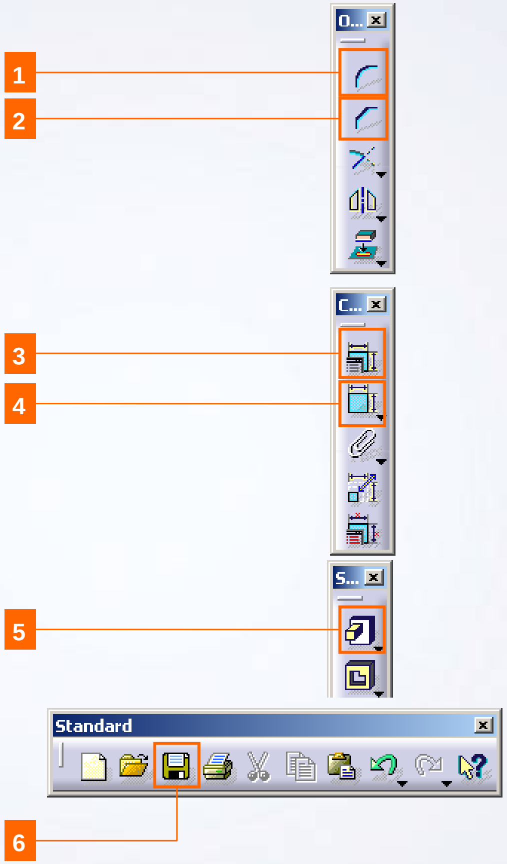
4 **Constraint:** quickly creates geometrical and dimensional constraints

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

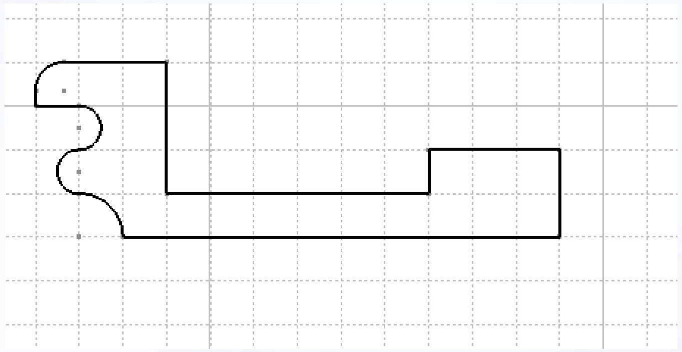
Recap Exercise

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.

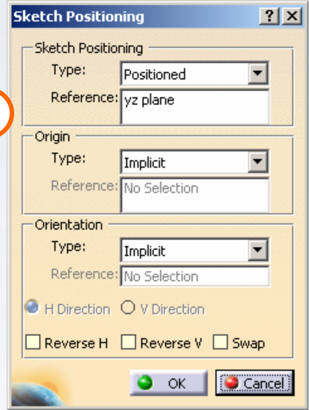
3. Ensure that the automatic constraints (located in the Sketch Tools toolbar) are deactivated.

4. Create the profile (as shown) using the Profile icon.

5. Create the corner (this is a Style type of radius).

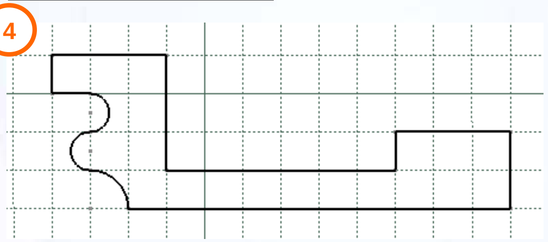
6. Close the document without saving it.

2

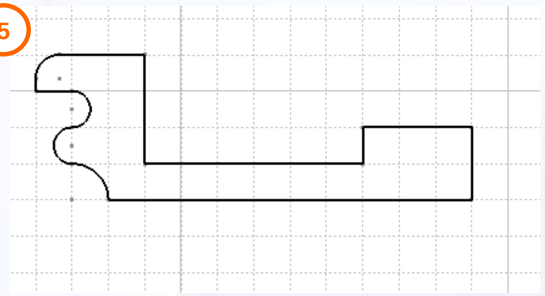


3

4

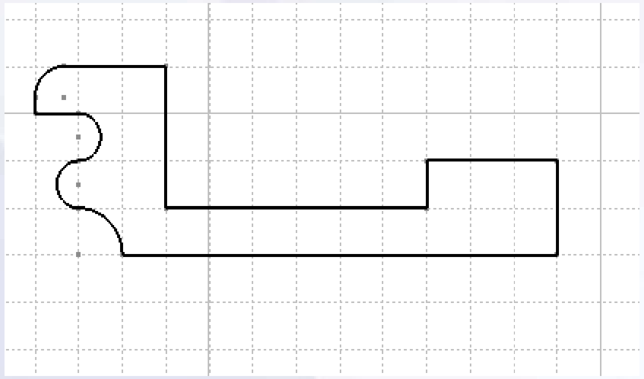


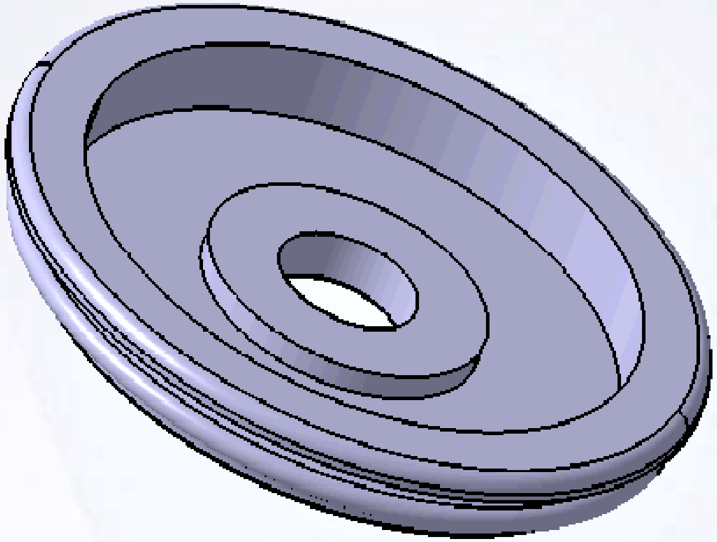
5



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: Multiple Profile Creation

Recap Exercise

10 min

In this exercise you will create a profile consisting of several shapes. You will use the tools from previous exercises to complete this exercise.

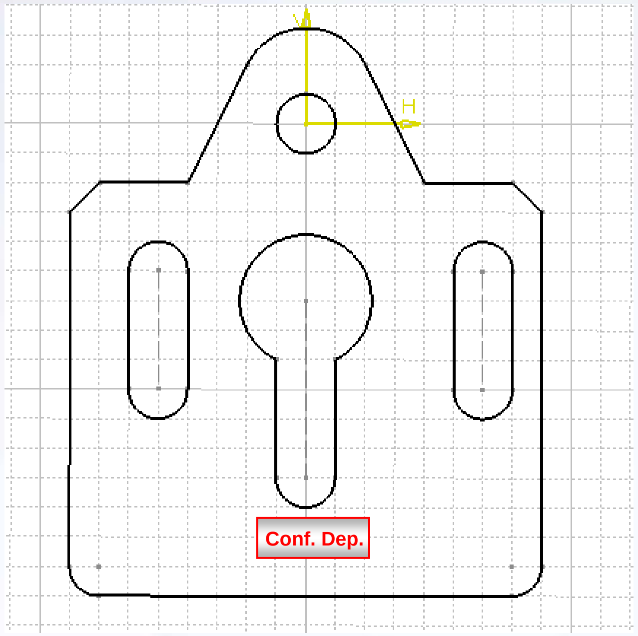
By the end of this exercise you will be able to:

- **Create a new part**
- **Access the Sketcher workbench**
- **Create sketched geometry**
- **Close the document without saving it**

CAUTION: To simplify the exercise, the sketch will contain several closed contours representing both the external shape of the 3D part (including fillets) and the internal forms (holes and pockets). 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 the profile shown, using the ZX plane as the sketch support.**

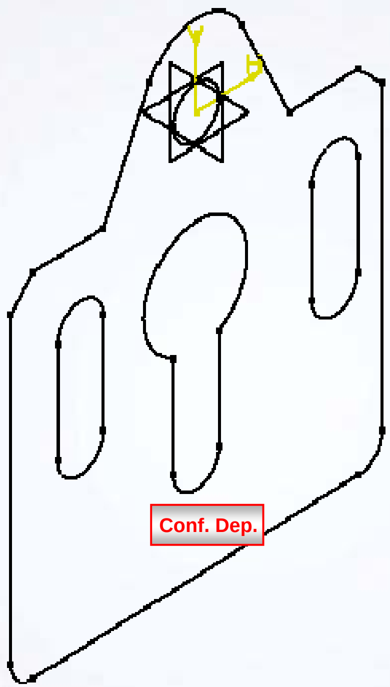


Exercise Recap: Multiple Profile Creation

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

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

- Create the profile using a series of lines and arcs.
- Create the basic shape using the Profile tool, then add corners and chamfers as separate operations.
- 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**

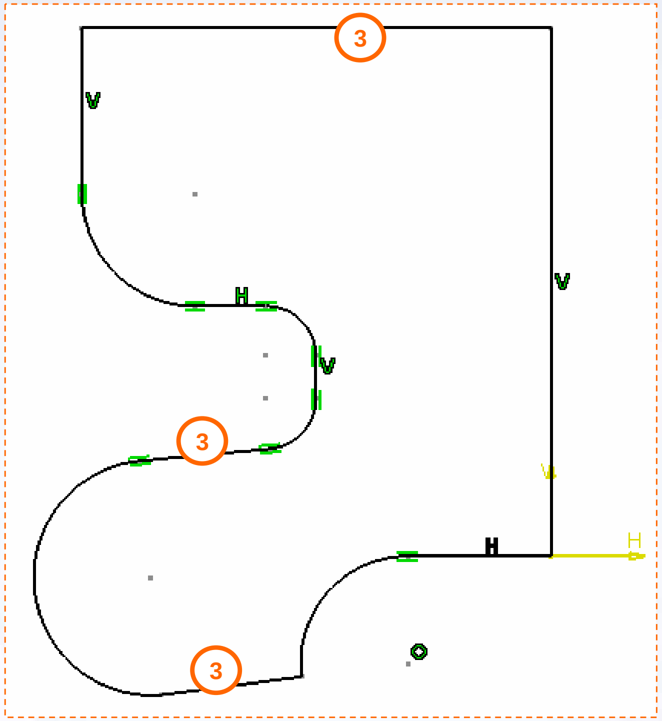
CAUTION: Generally, for complex parts it is recommended that you simplify the sketches using dedicated 3D features like fillets, chamfers, holes, drafts, etc. to better fit the design and manufacturing intents.

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.

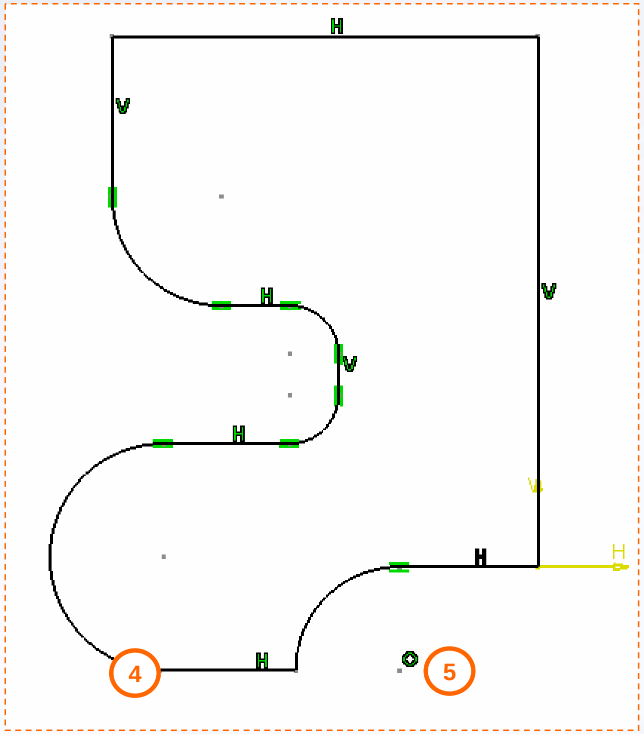
3. **Add Horizontal constraints.**



Do it Yourself (2/5)

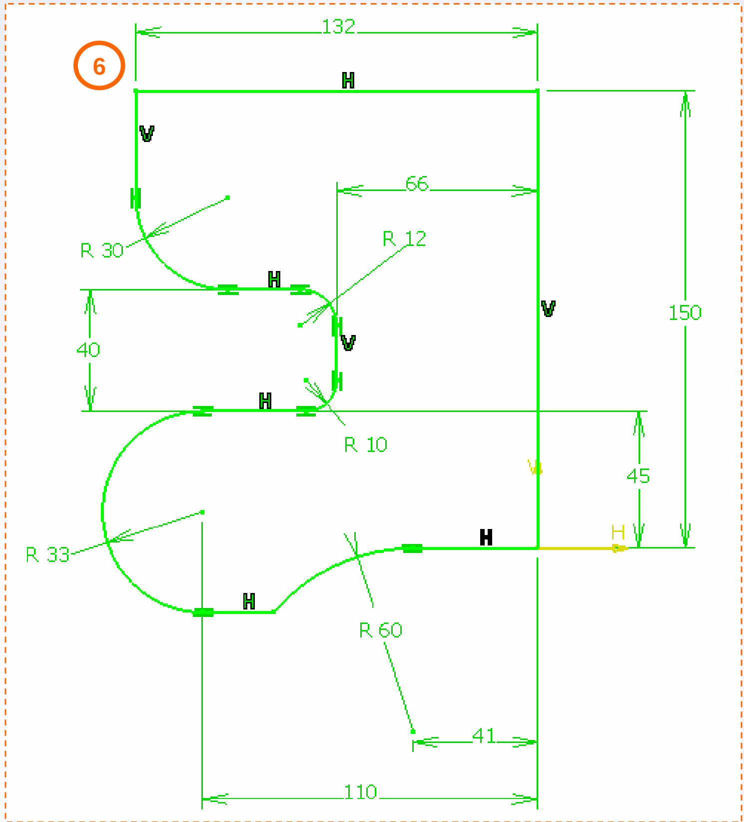
4. **Add Tangency constraint.**
   - Apply a tangency constraint between the bottom horizontal line and the arc.

5. **Remove coincidence constraints which are not required.**



Do it Yourself (3/5)

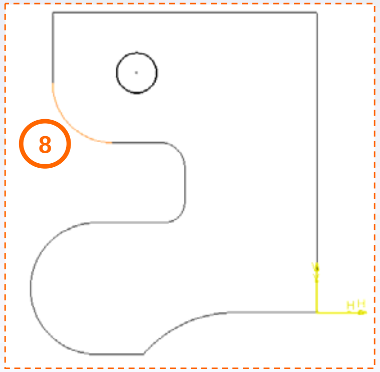
6. **Add dimensional constraints.**
   - Using proper techniques, add dimensional constraints to the sketch. Once all the dimensional constraints have been applied, the sketch will turn green, indicating that the sketch is fully-constrained.



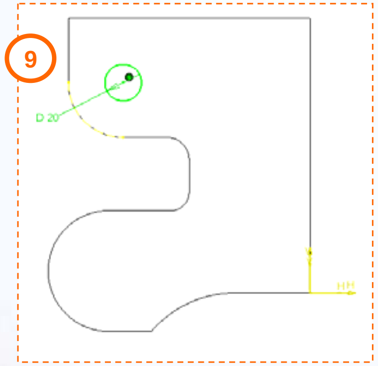
Do it Yourself (4/5)

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.

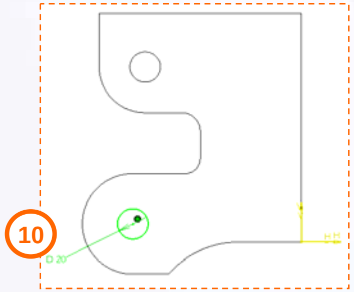
8. **Sketch in Context.**
   - Project the edge of sketch.1 and keep it as a construction element. Add Concentricity constraint.



9. **Add dimensional constraint.**
   - Add 20mm diameter dimension to make sketch.2 fully-constrained.



10. **Add dimensional constraint.**
    - Similarly, make sketch.3 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.

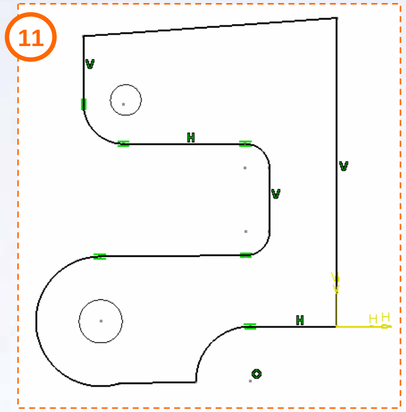
13. Geometrically and dimensionally constrain the sketches.

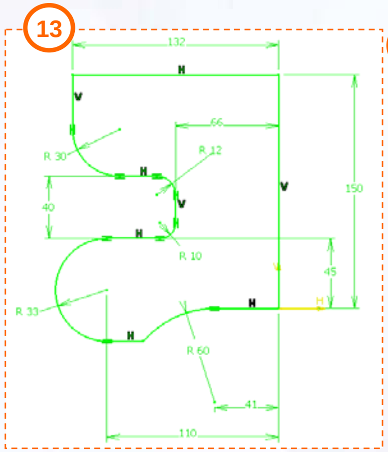
- Fully constrain the sketched circles with 20mm diameter dimensions as in the earlier instance.

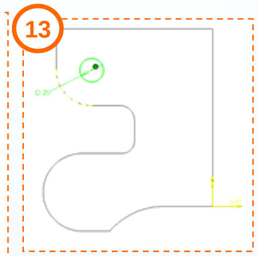
14. Compare sketches.

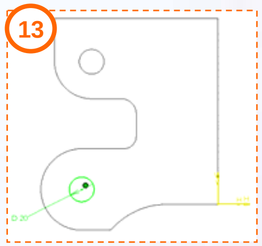
- Were the sketches easier to constrain compared to the earlier instance? Why?

15. Save and close both the documents.

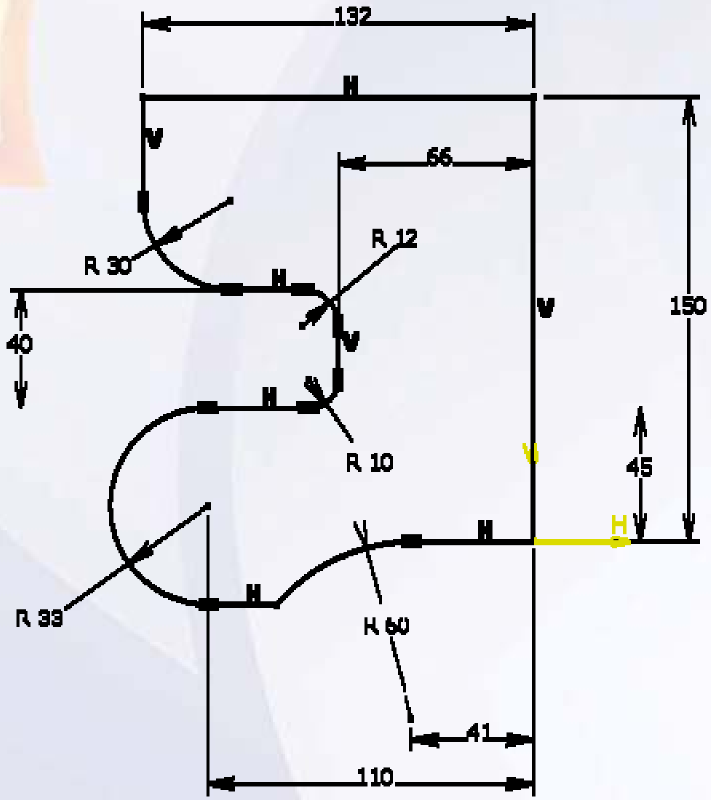


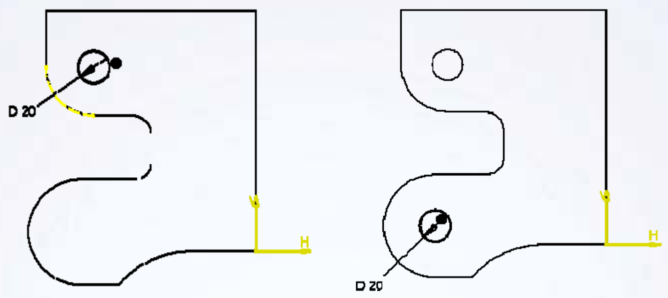






Exercise Recap: Sketch Constraints





- ✓ **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.

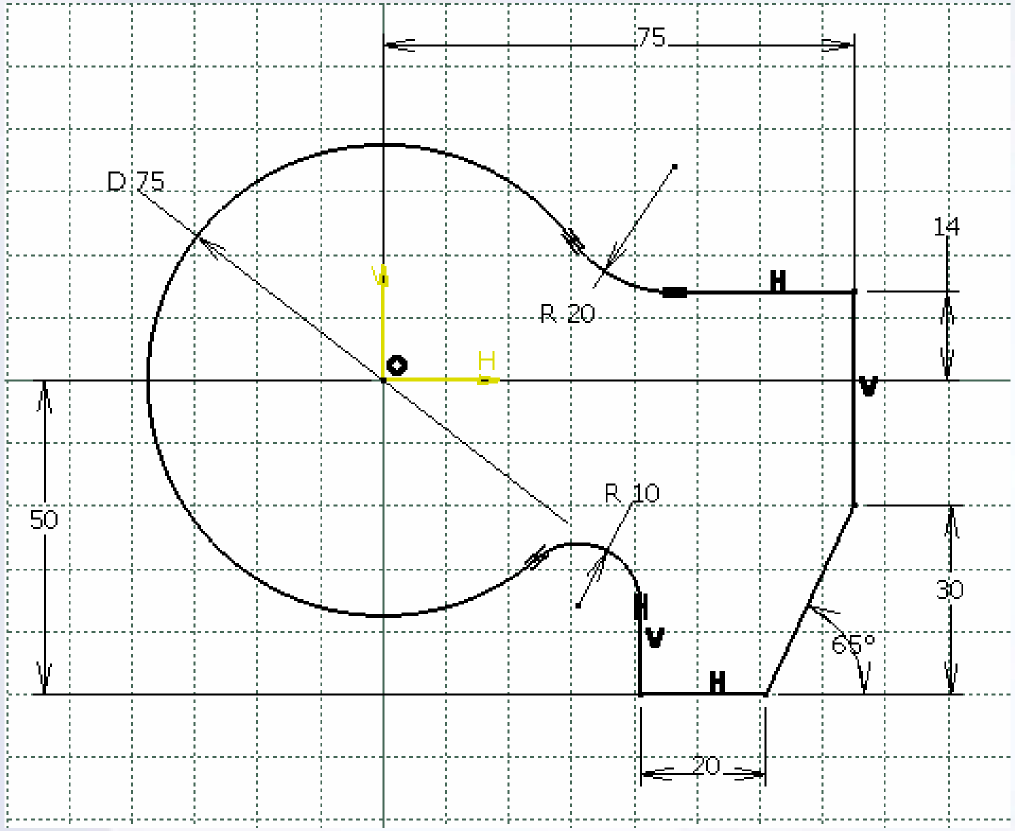
By the end of the exercise you will be able to:

- 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.

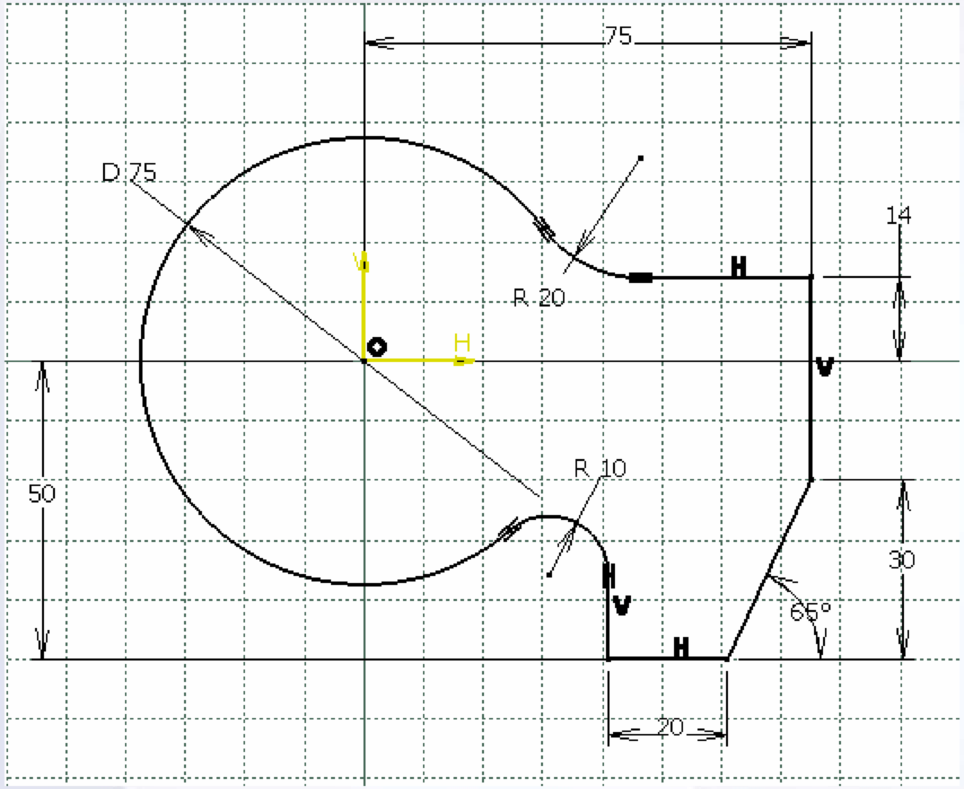
Do it Yourself

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



Exercise Recap: Sketch Constraints

- ✓ **Constrain a sketch**
- ✓ **Dimension a sketch**



Case Study: Profile Creation

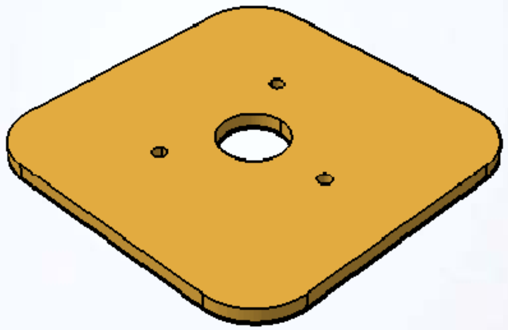
Recap Exercise

10 min

You will practice what you have learned by completing the case study model using only a detailed drawing as a guidance.

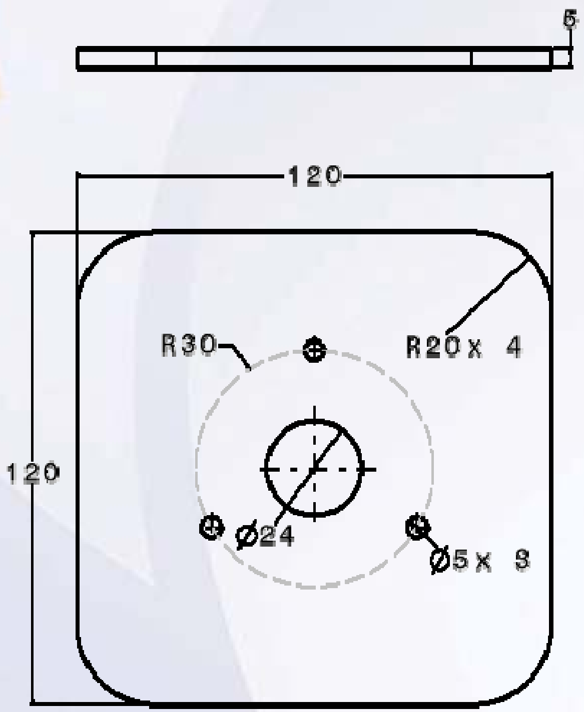
In this exercise, you will create the case study model. Recall the design intent of this model:

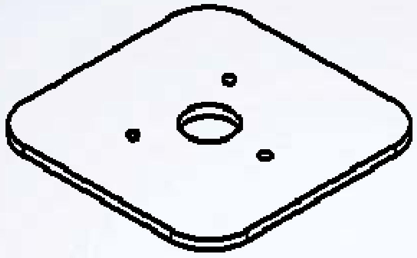
- ✓ The model must be created in one feature.
- ✓ The model must be centered along the YZ and ZX planes.
- ✓ The smaller holes must be 30mm away from the center hole.
- ✓ The center hole must remain in the center of the support.
- ✓ The model must be saved with the name Support_Plaque.



Using the techniques learnt so far, create the model without detailed instructions.

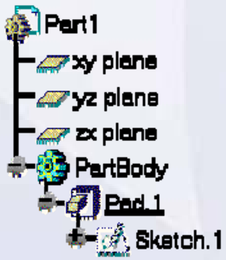
Do It Yourself: Drawing of the Support Plaque

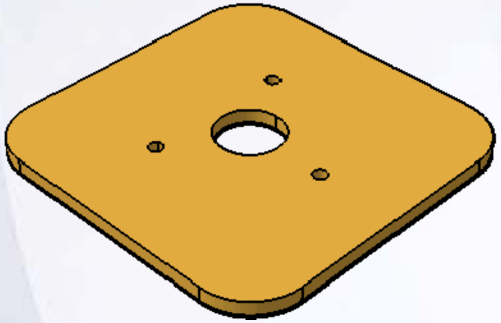


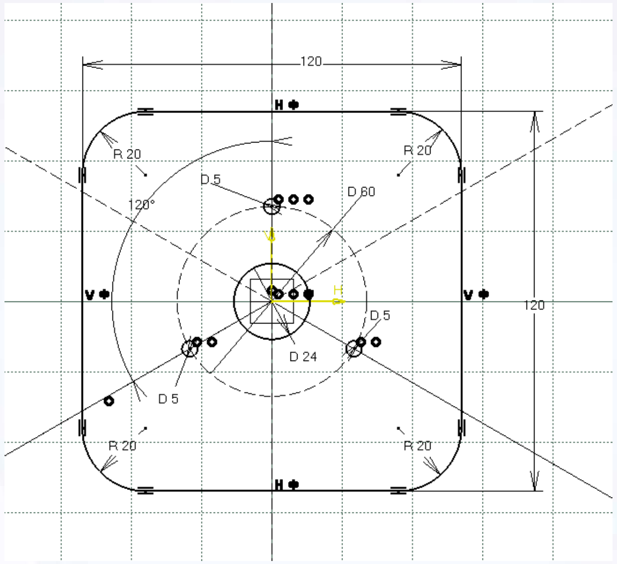


Case Study: Support Plaque Recap

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





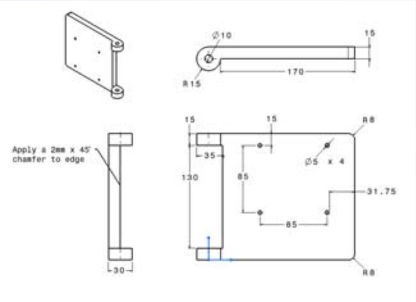


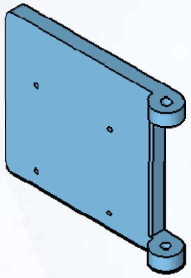
Basic Features 3

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Determine a suitable base feature.
- ✓ Create the pad and pocket features.
- ✓ Create holes.
- ✓ Create fillets and chamfers.





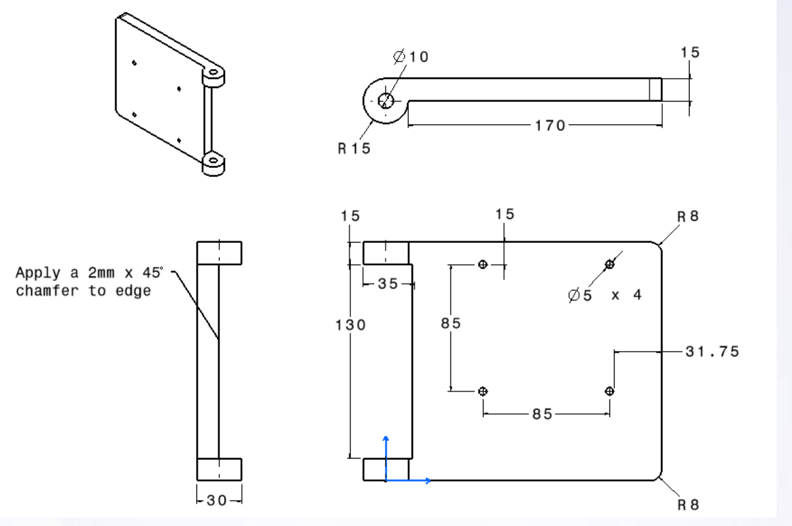
4 Hours

Case Study

The case study for this lesson is the engine support used in the drill support assembly.

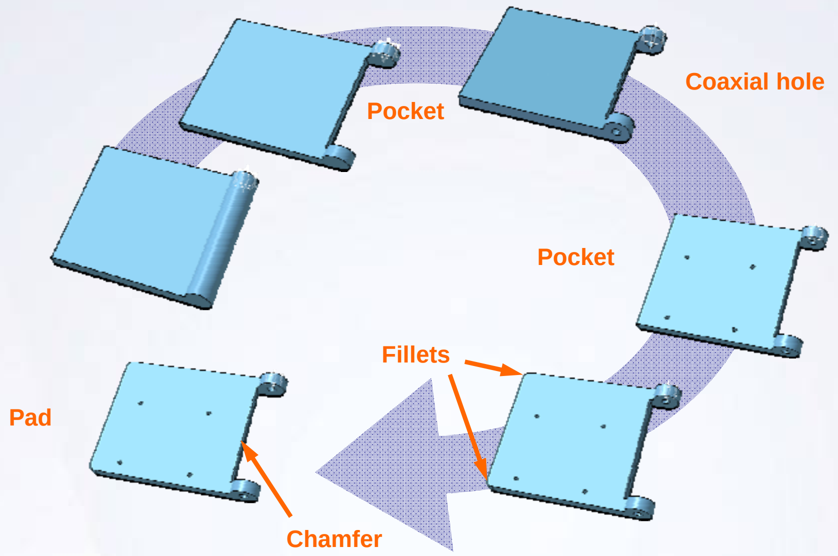
Design Intent

- ✓ Internal loops must not be created in a sketch.
  - Each element on this model must be created as a separate feature. This makes it easy to make modifications in the future.
- ✓ The four center holes must be created as one feature.
  - At first, one hole is created and then it is patterned to create the other three holes. Since the requirement is to have them created as one feature, a pocket must be used.
- ✓ The fillets and the chamfer may need to be removed in downstream applications.
  - The fillets and the chamfer cannot be created within the sketched profile; they must be created as separate features.



Stages in the Process

1. Determine a suitable base feature.
2. Create the pad and pocket features.
3. Create holes.
4. Create fillets and chamfers.



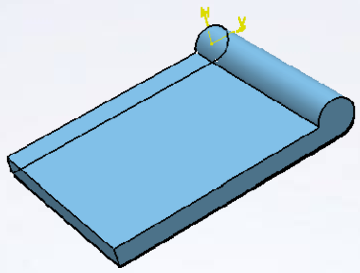
Determine a Suitable Base Feature

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.



Base Feature

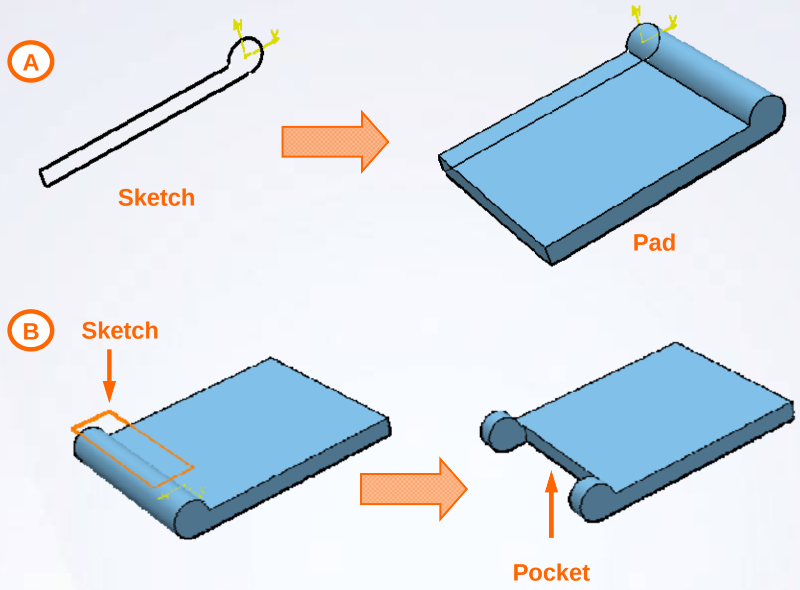
Create the Pad and Pocket Features

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.



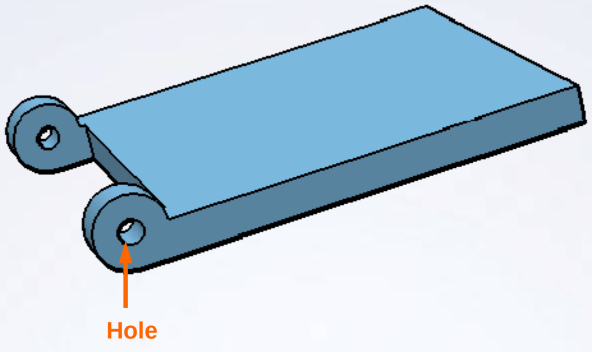
Create Holes

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.

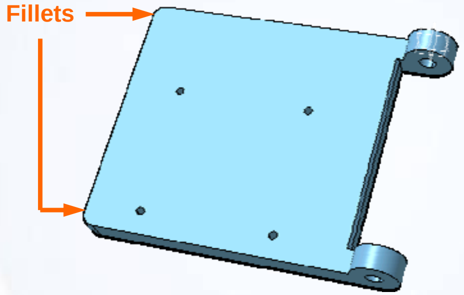
If there is a possibility that the profile for the cutout may change from circular to another shape then consider using a pocket instead of a hole.

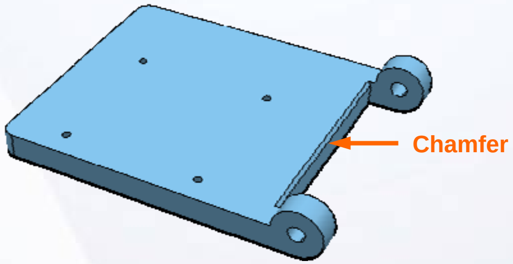


Create Fillets and Chamfers

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

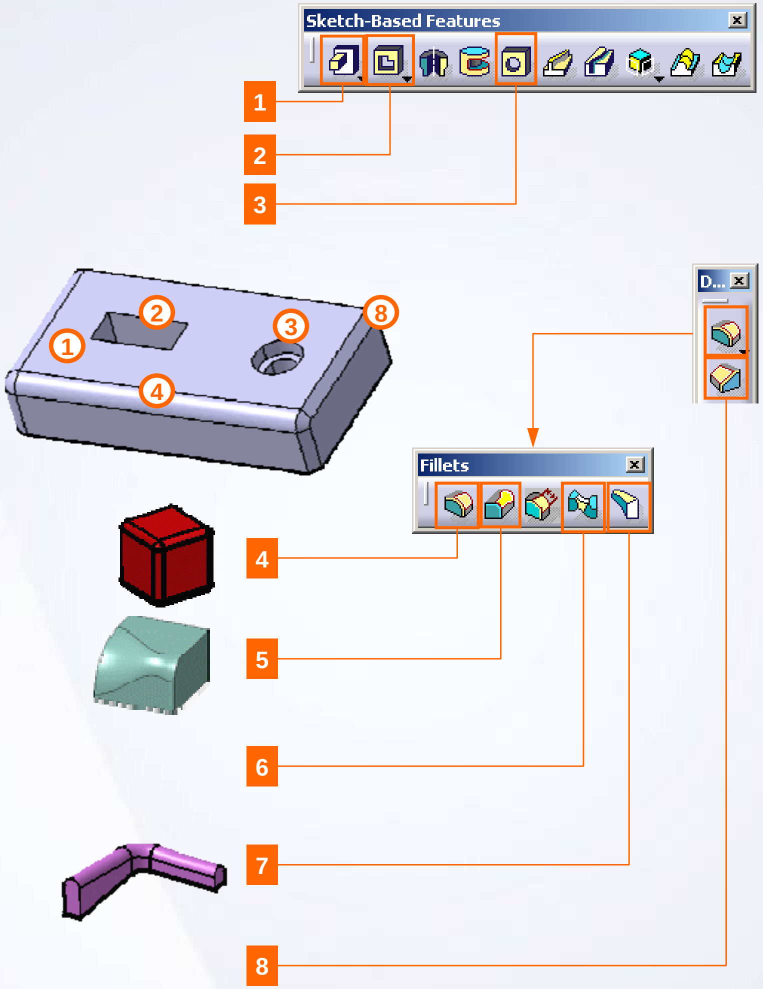
4 **Edge Fillet:** creates smooth transitional surfaces between two adjacent faces

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 Feature Creation

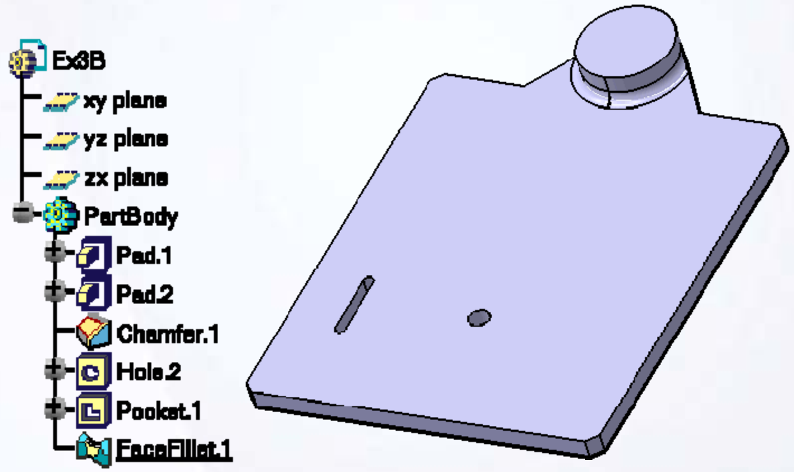
Recap Exercise

15 min

In this exercise you will open an existing part that contains a base pad feature. In the base feature you will create a pocket, a face-face fillet and chamfer. High-level instructions for this exercise are provided.

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

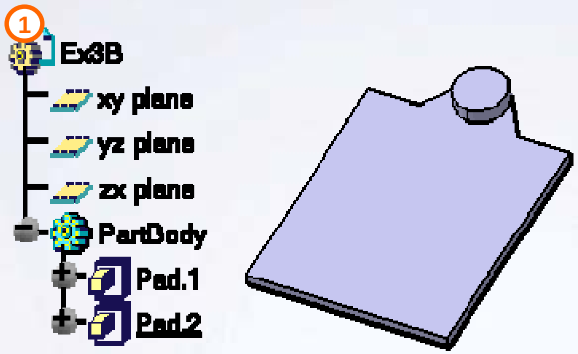
- Create a hole
- Create a pocket
- Create a face-face fillet
- Create a chamfer

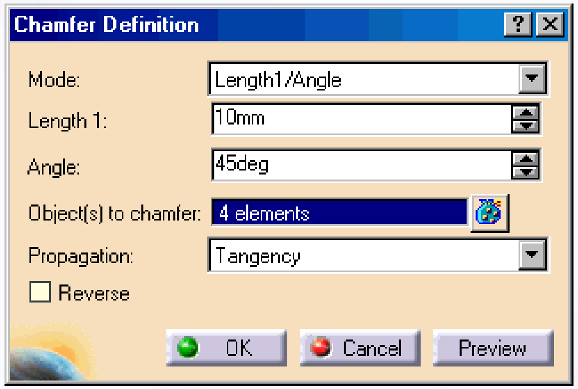


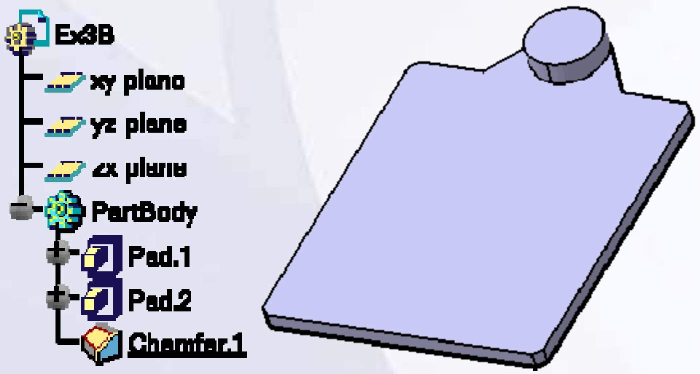
Do it Yourself (1/4)

1. **Open up the part Ex3B.CATPart.**
   - Open an existing part file using the **Open** tool. The part file constrains two pad features.

2. **Create four chamfers.**
   - Create chamfers on the four vertical edges of Pad.1.



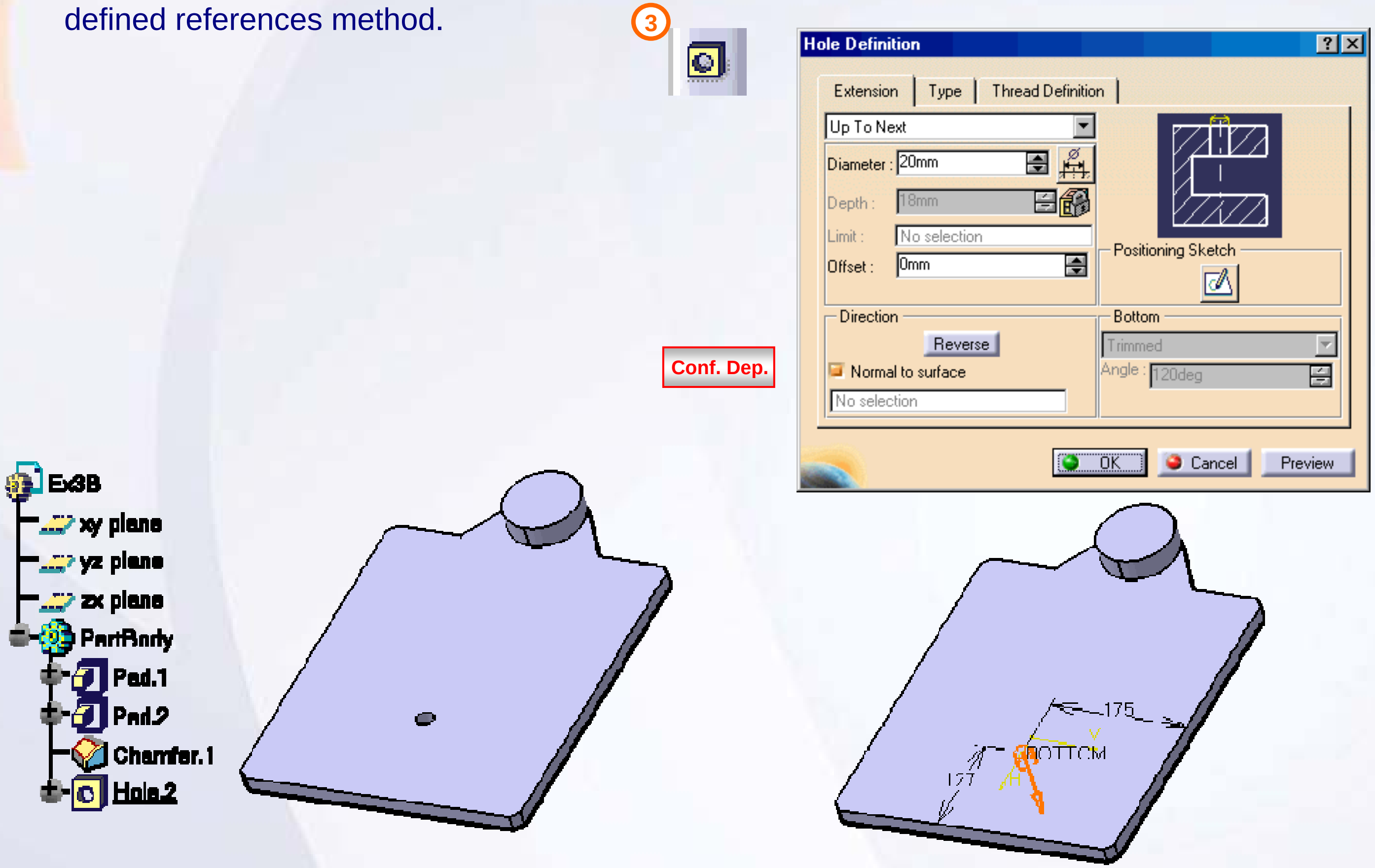




Do it Yourself (2/4)

3. Create a simple hole.

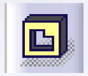
- Create a simple hole using the pre-defined references method.

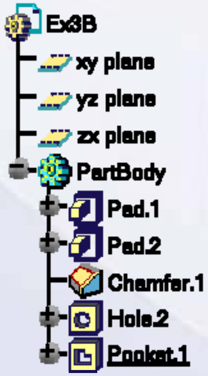


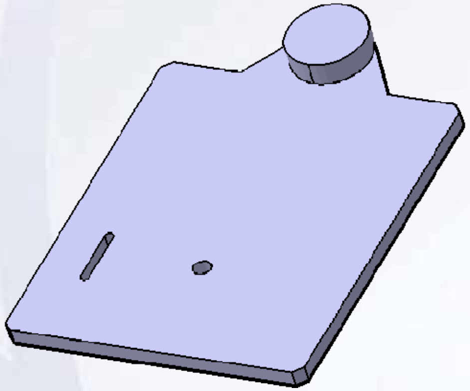
Do it Yourself (3/4)

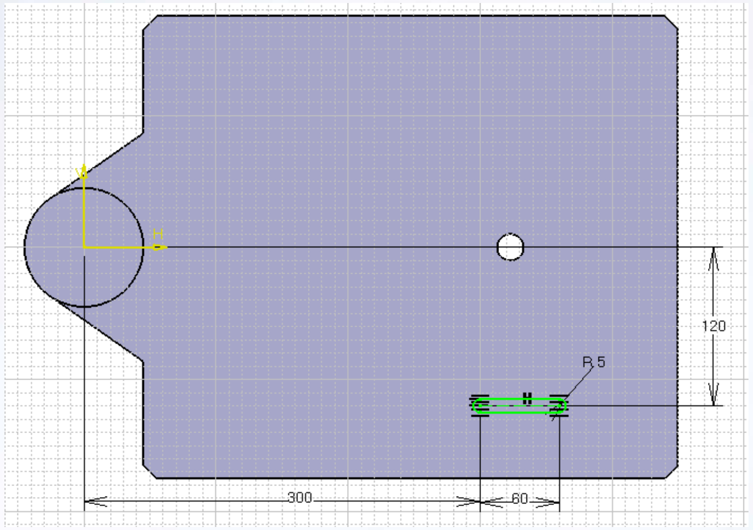
4. **Create a pocket.**
   - Create an **Up to Last** pocket using the dimension shown.

4







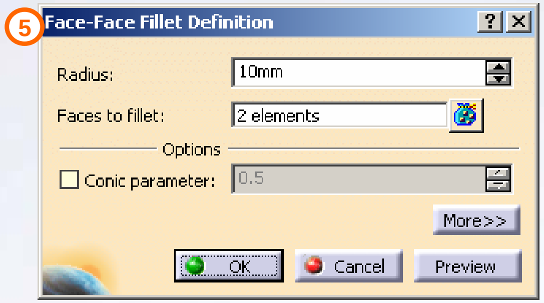


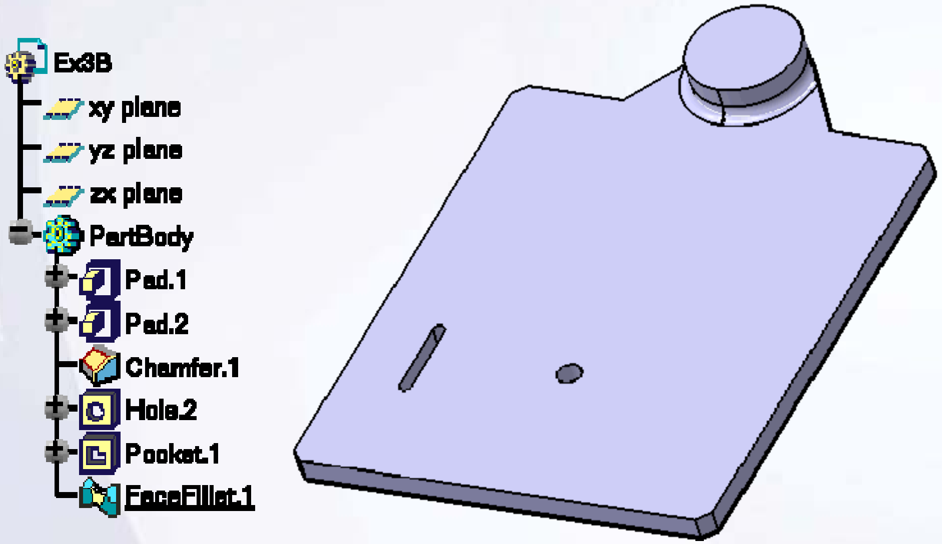
Do it Yourself (4/4)

5. Create a face-face fillet.

- Create a face-face fillet between surfaces on Pad.1 and Pad.2.

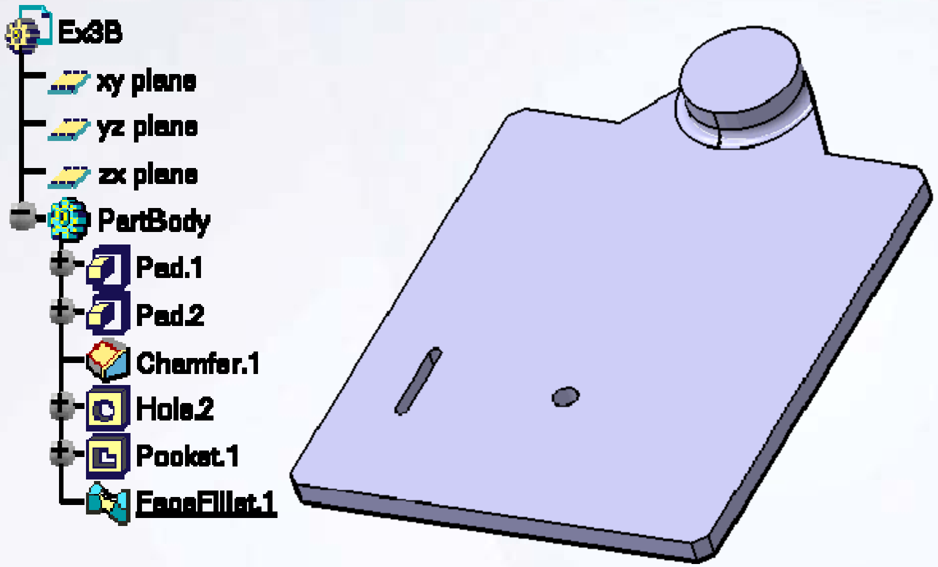
6. Save and close the file.





Exercise Recap: Basic Feature Creation

- ✓ Create a hole
- ✓ Create a pocket
- ✓ Create a face to face fillet
- ✓ Create a chamfer



Exercise: Basic Features Creation

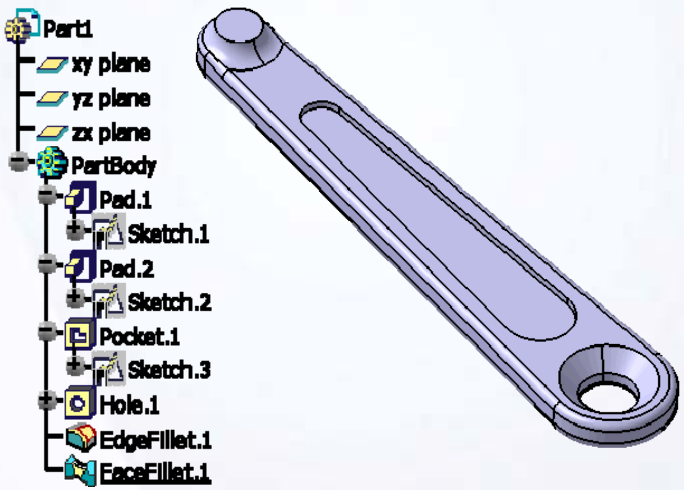
Recap Exercise

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.

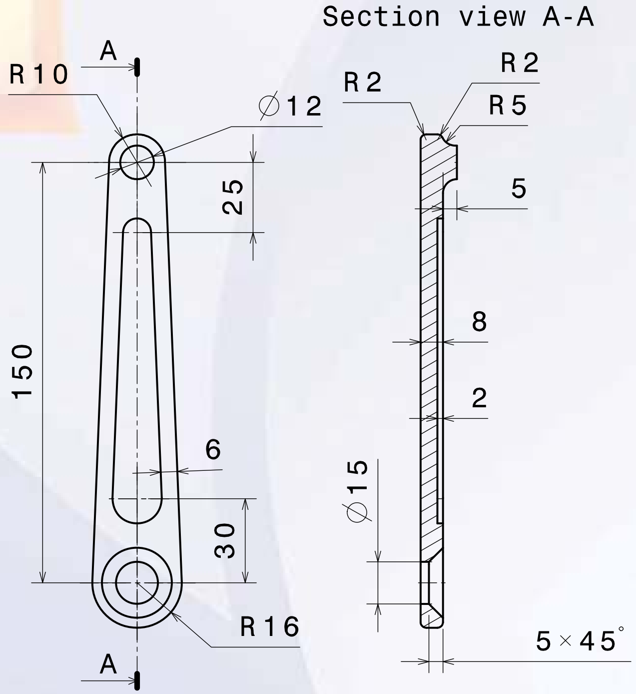
By the end of this exercise you will be able to:

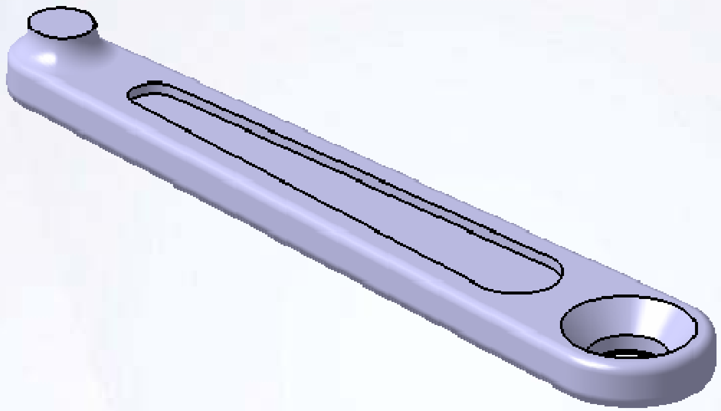
- **Create a pad**
- **Create a pocket**
- **Create a countersunk hole**
- **Create an edge fillet**



Do it Yourself

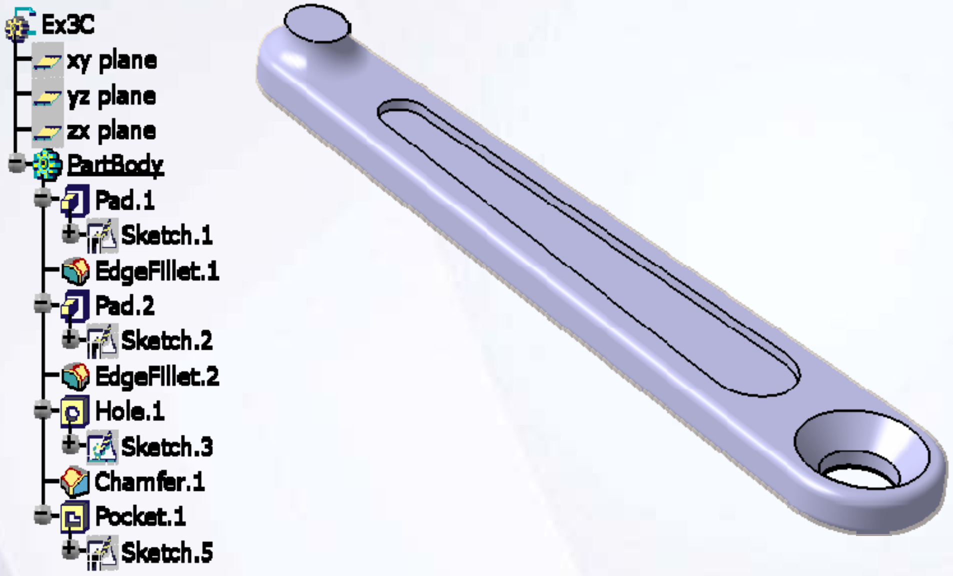
1. **Create the following part.**





Exercise Recap: Basic Features Creation

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



Exercise: Edge and Face-Face Fillets

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.

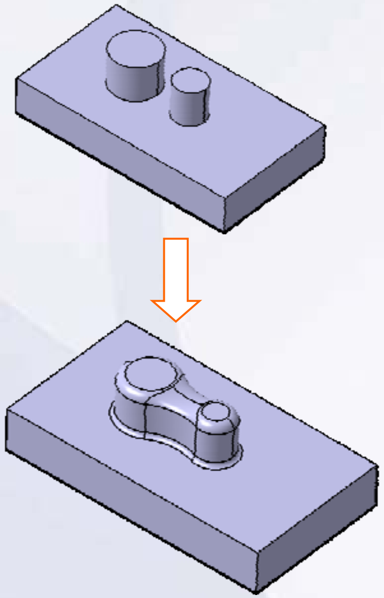
By the end of this exercise you will be able to:

- **Create a face-face fillet**
- **Create the necessary additional fillet in order to enable face-face fillet creation**

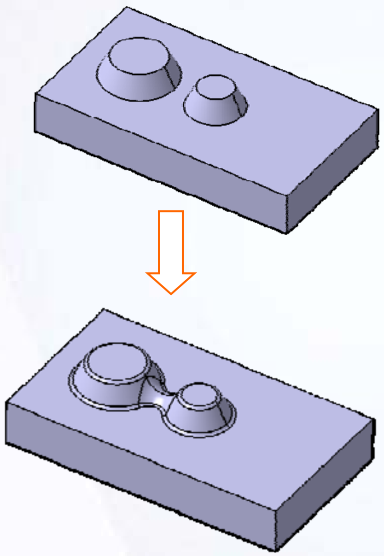
Do it Yourself

1. Add edge fillets to the top faces of the following parts.
2. Add face-face fillets by determining the radius yourself. Afterwards add bottom edge fillets.
3. Change the distance between the cylindrical / drafted pads and the preliminary edge fillet's radius and examine the impact on the face-face fillet

Ex3D_A.CATPart

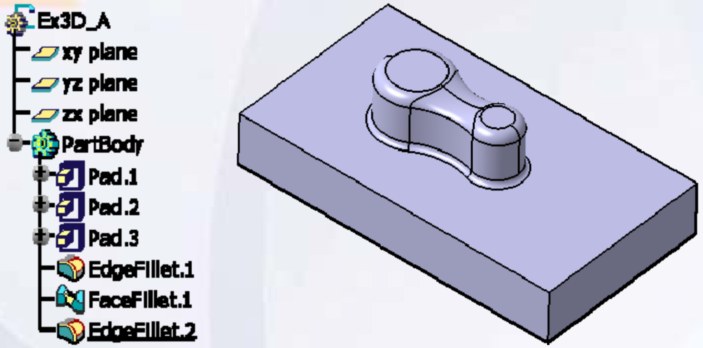


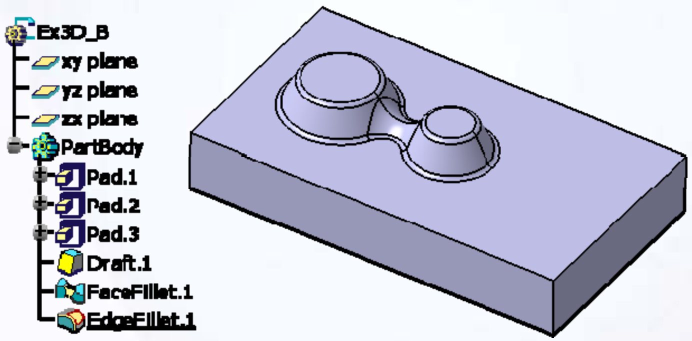
Ex3D_B.CATPart



Exercise Recap: Edge and Face-Face Fillets

- ✓ Create edge fillets in order to enable face-face fillet creation
- ✓ Create face-face fillets





Case Study: Basic Features

Recap Exercise

20 min

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

✓ The sketch must not contain any internal loops.

- Each element on this model will need to be created as a separate feature. Creating the elements separately makes it easy to make modifications later.

✓ The four center holes must be created as one feature.

- One hole would be created first and then patterned to create the other three holes. Since the requirement is to have them created as one feature, a pocket will need to be used.

✓ The fillets and the chamfer may need to be removed in downstream applications.

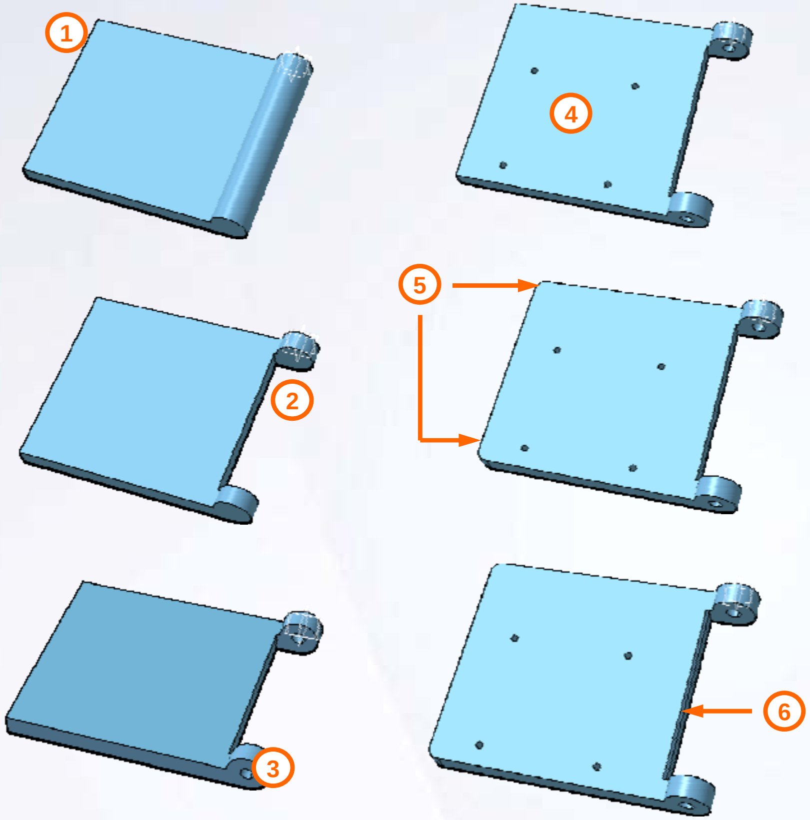
- The fillets and the chamfer cannot be created within the sketched profile; they will have to be created as separate features.

Using the techniques discussed so far, create the model without detailed instructions.

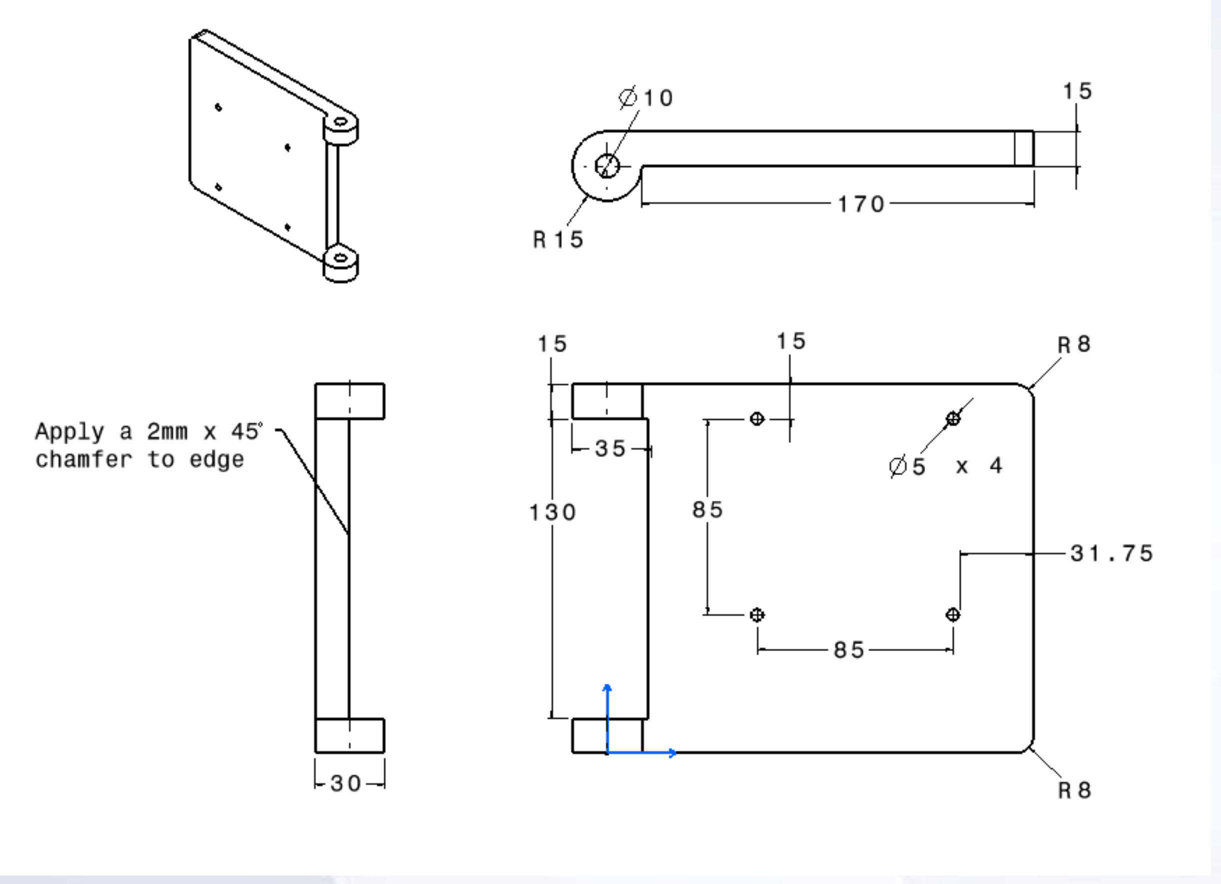
Do It Yourself: Drawing of the Engine Support (1/2)

You will be required to create the following features:

1. Pad
2. Pocket
3. Coaxial hole
4. Pocket
5. Fillets
6. Chamfer

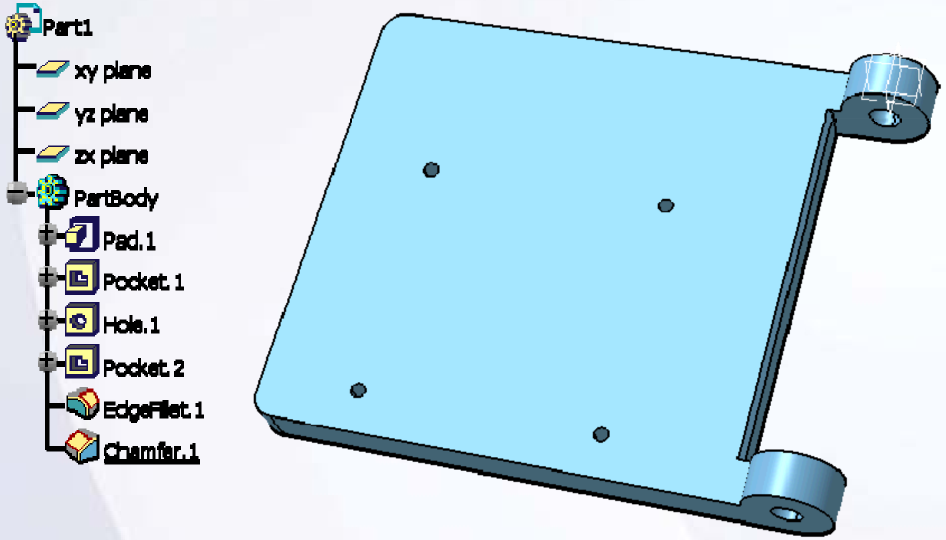


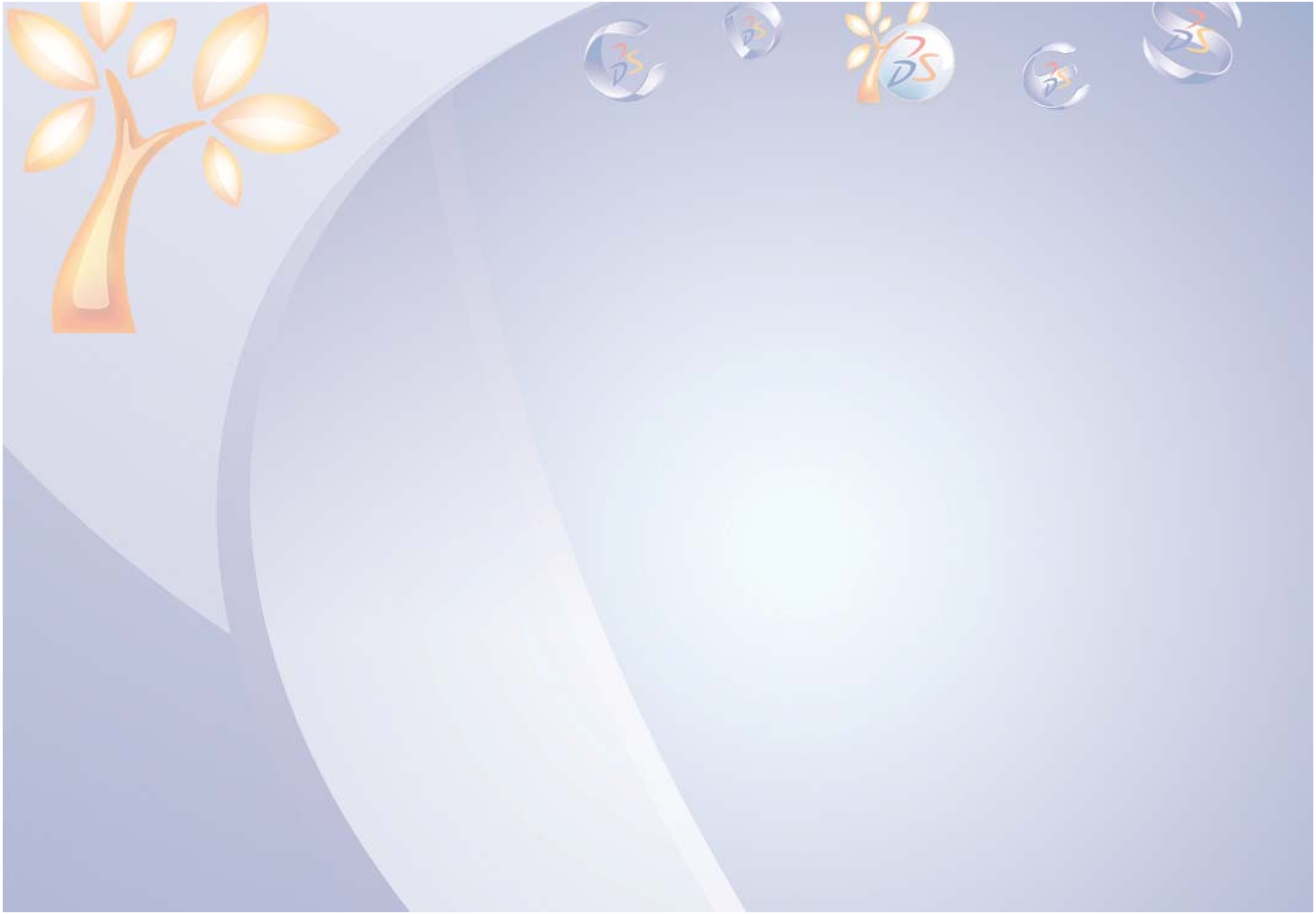
Do It Yourself: Drawing of the Engine Support (2/2)



Case Study: Engine Support Recap

- ✓ Select a base feature
- ✓ Create a pad
- ✓ Create a pocket
- ✓ Create holes
- ✓ Create edge fillets
- ✓ Create chamfers



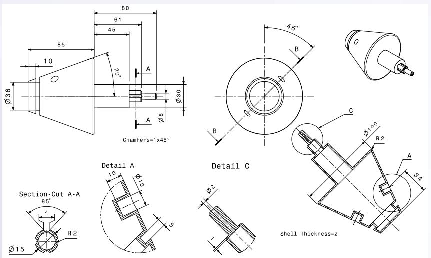


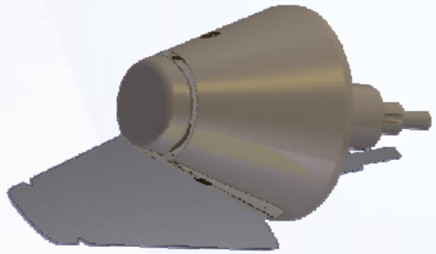
Additional Part Features 4

Learning Objectives

Upon completion of this lesson you will** be **able to:

- ✓ Create feature profiles and Axis system.
- ✓ Create Multi-profile Sketch Feature
- ✓ Create basic wireframe geometry.
- ✓ Create shaft and groove features.
- ✓ Shell the model.





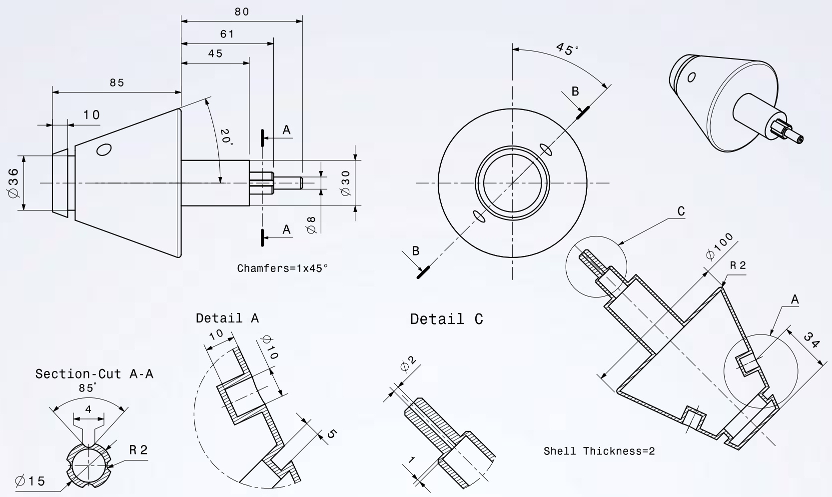
4 Hours

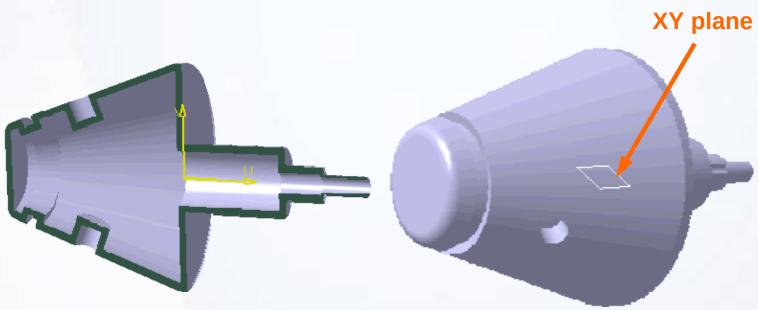
Case Study

The case study for this lesson is the Handle Block used in the Drill Press assembly.

Design Intent

- ✓ The top and the bottom portions of the model must be created as separate features.
- ✓ The holes must be created at an angle to the XY plane.
- ✓ The model must be hollow and must have a uniform thickness of 3mm, except the end which must have a thickness of 1mm.
- ✓ The holes must be normal to the sides of the handle block.

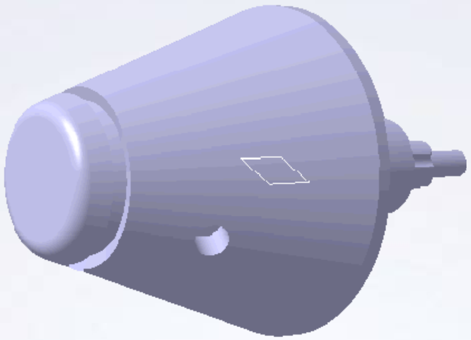




X-section of handle block

Stages in the Process

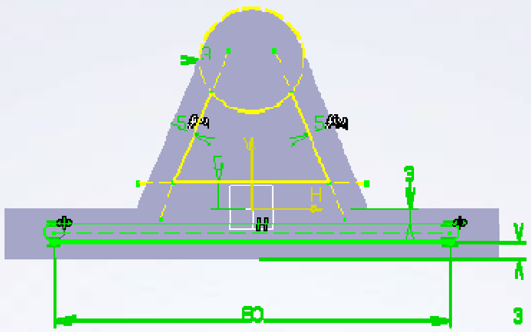
1. Create feature profiles and Axis system.
2. Create Multi-profile Sketch Feature
3. Create basic wireframe geometry.
4. Create shaft and groove features.
5. Shell the model.



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.

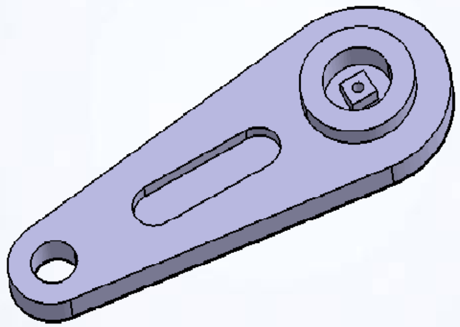
Sketcher includes the following additional tools:



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

Create Multi-profile Sketch Feature

Multi-pads and pockets are features that create several pads/pockets in one operation. These tools require a sketch with at least two closed profiles. Consider using these tools as a fast way to create multiple features

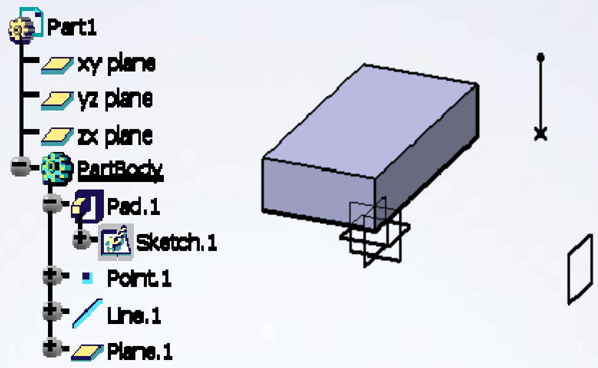


Careful thought must be given to the profiles created in the sketch when they are used to define a Multi-Pad/Pocket. The profiles cannot intersect, they must form a closed loop to avoid feature definition error.

Create Basic Wireframe Geometry

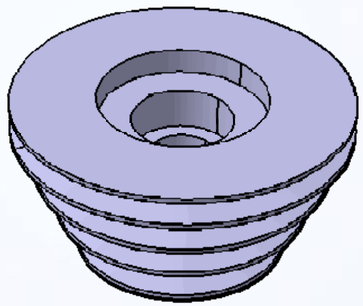
In the Part Design workbench, you have the ability to create points, lines, and planes outside of the Sketcher environment. These elements are called reference or 3D wireframe geometry.

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.



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:



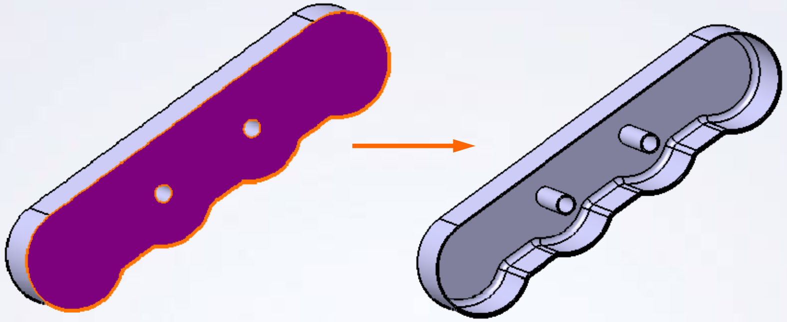
The axis of revolution for a revolved feature can be created inside the sketch containing the profile, using the Axis tool.

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.

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.

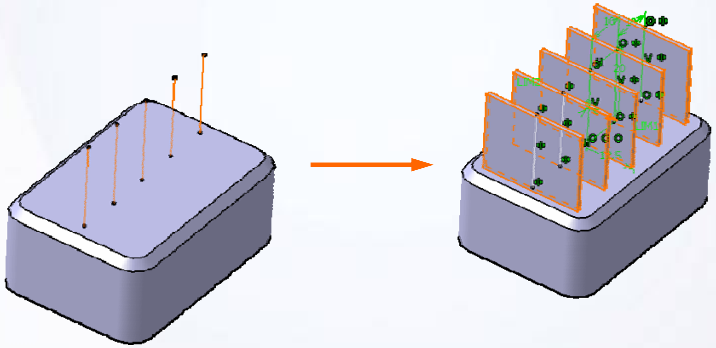
While shelling a model, it is important to consider the feature order. The Shell operation hollows all solid features in a model. If you do not want a feature to be shelled, it must be created after the shell operation.



Create Thin Features

A thin feature is created by applying a constant thickness to a profile. The definition dialog boxes for pads, pockets, shafts, and grooves contain a section for defining a thin feature.

- ✓ A thin feature can be created with a closed or open profile.
- ✓ Thickness can be applied to one side or both sides of the profile.



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:** helps to resolve problems with a sketch

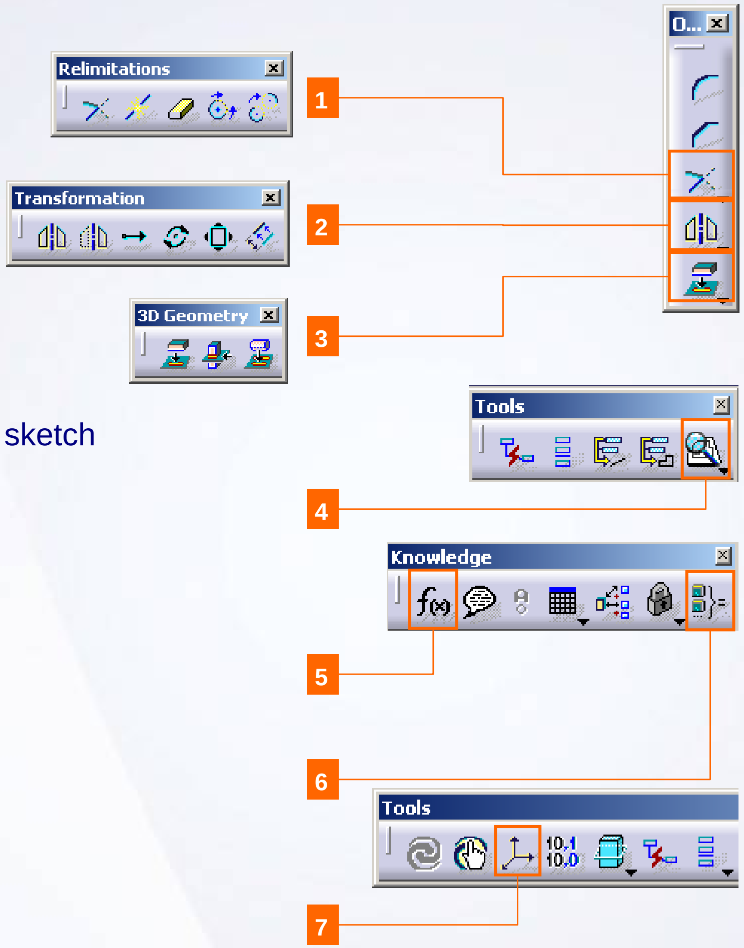
Knowledge

5 **Formula:** creates 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

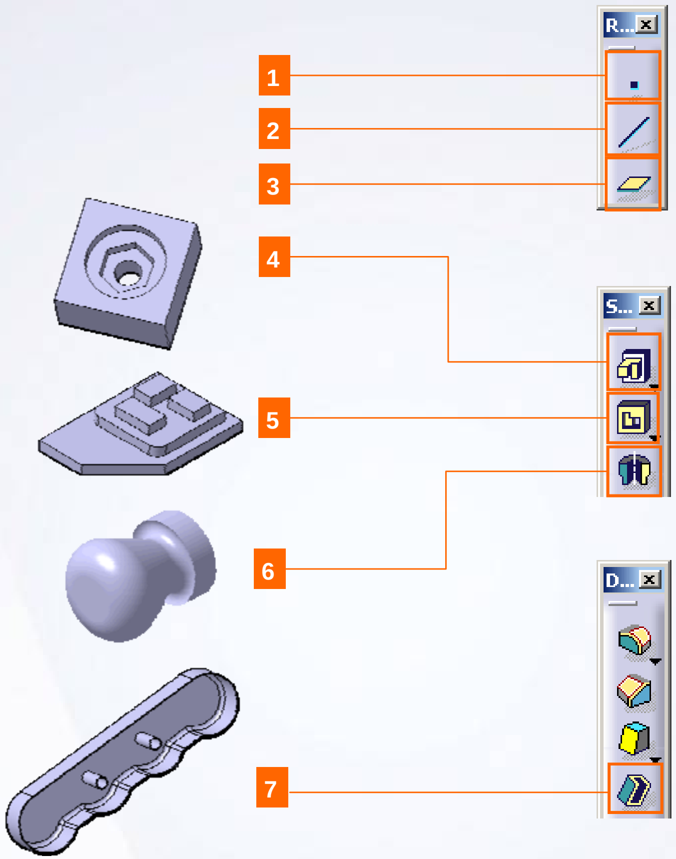
4 **Multi-pad:** creates several pads in one operation

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: Sketch Analysis and Pocket

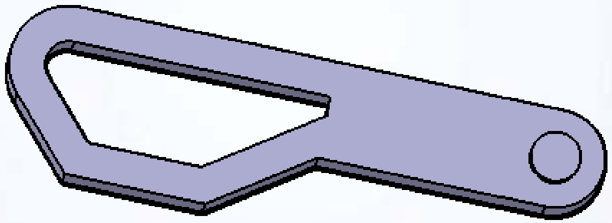
Recap Exercise

15 min

In this exercise, you will open an existing part that contains a multi-profile sketch. You will use this sketch to create several features. High-level instructions for this exercise are provided.

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

- Problem-solve a sketch
- Use the Sketch Analysis tool
- Create a pad using a sub-element of a sketch
- Create a multi-pocket

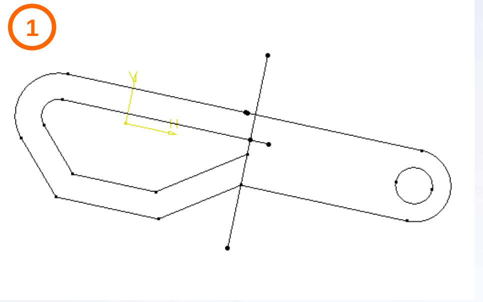


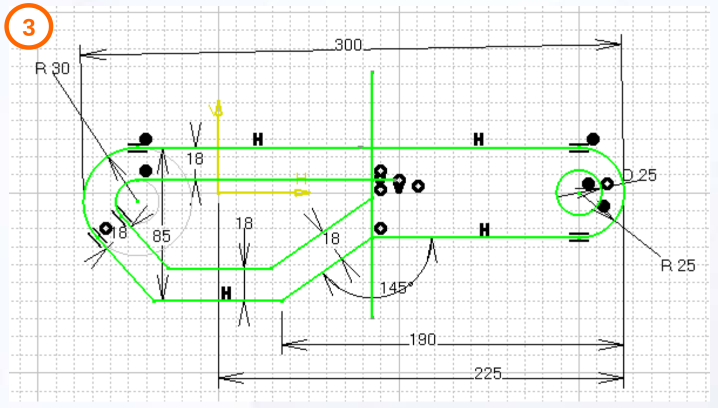
Do it Yourself (1/7)

1. **Open Ex4B.CATPart.**
   - Open an existing part file.

2. **Create multi-pad feature.**
   - Create the multi-pad feature using the sketch given. An error message appears, indicating that the sketch is not valid. Cancel the multi-pad creation.

3. **Edit the sketch.**
   - Access the Sketcher workbench for Sketch.1 to investigate the sketch.

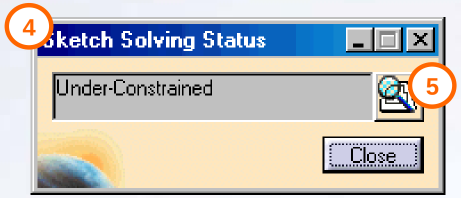




Do it Yourself (2/7)

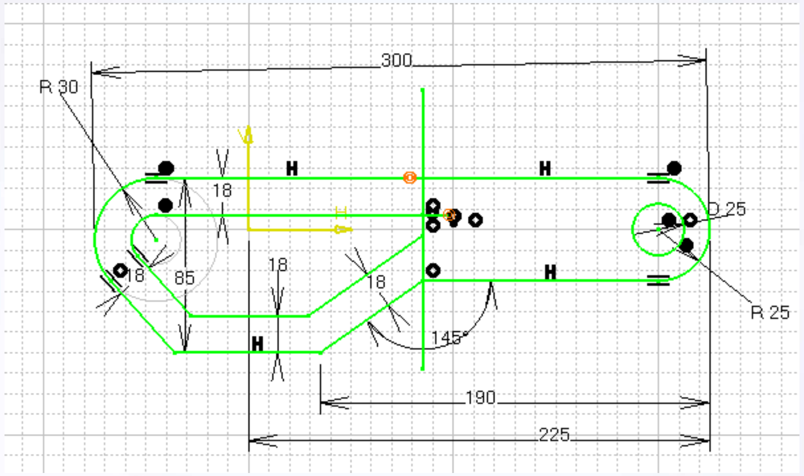
4. Use the Sketch Solving Status Tool.

- Use the Sketch Analysis tool to investigate what is wrong with the sketch. Although the sketch appears to be completely green (Iso-constrained), the status of the sketch is actually under-constrained. Observe the highlighted points.



5. Access the Sketch Analysis tool.

- Use the Sketch Analysis tool to further investigate the sketch.



Do it Yourself (3/7)

6. Review the geometry.

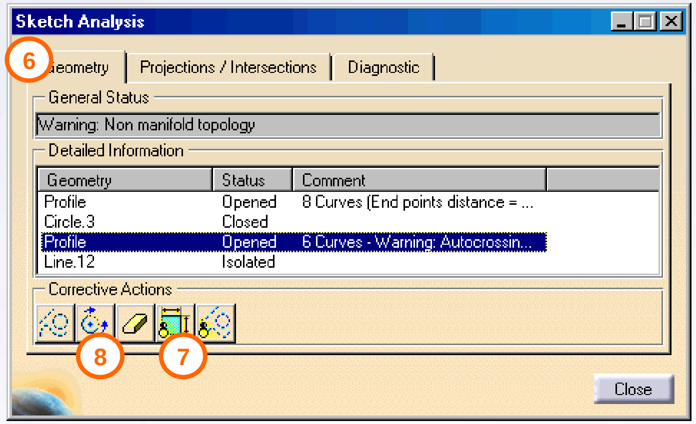
- Select the **Geometry** tab. The sketch contains two open profiles and an isolated line.

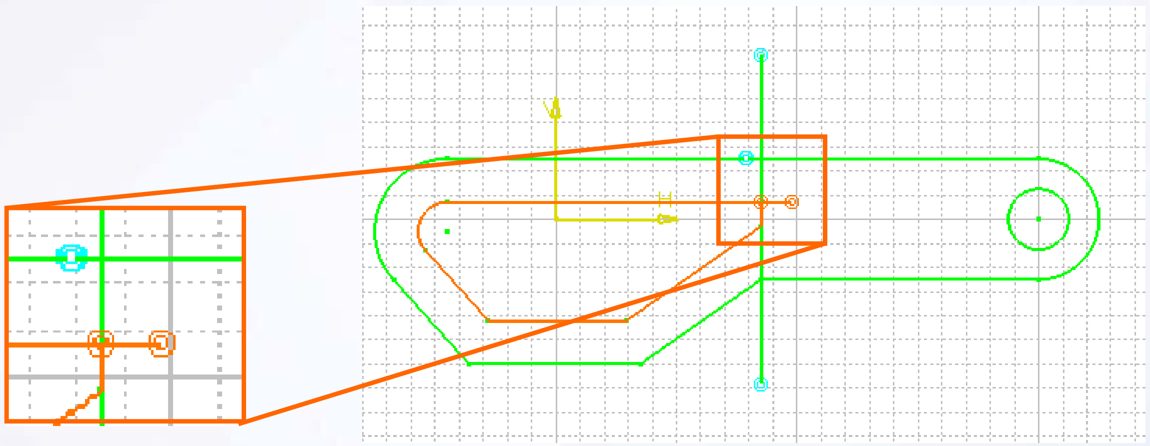
7. Remove the constraints from the display.

- To simplify the display, select the **Hide Constraints** icon.

8. Resolve the open profile.

- Select the inside open profile and use the **Close Opened Profile** icon to resolve the issue.

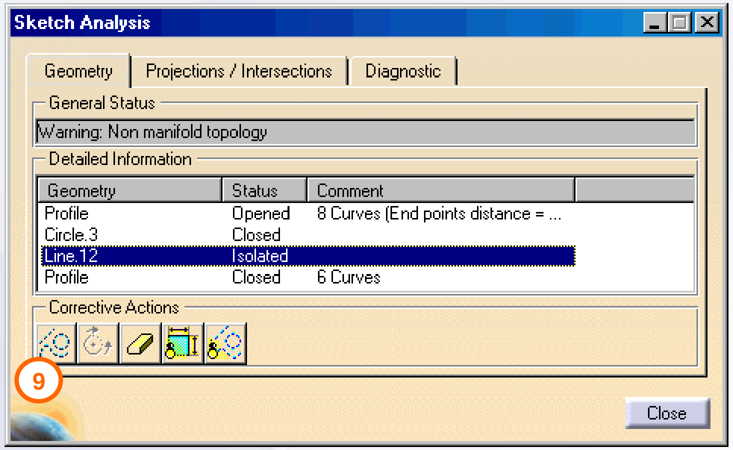


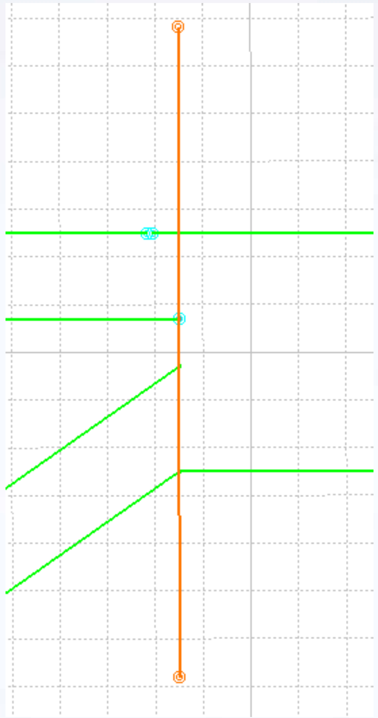


Do it Yourself (4/7)

9. Resolve the isolated line.

- Select the isolated line in the window. Observe where it is located in the model. This line should be a construction element. Use the **Set in Construction Mode** icon to convert the point.

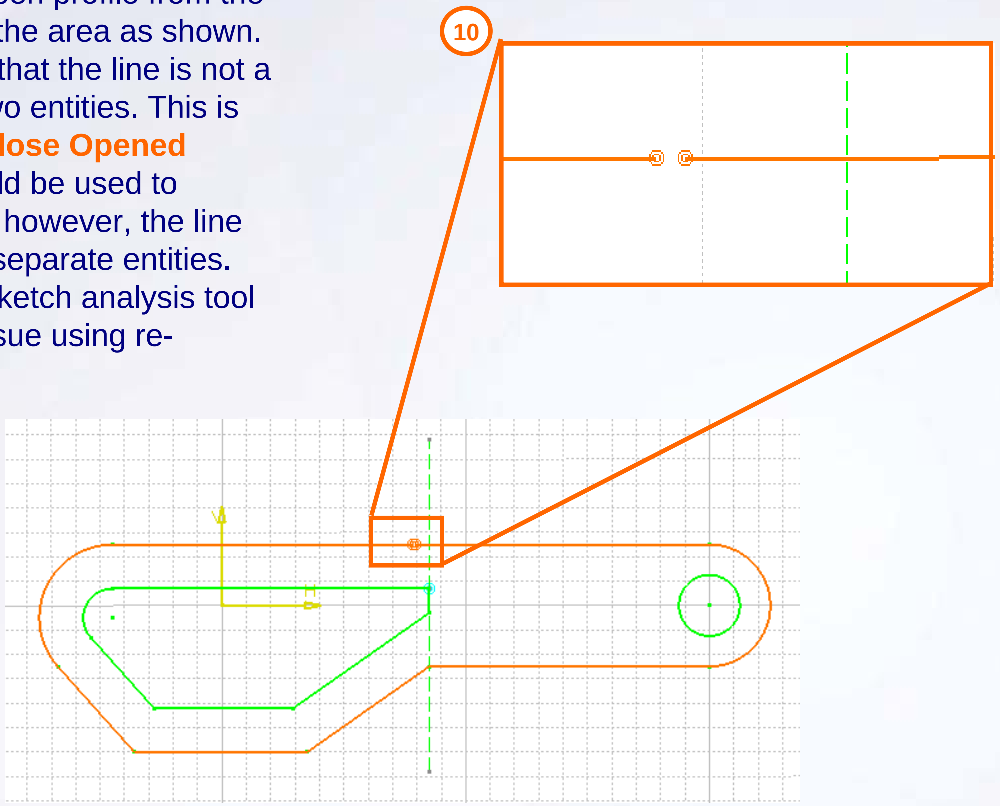




Do it Yourself (5/7)

10. Review the second open profile.

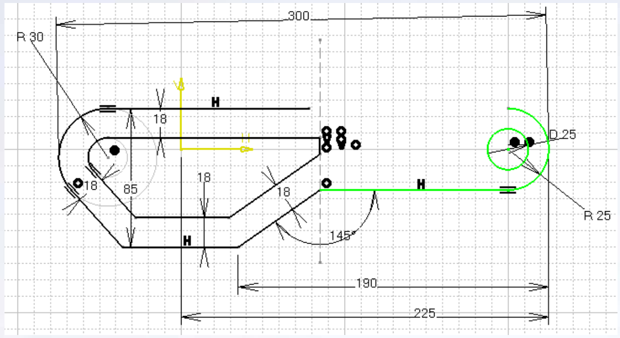
- Select the outer open profile from the window. Observe the area as shown. Zoom in; observe that the line is not a single entity but two entities. This is not correct. The **Close Opened Profile** option could be used to resolve this issue; however, the line would still be two separate entities. Instead, exit the Sketch analysis tool and resolve the issue using re-limitation tools.



Do it Yourself (6/7)

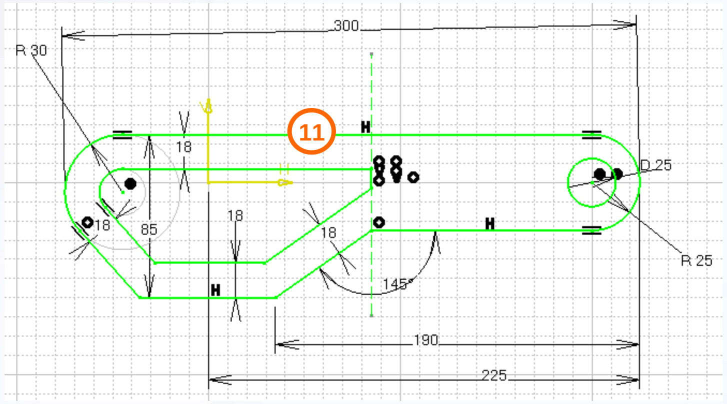
11. Resolve the second open profile.

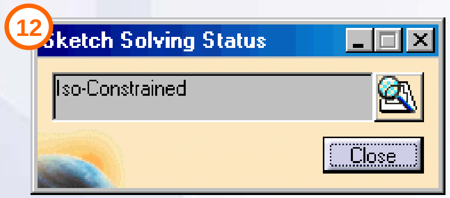
- Delete one of the top lines. Use the **Trim** tool to extend the remaining line. Remember to add tangency between the line and the arc.



12. Re-analyze the sketch.

- Return to the Sketch solving status window. The sketch should now be iso-constrained.

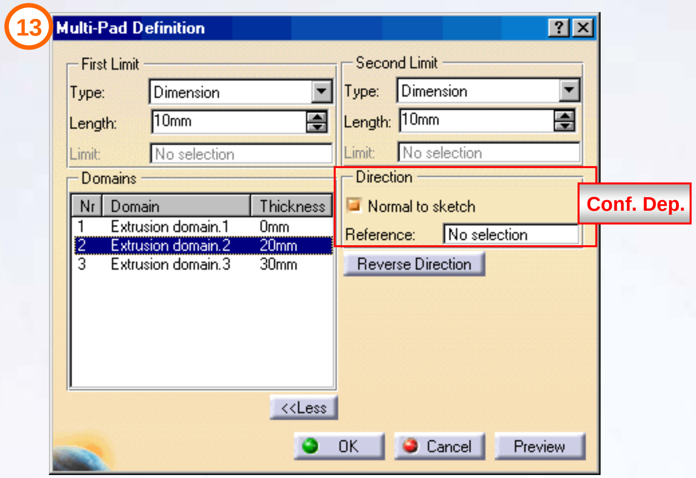


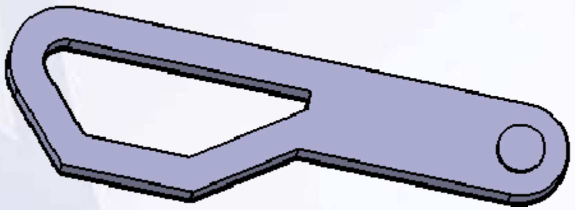


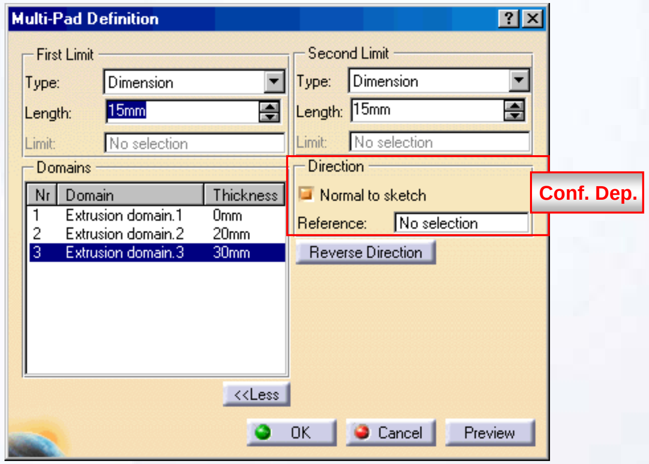
Do it Yourself (7/7)

13. Exit the sketcher and create the multi-pad feature as shown.

14. Save and close the file.

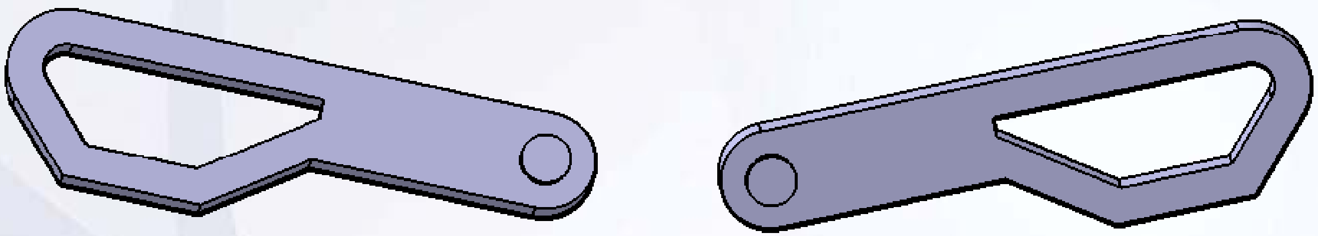






Exercise Recap: Sketch Analysis and Pocket

- ✓ **Problem-solve a sketch**
- ✓ **Use the Sketch Analysis tool**
- ✓ **Use re-limitation tools**
- ✓ **Create a multi-pad feature**



Exercise: Multiple Profile Sketch Features

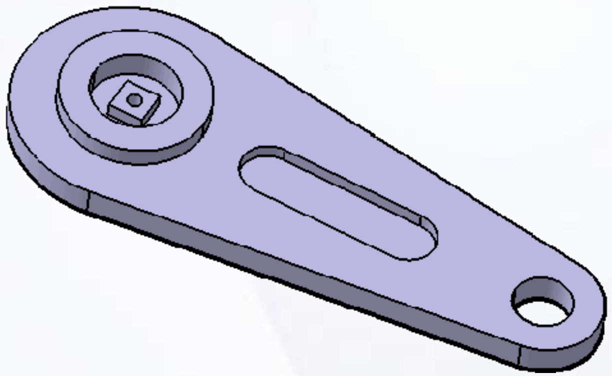
Recap Exercise

15 min

In this exercise, you will create a part that contains two features, a multi-pad, and a multi-pocket. 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 multi-pad
- Create a multi-pocket

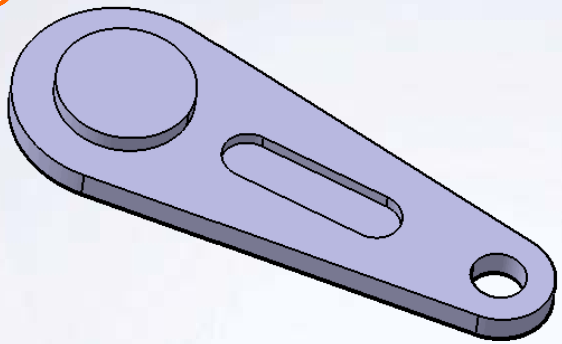


Do it Yourself (1/2)

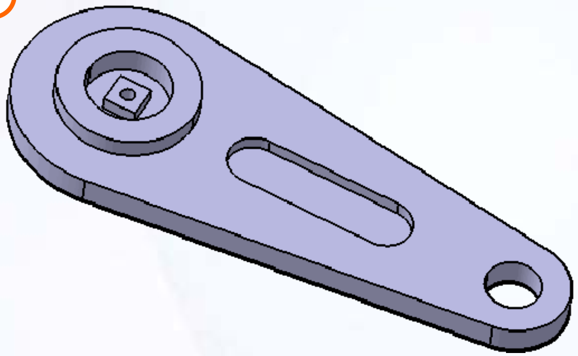
You need to create the following features:

1. Multi-pad
2. Multi-pocket

1

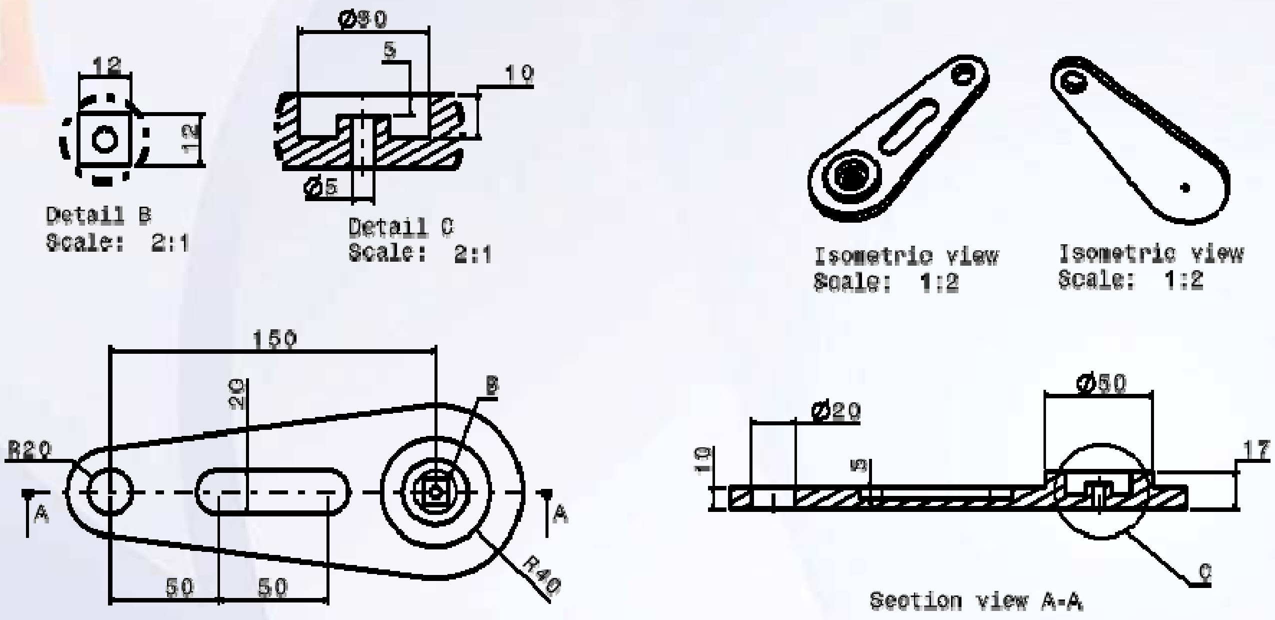


2



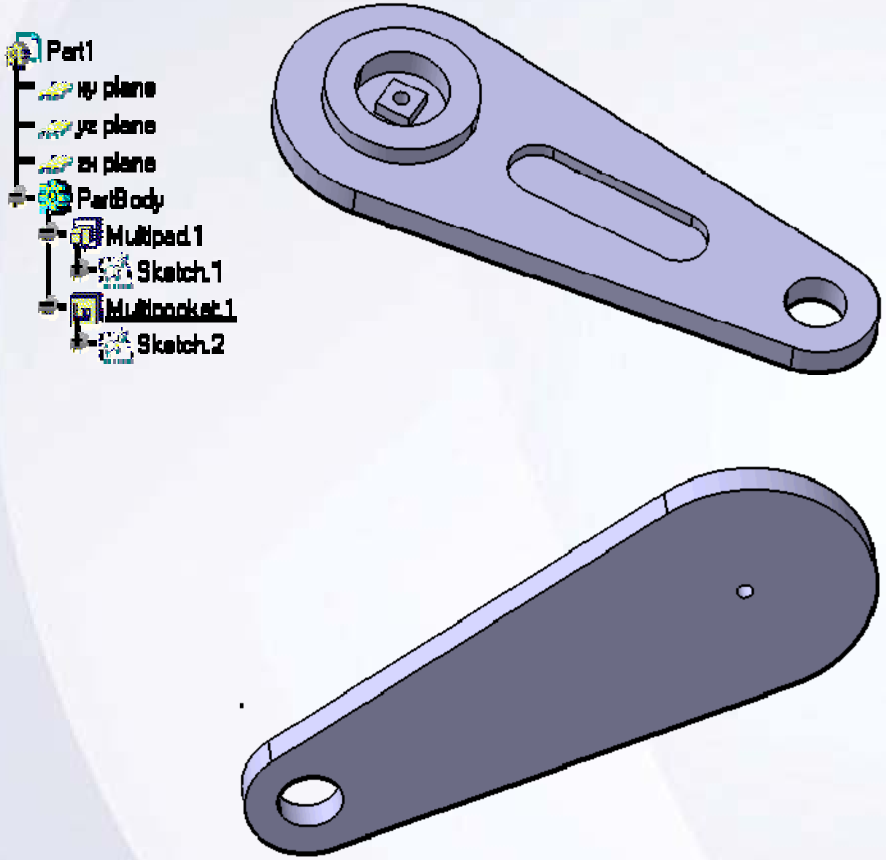
Do it Yourself (2/2)

1. Create the following part.



Exercise Recap: Multiple Profile Sketch Features

- ✓ Create a multi-pad
- ✓ Create a multi-pocket



Exercise: Shaft and Groove

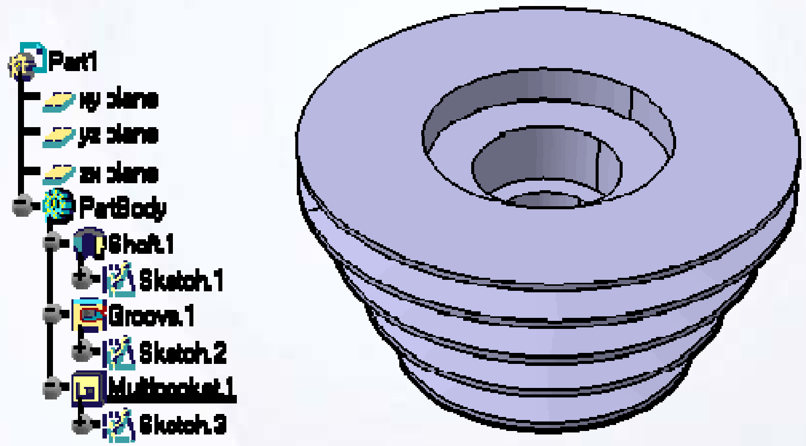
Recap Exercise

15 min

In this exercise you will create a new part. Using the shafts, grooves, and multi-pocket features, you will construct a pulley. High-level instructions for this exercise are provided.

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

- **Create a shaft**
- **Create a groove**
- **Create a multi-pad**



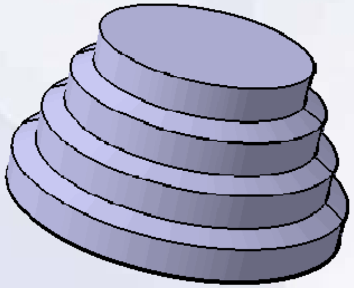
Do it Yourself (1/3)

1. Create a new part file.

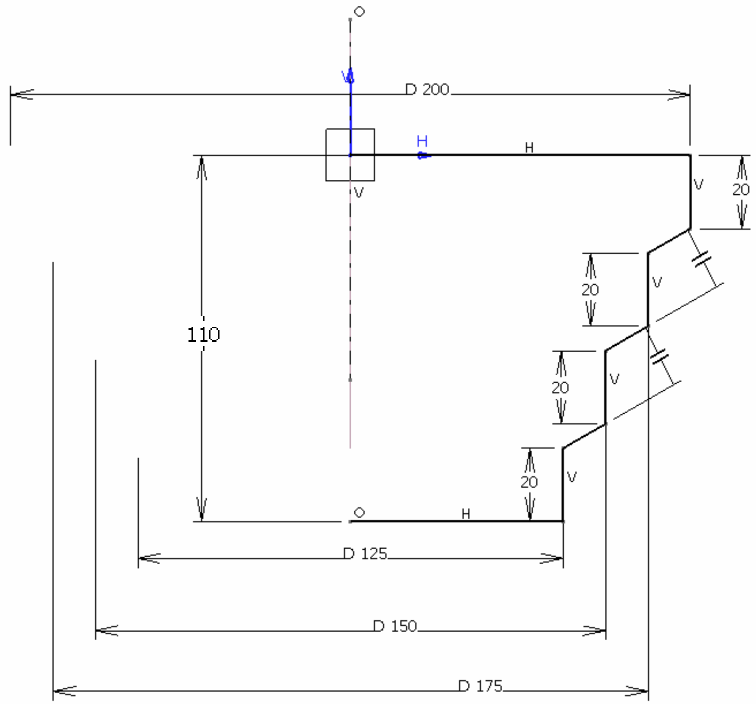
- Create a new part file called [Ex4E.CATPart].

2. Create a shaft feature.

- Use the dimension shown to construct a shaft feature.



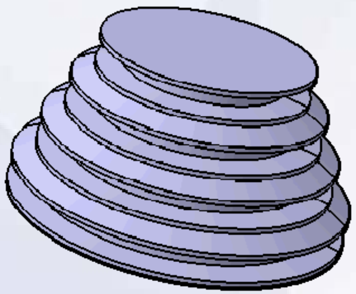
2

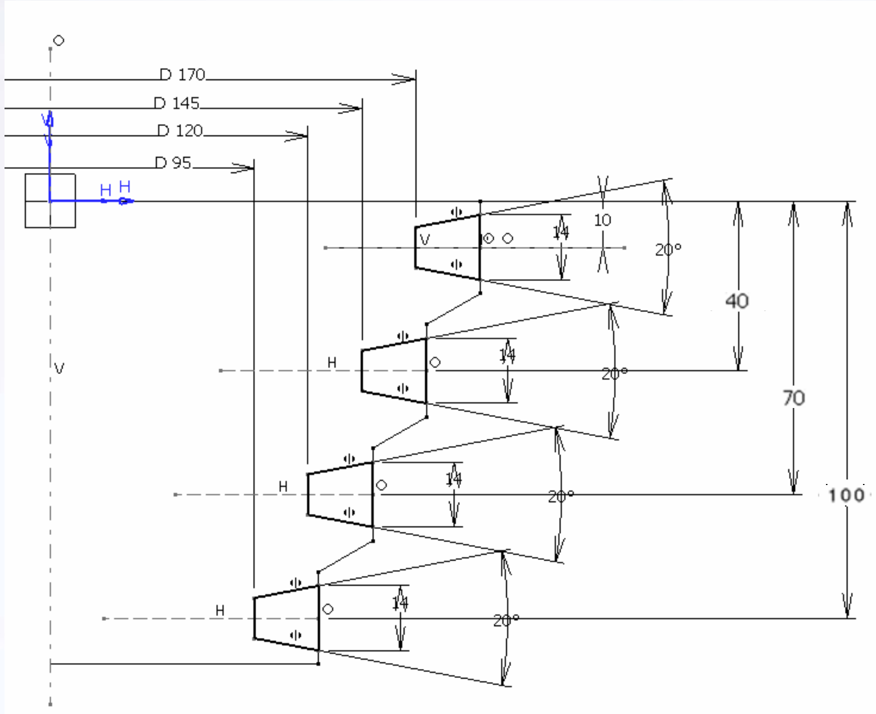


Do it Yourself (2/3)

3. Create a groove feature.

- Use the dimensions shown to create a groove feature. Remember to use the transformation tools while creating several identical profiles within one sketch. All profiles have the same internal dimensions.

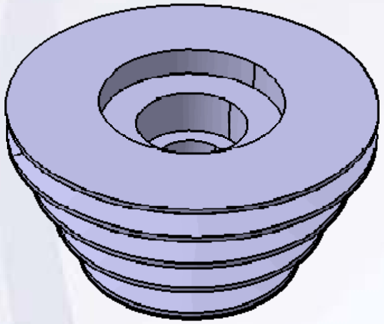


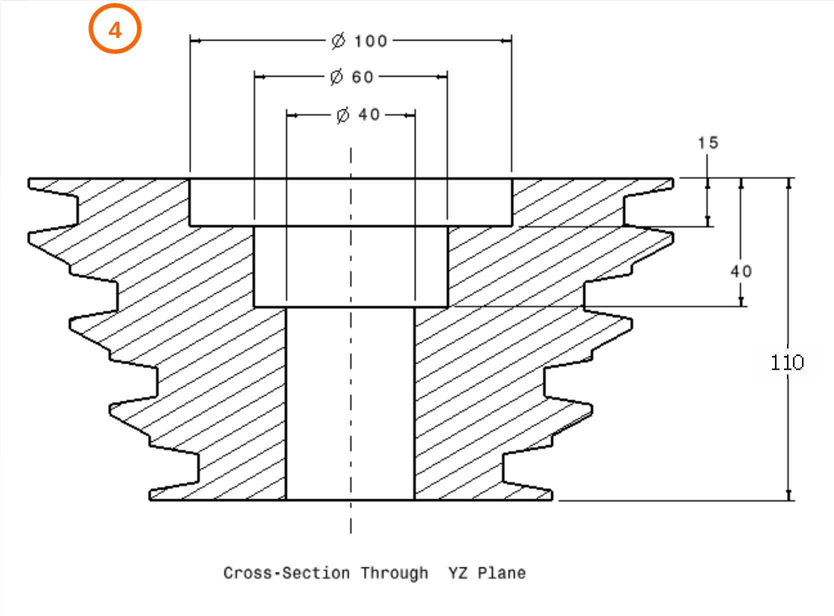


Do it Yourself (3/3)

4. **Create a multi-pocket feature.**
   - Use the dimensions shown to create a multi-pocket feature.

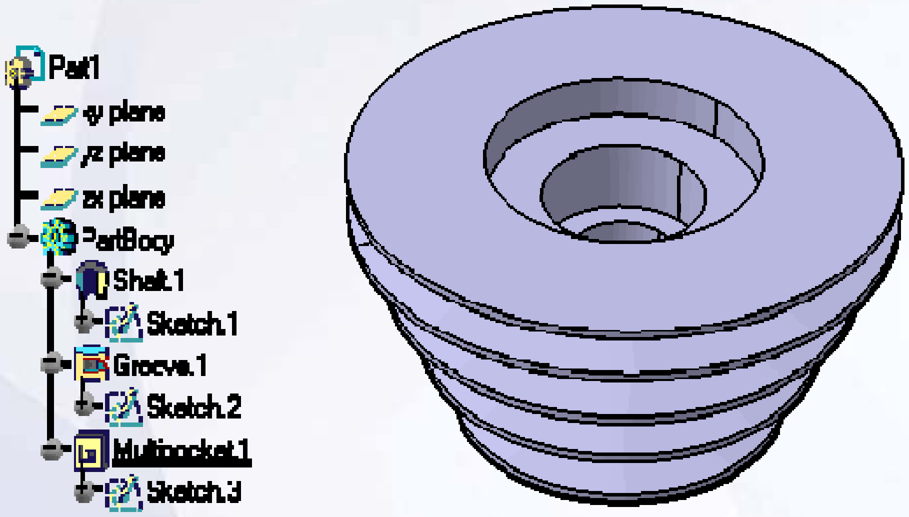
5. **Save and close the model.**





Exercise Recap: Shaft and Groove

- ✓ Create a shaft feature
- ✓ Create a groove feature
- ✓ Create a multi-pocket feature



Exercise: Shaft and Groove

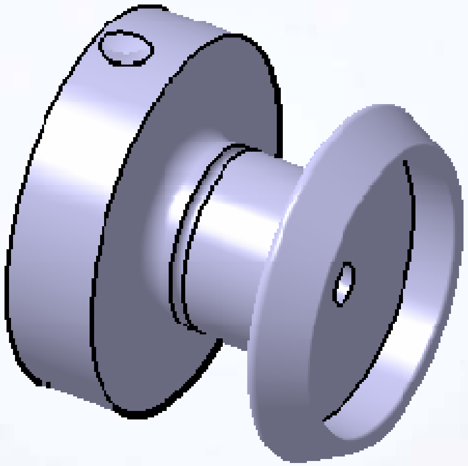
Recap Exercise

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.

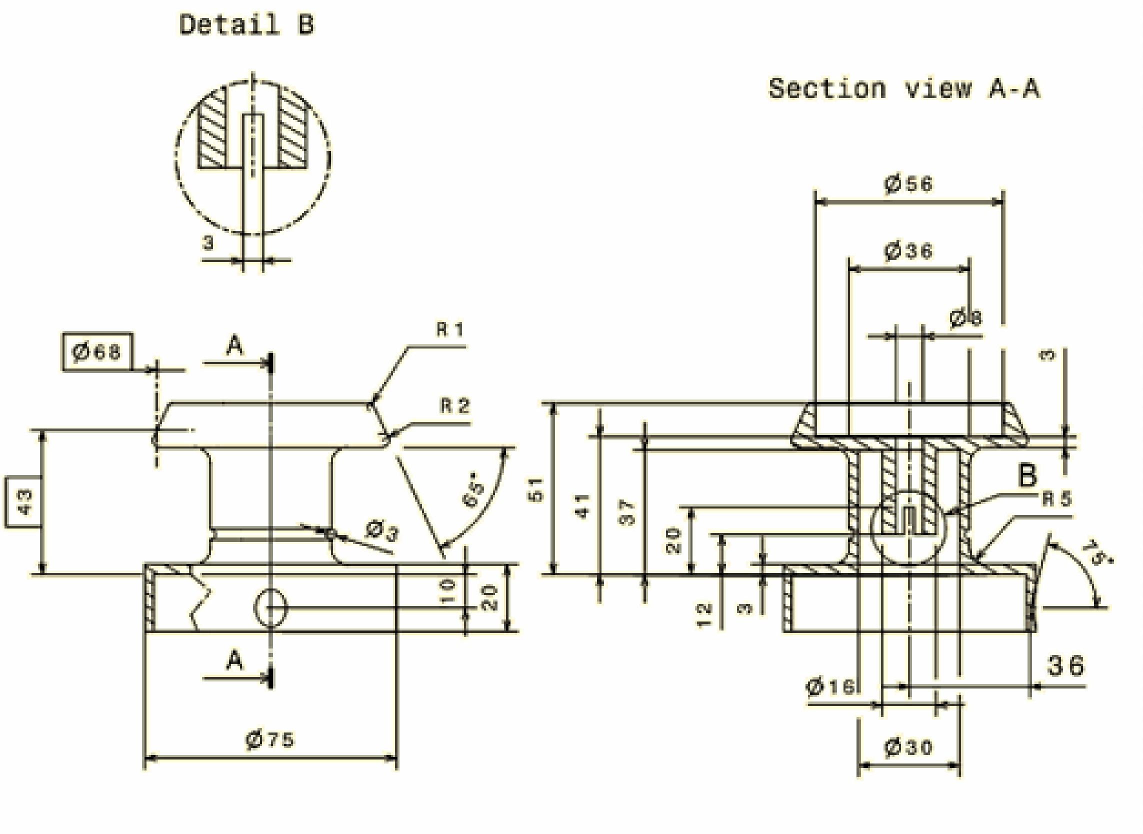
By the end of this exercise you will be able to:

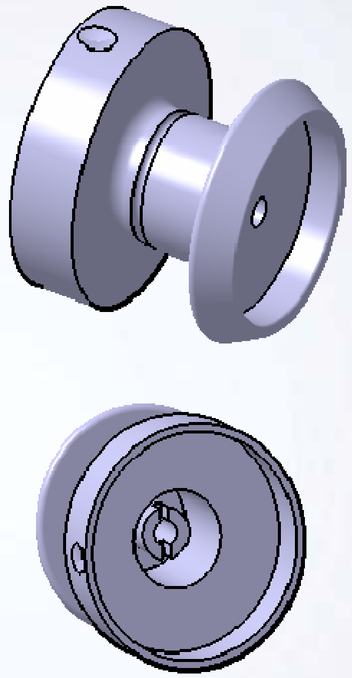
- **Create a shaft feature**
- **Create edge fillets**
- **Create internal and external groove features**
- **Create a pocket feature**
- **Create a reference point and line**
- **Create a cone-shaped groove feature**



Do it Yourself

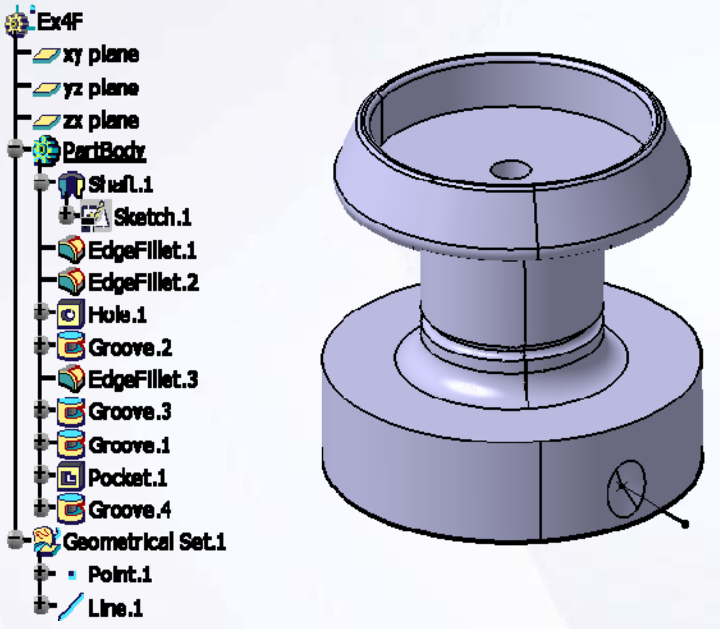
1. **Create the following spool part.**





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: Pad, Fillet, Hole and Shell

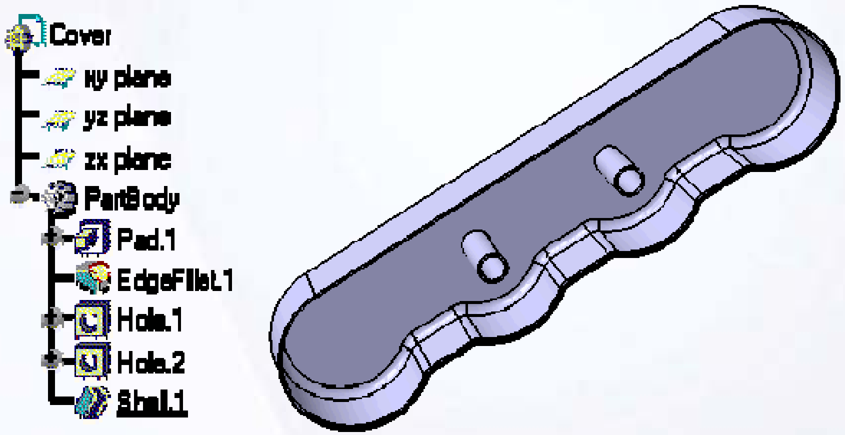
Recap Exercise

20 min

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

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

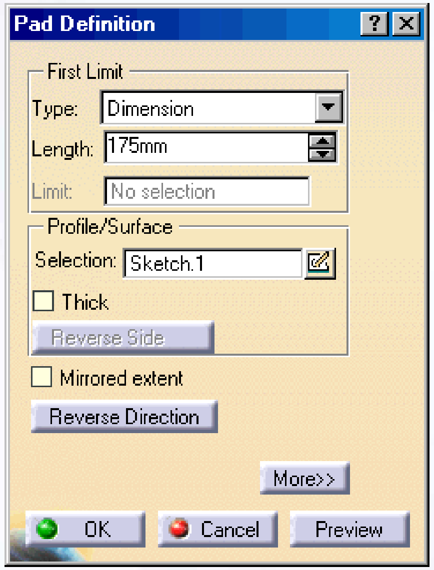
- Create a pad
- Create an edge fillet
- Create holes
- Create a shell feature

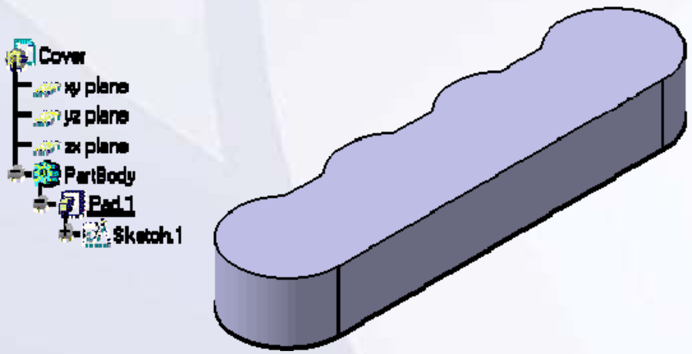


Do it Yourself (1/4)

1. **Open up the part Ex4H.CATPart.**
   - Open an existing part file using the Open tool. The part file contains a sketch.

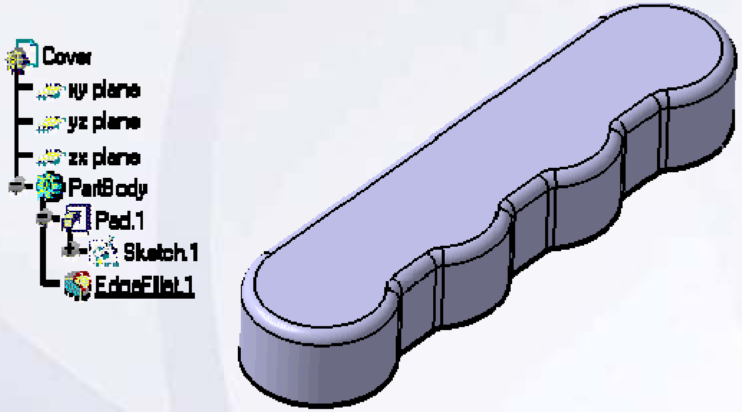
2. **Create a pad.**
   - Create a pad with a length of [175mm] using the existing sketch.

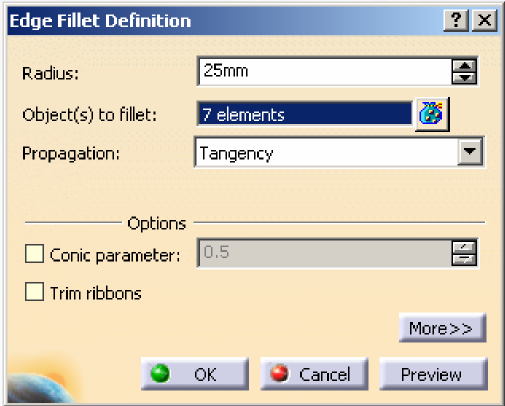




Do it Yourself (2/4)

3. **Create edge fillets.**
   - Create an edge fillet feature. Select the top surface and all the six vertical edges.

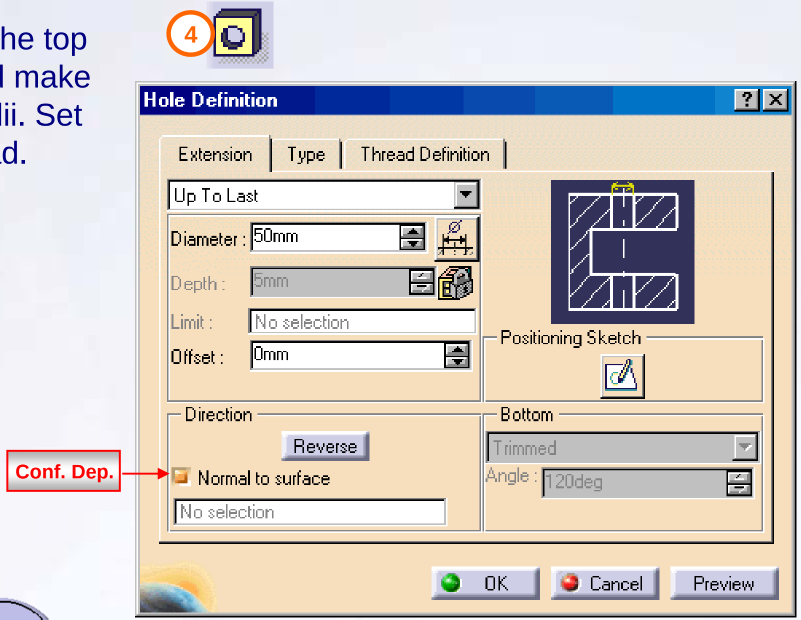


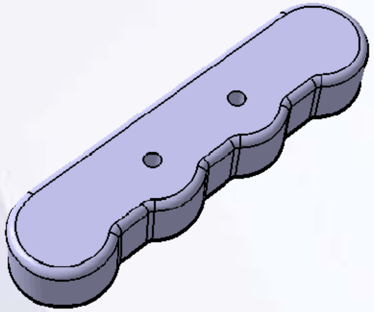


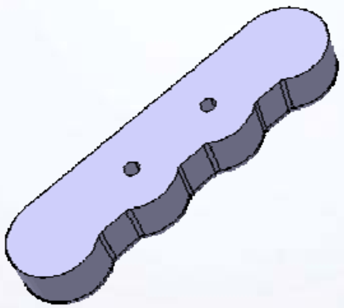
Do it Yourself (3/4)

4. Create holes.

- Create [50mm] diameter holes. Use the top of the pad as the starting surface and make them concentric to the two center radii. Set the depth to cut through the entire pad.

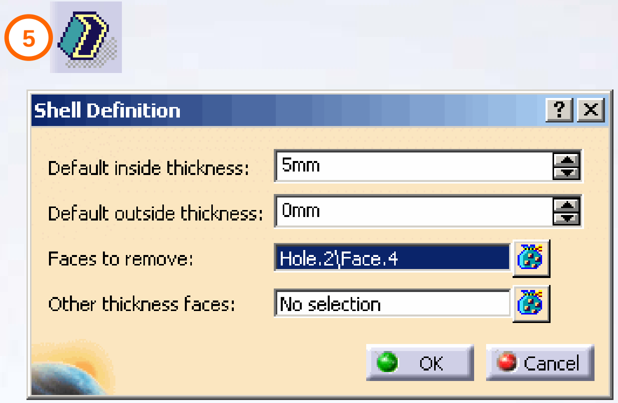


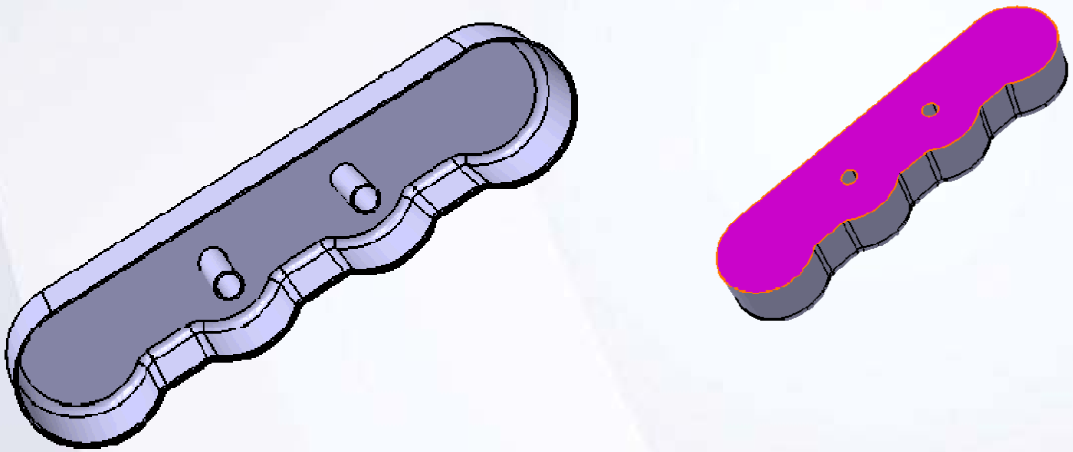




Do it Yourself (4/4)

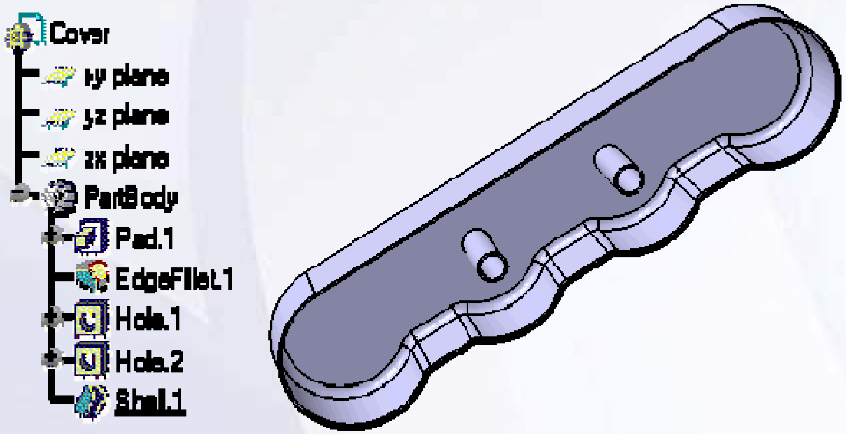
5. **Create a shell feature.**
   - Create a Shell and select the bottom surface to be removed and specify a thickness of [5mm] for the inside thickness.





Exercise Recap: Pad, Fillet, Hole and Shell

- ✓ Create a pad
- ✓ Create a fillet
- ✓ Create holes
- ✓ Create a shell feature



Exercise: Thin Pad, Shell and Holes

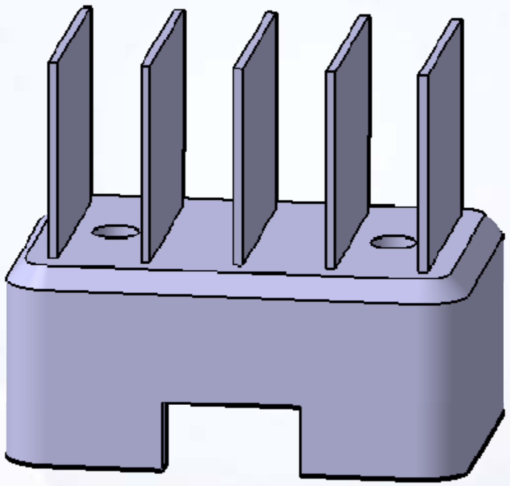
Recap Exercise

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 learned in this lesson to complete the exercise with no detailed instructions.

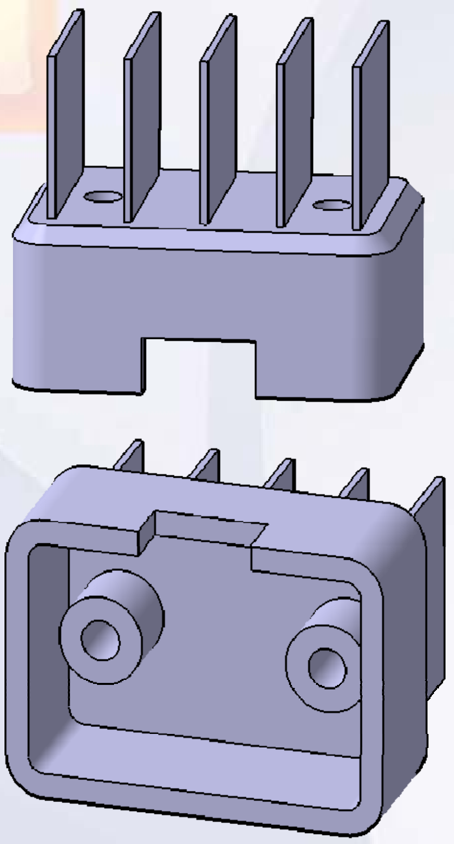
By the end of this exercise you will be able to:

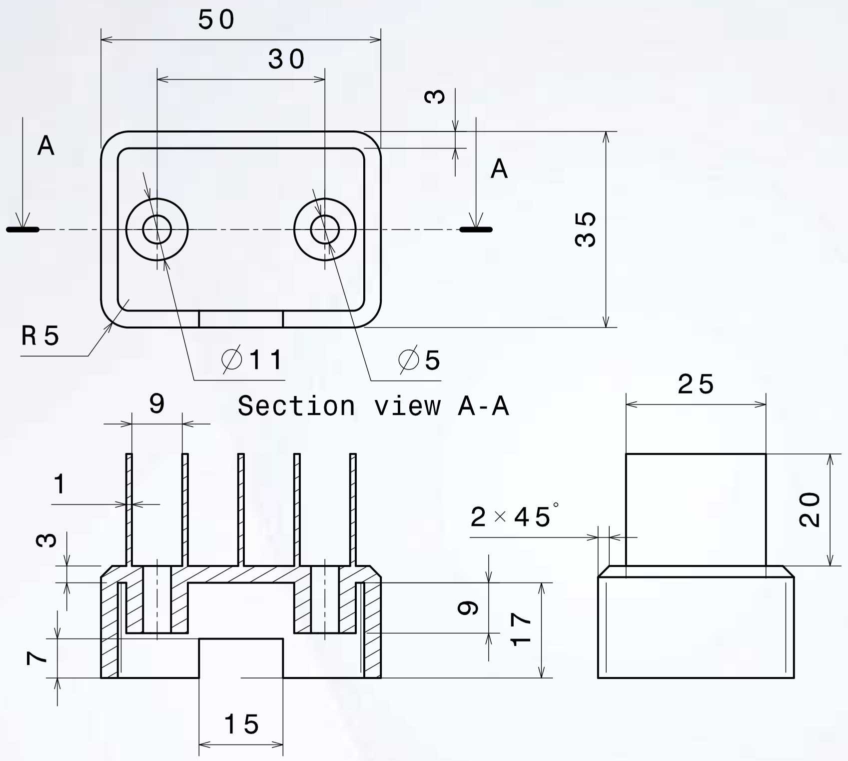
- **Create pads**
- **Create a shell**
- **Create a thick pad**
- **Create holes**
- **Create a chamfer**



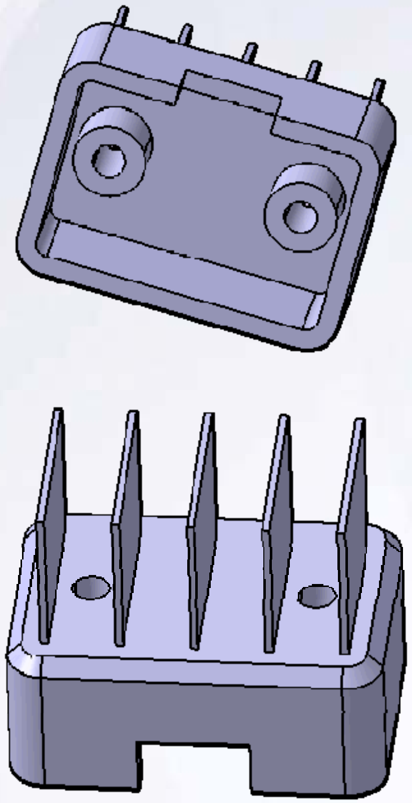
Do it Yourself

Create the following part.





Exercise Recap: Thin Pad, Shell and Holes



- ✓ Create pads
- ✓ Create a shell
- ✓ Create a thick pad
- ✓ Create holes
- ✓ Create a chamfer

Case Study: Additional Part Features

Recap Exercise

20 min

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

- ✓ The top portion and bottom portions of the model must be created as separate features.
- ✓ The holes must created at an angle to the XY plane.
- ✓ The model must be hollow.
- ✓ The holes must be drilled normal to the sides of the handle.

Using the techniques discussed so far, create the model without detailed instructions.

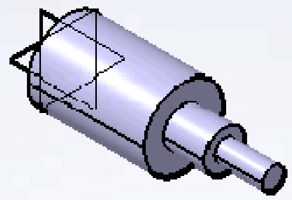
Do It Yourself: Drawing of the Handle Block (1/4)

You will need to create the following features:

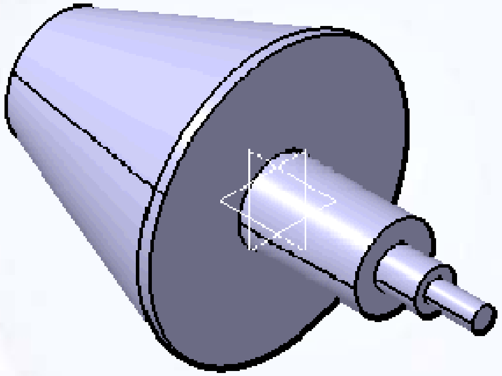
1. **Create a multi-pad feature.**
   - Create the bottom section of the model with the **Multi-pad** tool.

2. **Create a revolve feature.**
   - Create the top section of the model using a revolve feature.

1



2



Do It Yourself: Drawing of the Handle Block (2/4)

You will need to create the following features (continued):

3. Create a groove.

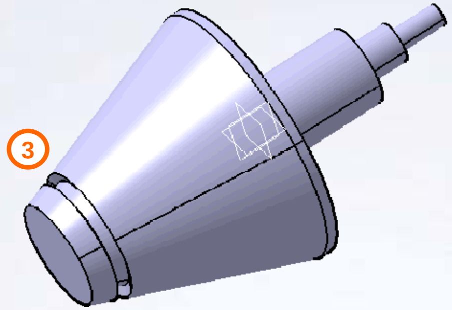
- Create a cut using the **Groove** tool. Use the Project 3D tools to associate the cut to the revolve feature.

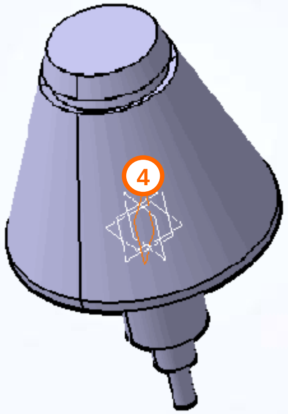
4. Create a plane.

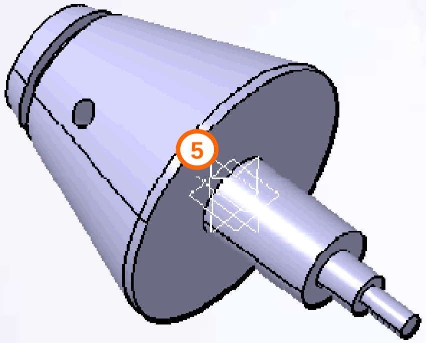
- Create a plane [45] degrees from the XY plane.

5. Create holes.

- Create holes that are coincident with the user-defined plane.



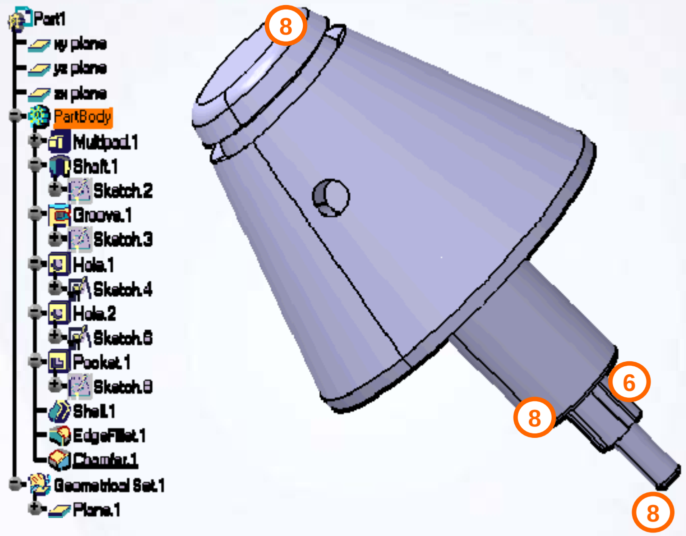




Do It Yourself: Drawing of the Handle Block (3/4)

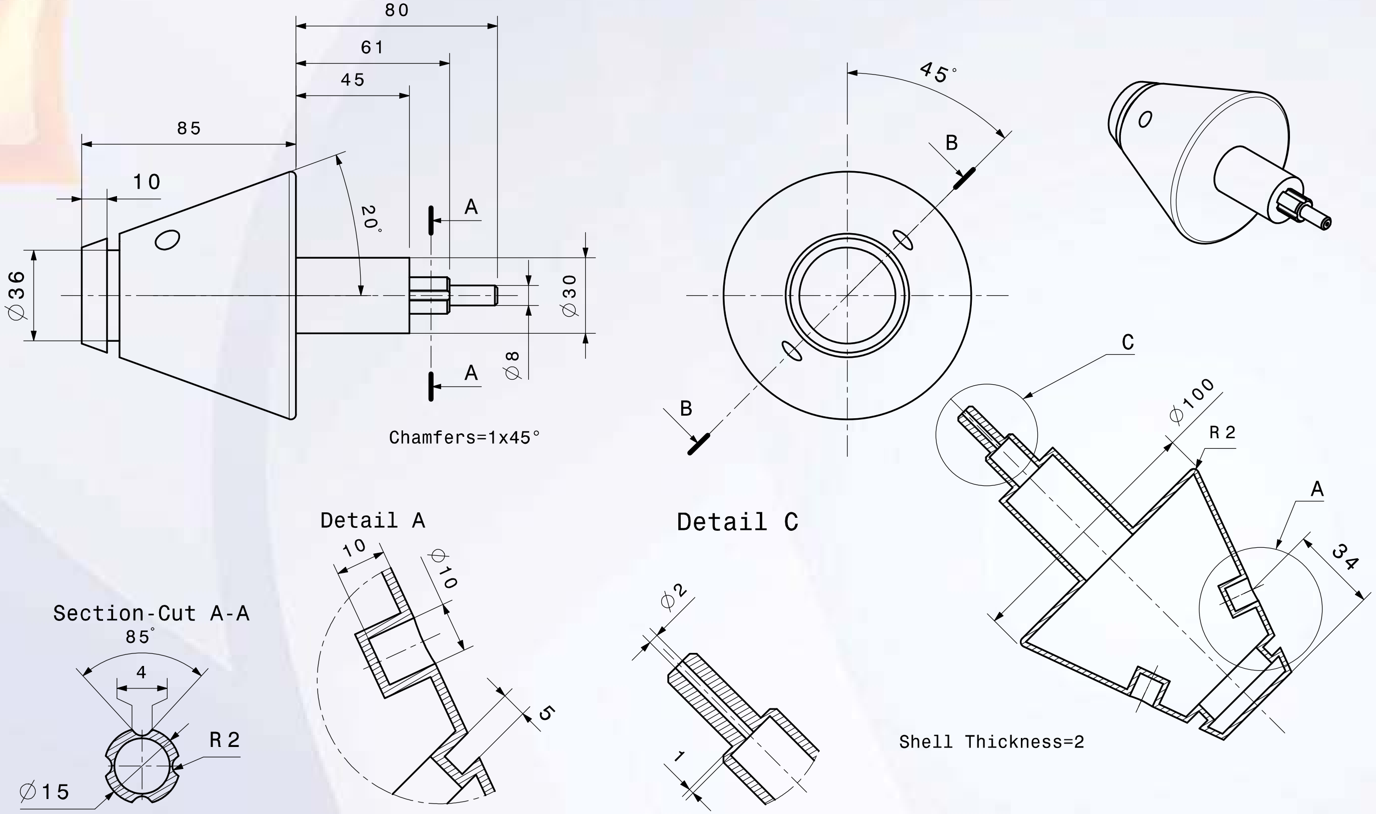
You will need to create the following features (continued):

6. **Create a pocket.**
   - Create the sketch for the pocket by creating one profile, use the **Rotate** tool to create the remaining three profiles.
7. **Shell the model.**
   - Shell the model to a thickness of 2mm, except at the bottom where the thickness should be different (see the drawing).
8. **Create dress-up features.**
   - Complete the model by adding a 5mm fillet to the top edge and a 1mm x 45 degree chamfer to the two edges shown.



Do It Yourself: Drawing of the Handle Block (4/4)

Use the dimensions shown to complete the Handle Block part.



Case Study: Handle Block Recap

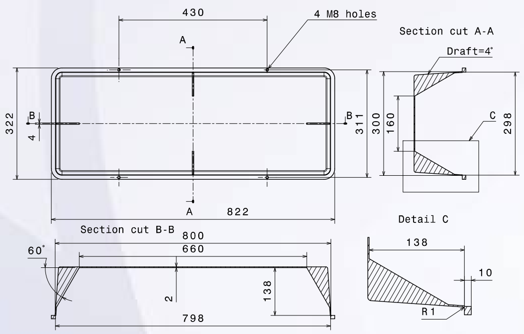
- ✓ Create a multi-pad feature
- ✓ Create a shaft feature
- ✓ Create a groove feature
- ✓ Create reference geometry
- ✓ Create holes
- ✓ Create a pocket
- ✓ Shell the model
- ✓ Create dress-up features

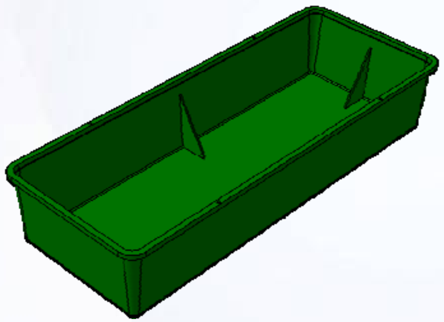
Dress-Up Features 5

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Apply a draft
- ✓ Create a stiffener
- ✓ Create threads and taps
- ✓ Edit features





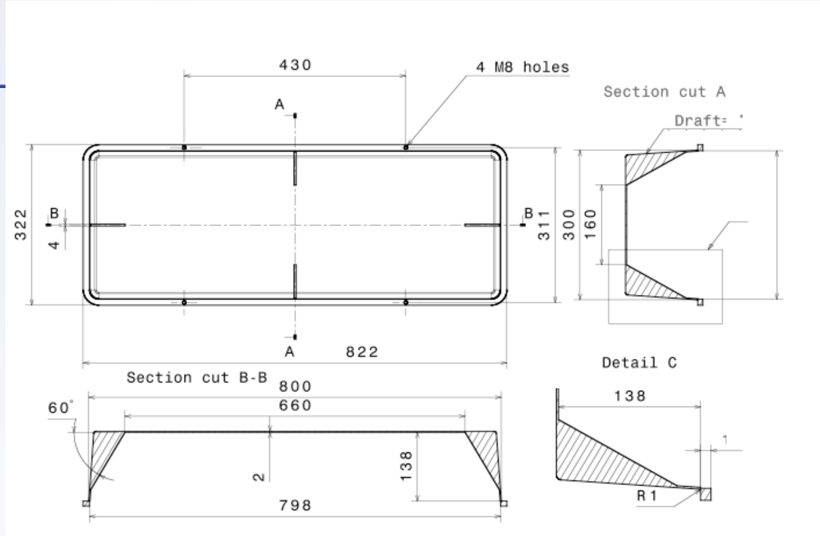
4 Hours

Case Study

The case study for this lesson is the Casing used in the Drill Press assembly.

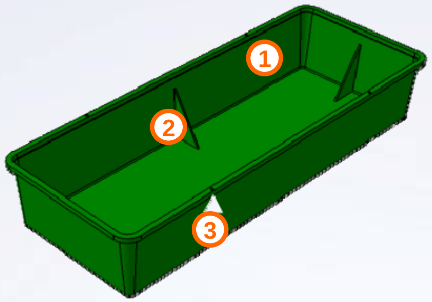
Design Intent

- ✓ The inner ribs of the Casing should be created using Stiffener features.
  - Stiffener features provide the most efficient method of creating this geometry.
- ✓ The Casing should have a 4 degree draft.
  - This part would most likely be manufactured using the Molding process, which requires drafts.
- ✓ The Casing should have taps defined for all the holes.
  - Taps can be represented in a simple manner without having to create the complex geometry, which can be time-consuming and resource-intensive during regeneration cycles.



Stages in the Process

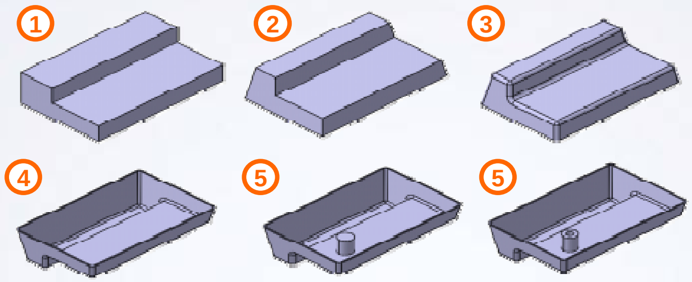
1. Apply a draft.
2. Create a stiffener.
3. Create threads and taps.
4. Edit features.



Apply a Draft

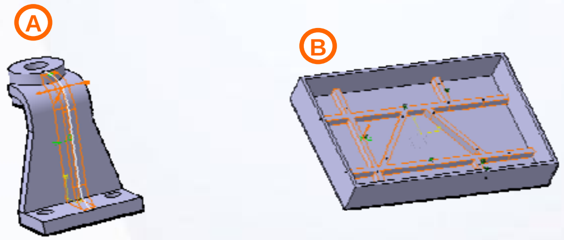
Draft features are used to apply an angle to a part surface relative to some reference. Material is added or removed depending on the draft angle and the pull direction applied during the operation.

- ✓ 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.



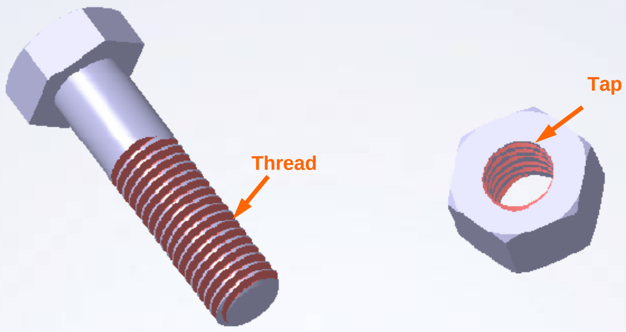
A. From Side
The sketch is extruded in the profile plane and thickened normal to it.

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

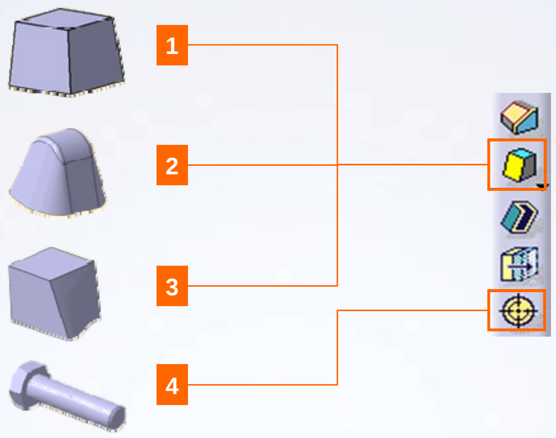
Dress-Up Features

1 **Draft Angle:** Creates a basic draft.

2 **Draft Reflect Line:** Creates drafts on non-planar surfaces, such as a cylinder.

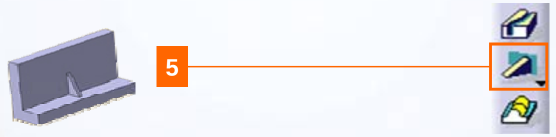
3 **Variable Angle Draft:** Creates a draft that has different angles at transition edges.

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: Reflect Draft

Conf. Dep.

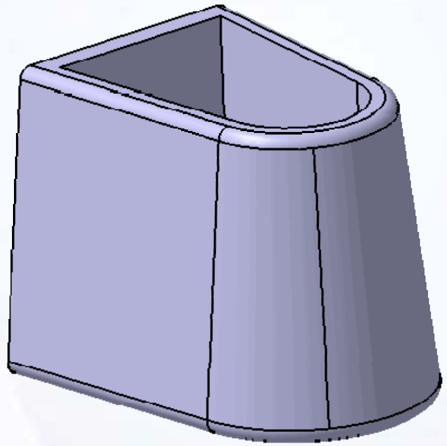
Recap Exercise

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)

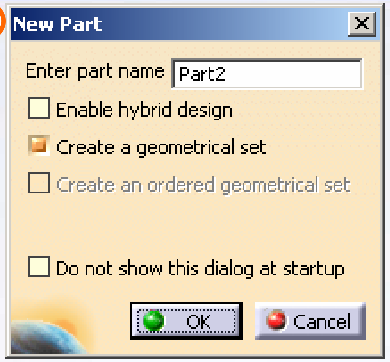
1. Create a new part.

- Create a new part with the geometrical set.

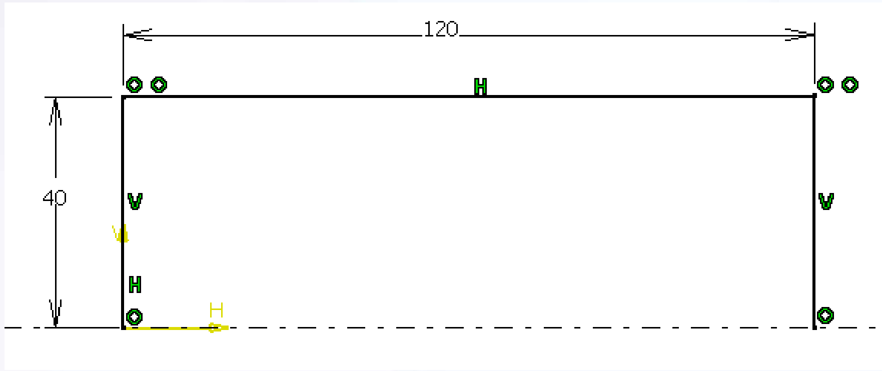
2. Create a shaft.

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

1

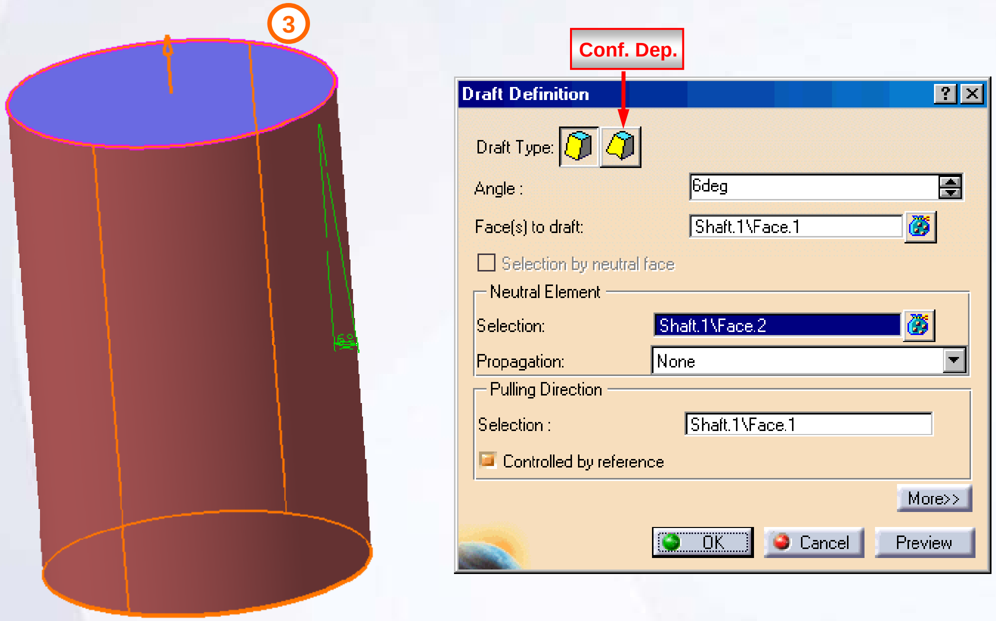


2



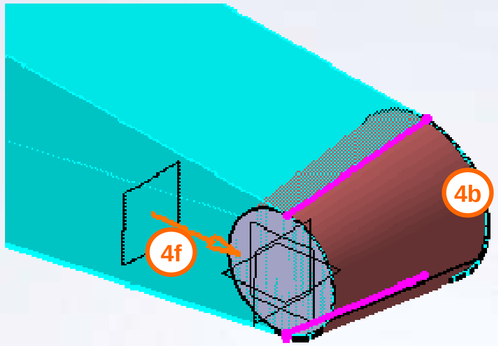
Do it Yourself (2/5)

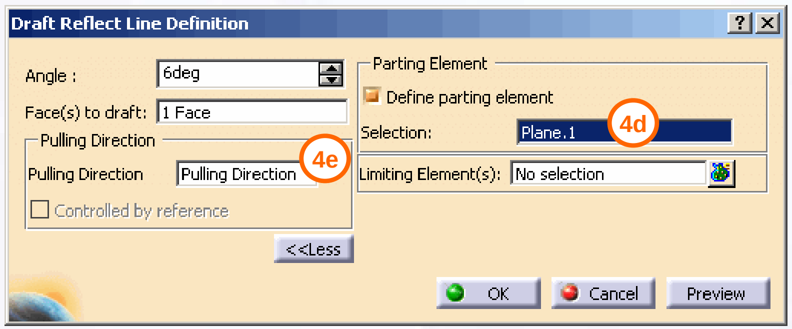
3. **Create a basic draft.**
    a. Select the walls of the cylinder as the faces to draft and the top surface as the neutral and pulling direction.
    b. Specify a [6deg] draft angle.



Do it Yourself (3/5) Conf. Dep.

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** on 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 that the pull direction is correct.



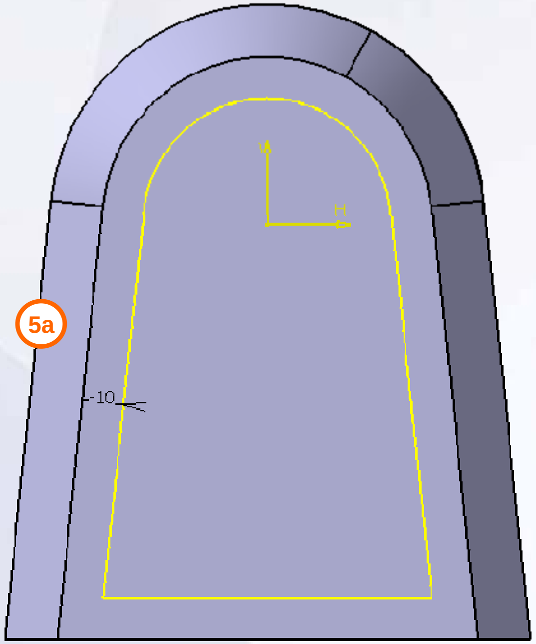


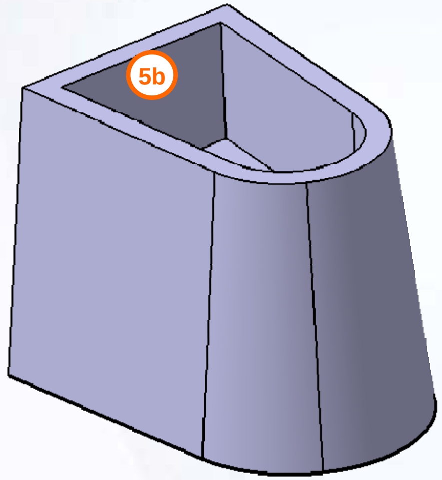
Do it Yourself (4/5)

5. Create a pocket.

a. Select the top surface of the pad and sketch the following profile. Use the existing face of the pad to create a [10mm] offset.

b. Create a pocket that is [50mm] deep.





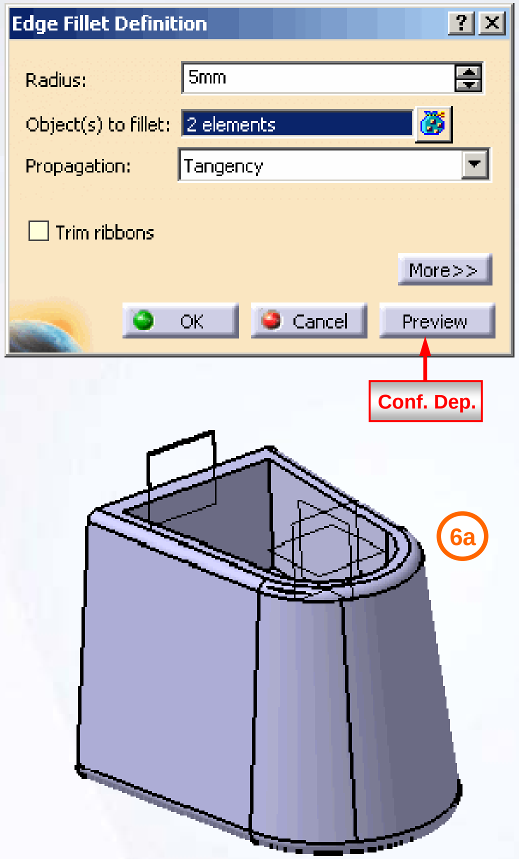
Do it Yourself (5/5)

6. **Create an edge fillet.**

   a. Select the edges around the top and bottom profiles and specify a [5mm] radius value.

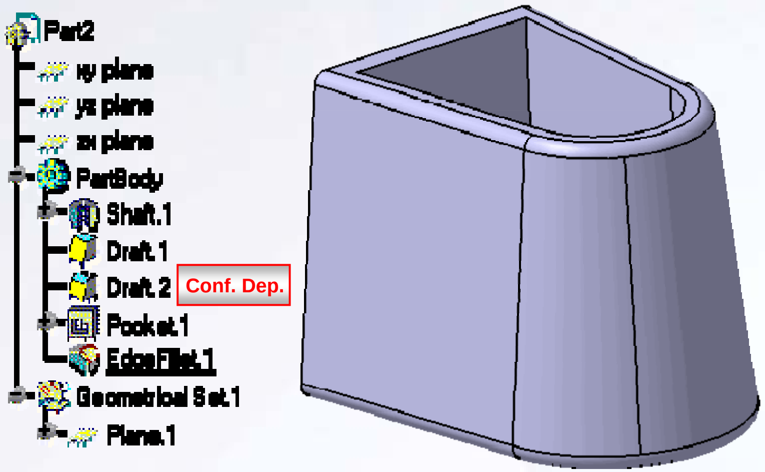
7. **Hide all the references plane.**

8. **Save and close the file.**



Exercise Recap: Reflect Draft

- ✓ Create a basic draft
- ✓ Create a reflect draft Conf. Dep.



Exercise: Stiffeners and Draft

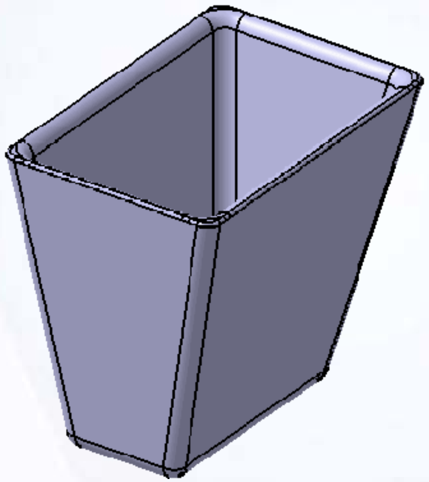
Recap Exercise

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.

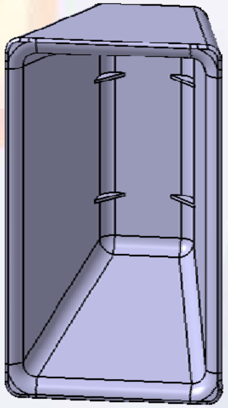
By the end of this exercise you will be able to:

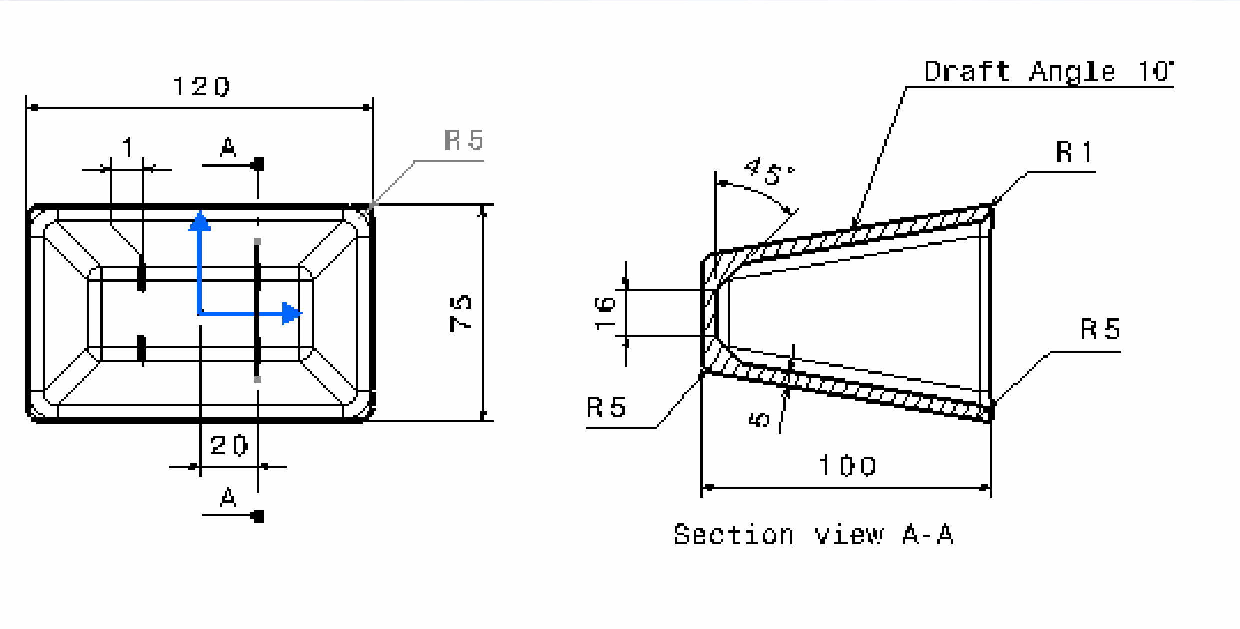
- **Create a new part**
- **Apply draft to a part**
- **Create stiffeners**



Do it Yourself

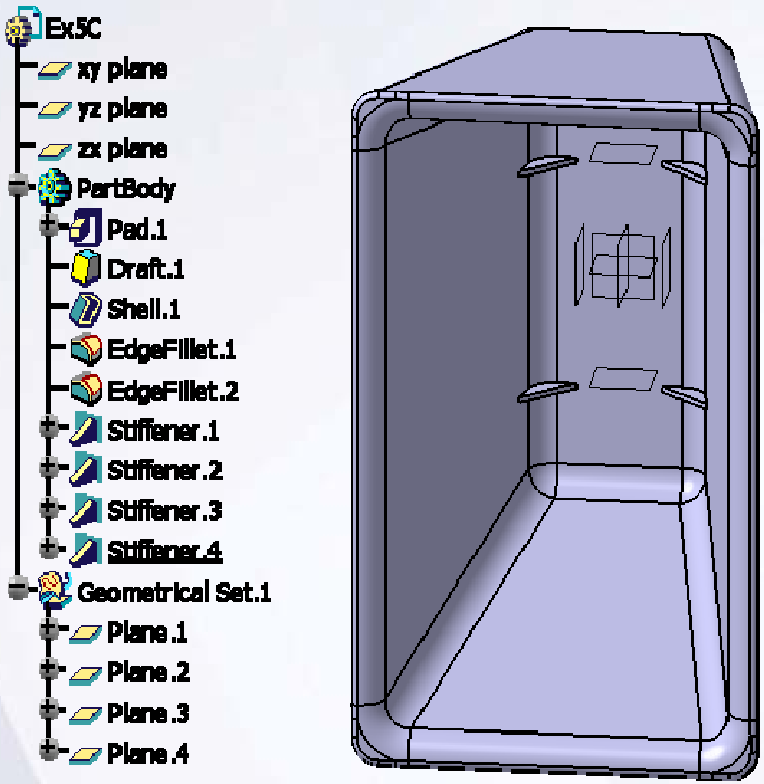
1. **Create the part shown below.**





Exercise Recap: Stiffeners and Draft

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



Exercise: Thread

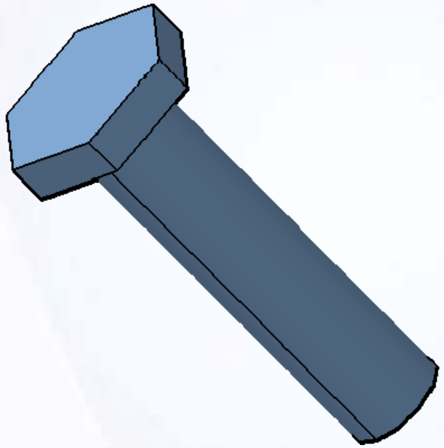
Recap Exercise

10 min

In this exercise you will create a bolt and complete the exercise with the techniques and tools you have already learned, without any detailed instructions.

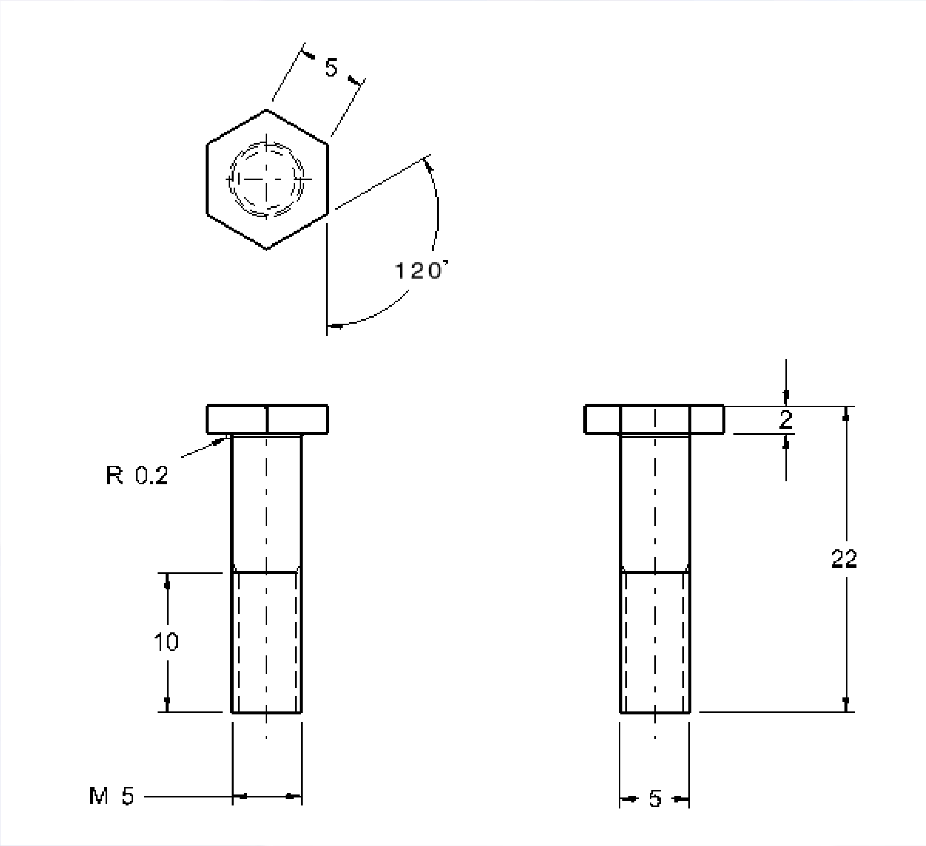
By the end of this exercise you will be able to:

- **Create threads.**



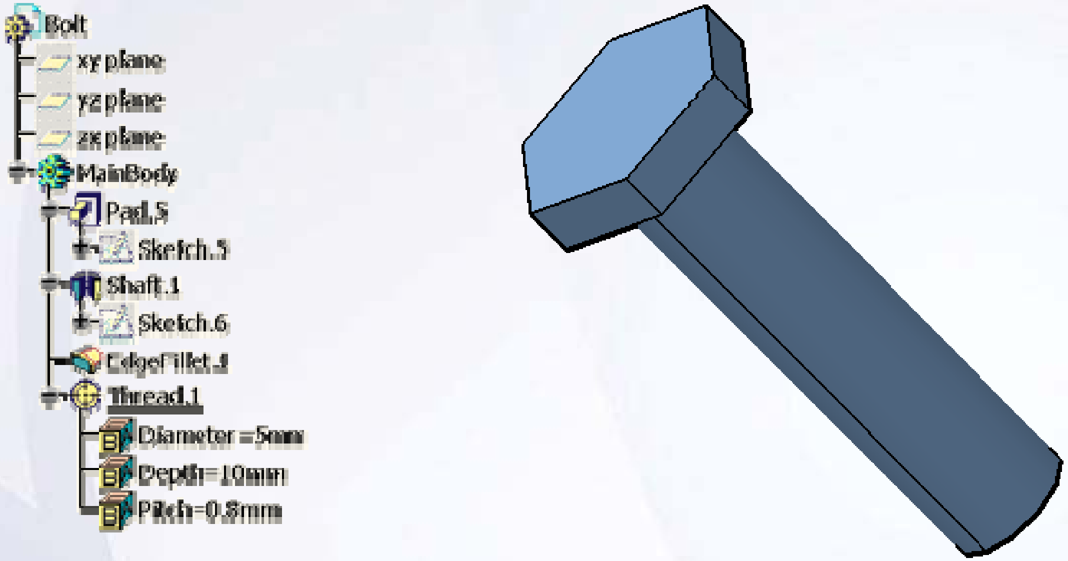
Do it Yourself

1. Create the bolt part with the dimensions given below.



Exercise Recap: Thread

✓ Create threads



Exercise: Features Activation

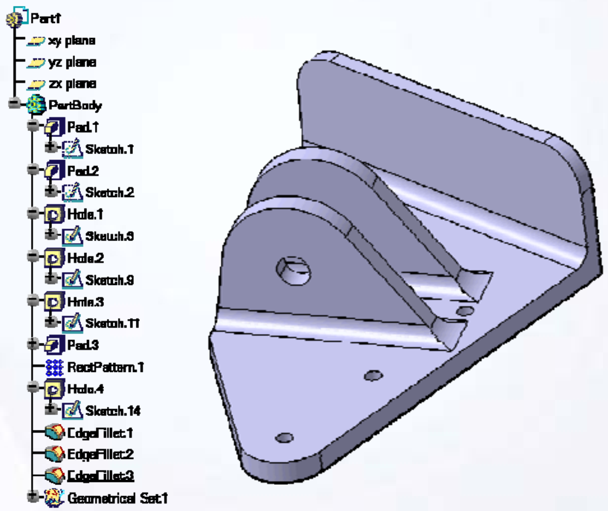
Recap Exercise

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**

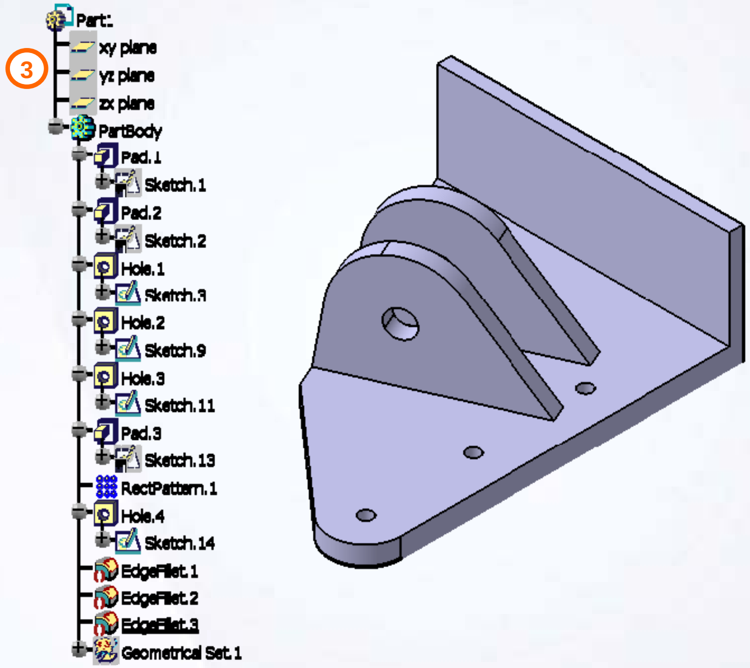


Do it Yourself (1/2)

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.

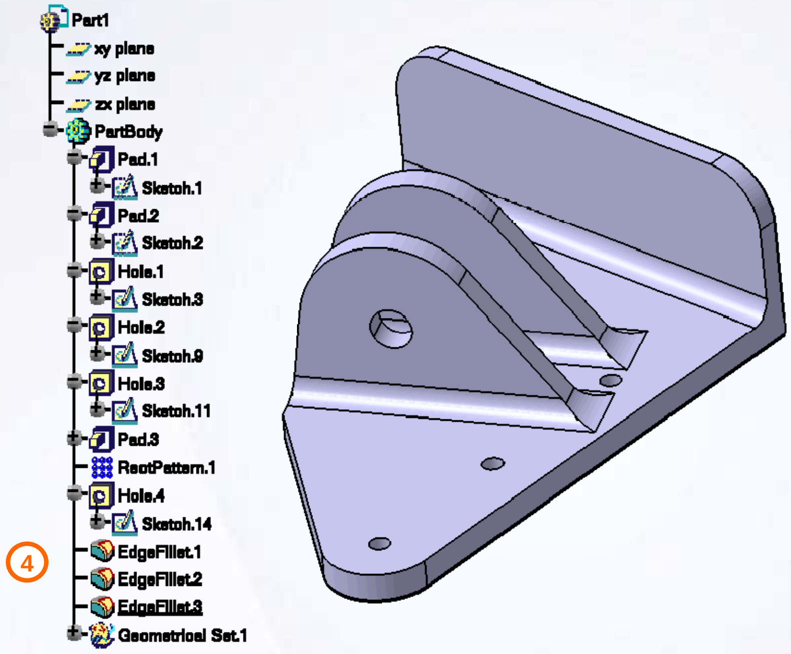


Do it Yourself (2/2)

4. Activate the edge fillets.

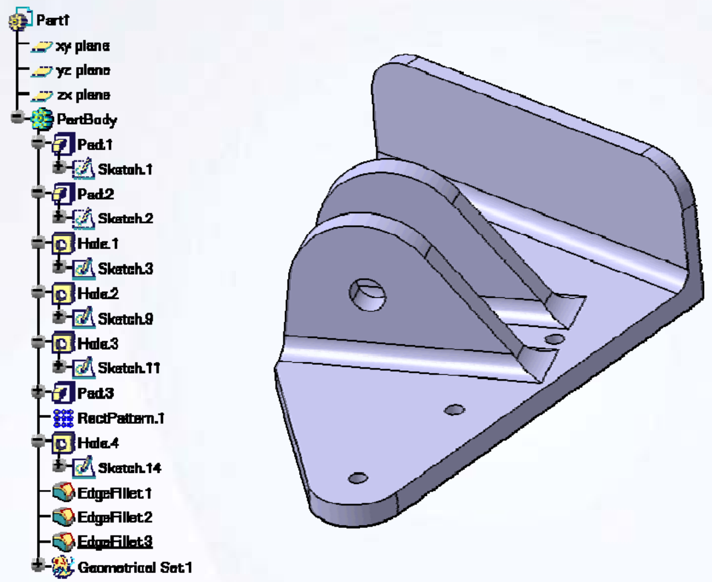
- The last three features in the specification tree have been deactivated. Activate these features.

5. Close the model.



Exercise Recap: Features Activation

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



Exercise: Update Error Management

Recap Exercise

20 min

In this exercise, you will open an existing part file, 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.

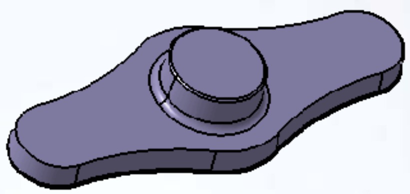
Do it Yourself (1/3)

In this exercise, you will open an existing part file, update the part and resolve any feature failures.

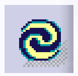
1. **Open Ex5e_error.CATPart.**
   - Open an existing part file using the **Open** tool and investigate the features in the specification tree.

2. **Update the model.**
   - The icons in the specification tree specify that model needs to be updated .

1

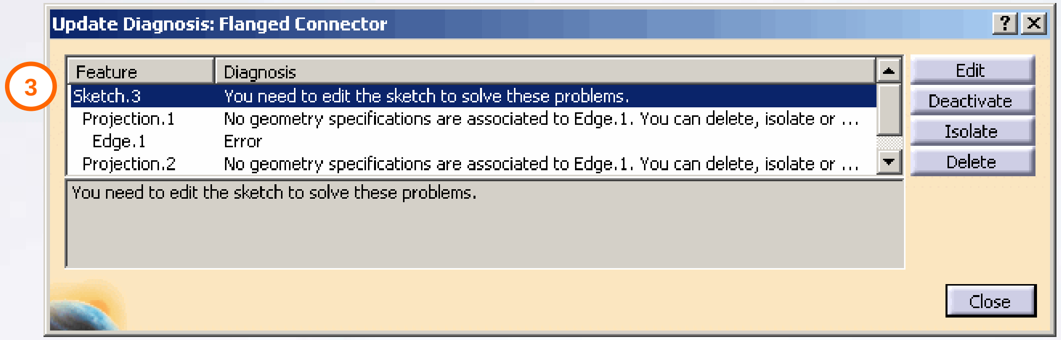


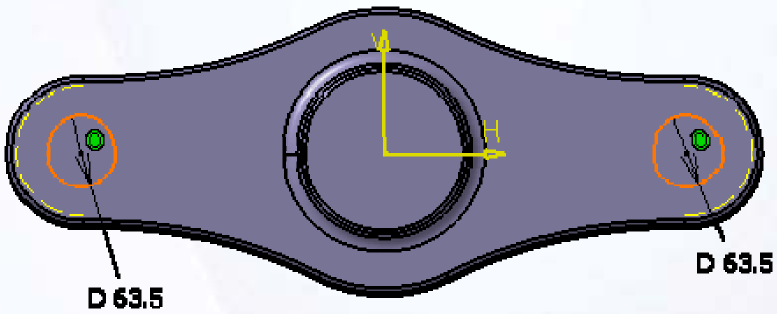
2



Do it Yourself (2/3)

3. **Resolve feature failures.**
   - Once CATIA tries to regenerate Pad.3, sketch.3 fails. CATIA prompts you to edit the sketch. Review the sketch and make a note of the missing references. Delete them and exit the sketcher workbench.

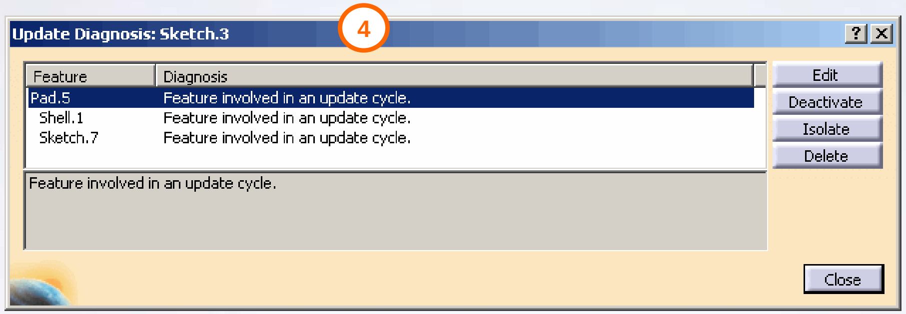


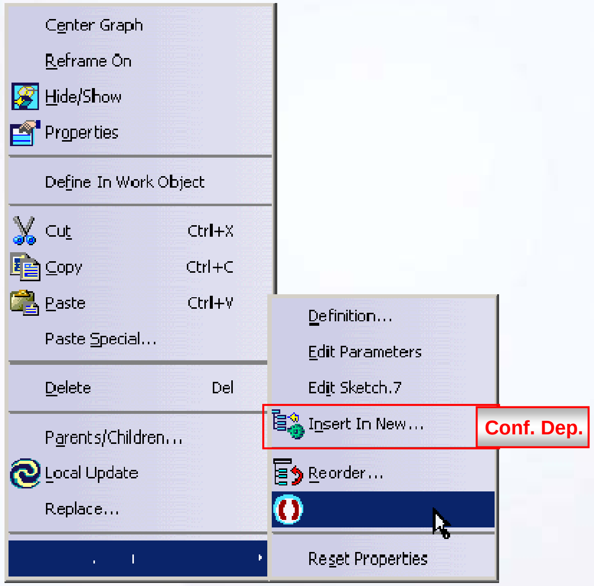


Do it Yourself (3/3)

4. **Resolve feature failures (continued).**
   - The second feature failure occurs because of an update cycle error between features Pad.5 and Shell.1. After reviewing the features, Shell.1 needs to be reordered to occur before Pad.5.
     a. Deactivate Pad.5 and then reorder Shell.1 to occur before it.
     b. Select **Deactivate** in the **Update Diagnosis** dialog box. Hole.2 and Hole.3 also need to be deactivated since they are children of Pad.5.
     c. Reorder Pad.5 to appear after Shell.1, since the Sketch for Pad.5 needs a face from Shell.1 feature.
     d. Activate all the three features that were deactivated.

5. **Save and close the file.**





Exercise Recap: Update Error Management

✓ Troubleshoot a part that contains features that fail.

Case Study: Dress-up Features

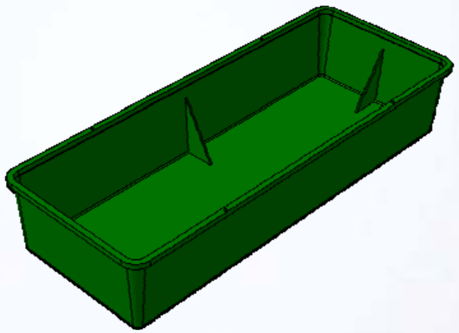
Recap Exercise

25 min

You will practice what you learned by completing the case study model using only a detailed drawing and hints as guidance.

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

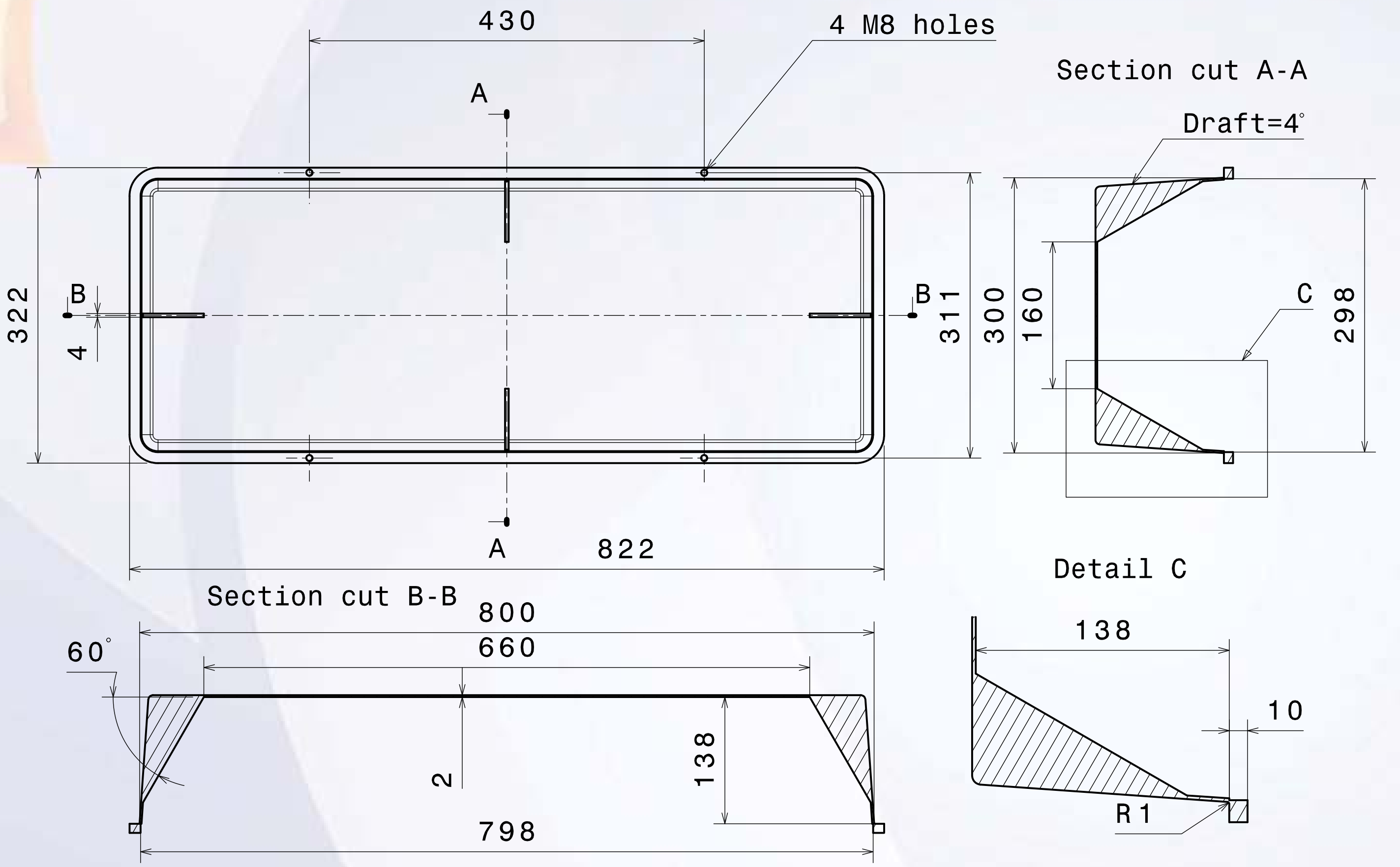
- ✓ The inner ribs should be created using stiffener features.
- ✓ The casing should contain a four degree draft.
- ✓ The casing should have taps defined for any holes.



Using the techniques discussed in this and the previous lessons, create the model without detailed instructions.

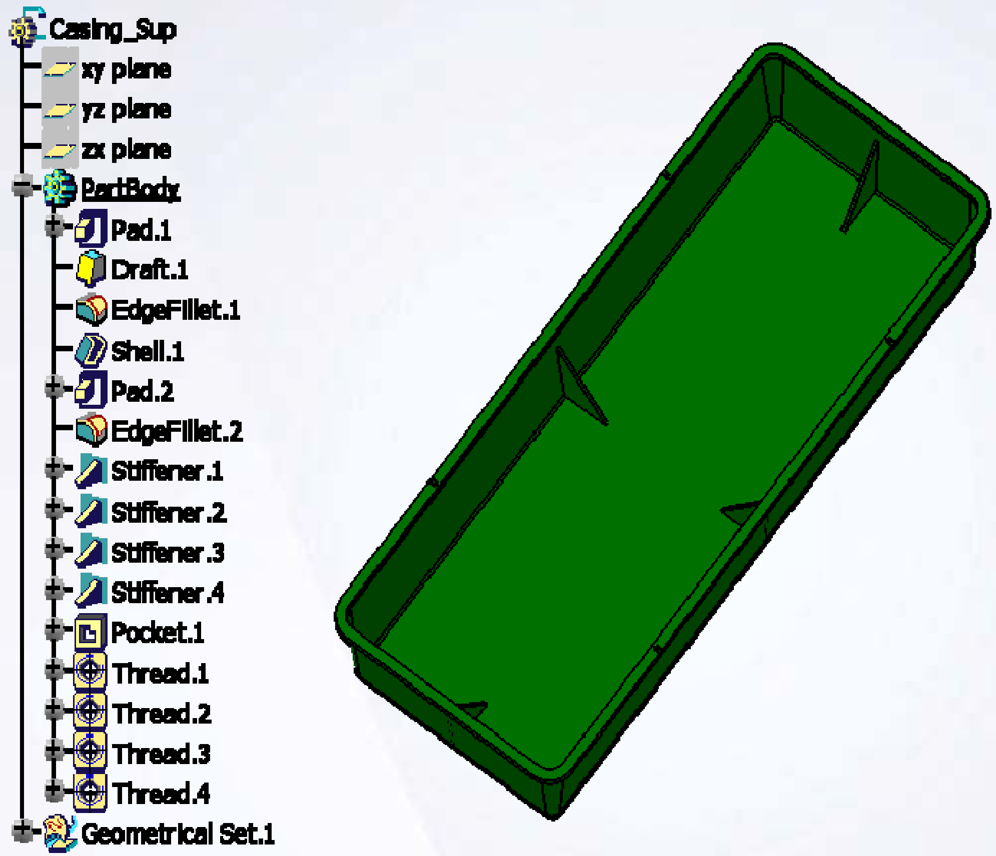
Do It Yourself: Drawing of the Casing

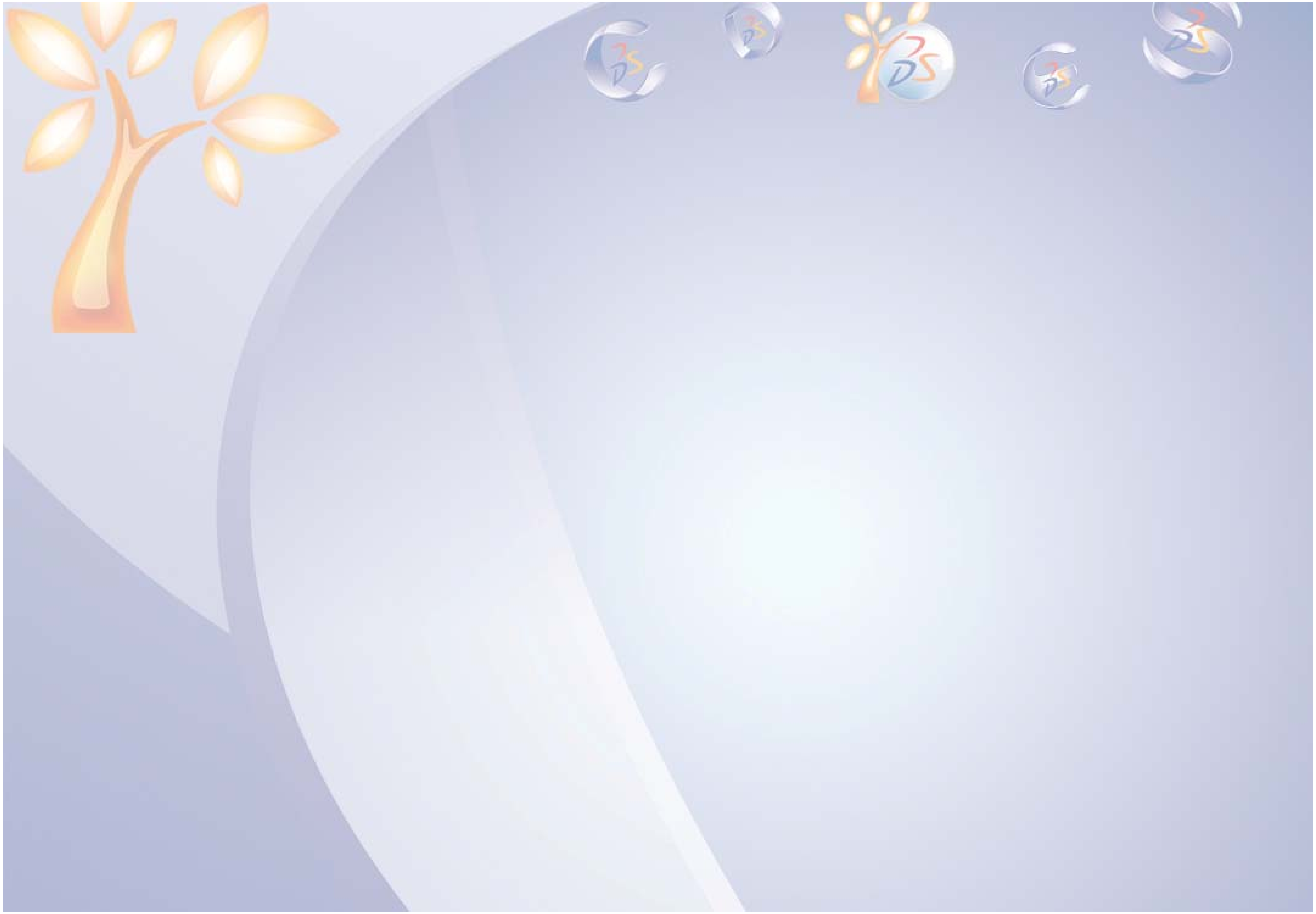
Create the model using the drawing provided here.



Case Study: Casing Recap

- ✓ The inner ribs should be created using stiffener features.
- ✓ The casing should contain a four degree draft.
- ✓ The casing should have taps defined for any holes.



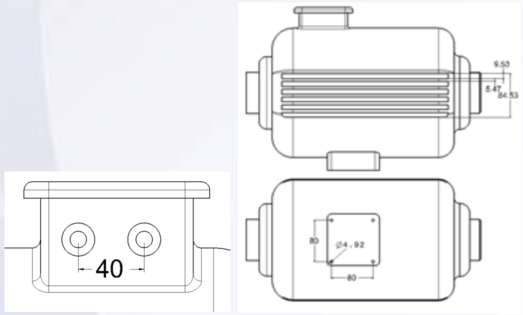


Reusing Data 6

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Duplicate Features
- ✓ Transform a Body
- ✓ Copy and Paste the Data
- ✓ Insert Data From a Catalog



4 Hours

Case Study

The case study for this lesson is the Engine used in the Drill Press assembly.

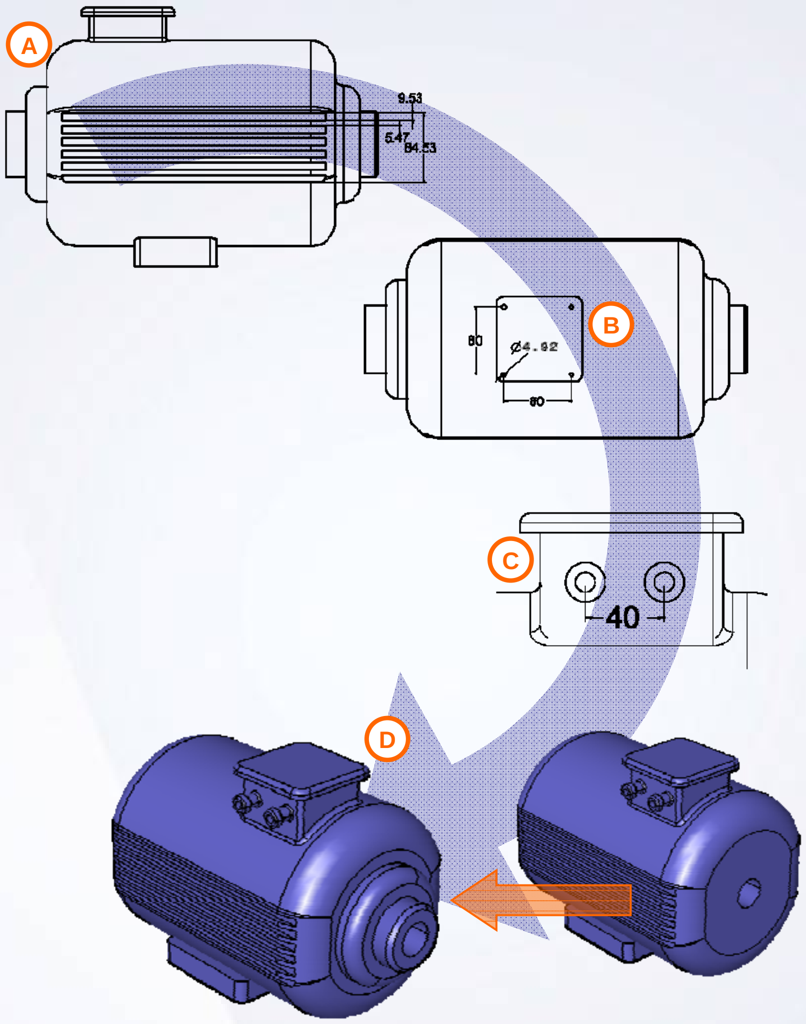
Design Intent

A. The side fins must be created using a rectangular pattern.
   - This avoids having to create and sketch each pocket individually.

B. The hole pattern must be created using a user-defined pattern.
   - This feature can also be created as a rectangular pattern; however, a user-defined pattern will enable you to customize the hole locations.

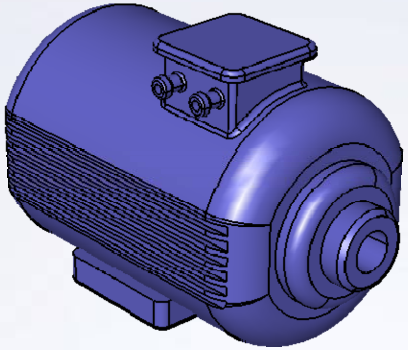
C. The model must be partially created by copy pasting the features.
   - This is one way of quickly recreating duplicate features.

D. The model must include features from a catalog.
   - This is an easy method to retrieve data from a source that is accessible to everyone.



Stages in the Process

1. Duplicate Features
2. Transform a Body
3. Copy and Paste the Data
4. Insert Data From a Catalog



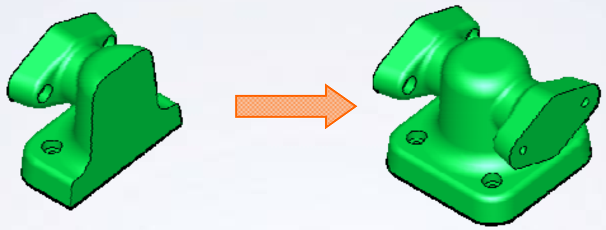
Duplicate Features

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.

A

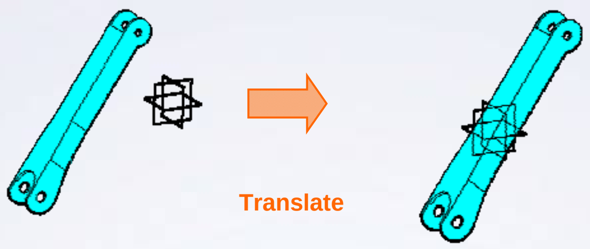


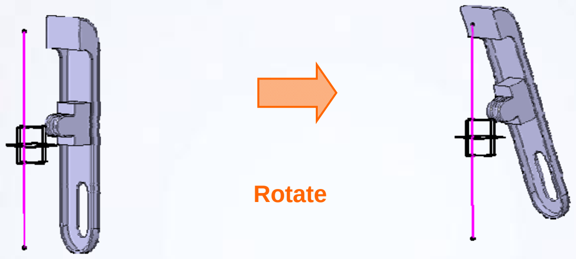
B

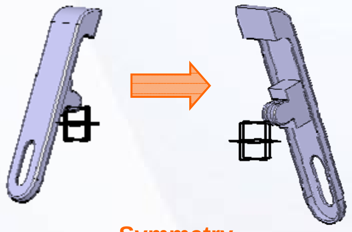
Transform a Body

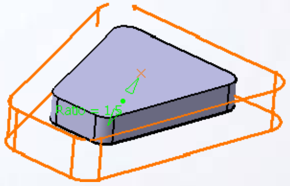
Transformations are used in a multi-body context. Transformations are required when you have some geometry that has been created in one location and which needs to be moved into a specific position. These transformations enable you to move a body by translating it along an axis, rotating it round an axis, or moving it symmetrically about a plane.

The Scaling option allows to shrink or expand an entire body based on a single reference. In the design of the model for an injection molded plastic part, the design part will often be scaled up, to account for material shrinkage.


Translate


Rotate

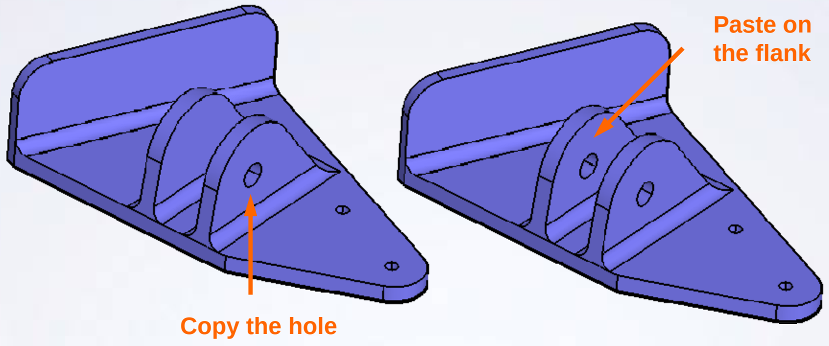

Symmetry


Scaling

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.



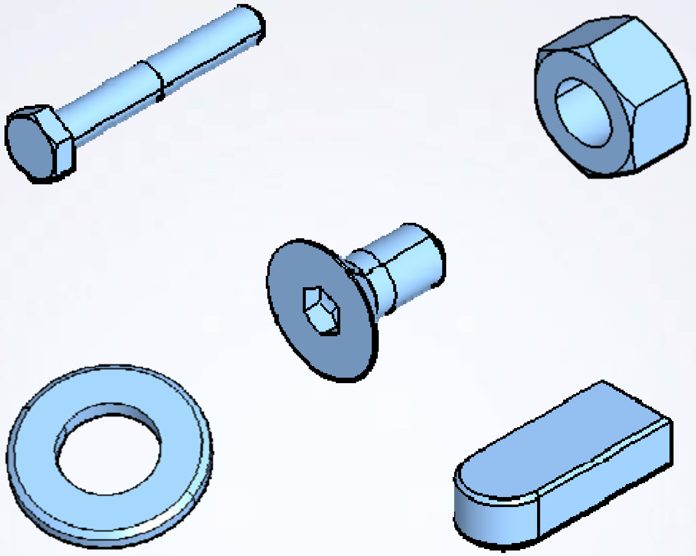
Insert Data From a Catalog

Catalogs are sets of frequently used features or components which are stored as a library of information. You can retrieve these features or components to avoid recreating geometries.

Catalogs can be built of two types of features:

A. **User Feature** is a group of features that exists as one entity and when placed in a model, it is represented by a single feature.

B. **PowerCopy** is also a group of features that exist as one entity. However, when placed in a model the original order and the state of the features are preserved. This makes it easy to modify the features after placing the PowerCopy.



Reusing Tools

Transformations

1 **Translate:** moves a body in a linear direction

2 **Rotate:** rotates a body around a selected axis

3 **Symmetry:** mirrors a body without duplicating it

4 **Mirror:** duplicates selected features across a reference plane to create mirror effect

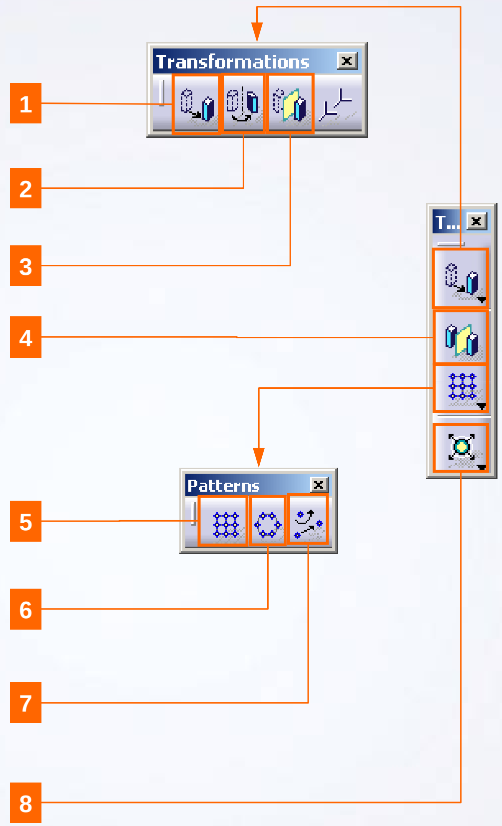
Patterns

5 **Rectangular Pattern:** creates array of selected features in linear directions

6 **Circular Pattern:** creates array of selected features in radial directions

7 **User Pattern:** creates array of selected features using locations defined in an existing sketch of points

8 **Scaling:** shrinks or expands an entire body based on a given ratio



Exercise: Patterns

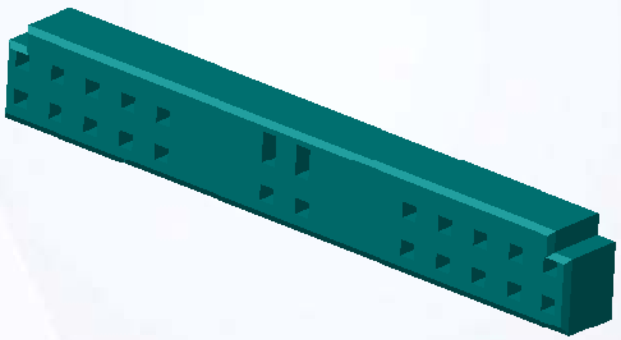
Recap Exercise

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** Conf. Dep.
- **Modify an instance of the pattern**

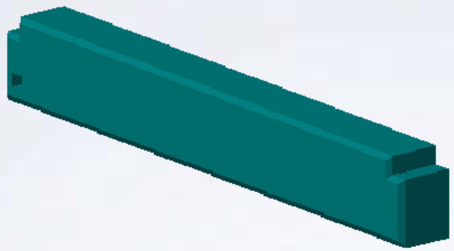


Do it Yourself (1/3)

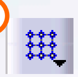
You will be opening up an existing part, creating a rectangular pattern, removing some instances, exploding the pattern, and modifying some instances of the pattern.

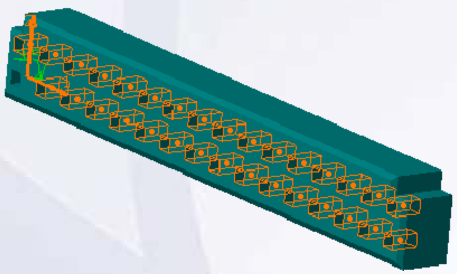
1. **Open the file Ex6B.CATPart.**
2. **Create a rectangular pattern of Pocket.1.**

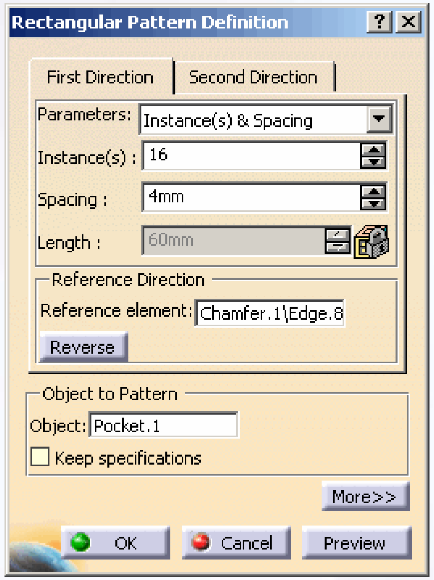
1

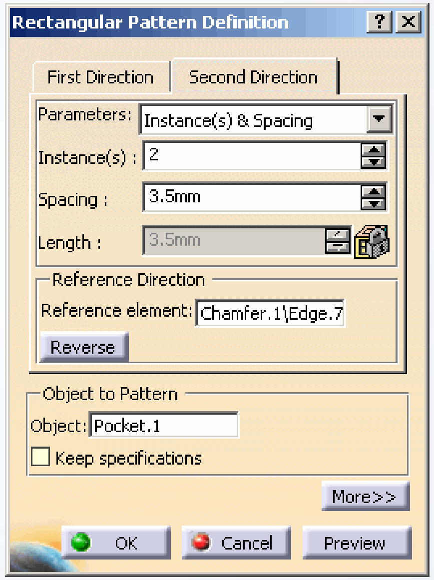


2



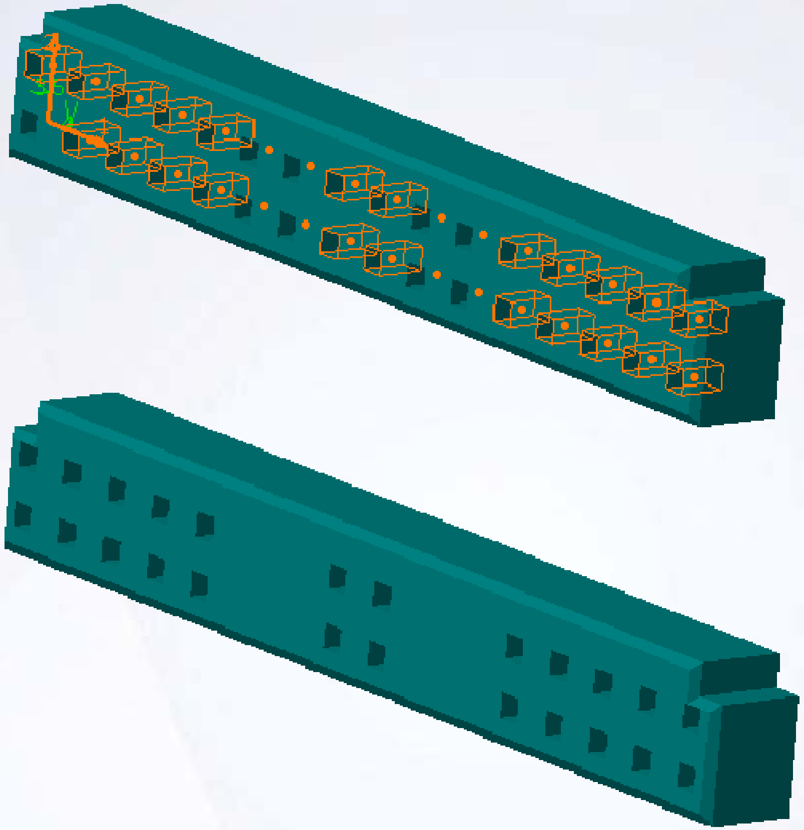






Do it Yourself (2/3)

3. **Remove the following instances from the pattern.**



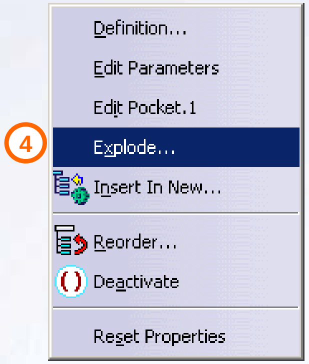
Do it Yourself (3/3)

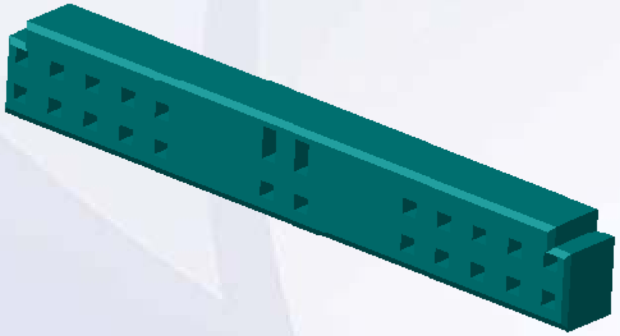
Conf. Dep.

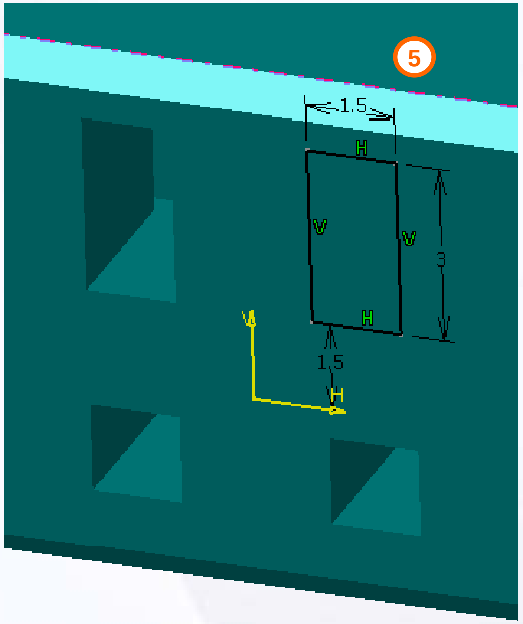
4. **Explode the pattern.**

5. **Modify the two pockets as per the following sketch.**

6. **Save and close the file.**

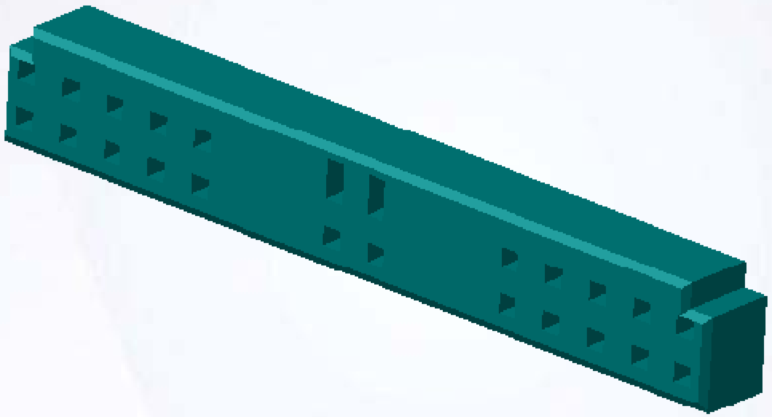






Exercise Recap: Patterns

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



Exercise: Patterns

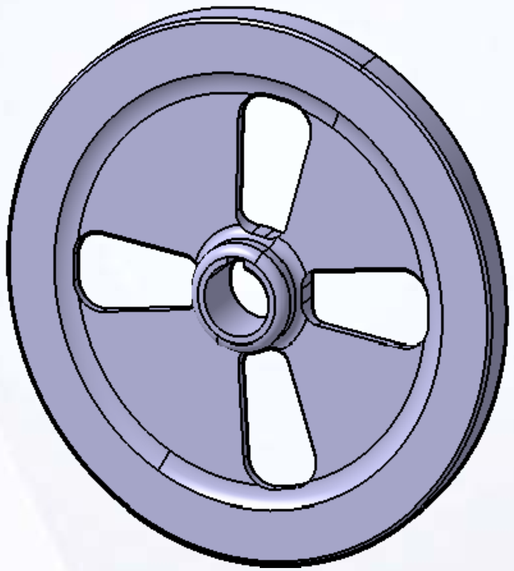
Recap Exercise

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.

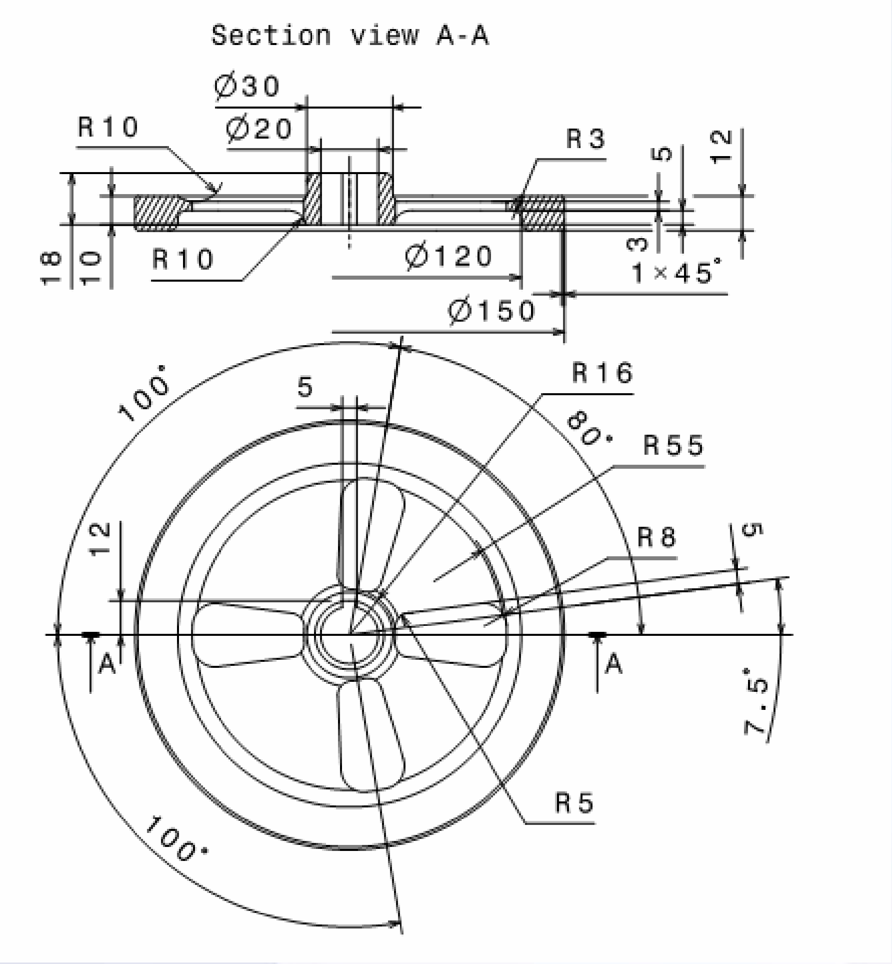
By the end of this exercise you will be able to:

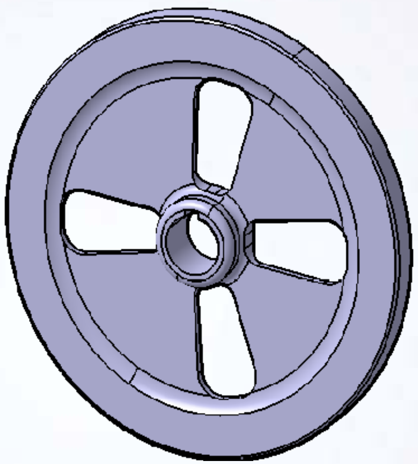
- **Create a new part**
- **Create a circular pattern**



Do it Yourself

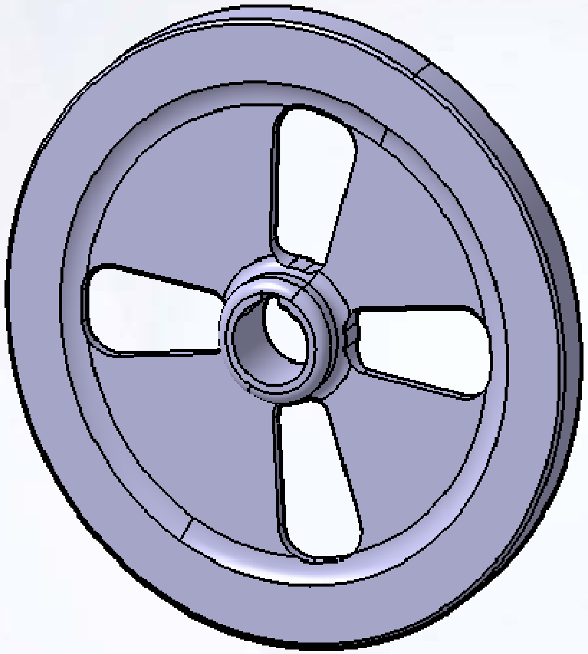
Create the part as shown below:





Exercise Recap: Patterns

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



Exercise: Patterns Modifications

Conf. Dep.

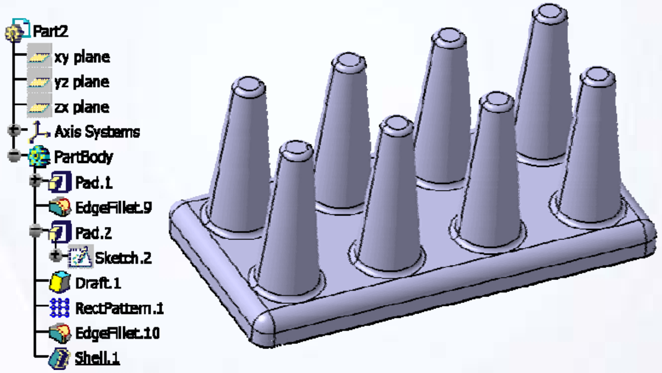
Recap Exercise

10 min

In this exercise you will modify an existing pattern by exploding and removing instances. You will also copy and paste one of the exploded instances and make changes in the copied feature. High-level instructions for this exercise are provided.

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

- **Explode a pattern**
- **Remove instances of the pattern**
- **Copy and paste features**
- **Modify the copied features**

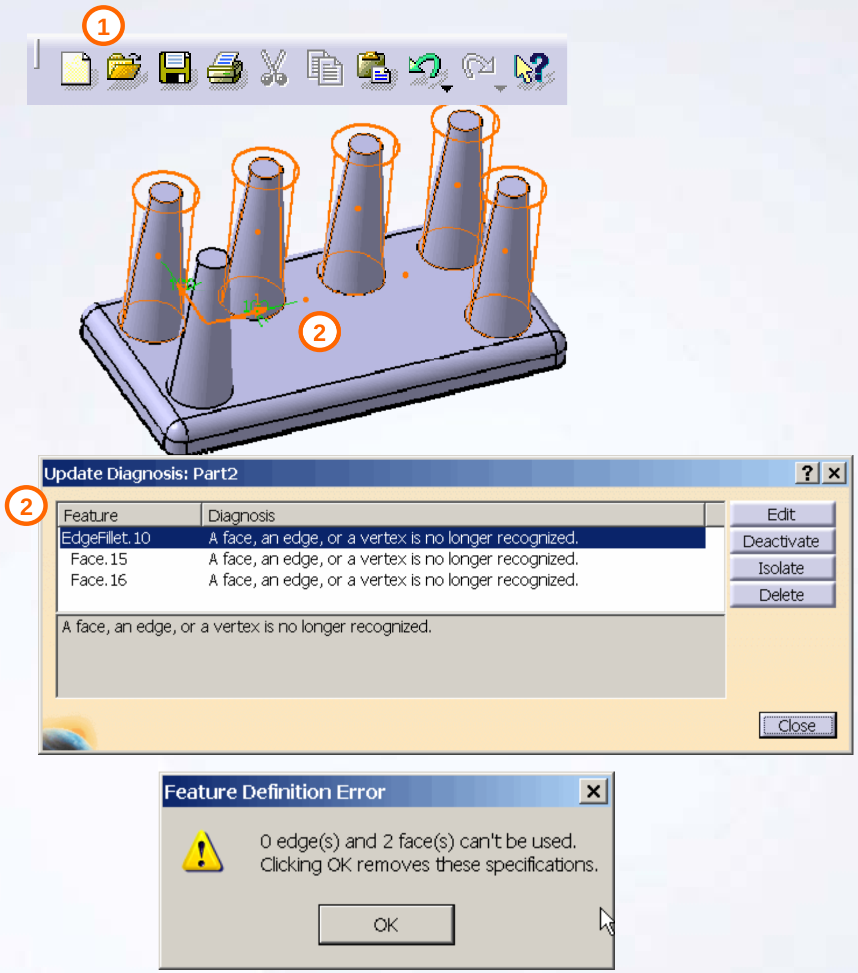


Do it Yourself (1/3)

In this exercise, you will open an existing part file and manipulate instances of a pattern.

1. **Open Ex6Econeholder.CATPart.**
   - Open an existing part file using the Open tool and investigate the features in the specification tree.

2. **Edit the pattern and remove instances.**
   - Remove two instances of the pattern by clicking on the pattern instance dots. Note that the fillet feature will fail because it was created after the pattern feature. Removing instances of the pattern will also remove references for the fillet feature. Select Edgefillet.10, select **Edit**, and click **OK** to remove the missing references. Click **OK** on the Edge Fillet definition dialog box.



Do it Yourself (2/3)

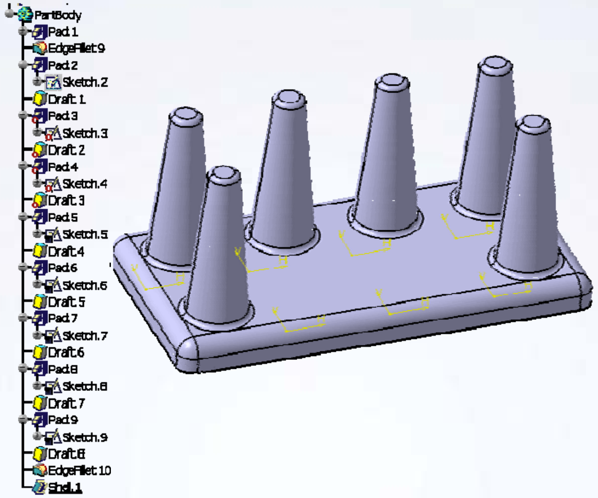
Conf. Dep.

3. **Explode the pattern.**
   - Explode the pattern and notice that the fillet and shell features that were created after the pattern are not deleted.

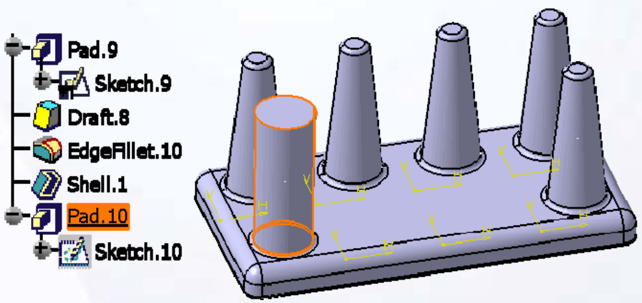
4. **Copy and paste an instance.**
   - Copy Pad.2 and paste it into the same part.

5. **Modify the copied instance.**
   - Edit the sketch of the copied pad and move the sketch to the centre of the part. Change the pad length to [100mm].

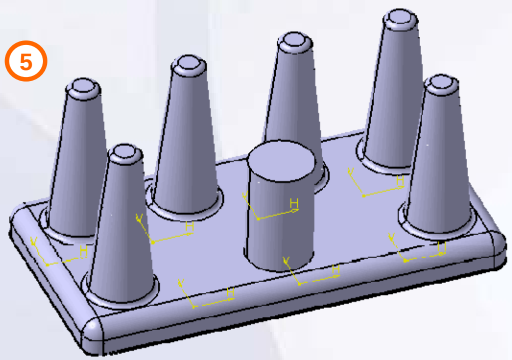
3



4



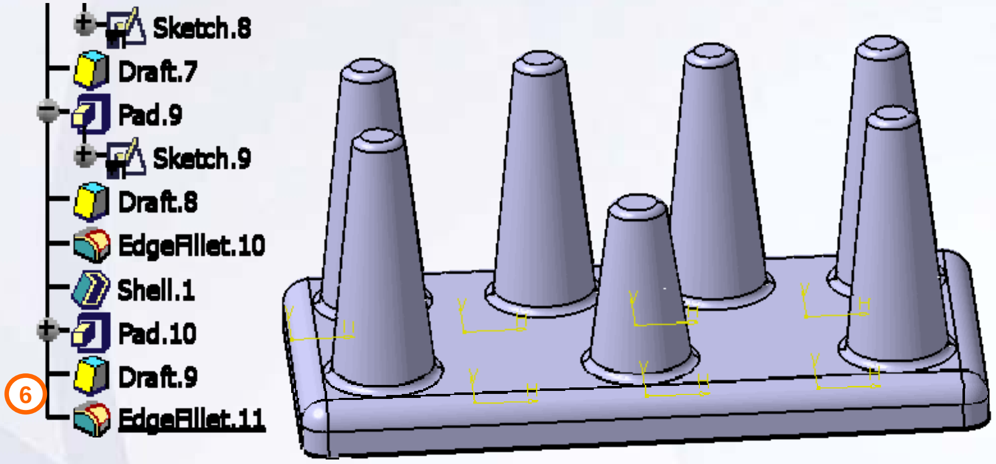
5



Do it Yourself (3/3)

6. **Add a draft and fillet to the instance.**
   - Add an [8 degree] draft and [5mm] fillets to the copied instance.

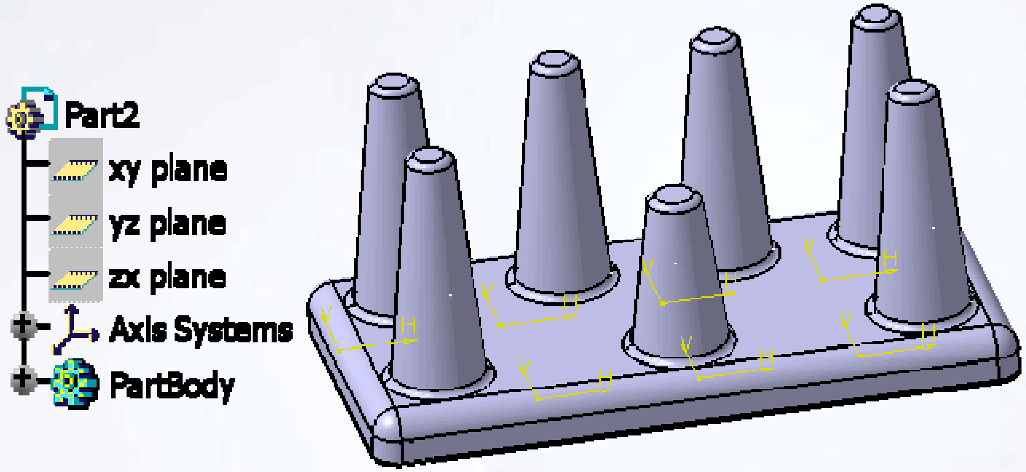
7. **Save and close the part.**



Exercise Recap: Patterns Modifications

Conf. Dep.

- ✓ Remove pattern instances
- ✓ Explode pattern
- ✓ Copy and paste a pad instance
- ✓ Modify the copied pad
- ✓ Create a draft and fillets



Exercise: Pattern and Catalog

Recap Exercise

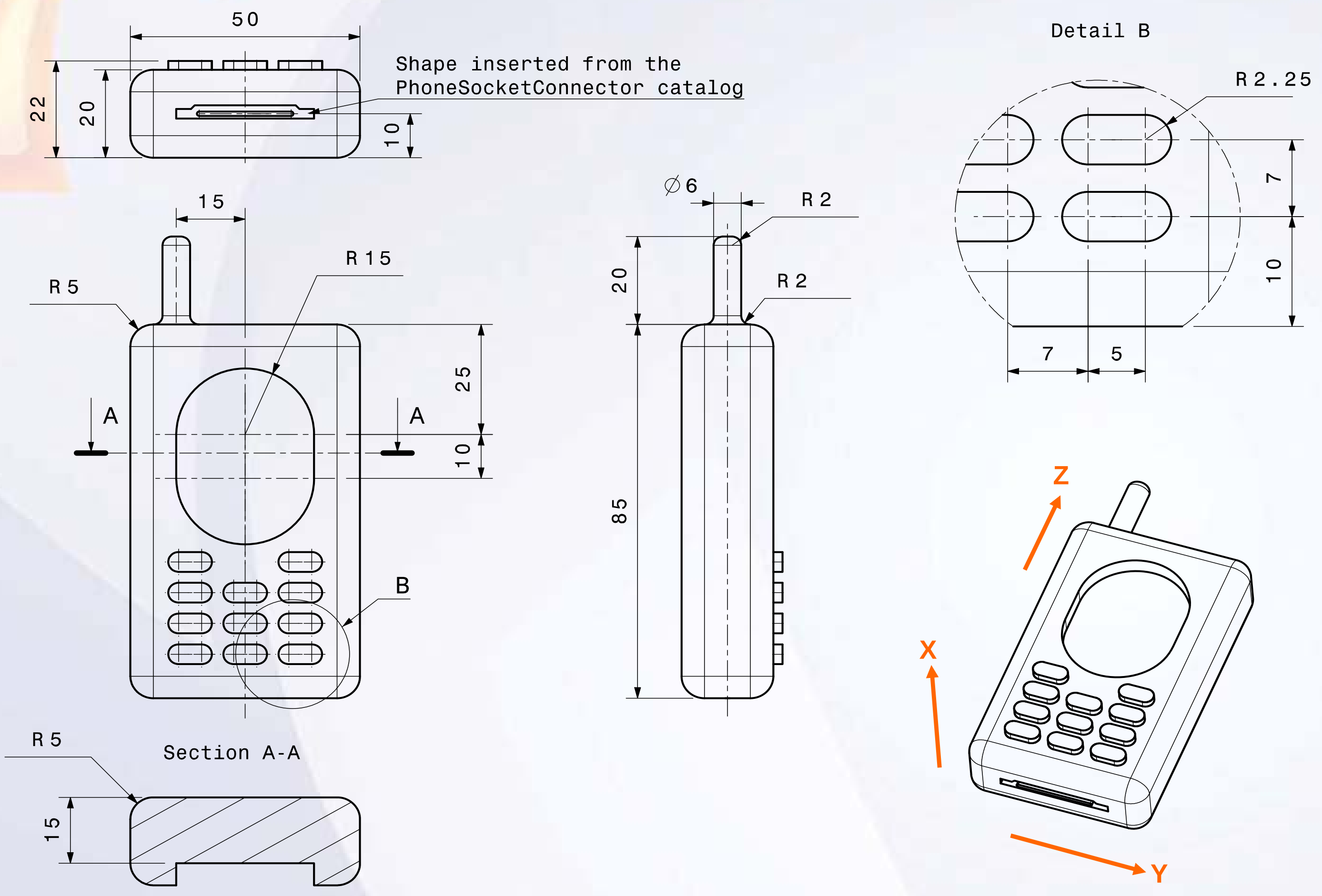
30 min

In this exercise you will create a phone model. 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 pattern**
- **Remove instances of a pattern**
- **Insert a feature from a catalog**

Do it Yourself



Exercise Recap: Pattern and Catalog

- ✓ Create a pattern
- ✓ Remove instances of a pattern
- ✓ Insert a pocket feature from a catalog

Case Study: Reusing Data

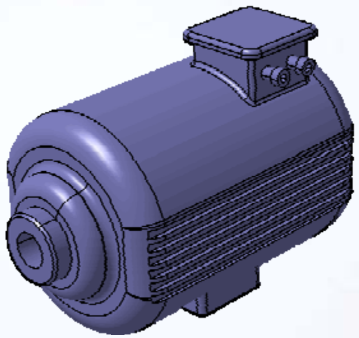
Recap Exercise

35 min

You will practice what you have learned by completing the case study model using only a detailed drawing and hints as guidance.

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

- ✓ The side fins should be created by a rectangular pattern
- ✓ The hole pattern should be created by a user pattern
- ✓ The model should be partially created by copying and pasting features
- ✓ The model should include features from a catalog



Using the techniques discussed so far, create the model without detailed instructions.

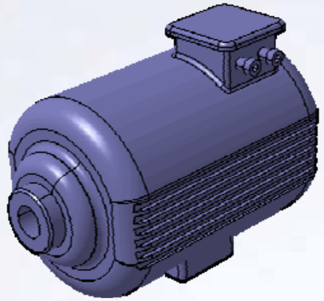
Do It Yourself: Drawing of the Engine (1/3)

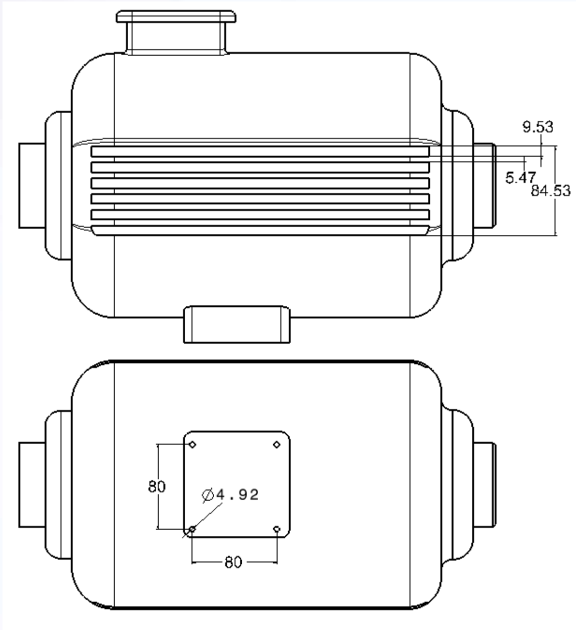
The following steps offer useful hints to guide you through the creation of the engine part:

1. Open CaseStudy.CATPart.

2. Create a rectangular pattern for the side fins.

3. Create a user pattern for the hole pattern.





Do It Yourself: Drawing of the Engine (2/3)

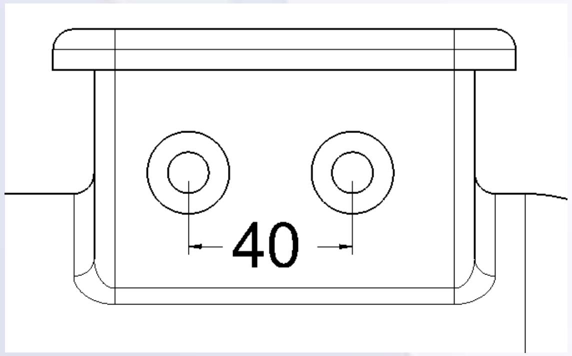
The following steps offer useful hints to guide you through the creation of the engine part (continued):

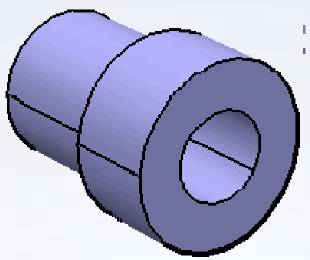
Conf. Dep.

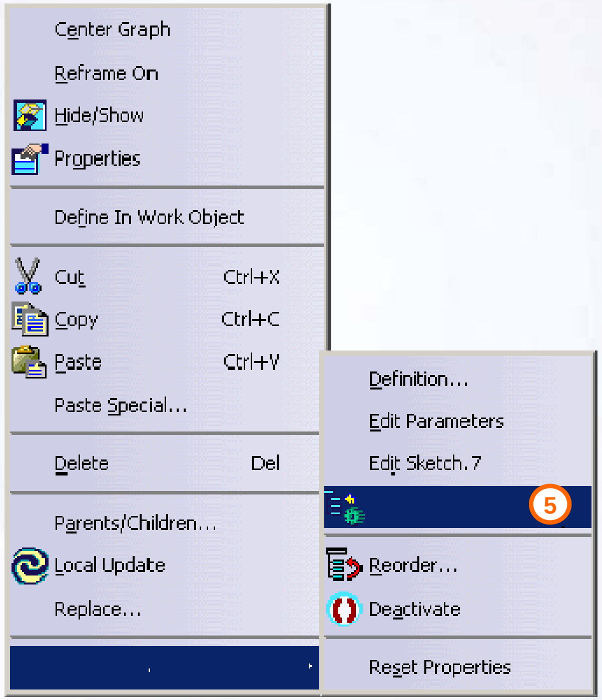
4. Create copies of Pad.3, Pad.5 and Hole.1 one after the other and paste them one by one into the model.

5. Move the pasted features into a separate body by selecting the features, and selecting **Insert In New..** from the contextual menu.

6. Translate the body to a distance of [40] in the X direction.





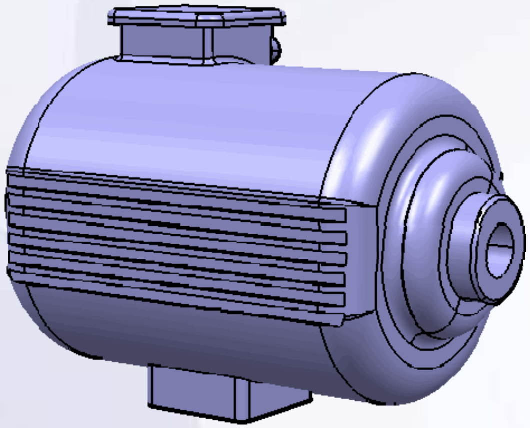


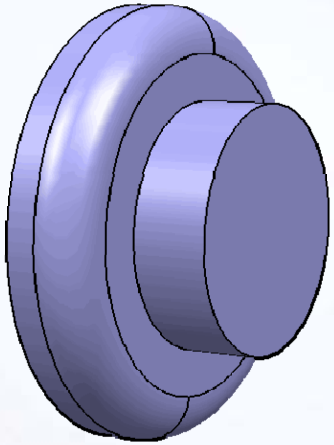
Do It Yourself: Drawing of the Engine (3/3)

The following steps offer useful hints to guide you through the creation of the engine part (continued):

7. Insert data from the supplied CaseStudy.catalog.

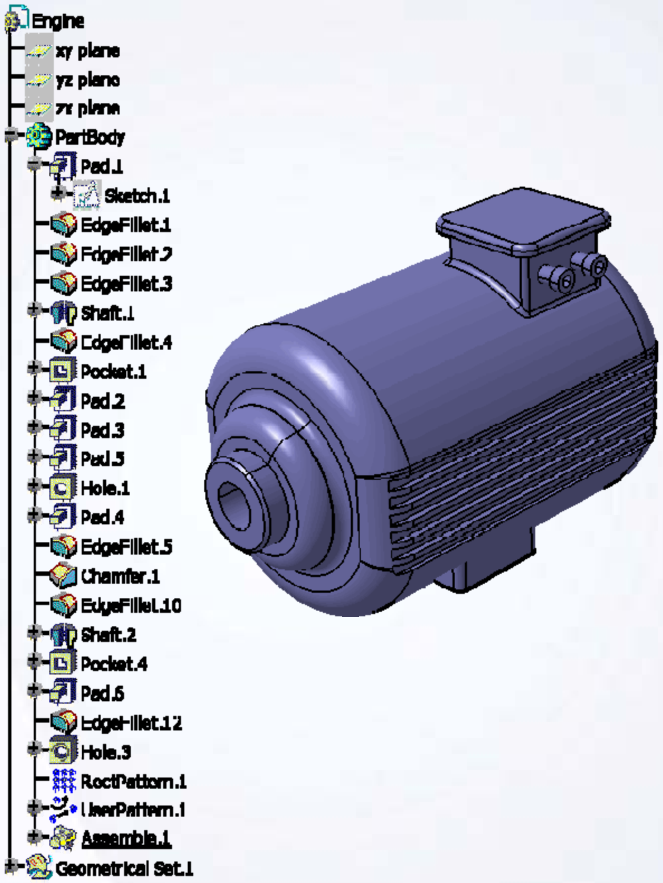
8. Reorder the catalog Shaft feature to occur before Pocket.4 in order to apply the pocket feature to the catalog geometry.

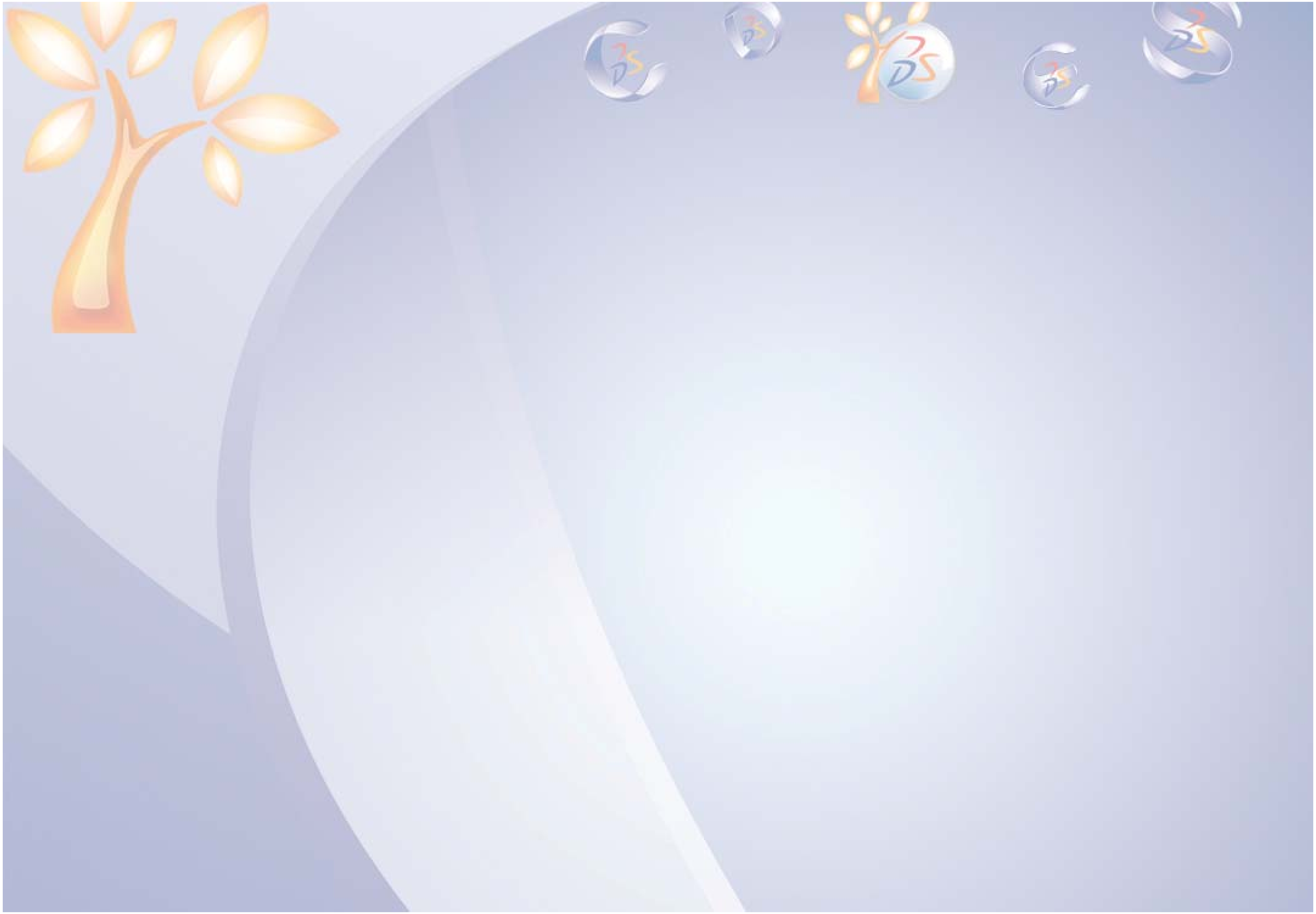




Case Study: Engine Recap

- ✓ The side fins should be created by a rectangular pattern
- ✓ The hole pattern should be created by a user-defined pattern
- ✓ The model should be partially created by copying and pasting the features
- ✓ The model should include features from a catalog



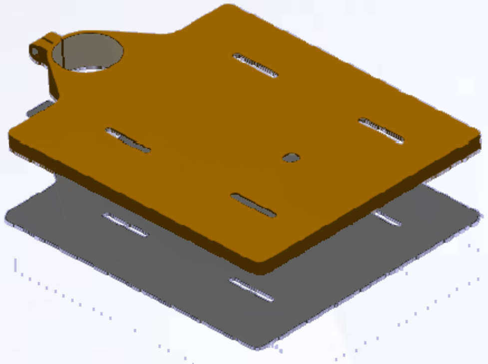


Finalizing Design Intent 7

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Apply Material Properties
- ✓ Analyze the Model
- ✓ Create Formulas and Parameters



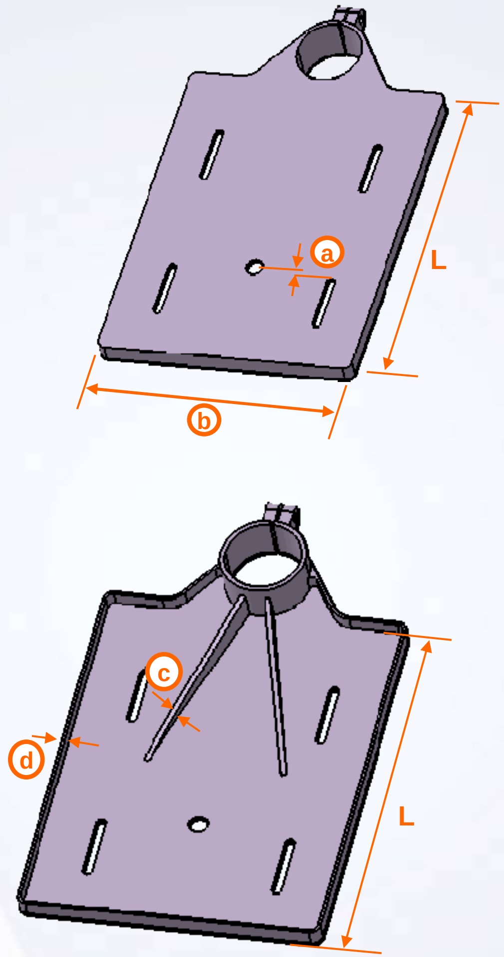
4 Hours

Case Study

The case study for this lesson is the table used in the Drill Support assembly.

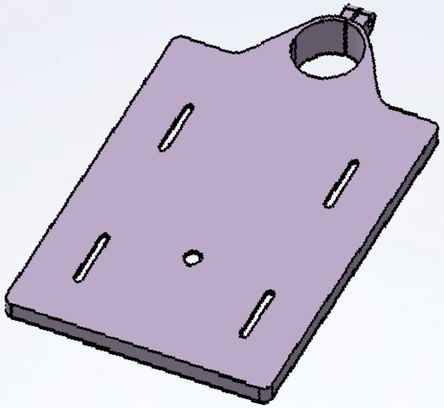
Design Intent

- ✓ The model must be made of aluminum.
  - The material selected for this part is aluminum. The material properties of aluminum in the CATIA library will meet the requirements.
- ✓ Model geometry must adhere to the following criteria (which can be verified using measurement tools and enforced using formulas):
  a. Create a hole that is always 2mm above the bottom oblong holes and is centered horizontally.
  b. Overall width must be 80% of the length (L).
  c. The thickness of the model is always 1% of the length (L).
  d. The thickness of the ribs is two times the thickness of the model.



Stages in the Process

1. Apply Material Properties
2. Analyze the Model
3. Create Formulas and Parameters



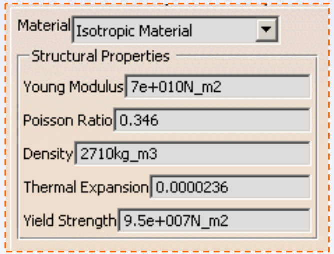
Apply Material Properties

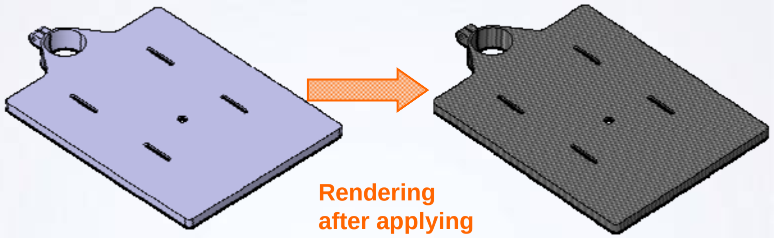
In CATIA you can apply material to any part. These material properties (e.g., density) affects the mass of the part. These properties play important role in the structural and thermal analyses of the part.

CATIA has a default library of materials already installed. Your company may have custom materials created to conform to your requirements.

You can render the material on the model using a customized view mode or shading with material mode.

Properties important for thermal and structural analyses





Rendering after applying material.

Analyze the Model

There are three types of measure modes:

A. The Measure Between tool

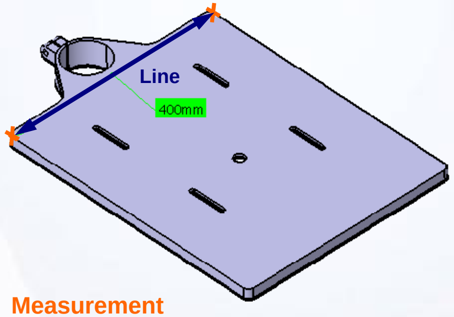
B. The Measure Item tool

C. The Measure Inertia tool

All measurements can be saved in the specification tree by selecting the Keep Measure option. All measurement tools have an option to create a geometry. Points, lines, and axis systems related to measurement can be created to illustrate the measurement. This is called as measurement geometry.

Second point

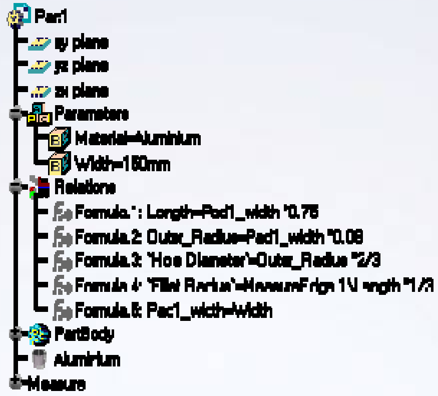
First point



Measurement Geometry

Create Formulas and Parameters

All features and elements in CATIA are unique. Once the features are created, they receive a unique identifier (parameter). These parameters can be used to create formulas. Formulas are equations that relate one parameter to another and ensure that the design intent is maintained. Formulas are stored under the Relations branch of the specification tree.



In addition to system generated parameters you can create new parameters. Such parameters are called as User-Defined parameters. User-Defined parameters are isolated until they are related to some geometric parameter in the model. User-defined parameters are stored under the Parameters branch of the tree.

It is important to consider units when you write formulas. If units are not specified in a formula, the default unit is used (i.e., meters). This is particularly important if you are adding or subtracting a numerical value.

Main Tools

Measure

1 **Measure Between:** measures the distance between the elements in a model

2 **Measure Item:** measures a specific element in a model

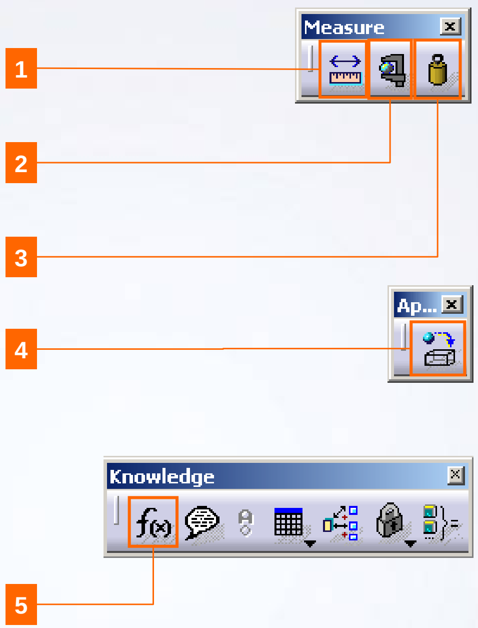
3 **Measure Inertia:** calculates the mass properties of the model

Material

4 **Apply Material:** applies material to any part in CATIA

Knowledge

5 **Formula:** creates new user-defined parameters and creates relation between different parameters



Exercise: Material and Measures

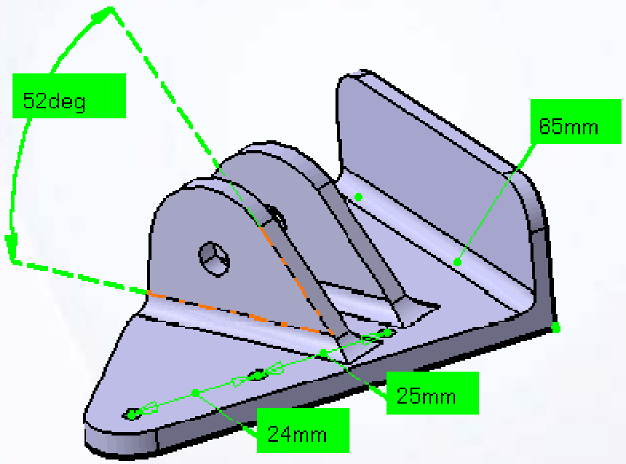
Recap Exercise

15 min

In this exercise you will use the measurement tools to determine specific dimensions on an existing model. High-level instructions for this exercise are provided.

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

- **Apply material to a model**
- **Take measurements**
- **Calculate mass properties**



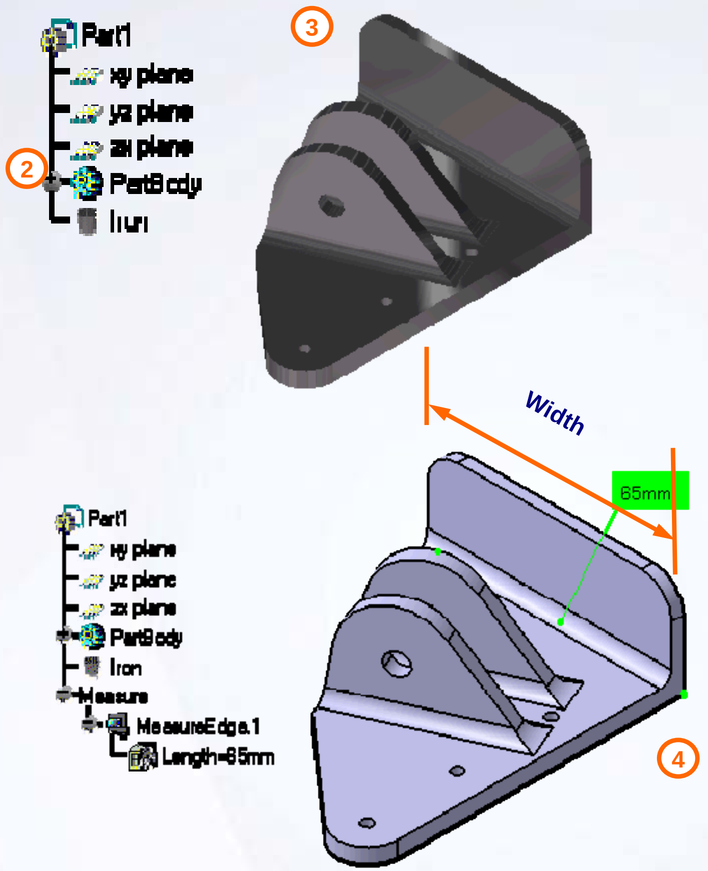
Do it Yourself (1/2)

1. Open Ex7B.CATPart.

2. Apply Iron material to the model.

3. View the applied material.

4. Determine the width of the part.



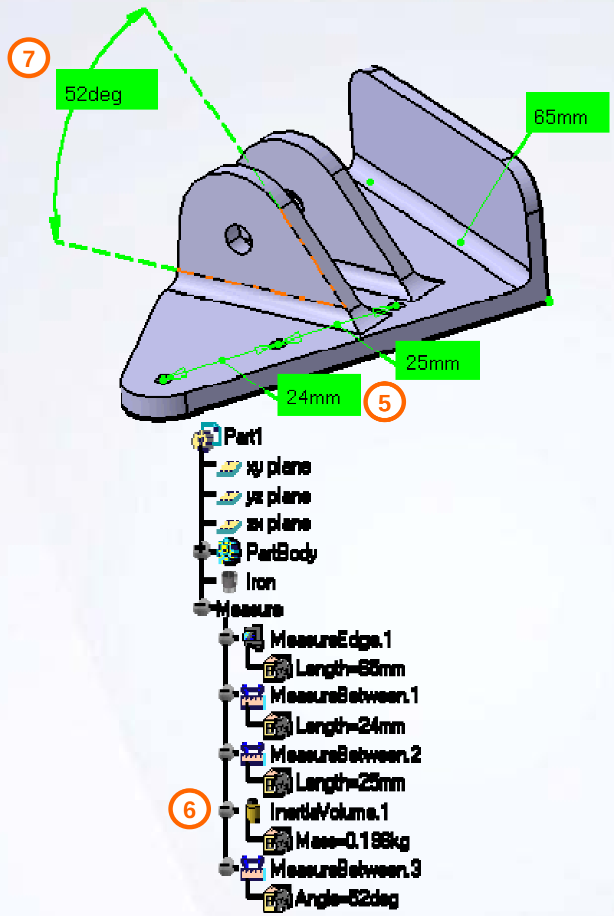
Do it Yourself (2/2)

5. Calculate the distance between the three center points of the three holes.

6. Determine the mass of the model.

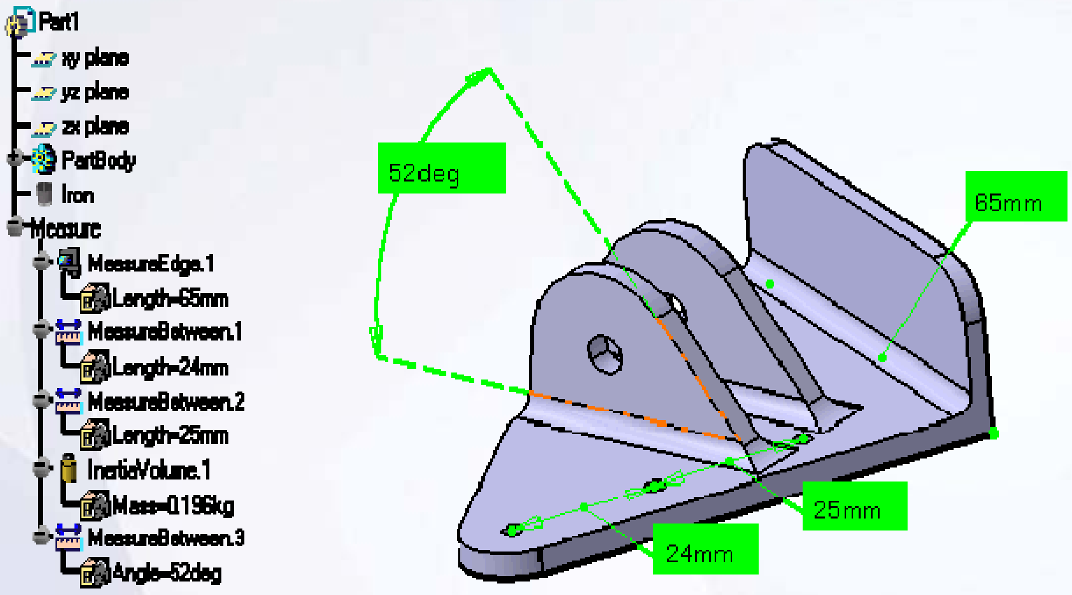
7. Measure the angle as shown.

8. Save and close the file.



Exercise Recap: Material and Measures

- ✓ Apply material to a model
- ✓ Take measurements
- ✓ Calculate mass properties



Exercise: Parameter and Formula

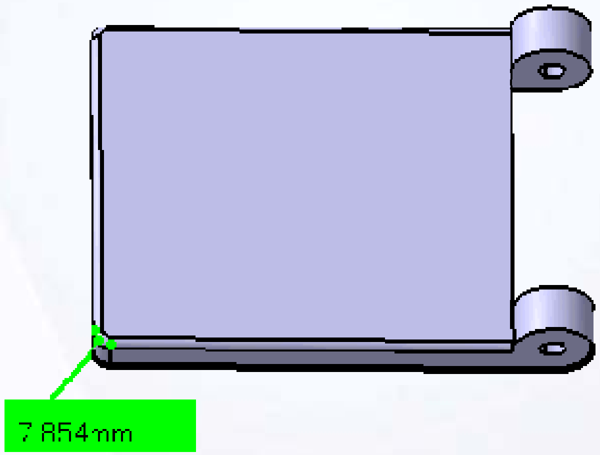
Recap Exercise

20 min

In this exercise you will practice maintaining design intent by creating formulas and parameters. You will use the tools used in the previous exercises to complete this exercise. High-level instructions for this exercise are provided.

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

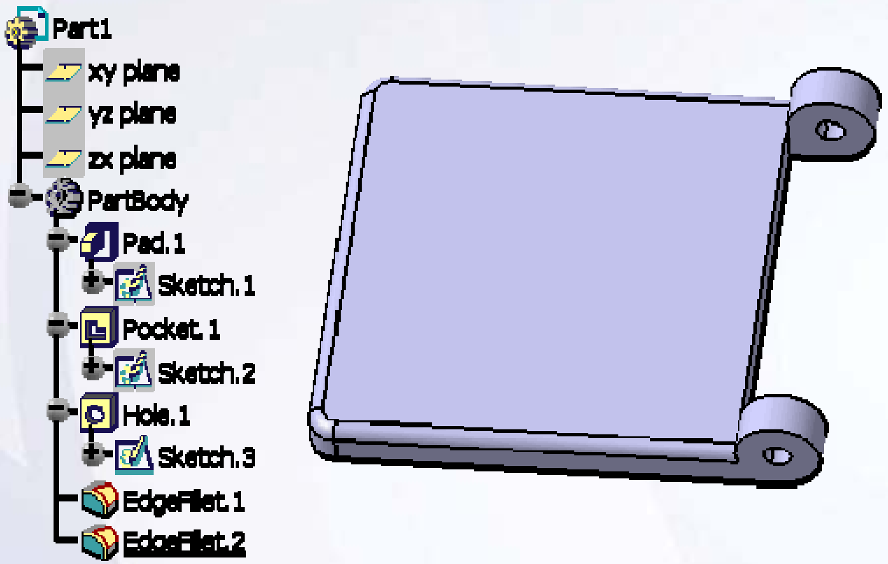
- Create formulas
- Create user-defined parameters



Do it Yourself (1/7)

1. **Open Ex7D.CATPart.**
   - Open an existing part file using the **Open** tool.

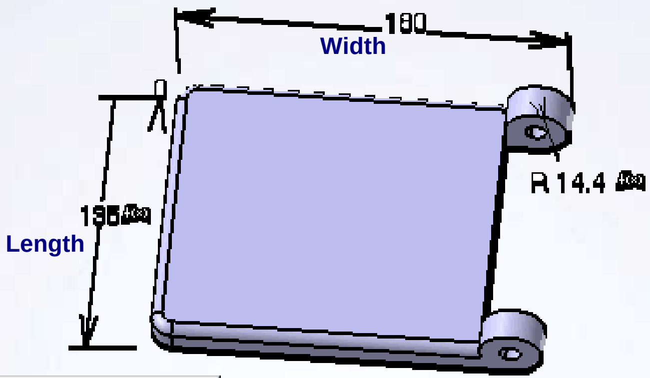
2. **Review the model.**
   - Review the model creation. Are there any existing formulas?

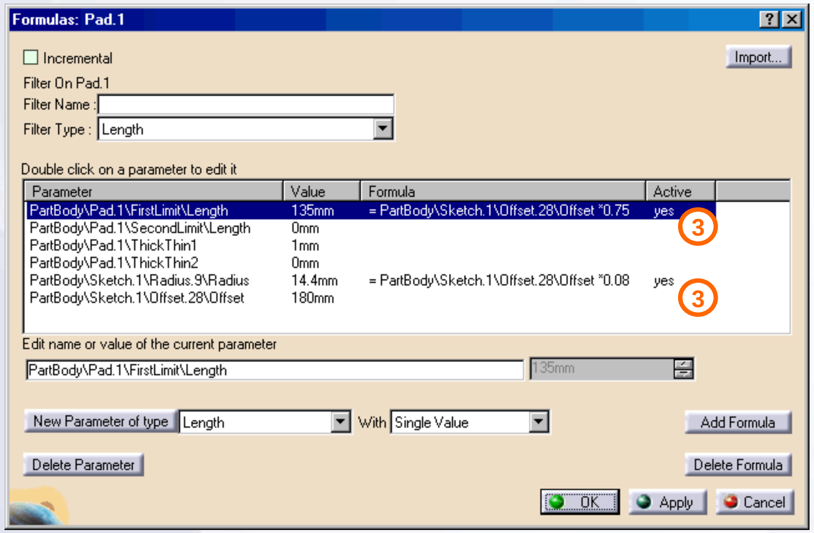


Do it Yourself (2/7)

3. Control all Pad.1 dimensions with the width.

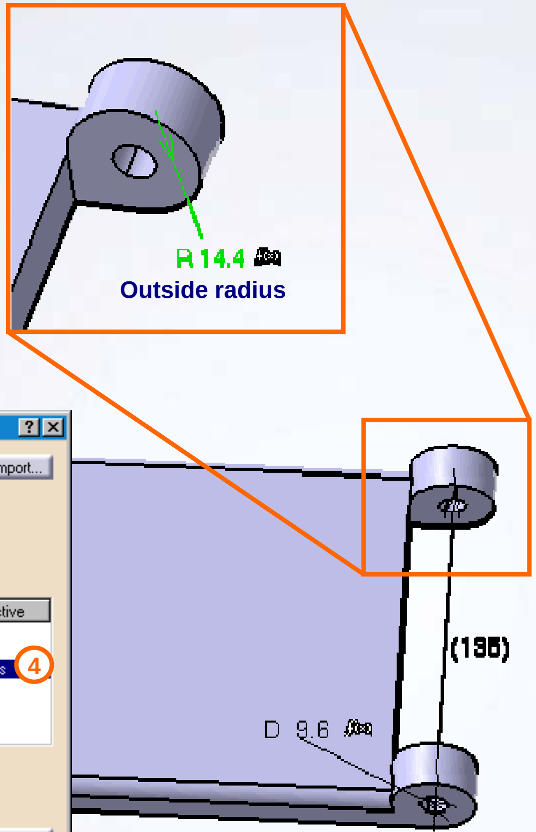
- Create formulas so that changing the width of pad.1 also updates the radius and the length. The radius should be 8% of the width and the length should be 75% of the width.

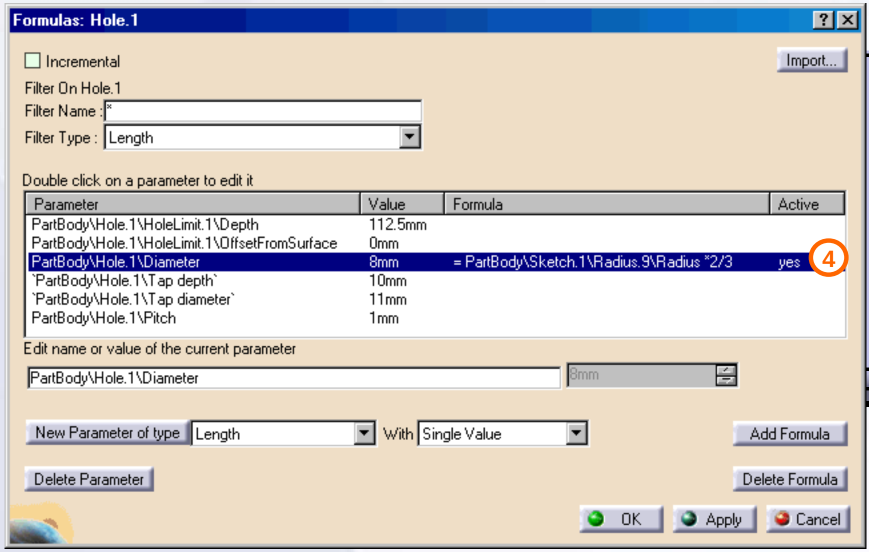




Do it Yourself (3/7)

4. **Create a formula.**
   - Create a formula to link the diameter of the co-axial hole with the outside radius. The co-axial hole's diameter should be two- third (2/3) of the outside radius.



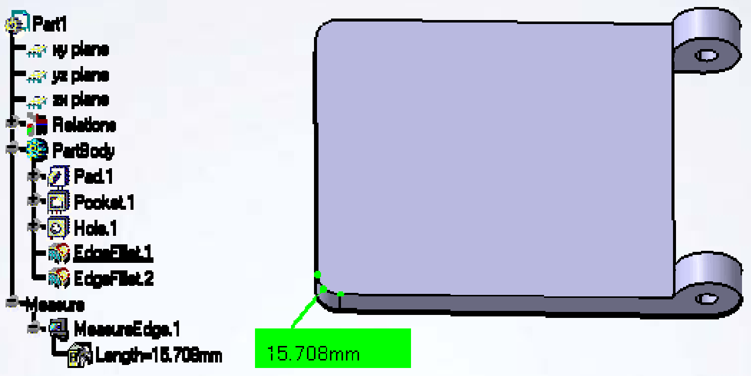


Do it Yourself (4/7)

The next two steps are used to create a formula that controls the radius of EdgeFillet.2 based on the arc length of EdgeFillet.1.

5. Measure arc length of the edge fillet.

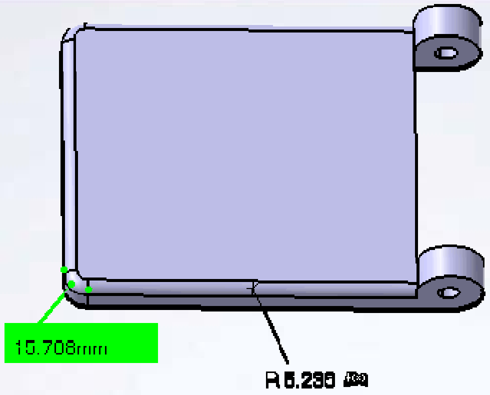
- In order to use a measurement in a formula, you must create the measurement before creating the feature where you want to use it. As a workaround, define EdgeFillet.1 as the object and take the measurement. By doing this, the measurement comes before EdgeFillet.2 in the regeneration cycle:
  a. Right-click on EdgeFillet.1 and select **Define in Work Object**.
  b. Calculate the arc length of EdgeFillet.1. You will need to customize the measurement to calculate the length.
  c. Save the measurement in the specification tree.

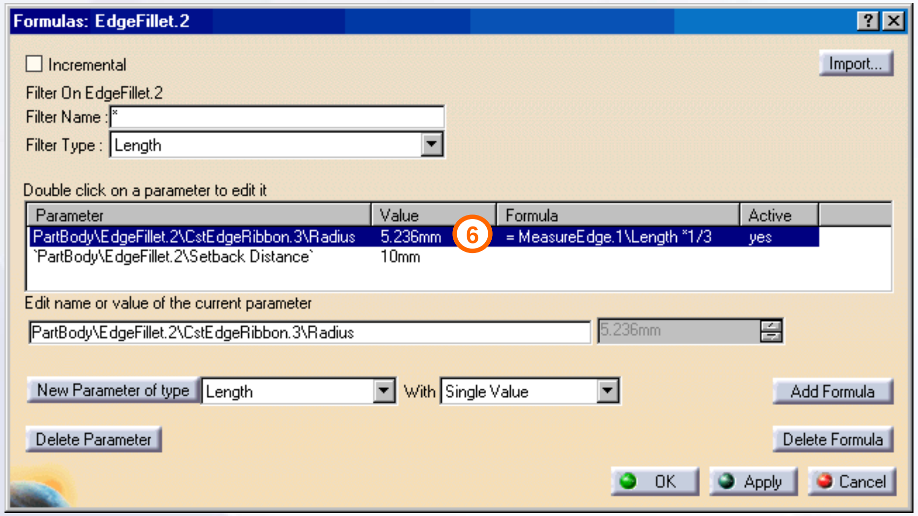


Do it Yourself (5/7)

6. Create a formula.

- Create a formula that equates the radius of EdgeFillet.2 to one-third (1/3) the arc length measured in the last step.

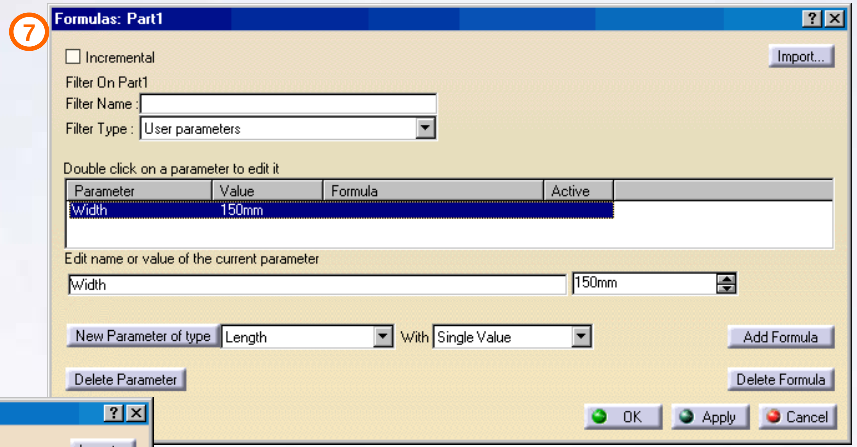


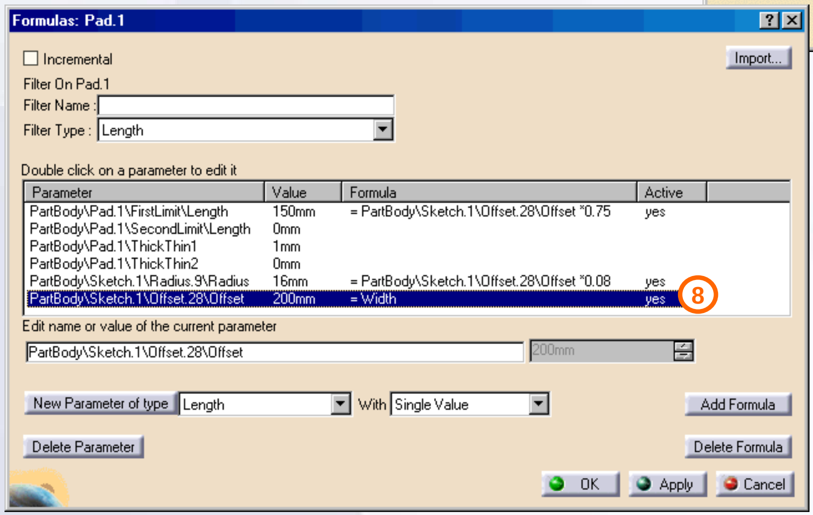


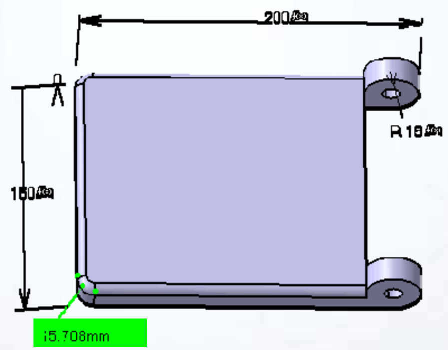
Do it Yourself (6/7)

7. **Create a user-defined parameter.**
   - Type = [Length]
   - Name = [Width]
   - Value = [200mm]

8. **Equate the width of the part to the new Width parameter.**
   - Set the Width dimension (PartBody\Sketch.1\Offset.28\Offset) of Pad.1 equal to the new parameter.





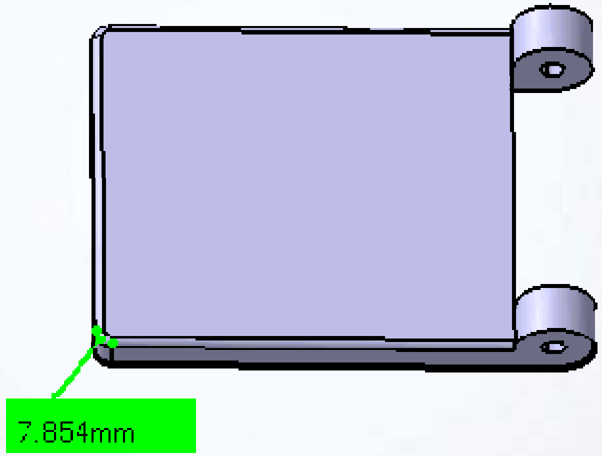


Do it Yourself (7/7)

9. Test the model.

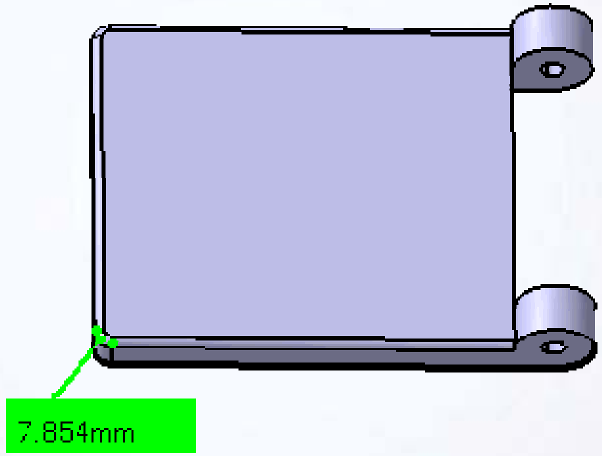
- Change the Width parameter to [150mm] and change the radius of EdgeFillet.1 to [5mm]. Update the model. Did the model update as expected?

10. Save and close the file.



Exercise Recap: Parameter and Formula

- ✓ Create formulas
- ✓ Create user-defined parameters



Exercise: Parameter and Formula

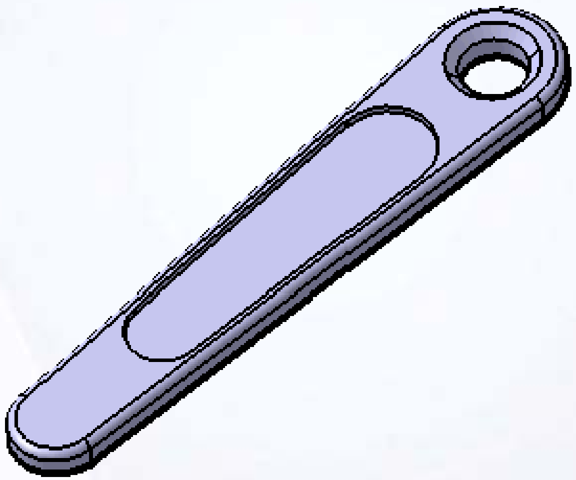
Recap Exercise

15 min

In this exercise, you will create formulas and parameters to control dimensions in the model. You will use the tools you have learned in this lesson to complete the exercise with no detailed instructions.

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

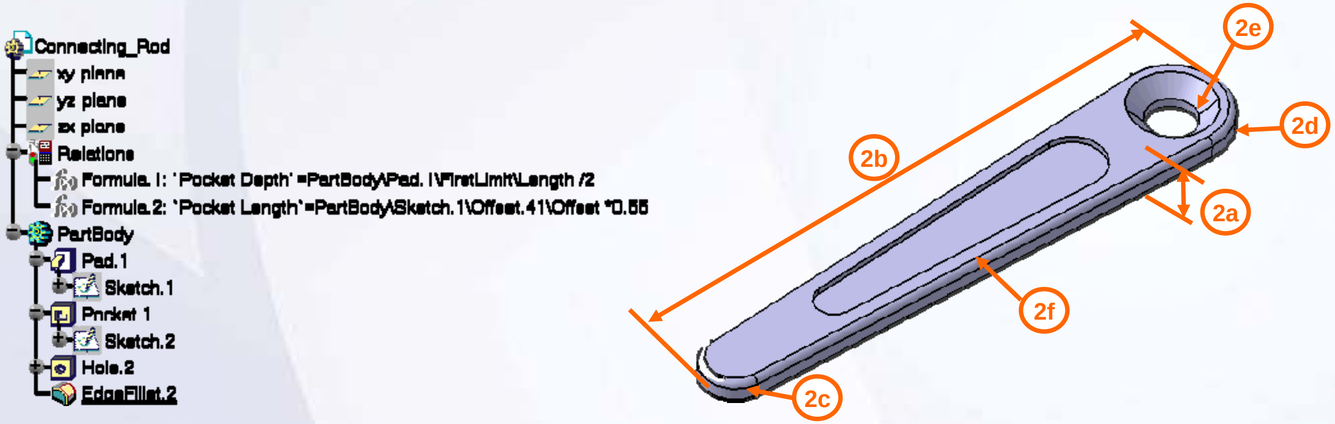
- **Rename parameters**
- **Create formulas**
- **Create user-defined parameters**



Do it Yourself (1/3)

1. **Open Ex_7E.CATPart.**
2. **Rename the following parameters to create the formulas:**

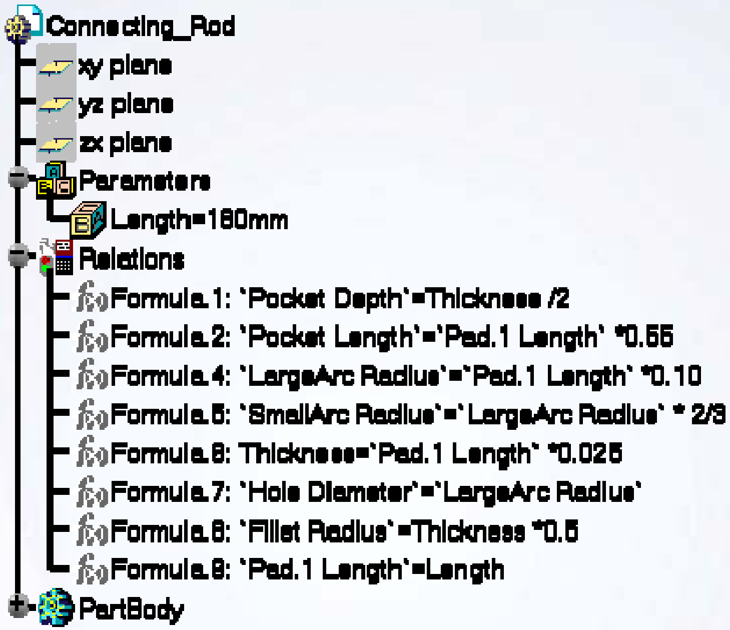
	Current Name	New Name
a.	PartBody\Pad.1\FirstLimit\Length	Thickness
b.	PartBody\Sketch.1\Offset.41\Offset	Pad.1 Length
c.	PartBody\Sketch.1\Radius.13\Radius	SmallArc Radius
d.	PartBody\Sketch.1\Radius.14\Radius	LargeArc Radius
e.	PartBody\Hole.2\Diameter	Hole Diameter
f.	PartBody\EdgeFillet.2\CstEdgeRibbon.16\Radius	Fillet Radius



Do it Yourself (2/3)

3. **Create a parameter.**
   - Type = [Length].
   - Name = [Length].
   - Value = 180mm].

4. **Create the following formulas:**
   - LargeArc Radius = 10% of Length.
   - SmallArc Radius = 2/3 LargeArc Radius.
   - Thickness = 2.5% Length.
   - Hole Diameter = LargeArc Radius.
   - Fillet Radius = ½ Thickness.
   - Pad.1 Length = Length.

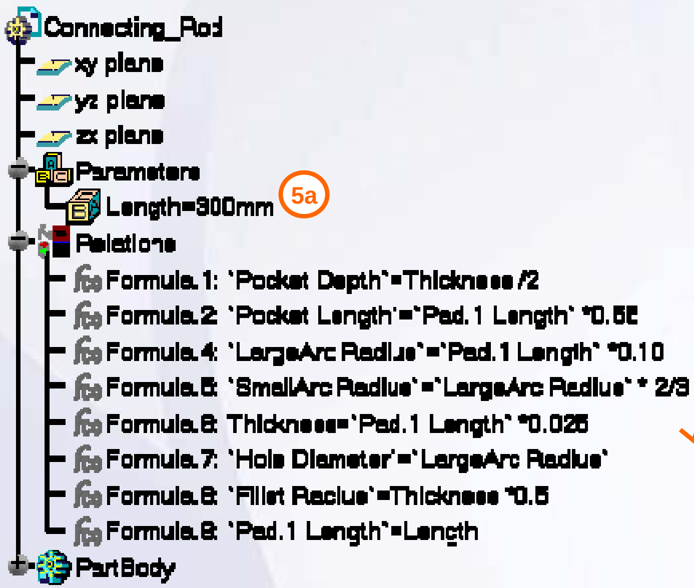


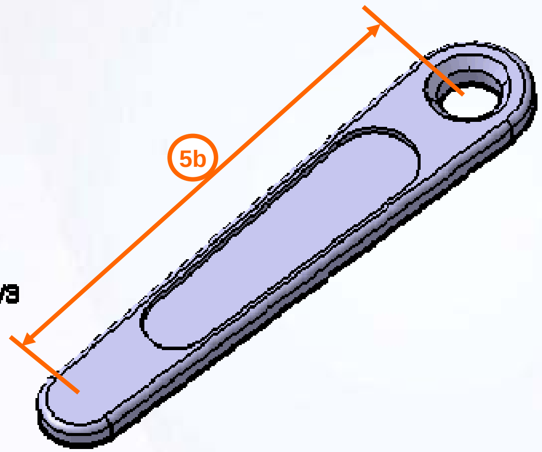
Do it Yourself (3/3)

5. Test the model.

a. Change the length parameter to a value of [300mm].

b. Use the **Measurement** tool to calculate the new distance between the center of the large arc and the center of the small arc.

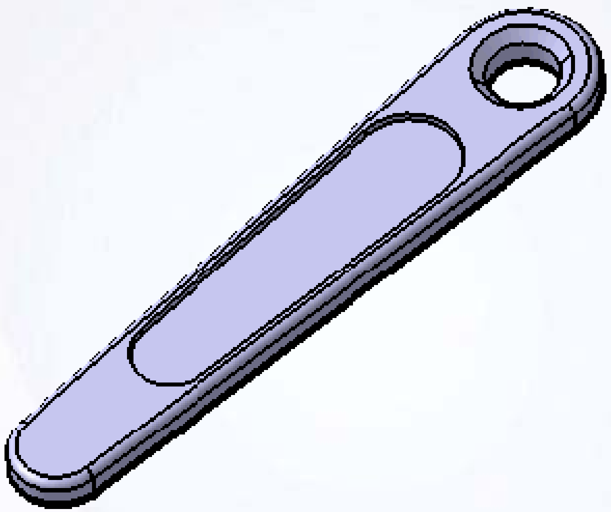




Exercise Recap: Parameter and Formula

- ✓ Rename parameters
- ✓ Create formulas
- ✓ Create user-defined parameters

Connecting_Rod
- xy plane
- yz plane
- zx plane
- Parameters
  - Length=300mm
- Relations
  - Formula.1: `Pocket Depth`=Thickness /2
  - Formula.2: `Pocket Length`=`Pad.1 Length` *0.55
  - Formula.4: `LargeArc Radius`=`Pad.1 Length` *0.10
  - Formula.5: `SmallArc Radius`=`LargeArc Radius` * 2/3
  - Formula.6: Thickness=`Pad.1 Length` *0.025
  - Formula.7: `Hole Diameter`=`LargeArc Radius`
  - Formula.8: `Fillet Radius`=Thickness *0.5
  - Formula.9: `Pad.1 Length`=Length
- PartBody



Case Study: Finalizing Design Intent

Recap Exercise

30 min

You will practice what you learned by completing the case study model using only a detailed drawing as a guidance.

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

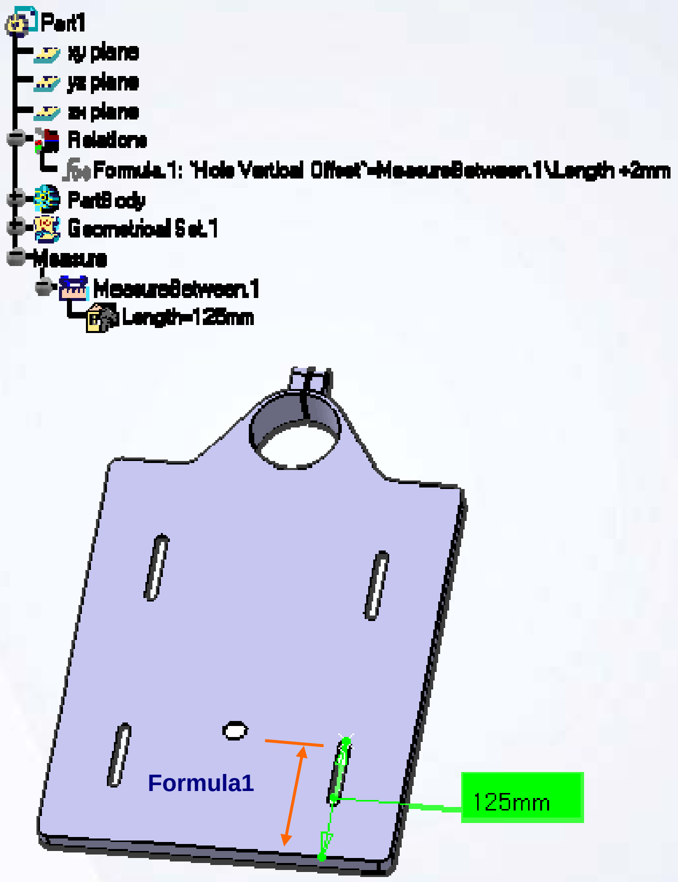
- ✓ The model must be made of aluminum.
- ✓ Create a hole that is centered on the part horizontally and 2mm above the top of the bottom oblong holes.
- ✓ When modifications are made to the model:
  - Overall width must be 80% of the length (L).
  - Thickness of the model is always 1% the length (L) of the model.
  - The thickness of the ribs is 2 times the thickness of the model.

Using the techniques discussed so far, create the model without detailed instructions.

Do It Yourself: Finalizing Design Intent (1/5)

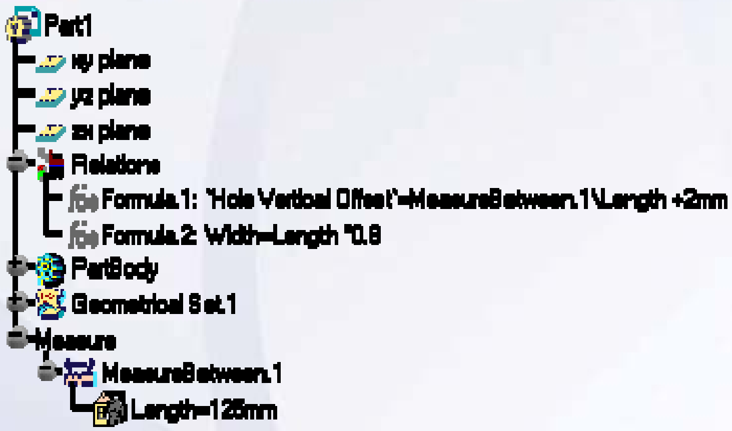
Use the following steps as hints to create the model:

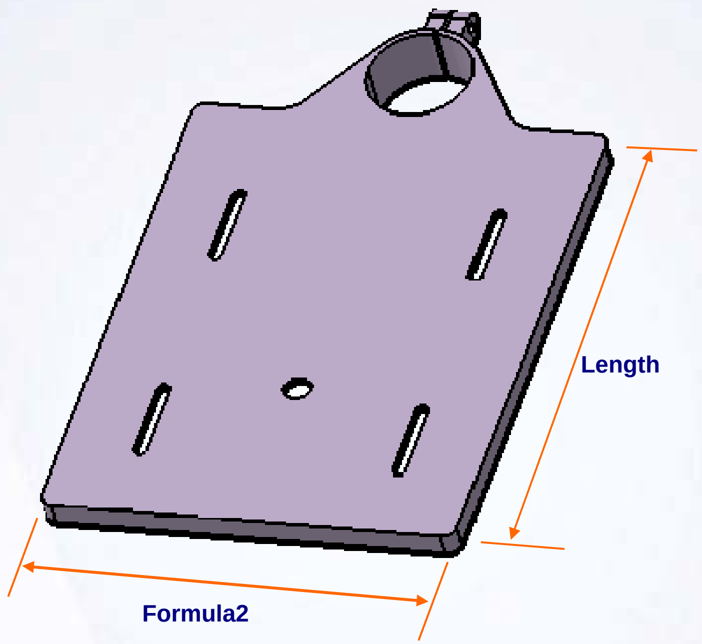
1. **Open CS_L7.CATPart.**
2. **Create a 20mm diameter hole with the following requirements:**
   - Hole must remain 2mm above the top of the bottom oblong holes.
     a. To do this, you can create a measurement to calculate the distance from the bottom of the model to the top of the bottom oblong holes. For the measurement a point has already been created.
   - Hole must remain centered horizontally on the model.
     a. This model has been created symmetric about the YZ plane.



Do It Yourself: Finalizing Design Intent (2/5)

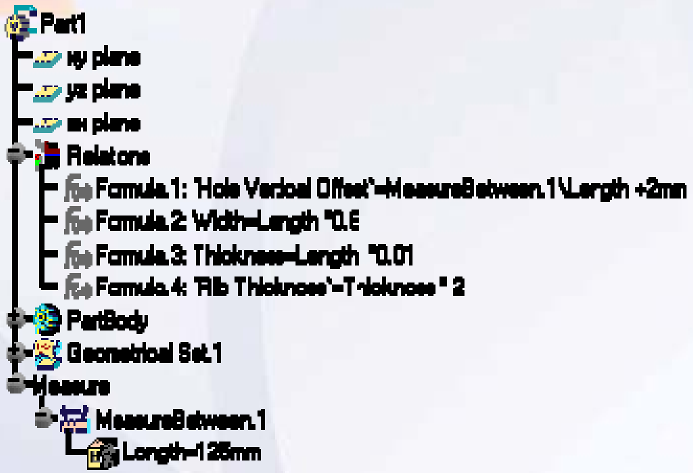
3. **Rename the parameters and create formulas to maintain the required design intent. The completed model is shown.**

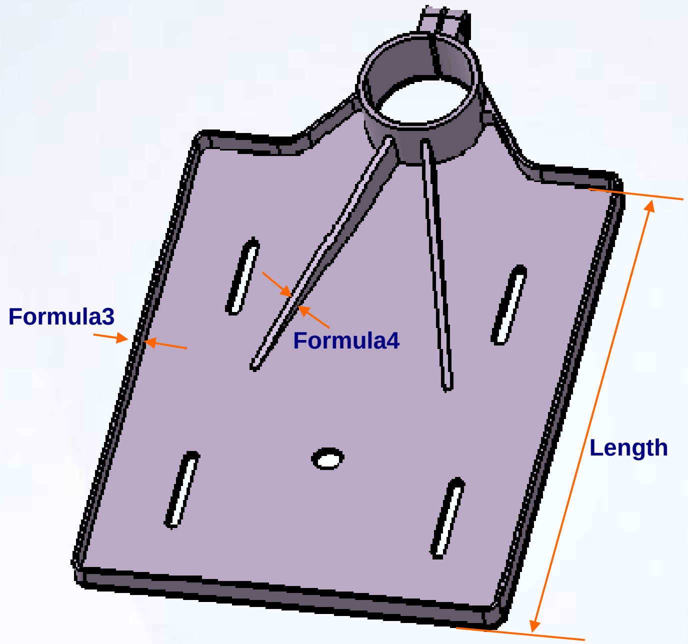




Do It Yourself: Finalizing Design Intent (3/5)

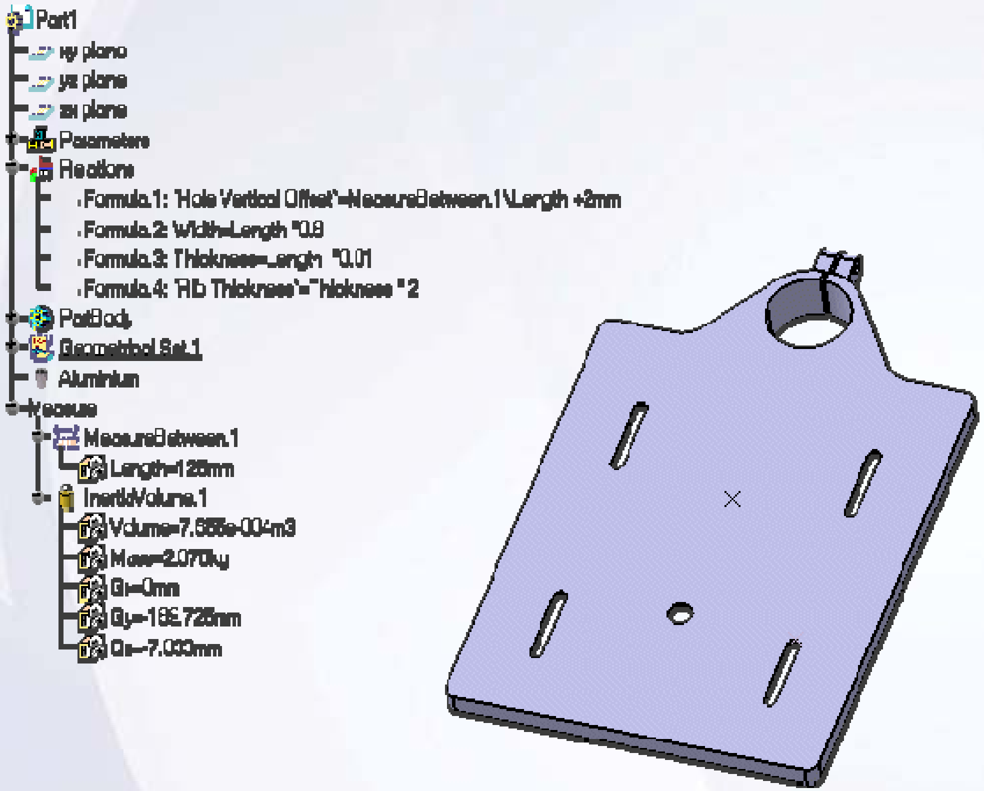
4. **Rename the parameters and create formulas to maintain the required design intent. The completed model is shown.**





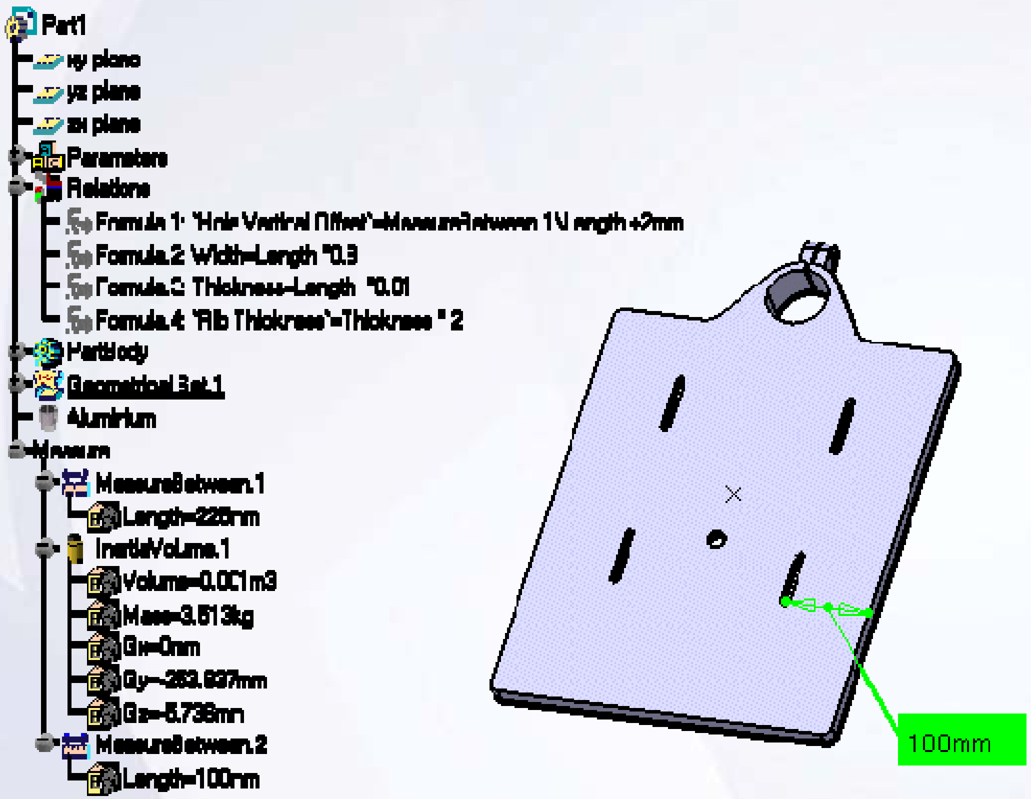
Do It Yourself: Finalizing Design Intent (4/5)

5. Calculate the volume and mass of the model. Create an associative point at the center of gravity.



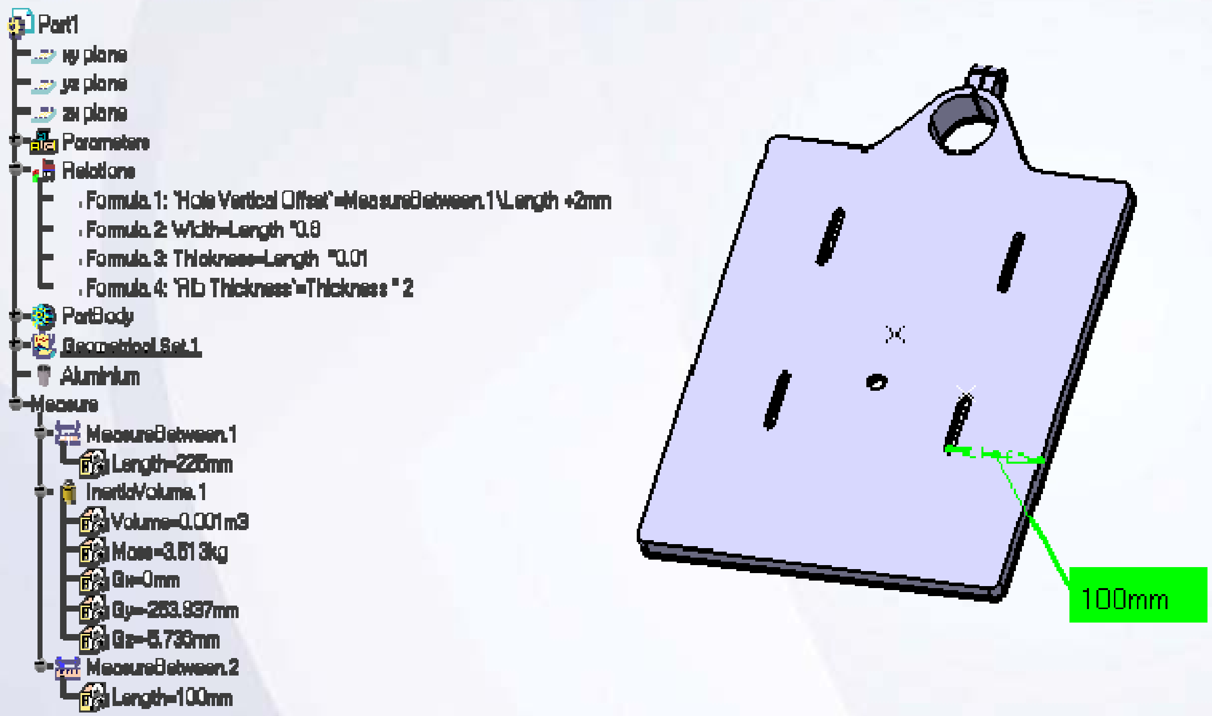
Do It Yourself: Finalizing Design Intent (5/5)

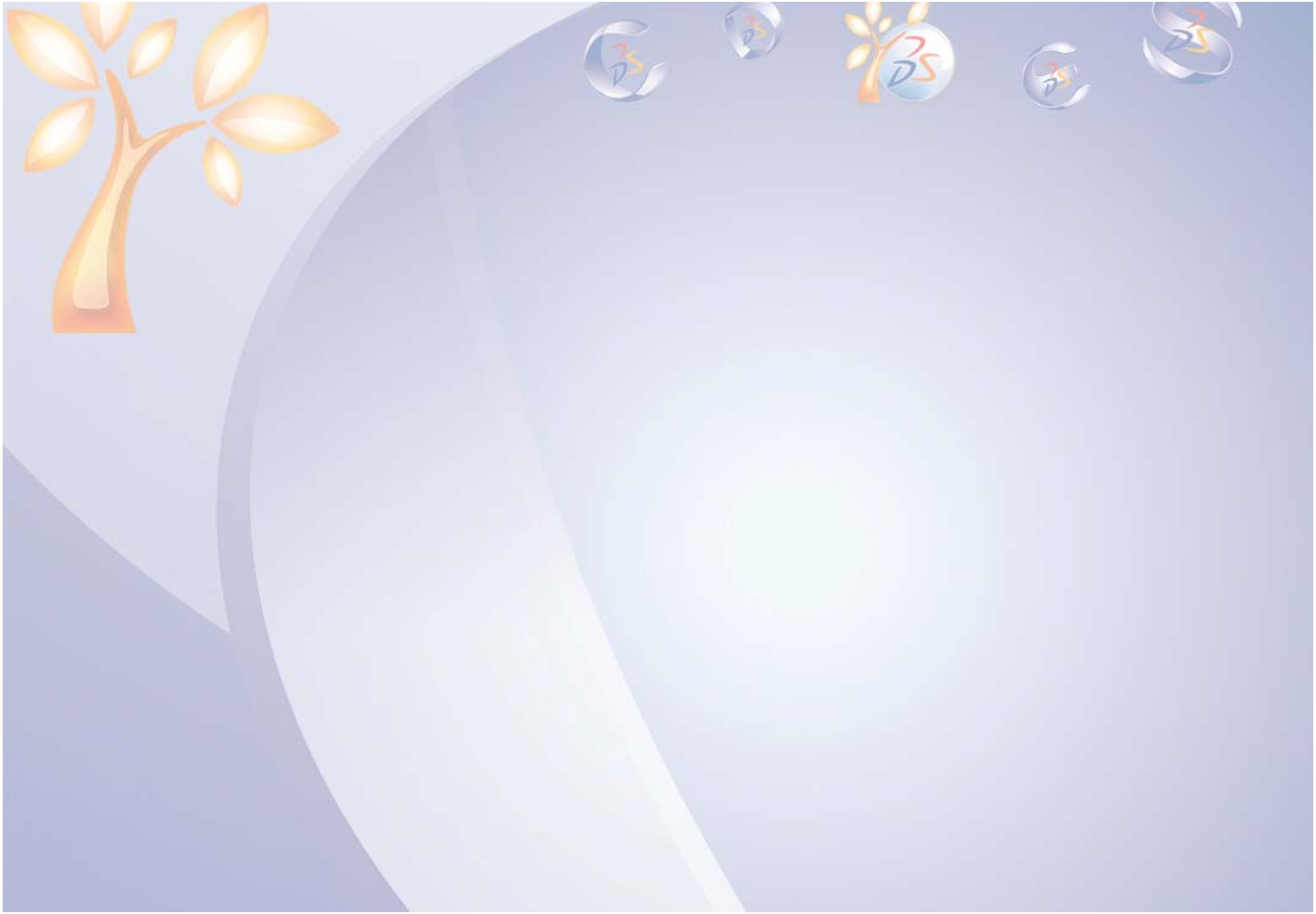
6. **Modify the length of the model to [500mm]. Calculate the distance between the center oblong hole and the side wall.**



Case Study: Finalizing Design Intent Recap

- ✓ **Apply material to the model**
- ✓ **Calculate mass properties**
- ✓ **Create formulas**
- ✓ **Take measurements**





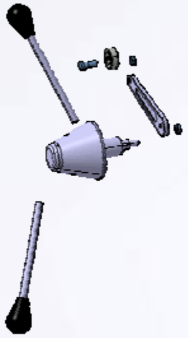
Assembly Design

8

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Create a New CATProduct
- ✓ Assemble in the Base Component
- ✓ Manipulate the Position of the Component
- ✓ Assemble & Fully Constrain Components
- ✓ Save the Assembly



4 Hours

Case Study

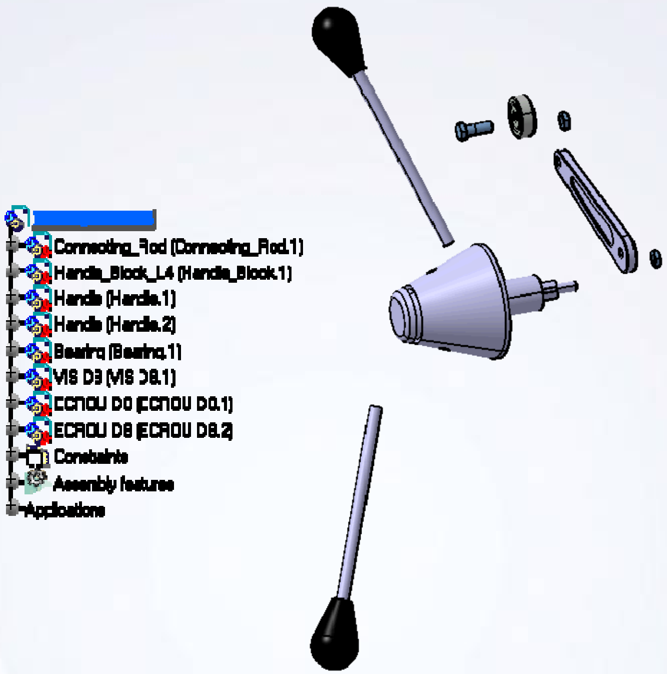
The case study for this lesson is the handle mechanism sub-assembly.

Design Intent

✓ Fix the first component.
- Although fixing the first component is not essential in CATIA, it is considered as a good practice.

✓ Fully constrain all components.
- Fully constraining the components is a good way to avoid undesired changes.

✓ Reuse existing components.
- Reusing the existing components decreases the design time.

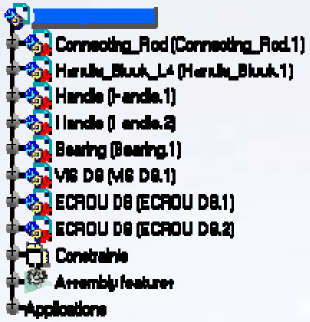


Stages in the Process

1. Create a New CATProduct
2. Assemble in the Base Component
3. Manipulate the Position of the Component
4. Assemble & Fully Constrain Components
5. Save the Assembly

Create a New CATProduct

A product is an 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

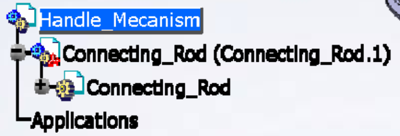
You can add a component to an assembly in one of the three ways:

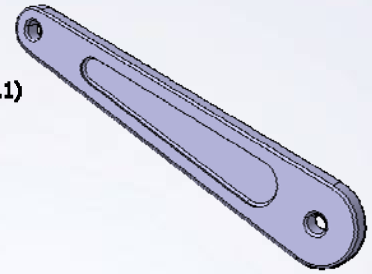
A. **Contextual menu**: Right-click the assembly that will receive the component and use the contextual menu to insert the component.

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.



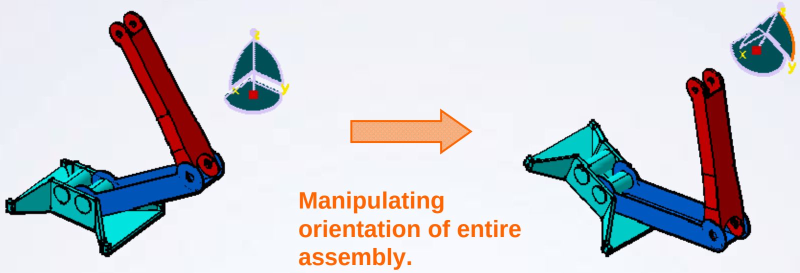


Manipulate the Position of the Component

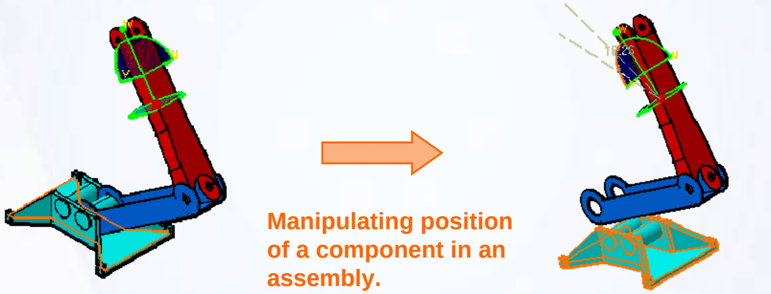
After inserting the base component, it can be left to “float” in space (without constraints), but it is a good practice to fix this component. Fixing it will serve as a reference for placing all other components that are assembled later. Once you have fixed a component, you can still temporarily manipulate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



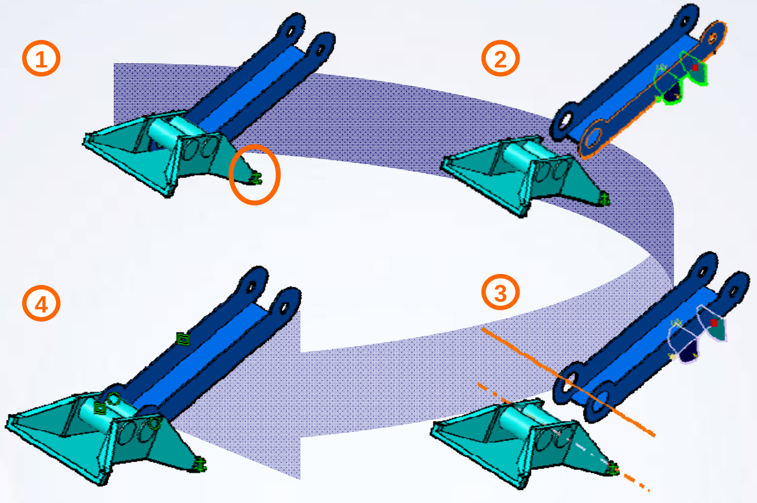
Manipulating orientation of entire assembly.



Manipulating position of a component in an assembly.

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.

Save the Assembly

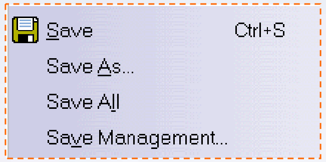
The following four options (found in the File menu) can be used to save the assembly and child documents:

A. The **Save** option saves the active component children of the active document.

B. The **Save As** option is similar to Save, except that you can also specify a name and a folder for the active document.

C. The **Save All** option saves all the open documents that have been modified since the last save.

D. The **Save Management** option prompts you to save all the open documents and children of these documents; you can also control their names and locations.



Assembly Design Tools

Product Structure tools

1 **Add components:** adds existing components such as parts, subassemblies etc. to a Product

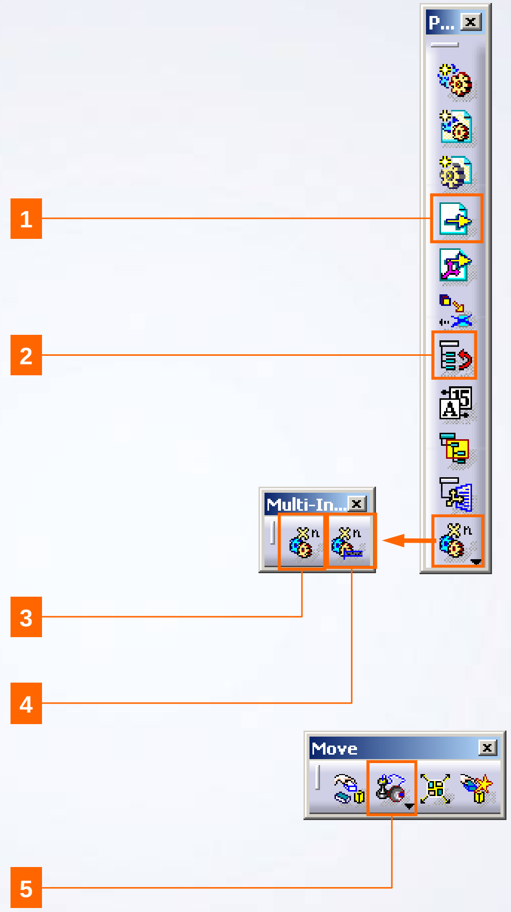
2 **Graph tree Reordering:** reorders components in the specification tree to meet your needs

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

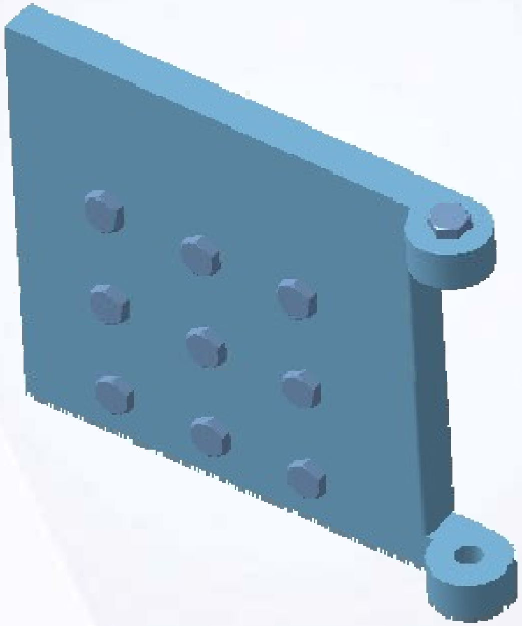
7 **Reuse Pattern:** duplicates a component reusing a pattern created in the Part Design workbench

Exercise: Reuse Components

Recap Exercise

20 min

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



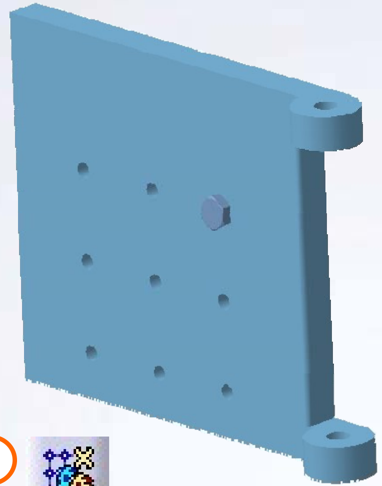
By the end of this exercise you will be able to:

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

Do it Yourself (1/3)

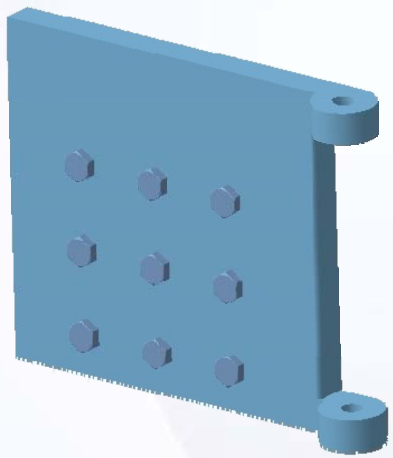
1. **Load Exercise10B.CATProduct.**

1



2. **Instantiate VISD8.CATPart using Reuse Pattern.**
   - Using Rectangular pattern defined in the **Engine_support_pattern.catpart**, instantiate **VISD8.CATPart**.

2

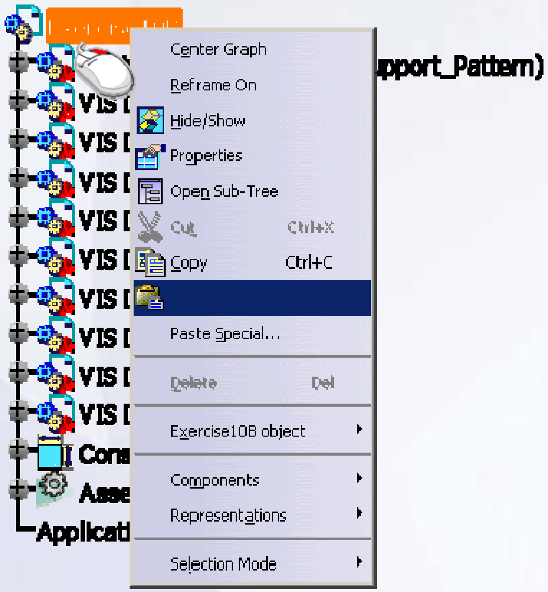


Do it Yourself (2/3)

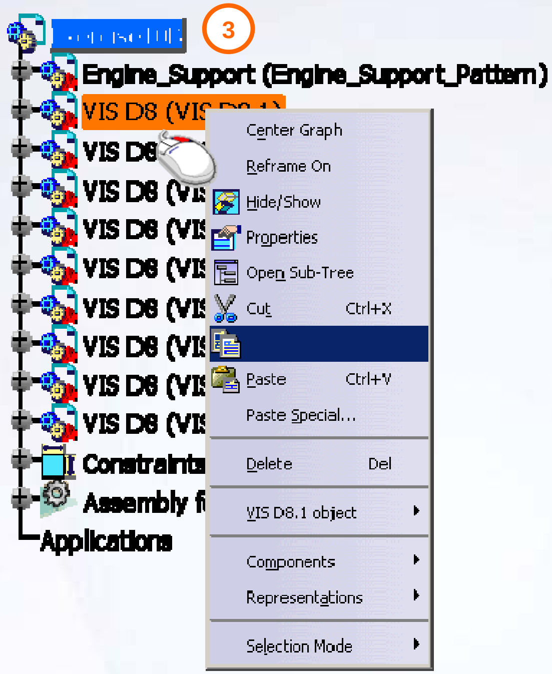
3. Create instance of VISD8.CATPart.

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

3



3

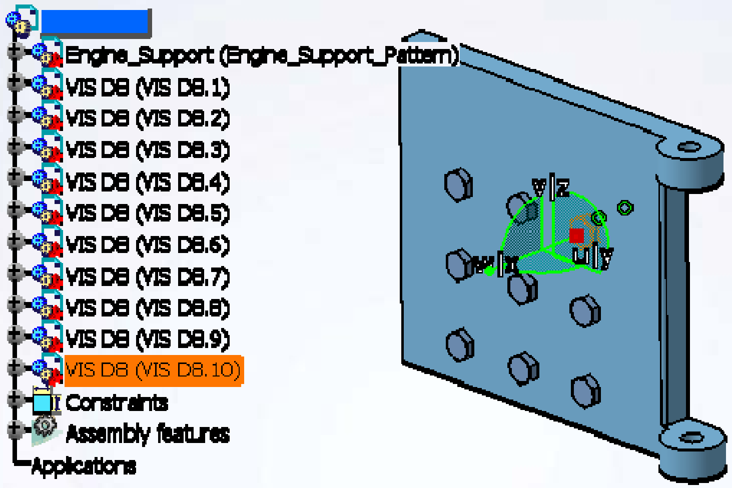


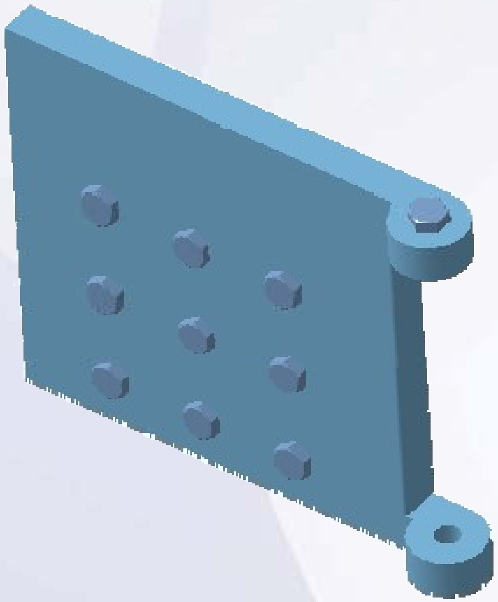
Do it Yourself (3/3)

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.**

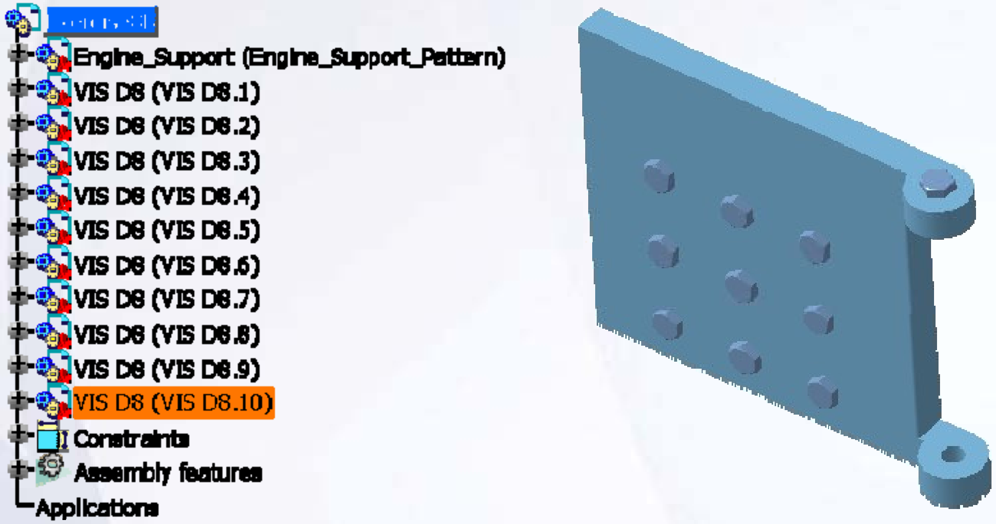
4





Exercise Recap: Reuse Components

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



Exercise: Component Positioning

Recap Exercise

10 min

In this exercise you will use the tools used in the previous exercises to create a new assembly.

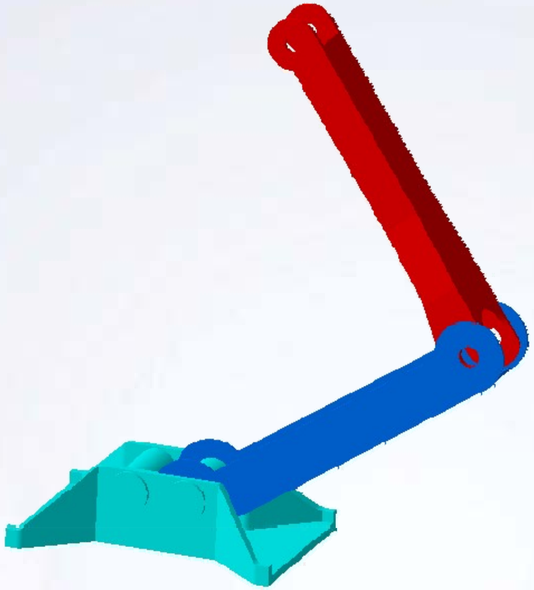
By the end of this exercise you will be able to:

- **Create a new product**
- **Add a component into the assembly**
- **Position the component using manual methods**

Do it Yourself

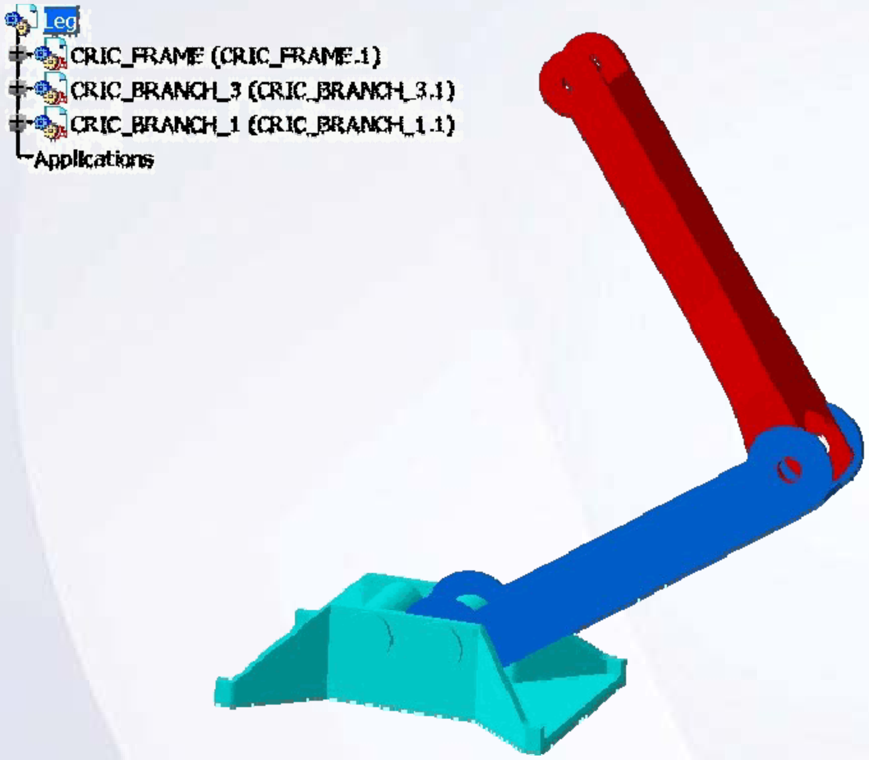
1. **Insert the cric_frame.catpart first and fix it in space.**

2. **Insert cric_branch_3.catpart and cric_branch_1.catpart, then move them into the positions shown.**



Exercise Recap: Component Positioning

- ✓ Create a new product
- ✓ Add a component into the assembly
- ✓ Position the component using manual methods

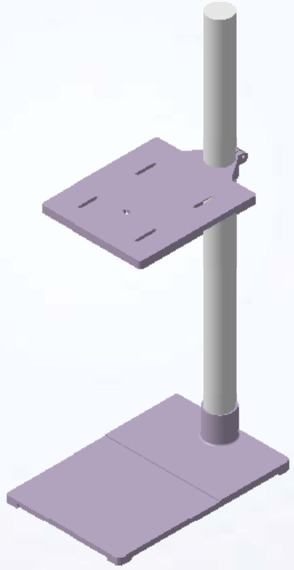


Exercise: Constraints Creation

Recap Exercise

20 min

In this exercise you will continue to practice assembling and constraining components while building an assembly. High-level instructions for this exercise are provided.



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

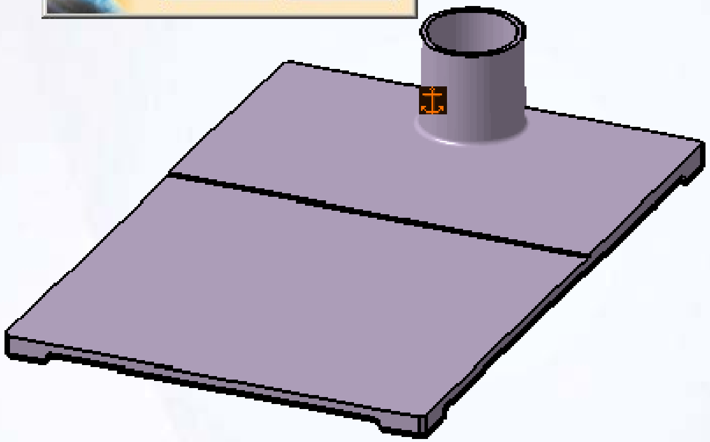
- Create a Contact constraint
- Create an Offset constraint
- Create a Coincidence constraint
- Create a Parallelism constraint

Do it Yourself (1/4)

1. **Create new product.**

2. **Save the file and specify [Stand] as the name.**

3. **Insert Base.CATPart and apply a Fix constraint.**

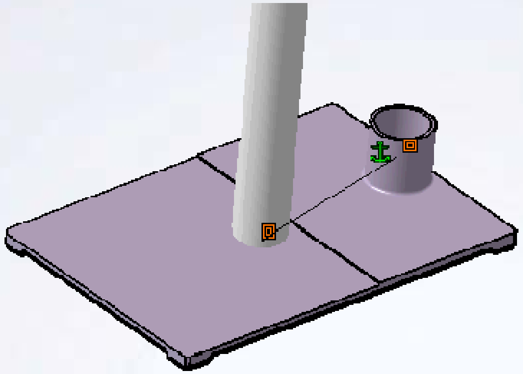


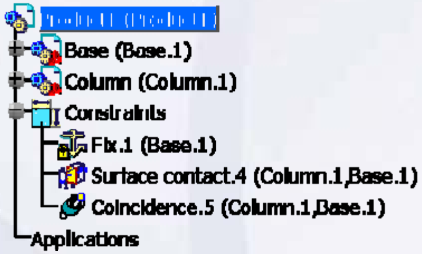
Do it Yourself (2/4)

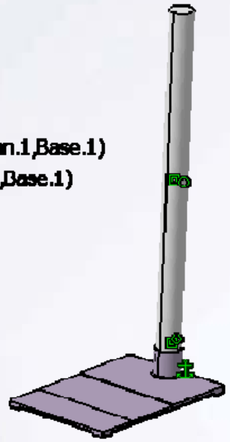
4. **Insert Column.CATPart and apply a Contact constraint.**

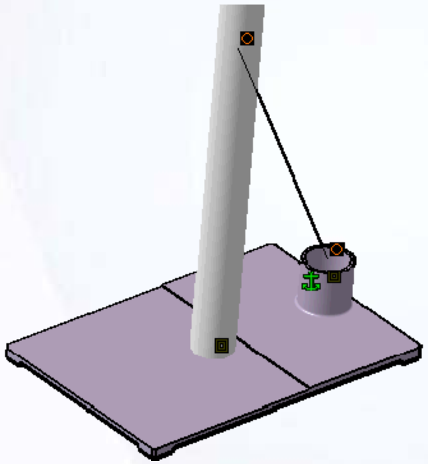
5. **Apply a Coincident constraint.**

6. **Update the assembly and view the structure and constraints.**





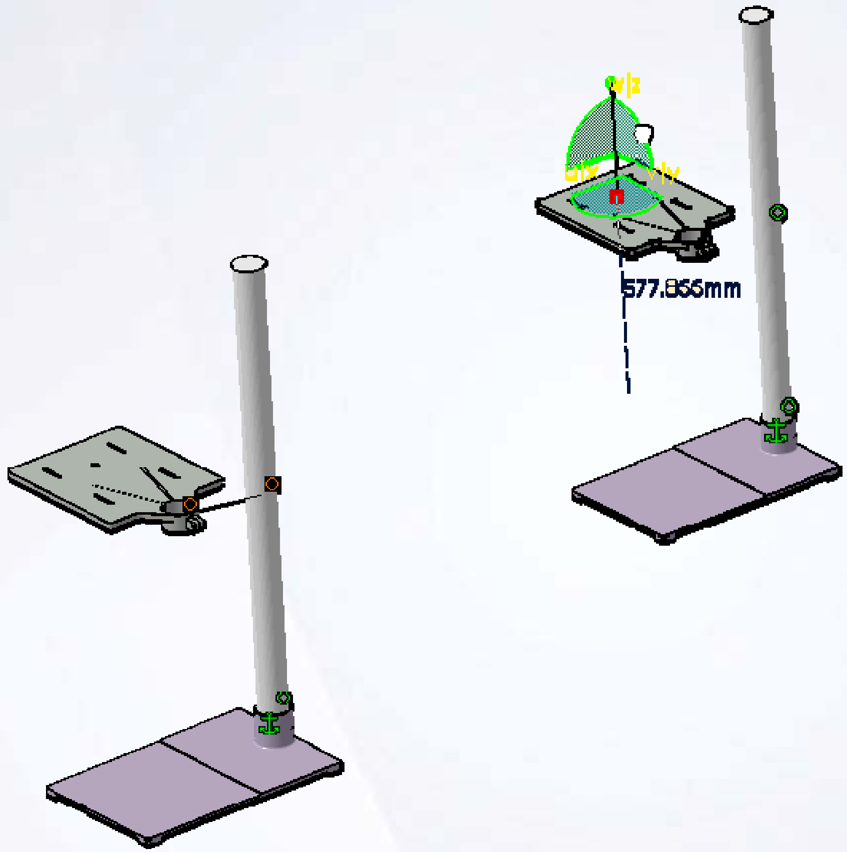




Do it Yourself (3/4)

7. **Insert Table.CATPart and position it with the compass as shown.**

8. **Apply a Coincident constraint.**

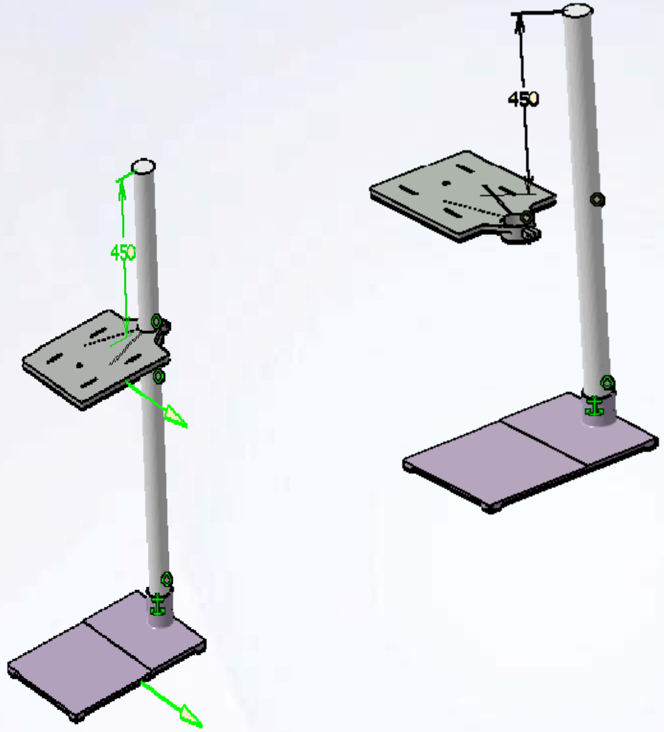


Do it Yourself (4/4)

9. Apply an offset constraint. Specify [450] as the offset value.

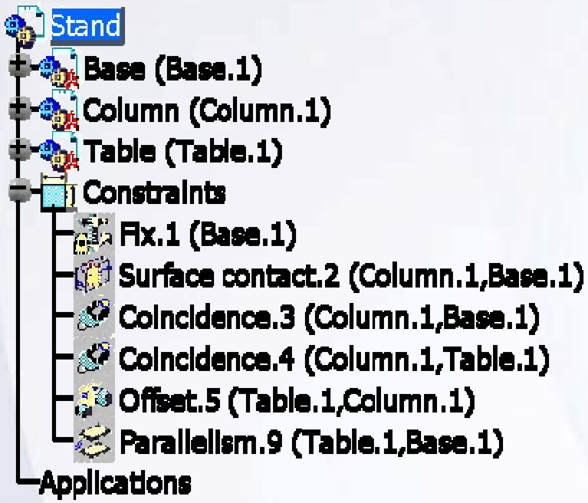
10. Apply a parallelism constraint.

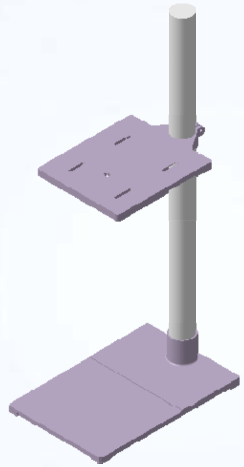
11. Save the product file.



Exercise Recap: Constraints Creation

- ✓ Create a Contact constraint
- ✓ Create an Offset constraint
- ✓ Create a Coincidence constraint
- ✓ Create a Parallelism constraint





Exercise: Degrees of Freedom

Conf. Dep.

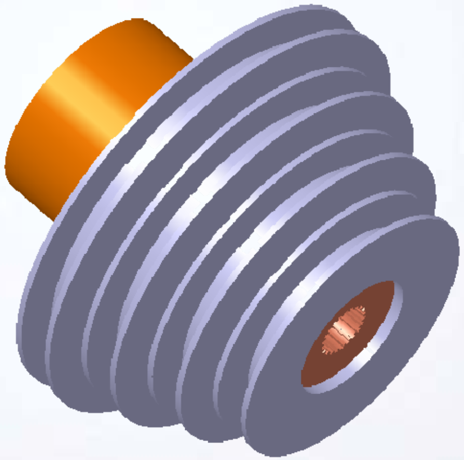
Recap Exercise

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:

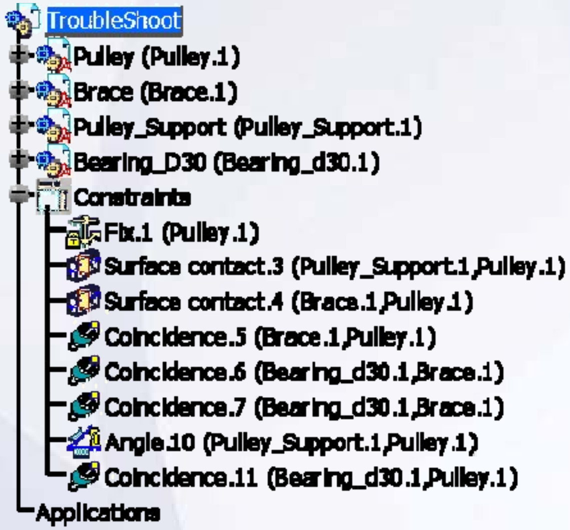
- **Determine the degrees of freedom of a component**
- **Fully constrain a component**

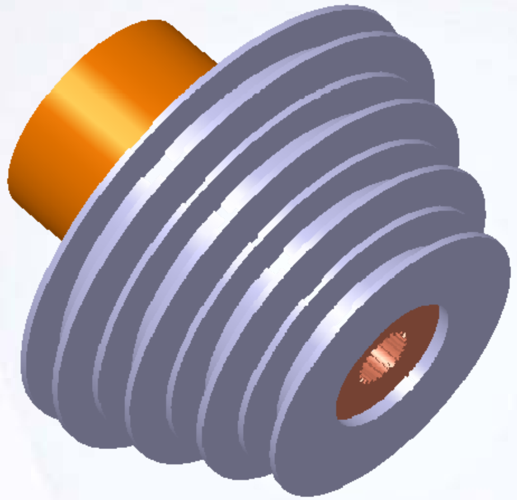


Do it Yourself

Conf. Dep.

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.

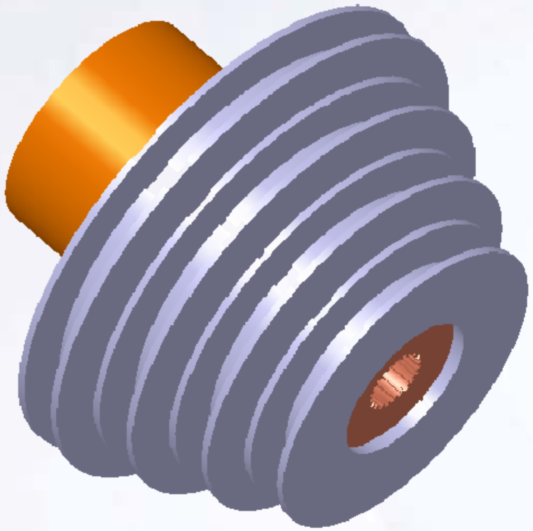




Exercise Recap: Degrees of Freedom

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

Conf. Dep.



Case Study: Assembly Design

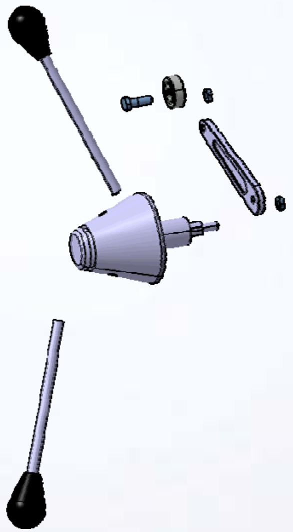
Recap Exercise

20 min

You will practice what you have learned by completing the case study model using only a detailed drawing as guidance.

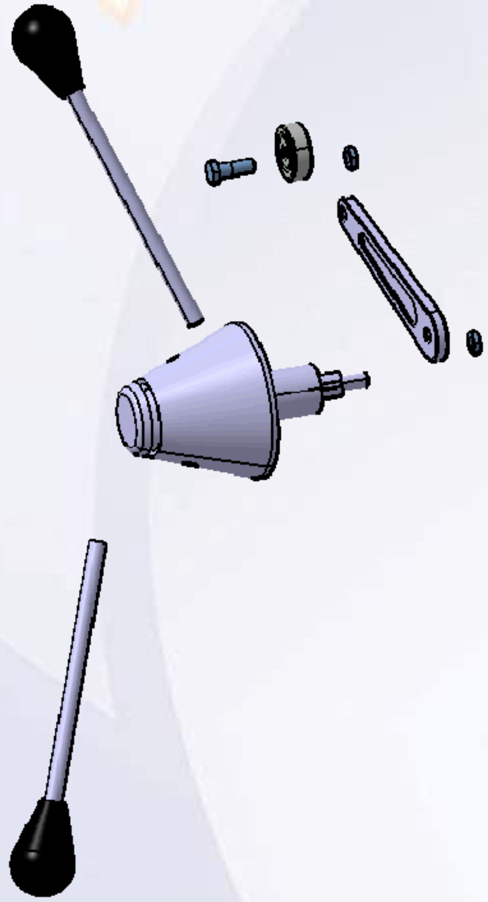
In this exercise you will create the case study model. Recall the design intent of this model:

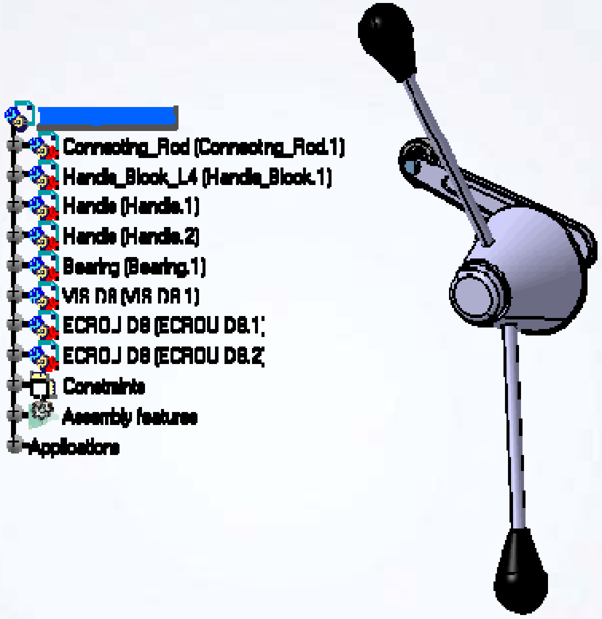
- ✓ First component is fixed
- ✓ All the components are fully-constrained
- ✓ Duplication tools are required



Using the techniques discussed so far, create the model without detailed instructions.

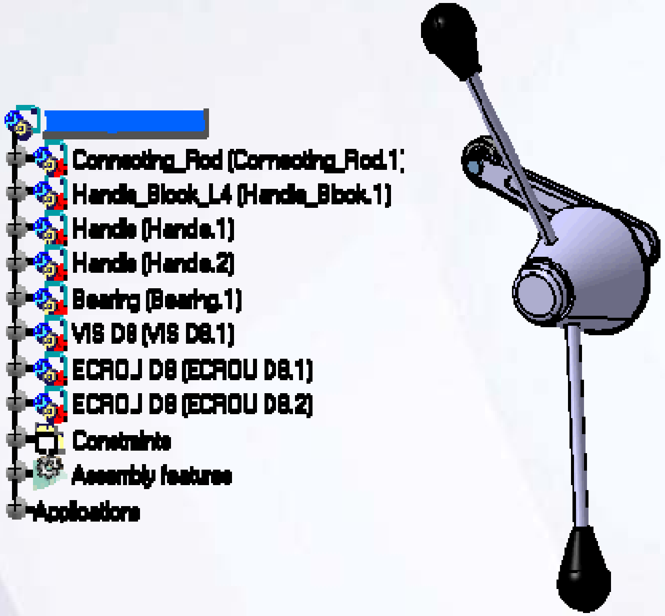
Do It Yourself: Drawing of the Handle Mechanism

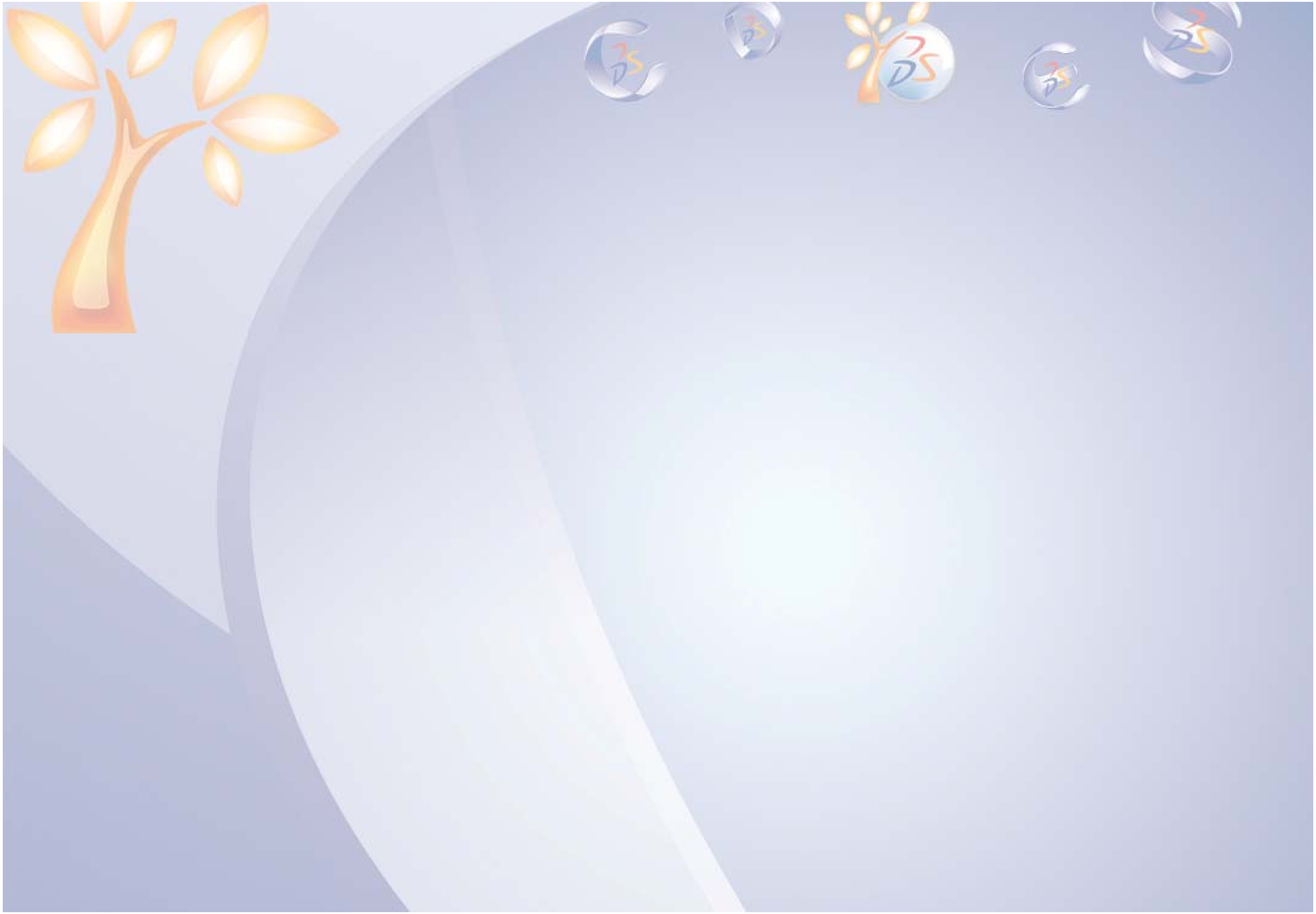




Case Study: Handle Mechanism Recap

- ✓ First component is fixed
- ✓ All the components are fully-constrained
- ✓ Duplication tools are required



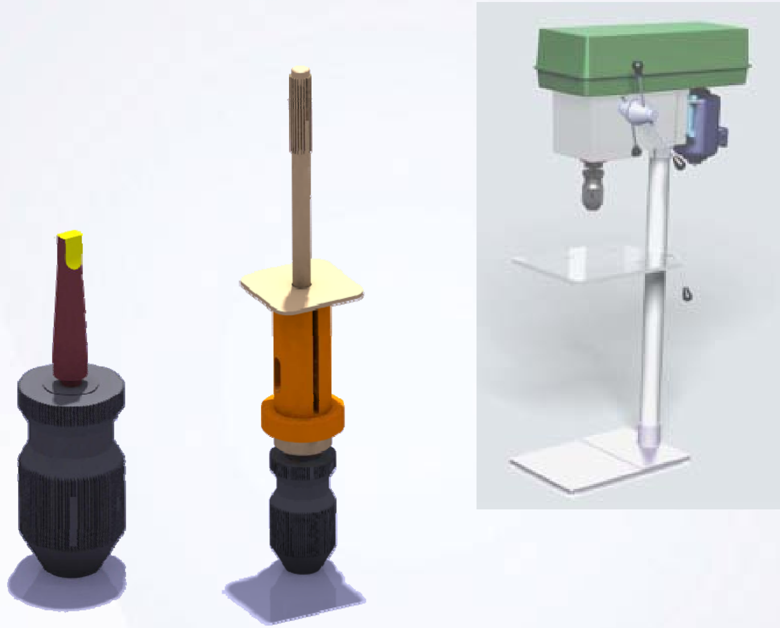


Designing in Context 9

Learning Objectives

Upon completion of this lesson you will be able to:

- ✓ Open an Existing Assembly
- ✓ Insert a New Model
- ✓ Create a Sketch in Context
- ✓ Create Assembly-Level Features



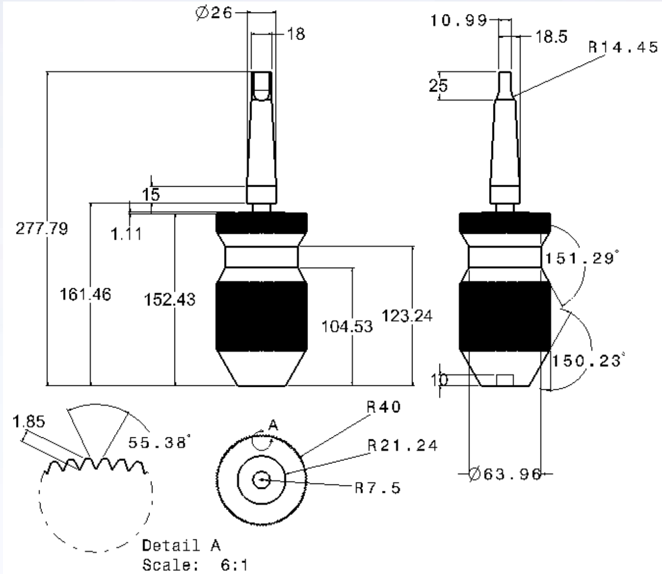
4 Hours

Case Study

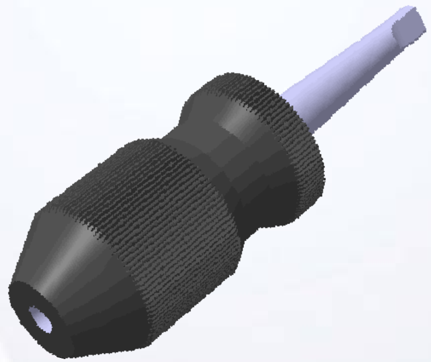
The case study for this lesson is the Chuck part used in the Drill Support assembly.

Design Intent

- ✓ The model must be created within an assembly.
- ✓ The base feature sketch support should refer a reference plane from another model.
- ✓ The axis of revolution for the shaft should be coincident with the axis of the base component.
- ✓ The chuck geometry will be used to define the volume within the canella_axis.



Stages in the Process

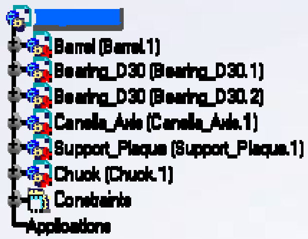


1. Open an Existing Assembly
2. Insert a New Model
3. Create a Sketch in Context
4. Create Assembly-Level Features

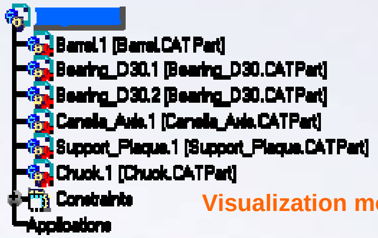
Open an Existing Assembly

Assemblies can contain components that reference individual parts and assembly files. If referenced files are moved from their original locations, CATIA may not be able to locate them when the top-level assembly is retrieved. CATIA prompts you to specify the new locations of the missing files. Using the Desk command, you can locate these files.

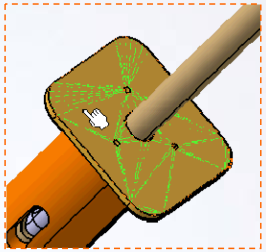
By default, the assemblies and their components are loaded into a CATIA session in Design mode. In this mode, the part definition of all the components are loaded into memory. The loading time may be large. To improve performance, assemblies can be loaded in the visualization mode, where CGR representations of the geometry are loaded instead of the actual geometry.



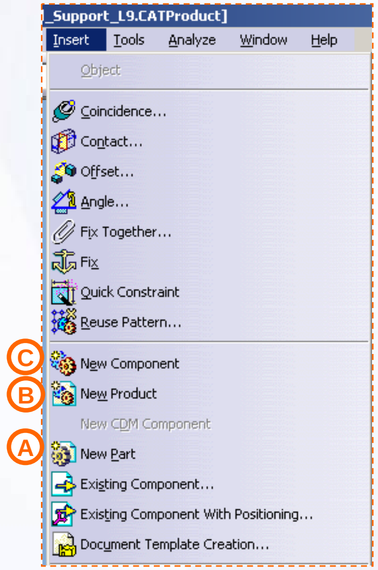
Design mode



Visualization mode



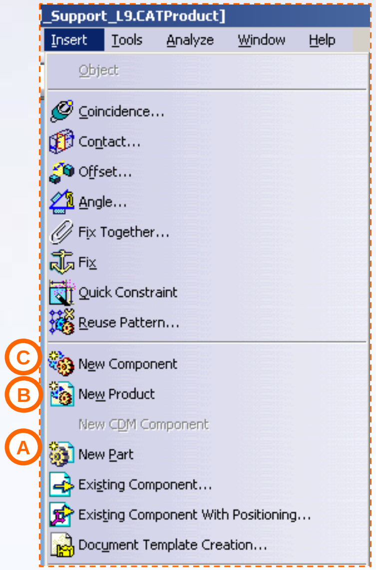
Product opened in Visualization mode



Insert a New Model

As mentioned in the previous lessons, CATIA enables you to insert previously created components into an assembly. New models can also be created directly in an assembly. You can create the following types of models:

A. **Part:** Create a new part file that exists as a separate file.
B. **Product:** Create a new product or sub-assembly that exists as a separate file.
C. **Component:** Create a new product that exists only in the top-level assembly.



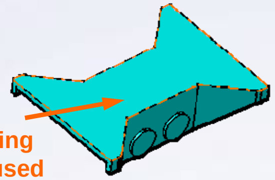
Create a Sketch in Context

While working in context of an assembly you can make use of the existing components in one of the following ways:

A. Sketching on a face of a component
B. Projecting / intersecting 3D elements onto the sketch plane
C. Defining sketch constraints using other components
D. Limiting features up to other components

As a general rule when designing in context, the components created within the context of an assembly is unique to the assembly and should not be inserted into another assembly nor moved to another position. However, if the components have to be moved as per the design requirement, you must break the external references. This is done by isolating the feature.

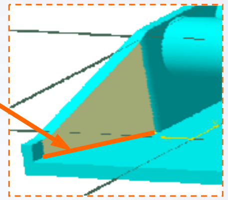
A



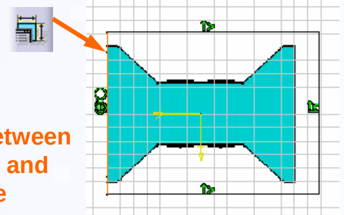
Face of existing component used to create a sketch

B

Edge created using intersection



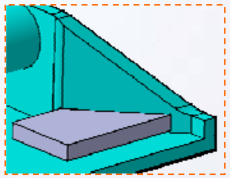
C



Coincident constraint between sketch entity and existing edge

D

Pad limited by existing face

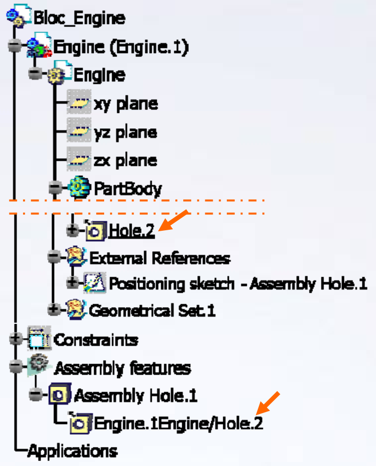


Create Assembly-Level Features

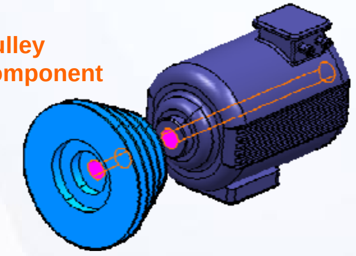
Assembly features are created inside an assembly and these features affect not only a single part but a set of components of the assembly. These features include: split, hole, pocket, add, remove, and symmetry.

Assembly-level split, hole, and pocket features can be created based on the same type of features that exist in the child component. These features cannot be patterned.

Assembly-level hole features appear in the assembly specification tree and the part specification tree in which it is applied. This is because, the hole dimensions are modified at the assembly level, but their position is modified at the part level.



Assembly-level hole features in the assembly specification tree and the part specification tree



Assembly level hole feature affects engine as well as pulley.

Design in Context Tools

Assembly Features

1 **Split:** splits one or more components using the splitting surface in a single operation

2 **Hole:** creates a hole passing through multiple components in a single operation

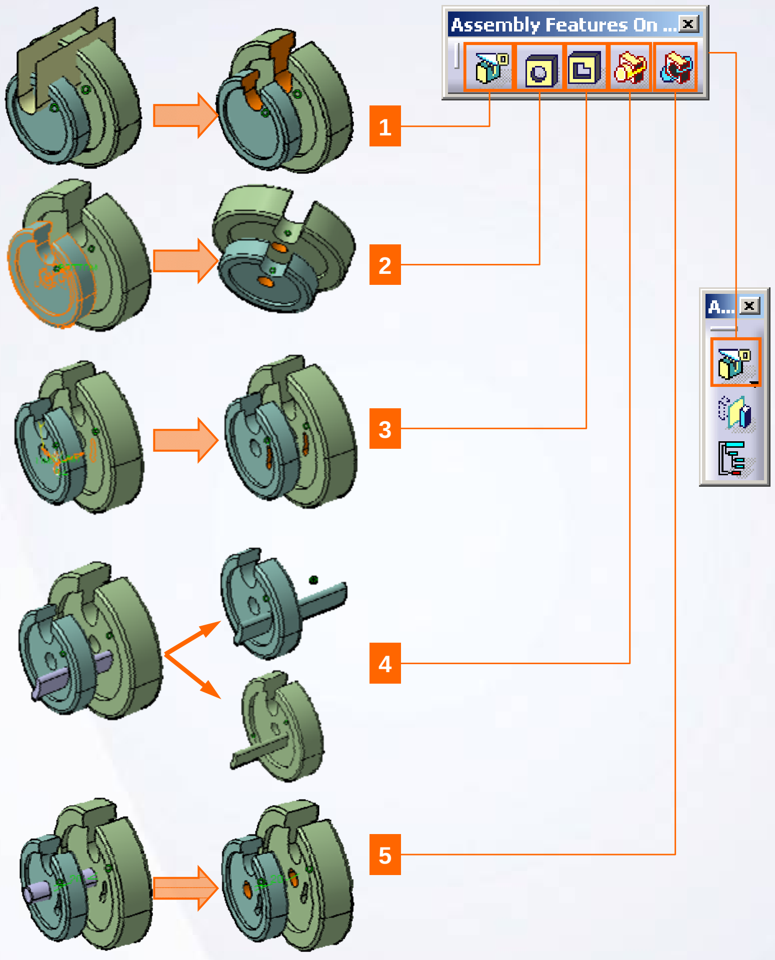
3 **Pocket:** creates a pocket passing through multiple components in a single operation

4 **Add:** adds a part body to multiple components in a single operation

5 **Remove:** removes material from all the affected components using geometry of a part body in a single operation

File Menu

6 **File > Desk:** helps to locate the missing files referenced by an assembly loaded in the session



Exercise: Visualization and Design Modes

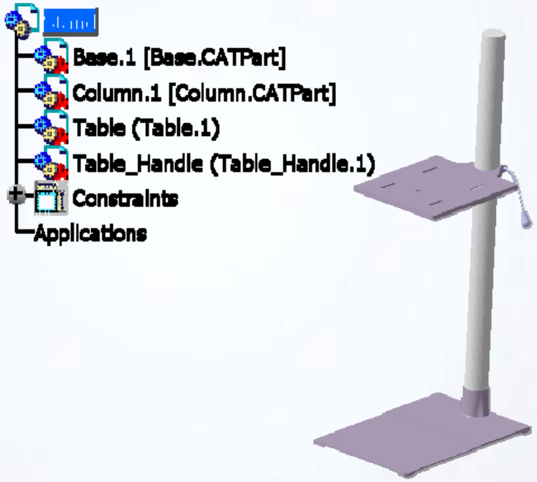
Recap Exercise

25 min

In this exercise you will open an existing assembly and change between the Visualization and Design modes, make modifications to a component, and switch back to Visualization mode. High-level instructions for this exercise are provided.

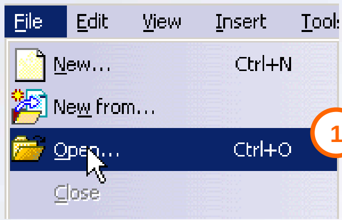
By the end of this exercise you will be able to:

- **Open an existing assembly in Visualization mode**
- **Change between the Visualization and Design modes**
- **Modify a component**



Do it Yourself (1/2)

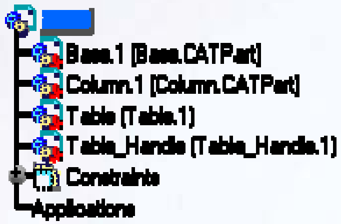
1. **Open the file Ex9B.CATProduct.**
   - Verify that the **Work with the cache system** option is activated for this exercise.



1

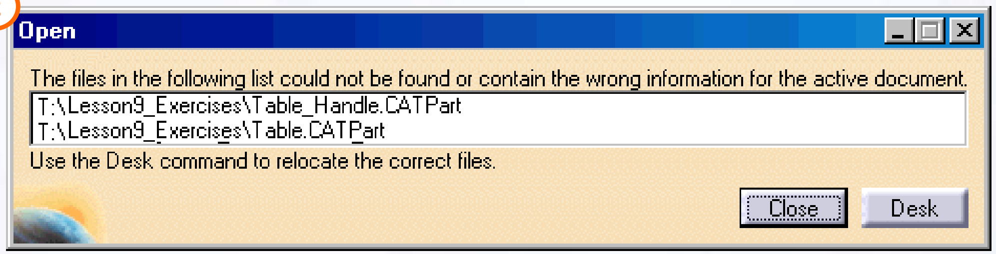
2. **Locate the missing files using the Desk command.**

3. **Switch all the components to Design mode.**



3

2



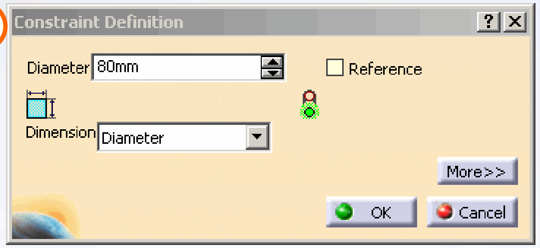
Do it Yourself (2/2)

4. Modify the diameter of the Column from [80mm] to [75mm].

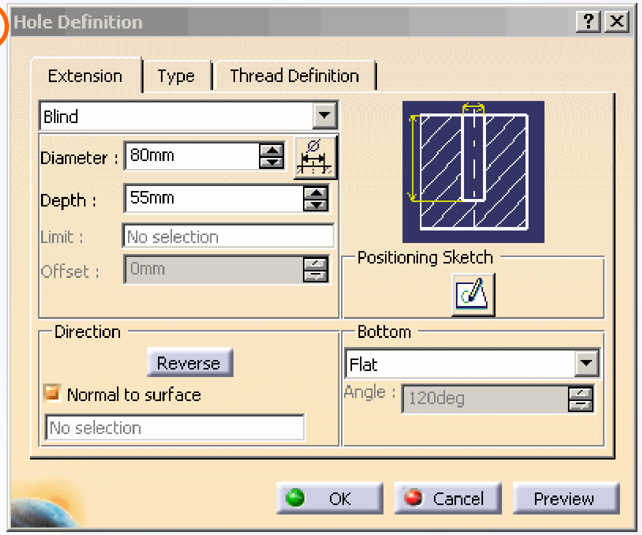
5. Modify the diameter of hole.1 in the Base component from [80mm] to [75mm].

6. Switch the Table.1 and Table_Handle.1 components back to Visualization mode.

4



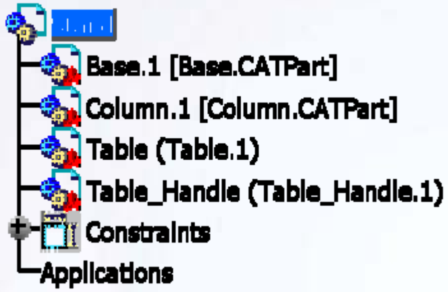
5

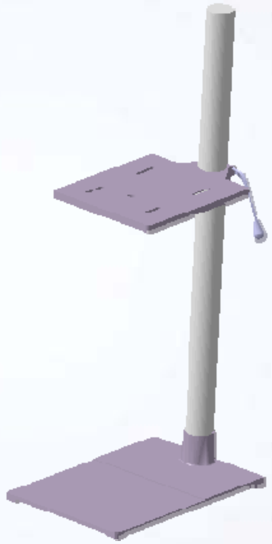


6

Exercise Recap: Visualization and Design Modes

- ✓ Open an existing assembly in Visualization mode
- ✓ Change between the Visualization and Design modes
- ✓ Modify a component





Exercise: Design in Context

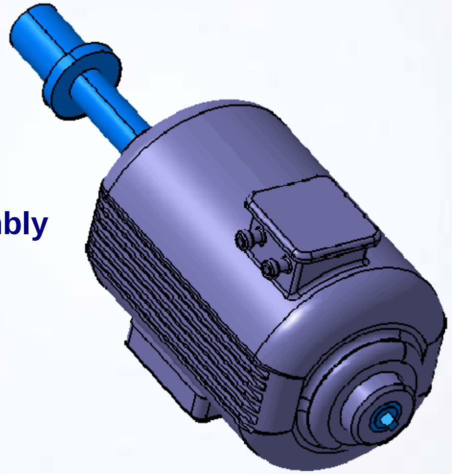
Recap Exercise

20 min

In this exercise, you will construct the Engine Axis part within the Bloc_Engine assembly. High-level instructions for this exercise are provided.

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

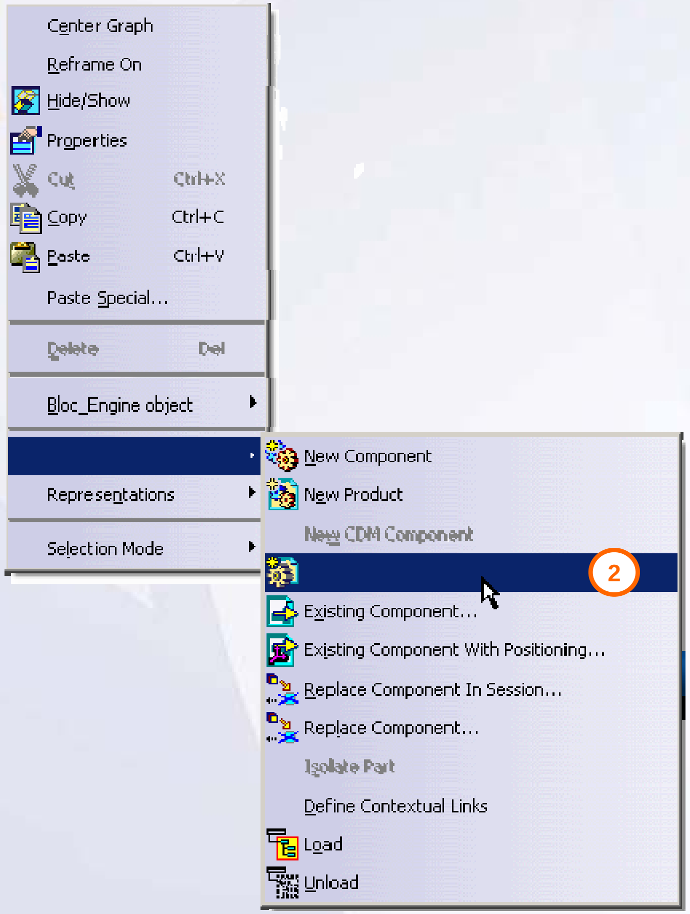
- **Create a new part within an assembly**
- **Create a part geometry referencing other components in the assembly**
- **Modify the geometry of a source and update the target component**

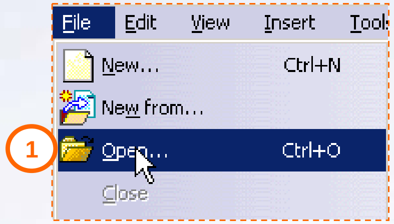


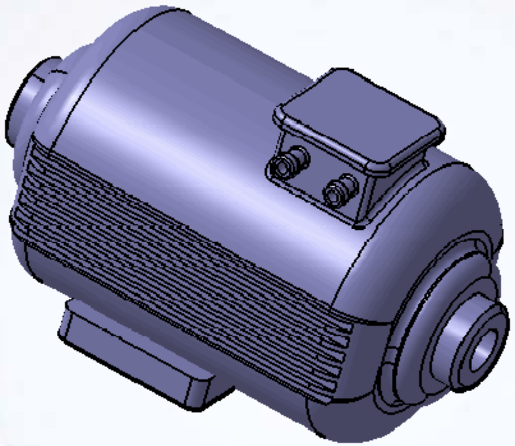
Do it Yourself (1/5)

1. **Open the file Ex9D.CATProduct.**

2. **Insert a new part file and make it the active model.**





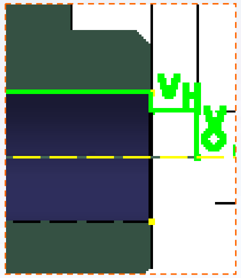


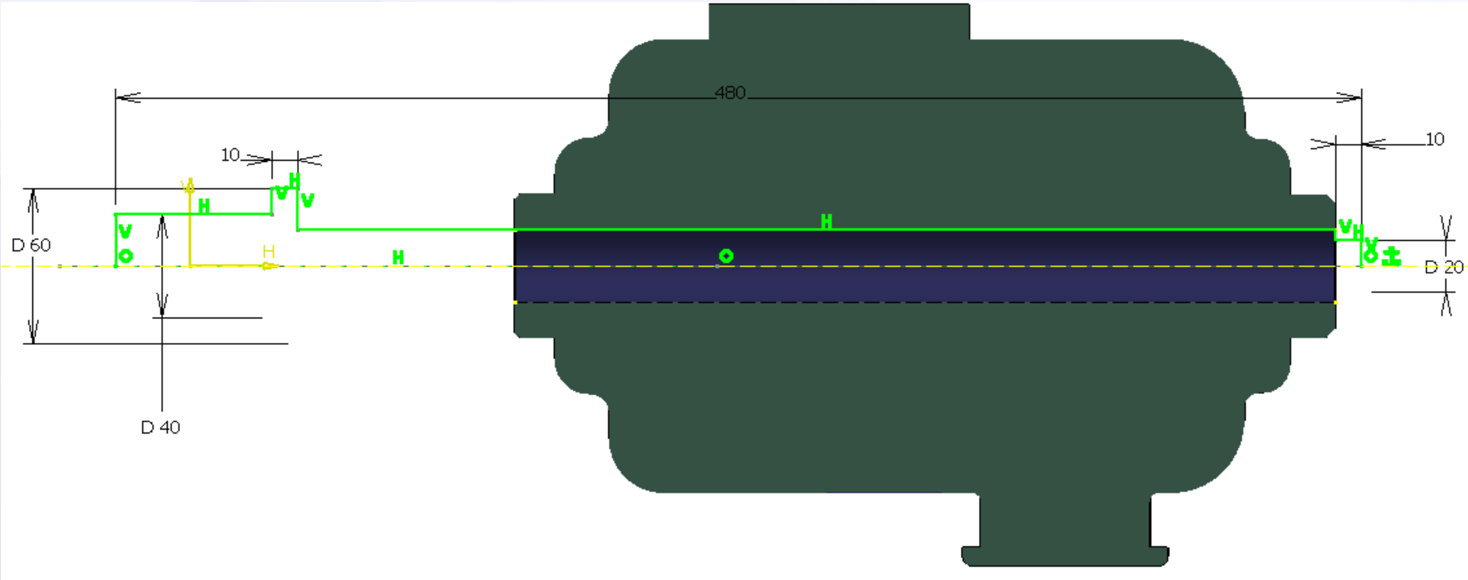
Do it Yourself (2/5)

3. Create a shaft inside the new part with the following sketch.

- Create the sketch on the zx plane that exists in the engine part.
- Use Intersect 3D elements to create an external reference between the surface of the hole and the profile.
- Use Project 3D elements for the creation of the axis and the vertical side of the profile.
- Inside the Sketcher workbench, select the **Cut Part by Sketch Plane** icon from the Visualization toolbar, to better view the part.

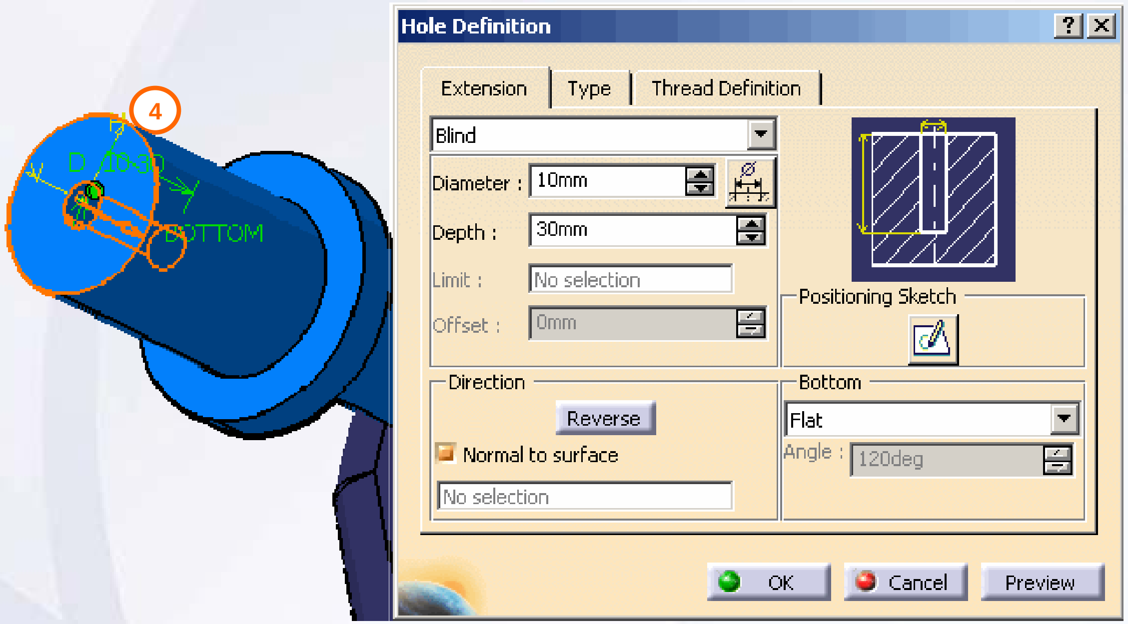
3





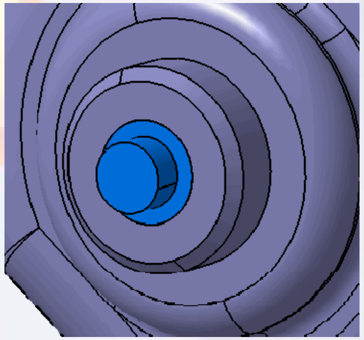
Do it Yourself (3/5)

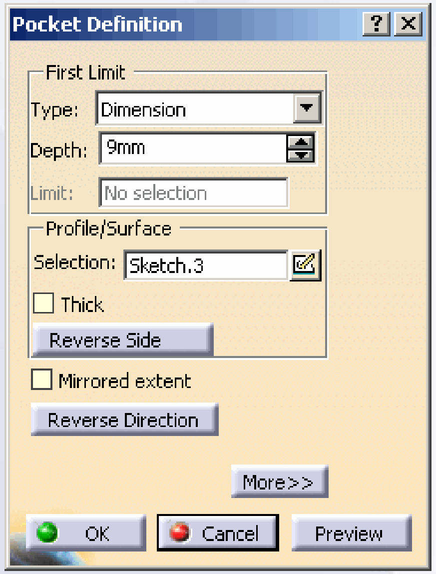
4. **Create a hole using the criteria shown.**

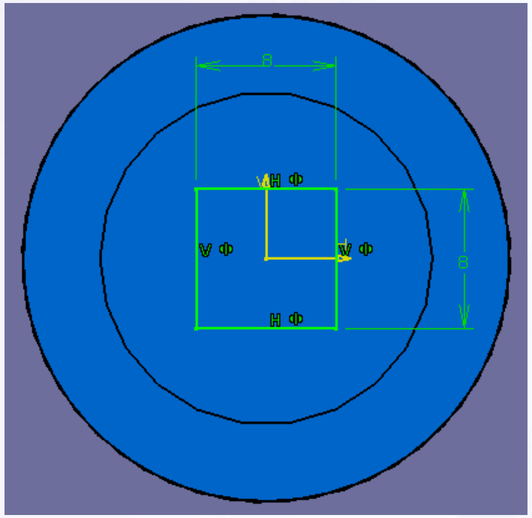


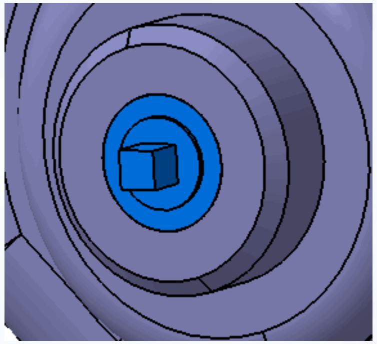
Do it Yourself (4/5)

5. **Create a pocket as shown.**





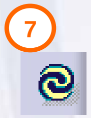


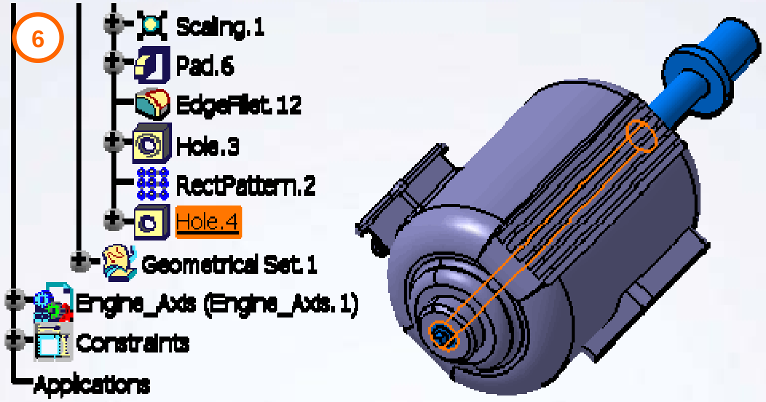


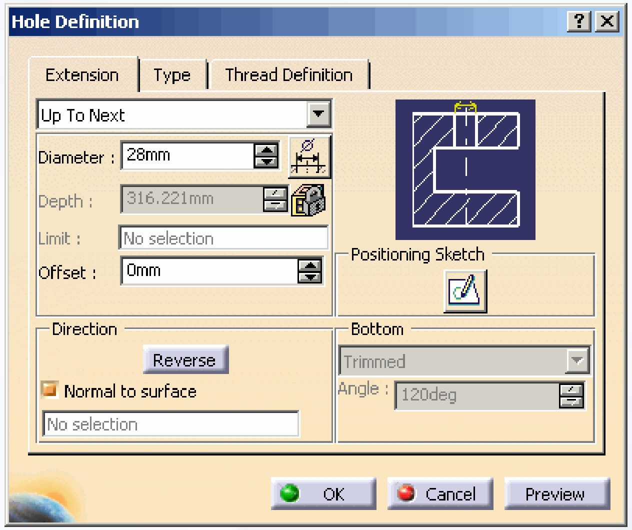
Do it Yourself (5/5)

6. Activate the Engine part and modify the hole.4 diameter from 28mm to 30mm.

7. Update the assembly.

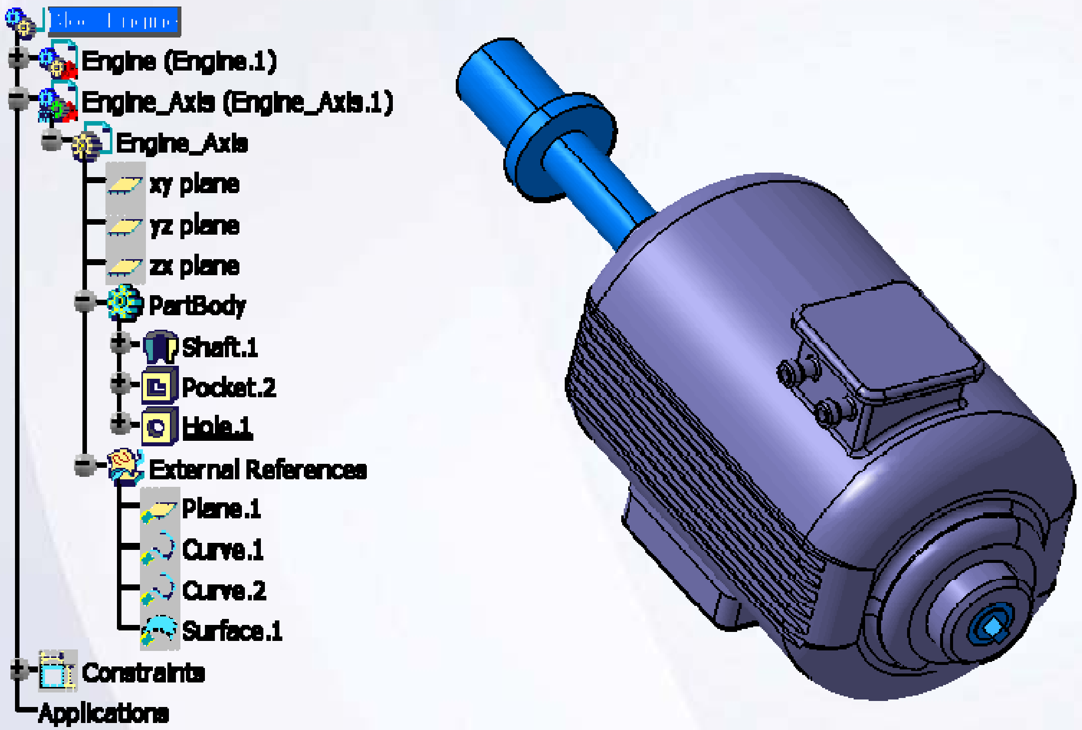






Exercise Recap: Design in Context

- ✓ Create a new part within an assembly
- ✓ Create a part geometry referencing other components in the assembly
- ✓ Modify the geometry of a source and update the target component



Exercise: Design in Context

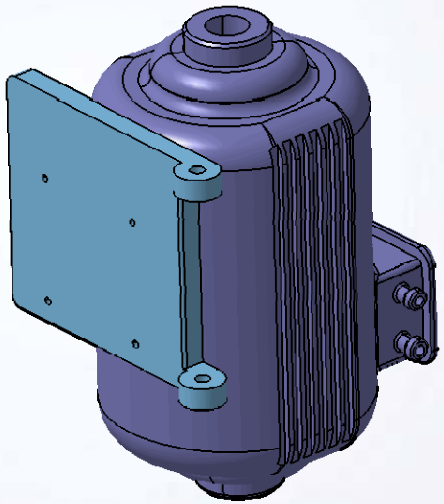
Recap Exercise

20 min

In this exercise you will create a part in the context of the assembly then compare it with an existing model where the components were created separately. You will use the tools you have learned in this lesson to complete the exercise with no detailed instructions.

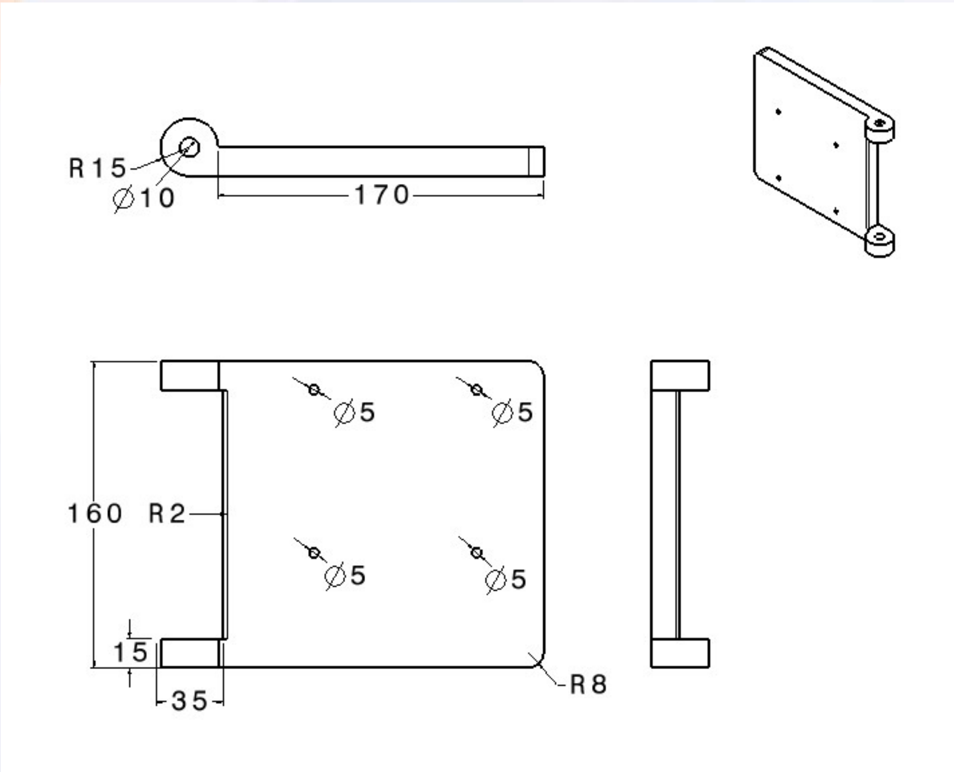
By the end of this exercise you will be able to:

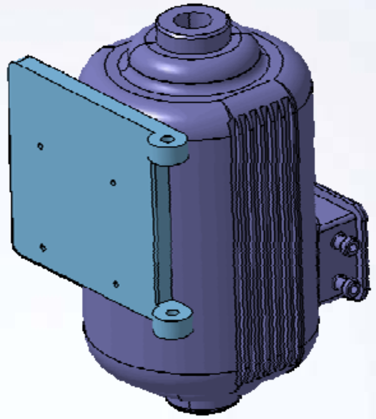
- **Create a part within the context of an assembly**
- **Understand some of the differences between the two design approaches**

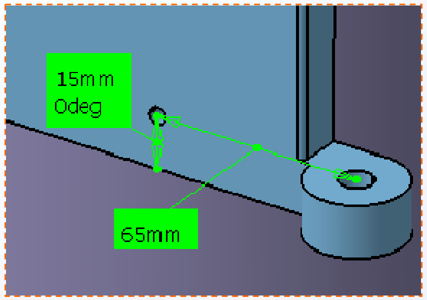


Do it Yourself (1/2)

1. Open Ex9E.CATProduct and create the following engine support in the context of the assembly. Reference the engine component while creating the engine support.







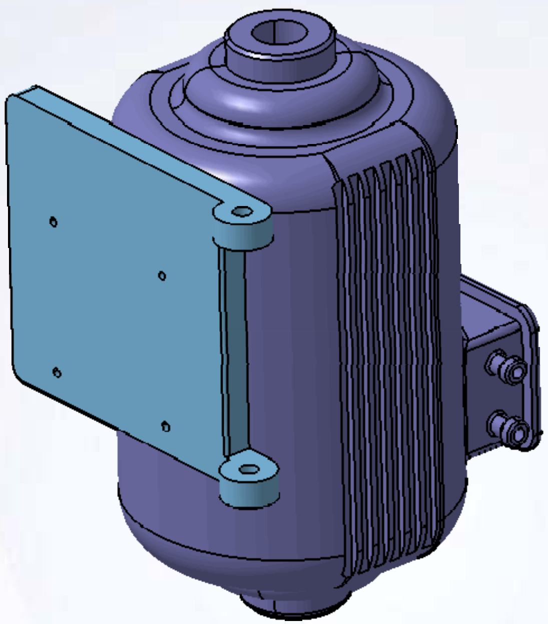
Do it Yourself (2/2)

2. Open Bloc_Engine.CATProduct and compare this model with the part you created. Investigate the following:

A. Can geometry changes made to the engine part cause geometry changes in the engine support?

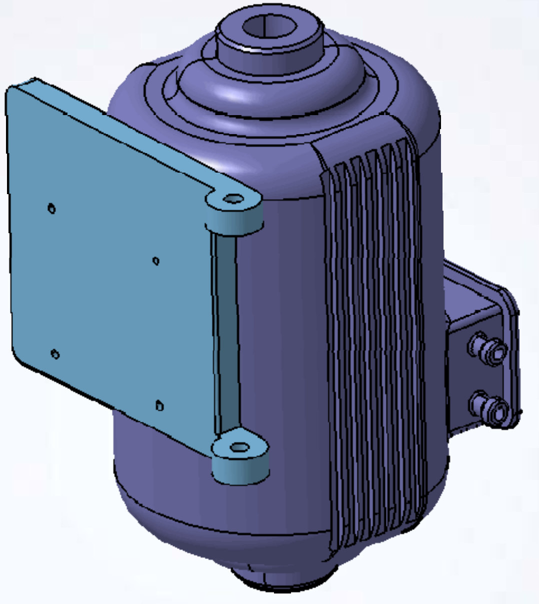
B. Are you able to move the components using the compass in an assembly?

C. What happens if the engine part is deleted?



Exercise Recap: Design in Context

- ✓ Create a part within the context of an assembly
- ✓ Understand some of the differences between the two design approaches



A. Can geometry changes made to the engine part cause geometry changes in the engine support?

With Bloc_Engine.CATProduct, the only references that exist between the components are through assembly constraints, which cannot modify the actual geometry.

B. Are you able to move components using the compass in an assembly?

You are able to move components in both assemblies; however, you will be breaking the assembly constraints in the Bloc_Engine.CATProduct. The components need to be isolated before they can be moved.

C. What happens if the engine part is deleted?

In Ex9E.CATProduct the engine support will lose references that were used to create its geometry. The part will open, but features that reference the engine will not be modifiable.

Case Study: Designing in Context

Recap Exercise

25 min

You will practice what you learned by completing the case study model using only a detailed drawing as guidance.

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

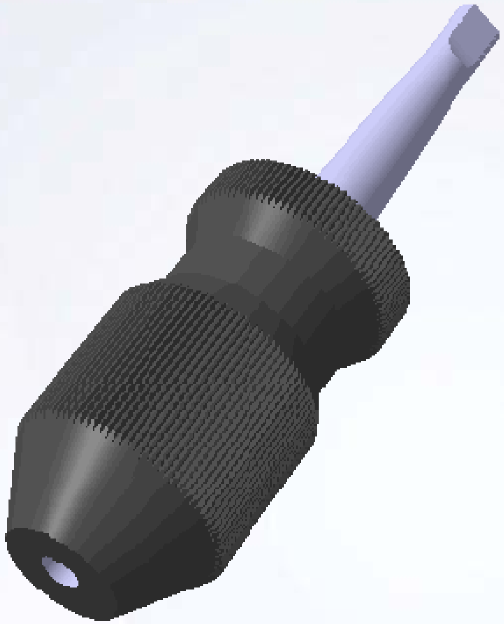
- ✓ The model must be created within the assembly.
- ✓ The base feature sketch support should reference a datum plane from another model.
- ✓ The axis of revolution for the shaft should be coincident with the axis of the base component.
- ✓ The Chuck geometry will be used to define the volume within the Canella_axis.

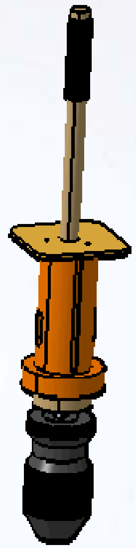
Using the techniques discussed so far, create the model without detailed instructions.

Do It Yourself: Designing in Context (1/2)

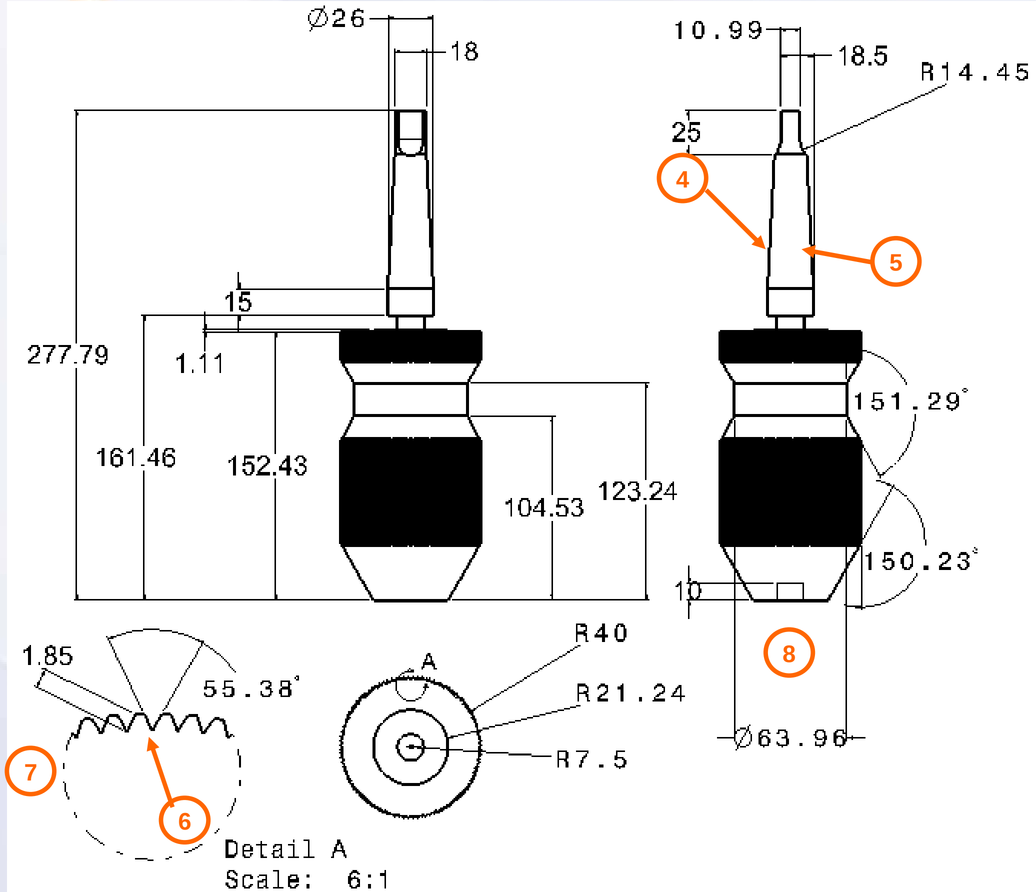
You must complete the following tasks:

1. Open the supplied file **CS-L9.CATProduct** from the case study directory.
2. Insert a new part into the assembly.
3. Create the revolved base feature.
4. Create a pocket to define a cut.
5. Mirror the pocket.
6. Create a pocket to define one of the ridges.
7. Pattern the pocket as a complete crown.
8. Create an assembly-level hole.
9. Perform a Remove Boolean operation at the assembly level to remove the Chuck part from the Canella_axis component.



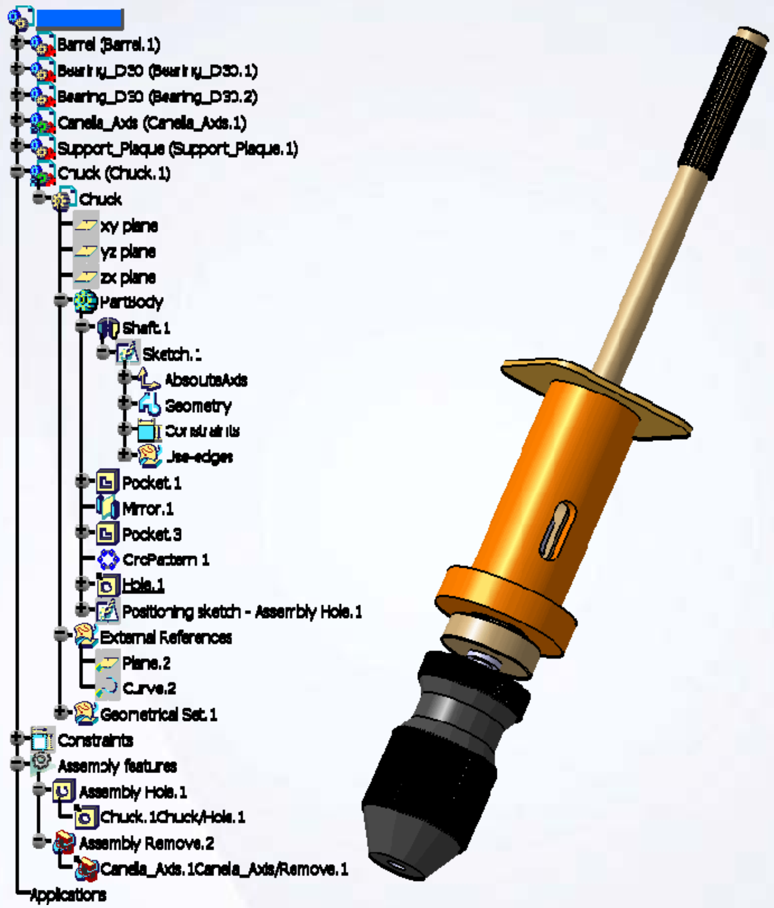


Do It Yourself: Designing in Context (2/2)



Case Study: Chuck Recap

- ✓ Insert a new part into the reference assembly.
- ✓ Create a revolved base feature.
- ✓ Create a pocket to define a cut.
- ✓ Create a pocket.
- ✓ Pattern the pocket.
- ✓ Create an assembly-level hole.
- ✓ Perform a remove at the assembly level that removes the Chuck part from the barrel component.

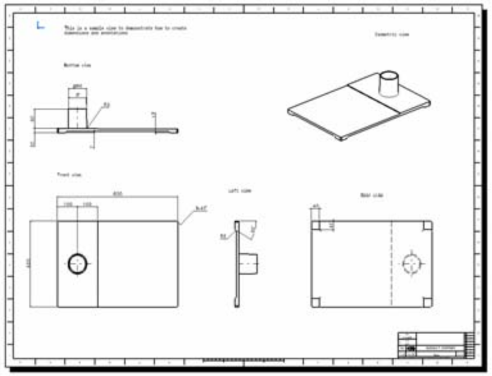


Drafting (ISO) 10

Learning Objectives

Upon completion of this lesson you will be able to:

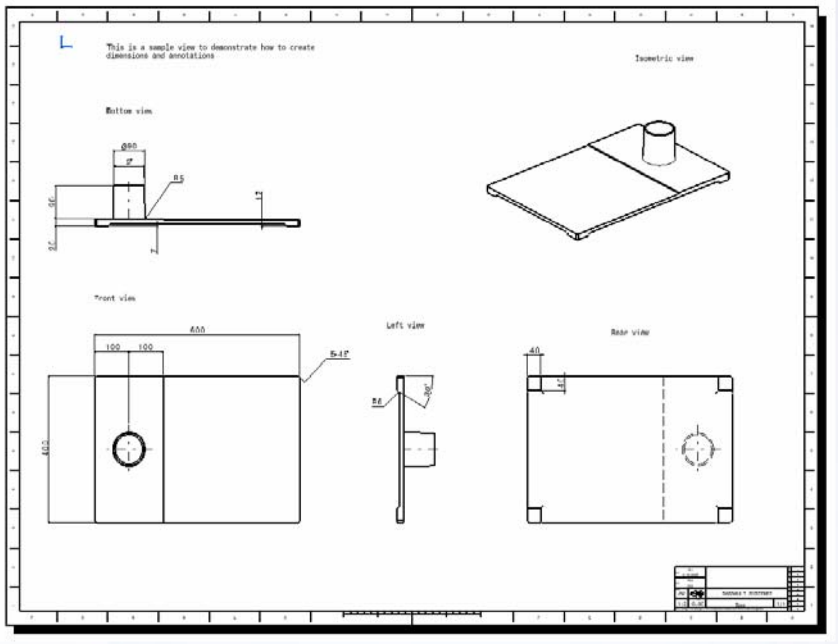
- ✓ Start a New Drawing
- ✓ Create Views
- ✓ Create Dimensions and Annotations
- ✓ Create Additional Views
- ✓ View Modifications
- ✓ Save the Drawing
- ✓ Print the Drawing



2 Hours

Case Study

The case study for this lesson is the drawing of base part used in the Drill Press 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. Create Views
3. Create Dimensions and Annotations
4. Create Additional Views
5. View Modifications
6. Save the Drawing
7. Print the Drawing

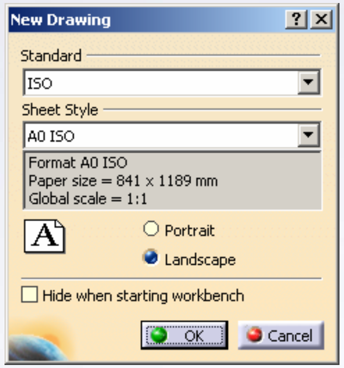
Start a New Drawing

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.

A. Title Blocks of the organization, which may contain information such as name of the organization, part number, etc.
B. Drafting standards such as dimension styles, line types, etc.



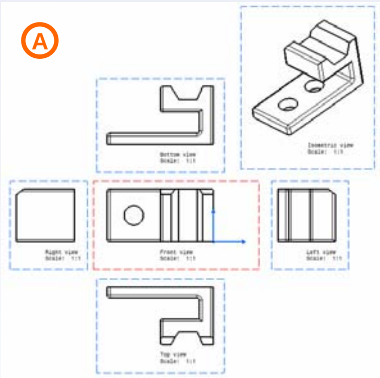
Create Views

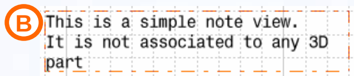
Views represent different orientations of a part, which help to convey its design intent. Two types of views can be created in CATIA:

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. Among different modes of view generation exact view generation mode is the best option in many cases. For creating views of large assemblies you can use CGR mode.





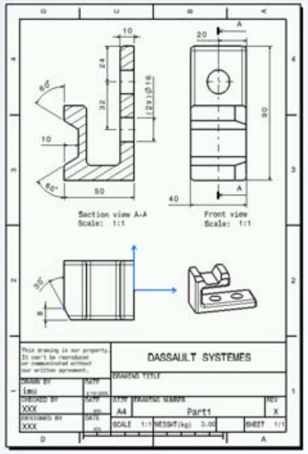
Create Dimensions and Annotations

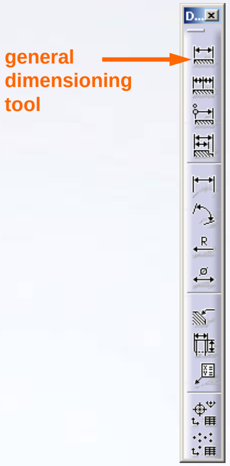
Dimensions define the size and functional intent of a part and are often used to create a fabrication drawing for a manufacturer.

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.

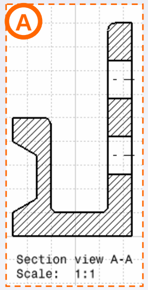


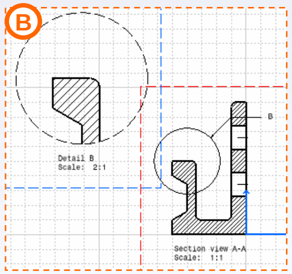


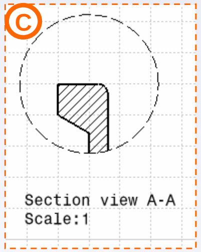
Create Additional Views

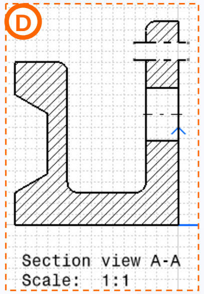
Secondary Views are added to improve the clarity of the description of a part through better visualization and/or to aid in dimensioning.

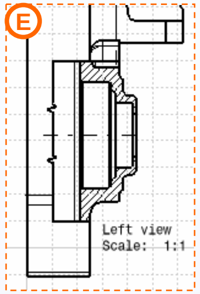
A. **Section View:** created by cutting the solid by section plane
B. **Detail View:** created by defining a "callout" on an existing view around the area to be enlarged, creates new view
C. **Clipping View:** created by defining a "callout" on an existing view around the area to be enlarged, modifies 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

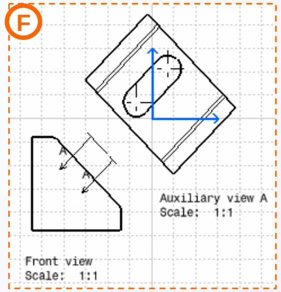












View Modifications

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.

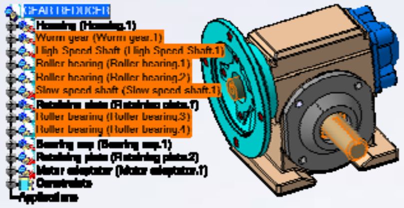
B. To delete the unnecessary views: Select the view frame and select Edit > Delete.

C. To modify the properties of a view: Select the view frame and select Properties from the contextual menu.

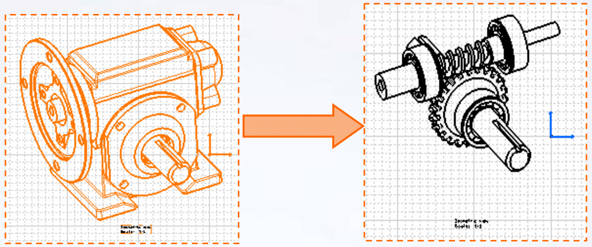
D. To change the content of a view: Select the view frame and select Modify Links from contextual menu.

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.



changing the content of a view



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

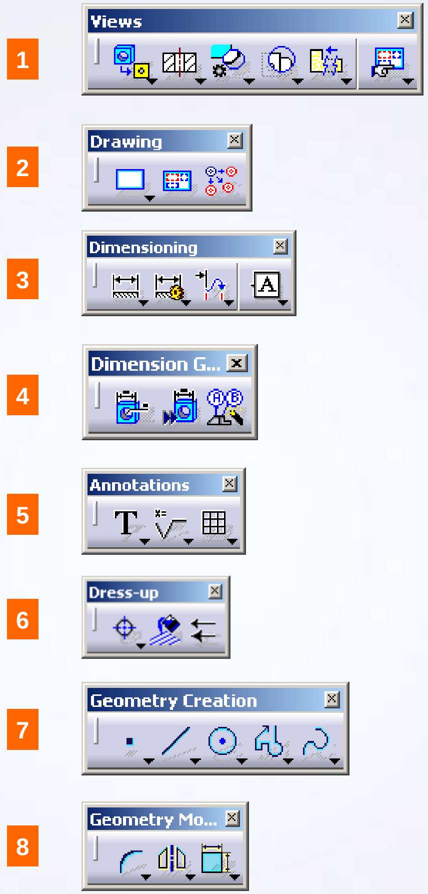
4 **Generation:** generate dimensions and balloons

5 **Annotations:** add annotations to existing views

6 **Dress-Up:** add dress-up elements on the drawing

7 **Geometry Creation:** creates 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

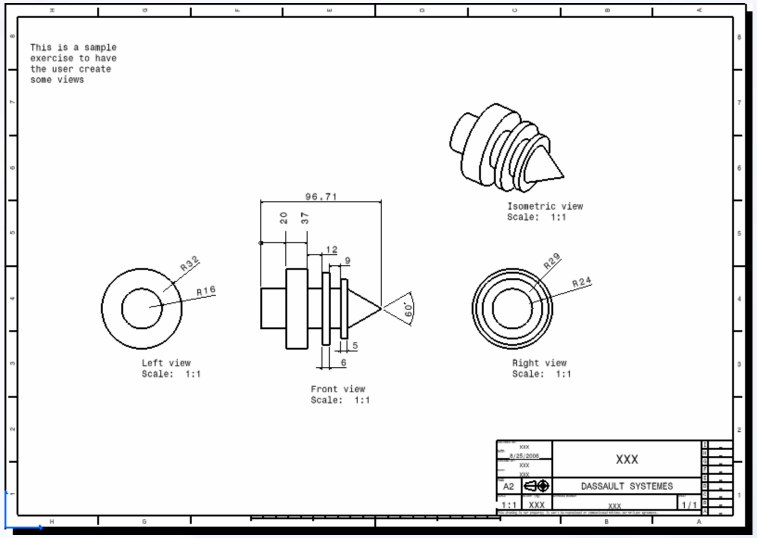
Recap Exercise

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:

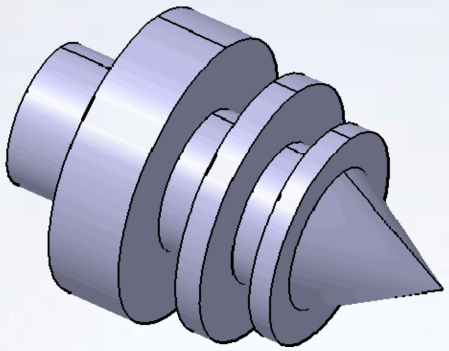
- **Create a drawing**
- **Insert a title block using catalog**
- **Create views using the view wizard**
- **Move and delete views**
- **Dimension geometry**



Do it Yourself (1/3)

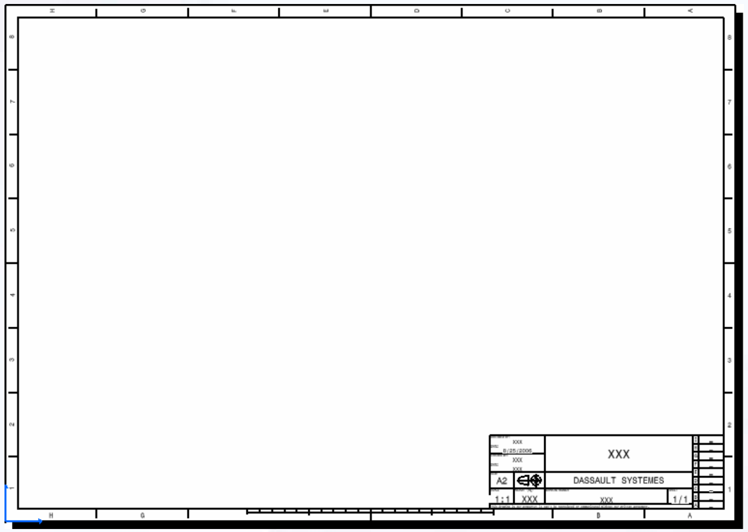
1. **Load Ex12B.CATPart .**

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)

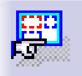
4. Use the view wizard to create views.

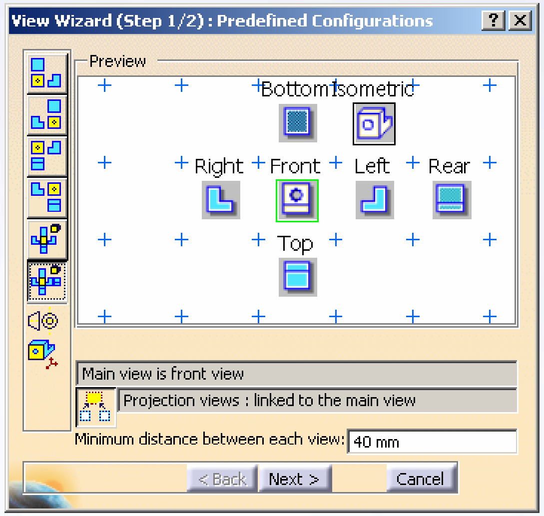
- Place the pre-defined layout of Configuration 6 with a third angle projection.

5. Move and delete some views.

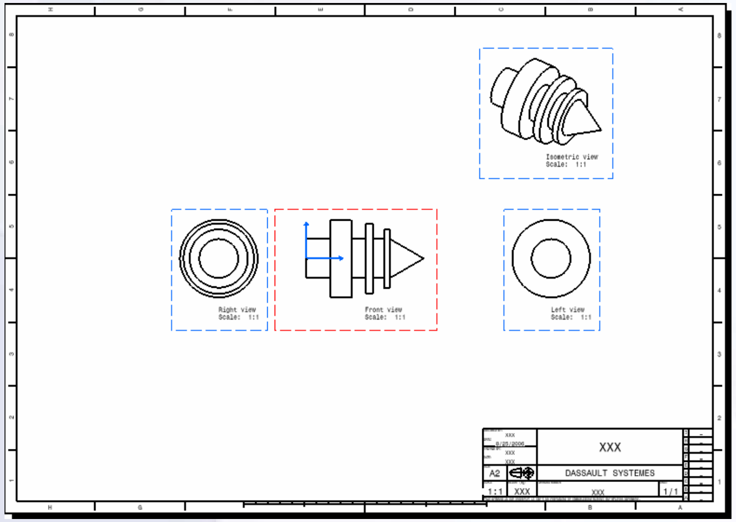
- Delete the top, bottom, and rear views.
- Position the views so that they appear evenly spaced out in the drawing.

4



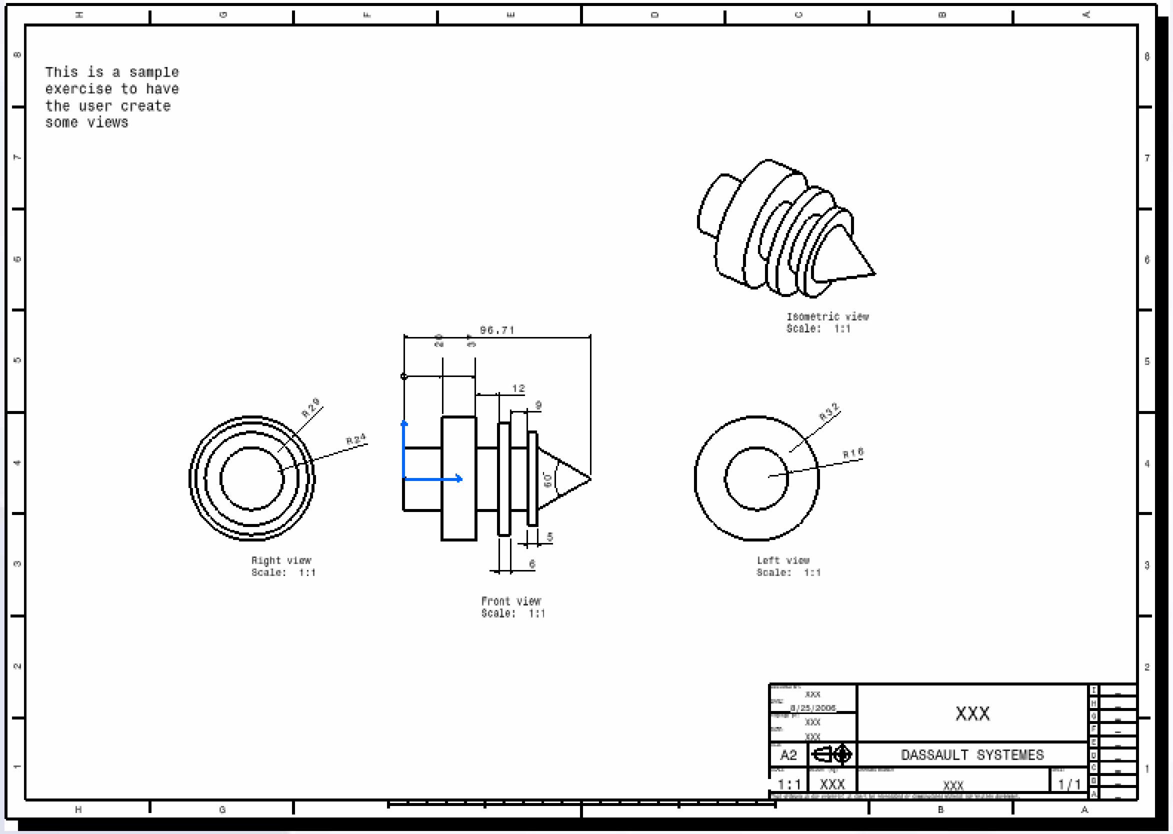


5



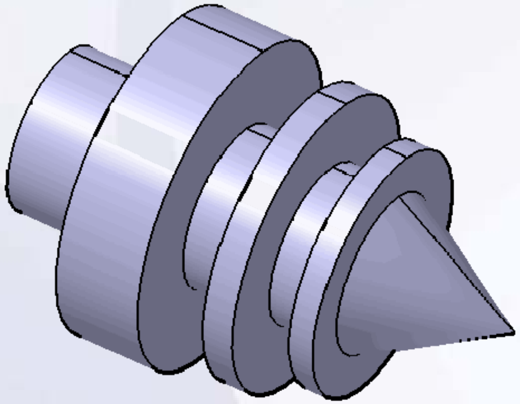
Do it Yourself (3/3)

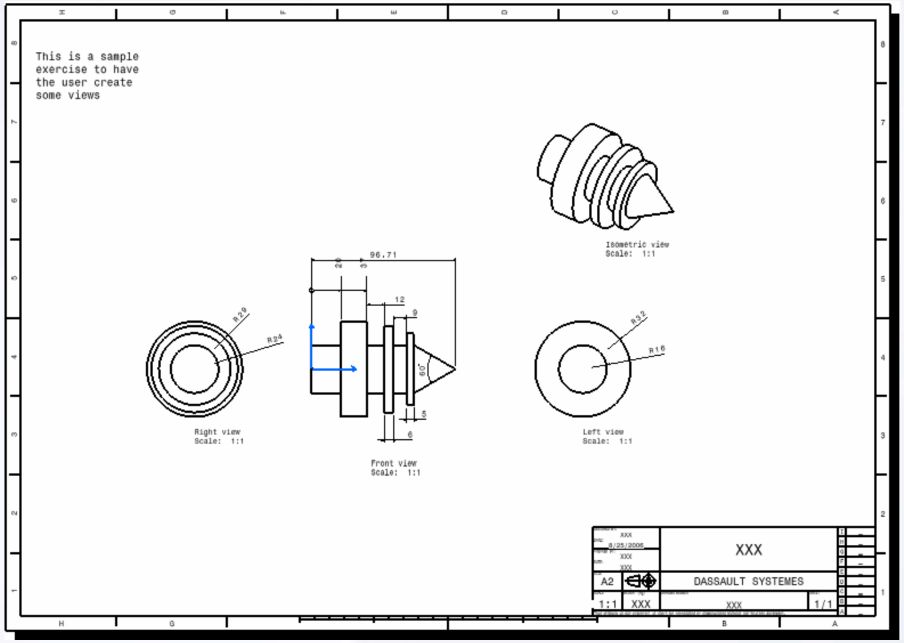
6. Dimension and annotate the drawing as shown.
7. Close the drawing without saving.



Exercise Recap: Drawing Creation

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



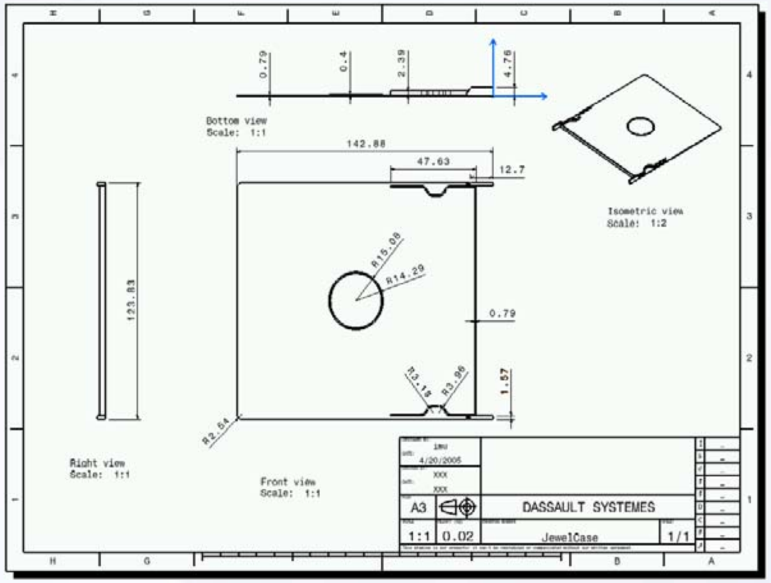


Exercise: Drawing Creation

Recap Exercise

20 min

In this exercise you will use the new skills you have gained to create a drawing of a jewel case part. You will use the tools used in the previous exercises to complete this exercise without any detailed instruction.

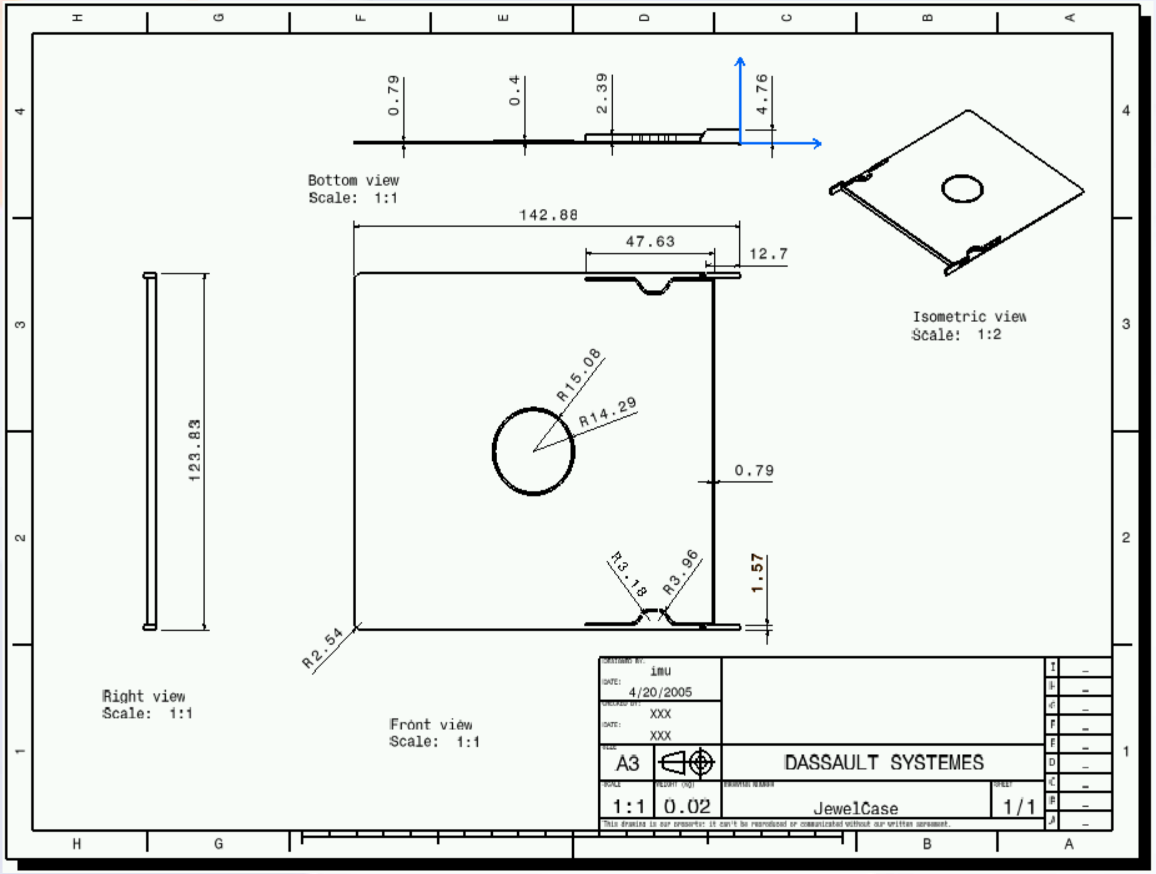


By the end of this exercise you will be able to:

- **Create a new drawing**
- **Insert a title block**
- **Add views**
- **Dimension and annotate the drawing**
- **Save the drawing**

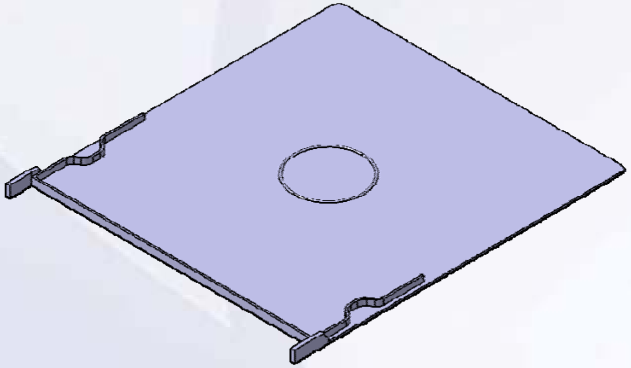
Do it Yourself

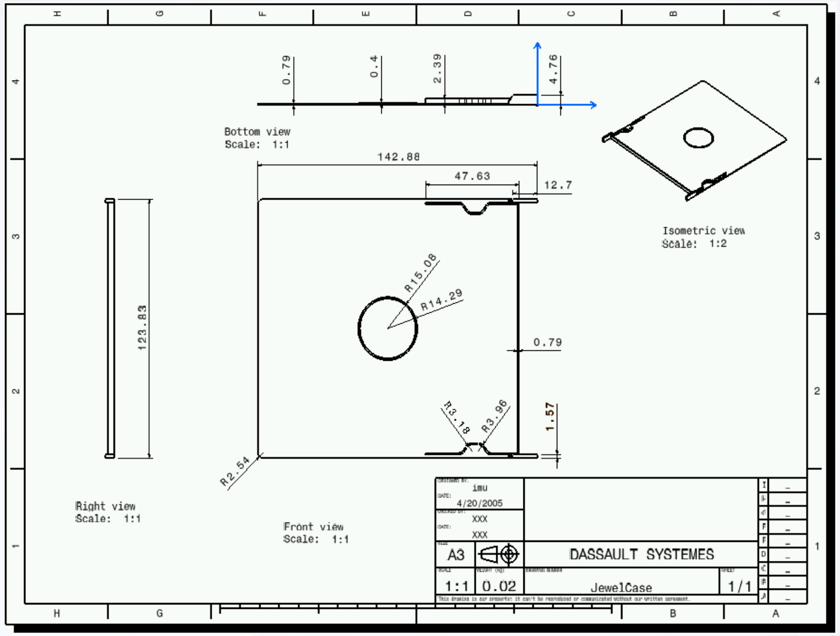
1. **Create drawing of the Jewel_Case_Done part, as shown.**



Exercise Recap: Drawing Creation

- ✓ Create a new drawing
- ✓ Insert a title block
- ✓ Add views
- ✓ Dimension and annotate the drawing
- ✓ Save the drawing





Case Study: Drafting

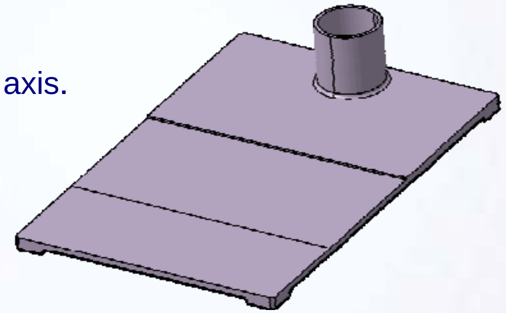
Recap Exercise

20 min

You will practice what you learned by completing the case study model using only a detailed drawing and hints as guidance.

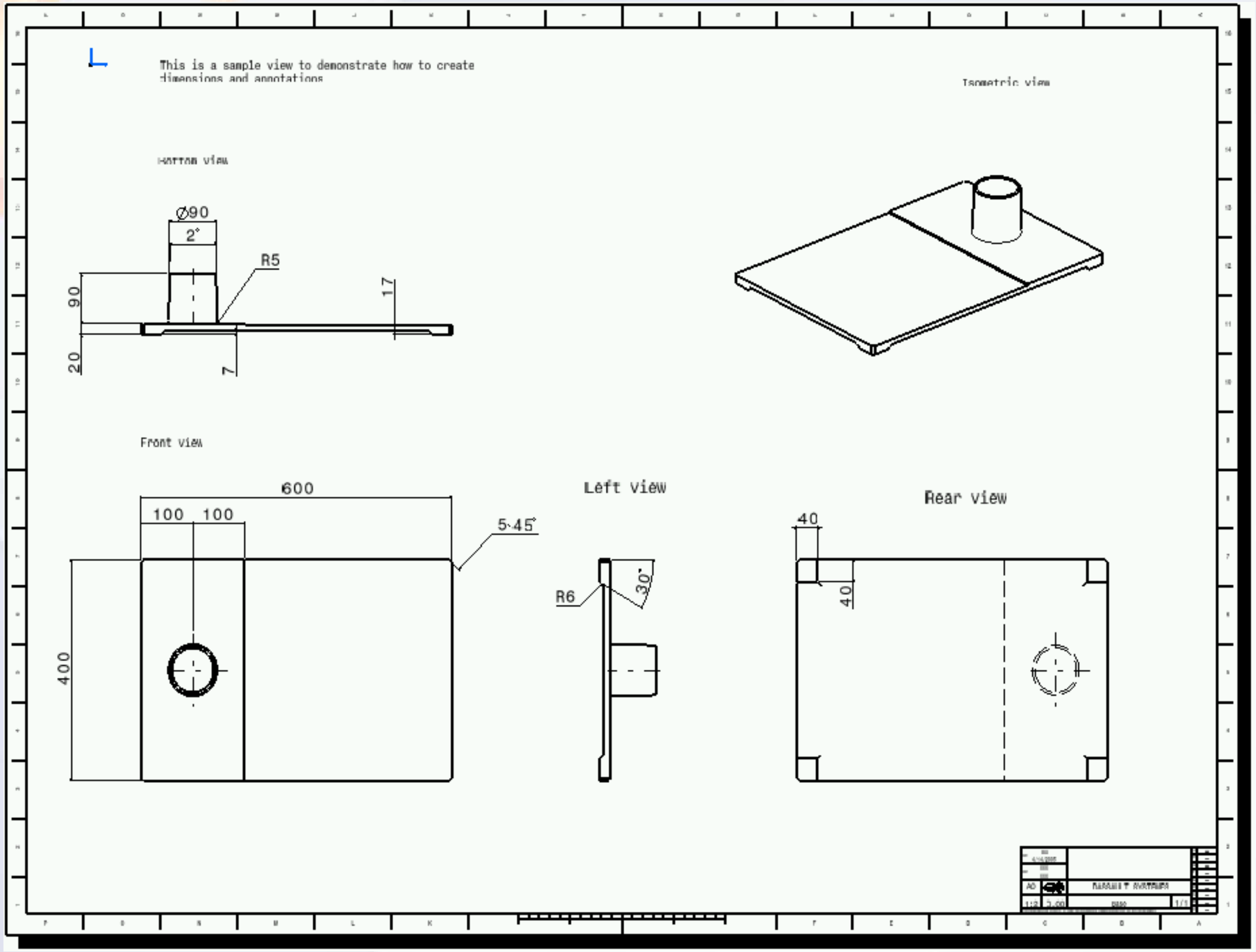
In this exercise, you will create the case study model drawing. Recall the design intent of this model:

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



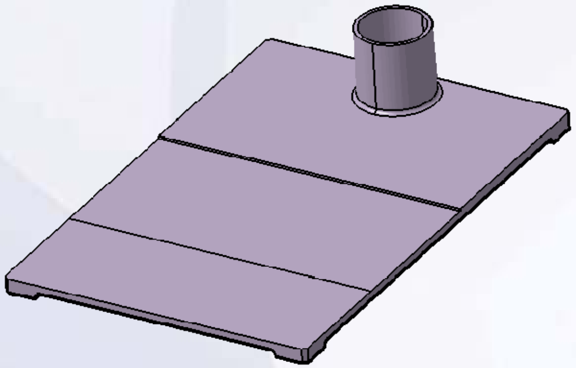
Using the techniques you have learned in this and previous lessons, create the model without detailed instruction.

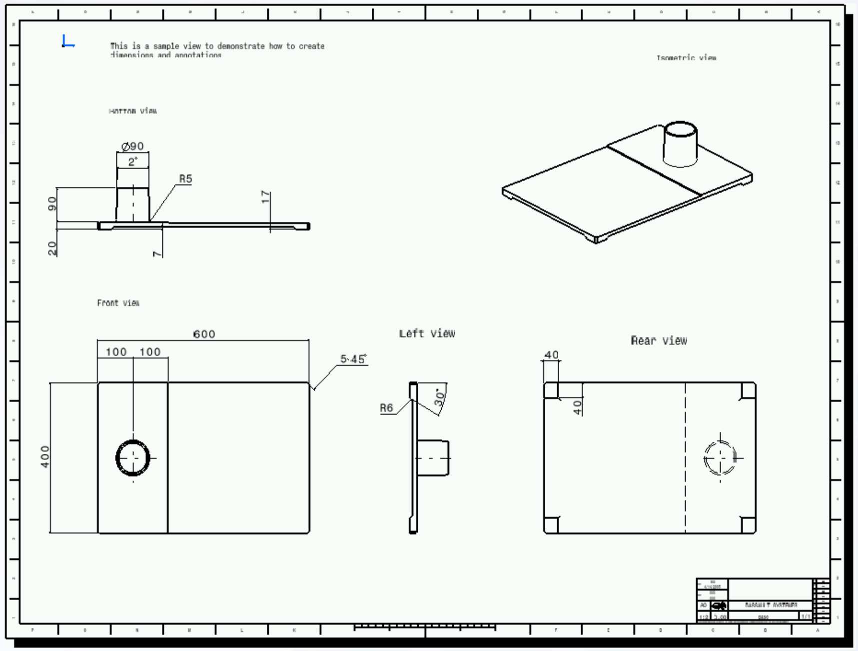
Do It Yourself: Drawing of the Base Part



Case Study: Base Recap

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





Master Project

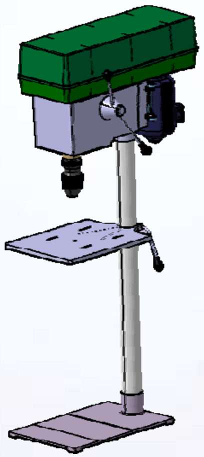
Drill Press Assembly

30 min

The objectives of this project are to create a part, analyze it, add the part to an assembly, modify the part within the context of the assembly, and to create an assembly drawing. The assembly used in this project is the Drill Press. You have created several components used in the assembly in case studies throughout the course.

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

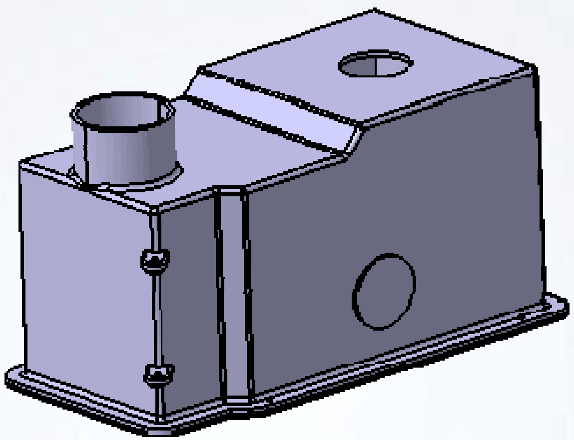
- **Create the support part for the assembly**
- **Finalize the part**
- **Create the drill press assembly**
- **Edit a part within the context of the assembly**
- **Create a drawing of the Drill Press assembly**



Master Project: Overview (1/3)

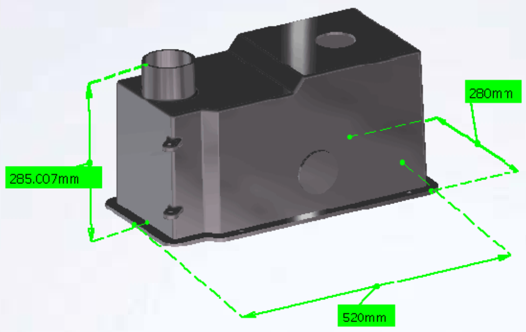
Following is a list of steps that are required to complete the master project:

1. **Create a part.**
   - Create the support used in the Drill Press assembly. The part uses features learned in this course.
   - Design Intent:
     - ✓ A stable base feature must be selected.
     - ✓ The model must be created symmetrically about the YZ and the ZX planes, and must sit on the XY plane.
     - ✓ Fillets must be created as separate features (they cannot be created within the profile sketches).
     - ✓ Avoid complicated profiles.

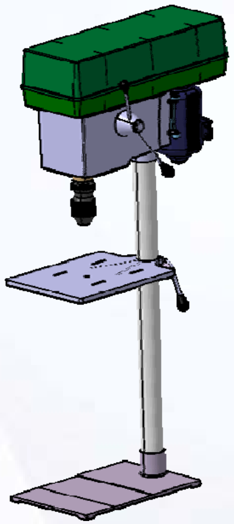


Master Project: Overview (2/3)

2. **Finalize the part.**
   - Apply material and take measurements of the model.
   - Design Intent:
     - ✓ The model must be made of aluminum.
     - ✓ Overall dimension, the mass and center of gravity must be clearly displayed on the model.
     - ✓ The length of the model must be double the width of the model.
     - ✓ A parameter must be created to control the length of the model.

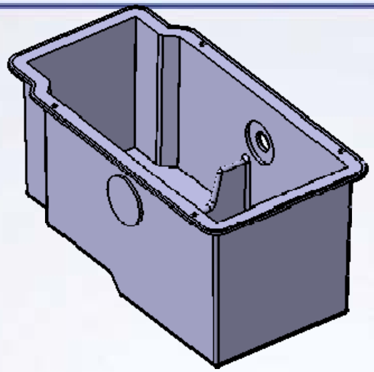


3. **Create the Drill Press assembly.**
   - Using the models you have built in the case studies, and additional models, create the Drill Press assembly.
   - Design Intent:
     - ✓ Select the most stable base component.
     - ✓ Fully constrain all the components in the assembly.

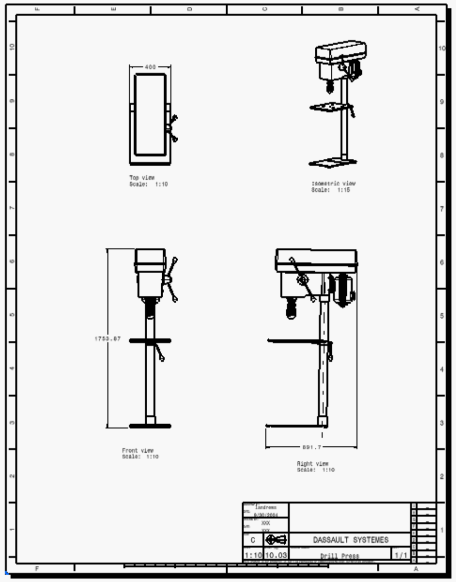


Master Project: Overview (3/3)

4. **Modify a part in the context of the assembly.**
   - Make changes to the support within the context of the assembly.
   - Design Intent:
     - ✓ The diameter of the hole must update when changes are made to the handle block.

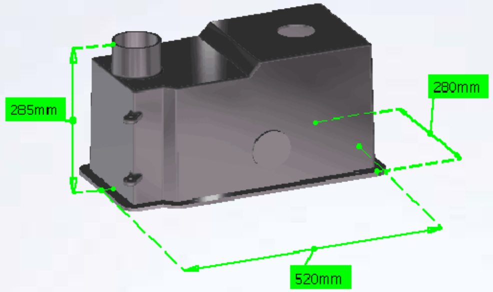


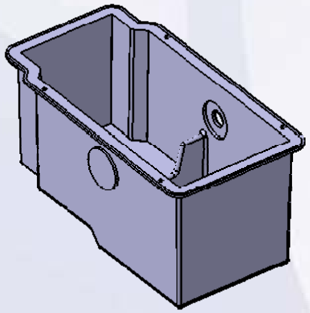
5. **Create an assembly drawing.**
   - Create a simple assembly drawing of the Drill Press assembly.
   - Design Intent:
     - ✓ Three main views of the drawing must be shown.
     - ✓ Overall dimensions for the Drill Press must be displayed on the drawing.
     - ✓ The drawing must contain a title block.

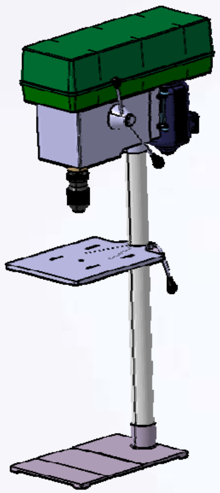


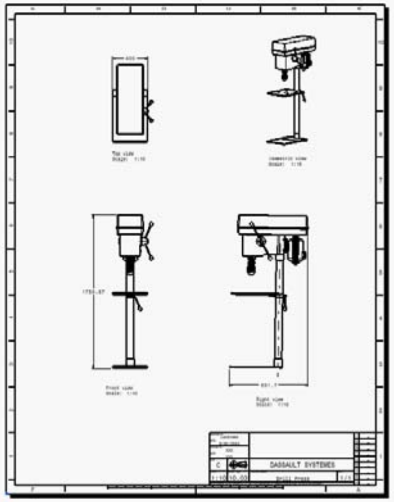
Master Project: Overview Recap

- ✓ Create the support
- ✓ Finalize the part file
- ✓ Create the Drill Press assembly
- ✓ Edit a part within the context of the assembly
- ✓ Create an assembly drawing









Master Project: Part Creation

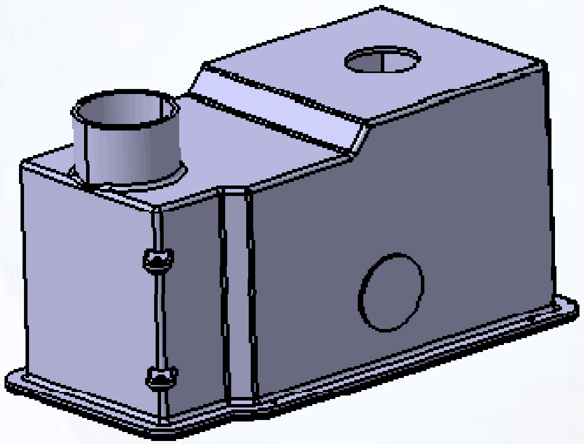
Drill Press Assembly

60 min

The objective of this step is to plan and create the support part. The support part is a part of the Drill Press assembly. High-level instructions for this exercise are provided.

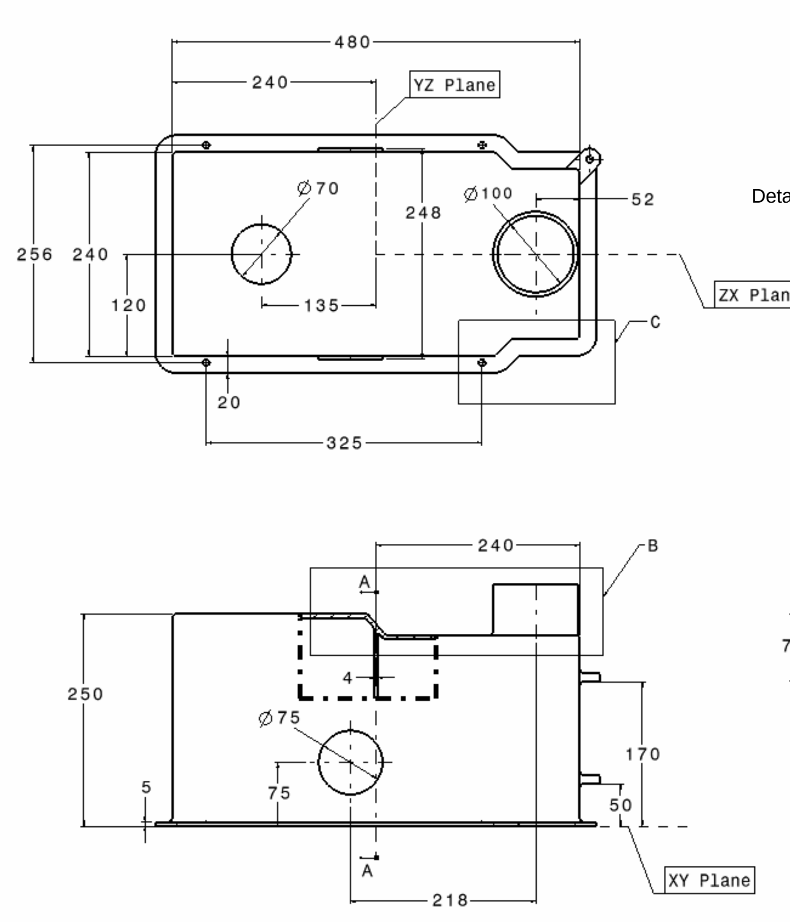
By the end of this step, you will be able to:

- **Create a new part**
- **Determine the best base feature**
- **Determine the best tool for each feature**
- **Save a file**



Master Project: Part Creation (1/10)

Create the support part.



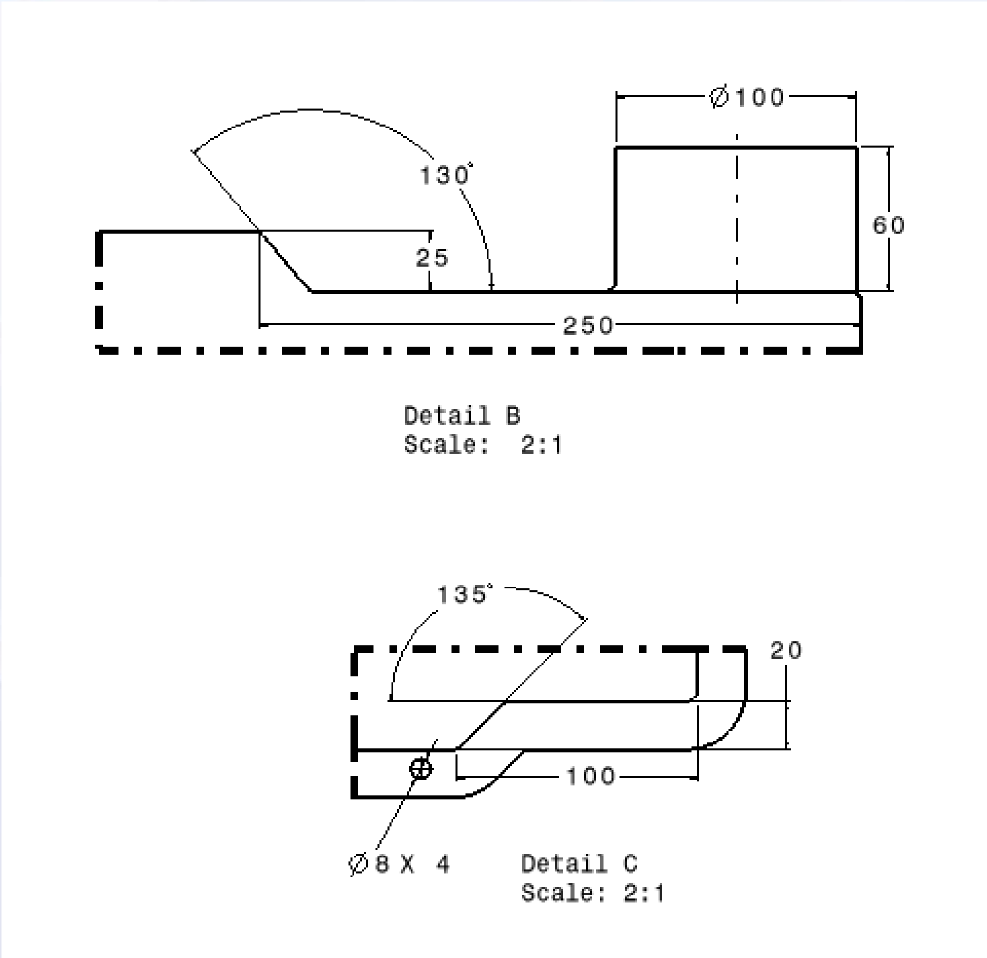
GENERAL NOTES:

All radii are 5mm
All draft surfaces are to 1 degree

Detail views on the next page.

Master Project: Part Creation (2/10)

Create the support part (continued).



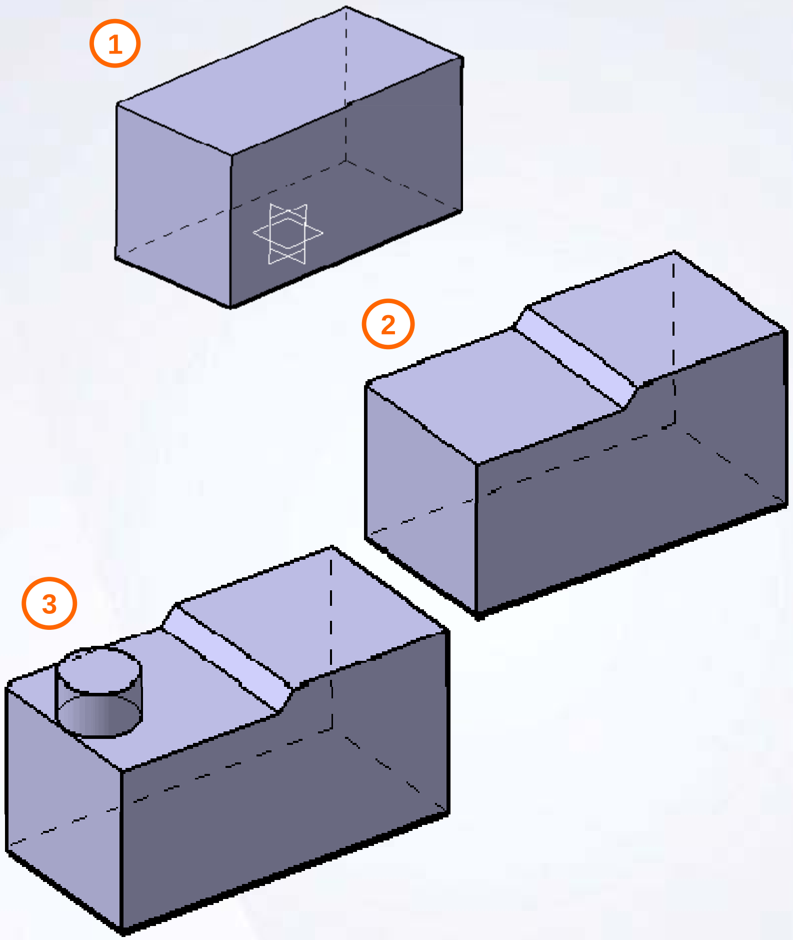
Master Project: Part Creation (3/10)

Here is a list of tasks to guide you:

1. Create a base feature.
   - Select a stable base feature. For the support, a simple block shape is best. Create the block so that the bottom of the block is located on the XY plane, and the box is symmetric about the YZ and ZX planes.

2. Remove material.
   - Remove the material using a pocket feature.

3. Create cylindrical pad.
   - Create the top cylinder using a pad feature. This feature could also be constructed with a shaft. What are the benefits of using a pad?



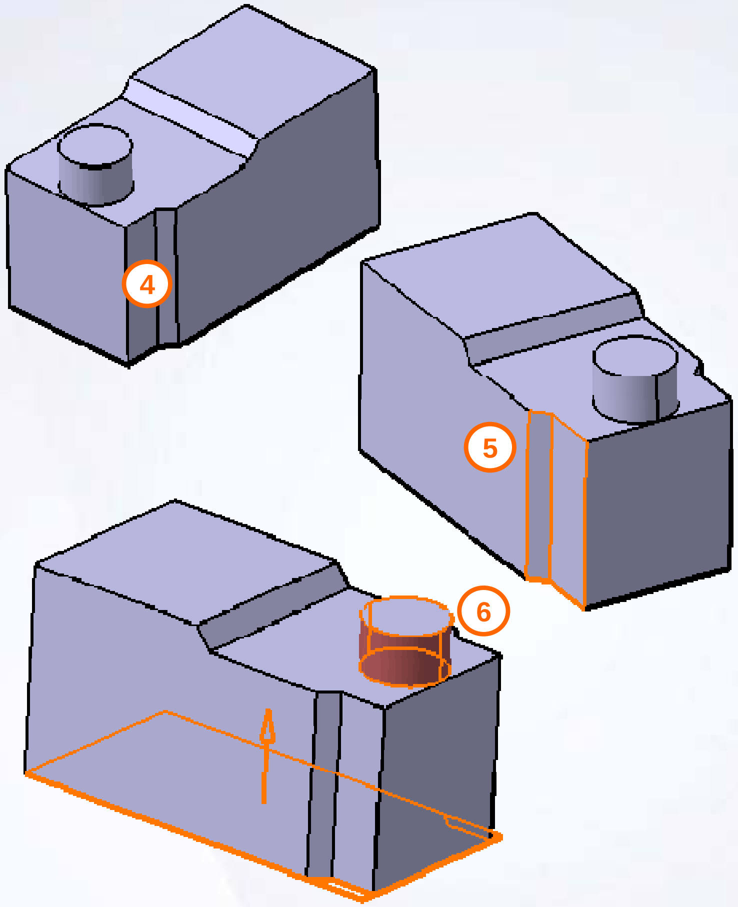
Master Project: Part Creation (4/10)

Here is a list of tasks to guide you (continued):

4. Remove side material.
   - Remove the side material using a pocket feature.

5. Remove material from the other side.
   - Remove the material from the other side by mirroring the pocket created in step 4 about a reference plane.

6. Apply 1degree draft to the cylindrical surface.
   - Apply a 1 degree draft to side surface of the cylindrical pad. Use the bottom surface of the block as the neutral element. Make sure the pull direction appears as shown.



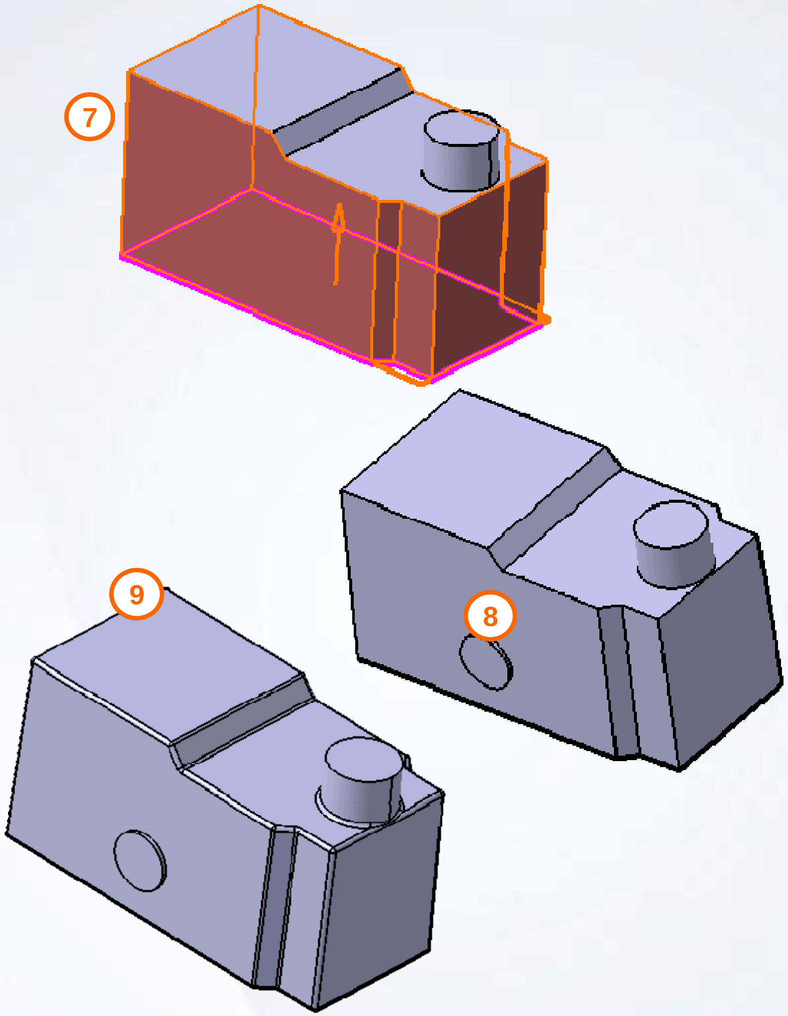
Master Project: Part Creation (5/10)

Here is a list of tasks to guide you (continued):

7. Apply a 1 degree draft to the sides of the base.
   - Apply a 1 degree draft to 8 side surfaces of the block. Use the bottom surface of the block as the neutral element. Make sure the pull direction is as shown.

8. Create Pad.
   - Create a pad through the block. Create the pad in the center of the block, then extrude in both the directions equally.

9. Apply fillets to the model.
   - Apply fillets to the model. All fillets must be 5mm.



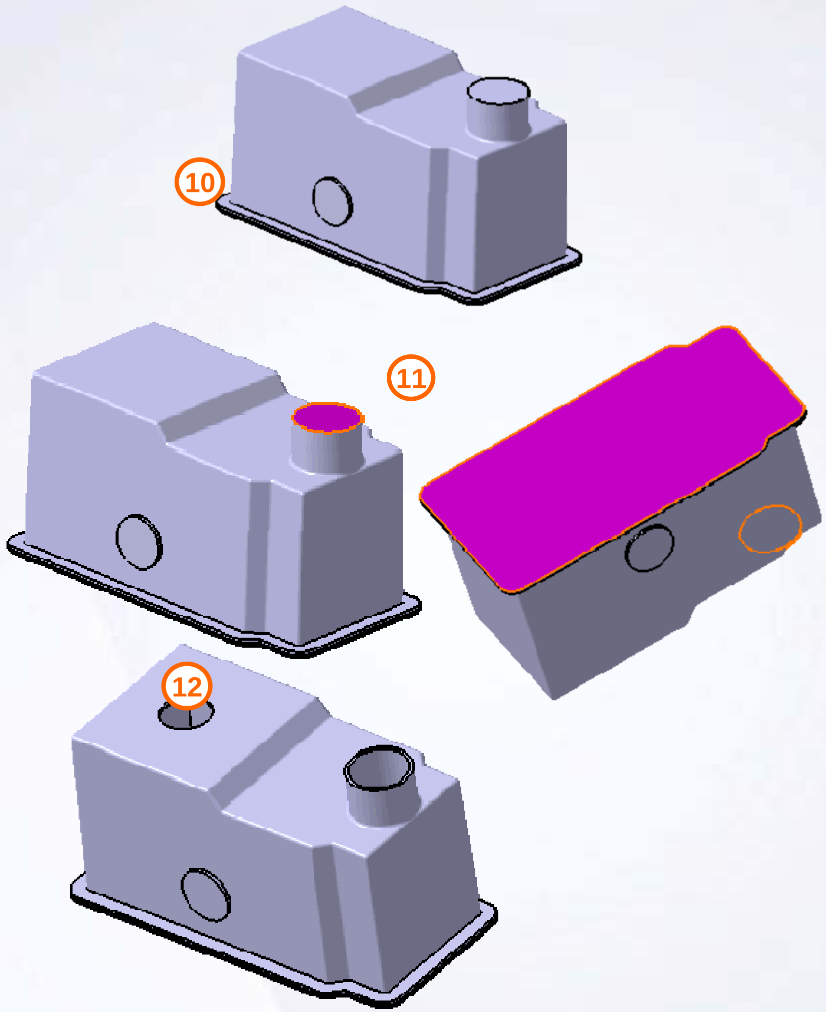
Master Project: Part Creation (6/10)

Here is a list of tasks to guide you (continued):

10. Create lip.
    - Create the lip at the base of the block by offsetting the bottom surface of the block in the Sketcher workbench. Recall the design intent: the bottom of the support must sit on the XY plane.

11. Shell the model.
    - Shell the model leaving a 5mm inside thickness. Remove the two surfaces as shown.

12. Create a hole.
    - Create a 70mm diameter hole at the top of the block.



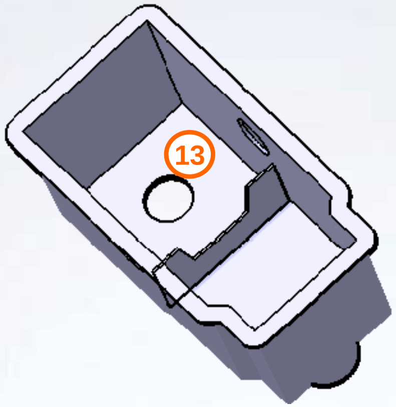
Master Project: Part Creation (7/10)

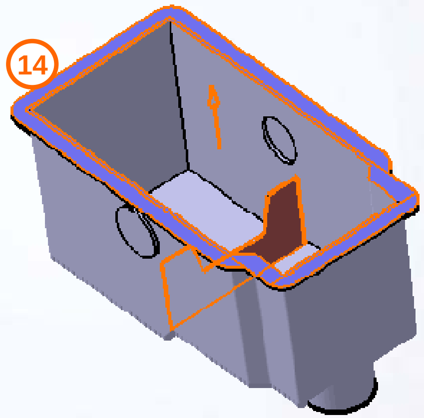
Here is a list of tasks to guide you (continued):

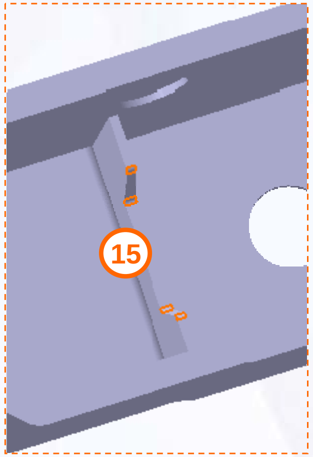
13. Create a stiffener.
    - Create a stiffener in the center of the support.

14. Create 1 degree draft on stiffener.
    - Apply a 1 degree draft to the sides of the stiffener. Use the bottom surface of the support as the neutral element. Ensure that the pull direction appears as shown.

15. Create edge fillets on the four edges.
    - Add 5mm edge fillets to the inside four edges of the stiffener as shown.







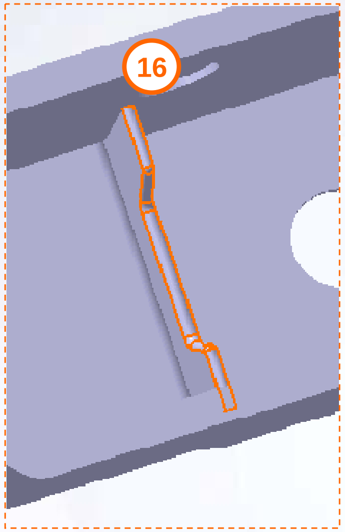
Master Project: Part Creation (8/10)

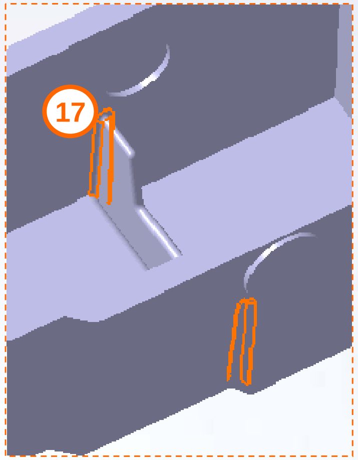
Here is a list of tasks to guide you (continued):

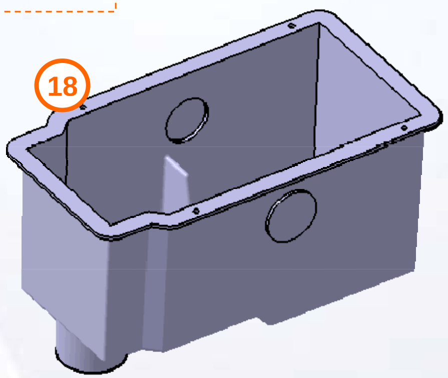
16. Create a Tritangent fillet.
    - Add a tritangent fillet to the stiffener.

17. Create edge fillets on the two outside edges of the stiffener.
    - Complete the stiffener by adding 5mm edge fillets to the end edges.

18. Create and pattern a hole.
    - Create an 8 mm diameter hole on the lip of the support and use the **Rectangular Pattern** tool to create the other three holes.



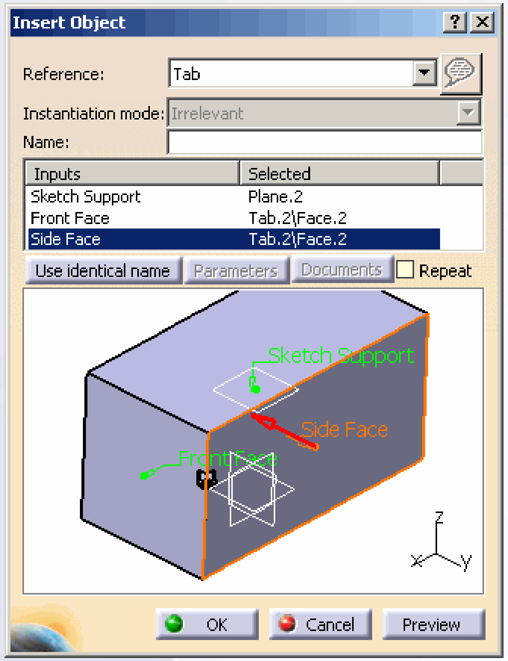


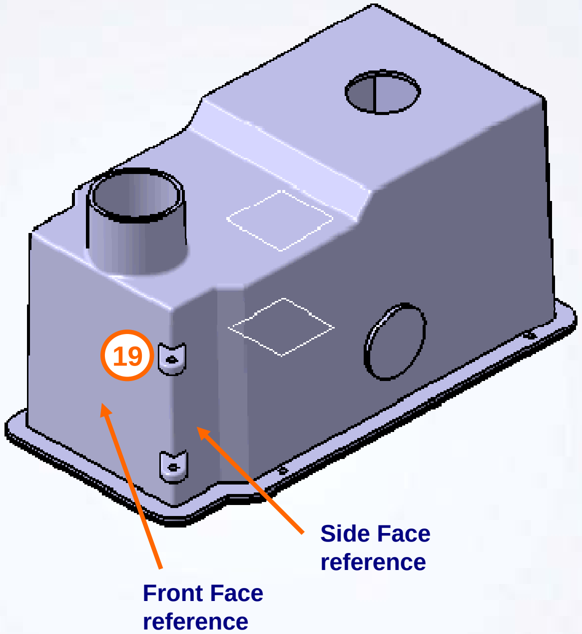


Master Project: Part Creation (9/10)

Here is a list of tasks to guide you (continued):

19. Add two cataloged features.
    - The tabs have been created for you and are located in the Tabs.Catalog file. Use the references for the user feature as shown.
    - Create two planes as the sketch support reference. The first plane is offset from the XY plane 50mm, the second plane is offset 170mm from the XY plane.

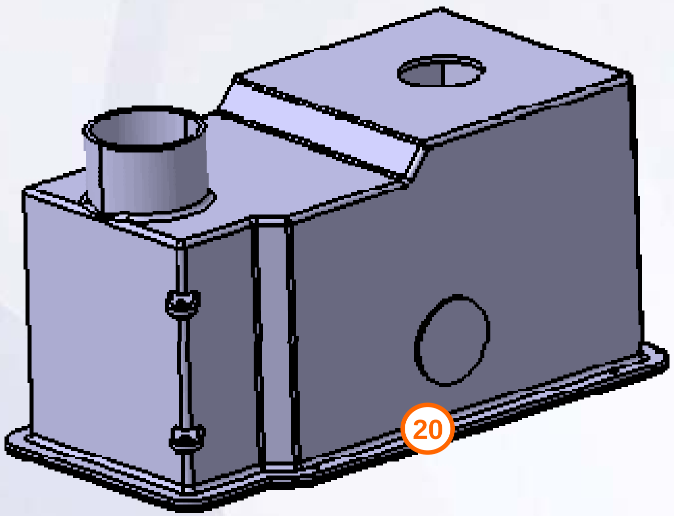


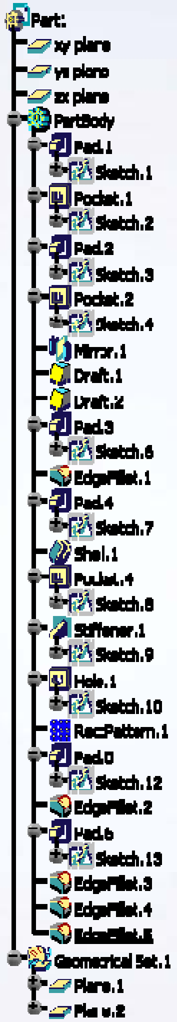


Master Project: Part Creation (10/10)

Here is a list of tasks to guide you (continued):

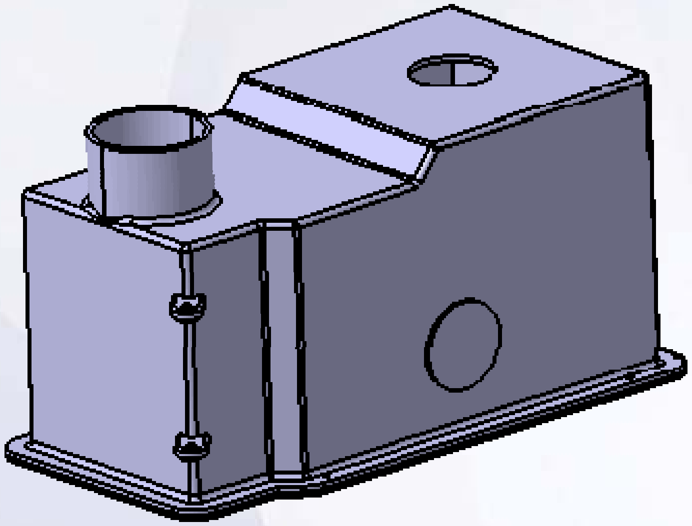
20. Add edge fillets to both tabs and lip edge.
    - Finish the model by adding 5mm fillets to the tabs and the edge between the lip and the block.





Master Project: Part Creation Recap

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



Master Project: Finalize the Part

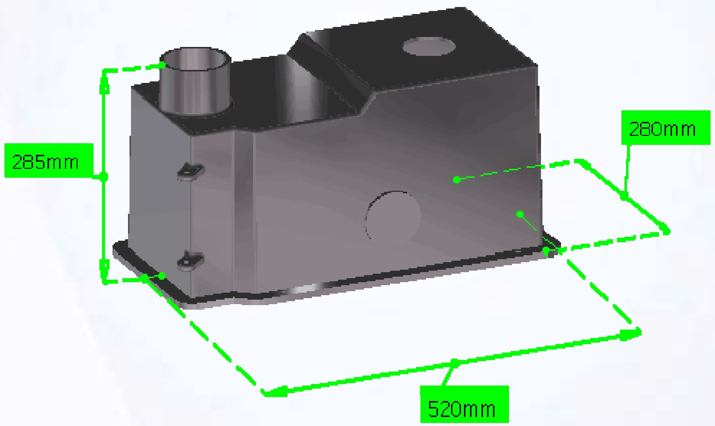
Drill Press Assembly

60 min

The objective of this step is to finalize the support part. Material will be applied, measurements will be taken, and formulas will be developed. High-level instructions for this exercise are provided.

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

- Apply material to the model
- Save overall length and width measurements
- Calculate the mass of the model
- Locate the center of gravity
- Create formulas



Master Project: Finalize the Part (1/4)

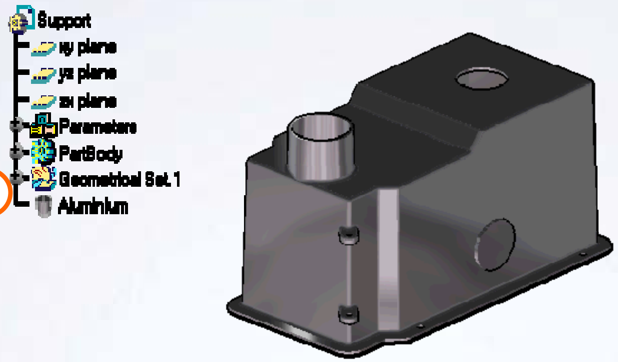
Continue with the support part created in step 1. If you did not complete step 1, use Support_Step1.CATPart from the completed folder.

Here is a list of tasks to guide you:

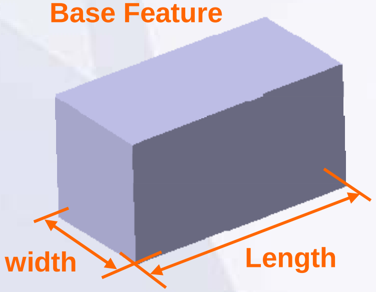
1. Apply material to the model.
   - Apply aluminum to the model.

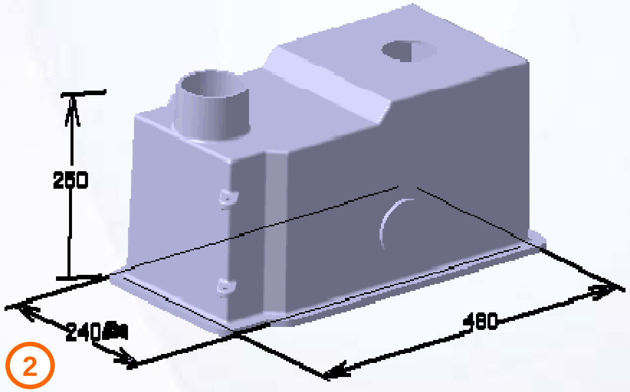
2. Create formula to control the width of the model.
   - Create a formula that equates the width of the model to ½ the length. Use the width and length dimension for the base feature.

1



Base Feature





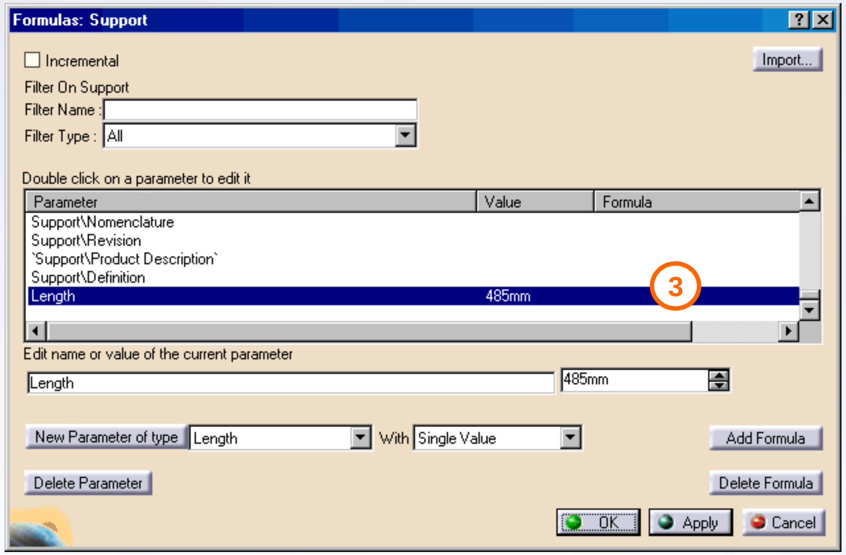
2

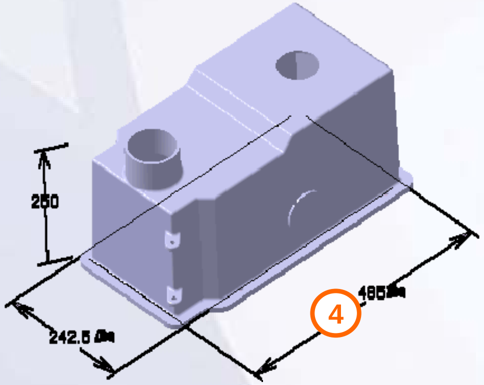
Master Project: Finalize the Part (2/4)

Here is a list of tasks to guide you (continued):

3. Add a user-defined parameter to the model.
   - Create a user-defined parameter of type [length]. Name the parameter [Length]. Give the parameter a value of [485mm].

4. Equate the length of the model to the user-defined parameter.
   - Equate the length dimension to the new parameter. Test the parameter by changing the Length parameter value back to [480mm].



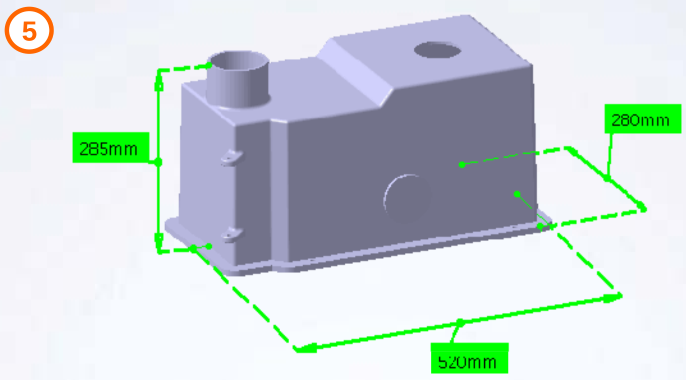


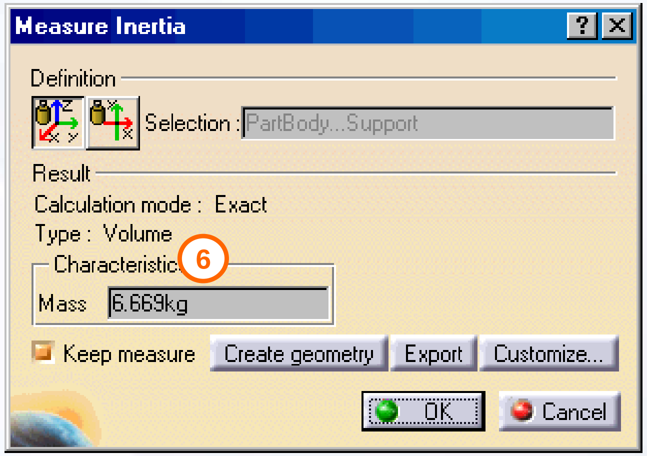
Master Project: Finalize the Part (3/4)

Here is a list of tasks to guide you (continued):

5. Calculate the overall length, width and height of the model.
   - Calculate the measurements as shown. Customize the measurements to calculate the components. Save the measurements.

6. Determine the mass of the support.
   - Customize the **Measure Inertia** tool to only calculate mass. Save the measurement.



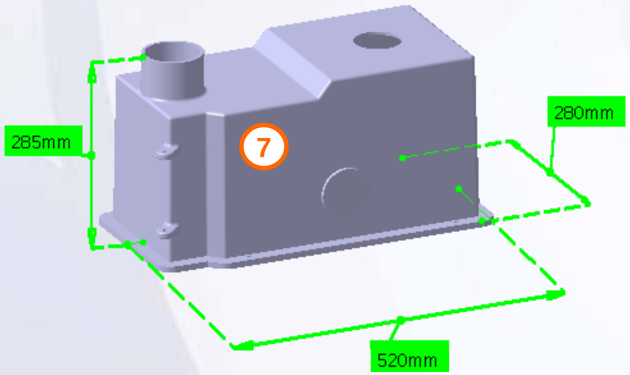


Master Project: Finalize the Part (4/4)

To help, here is a list of tasks (continued):

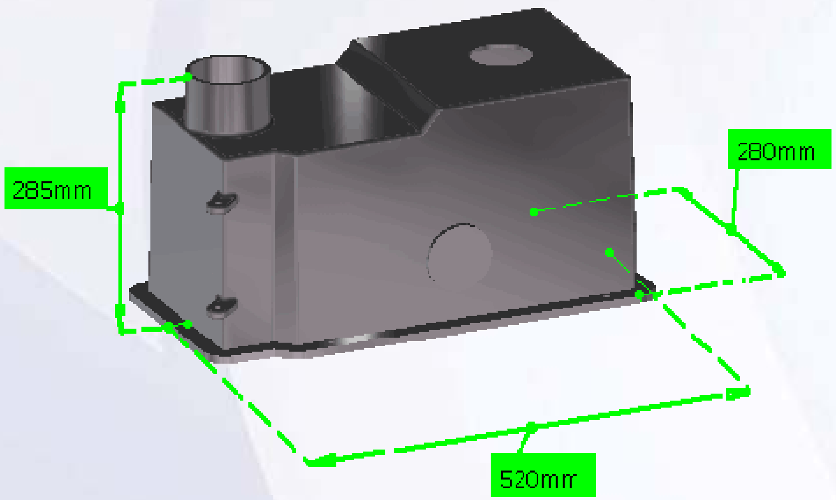
7. Locate the center of gravity.
   - Create a point at the center of gravity using the **Create Geometry** option from the Measure Inertia window.

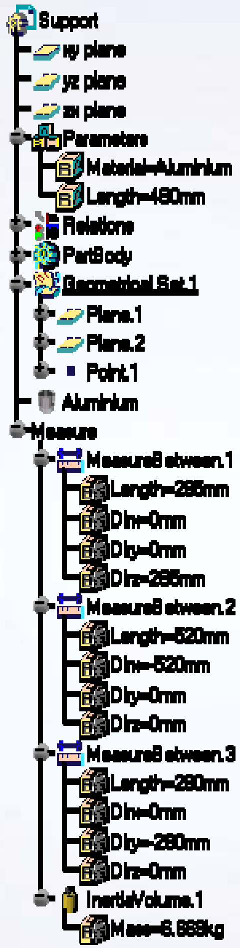
8. Hide the Geometrical set and the Measurements.
   - For clarity, hide the Geometrical Set.1 and Measure branches of the specification tree.



Master Project: Finalize the Part Recap

- ✓ **Apply material to the model**
- ✓ **Save overall length and width measurements**
- ✓ **Calculate the mass of the model**
- ✓ **Locate the center of gravity**
- ✓ **Create formulas**



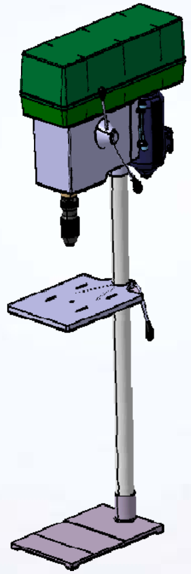


Master Project: Create an Assembly

Drill Press Assembly

60 min

The objective of this step is to create the Drill Press assembly. This assembly contains all the parts you have created in the case studies also the support part. High-level instructions for this exercise are provided.



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

- **Create a new assembly**
- **Add components to the assembly**
- **Fully constrain the assembly**
- **Modify display properties**
- **Save the file**

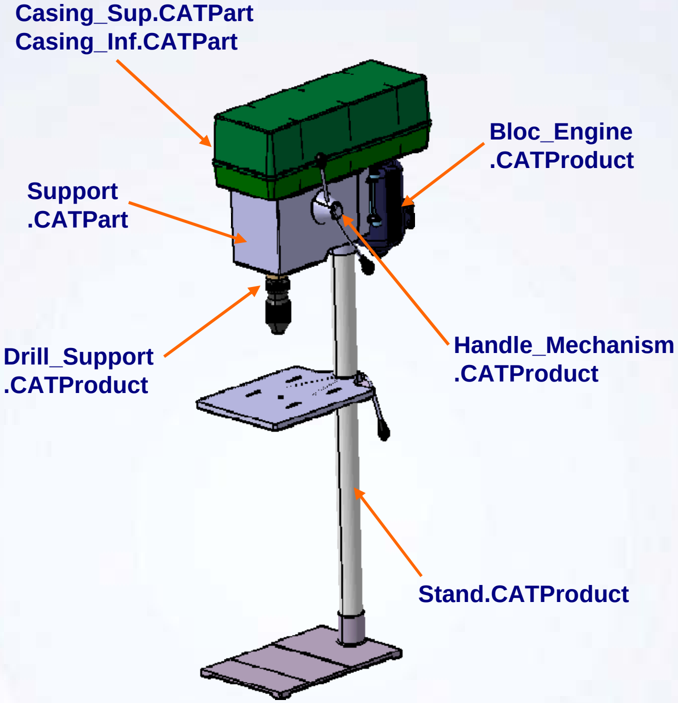
Master Project: Create an Assembly (1/7)

All sub-assemblies have already been created for you. The structure of the Drill Press assembly is shown.

Here is a list of tasks to guide you:

1. Create a new assembly.
   - Create a new assembly. Name the assembly [Drill_Press].

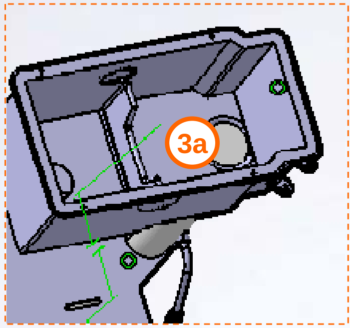
2. Add the Stand.CATProduct.
   - The first component assembled into the file should be the most stable. In this case, the Stand sub-assembly is the best choice. Use the Fix constraint to fully constrain the component.

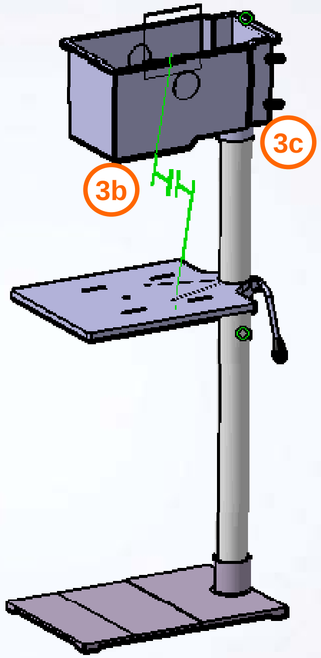


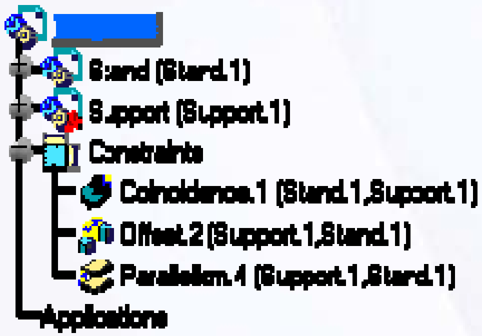
Master Project: Create an Assembly (2/7)

Here is a list of tasks to guide you (continued):

3. Insert Support.CATPart.
   a. Add the Support to the assembly. If you did not complete step 2, use the Support_Step2.CATPart in the completed folder instead.
   b. The inside bottom surface of the support should be on the same plane as the top of the column.
   c. The ZX plane in the support should be parallel with the side of the table.
   d. The axis of the pad on the support should be coincident with the axis of the column.



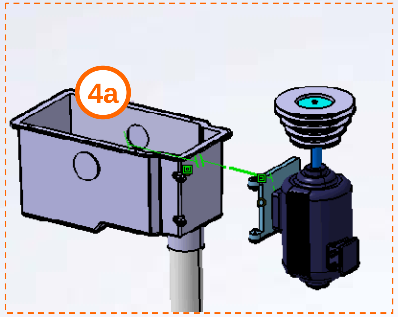


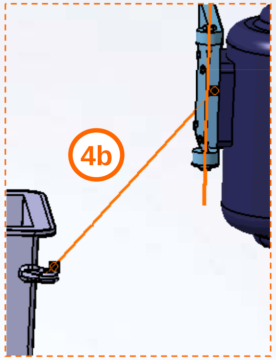


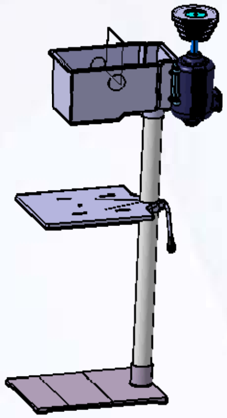
Master Project: Create an Assembly (3/7)

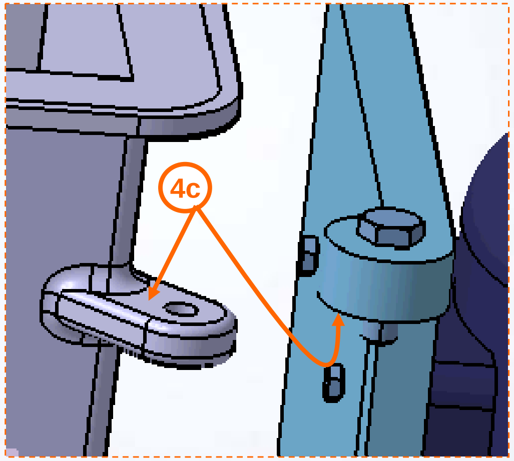
Here is a list of tasks to guide you (continued):

4. Insert Bloc_Engine.CATProduct.
   a. Make the surface of the Engine support parallel with the YZ plane in the support.
   b. Make the axis of the hole coincident with the hole in the tab on the support.
   c. The bottom surface should be in contact with the top surface of the tab.





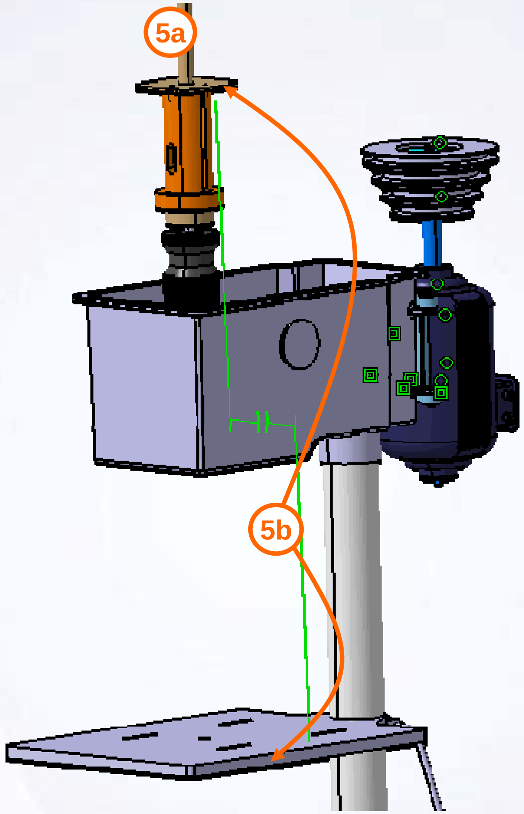




Master Project: Create an Assembly (4/7)

Here is a list of tasks to guide you (continued):

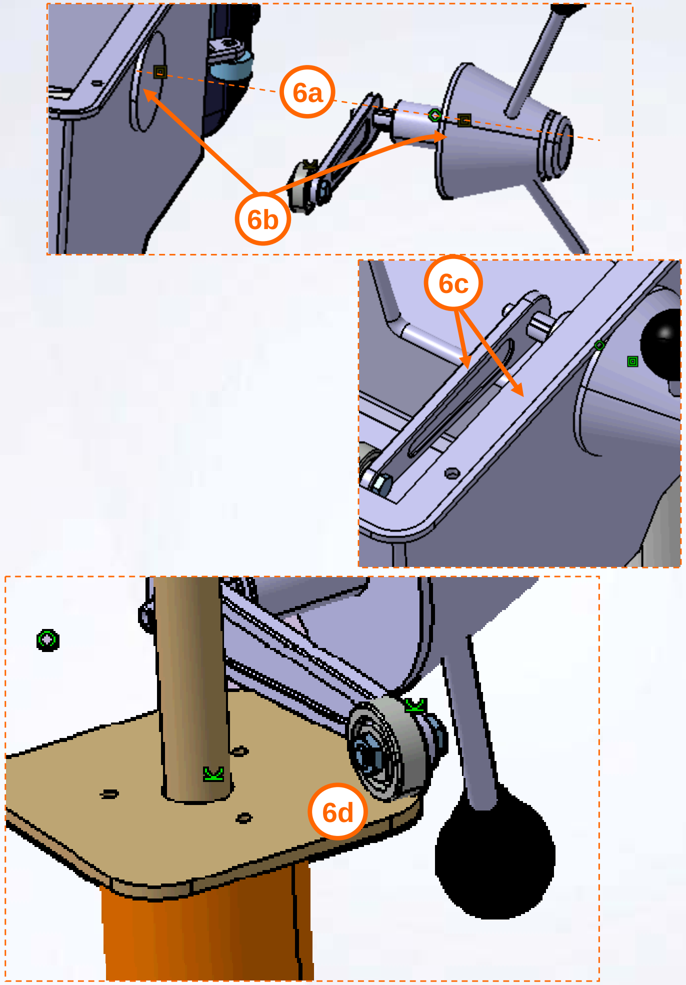
5. Insert Drill_Support_L9.CATProduct.
   a. Align the axis of the Canella_Axis with the axis of the hole in the support.
   b. Make the side of support plaque parallel to the side of the side of the table.



Master Project: Create an Assembly (5/7)

Here is a list of tasks to guide you (continued):

6. Insert Handle_Mechanism.CATProduct.
    a. Make the axis of the handle block coincident with the axis in the circular pad on the support.
    b. Make the bottom surface on the revolved feature in the handle block in contact with the circular pad on the support.
    c. Make the top surface of the connecting rod parallel to the top surface of the support.
    d. Add an external line contact constraint, using the surface contact tool between the circular surface of the bearing and the top surface of the support plaque.

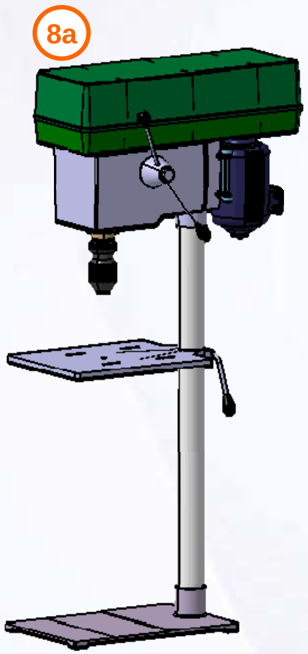


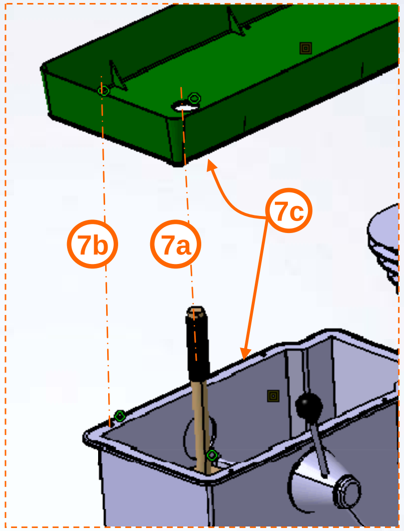
Master Project: Create an Assembly (6/7)

Here is a list of tasks to guide you (continued):

7. Add Casing_inf.CATPart.
   a. Make one of the center holes in the casing coincident with the axis of the Canella axis.
   b. Make one of the outer holes coincident with the corresponding hole on the support.
   c. Have the bottom surface of the casing in contact with the top surface of the support.

8. Add Casing_sup.CATPart.
   a. Use the default reference planes on both the casing parts to fully constrain the component as shown.

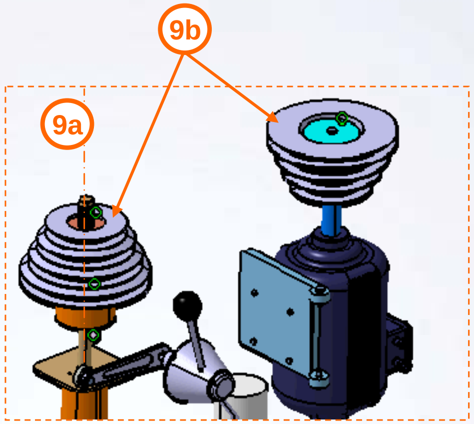




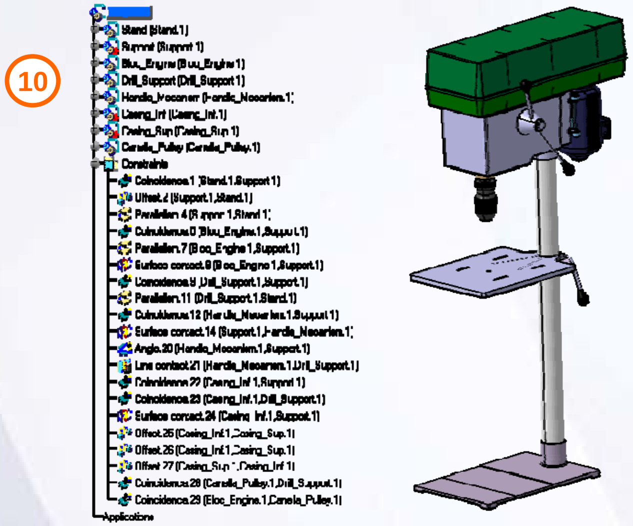
Master Project: Create an Assembly (7/7)

Here is a list of tasks to guide you (continued):

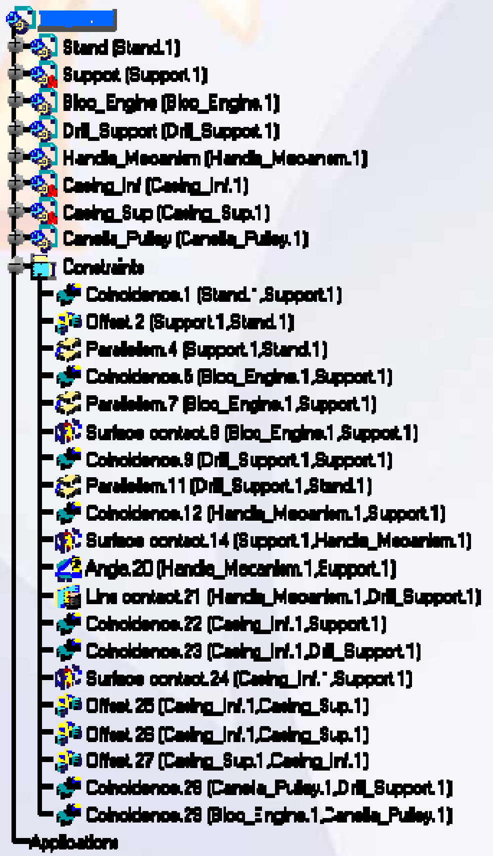
9. Insert Canella_Pulley.CATProduct.
   - For clarity, hide the support, and both the casing parts from display.
     a. Make the axis of the pulley support with the axis of the canella axis.
     b. Make the top surface of the pulley coincident with the top surface of the pulley in the Bloc_Engine.

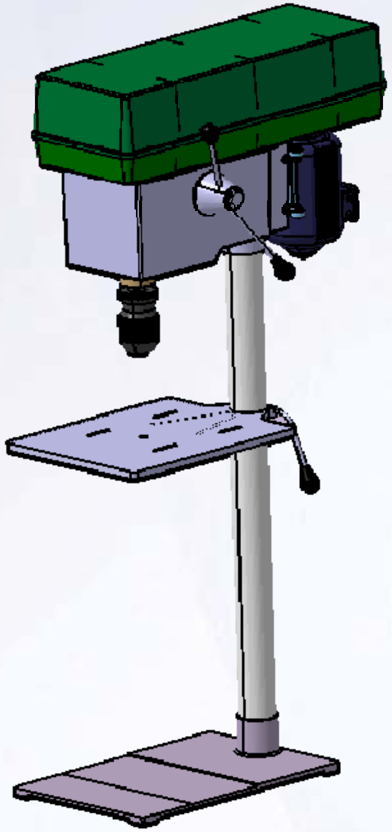


10. Modify the display.
    - Re-display all the components in the assembly. Hide all the constrains and visible reference planes if any.



Master Project: Create an Assembly Recap





- ✓ **Create a new assembly**
- ✓ **Add components to the assembly**
- ✓ **Fully constrain the assembly**
- ✓ **Modify display properties**
- ✓ **Save the file**

Master Project: Edit a Part in an Assembly

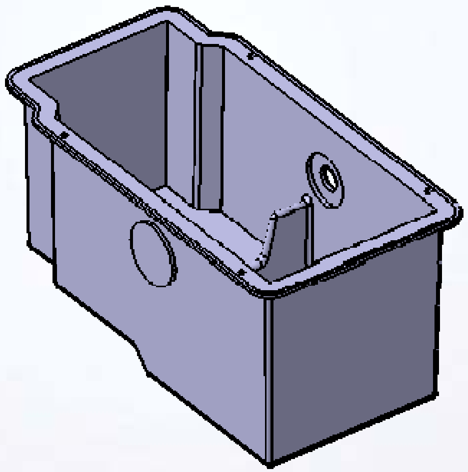
Drill Press Assembly

40 min

The objective of this step is to use the skills learned in this course to add a feature to the support part in the context of the assembly. Currently, there is no hole on the support for the handle block to the inserted. In this step, you will create the hole by referencing the diameter of the handle block. High-level instructions for this exercise are provided.

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

- **Edit a part in the context of the assembly**
- **Create a feature using references from other components in the assembly**



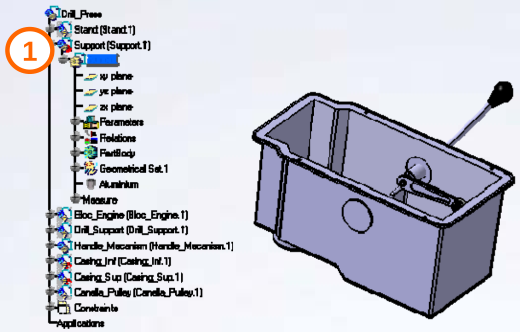
Master Project: Edit a Part in an Assembly (1/2)

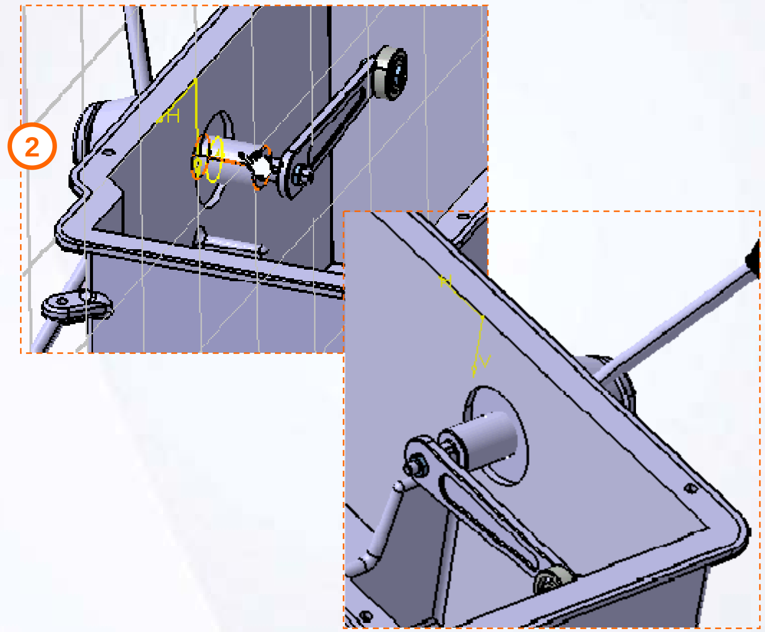
Continue with the Drill Press assembly created in step 3. If you have not completed step 3, use Step3_complete.CATProduct instead.

Here is a list of tasks to guide you:

1. Activate the support part.
   - To create a feature inside the support model, the model must be active.

2. Create profile.
   - Create a profile by projecting the edges of the handle block. This ensures that the diameter of the hole is always correct. To create the projection, rotate the model to 3D orientation, select the cylindrical surface, and use the **Intersect 3D elements** tool.





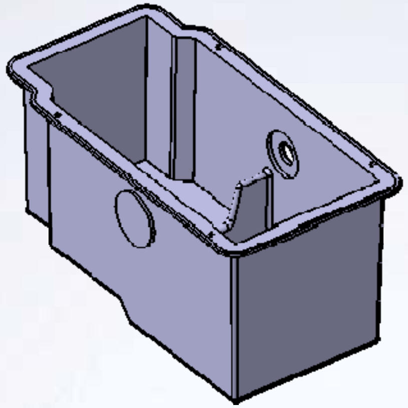
Master Project: Edit a Part in an Assembly (2/2)

Here is a list of tasks to guide you (continued):

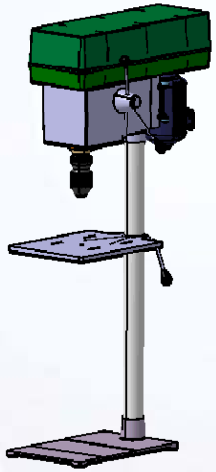
3. Remove material.
   - Use a pocket feature to remove the material from the support.

4. Reactivate the assembly.
   - Reactivate the assembly. Re-display the components you may have hidden.

3

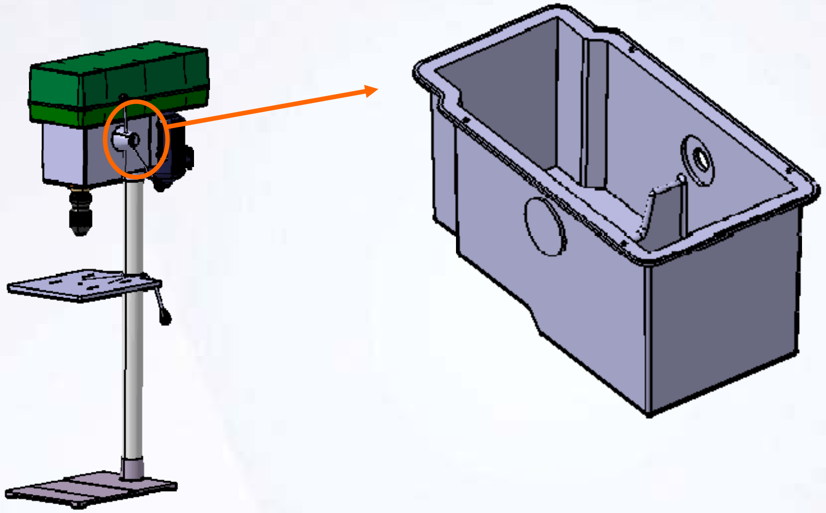


4



Master Project: Edit a Part in an Assembly Recap

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

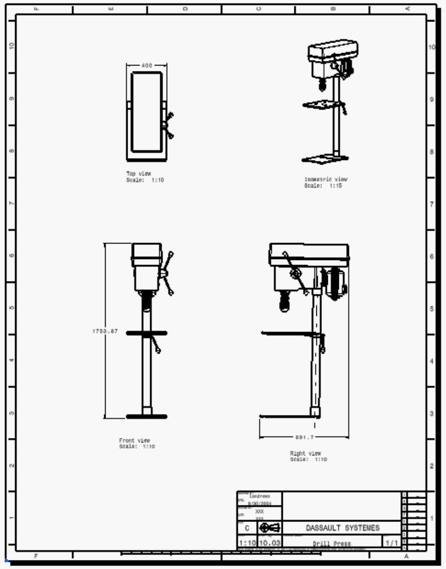


Master Project: Create an Assembly Drawing

Drill Press Assembly

20 min

The objective of this step is to create a drawing of the drill press 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:

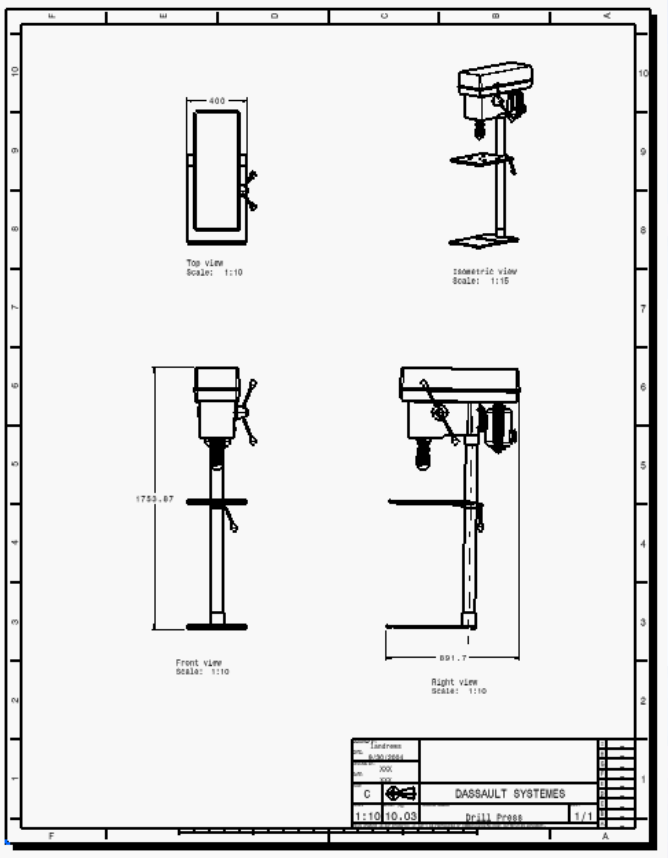
- **Create a new drawing**
- **Create three main views**
- **Add dimensions**
- **Apply title block**
- **Save a file**

Master Project: Create an Assembly Drawing

Create a drawing of the drill press assembly as shown. If you have not completed step 4, use Step4_complete.CATProduct.

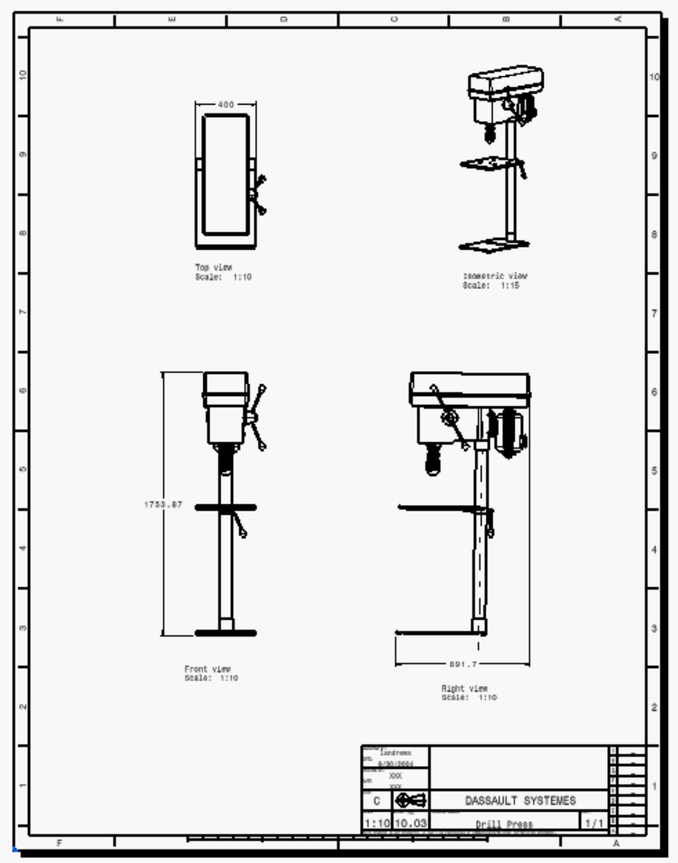
Use the following criteria:

- **Standard: ANSI**
- **Format: C ANSI**
- **Orientation: Portrait**
- **Sheet Scale: 1:10**
- **Isometric view's scale 1:15**
- **Title Block: Drawing_TitleBlock_Sample1.**



Master Project: Create an Assembly Drawing Recap

- ✓ Create a new drawing
- ✓ Create three 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 between and including the 2 selected elements
Shift F2	Overview of the specification tree		
F3	Hide/Show the specification tree		
Ctrl + Tab	Change CATIA V5 window	**Alt** F8	Macros
		Alt F11	Visual Basic editor
Ctrl N	New file	**Alt** + Enter	Properties
Ctrl O	Open file	**Alt** + MB1	Pre-selection Navigator
Ctrl S	Save file	**Ctrl** F11	Pre-selection Navigator
Ctrl P	Print	**Up/Down** or **Left/Right** arrow	Pre-selection Navigator
Ctrl Z	Undo	**Shift** + MB2	Local zoom and change of viewpoint
Ctrl Y	Redo		
Ctrl C	Copy	**Shift** + manipulation with compass	Displacement respecting constraints
Ctrl V	Paste		
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.

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.

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

Glossary

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.

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).

Glossary

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.

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

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.

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.

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 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.

Glossary

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

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

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

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.

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.

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.

User Companion
CATIA | ENOVIA | DELMIA | SIMULIA | 3DVIA

Your everyday companion!

Companion is an essential tool which allows you to continuously enhance your skills and optimize your performance with Dassault Systemes products – right at your desk! The Companion includes theory, demonstrations, exercises, and methodology recommendations that enable you to learn proven ways to perform your daily tasks. Every release the Companion is updated by Dassault Systemes experts to ensure that your knowledge remains current.
For more details please visit www.3ds.com/education/

Show them what you know!

Get Certified!

Research shows, and industry experts agree, that an IT certification increases your credibility in the Information Technology workplace. It provides tangible evidence to show that you have the proficiency to provide a higher level of support to your employer. Are you ready to get certified and affirm the knowledge, skills, and experience you possess and gain a worldwide recognized credential leading to success?
For complete details please visit http://www.pearsonvue.com/dassaultsystemes/