Basic Features

In this lesson you will learn how to create basic CATIA features.

Lesson Contents:

- Case Study: Basic Features
- Design Intent
- Stages in the Process
- Determine a Suitable Base Feature
- Create Pad and Pocket Features
- Create Holes
- Create Fillets and Chamfers

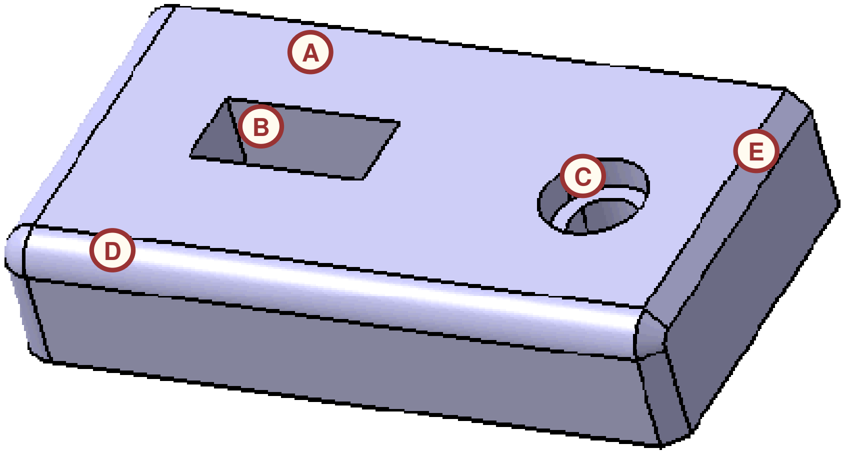
Duration: Approximately 0.33 day

Student Notes:

Basic Features in Part Design

Part design includes many features that help the user to create a model. The most common features will be introduced in this lesson:

A. Pad
B. Pocket
C. Hole
D. Fillet
E. Chamfer

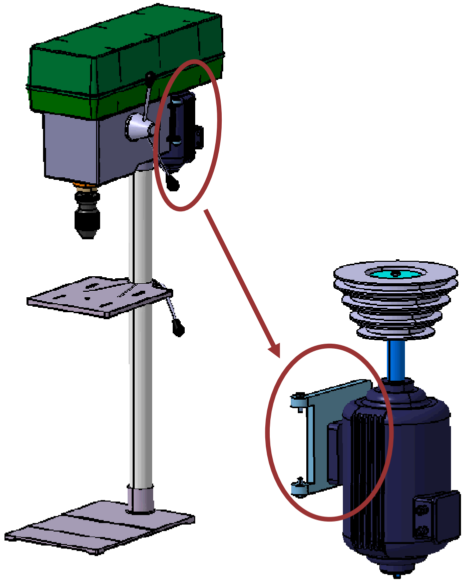


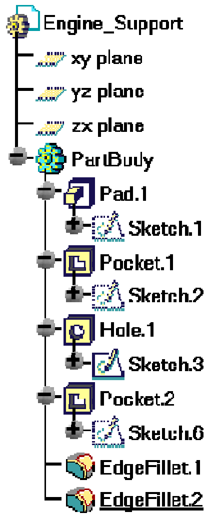
Student Notes:

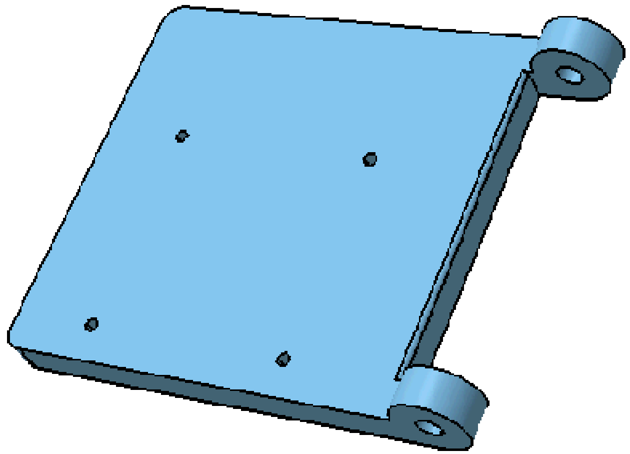
Student Notes:

Case Study: Basic Features

The case study for this lesson is the engine support used in the drill support assembly as is shown below. The engine support is part of the Block Engine sub-assembly. The focus of this case study is the creation of a feature that incorporate the design intent for the part. The engine support will consist of a pad, pockets, a hole, fillets, and a chamfer, and all these can be accessed using the Part Design Workbench.







Student Notes:

Design Intent

The engine support must meet the following design intent requirements:

✓ Internal loops must not be created in a sketch.

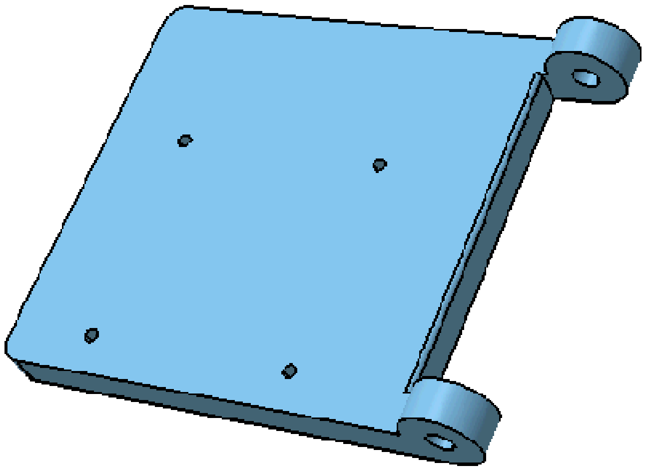
- Each element on this model must be created as a separate feature. This makes it easy to make modifications in the future.

✓ 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

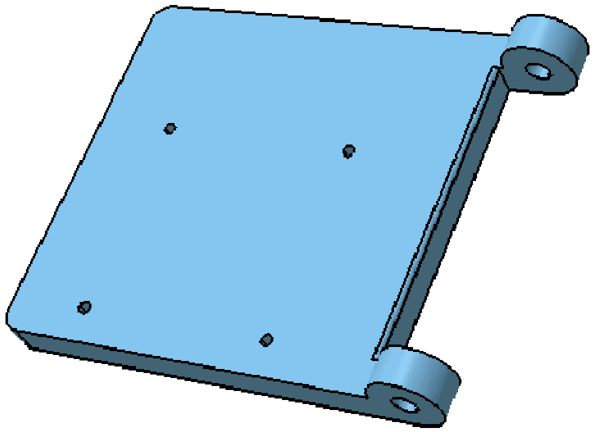


Student Notes:

Stages in the Process

Use the following steps to create the engine support:

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



Student Notes:

Determine a Suitable Base Feature

In this section you will learn how to create the base feature in a model.

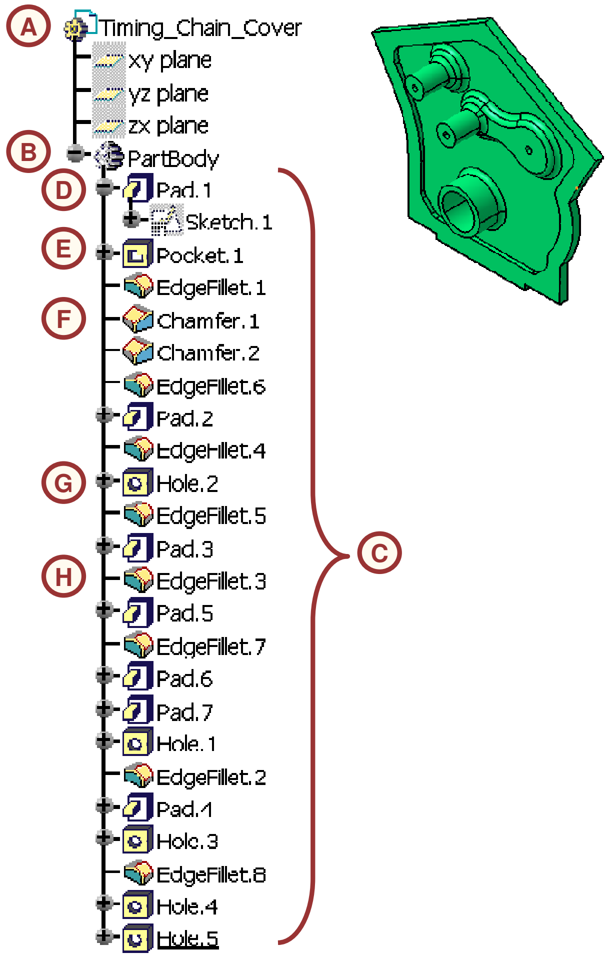
Use the following steps:

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

Student Notes:

Part Design Terminology

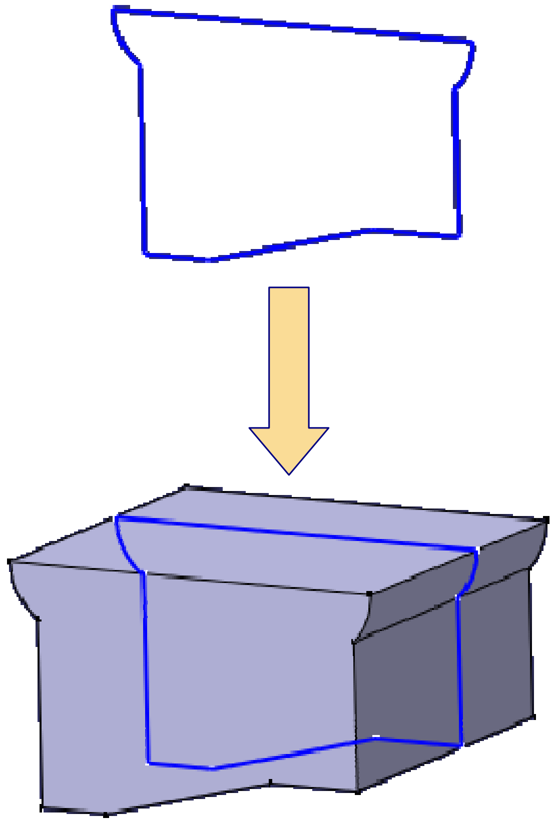
Term	Description
A. Part	The document containing the model. The document can consist of one or more features and bodies.
B. PartBody	A default container containing the features that make up a part.
C. Feature	Elements that make up a part. They can be based on sketches (sketch-based) or features that build on existing elements (dress-up and transformation). They can also be generated from surfaces (surface-based).
D. Pad	A solid feature created by extruding a sketched profile.
E. Pocket	A feature that removes material by extruding a sketched profile.
F. Hole	A feature that removes material through the extrusion of a circular profile.
G. Fillet	A curved surface of a constant or variable radius that is tangent to, and that joins two surfaces. Together, these three surfaces form an inside corner or outside corner.
H. Chamfer	A cut through the thickness of the feature at an angle, giving a sloping edge.



Student Notes:

Creating a Base Feature

It is important to begin with a strong *base feature*. Typically, this feature represents the primary shape or the foundation from which other geometries can be added/removed.



The base feature usually starts from a sketch or a surface element. This lesson describes how to create the base feature from a sketch.

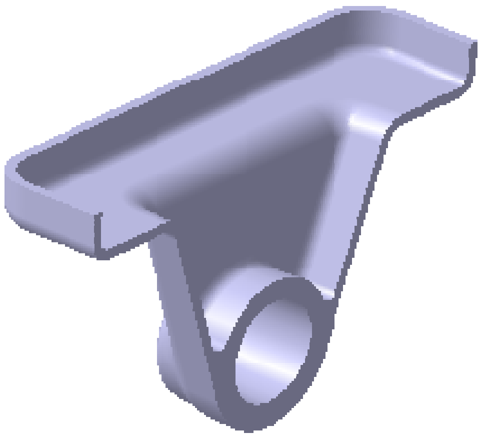
Student Notes:

Selecting a 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 select a base feature:

1. Identify the part features.
2. Select one feature to represent the base element.
3. Identify the CATIA tools (features) needed to create it.
4. Create the feature.

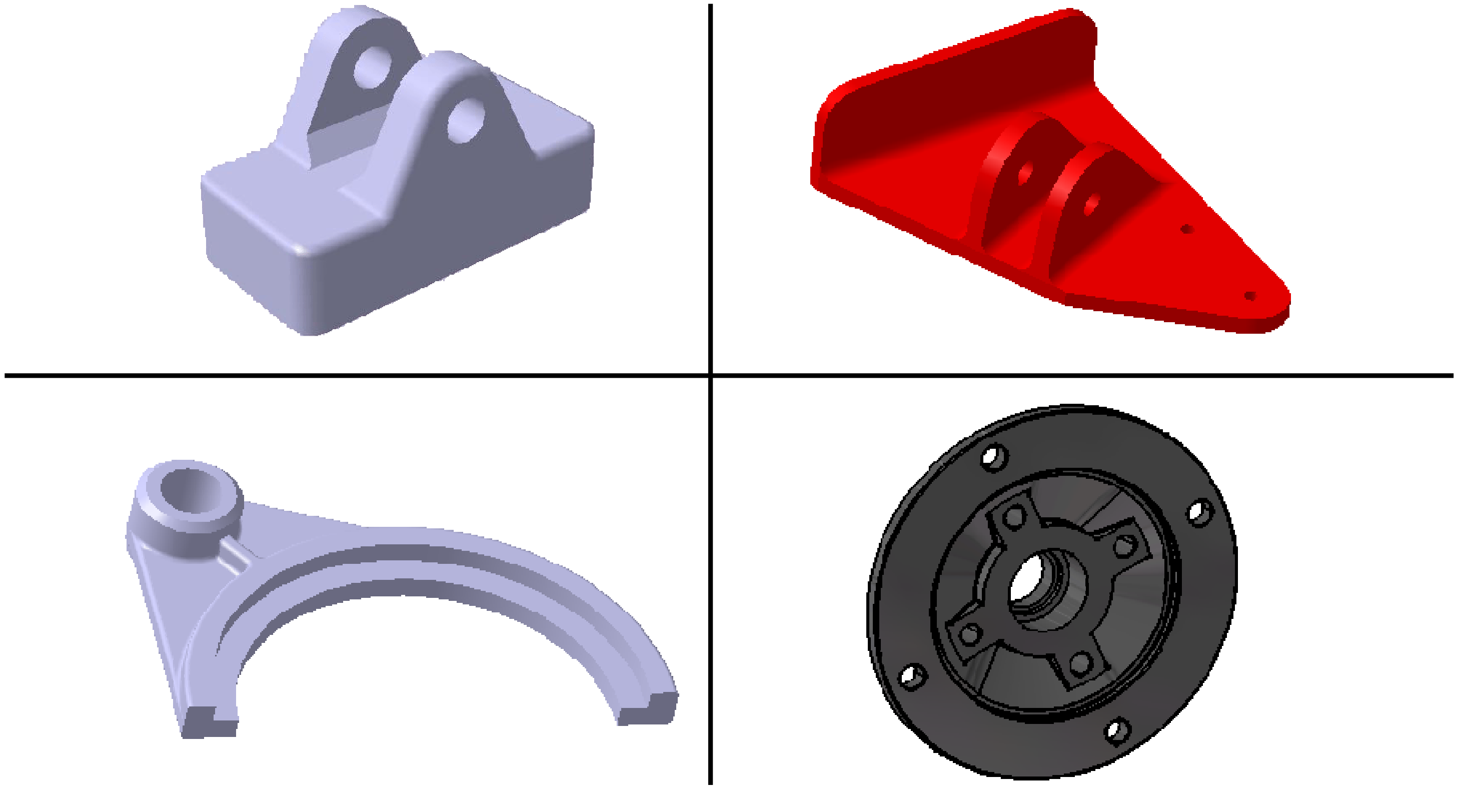


What would be the base feature for this part?

Student Notes:

Selecting a Base Feature - Exercise

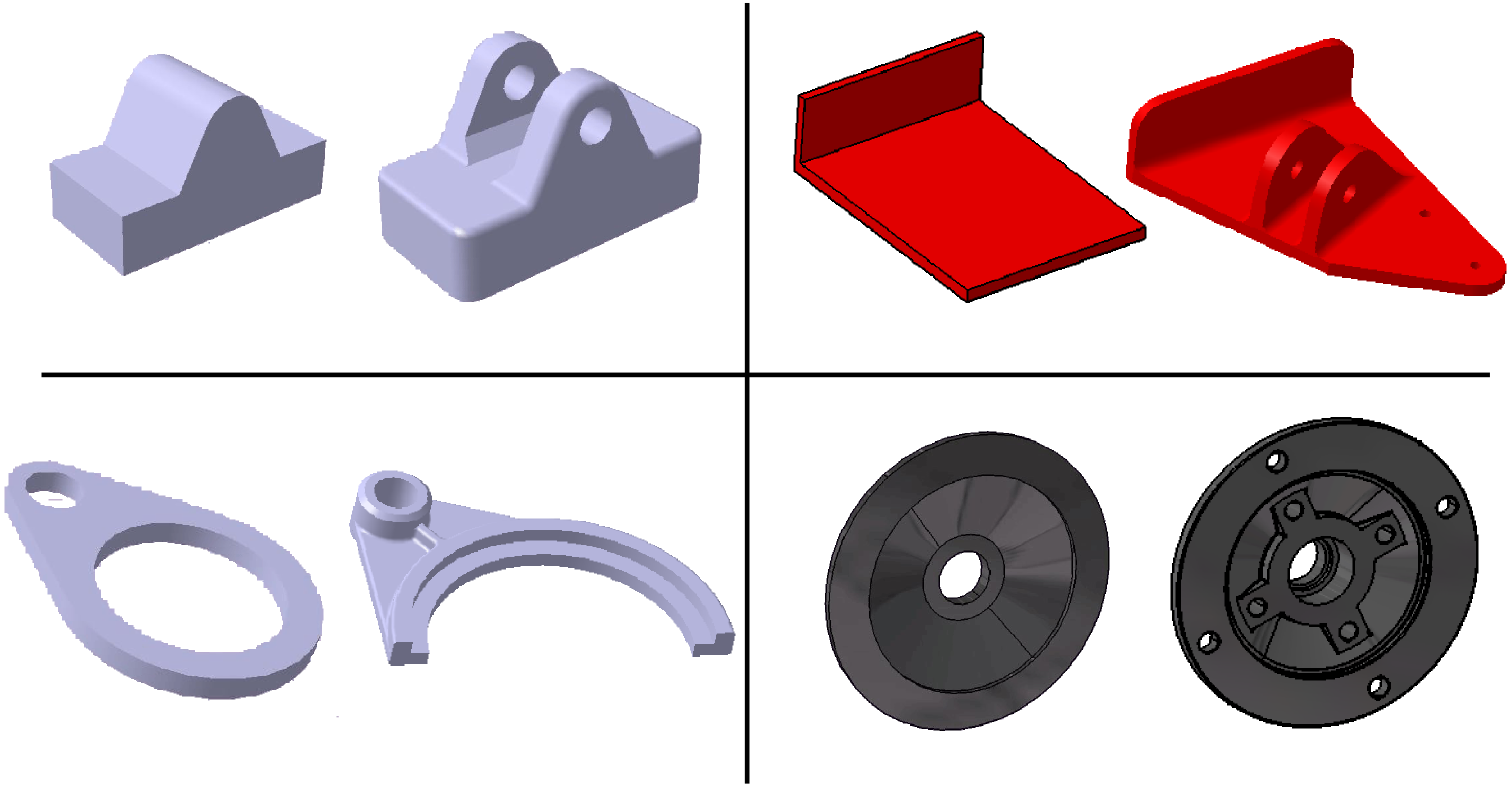
What would be the base feature for the following parts?



Student Notes:

Selecting a Base Feature - Answers

Here are some possible base features:

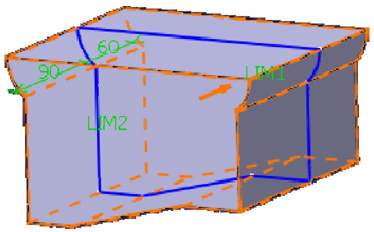


Student Notes:

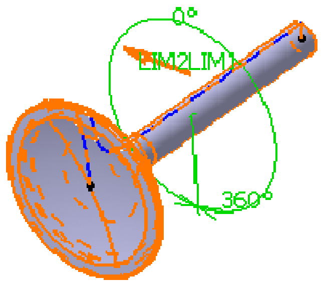
Features that Add or Remove Material (1/2)

Once the base feature is selected, it needs to be defined by *adding or removing material* to complete the design. The following is a list of features that add material:

- Pad (material added by extruding a sketch)
- Shaft (material added by revolving a sketch)
- Rib
- Multi-sections Solid
- Stiffener



Pad



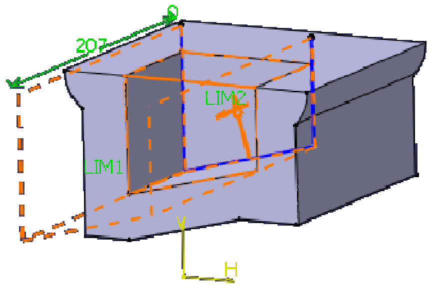
Shaft

Student Notes:

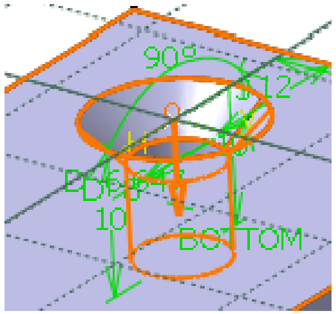
Features that Add or Remove Material (2/2)

The following is a list of features that remove material:

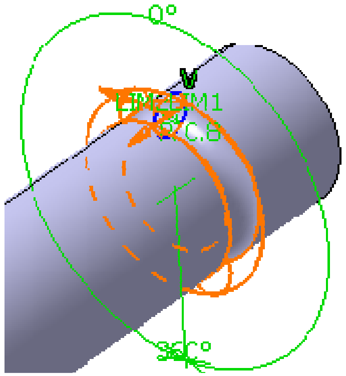
- Hole
- Pocket (material removed by extruding a sketch)
- Groove (material removed by rotating a sketch)
- Slot
- Removed Multi-sections Solid



Pocket



Hole

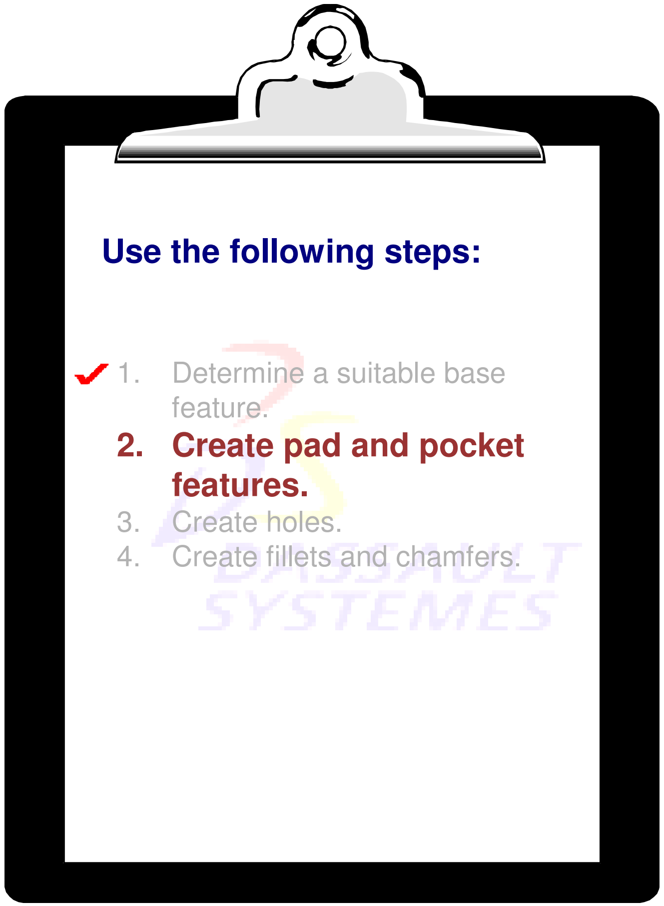


Groove

Student Notes:

Create Pad and Pocket Features

In this section, you will learn how to create simple pads and pockets from a 2D profile (or sketch).



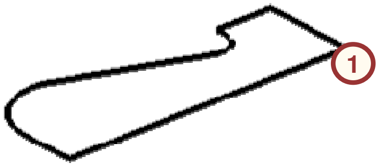
Student Notes:

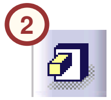
Creating Pads

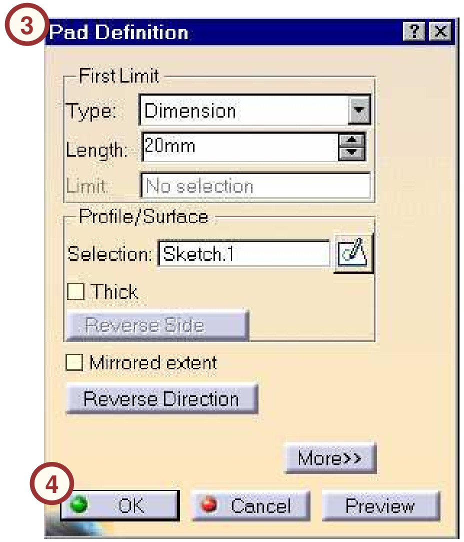
A pad is a sketched-based feature that adds material to a model.

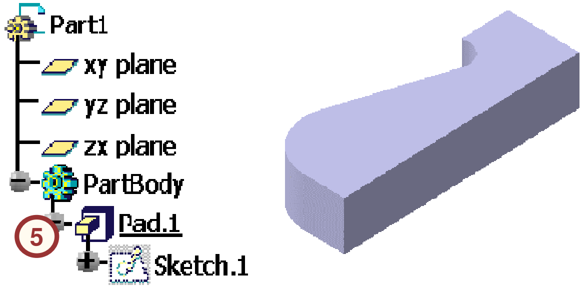
Use the following steps to create a pad feature:

1. Select the profile sketch.
2. Click the **Pad** icon.
3. Modify the pad definition.
4. Click **OK** to complete the feature. The pad feature is added to the specification tree. The profile sketch is moved under the pad in the tree.







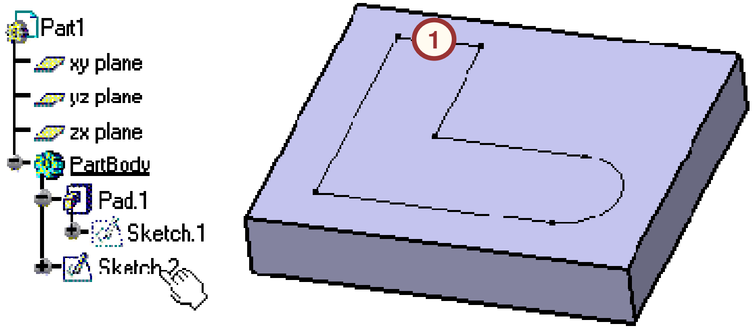


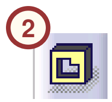
Student Notes:

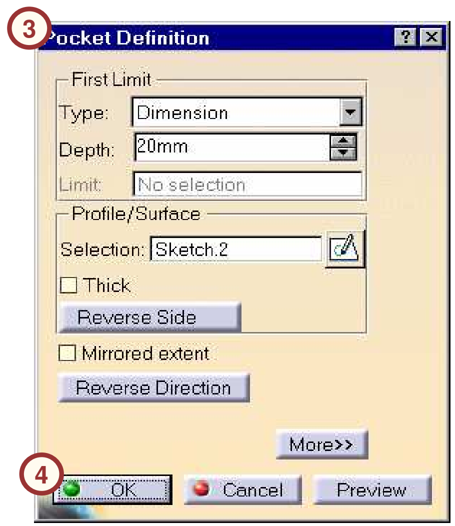
Creating a Simple Pocket

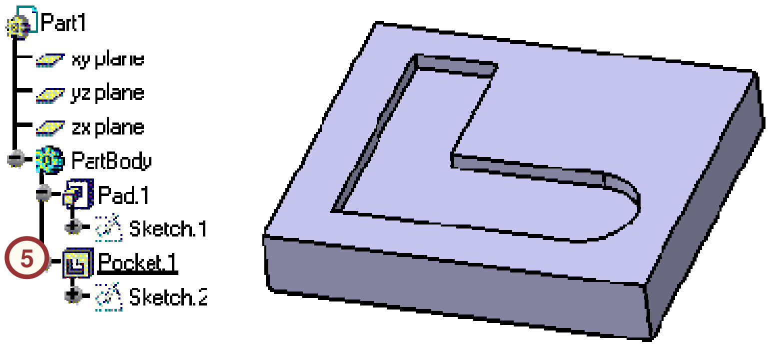
A pocket is a sketched-based feature that removes material from a model. Use the following steps to create a pocket feature:

1. Select the profile sketch.
2. Click the **Pocket** icon.
3. Modify the pocket definition.
4. Click **OK** to complete the feature. The pocket feature is added to the specification tree. The profile sketch is moved under the pocket in the tree.









Student Notes:

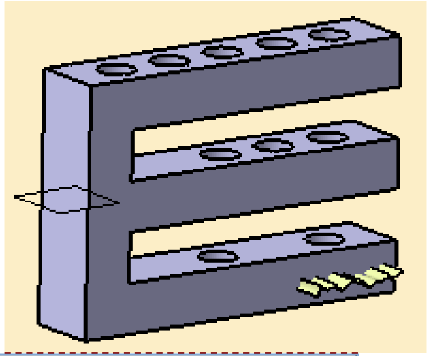
Pad and Pocket Limits

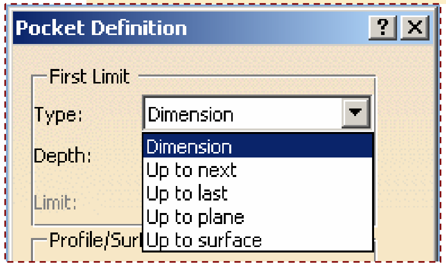
The length of a pad or pocket can be defined by dimensions or with respect to existing 3D limiting elements. If the pad/pocket feature is defined by a limiting element, it becomes associative to that element.

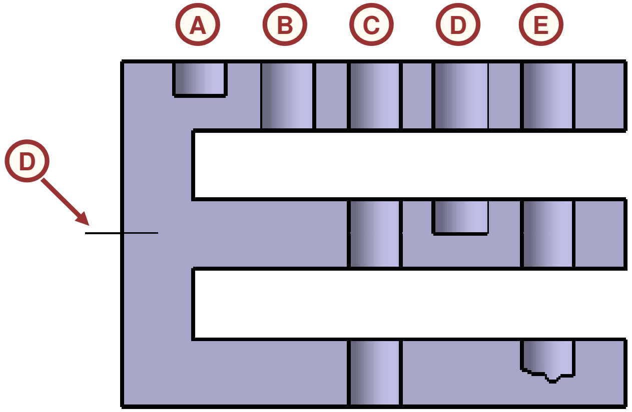
The following are types of depth options:

A. **Dimension**
B. **Up to Next**
C. **Up to Last**
D. **Up to Plane**
E. **Up to Surface**

Example

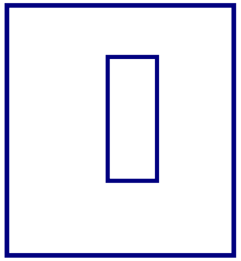
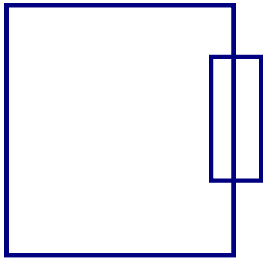
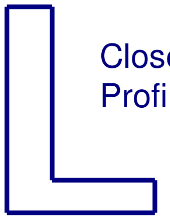






Restrictions for Pad/Pocket Profile Sketches

In general, 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.

Valid Sketch	***Invalid Sketch***	***Notes***
		Multiple profiles are acceptable, but they cannot intersect unless the **Thick** option is used.
 Closed Profile	 Open Profile  Multiple Open Profile	Open profiles cannot be used as the base feature of a part, unless the **Thick** option is used.

__Student Notes:__

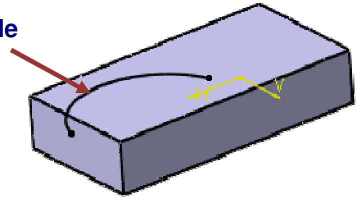
Student Notes:

Open Profiles

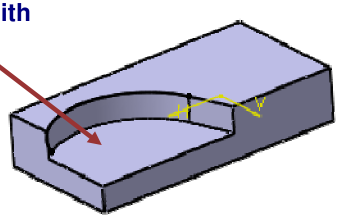
Open profiles can be used to create pads, pockets, or groove features. Consider using an open profile when the existing geometry is available to limit the new feature.

Using the existing geometry to re-limit a feature eliminates the need to create and constrain the additional sketched geometry. Always ensure that the re-limiting feature is stable. Major modifications or removal of the re-limiting feature will cause the profile to fail.

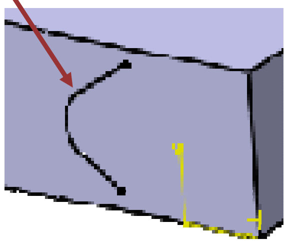
Open profile



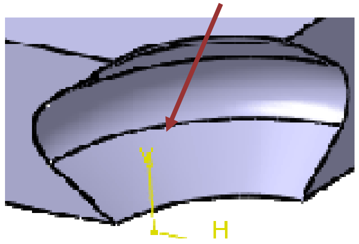
Pocket created with the open profile



Open profile



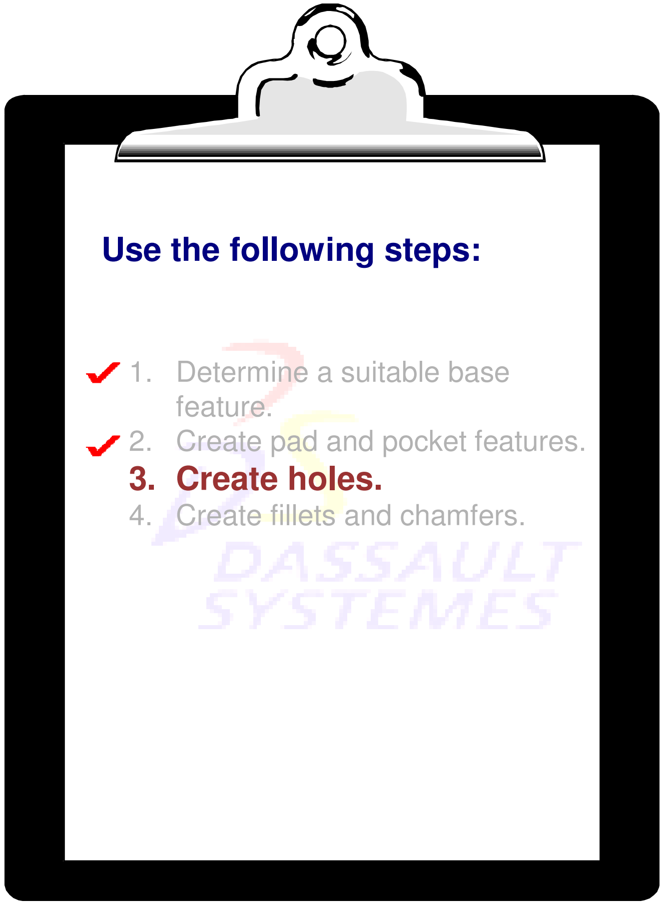
Groove created with the open profile



Student Notes:

Create Holes

In this section, you will learn how to create different types of holes and locate them on existing features.



Student Notes:

What is a Hole?

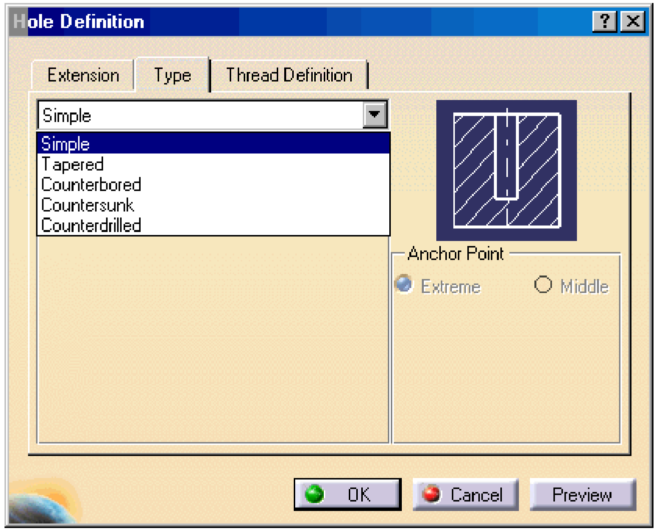
A hole removes circular material from an existing solid feature. A hole does not require a profile sketch. Like a pocket, its length can be defined using dimensions or with respect to the existing 3D elements.

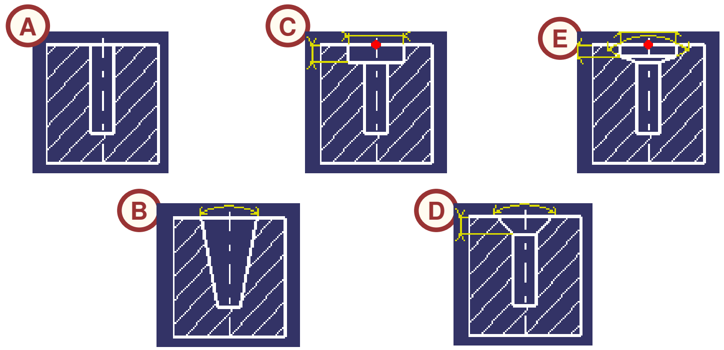
The hole type is defined using the **Type** tab of the **Hole Definition** dialog box. Several types of holes are available:

A. **Simple**
B. **Tapered**
C. **Counterbored**
D. **Countersunk**
E. **Counterdrilled**

Hole placement is defined using one of the two methods:

A. Placement using a positioning sketch.
B. Placement using pre-defined references.





Student Notes:

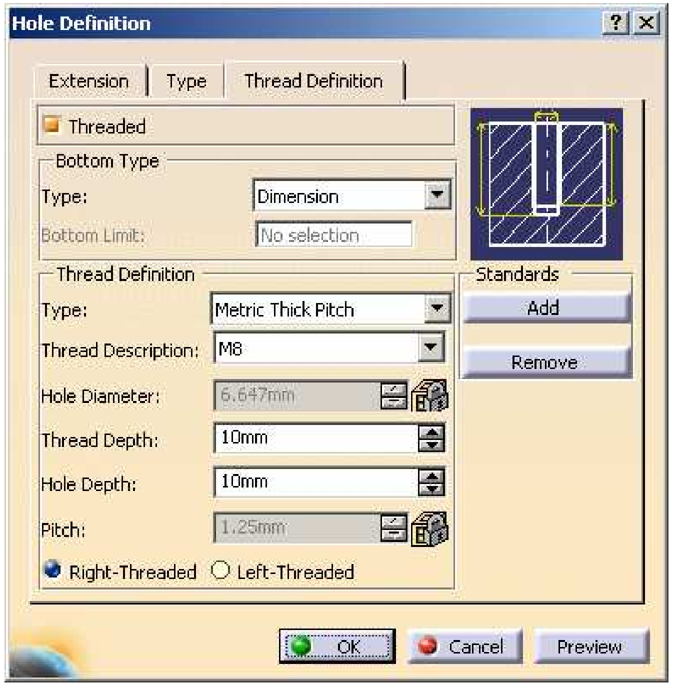
Using Pockets or Holes

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.

Threaded Holes

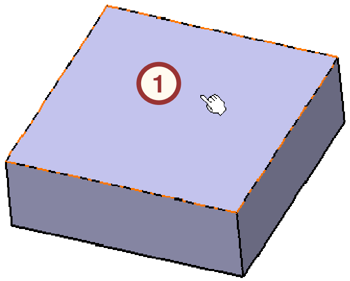


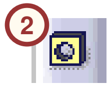
Student Notes:

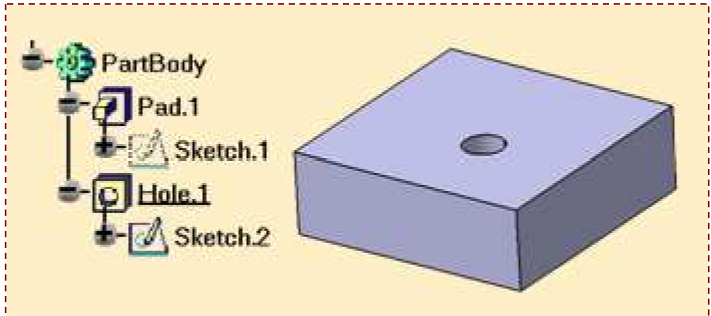
Hole Creation Using a Positioning Sketch

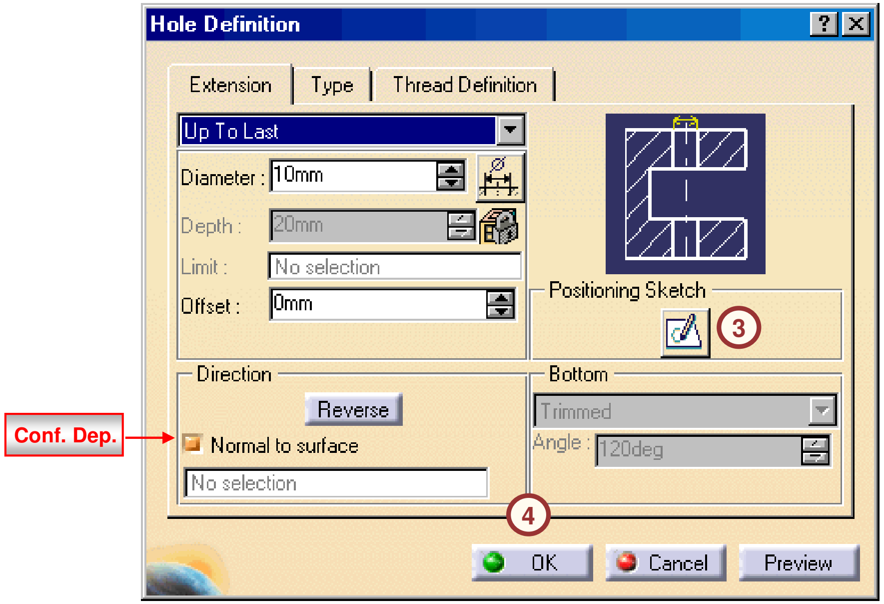
Use the following steps to define the hole placement using a positioning sketch:

1. Select a planar face on which the hole will be located.
2. Select the **Hole** icon.
3. Locate the center of the hole precisely inside the sketching workbench by selecting the **Positioning Sketch** icon.
4. Click **OK** to complete the feature. A sketch of the center point for the hole is automatically created under the hole feature in the specification tree.







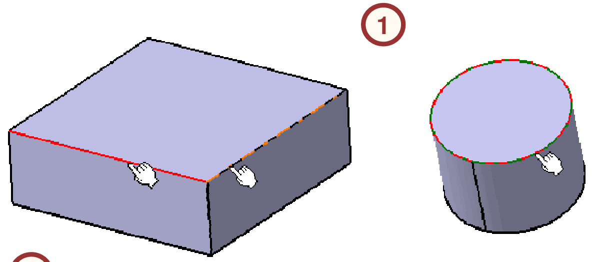


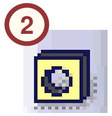
Student Notes:

Hole Creation Using Pre-defined References (1/2)

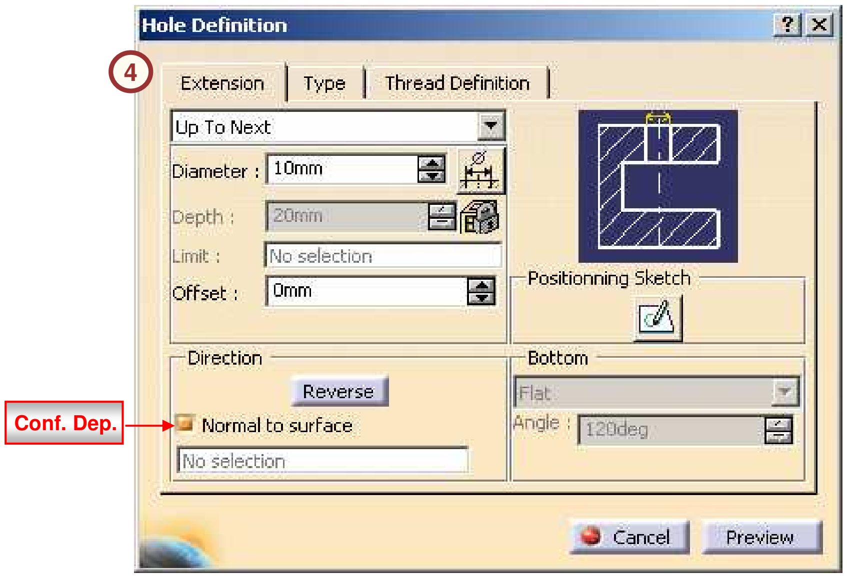
Use the following steps to define the hole placement using pre-defined references:

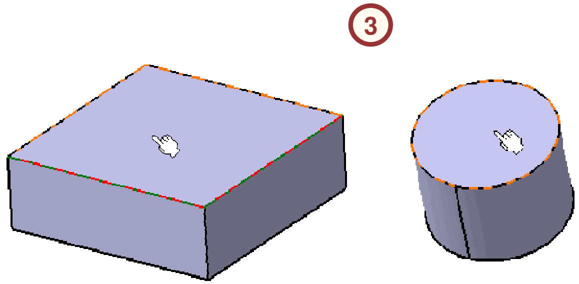
1. Multi-select two edges as linear positioning references. For a concentric hole, pre-select a circular edge as the reference.
2. Select the **Hole** icon.
3. Select the face where the hole will start.
4. Modify the hole definition.





The dialog box is same for a linear and concentric hole type.



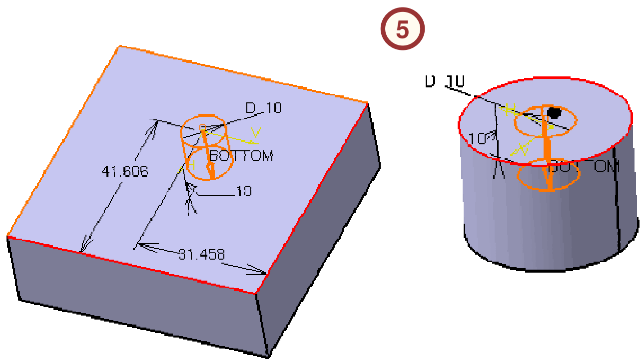


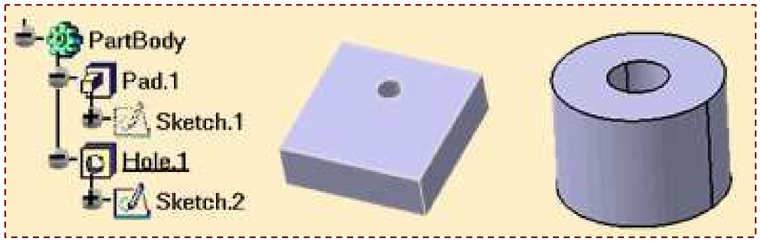
Student Notes:

Hole Creation Using Pre-defined References (2/2)

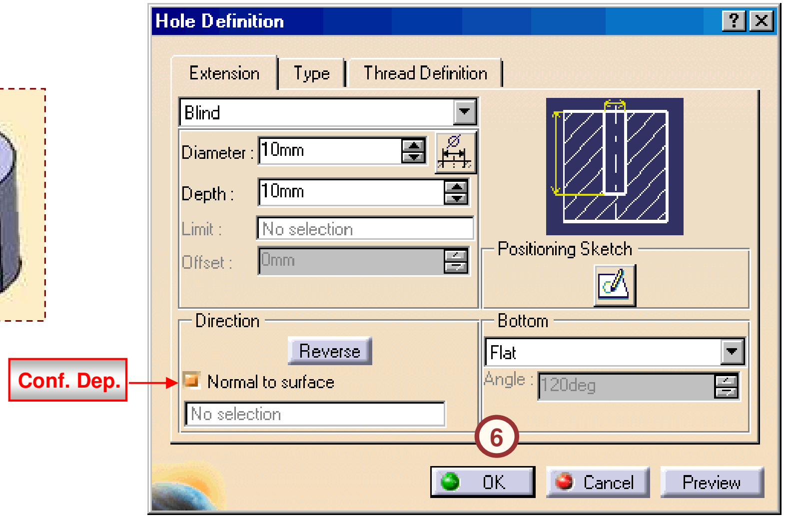
Use the following steps to define the hole placement using pre-defined references (continued):

5. Modify the reference dimensions by double-clicking on the dimensions. You can also modify the references by clicking the **Positioning Sketch** icon and editing the dimensions in the Sketcher workbench.
6. Click **OK** to complete the feature. The hole feature is added to the specification tree.





The specification tree will appear the same for a linear or concentric hole.



Student Notes:

Create Fillets and Chamfers

In this section, you will learn how to create fillets and chamfers.

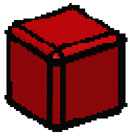
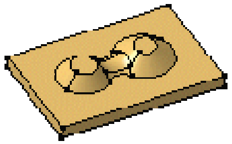
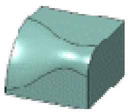
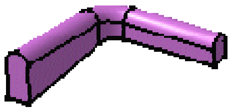
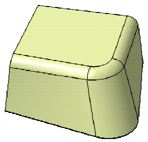
Use the following steps:

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

Student Notes:

What is a Fillet?

A fillet is a curved face of a constant or variable radius that is tangent to, and that joins, two surfaces. Together, these three surfaces form either an inside corner (fillet) or an outside corner (round). Several types of fillets are available in CATIA:

Type			Description
Edge			• Smooth transitional surfaces between two adjacent faces
Face-Face			• Used when there is no intersection between the faces or when there are more than two sharp edges between the faces
Variable			• Curved surfaces defined according to a variable radius
Tritangent			• Removes one of the three faces which are selected.
Chordal			• Controls the width of the fillet instead of radius.

Student Notes:

Selection and Propagation Modes

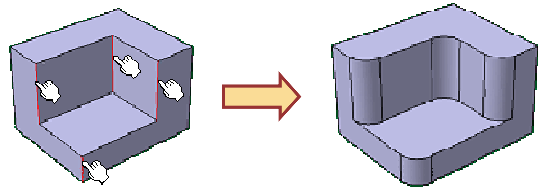
Edge Selection

Edges to be filleted can be selected using two different methods:

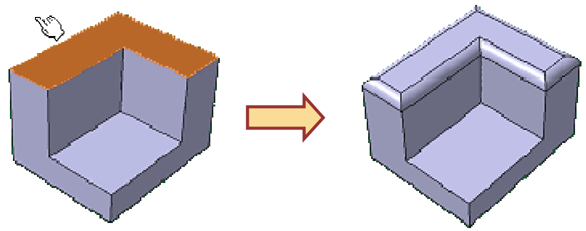
A. Select individual edges.

B. Select surfaces – Edges associated with the surface will be filleted (including internal edges).

A



B



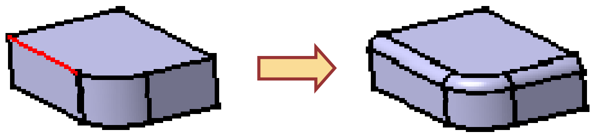
Propagation modes

While creating a fillet, you can use two different propagation modes:

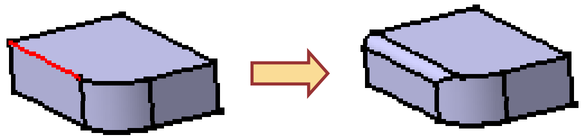
C. With the **Tangency** mode, the fillet is applied to the selected edge and all the edges tangent to the selected edge.

D. With the **Minimal** mode, the fillet is applied only to the selected edge.

C



D

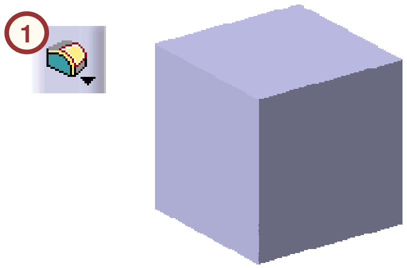


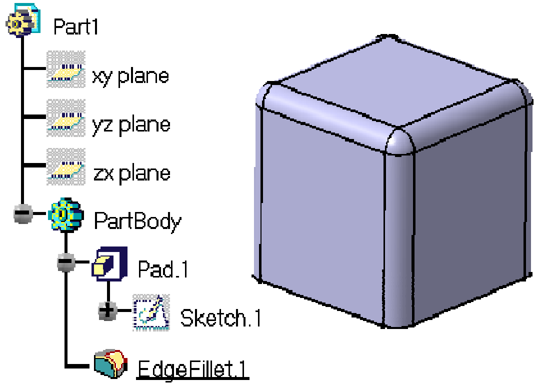
Student Notes:

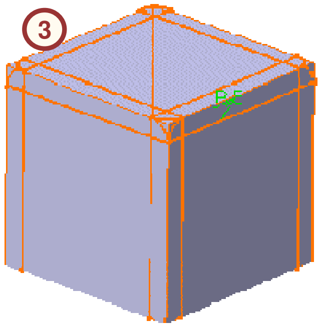
Filleting an Edge

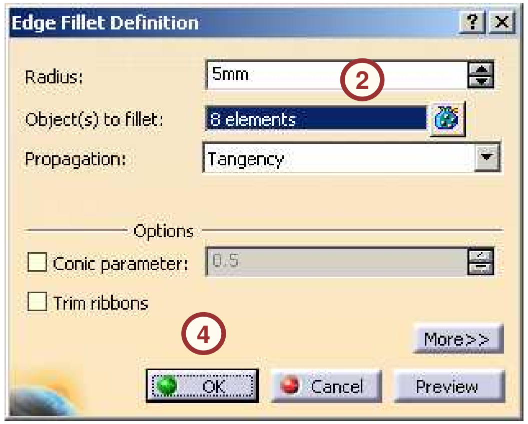
An edge fillet is a constant radius fillet that creates a smooth transitional surface between two adjacent faces. Use the following steps to create an edge fillet:

1. Click the **Edge Fillet** icon.
2. Specify the fillet radius.
3. Select the objects to fillet.
4. Click **OK** to complete the feature. The edge fillet is added to the specification tree as a separate feature.







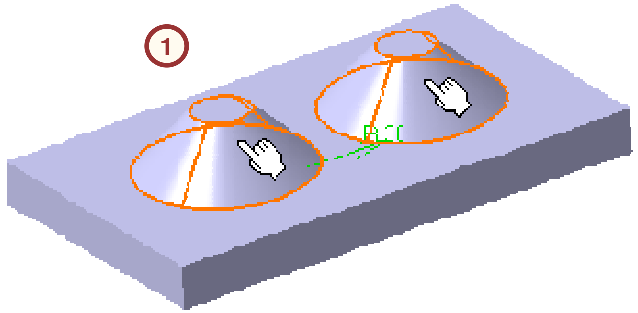


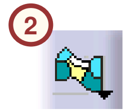
Student Notes:

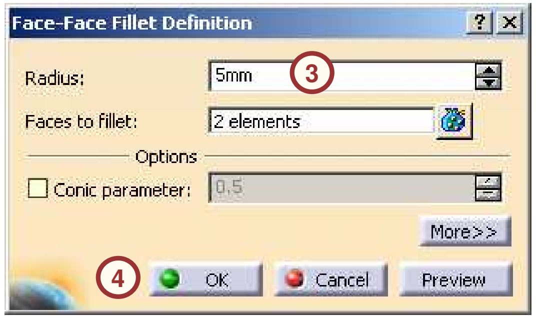
Face-Face Fillets (1/2)

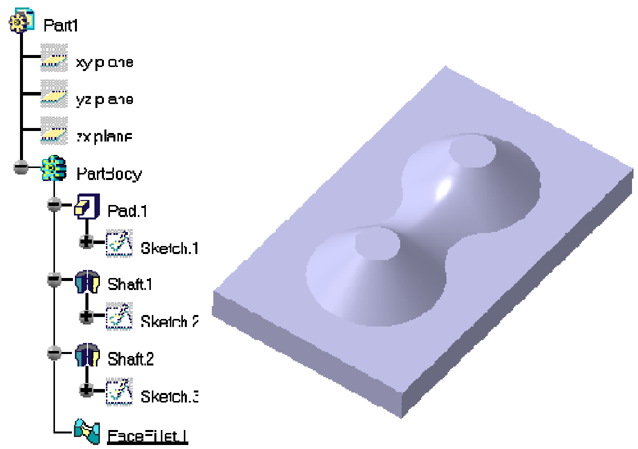
A face-face fillet is used when there is no intersection between the faces, or when more than two sharp edges exist between the faces. Use the following steps to create a face-face fillet:

1. Multi-select faces to be filleted.
2. Click **Face-Face Fillet** icon.
3. Specify the fillet radius.
4. Click **OK** to complete. The edge fillet is added to the specification tree as a separate feature.







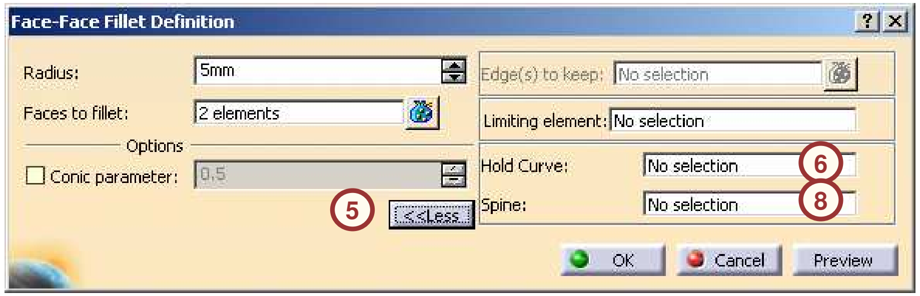


Student Notes:

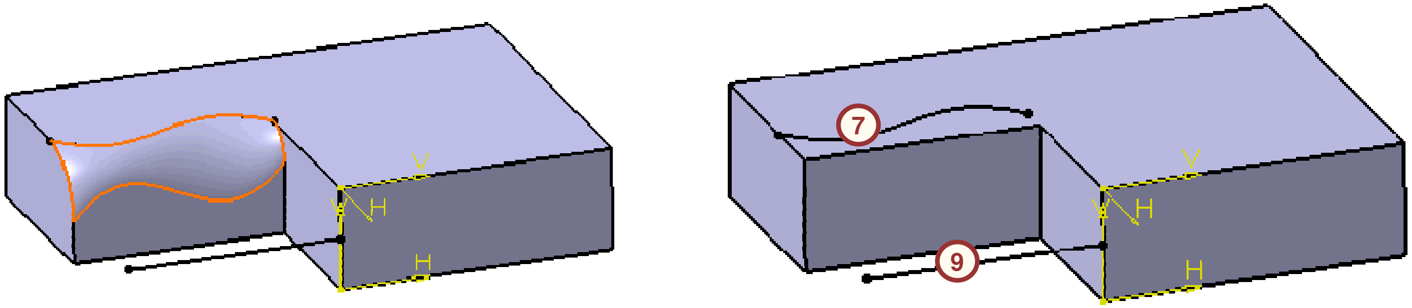
Face-Face Fillets (2/2) Conf. Dep.

Instead of specifying the radius value, the fillet radius can also be defined using a hold curve:

5. Expand the Dialog box to access the **Hold Curve** option.
6. Click on the **Hold Curve** field.
7. Select the curve.
8. Click on the **Spine** field.
9. Select the curve.



Result:

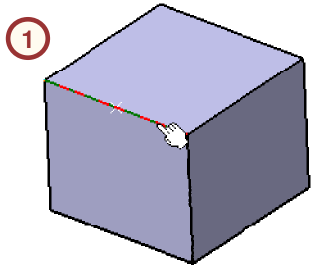


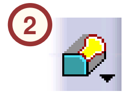
Student Notes:

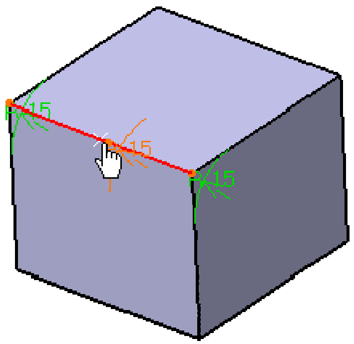
Variable Radius Fillets (1/2)

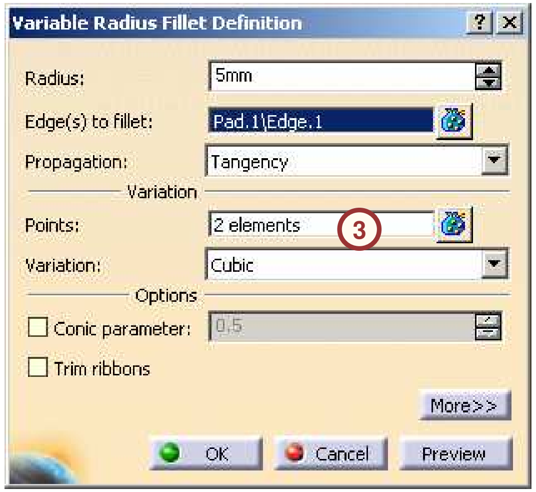
A variable radius fillet creates a curved surface defined according to a variable radius. Use the following steps to create a variable radius fillet:

1. Select the edge(s) to be filleted.
2. Click the **Variable Radius Fillet** icon
3. If required, click in the **Points** field and click additional variation points between the start and endpoints.





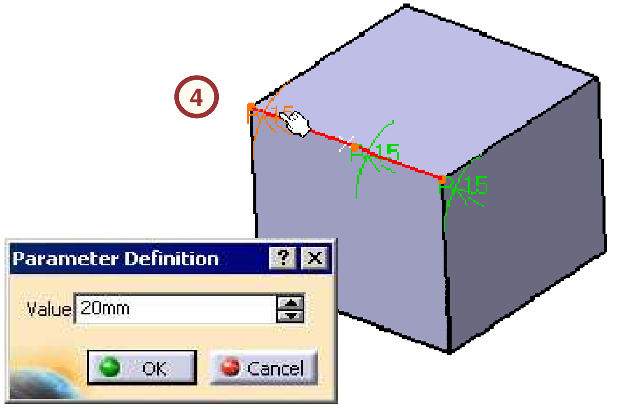


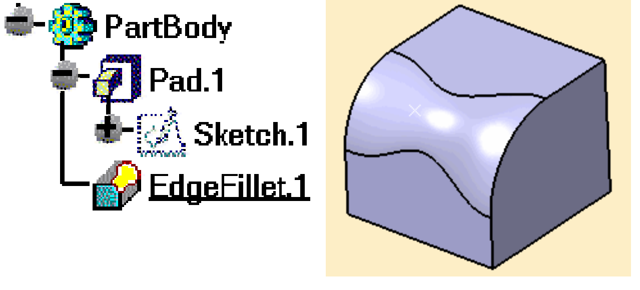


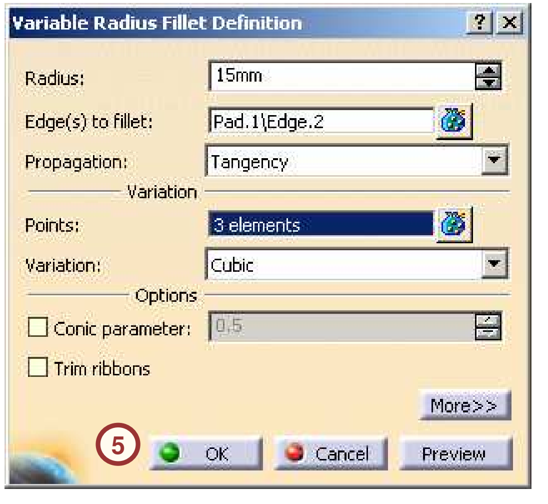
Variable Radius Fillets (2/2)

Use the following steps to create a variable radius fillet (continued):

4. Modify the radius at the points by double-clicking on the dimensions.
5. Click **OK** to complete. The edge fillet is added to the specification tree as a separate feature.







Student Notes:

Student Notes:

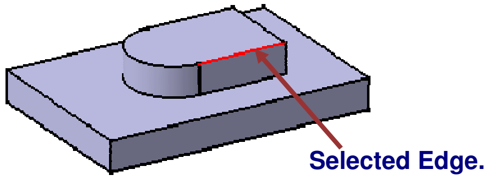
What is a Chamfer?

A chamfer removes or adds a flat section from a selected edge to create a beveled surface between the two original faces, which are common to that edge.

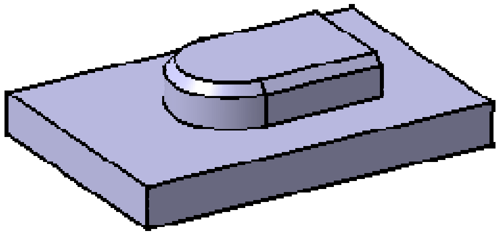
Like fillets, chamfers have two types of propagation options:

A. With the **Tangency** mode, the chamfer is applied to the selected edge and all the edges tangent to the selected edge.

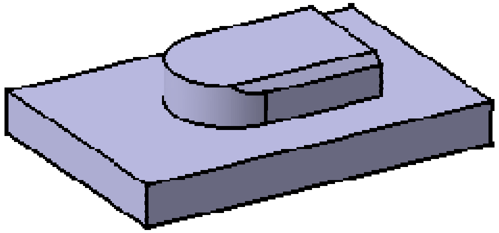
B. With the **Minimal** mode, the chamfer is applied only to the selected edge.



A



B



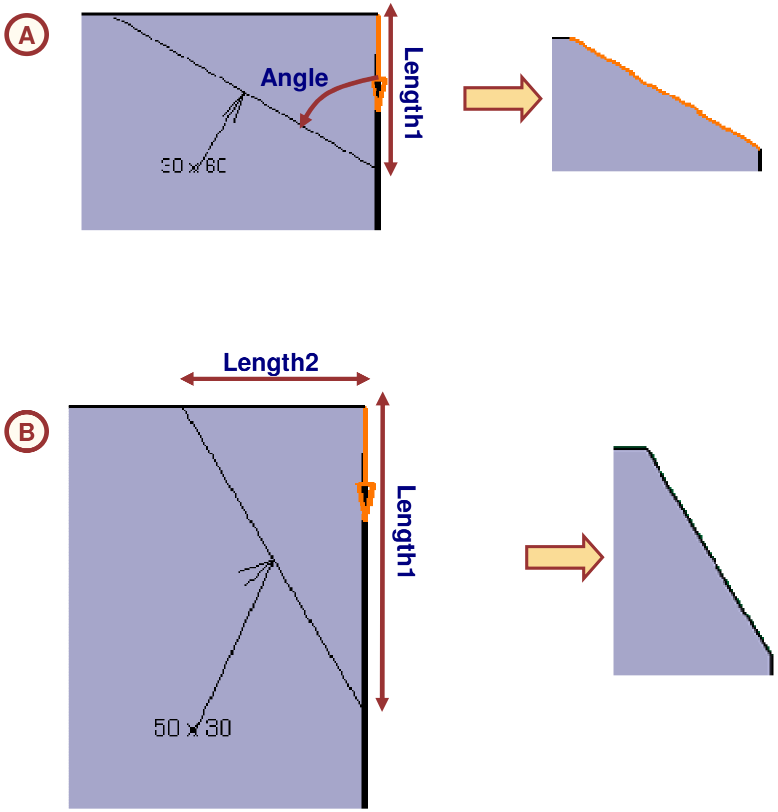
Student Notes:

Chamfer Dimensioning Mode

There are two dimensioning schemes available while creating a chamfer:

A. For **Length1**/**Angle**, the length is the distance along the selected edge to the edge of the bevel. The angle is measured with respect to Length1.

B. For **Length1**/**Length2**, the lengths are measured along the edges to be chamfered to the edge of the bevel.

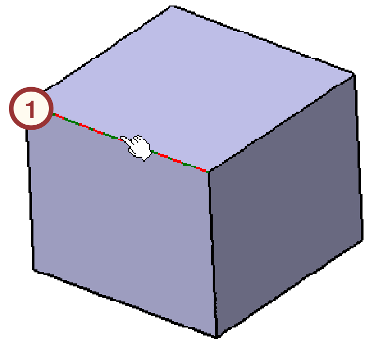


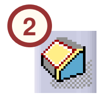
Student Notes:

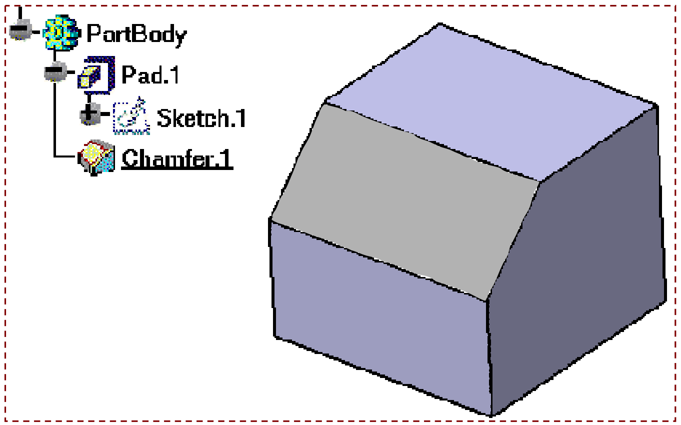
Creating a Chamfer

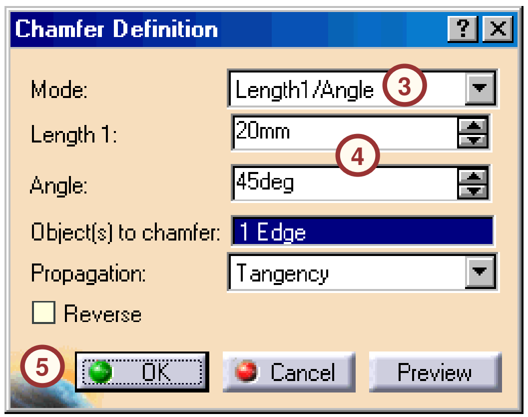
Use the following steps to create a chamfer:

1. Select the edge(s) to chamfer.
2. Click the **Chamfer** icon.
3. Select dimensioning scheme from the **Mode** menu.
4. Specify dimensional values.
5. Click **OK** to complete the chamfer. The chamfer is added to the specification tree as a separate feature.









Student Notes:

Recommendations for Fillets

In this section, you will be given a recommendation to help during the creation of fillets.

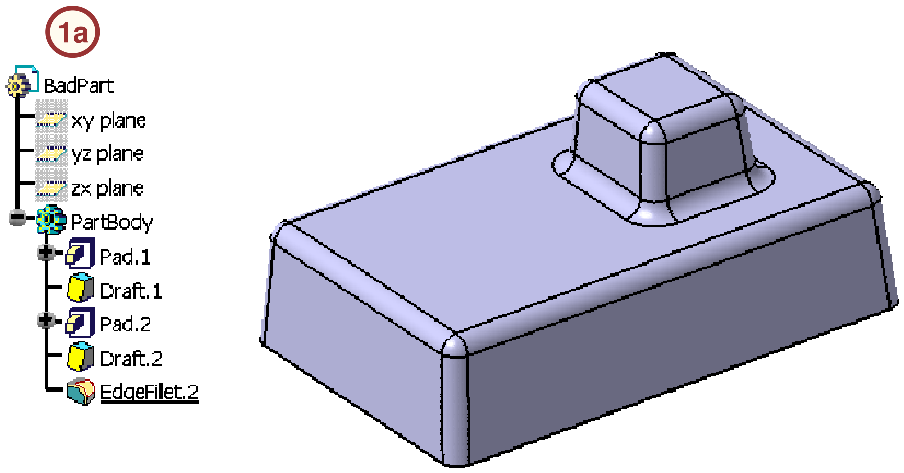
Student Notes:

Why One Fillet for Few Edges? (1/2)

It is recommended to group the edges according to the function and create the fillet.

In the example shown,

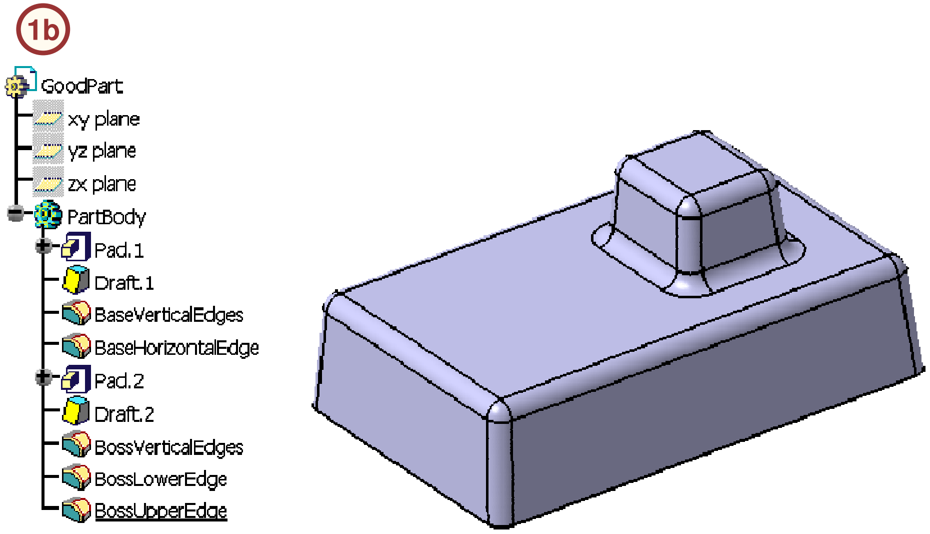
A. All the edges are grouped into a single fillet; therefore modification of the value of lower vertical edges cannot be done independently. These edges will have to be de-selected in the original fillet and a new fillet will have to be created.



Why One Fillet for Few Edges? (2/2)

Student Notes:

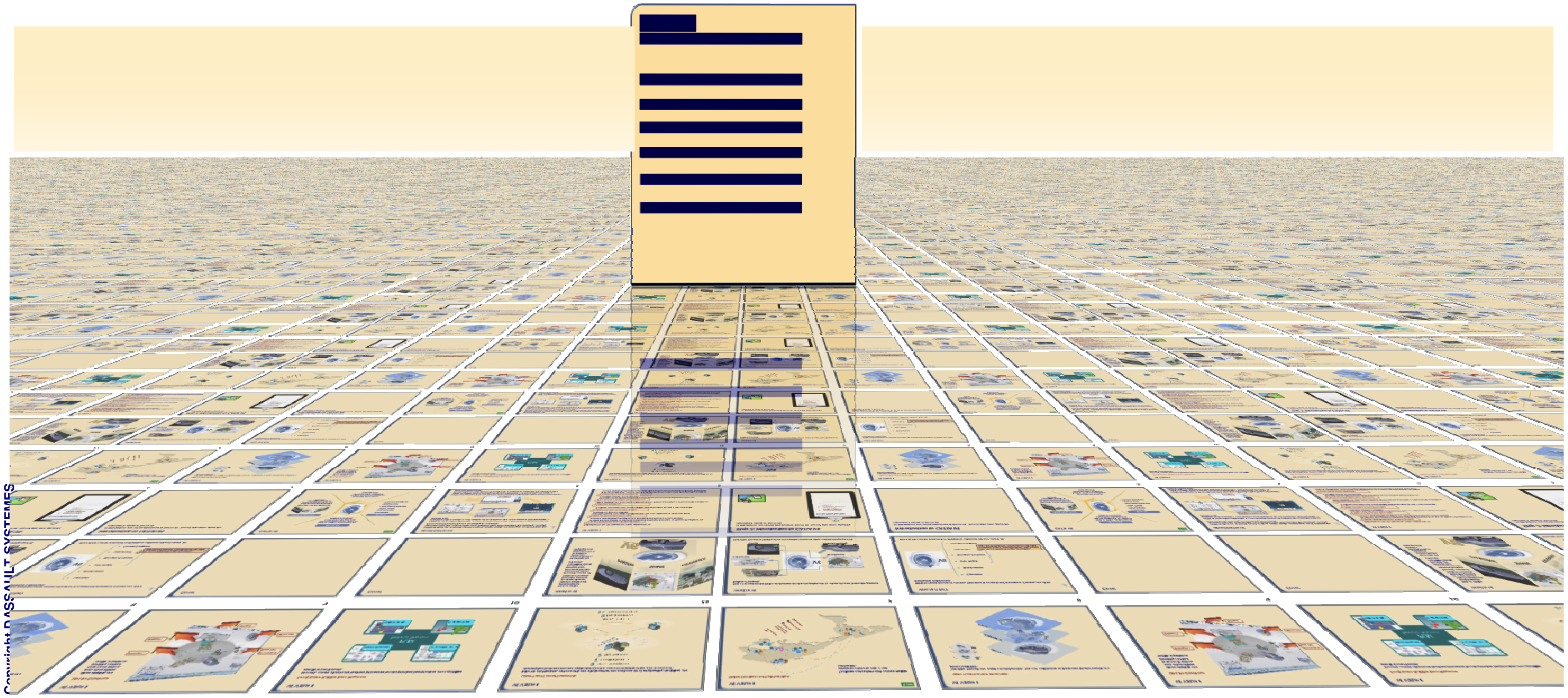
B. Edges are grouped by function; therefore the fillet radius for the lower vertical wall can be modified independently.



Student Notes:

To Sum Up

In the following slides you will find a summary of the topics covered in this lesson.



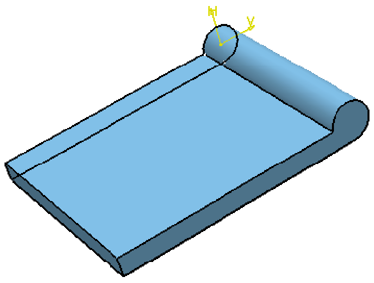
Determine a Suitable Base Feature

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

Student Notes:

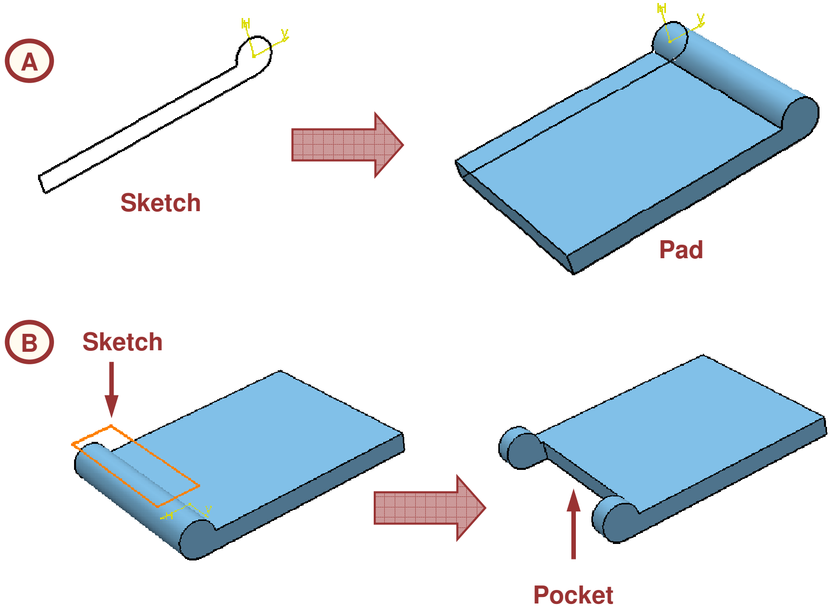
Student Notes:

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.



Student Notes:

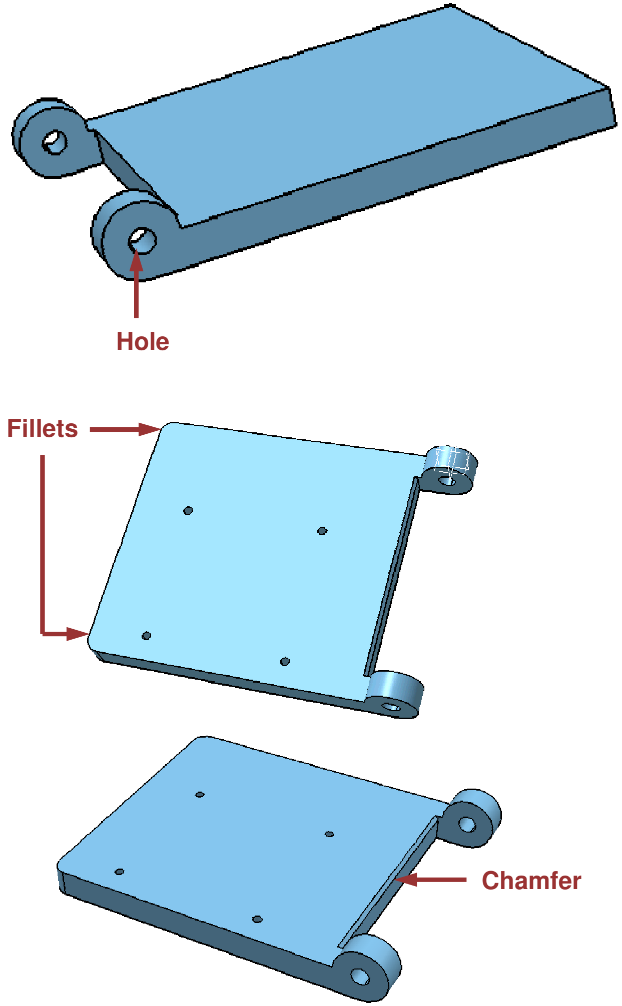
Create Holes

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.

Student Notes:

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

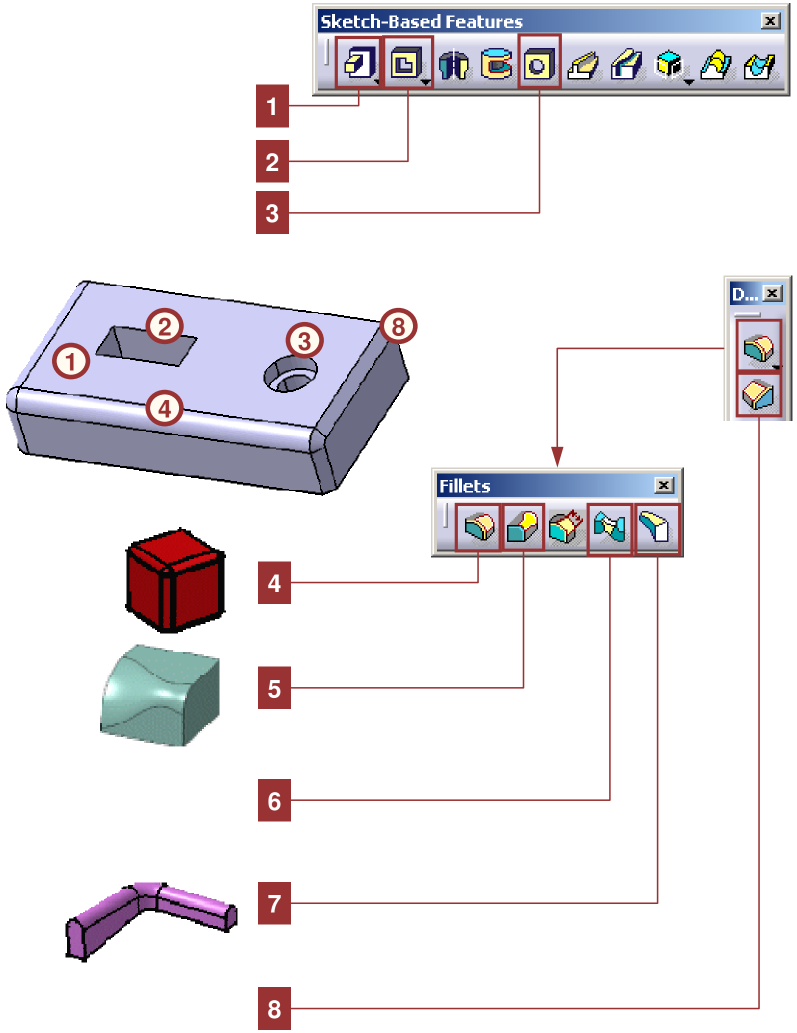
4 **Edge Fillet:** creates smooth transitional surfaces between two adjacent faces

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



Student Notes:

Exercise: Basic Features Creation

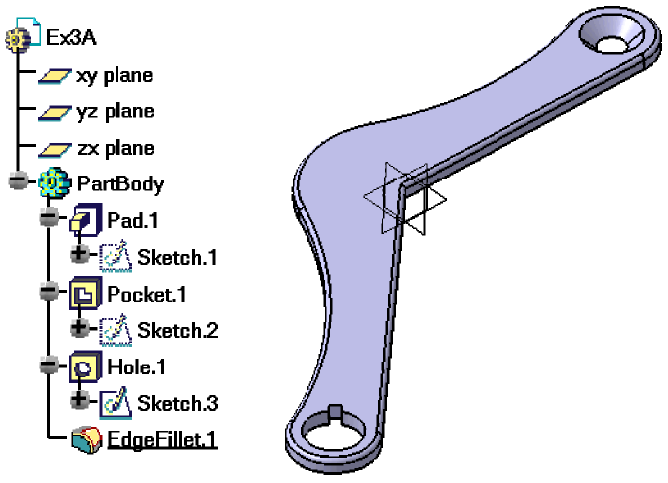
Recap Exercise

15 min

In this exercise you will load an existing part that contains two sketched profiles. You will use the tools learned in this lesson to create a pad, pocket, coaxial hole and fillet. Detailed instructions for this exercise are provided.

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

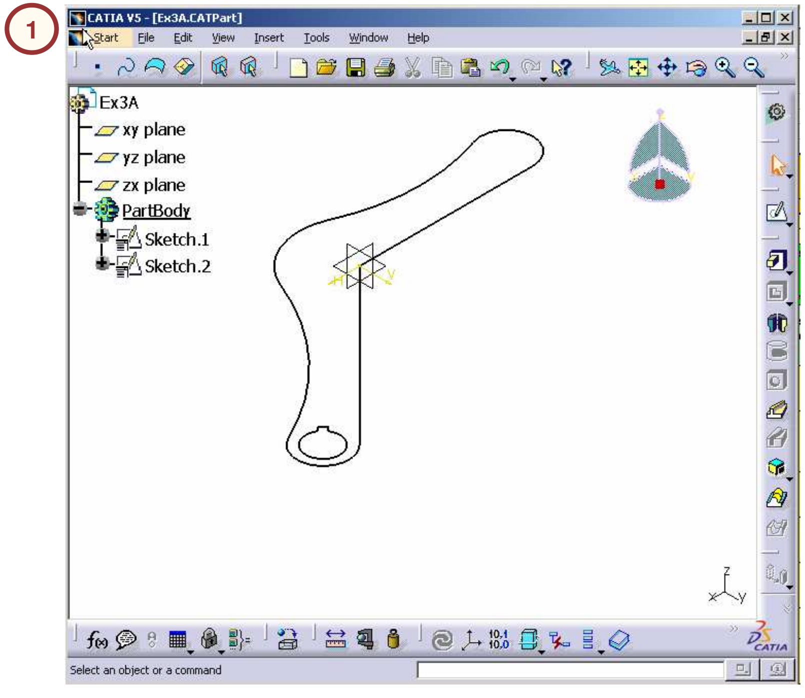
- Create a pad
- Create a pocket
- Create a coaxial hole
- Create an edge fillet



Student Notes:

Do it Yourself (1/7)

1. **Load Ex3A.CATPart.**
   - Load Ex3A.CATPart.

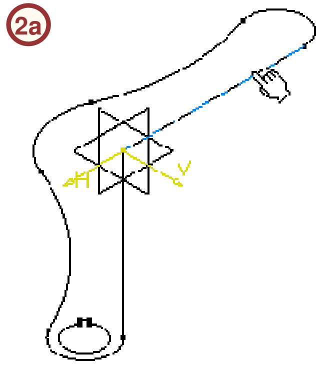


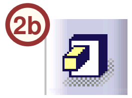
Student Notes:

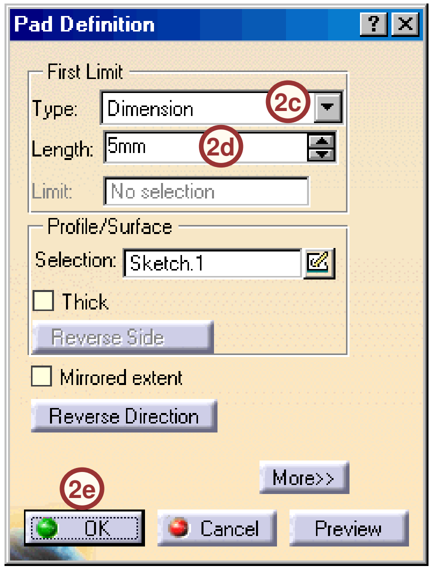
Do it Yourself (2/7)

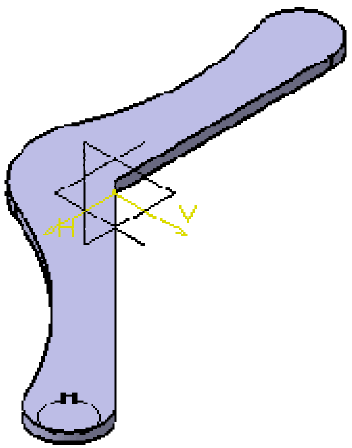
2. Create a pad.

- The first feature in a model must add material. Add a pad feature using Sketch.1 as the profile.
  a. Select Sketch.1
  b. Click the **Pad** icon.
  c. Select **Dimension** from the **Type** list.
  d. Type [5] for the length.
  e. Click **OK** to complete the feature.







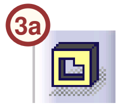


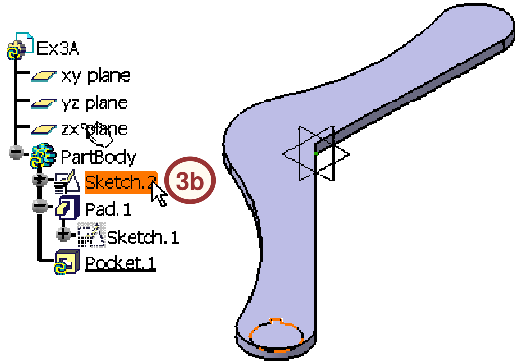
Student Notes:

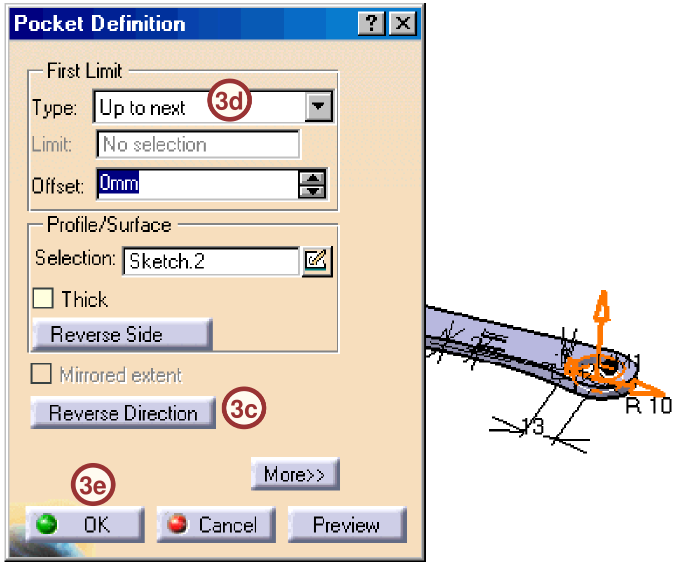
Do it Yourself (3/7)

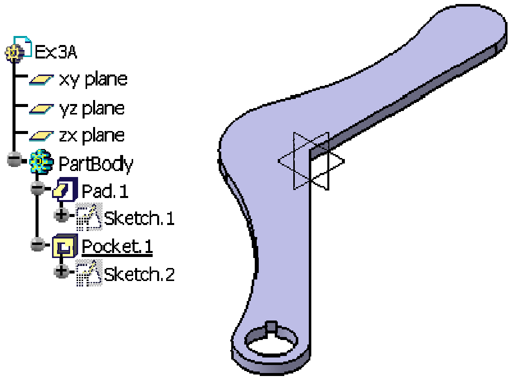
3. Create a pocket.

- Create a pocket using Sketch.2 as its profile.
  a. Click the **Pocket** icon.
  b. Select Sketch2.
  c. Ensure that the arrow in the preview is pointing upwards. This means that the material will be removed in this direction. If the arrow points downwards, select **Reverse Direction**.
  d. Select **Up to Next** from the **Type** list.
  e. Click **OK** to complete the feature.





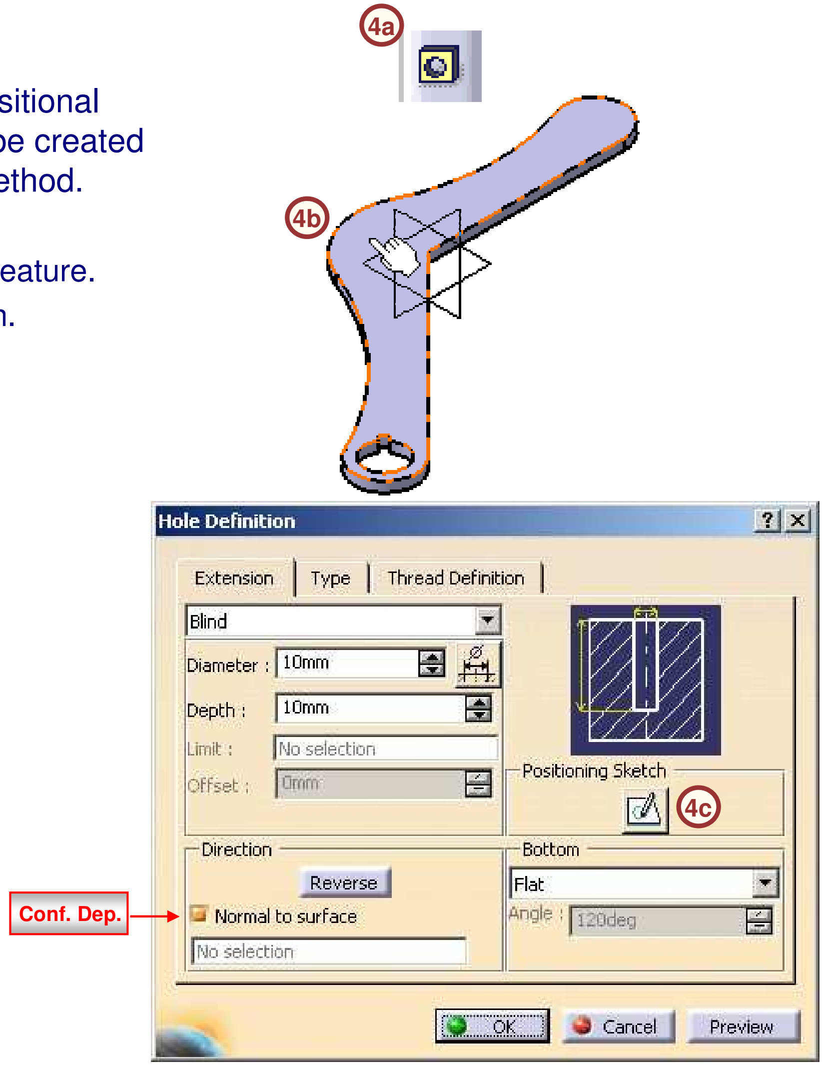




Do it Yourself (4/7)

4. Create a coaxial hole.

- Create a coaxial hole. Using the Positional Sketch method. The hole can also be created using the pre-defined references method.
  a. Click the **Hole** icon.
  b. Select the top surface of the pad feature.
  c. Click the **Positioning Sketch** icon.



Student Notes:

Student Notes:

Do it Yourself (5/7)

4. Create a Coaxial Hole (continued).

d. Select the hole center.

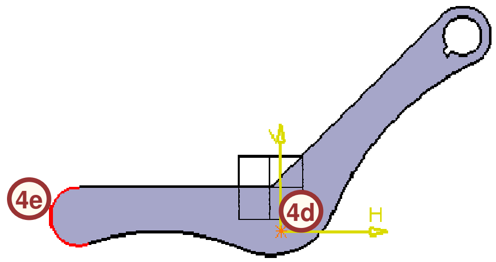
e. Press and hold the <Ctrl> key and select the edge of the arc.

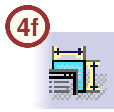
f. Click the **Constraints Defined in Dialog box** icon.

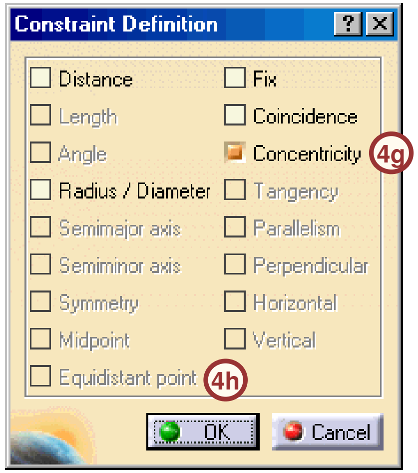
g. Select **Concentricity** constraint.

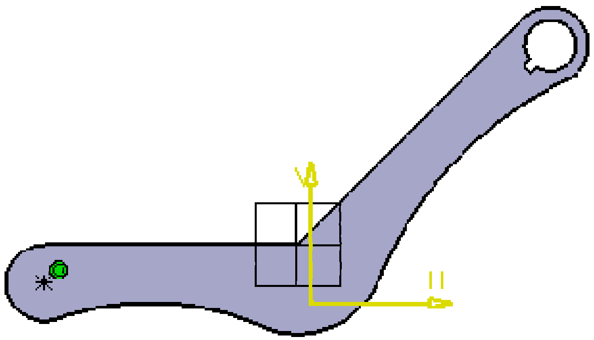
h. Click **OK**.

i. Exit the Sketcher.









Student Notes:

Do it Yourself (6/7)

4. Create a Coaxial Hole (continued…).

j. Change the depth of hole from **Blind** to **Up to Last**.

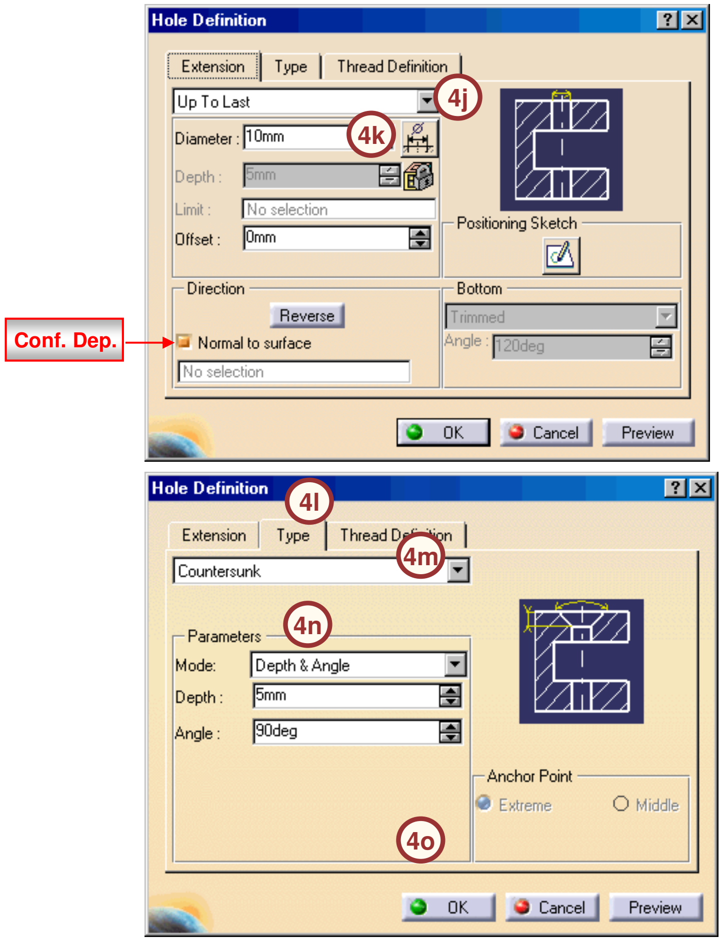
k. Specify a diameter of [10].

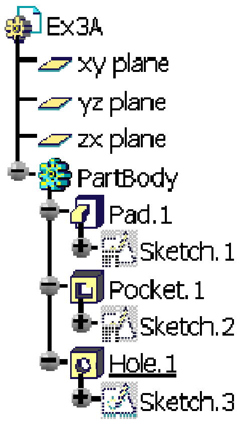
l. Select the **Type** Tab.

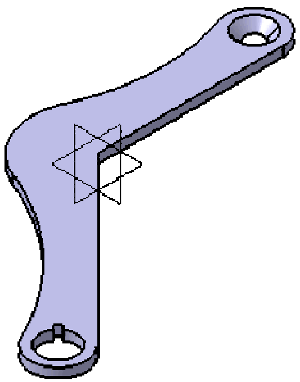
m. Change the hole type from **Simple** to **Countersunk**.

n. Type Depth [5] and Angle [90] values as shown.

o. Click **OK** to complete the feature.





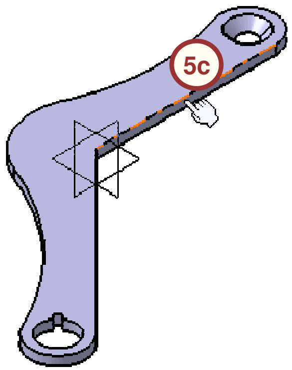


Student Notes:

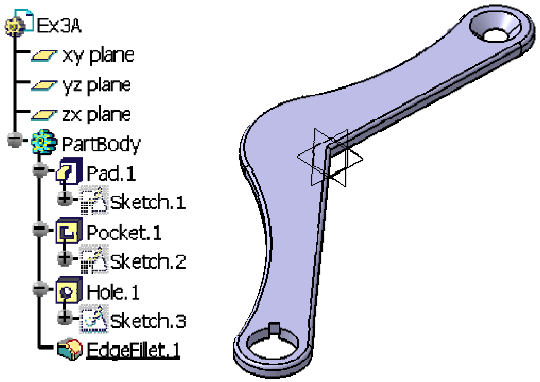
Do it Yourself (7/7)

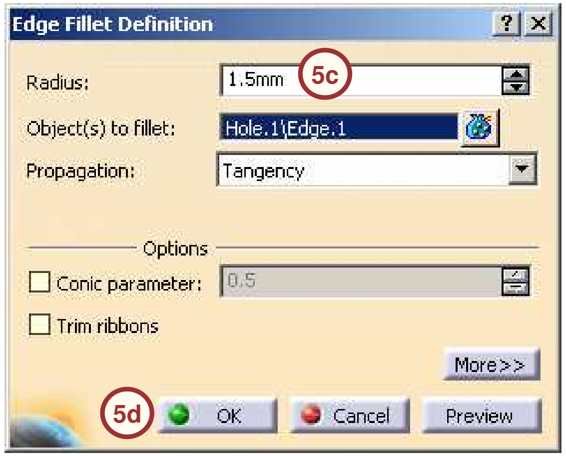
5. Create an edge fillet.

- Finally, apply an edge fillet to the outer edges of the pad feature using the **Edge Fillet** tool.
    a. Click the **Edge Fillet** icon.
    b. Select the edge as shown. Because of the tangency propagation type, all tangent edges are selected.
    c. Specify [1.5] for the radius value.
    d. Click **OK** to complete the feature.



6. Save and close the file.

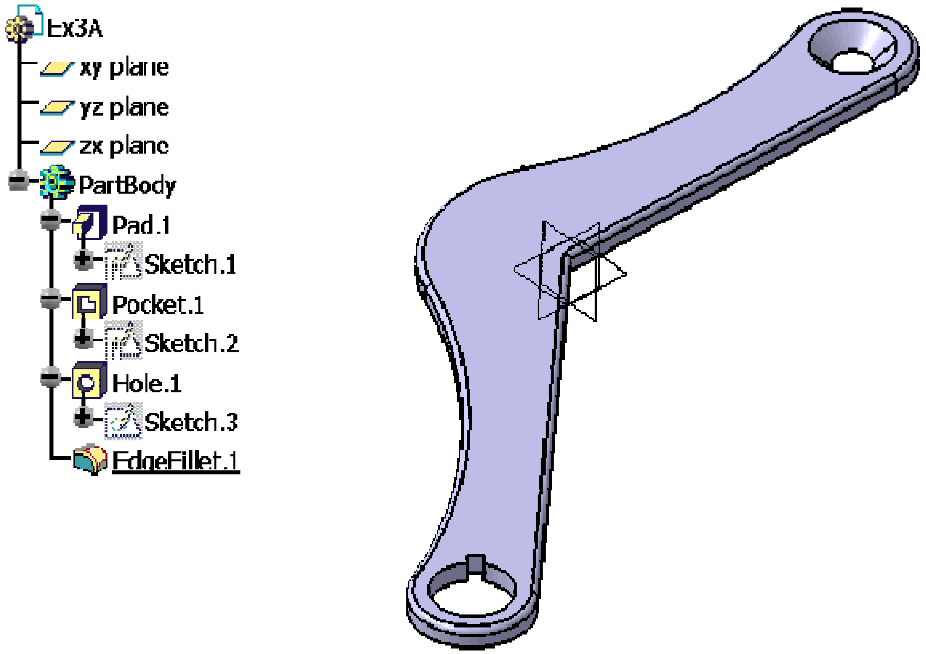




Student Notes:

Exercise Recap: Basic Features Creation

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



Student Notes:

Exercise: Basic Feature Creation

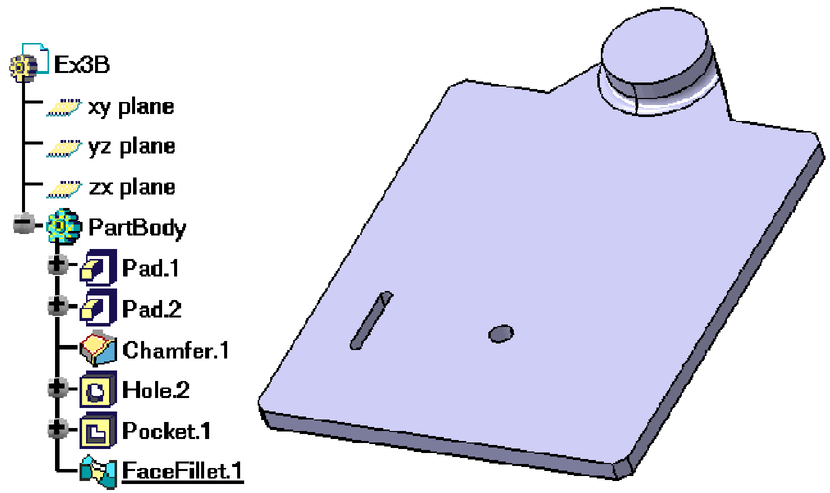
Recap Exercise

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**

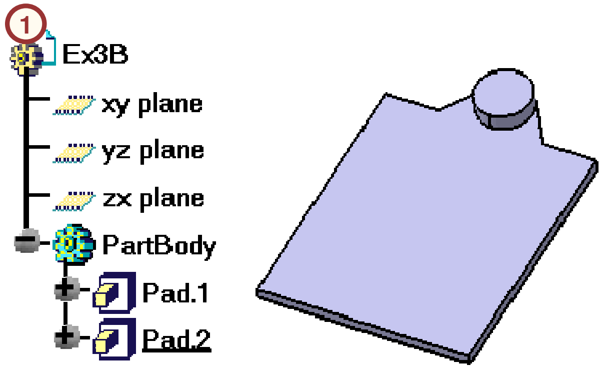


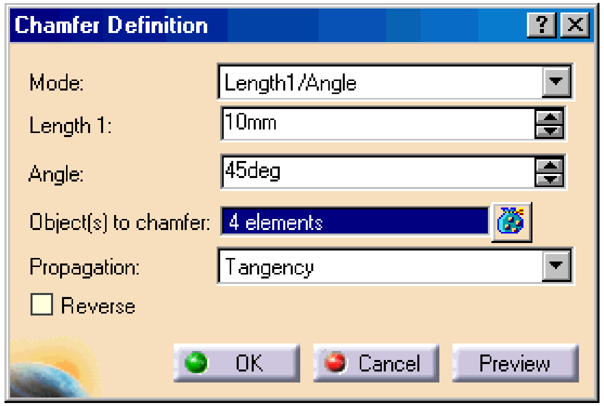
Student Notes:

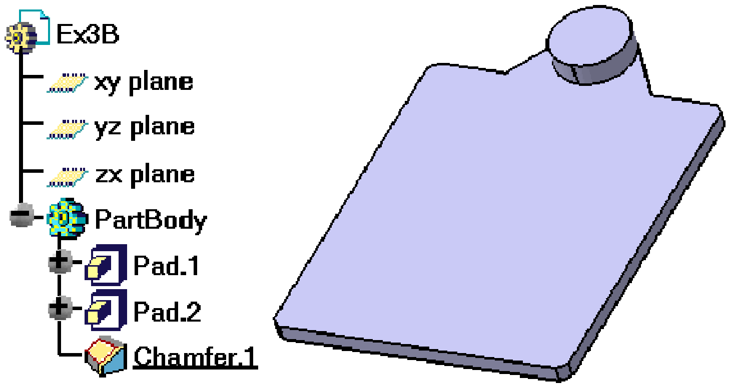
Do it Yourself (1/4)

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.







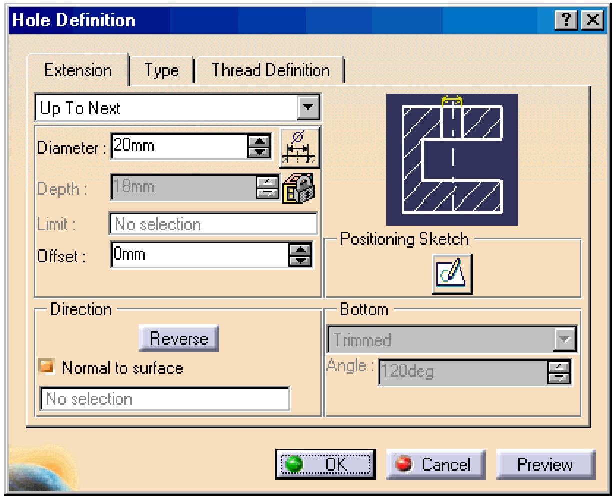
Student Notes:

Do it Yourself (2/4)

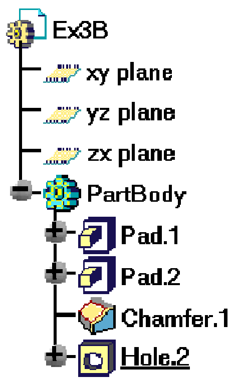
3. Create a simple hole.

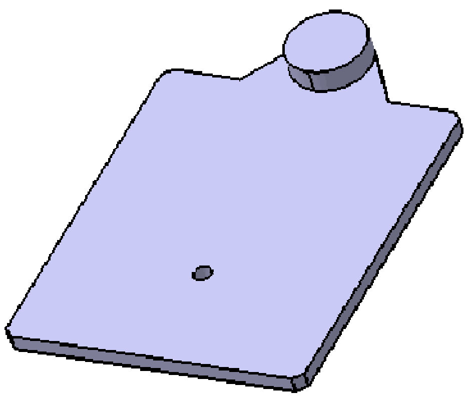
- Create a simple hole using the pre-defined references method.

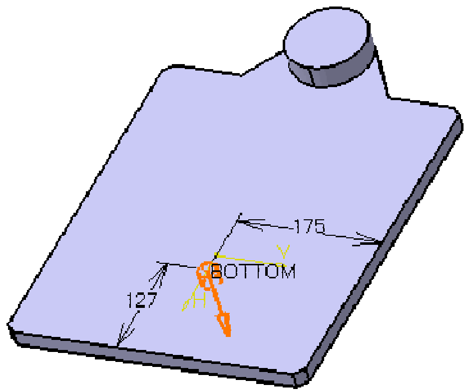
3



Conf. Dep.



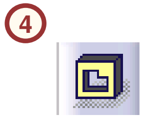


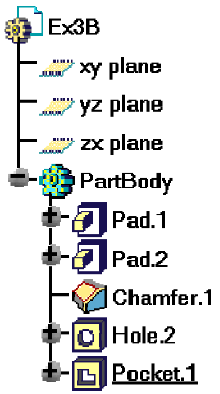


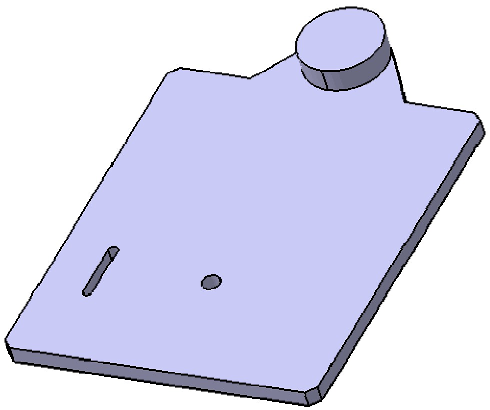
Student Notes:

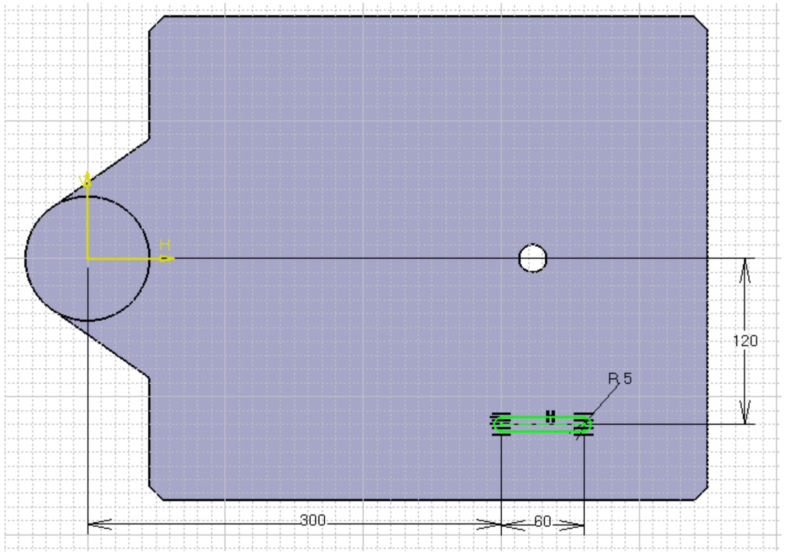
Do it Yourself (3/4)

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









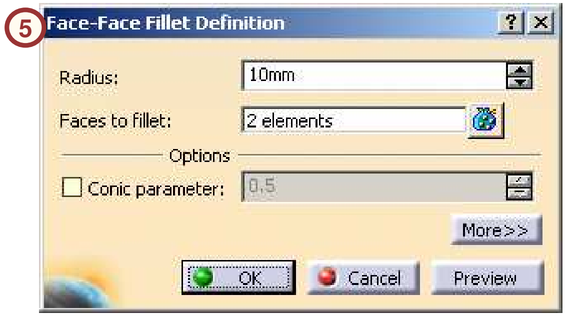
Student Notes:

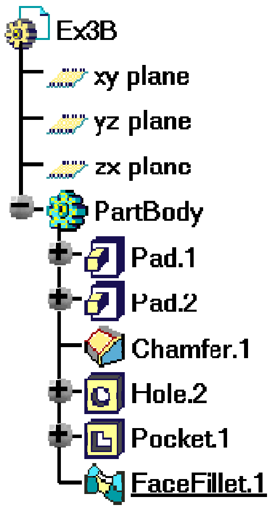
Do it Yourself (4/4)

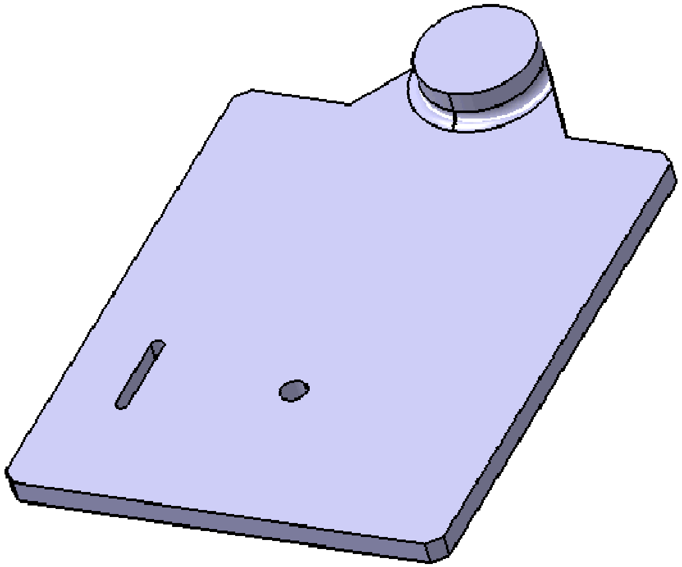
5. Create a face-face fillet.

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

6. Save and close the file.



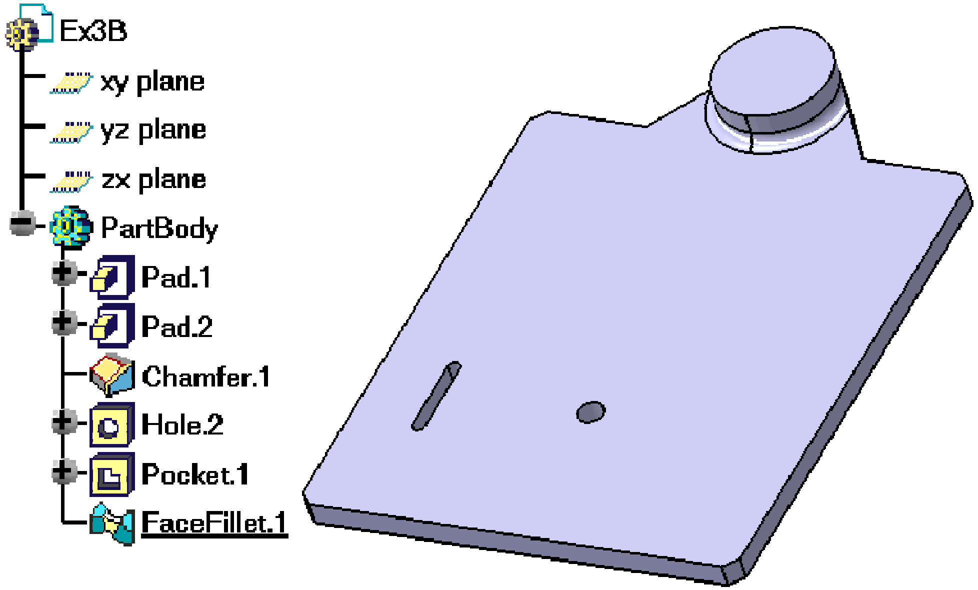




Student Notes:

Exercise Recap: Basic Feature Creation

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



Student Notes:

Exercise: Basic Features Creation

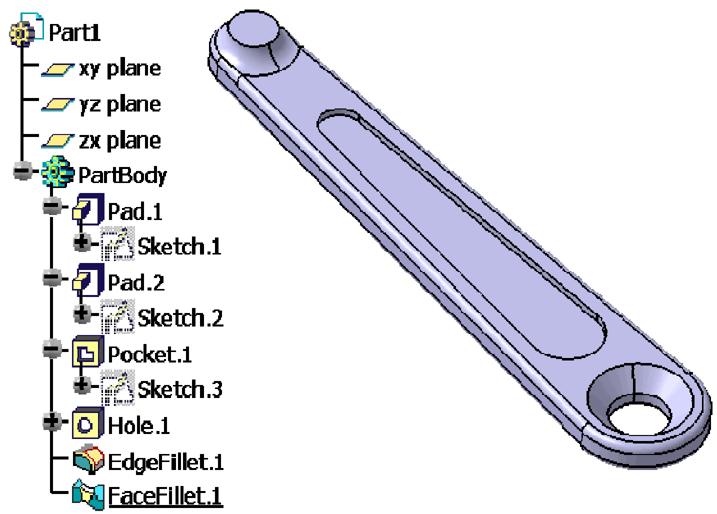
Recap Exercise

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:

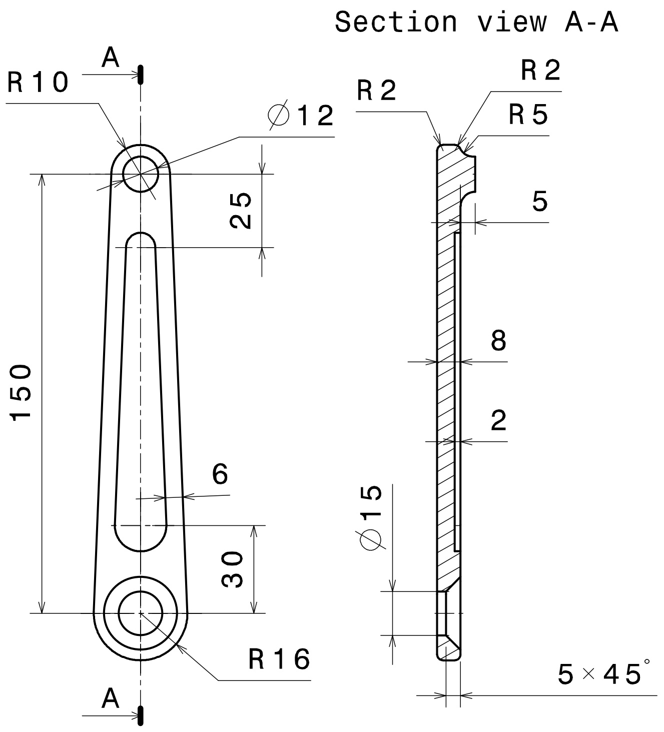
- **Create a pad**
- **Create a pocket**
- **Create a countersunk hole**
- **Create an edge fillet**

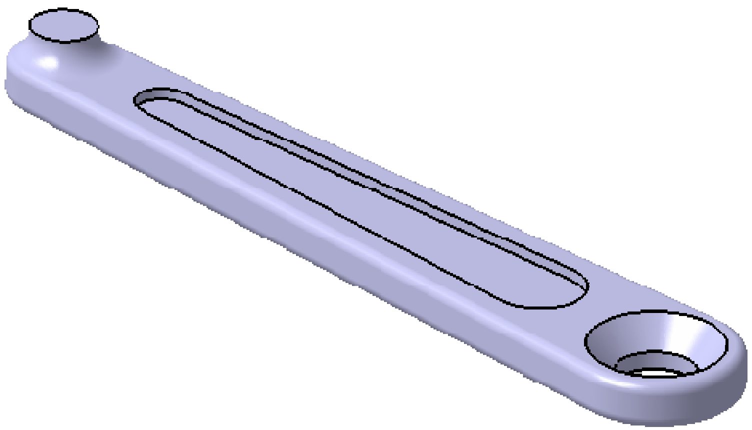


Student Notes:

Do it Yourself

1. Create the following part.

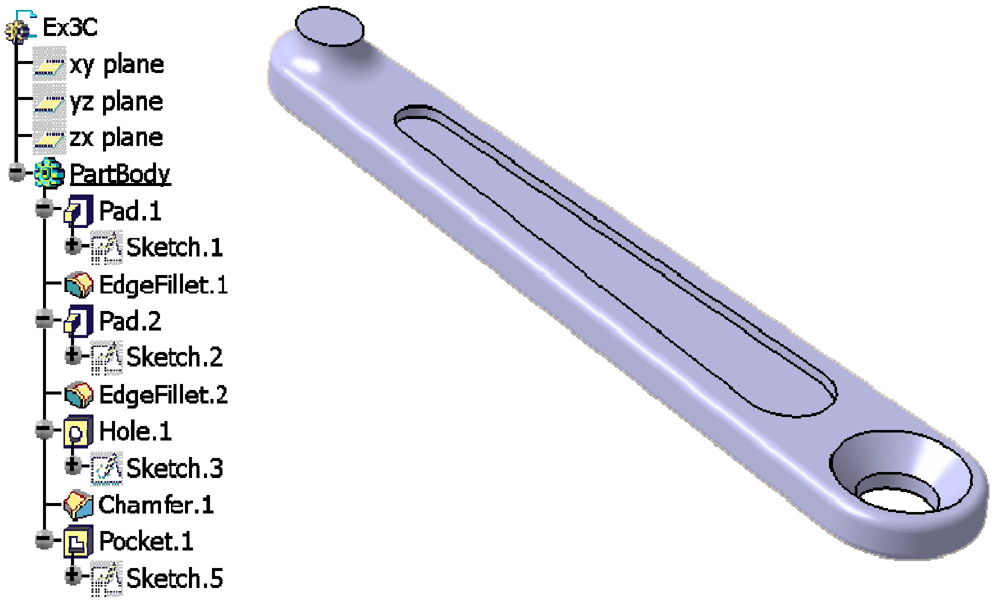




Student Notes:

Exercise Recap: Basic Features Creation

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



Student Notes:

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.

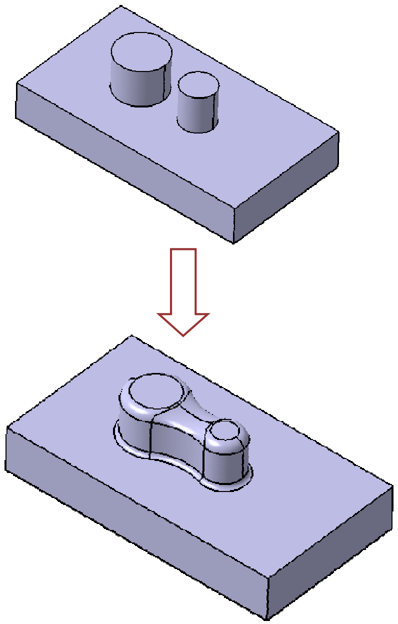
By the end of this exercise you will be able to:

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

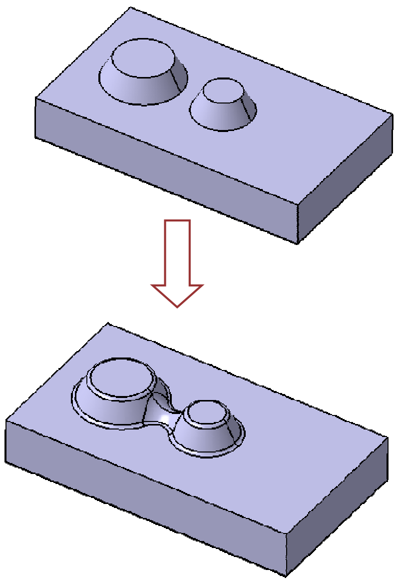
Do it Yourself

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

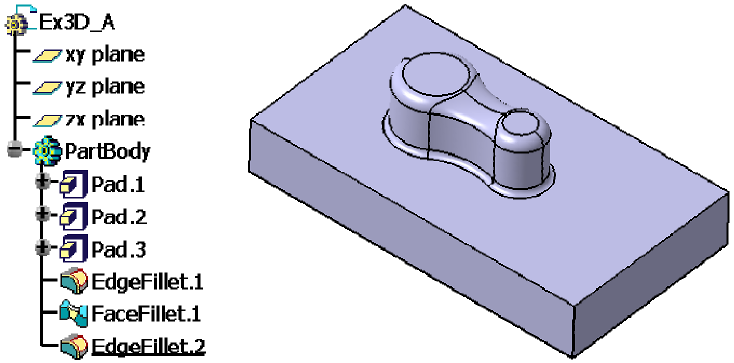


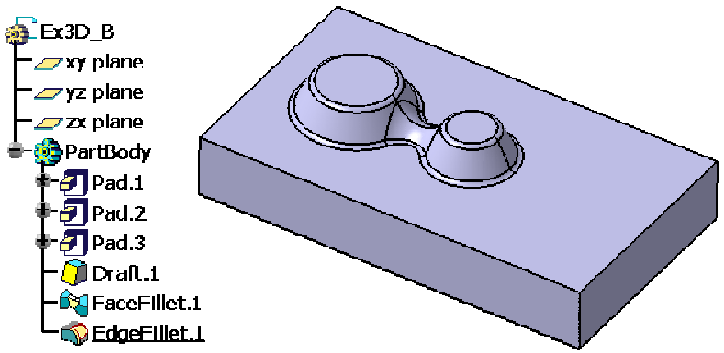
Student Notes:

Student Notes:

Exercise Recap: Edge and Face-Face Fillets

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





Student Notes:

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.

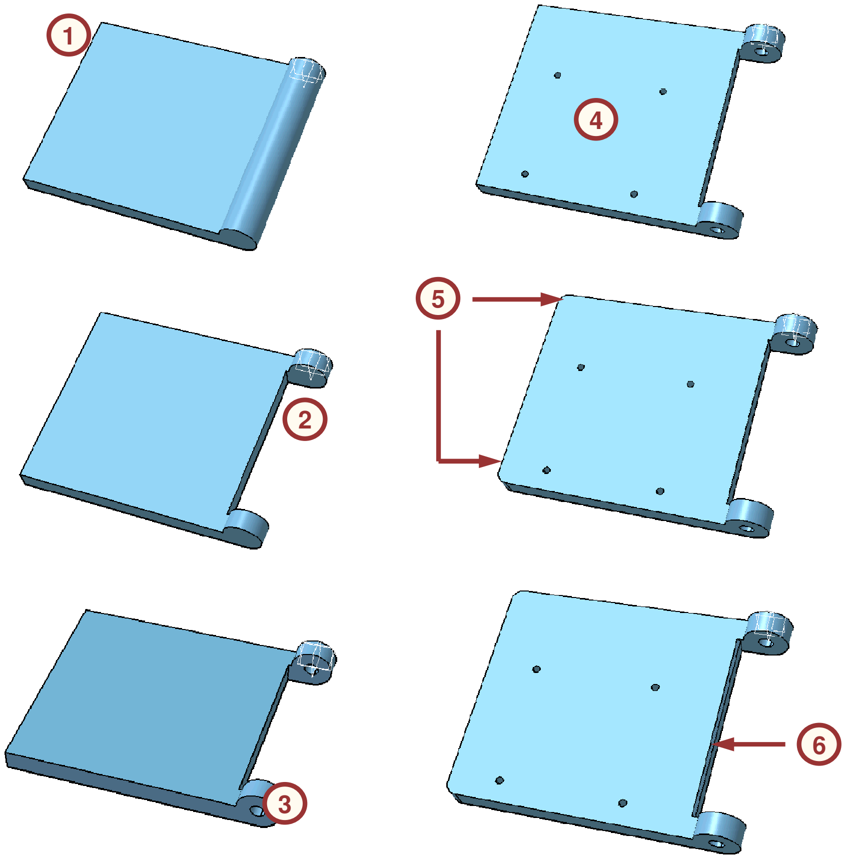
Using the techniques discussed so far, create the model without detailed instructions.

Student Notes:

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

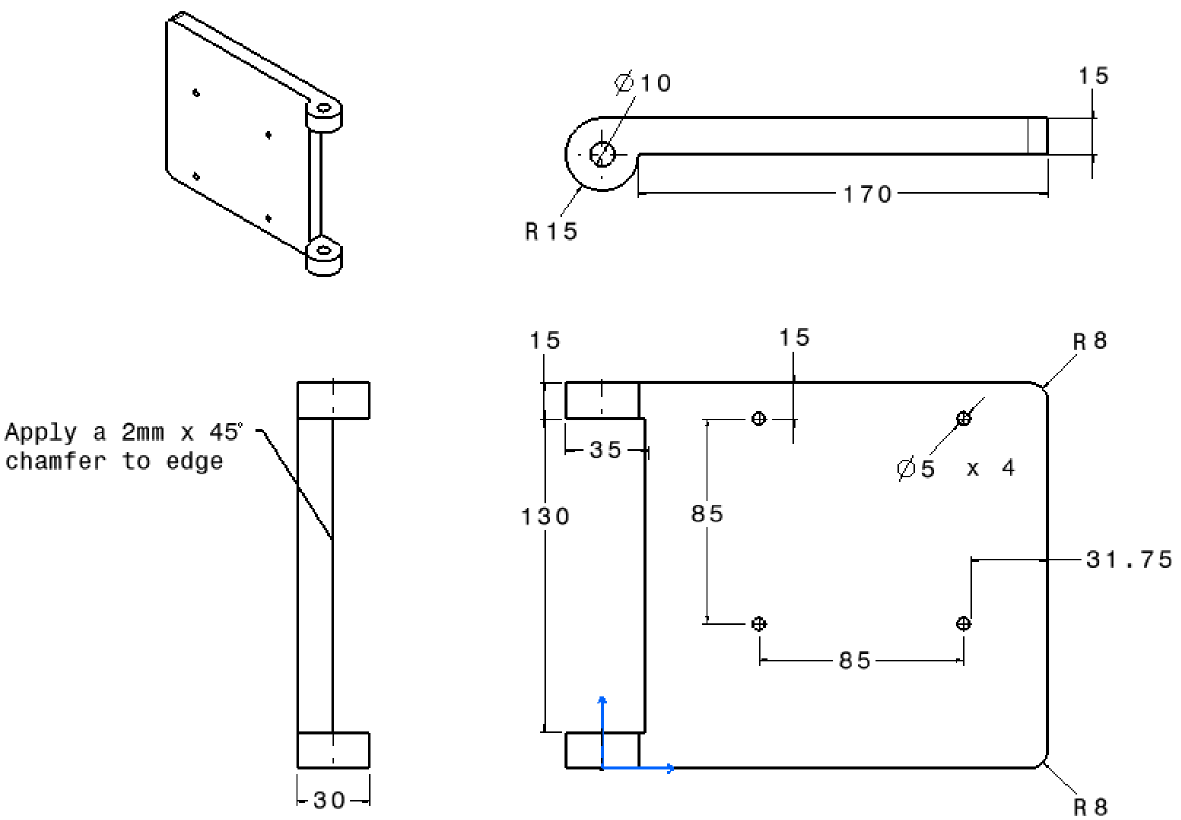
You will be required to create the following features:

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



Student Notes:

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



Student Notes:

Case Study: Engine Support Recap

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