Student Notes:

Additional Part Features

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

Lesson Contents:

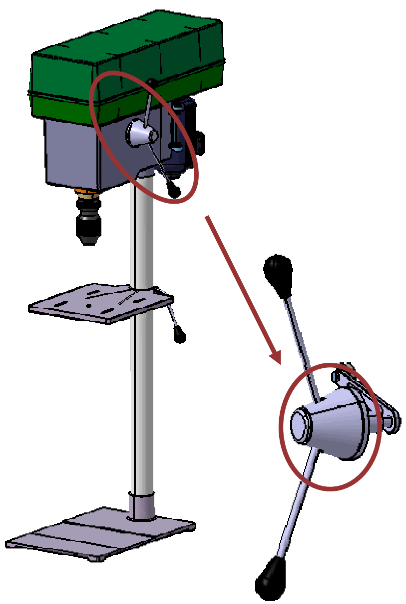
- Case Study: Additional Features
- Design Intent
- Stages in the Process
- Create Feature Profiles and Axis system
- Create Multi-profile Sketch Features
- Create Wireframe Geometry
- Create Shaft and Groove Features
- Shell the Model

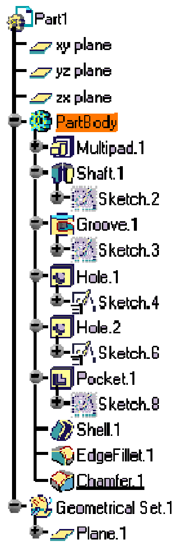
Duration: Approximately 0.5 day

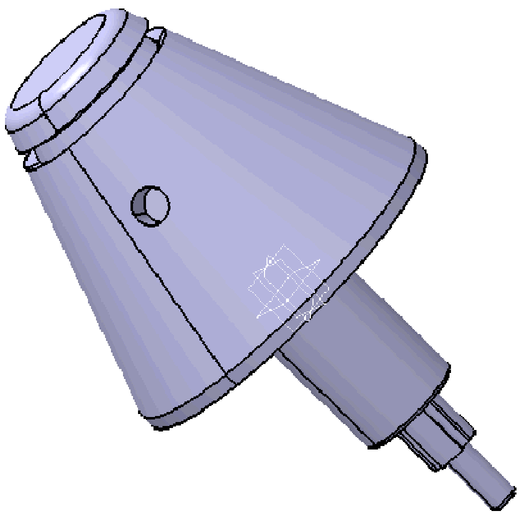
Student Notes:

Case Study: Additional Features

The case study for this lesson is the Handle Block used in the Drill Press assembly shown below. The Handle Block is part of the Handle Mechanism sub-assembly. This case study focuses on creating features that incorporate the design intent of the part. The Handle Block will consist of shafts, grooves, multi-profiles, fillets, chamfers, and a shell feature.







Student Notes:

Design Intent (1/2)

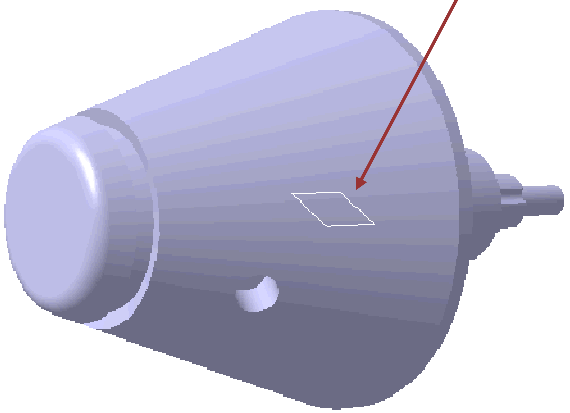
The Handle Block must meet the following design intent requirements:

✓ The top and the bottom portions of the model must be created as separate features.

- The top portion of the model will be created as a shaft, the bottom section will be created as a multi-pad.

✓ The holes must be created at an angle to the XY plane.

- Create the holes on the shaft surface, aligned to a user-defined plane which is created at an angle to the XY plane. Creating the holes on a user-defined plane gives more flexibility in the hole placement as the angle of plane can be changed as required.



Student Notes:

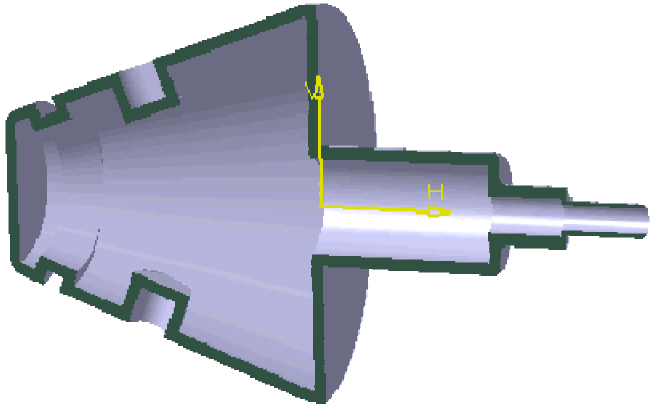
Design Intent (2/2)

The Handle Block must meet the following design intent requirements (continued):

✓ The model must be hollow and must have a uniform thickness of 3mm, except the end, which must have a thickness of 1mm.

- The Shell option will hollow out the model as required.

✓ The holes must be normal to the sides of the handle block.



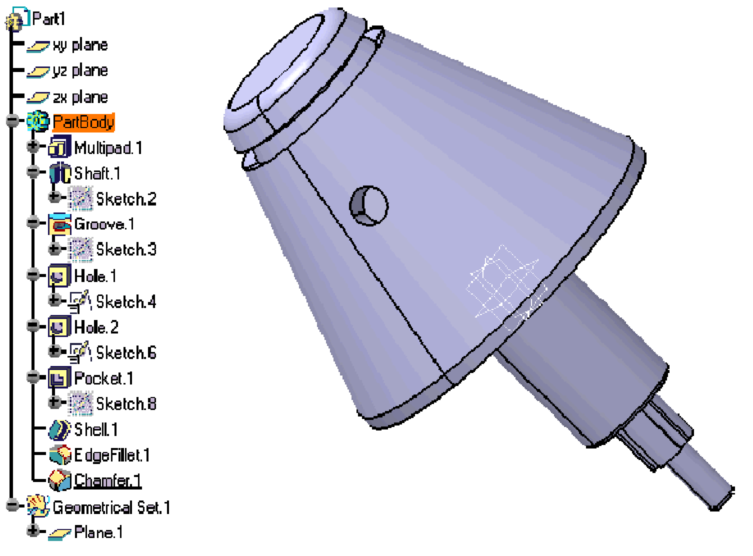
X-section of handle block

Student Notes:

Stages in the Process

Use the following steps to create the handle block:

1. Create feature profiles.
2. Create multi-profile sketch features.
3. Create reference geometry.
4. Create shaft and groove features.
5. Shell the model.



Create Feature Profiles and Axis system

In this section you will learn about additional sketch tools.

Use the following steps:

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

Student Notes:

Student Notes:

Additional Sketcher Tools

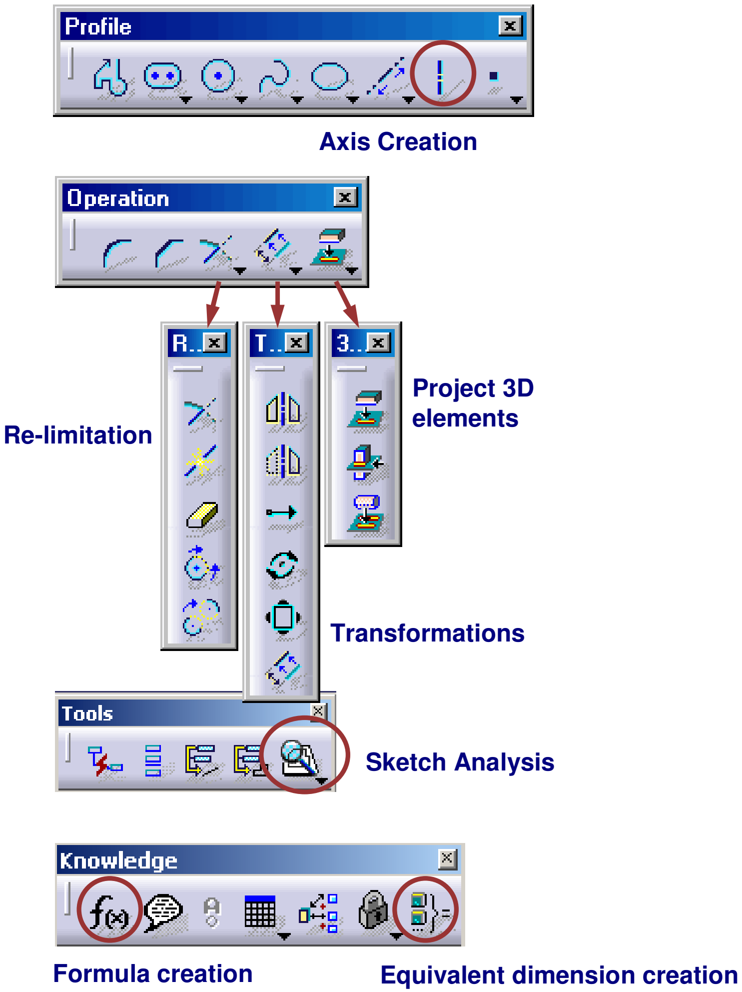
Lesson 2 introduced you to the basic Sketcher tools and the Sketcher environment. This lesson will introduce you to the advanced Sketcher tools.

Sketcher includes the following additional tools:

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

In addition, you will learn how to:

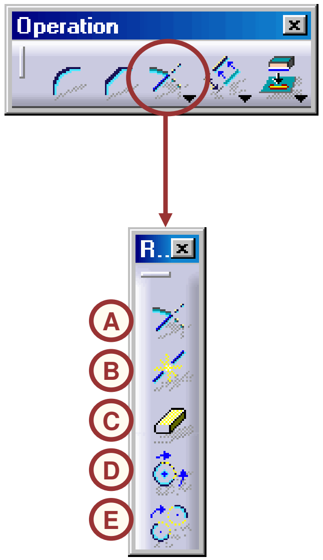
- Create Equivalent dimensions
- Create Formula



Student Notes:

Sketcher Re-limitation Tools

The **Re-limitation** tools trim or extend the existing sketched geometry. They can be found in the Re-limitation toolbar, which is a flyout menu in the **Operation** toolbar.



Available re-limitation tools include the following:

A. Trim

B. Break

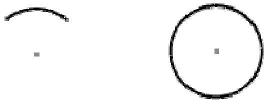
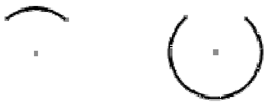
C. Quick Trim

D. Close

E. Complementary Angle

Student Notes:

Re-limitations

Tool	Geometry	Description
Trim		Trims two curves. Keeps the part of the curves you selected. This option can also be used to extend to elements.
Break		Breaks a curve at a selected point.
Quick Trim		Trims an intersected element.
Close		Closes the selected arc.
Complement		Creates the complementary arc.

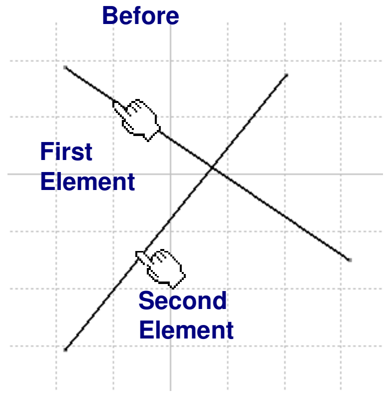
Student Notes:

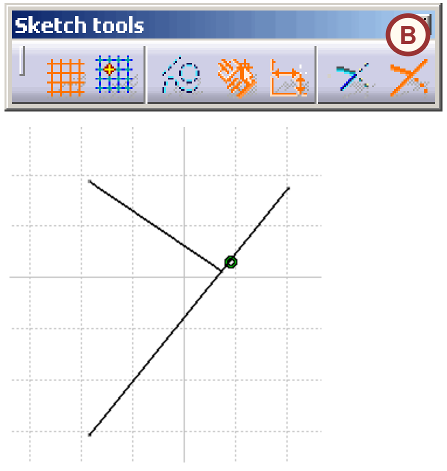
Trim Options

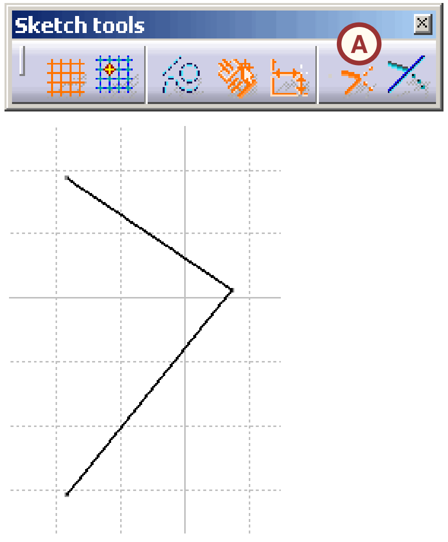
Once the **Trim** tool is selected, the Sketch Tools toolbar expands to display two modes for trim:

A. The *Trim All Elements* mode trims both the selected elements.

B. The *Trim First Element* mode trims only the first selected element; the second element is left unchanged.







Student Notes:

Quick Trim Options

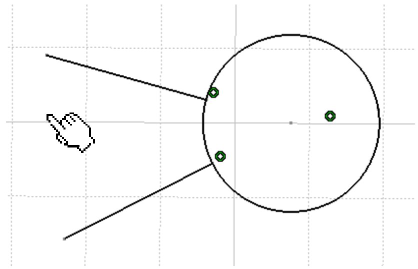
Once the **Quick Trim** tool is selected, the Sketch Tools toolbar expands to display several modes for quick trim:

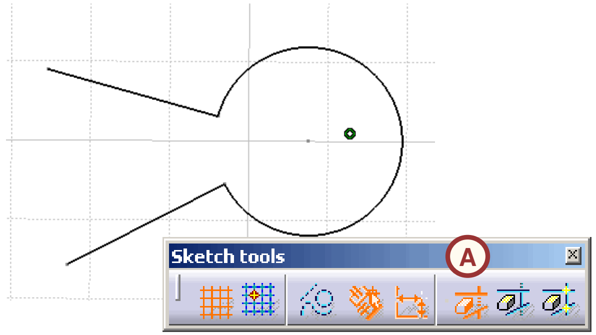
A. The *Break and Rubber In* mode removes a selected portion of an element up to its intersection with other elements.

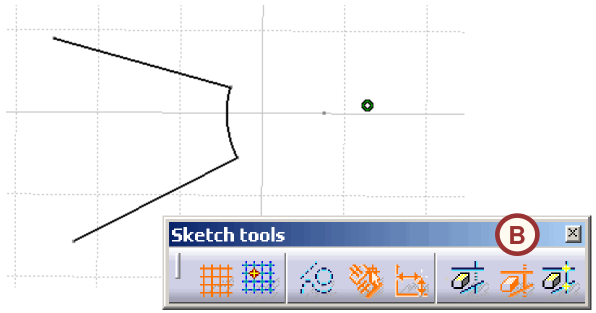
B. The *Break and Rubber Out* mode keeps the selected portion of an element up to its intersection with other elements.

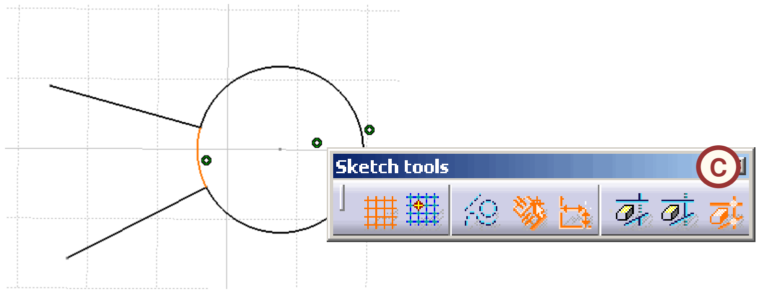
C. The *Break and Keep* mode keeps the entire elements but breaks the element at the intersection with other elements.

Before









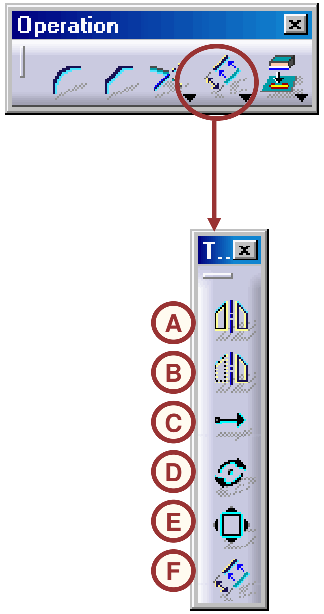
Student Notes:

Sketcher Transformation Tools

Transformation tools are used to modify existing sketcher geometry. They can also be used to create a duplicate of the existing sketcher geometry.

Transformation tools are found in the **Transformation** toolbar, which is a flyout menu in the **Operation** toolbar. Available Transformation tools include the following:

A. Mirror
B. Symmetry
C. Translate
D. Rotate
E. Scale
F. Offset



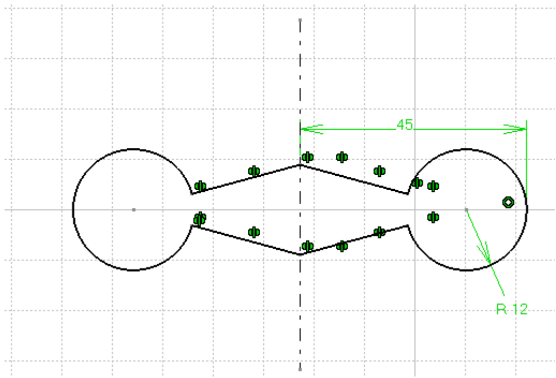
Student Notes:

Mirror and Symmetry Options

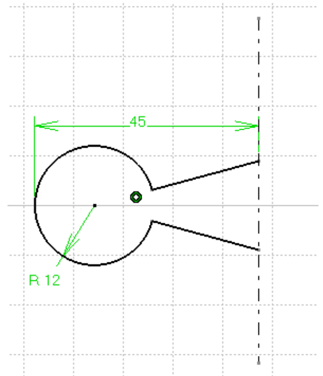
Both options, **Mirror** and **Symmetry**, allow you to mirror the selected geometry about an axis. The **Mirror** option retains the original geometry, while the **Symmetry** option removes it.

Use the following steps to use the **Mirror** and **Symmetry** tools:

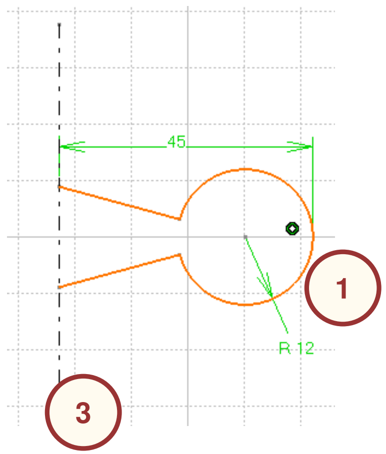
1. Select the geometry to mirror.
   Use the <Ctrl> key to select multiple items.
2. Select the tool.
   a. **Mirror**
   b. **Symmetry**
3. Select the symmetry axis.

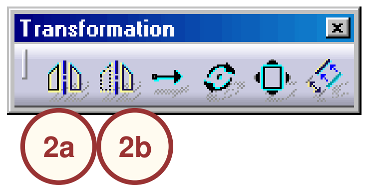


Result of Mirror



Result of Symmetry



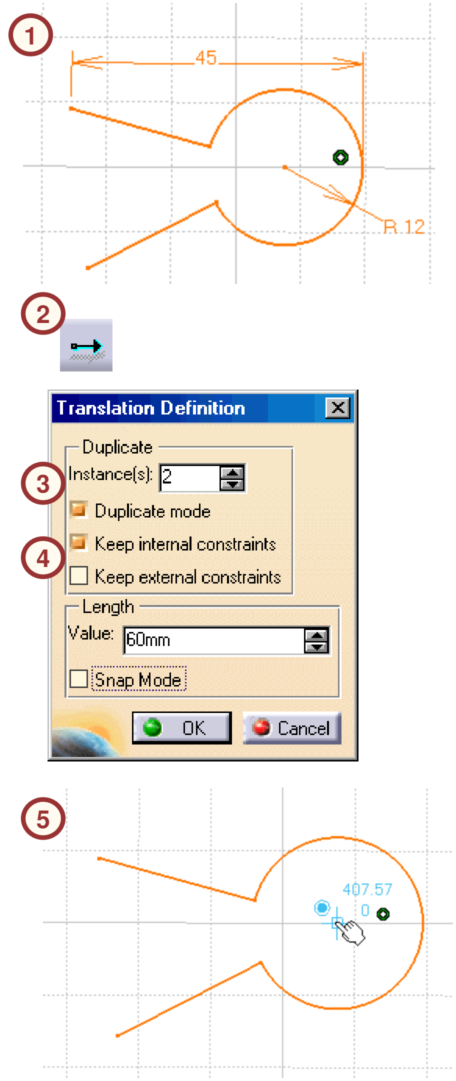


Translation (1/2)

The Translation tool moves the selected geometry along a translation vector.

Use the following steps to translate geometry:

1. Select the entities to move.
2. Select the **Translate** tool.
3. Select the **Duplication mode** option.
   - When the **Duplication mode** option is selected, the original geometry is unchanged and a copy of the geometry is created in the new location. You can also create multiple instances which are equidistant to each other. In this example, two instances are created.
4. If in the duplicate mode, specify the constraint conditions.
   - You may choose to keep all internal constraints, and/or all external constraints.
5. Select a point on the screen to act as the start point.



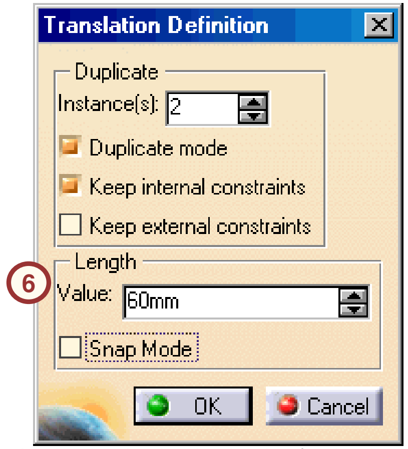
Student Notes:

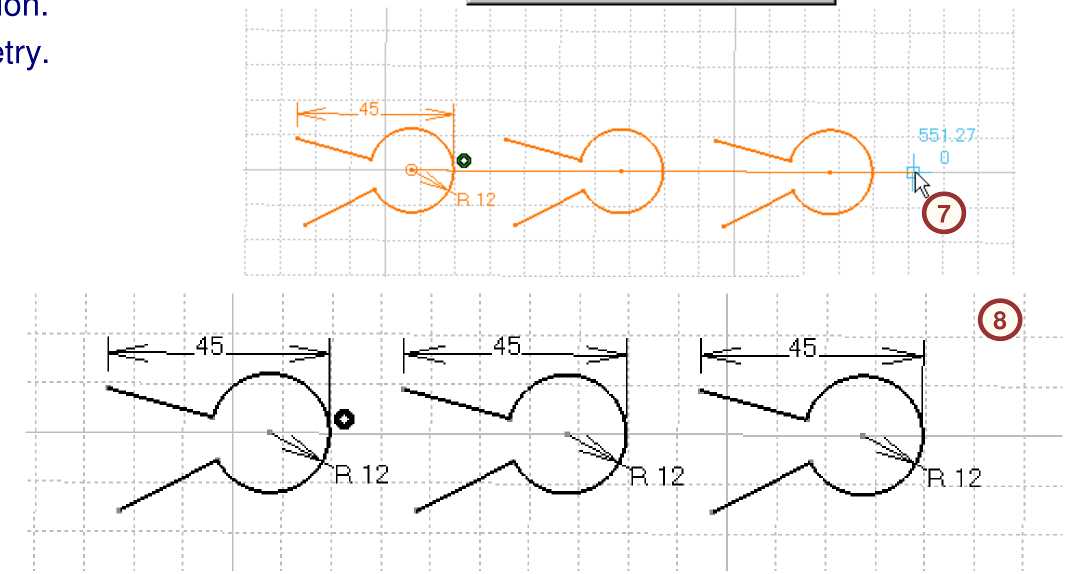
Student Notes:

Translation (2/2)

Use the following steps to translate geometry (continued):

6. Optionally, enter a distance value in the length field and click **OK**.
7. Move the start point on the screen. If no distance has been specified (step 6), you can place the selected elements anywhere. If a distance has been applied and **OK** is selected, you will have to define the direction.
8. Click to place the geometry.



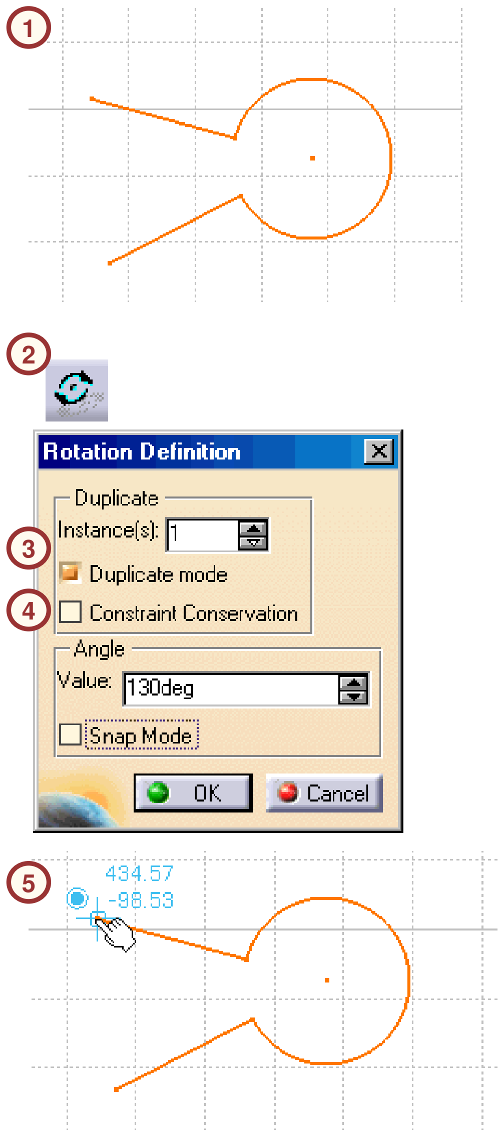


Rotation (1/2)

The **Rotate** tool lets you rotate selected sketched element(s) about a point.

Use the following steps to rotate geometry:

1. Select the entities to rotate.
2. Select the **Rotate** tool.
3. Select the **Duplication mode** option.
   - When the **Duplication mode** option is selected, the original geometry is unchanged and a copy of the geometry is created in the new location. You may create multiple instances which are equidistant to each other. In this example, one instance is created.
4. If in duplicate mode, specify **Constraint Conservation**. If selected, all internal constraints will be maintained.
5. Select a point on the screen to act as the center of rotation.



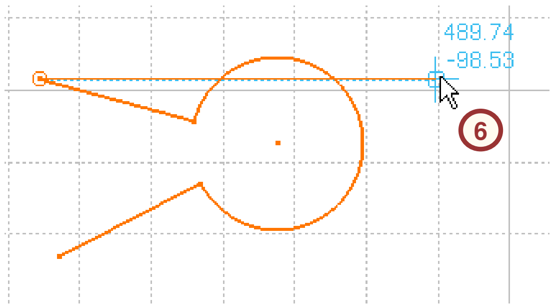
Student Notes:

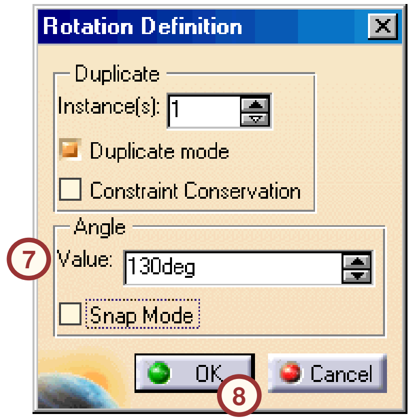
Student Notes:

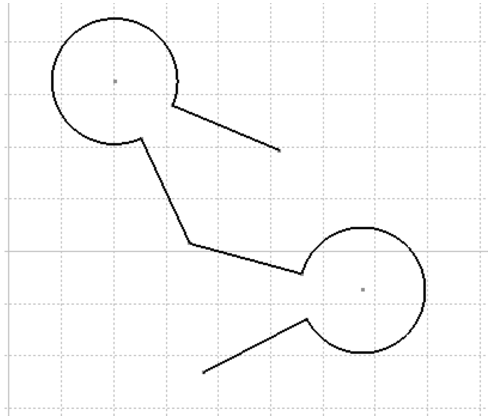
Rotation (2/2)

Use the following steps to rotate geometry (continued):

6. Select a point on the screen to define a reference line for the angle.
7. Specify a value in the Angle field or move the mouse to rotate the elements.
8. Click **OK** or click on the screen to complete the rotation.







Student Notes:

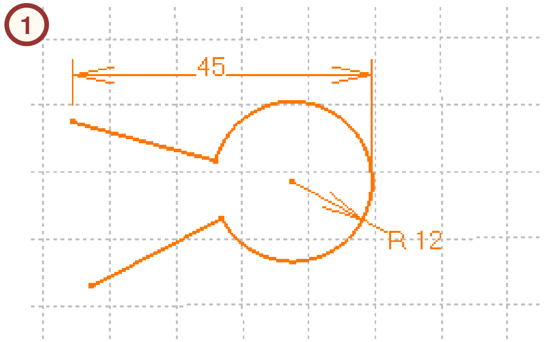
Scale (1/2)

The **Scale** tool allows you to resize the selected sketched element(s).

Use the following steps to scale sketched element(s):

1. Select the entities to scale.
2. Select the **Scale** tool.
3. Select the **Duplication mode** option.
   - When the **Duplication mode** option is selected, the original geometry is unchanged and a copy of the geometry is created in the new location.
4. If in duplicate mode, specify **Conservation of the constraints**. If selected, all the constraints will be maintained, but they will be converted into reference dimensions.

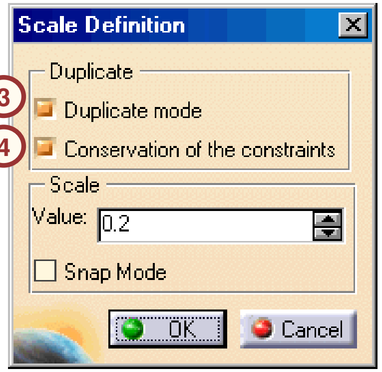
1



2

3

4

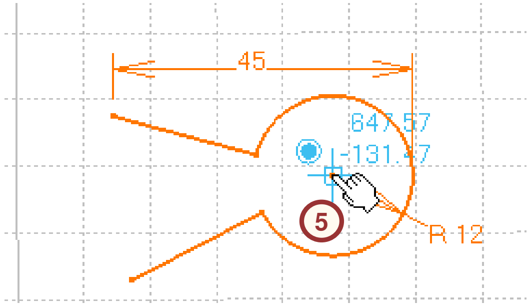


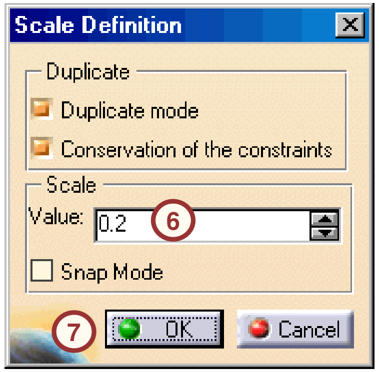
Student Notes:

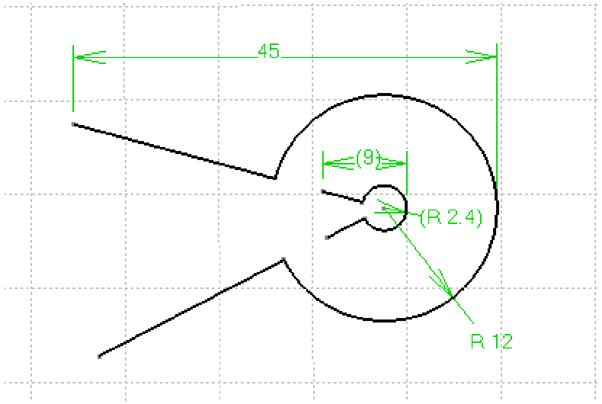
Scale (2/2)

Use the following steps to scale sketched element(s) (continued):

5. Select a point on the screen to act as the center point for scaling.
6. Specify a value in the Scale field or move the mouse to scale the elements.
7. Click **OK** or click on the screen to complete the scaling.







Student Notes:

Offset Propagation Modes

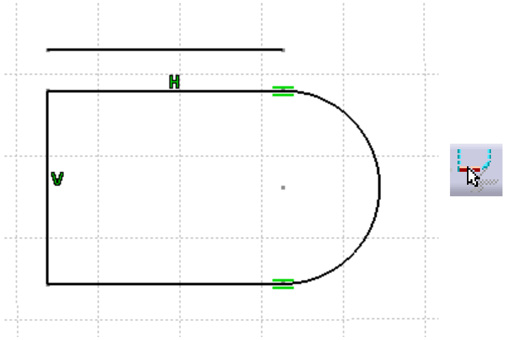
The **Offset** tool lets you offset one or more sketched elements. Once the Offset tool is selected, three propagation modes become available from the Sketch Tools toolbar:

A. In *No Propagation* mode, only the selected element(s) is offset.

B. In *Tangent Propagation* mode, the selected element(s) and all elements tangent to it are offset.

C. In *Point Propagation* mode, the selected element(s) and all elements that form a chain with it are offset.

A



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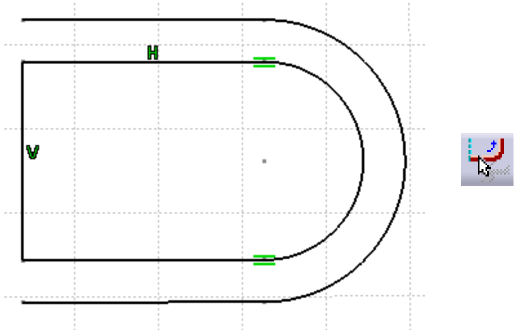
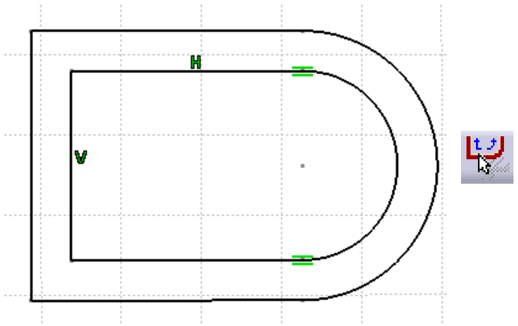


C

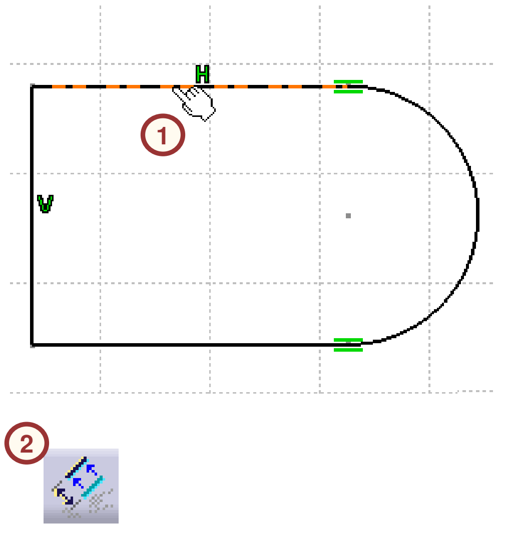


Student Notes:

Offset (1/2)

Use the following steps to offset sketched element(s):

1. Select the sketched element(s) to offset.
2. Click the **Offset** icon.
3. Select the Propagation mode.
4. Click the **Both sides** icon if you want to offset the element on both sides.
5. Enter the number of instances. Each instance will be equi-distant from each other. In this example, two instances are created.

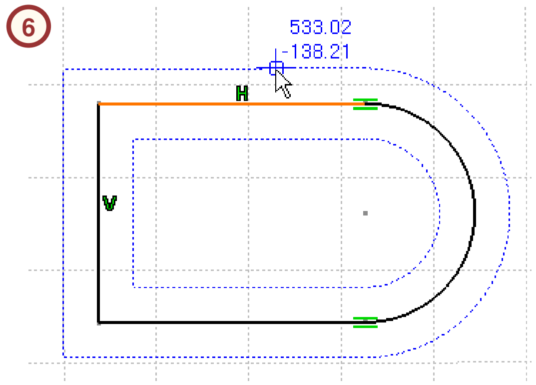


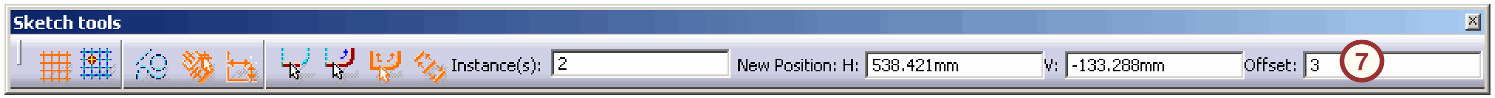
Student Notes:

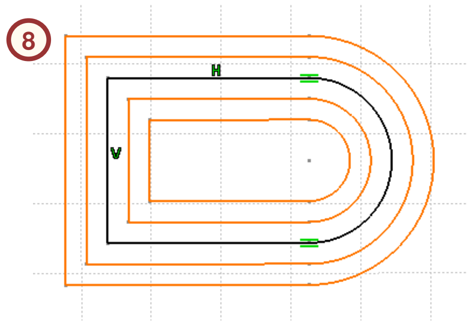
Offset (2/2)

Use the following steps to offset sketched element(s) (continued):

6. Move your pointer to the side on which you want to create the offset.
7. Press the <Tab> key until the Offset field is highlighted. Specify the offset distance.
8. Press the <Enter> key to place the offset.





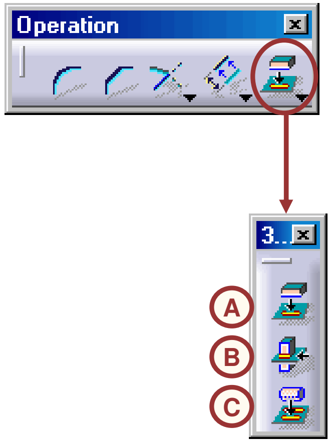


Student Notes:

Project 3D Elements

Several tools are available to project the existing 3D elements onto the sketch plane. These projected elements can be used as a standard sketch geometry, or converted into a construction geometry.

3D projection tools are found in the **3D Geometry** toolbar, which is a fly-out menu in the **Operation** toolbar. Available projection tools include the following:



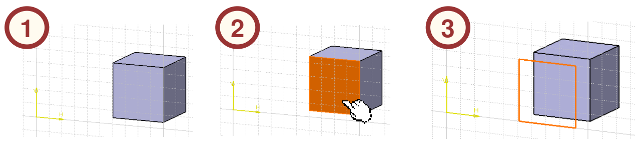
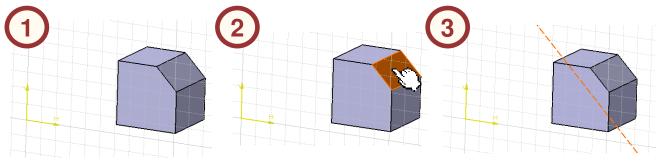
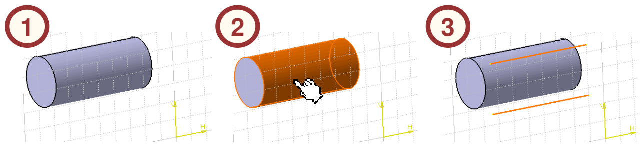
A. Project 3D Elements

B. Intersect 3D Elements

C. Project 3D Silhouette Edges

Student Notes:

3D Geometry Elements

Tool	Geometry	Description
Project 3D Elements		Project 3D elements onto the sketch plane.
Intersect 3D Elements		Intersect 3D elements with the sketch plane.
Project 3D Silhouette Edges		Project the silhouette of a cylindrical element onto the sketch plane. The axis of revolution for the projected element must be parallel to the sketch plane.

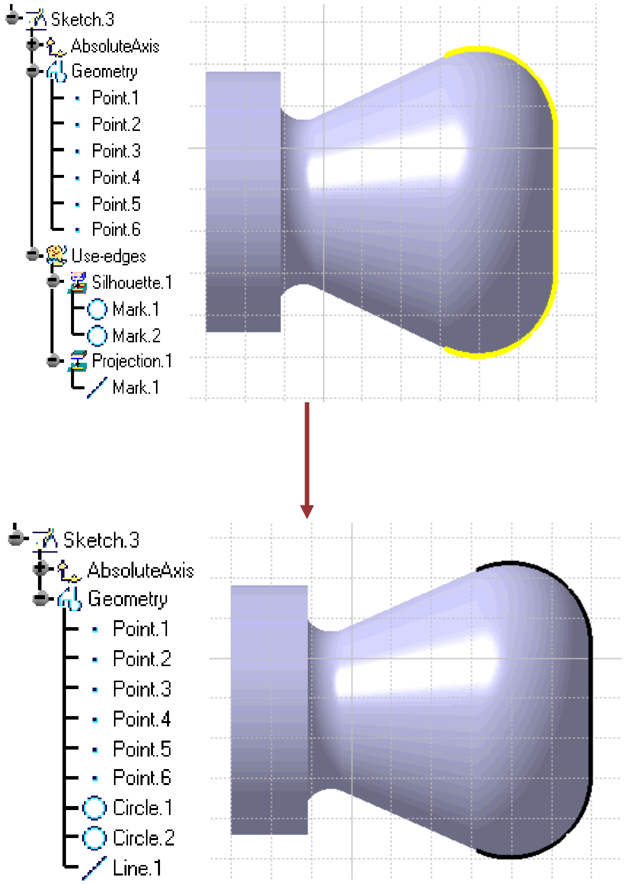
Student Notes:

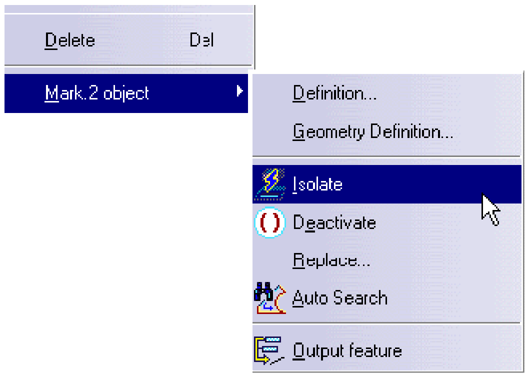
Isolate Projected Elements

By default, projected elements are linked to the 3D geometry from which they were created.

You can break this link by right-clicking on the projected element and clicking **Mark.x object > Isolate** from the contextual menu. Once the element is isolated, it will no longer be associative with the 3D geometry from which it was projected. This means that modifications to the 3D geometry will not impact the sketched elements created from it.

Once isolated, the projected geometry converts into standard sketched elements (e.g., lines, points, arcs).





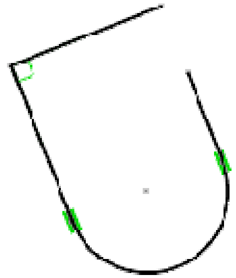
Student Notes:

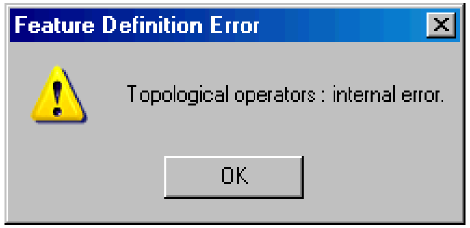
Sketch Analysis

The **Sketch analysis** tool can be used to help resolve problems with the sketch.

This tool can be used to determine a sketch's constraint status (i.e., Under-constrained, Iso-constrained, Over-constrained, or Inconsistent), and where degrees of freedom still exist in the sketch.

The Sketch Analysis tool can also be used to determine whether a profile is open or closed. This is useful if you receive an error while trying to create sketched-based features.



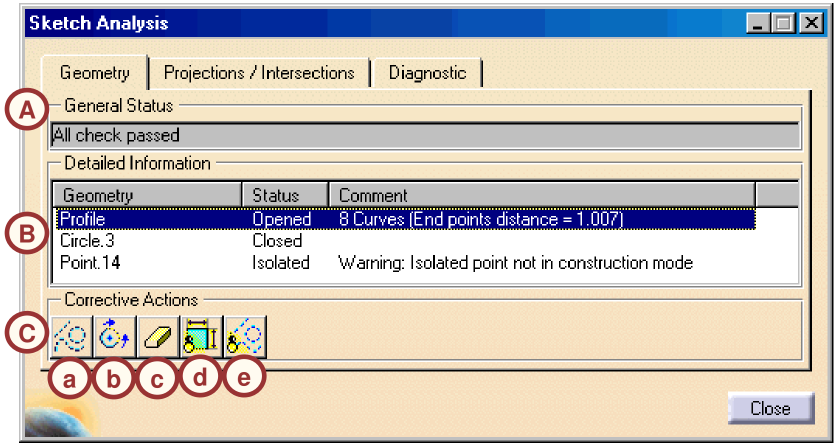


Student Notes:

Sketch Analysis Window (1/3)

The sketch analysis window has three tabs. Each tab contains information to analyze the sketch.

The **Geometry** tab is used to determine whether the sketch geometry is valid or not:



A. The *General Status* area analyzes several elements in the context of the entire sketch.

B. The *Detailed Information* area provides the status and comment on each geometric element in the sketch.

C. The *Corrective Actions* area lets you correct geometry. You can:

   a. Convert an element into a construction element.

   b. Close an open profile.

   c. Erase unwanted geometry.

   d. Hide all the constraints.

   e. Hide all the construction geometries.

Student Notes:

Sketch Analysis Window (2/3)

The **Projections/Intersections** tab is used to determine the status of all the projected elements:

A. The *Detailed Information* area provides a the status and comment on each projected or intersected element in the sketch.

B. The *Corrective Action* area lets you correct geometry. You can:

   a. Isolate geometry.

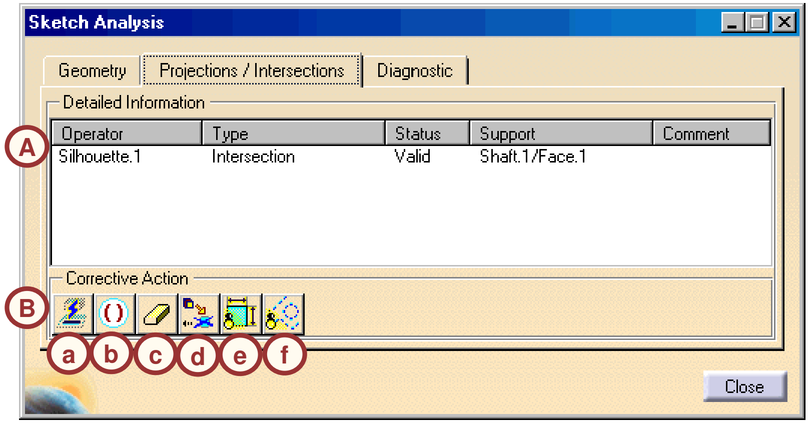
   b. Activate or Deactivate a constraint.

   c. Erase geometry.

   d. Replace a 3D geometry.

   e. Hide all the constraints.

   f. Hide all the construction geometries.

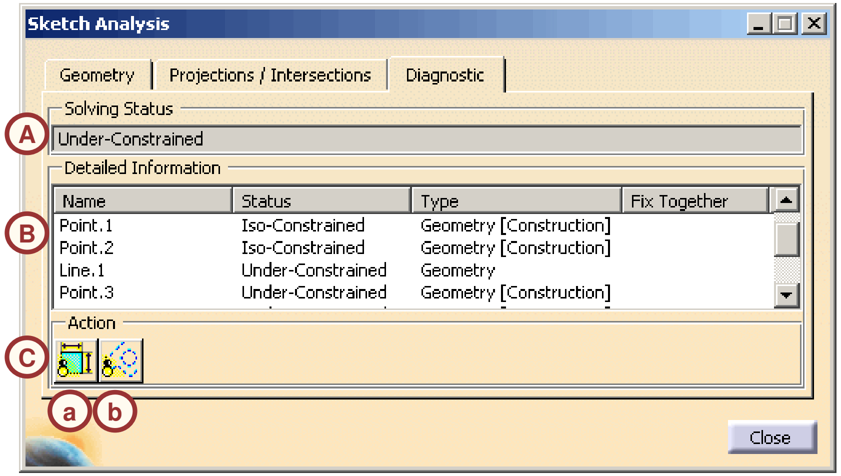


Student Notes:

Sketch Analysis Window (3/3)

The **Diagnostics** tab displays a full diagnosis of all the sketched geometries. It provides an analysis of the sketch as well as information on individual geometrical elements:

A. The *Solving Status* area provides an overall analysis of the sketched geometry.
B. The *Detailed Information* area provides a description and status on each constraint and geometric element in the sketch.
C. The *Action* area enables you to:
    a. Hide all constraints.
    b. Hide all the construction geometries.

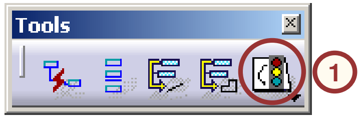


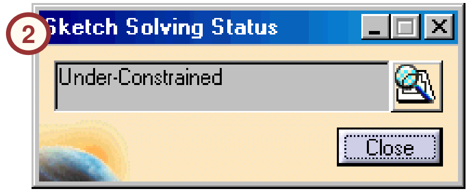
Student Notes:

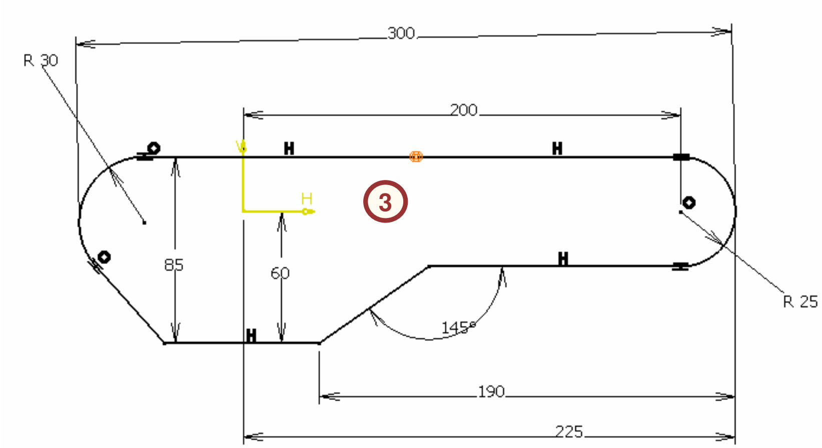
Performing a Quick Geometry Diagnosis (1/2)

Use the following steps to analyze a sketch:

1. Click the **Sketch Solving Status** icon.
2. The **Sketch Solving Status** dialog box appears. It indicates the overall status of the Sketch Geometry. In this case, the sketch is under-constrained even though the sketch appears to be green (iso-constrained).
3. Under- and over-constrained geometrical elements are highlighted on the sketch and in the specification tree.





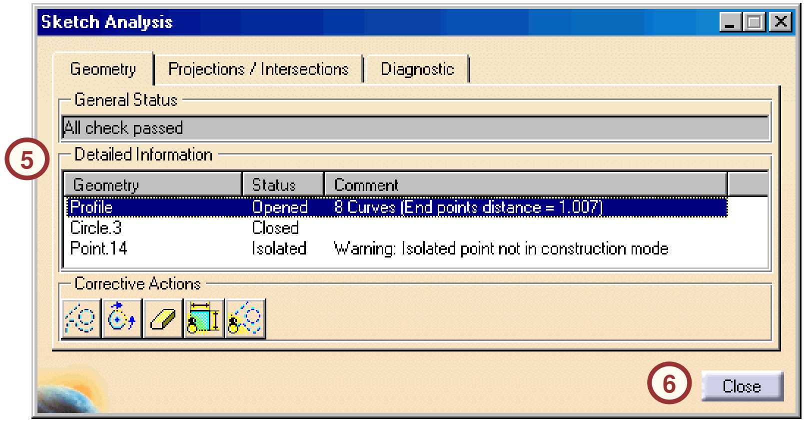


Student Notes:

Performing a Quick Geometry Diagnosis (2/2)

Use the following steps to analyze a sketch (continued):

4. Click the **Sketch Analysis** icon in the window or in the toolbar.
5. The Sketch Analysis window appears. In this example, the profile needs to be closed and the point needs to be changed to a construction element.
6. Click **Close** to close the Sketch Analysis window.



Student Notes:

Create Relationships between Dimensions

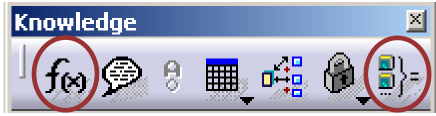
Relationships between Dimensions can be created by using-

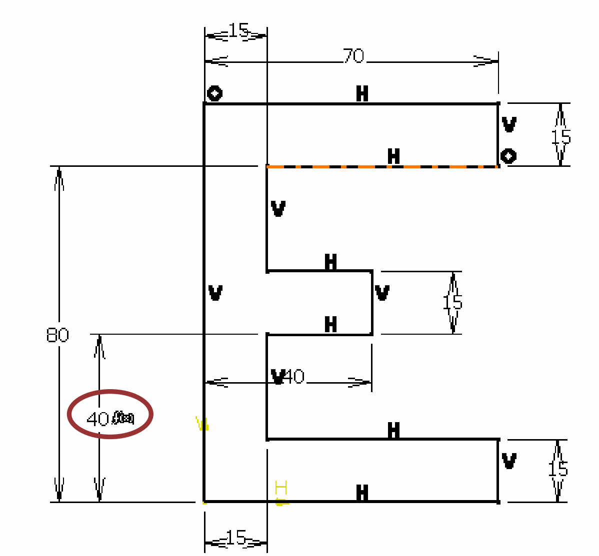
a. Equivalent Dimensions

b. Formula

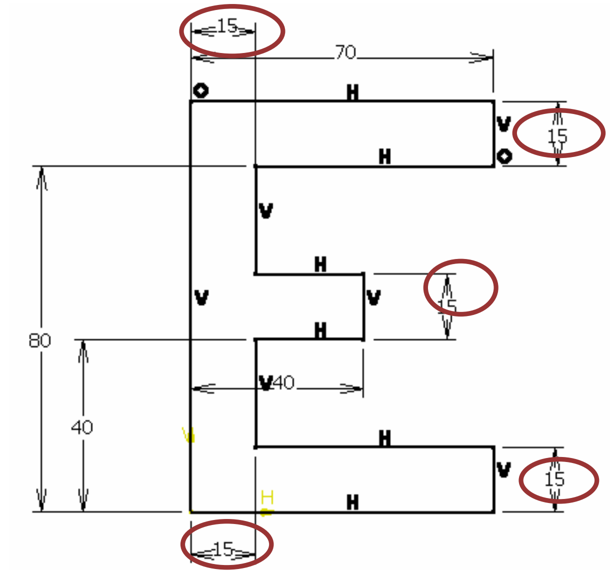
The Equivalent Dimensions feature can be used to define an equality between a set of Angles or Length parameters.

The formula can be used to relate one parameter to another.





Highlighted dimension is driven by a formula

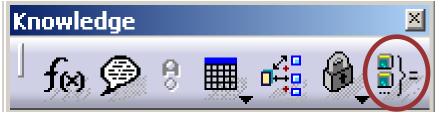


Highlighted dimensions are Equivalent Dimensions

Student Notes:

Equivalent Dimensions

The Equivalent Dimensions feature can be found in the **Knowledge** toolbar, which can be accessed in any workbench (such as Sketcher, Part Design) .

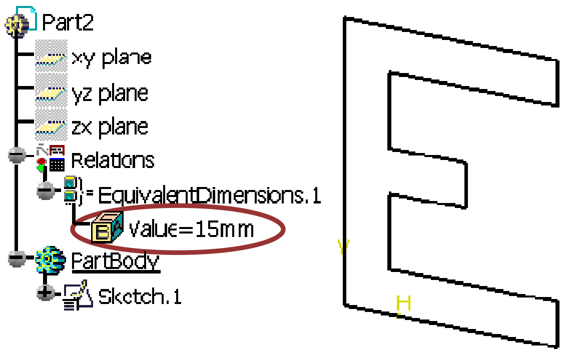


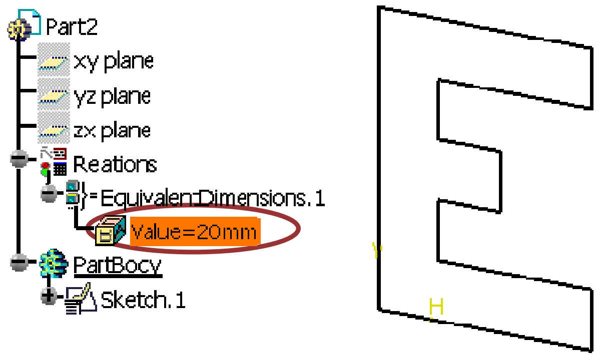
The value of Length or Angle can be modified through the editor and is propagated to all the parameters belonging to the equivalence.

Equivalent Dimensions feature help to-

a. Increase designers' productivity.

b. Reduce the model size.



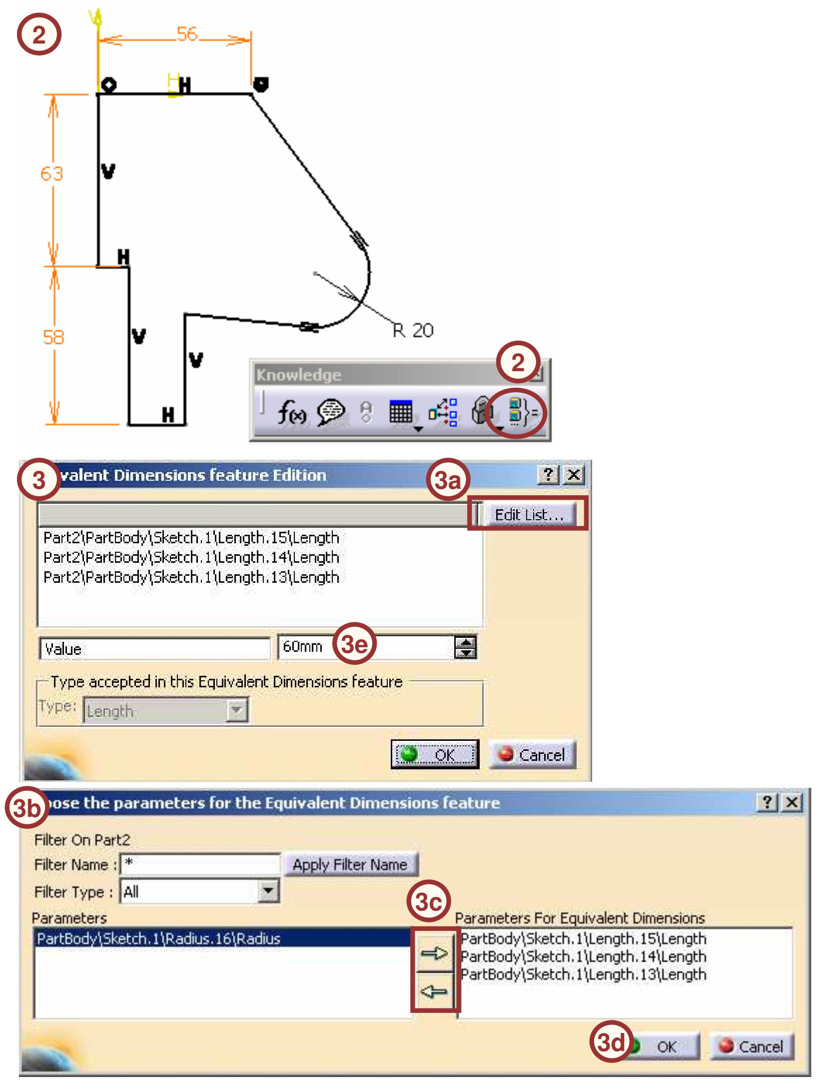


Student Notes:

Creating Equivalent Dimensions

Use the following steps to create Equivalent Dimensions through a sketcher:

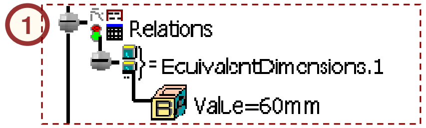
1. Edit the sketch to enter the Sketcher Workbench.
2. Select the dimensions that you want to equalize and click **Equivalent Dimensions** icon.
3. Equivalent Dimension edition dialog box is displayed.
   a. Click **Edit List** to add/remove parameters for equivalent dimensions.
   b. A dialog box is displayed for you to select the equivalent parameters.
   c. Use arrows to add/remove parameters for equivalent dimensions.
   d. Click **OK** to go back to Equivalent Dimension edition dialog box.
   e. In Equivalent Dimension edition dialog box, specify the value of equality.

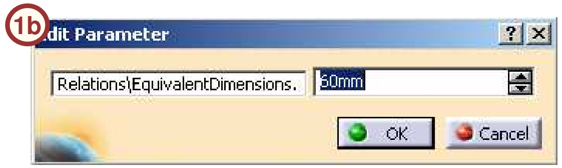


Student Notes:

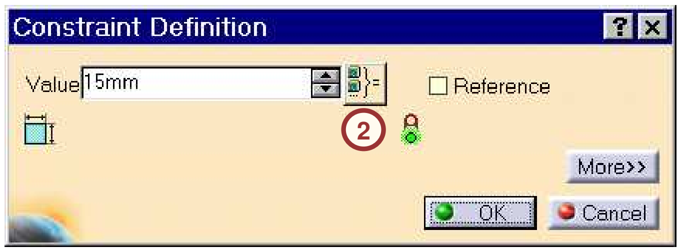
Editing Equivalent Dimensions through Feature Tree

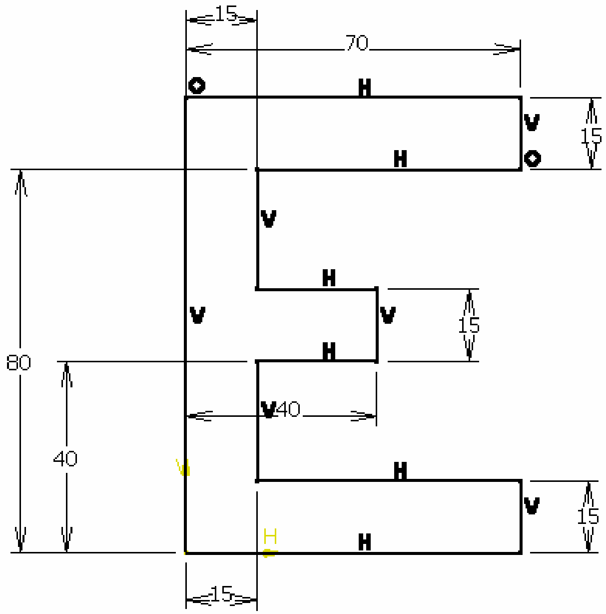
1. The Equivalent dimension feature is displayed in the Relations node of the feature tree.
   a. Double-click on it to view the list of parameters, modify it or change the value.
   b. Double-click on Value to change the value of equality.





2. Any dimension in the list of equivalent dimensions can be selected graphically and edited in order to modify all the dimensions. The big advantage is that there is no unique driving dimension.
   a. Double-click the value: the dialog box displays an icon next to the value which shows that this is an equivalent dimension. The value can be modified in the same way as you modify a standard constraint





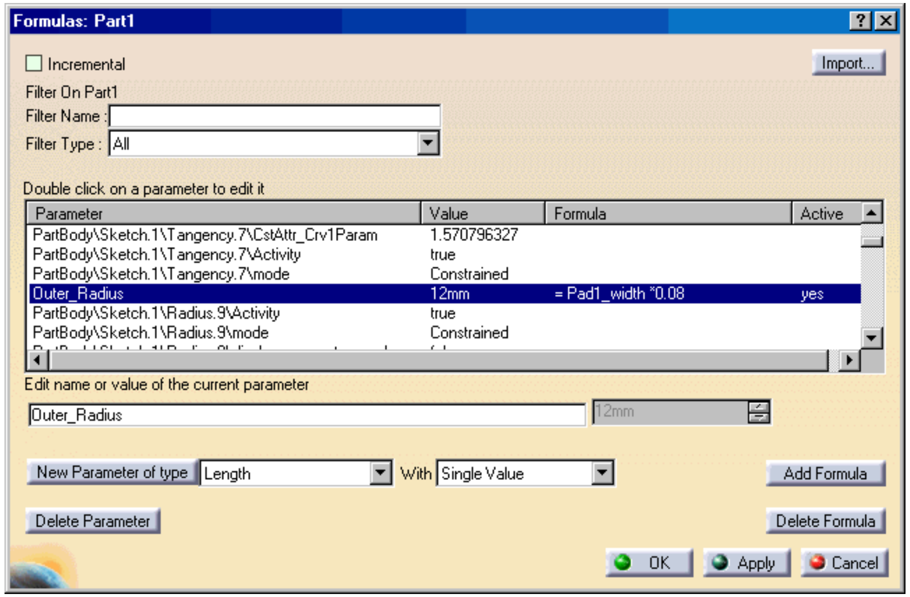
Student Notes:

Formula

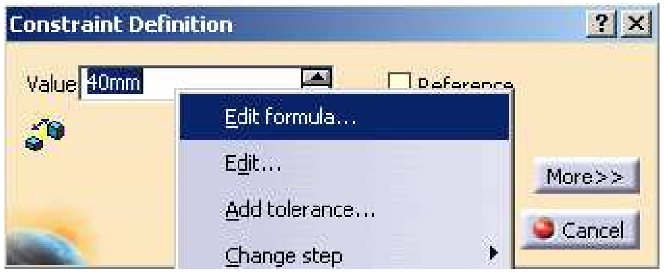
A formula is used to relate one parameter to another. It can be created by:

1. Using the formula window.
2. Editing the dimension value with the contextual menu.

1



2

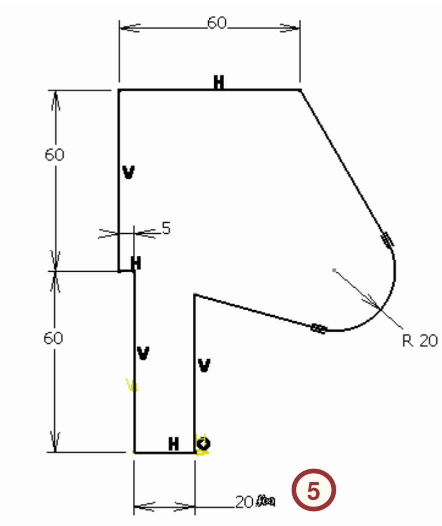


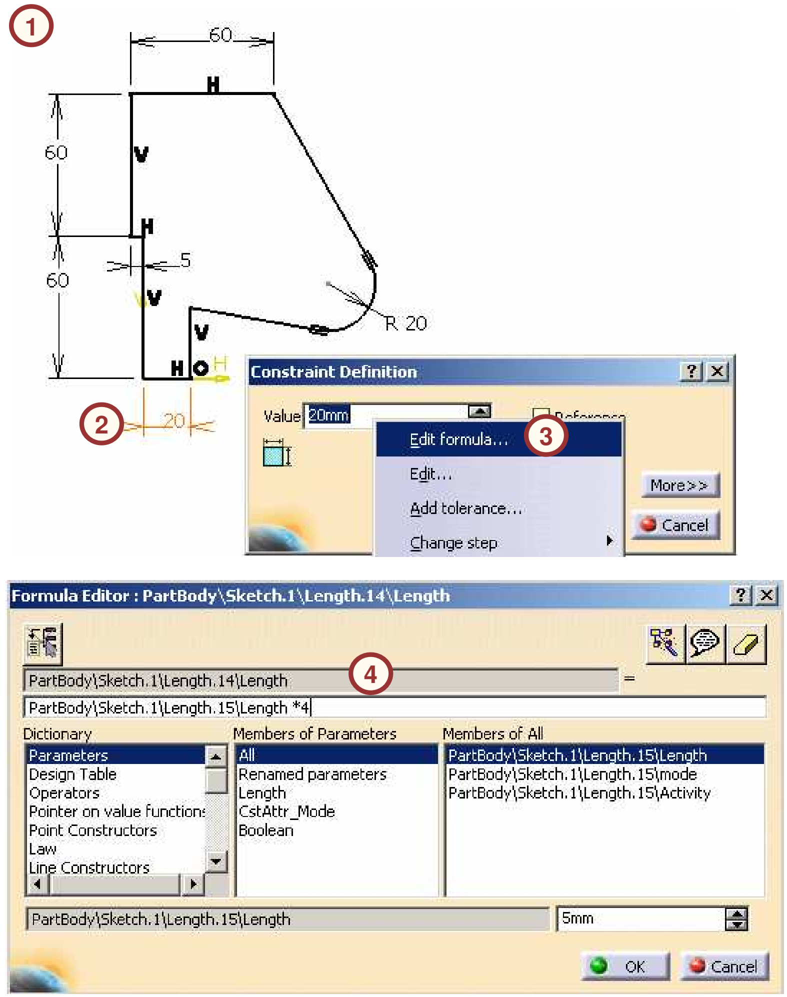
Student Notes:

Creating a Formula

Use the following steps to create a formula through a sketcher:

1. Edit the sketch to enter the Sketcher Workbench.
2. Double-click the dimension to which you want to associate a formula.
3. From Contextual menu in the value field select **Edit Formula**.
4. In the **Formula Editor** dialog box, add a relation.
5. Symbol f(x) appears in front of a dimension to which the formula is associated.

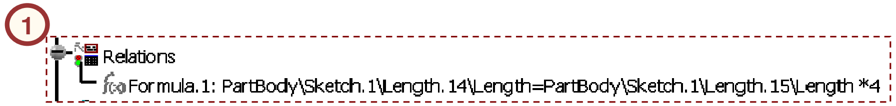


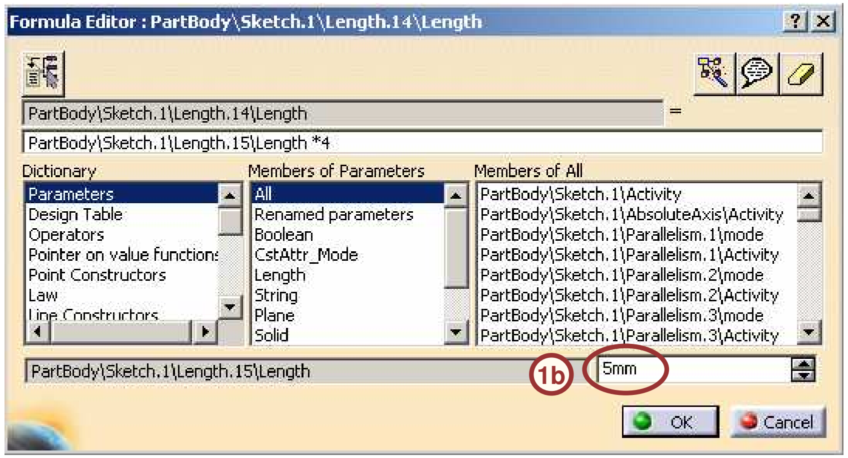


Student Notes:

Editing Formula through Feature Tree

1. The Formula feature is displayed in the Relations node of the feature tree.
   a. Double-click it to view the relation and to modify it.
   b. The highlighted value is a driving dimension.

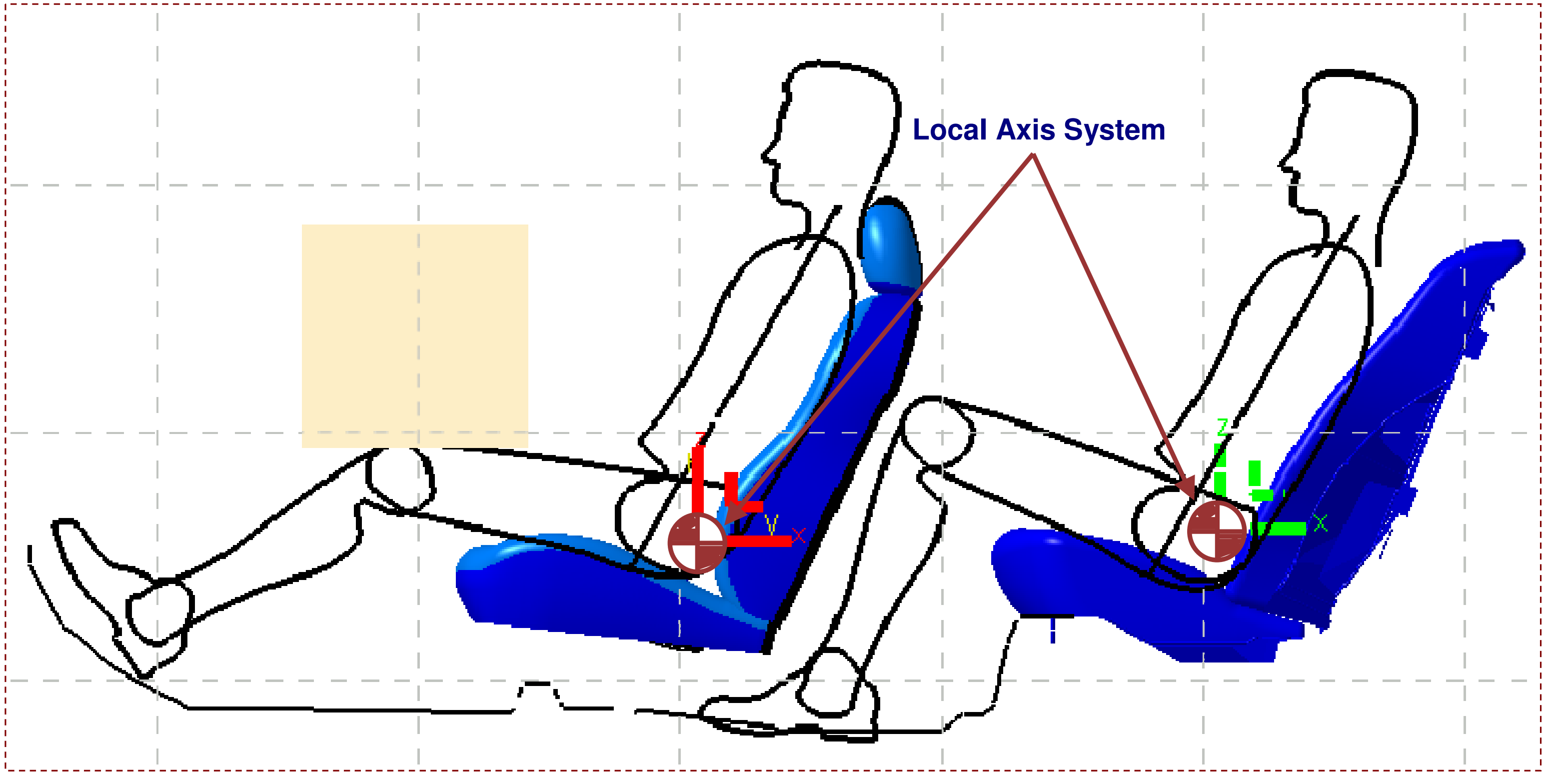




Student Notes:

Create an Axis System

A local axis is a user-defined axis system that can be used to define local coordinates. For example, it is often easier to build a point by coordinates, with respect to a local axis rather than creating it in the absolute coordinates system. An axis system can automatically be generated when a new part is created. This axis system is defined at the origin of the model and uses the default reference planes for direction.



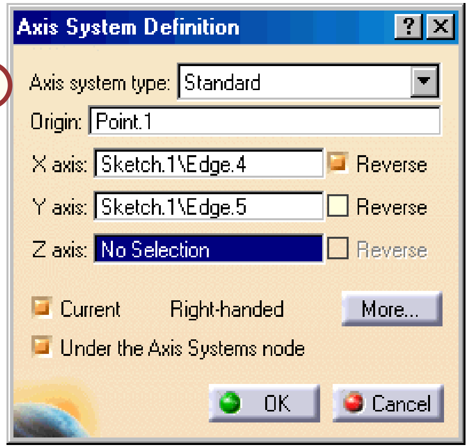
Student Notes:

Types of Axis System

The following types of local axis systems can be defined:

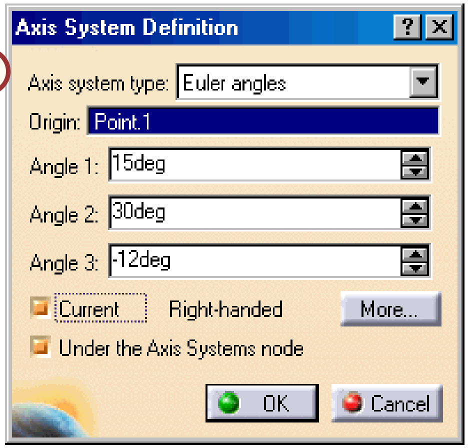
1. Standard Axis System: is defined by a origin and three orthogonal directions.
2. Rotation Axis System: is defined by an origin, three orthogonal directions, and an angle based on a selected reference.
3. Euler Axis System: is defined using Euler angles to specify its orientation.

1



2

3

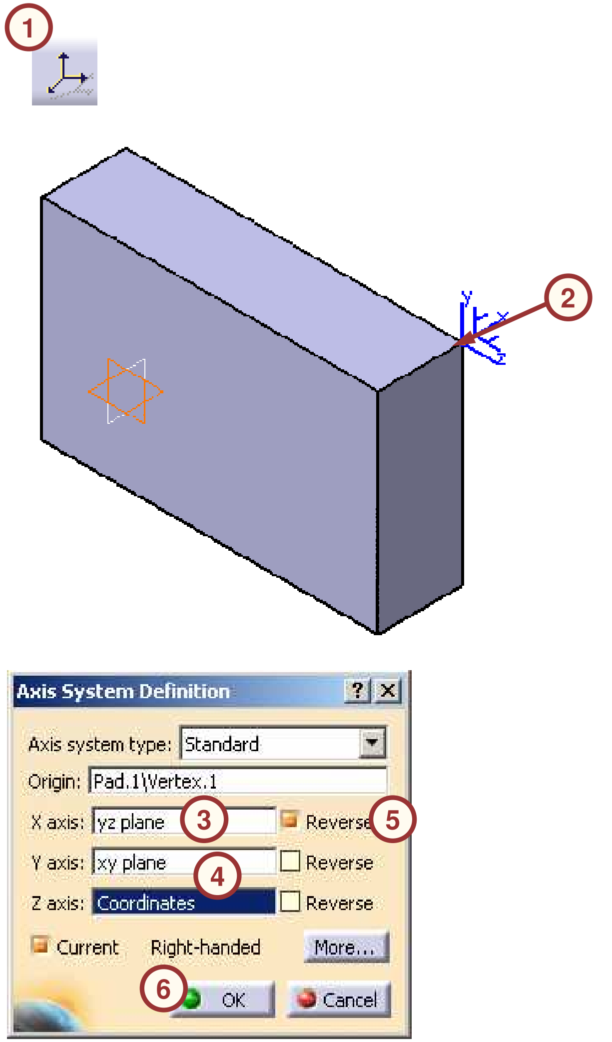


Student Notes:

Creating an Axis System

Use the following steps to create a local standard axis system:

1. Click the **Axis System** icon.
2. Select the pad vertex as origin point.
3. To define an axis direction, click on the appropriate axis field and select an element to define direction. For example, to define the direction of the X axis, click on the X axis field and select the element to define the direction.
4. Click on a second axis field and define its direction. The direction of the third axis will automatically be defined based on the previous selections.
5. Select the Reverse option to reverse the axis direction, if necessary. In this example, the X axis is reversed.
6. Click **OK** to create the axis.

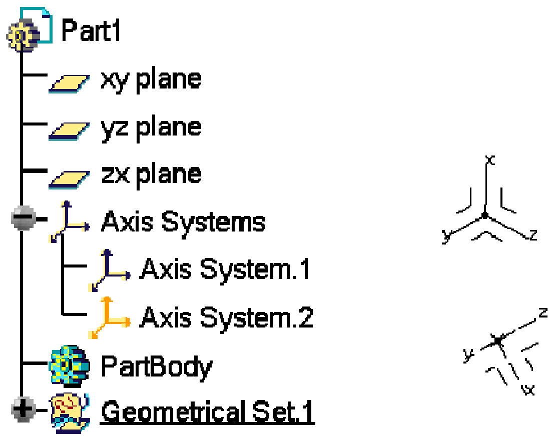


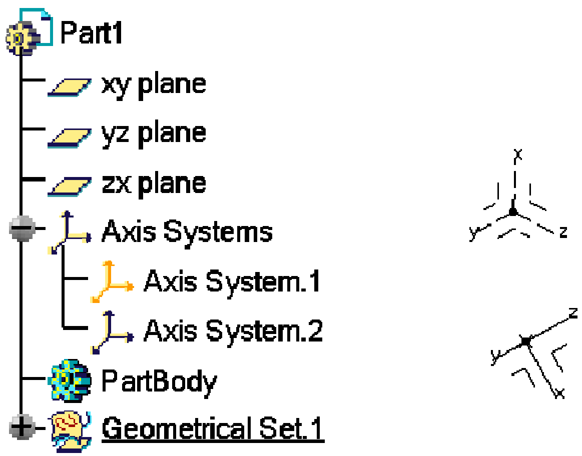
Axis System: Set as Current

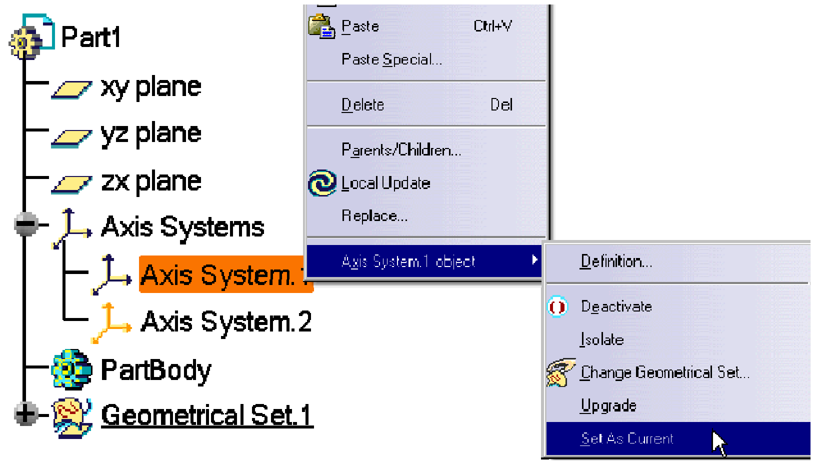
Student Notes:

By default, the last created axis system becomes the active system. The current axis system is highlighted in the specification tree, and is displayed with solid lines on the model. All the other axis systems are dashed lines on the model.

To change the active axis system, right-click on the system to be made current and click **Axis System.x object** > **Set as Current**.



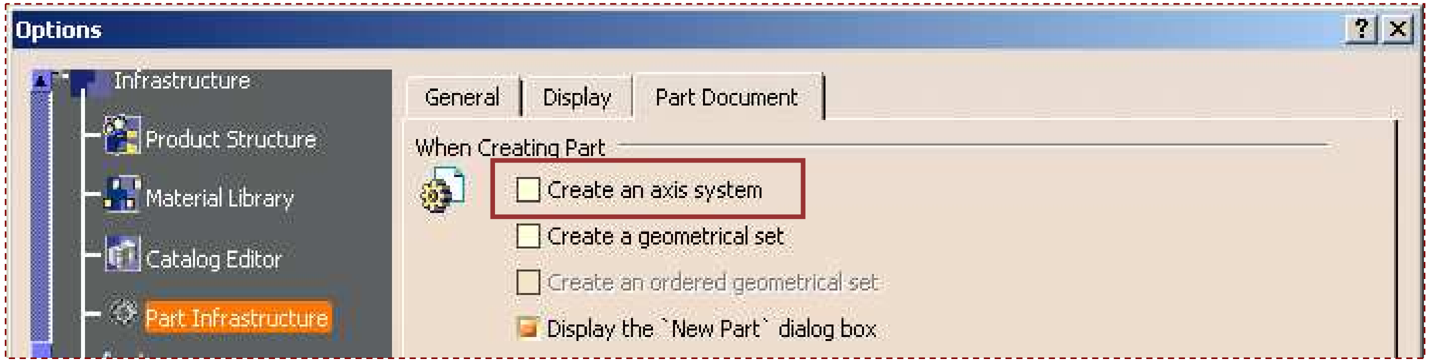




Student Notes:

Axis System with New Part

An axis system can automatically be generated when a new part is created. This axis system is defined at the origin of the model and uses the default reference planes for direction. If this option needs to be changed, click **Tools** > **Options** > **Infrastructure** > **Part Infrastructure**. Then from the **Part Document** tab, select the **Create an Axis System when creating a new part option**.



Student Notes:

Create Multi-profile Sketch Features

In this section, you will learn how to create several features using one sketch.

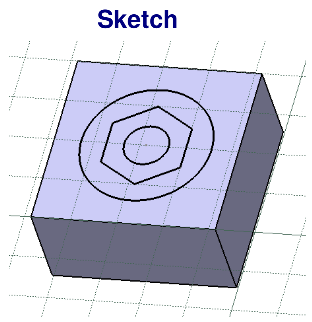
Use the following steps:

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

Student Notes:

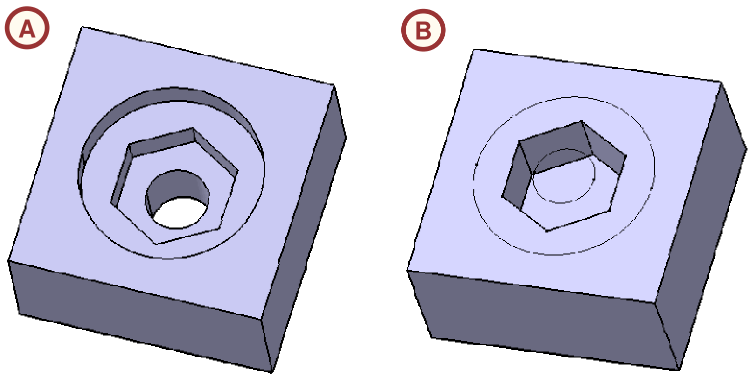
Multiple Profiles

Multi-profile sketches are sketches that contain more than one closed profile. This helps to quickly create multiple features using only one sketch. Therefore, if the sketch is removed, the corresponding features are also removed. This method is not recommended if the sketched profiles are complicated because editing all the individual profiles can be difficult when they are within one sketch.



So far, you have used multi-profile sketches to create multiple pockets using a single pocket feature. Multi-profile sketches can also be used to create the following:

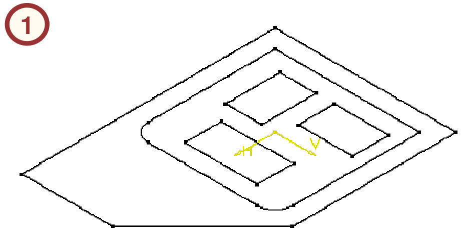
A. Multi-pads/pockets
B. Sub-elements of a Sketch



Student Notes:

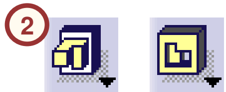
Multi-Pads/Pockets (1/2)

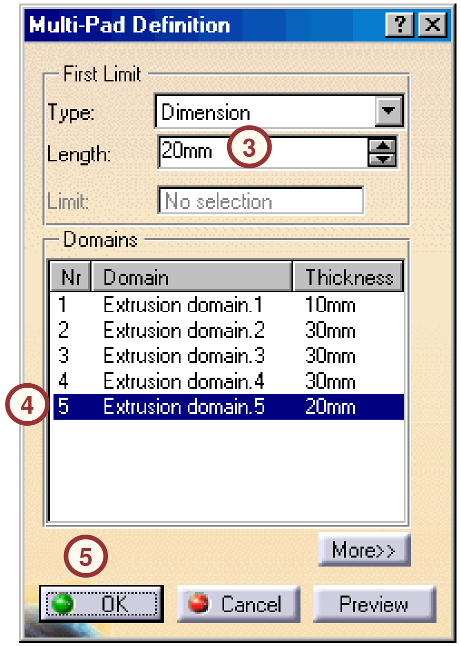
Multi-pads and pockets are features that create several pads/pockets in one operation. These tools require a sketch with at least two closed profiles that do not intersect. Consider using these tools as a fast way to create multiple features.

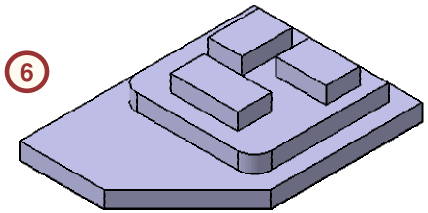


Use the following steps to create a multi-pad:

1. Create a sketch containing at least two closed profiles.
2. Select the sketch and click the **Multi-pad** or **Multi-Pocket** icon. In this example, Multi-Pad is selected.
3. Select the first extrusion domain. Specify a depth value for the closed profile.
4. Repeat step 3 for each extrusion domain.
5. Click **OK** to create the Multi-pad.
6. The Multi-pad feature is created in the specification tree.







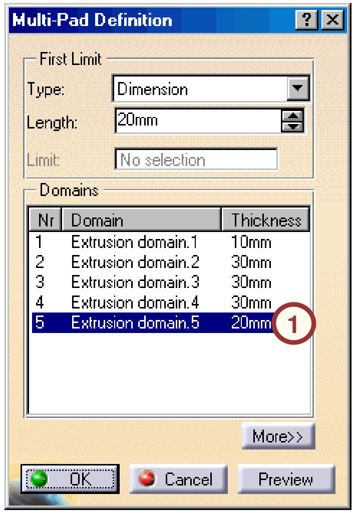
Student Notes:

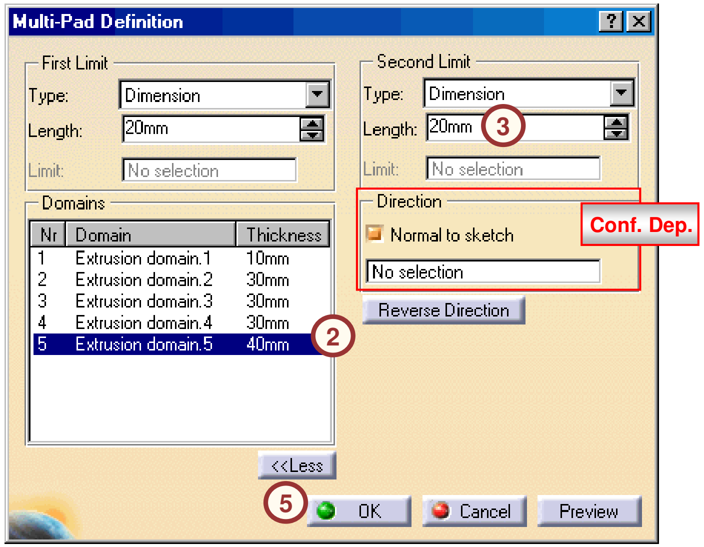
Multi-Pads/Pockets (2/2)

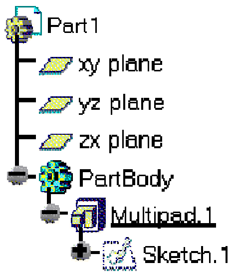
Like standard pads and pockets, multi-pad/pockets can be extruded in two directions.

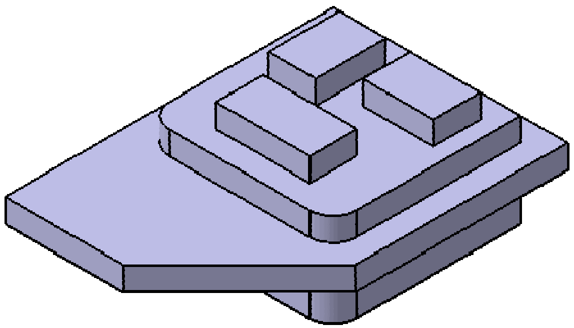
Use the following steps to extrude the multi-pad in a second direction:

1. Select **More>>**.
2. Select extrusion domain. In this example, Extrusion domain.5 is extruded in two directions.
3. Specify the second depth for the profile.
4. Repeat step 2 for each extrusion domain. In this example, only Extrusion domain.5 is extruded in two directions.
5. Click **OK** to complete the feature.







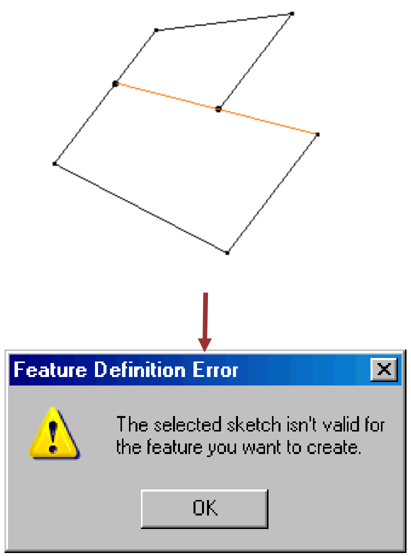


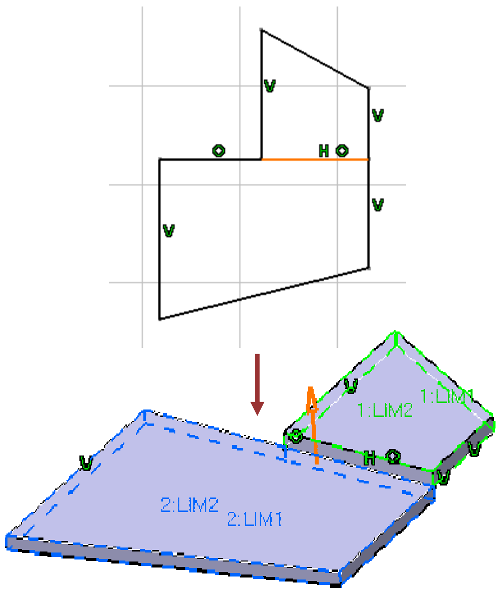
Student Notes:

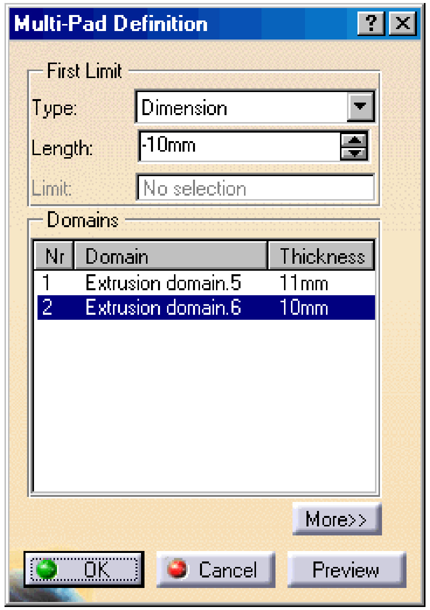
Solving Ambiguity for Multi-Pads/Pockets

Careful thought should 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. Use the **Break** tool in the Sketcher workbench to create proper profiles if necessary.

For example, two profiles are created as shown within the same sketch. If the shared line between the two profiles is created as one geometric element, the multi-feature fails. The top profile does not form a closed loop. By breaking the shared line into two separate segments, the top profile is now closed.







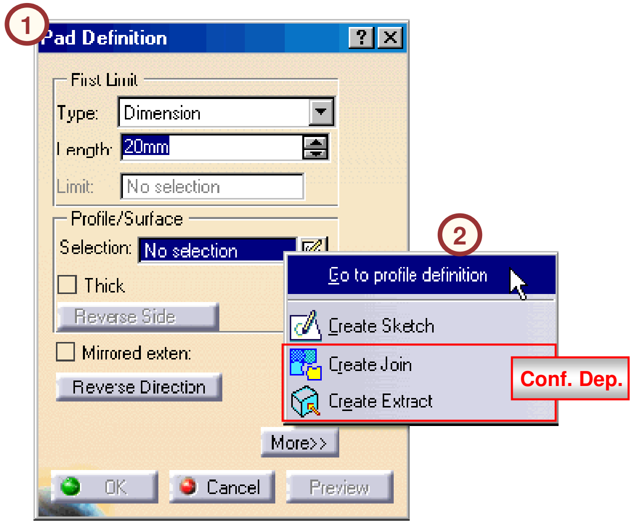
Student Notes:

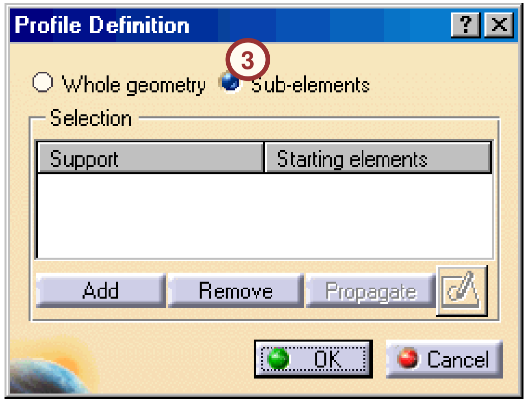
Sub-Elements of a Sketch (1/2)

In some cases, you may need to create a feature that uses only one profile of a multi-profile sketch. This is done using the **Sub-Elements** option. This tool allows you to extract only the profile you need from the sketch. Deleting or modifying the sketch will affect all features associated with it, because several features can be based on the same sketch.

Follow these steps to use a sub-element of a sketch:

1. Access the feature creation window (in this example, a Pad feature). Do not select the sketch before selecting the icon.
2. Right-click the Profile Selection field and select **Go to Profile Definition**.
3. Select **Sub-elements** radio button.

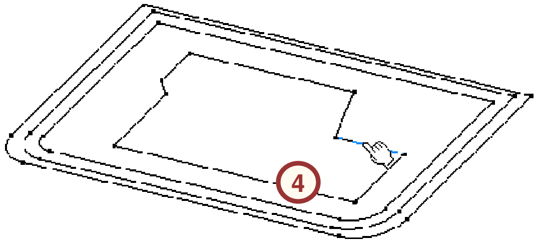




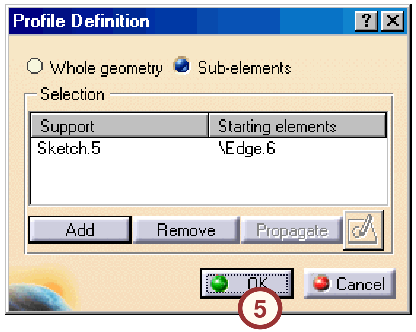
Student Notes:

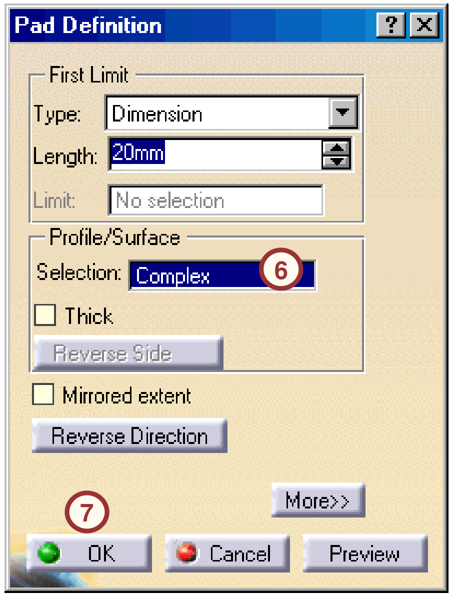
Sub-Elements of a Sketch (2/2)

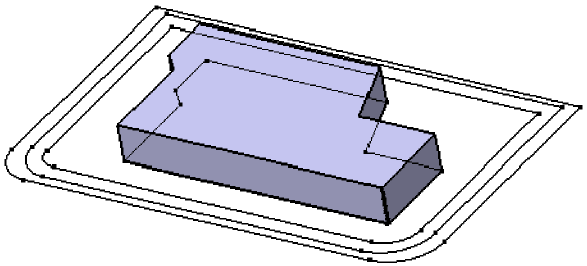
Follow these steps to use a sub-element of a sketch (continued):



4. Select an element on the required profile. All the entities that are part of the same loop as the selected geometry will get highlighted.
5. Click **OK** to return to the **Pad Definition** dialog box.
6. The Profile Selection field now says gets updated to “Complex”.
7. Click **OK** to complete the feature.







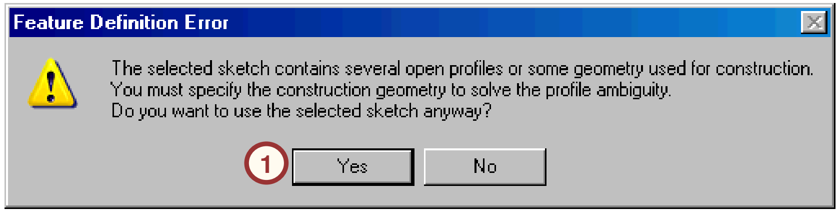
Student Notes:

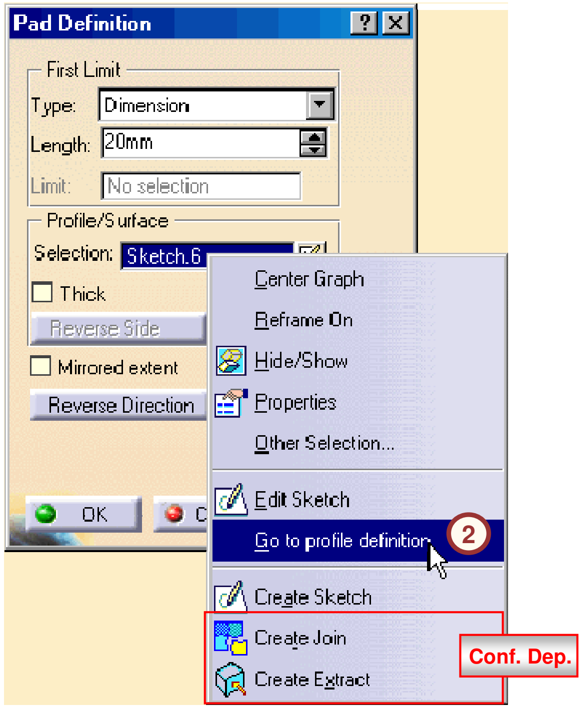
Solving Ambiguity for Sub-Elements (1/2)

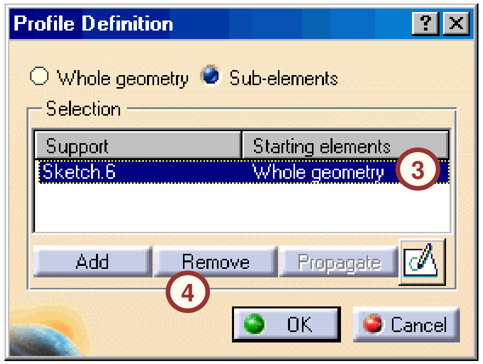
If you select a multi-profile sketch before selecting the feature tool, an error appears, indicating profile ambiguity. This is because multiple profiles are contained within the sketch and CATIA is unsure how to create the feature.

Use the following steps to solve the error:

1. Select **Yes** on the error window.
2. Right-click the Profile field and click **Go to Profile definition**.
3. Select the geometry in the selection window
4. Click the **Remove** button.



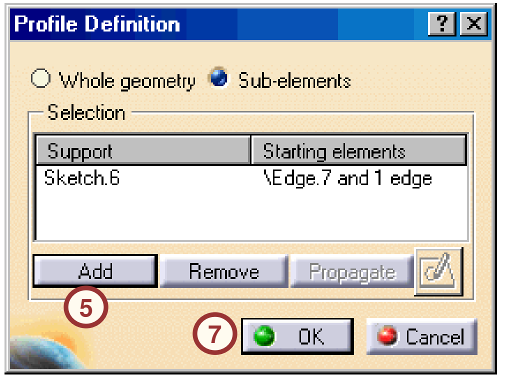


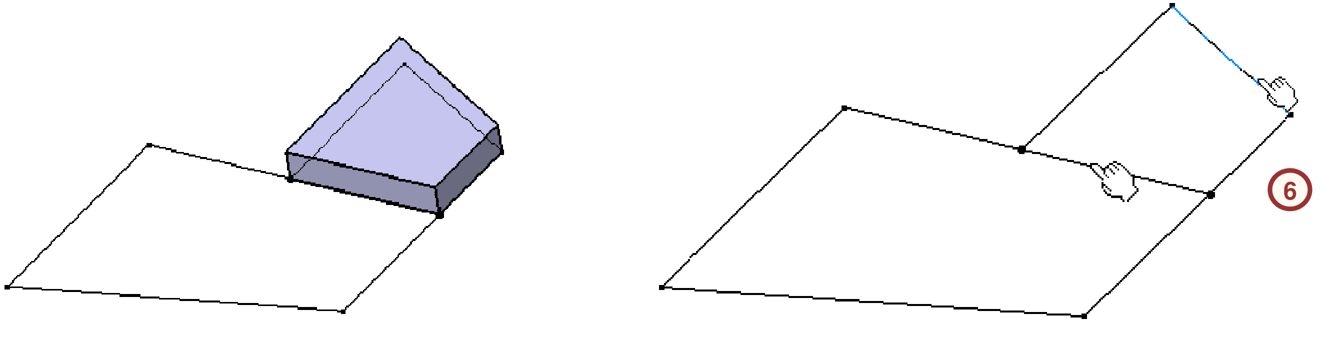


Solving Ambiguity for Sub-Elements (2/2)

Use the following steps to solve the error (continued):

5. Click the **Add** button.
6. Add the correct profile.
7. Click **OK** to return to the feature definition window.
8. Complete the definition and click **OK**.





Student Notes:

Student Notes:

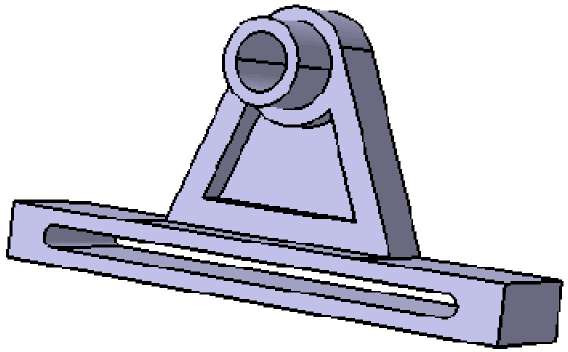
Exercise: Multiple Profile Sketch Features

Recap Exercise

In this exercise you will create a multi-pad feature and two pocket features using only sub-elements of a sketch. You will use some of the additional sketcher tools you have learned in this lesson to complete the exercise. Detailed instructions for this exercise are provided.

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

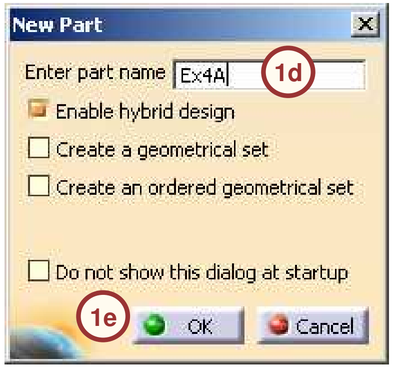
- **Create a multi-profile sketch**
- **Create a multi-pad feature**
- **Create a pocket using a sub-element of a sketch**
- **Use the Re-limitation, Projection, and Transformation tools in the Sketcher workbench**



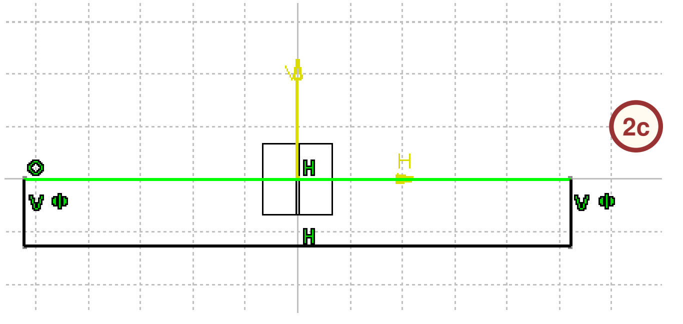
Student Notes:

Do it Yourself (1/16)

1. **Create a new part.**
   - Create a new part named Ex4A.CATPart.
     a. Click **File > New.**
     b. Select **Part**.
     c. Click **OK**.
     d. Type [Ex4A] as the part name.
     e. Click **OK**.



2. **Create a multi-profile sketch.**
   - Create a sketch that contains more than one closed profile. You will use this sketch to create a Multi-pad in a later step.
     a. Select the YZ plane.
     b. Click the **Sketcher** icon.
     c. Create the rectangle as shown. Make it symmetric about the ZX plane.



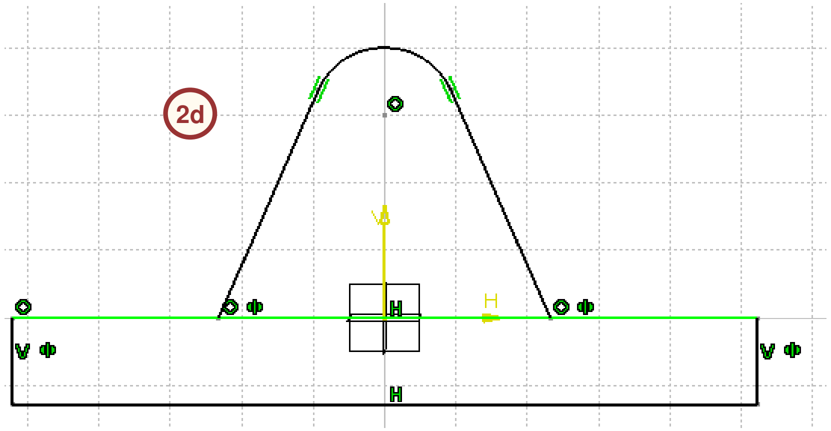
Student Notes:

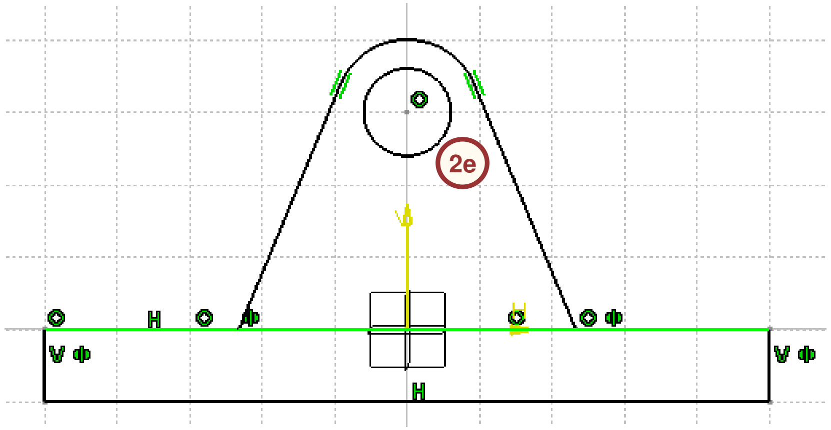
Do it Yourself (2/16)

2. Create a multi-profile sketch (continued...).

d. Create a profile as shown. The profile contains two lines and a tangent arc. Ensure that the lines are coincident with the top line of the rectangle and symmetric about the ZX plane. Make the center of the arc coincident with the ZX plane.

e. Create a circle whose centre is coincident with the centre of the arc.





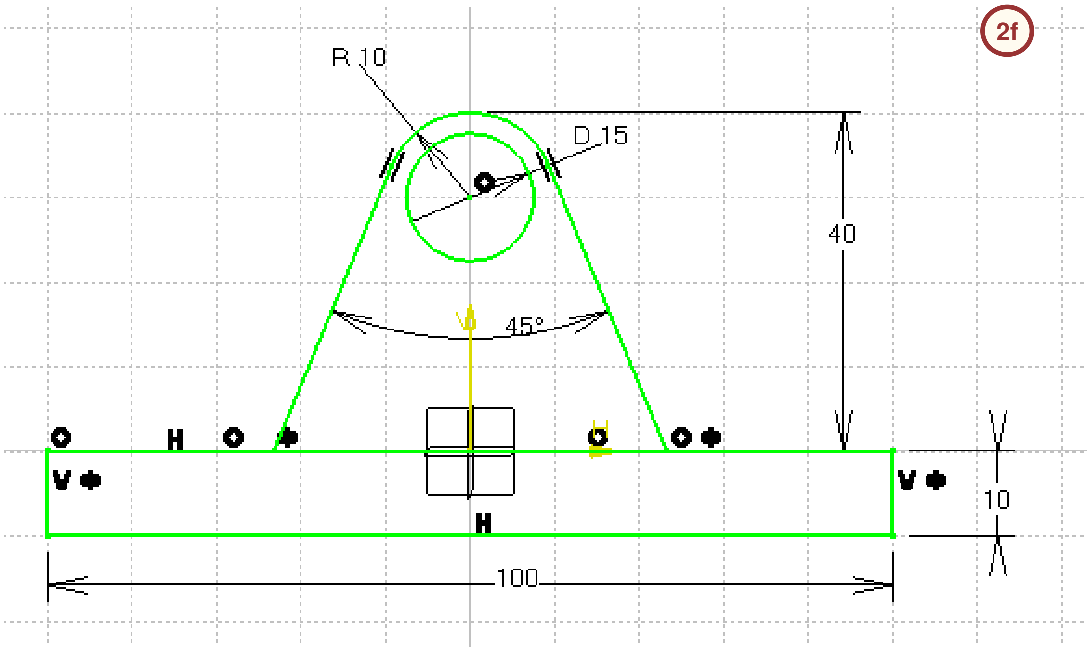
Student Notes:

Do it Yourself (3/16)

2. Create a multi-profile sketch (continued...).

f. Dimension the sketch as shown.

g. Exit the Sketcher.



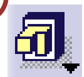
Student Notes:

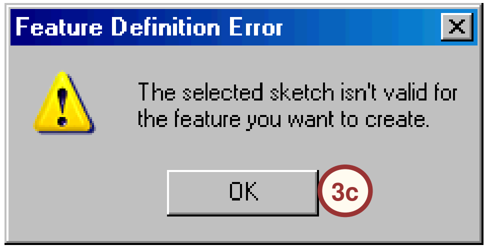
Do it Yourself (4/16)

3. Create a multi-pad feature.

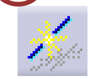
- Create a multi-pad feature using the sketch created in the last step.
  a. Select the sketch.
  b. Click the **Multi-Pad** icon.
  c. Read the error and click **OK**.
  d. The error indicates that the sketch is not valid. Can you guess what is the problem?
  e. Cancel the multi-pad creation by selecting the **Multi-Pad** icon again.

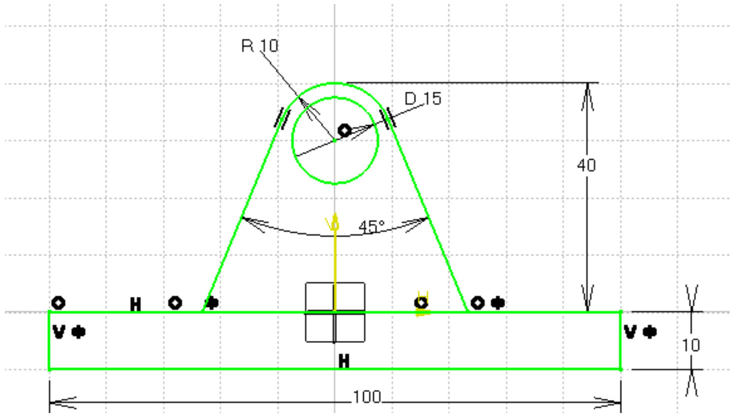
3b





4b





4. Correct the sketch.

- The sketch is not valid because the top line of the rectangle is one piece and needs to be shared by two profiles. It is not possible to create the top profile using this sketch as it is.
  a. Double-click on the sketch to edit.
  b. Click the **Break** icon.

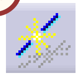
Student Notes:

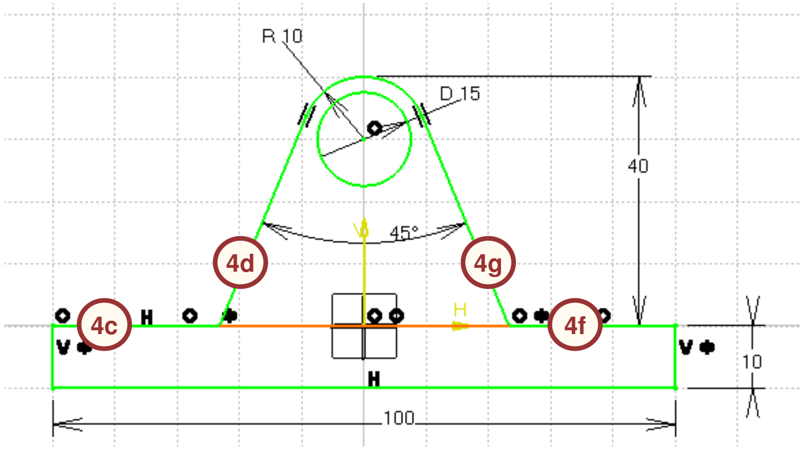
Do it Yourself (5/16)

4. Correct sketch (continued...).

c. Select the top line of the rectangle as the line to break.
d. Select one of the angled lines as the breaking element.
e. Click the **Break** icon again.
f. Select the top line of the rectangle near the other angled line.
g. Select the other angled line as the breaking element.
h. The line now consists of three separate entities. The top profile can now be constructed.
i. Exit the Sketcher workbench.

4e



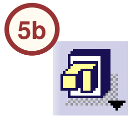


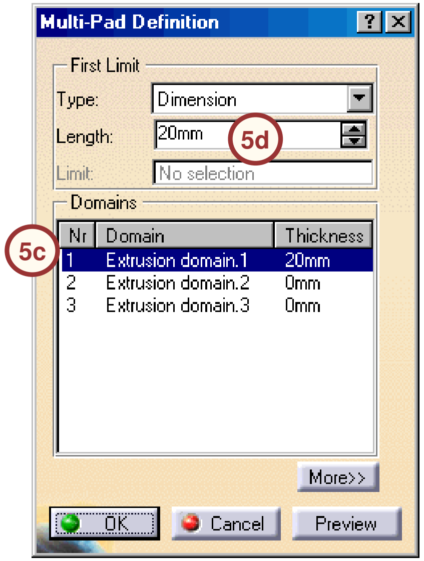
Student Notes:

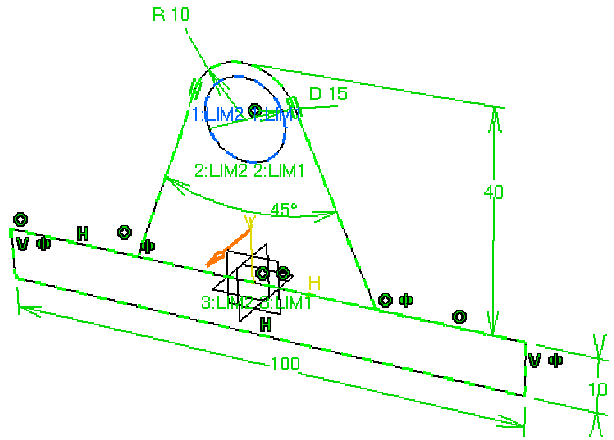
Do it Yourself (6/16)

5. Create multi-pad.

- Now that the sketch is correct. Create the multi-pad feature.
  - a. Select the sketch.
  - b. Click the **Multi-Pad** icon.
  - c. Select the first domain. The circle highlights on the model.
  - d. Specify a depth of [20mm].







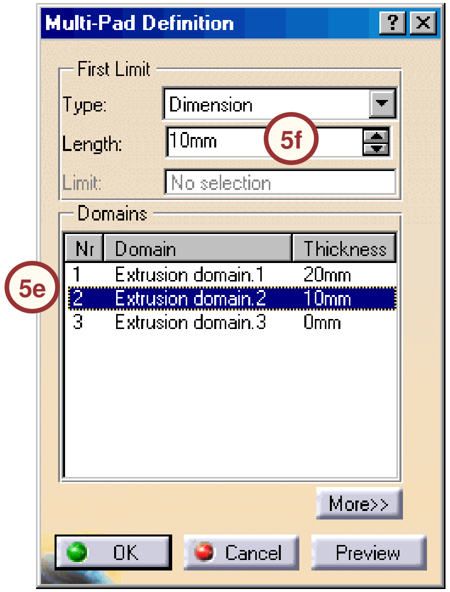
Student Notes:

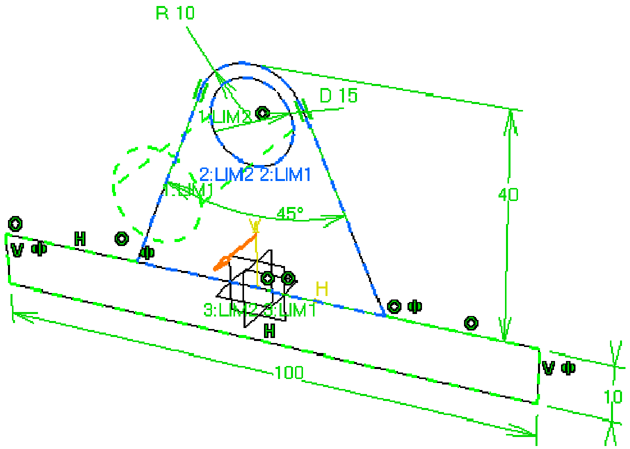
Do it Yourself (7/16)

5. Create multi-pad (continued...).

e. Select the next profile in the dialog box. The top profile is highlighted on the screen.

f. Specify a depth of [10mm].





Student Notes:

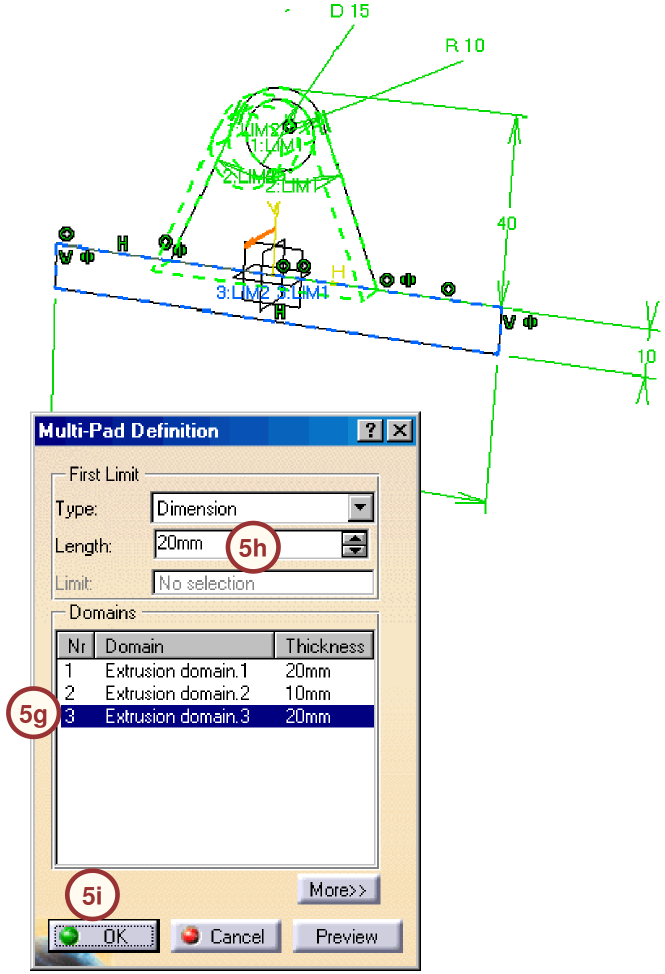
Do it Yourself (8/16)

5. Create multi-pad (continued...).

g. Select the final profile in the dialog box. The rectangular profile is highlighted.

h. Specify a depth of [20mm].

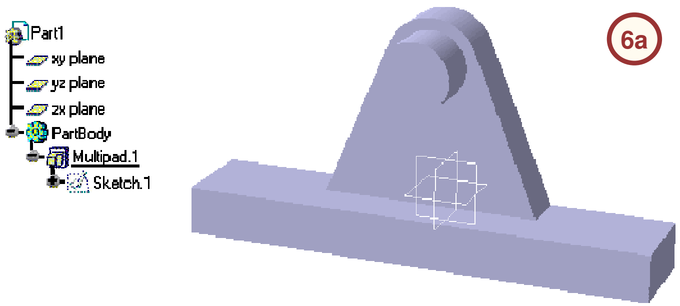
i. Click **OK** to complete the feature.



6. Create a multi-profile sketch.

- Create a multi-profile sketch that will be used to create separate pocket features.

  a. Select the top face of the rectangle and access the Sketcher workbench.



Student Notes:

Do it Yourself (9/16)

6. Create a multi-profile sketch (continued...).

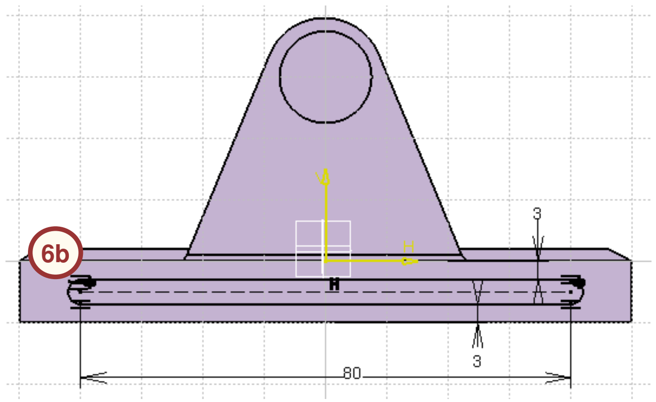
b. Create an elongated hole as shown.

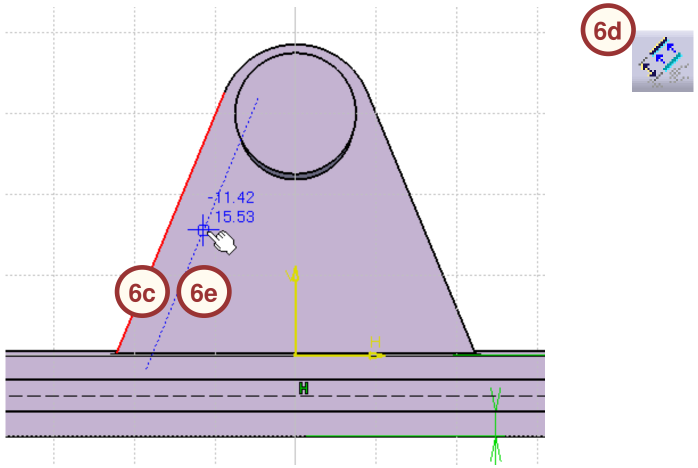
c. Select the 3D geometry edge as shown.

d. Click the **Offset** icon.

e. Move the pointer towards the center of the model. This indicates the direction of the offset.

f. Press the <Tab> key several times until the Offset field is highlighted in the **Sketch Tools** toolbar. Specify an offset of [5mm].





Student Notes:

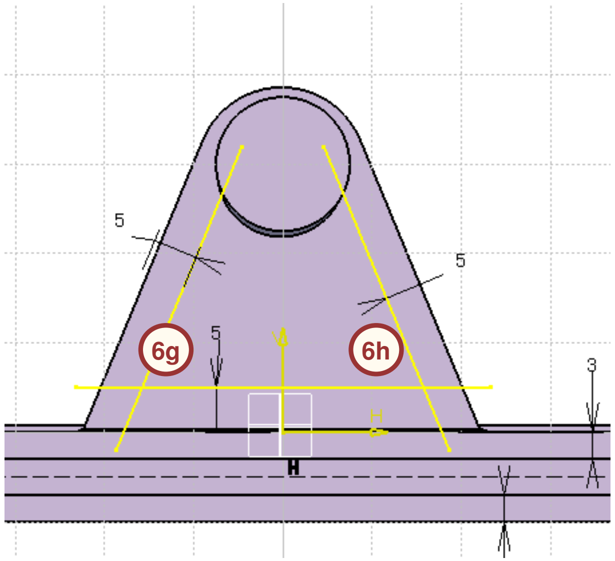
Do it Yourself (10/16)

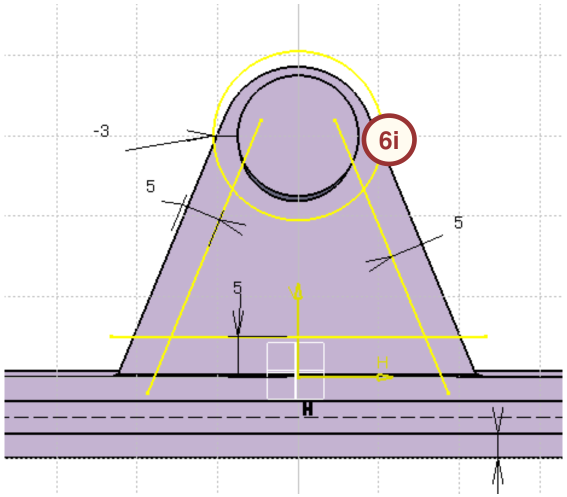
6. Create a multi-profile sketch (continued...).

g. Press the <Enter> key to create the offset geometry. Notice that the offset line is yellow. This indicates that it is projected from the 3D geometry.

h. Offset the other edges of the 3D geometry as shown.

i. Offset the edge of the circular pad [3mm].





Student Notes:

Do it Yourself (11/16)

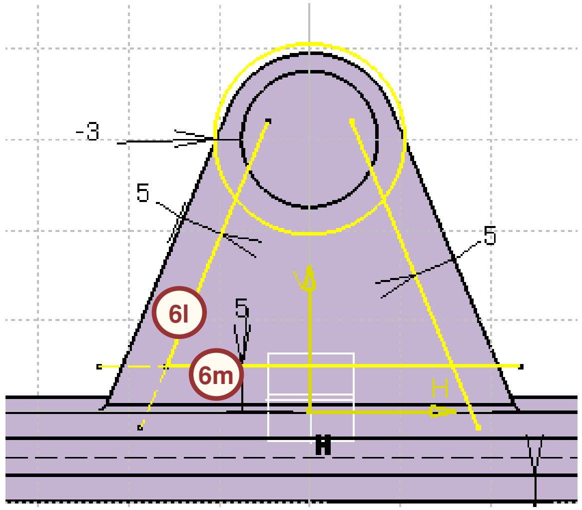
6. **Create a multi-profile sketch (continued...).**

   j. Click the **Trim** icon.

   k. Click **Trim All Elements** from the Sketch tools toolbar.

   l. Select the first edge in the area to be kept.

   m. Select the second edge in the area to be kept. The entities are trimmed.



Student Notes:

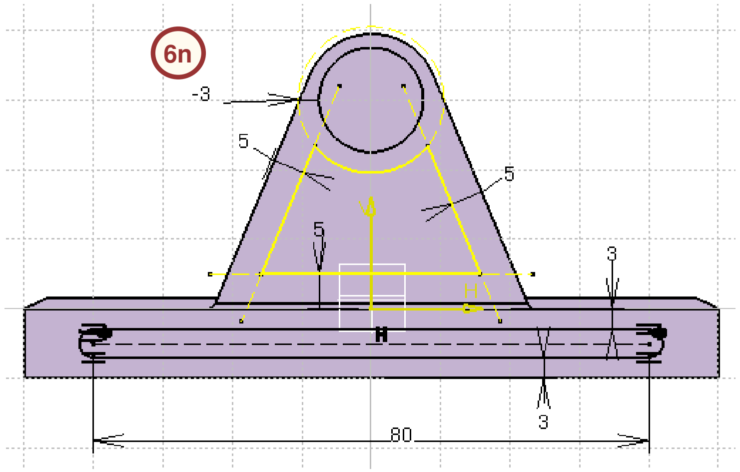
Do it Yourself (12/16)

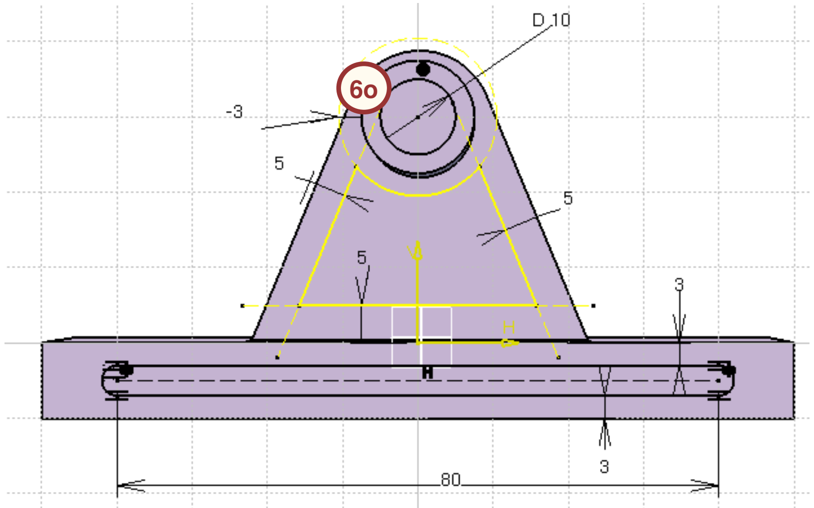
6. **Create a multi-profile sketch (continued...).**

n. Repeat the trim for the other elements.

o. Create a circle that is concentric with the circular pad.

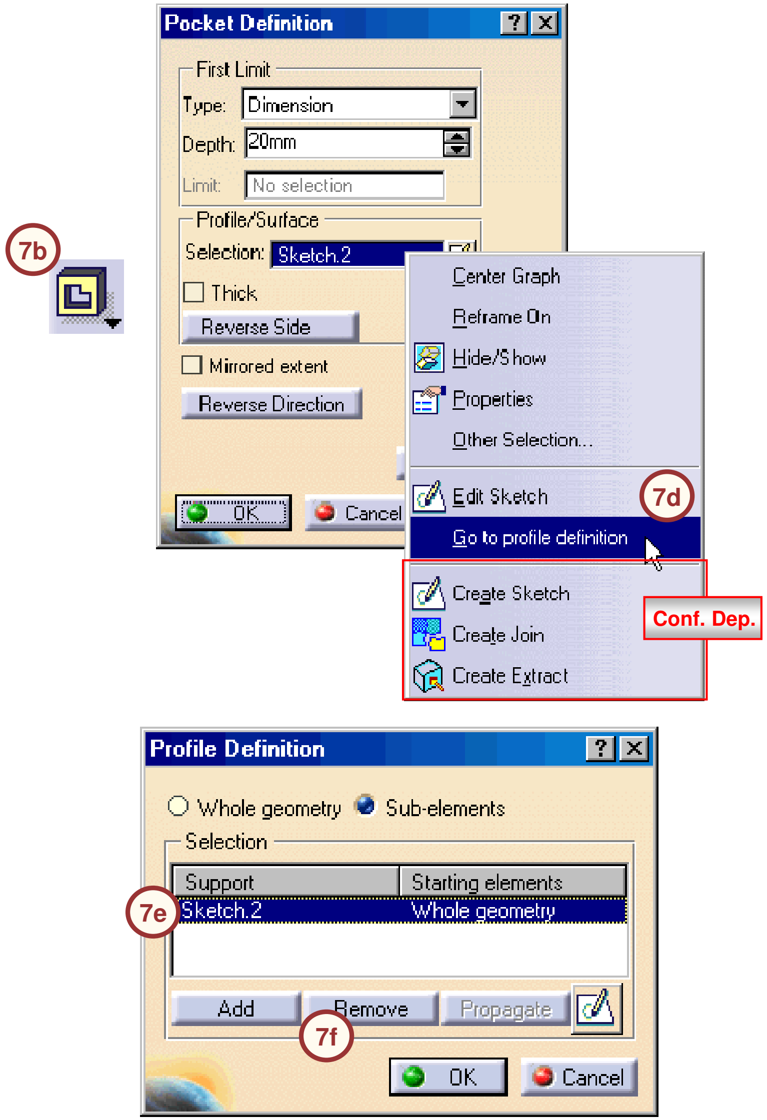
p. Exit the Sketcher workbench.





Do it Yourself (13/16)

7. **Create pockets.**
   - Use the sketch created in the last step to create two pocket features that cut through all the material. The third profile will be used in a separate feature.
     - a. Select the sketch created in step 6.
     - b. Click the **Pocket** icon.
     - c. All the profiles are highlighted. In this exercise, two of the profiles should have different depths than the third.
     - d. Right-click on the Profile Selection field and click **Go to Profile**.
     - e. Select Sketch.2 in the dialog box.
     - f. Select **Remove**.



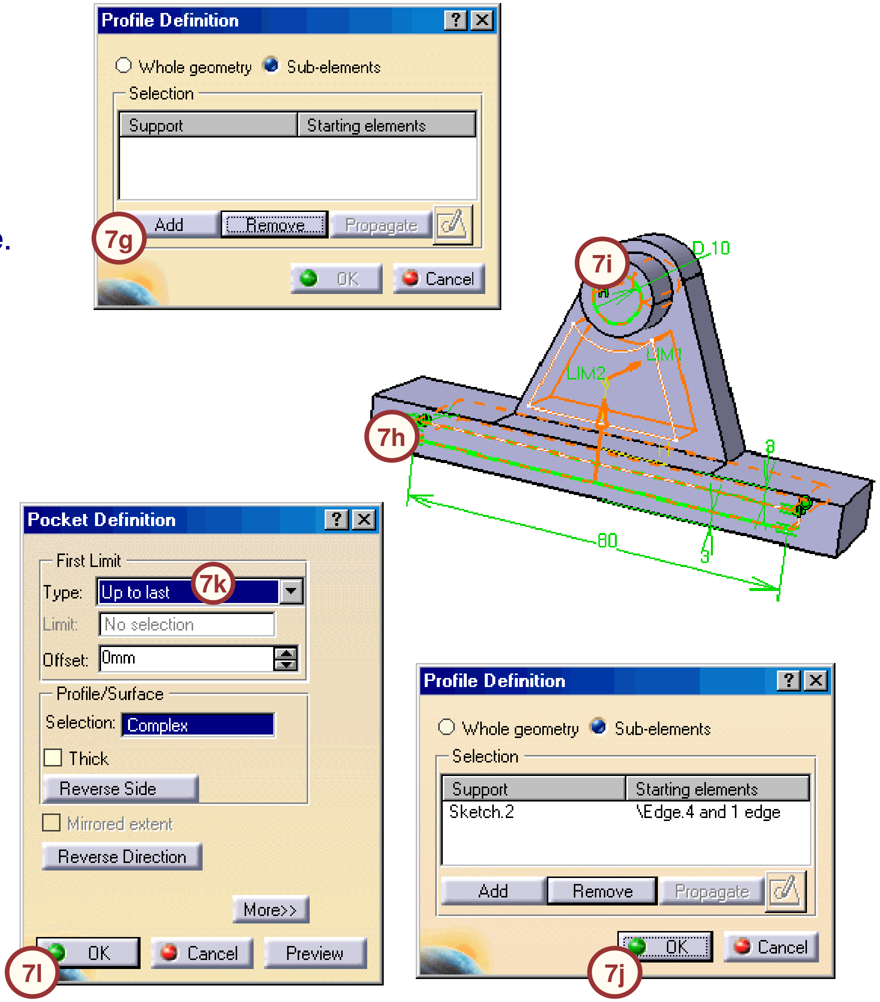
Student Notes:

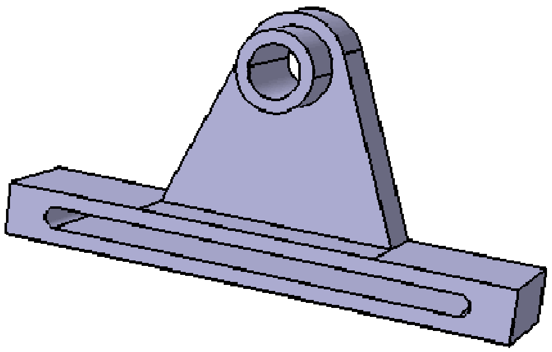
Student Notes:

Do it Yourself (14/16)

7. Create pockets (continued...).

g. Click **Add**.
h. Select the elongated hole profile.
i. Select the circle profile.
j. Click **OK**.
k. Select a depth of **Up to Last**.
l. Click **OK**.



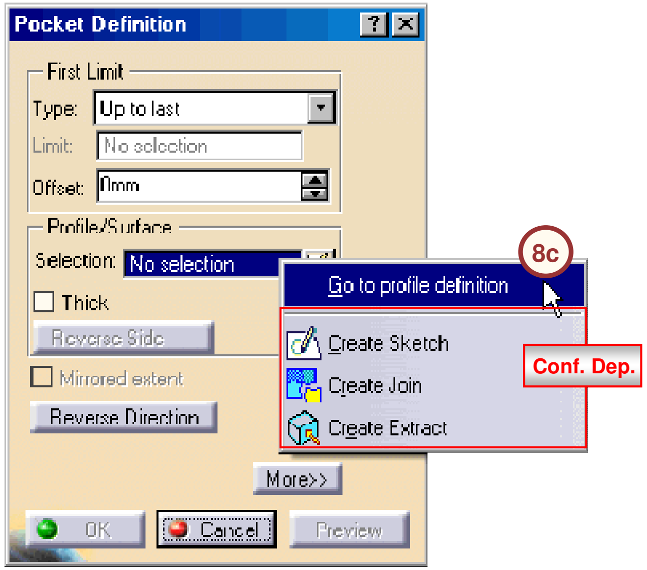


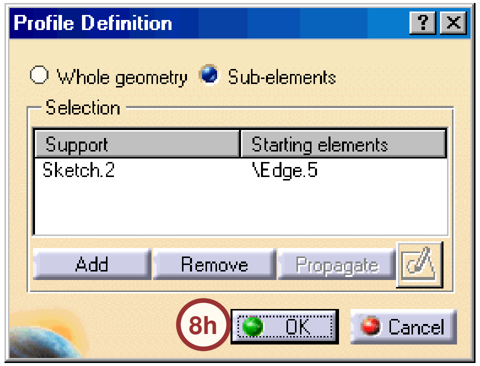
Student Notes:

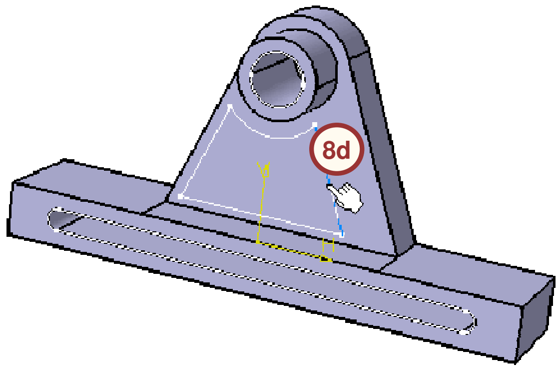
Do it Yourself (15/16)

8. Create pocket.

- Use the sketch created in step 6 to create the final pocket feature.
  a. Show Sketch.2 but do not select it.
  b. Click the **Pocket** icon.
  c. Right-click on the Profile Selection field and click **Go to Profile definition**.
  d. Select the center profile.
  e. Click **OK** to the Profile Definition dialog box.







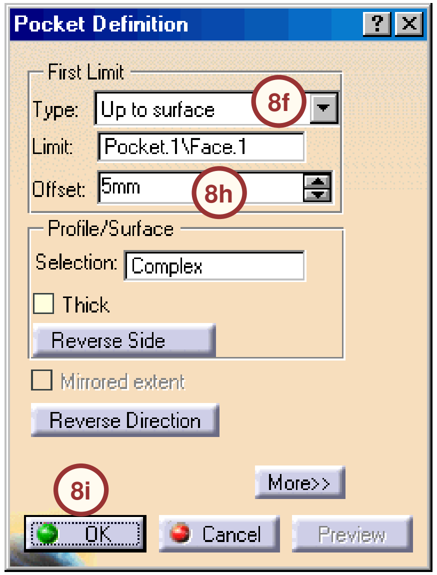
Student Notes:

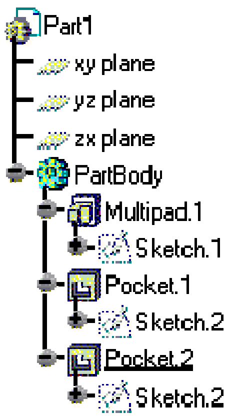
Do it Yourself (16/16)

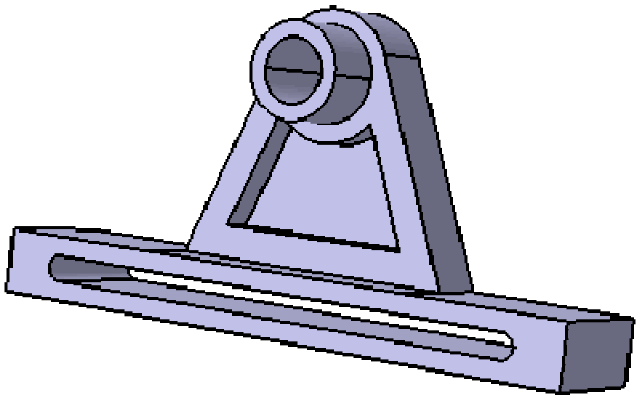
8. Create pocket (continued...).

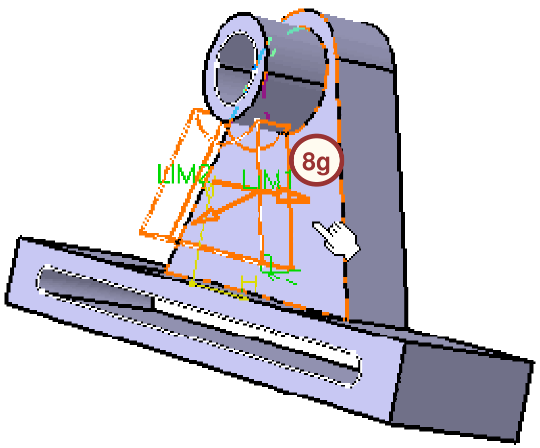
f. Select a depth of **Up To Surface**.
g. Select the surface shown.
h. Specify an offset of [5mm].
i. Click **OK**.
j. Hide Sketch.2 again.

9. Save and close the file.





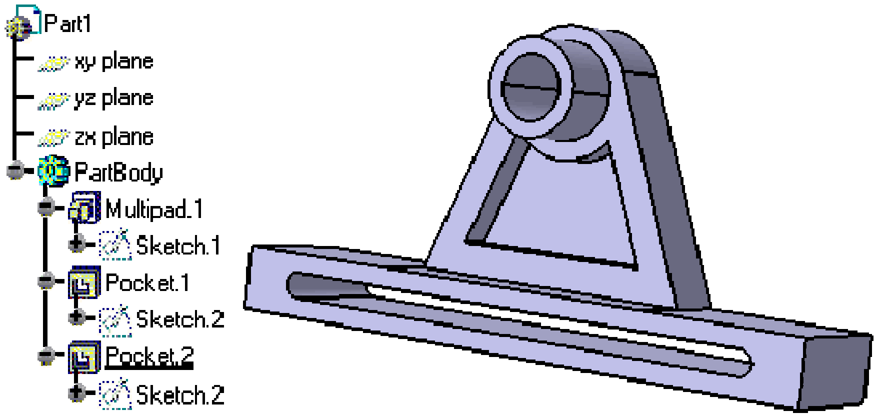




Student Notes:

Exercise Recap: Multiple Profile Sketch Features

- ✓ Create a multi-profile sketch
- ✓ Create a multi-pad
- ✓ Create pockets using sub-elements of a sketch
- ✓ Use re-limitation and transformation tools in the Sketcher workbench



Student Notes:

Exercise: Sketch Analysis and Pocket

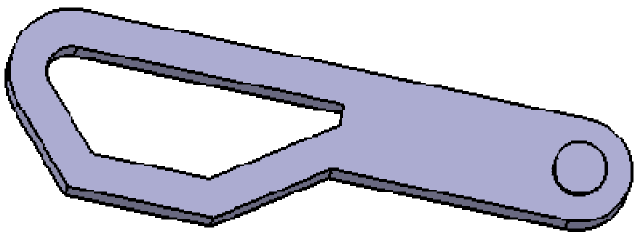
Recap Exercise

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**



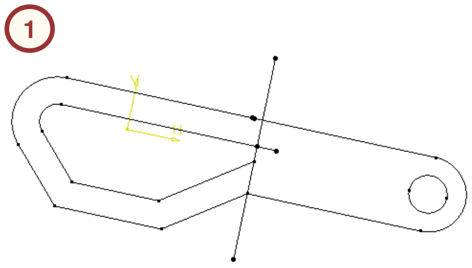
Student Notes:

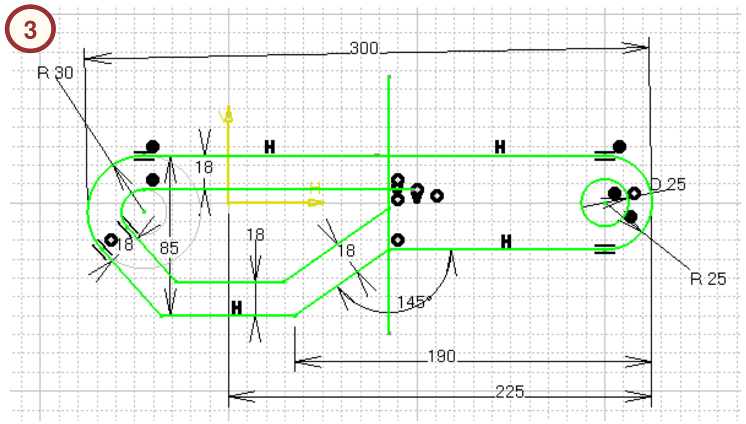
Do it Yourself (1/7)

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.

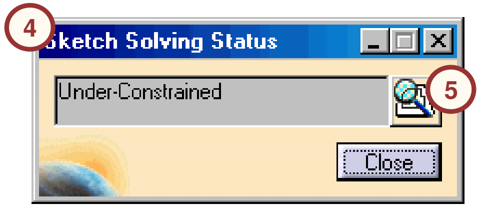
3. **Edit the sketch.**
   - Access the Sketcher workbench for Sketch.1 to investigate the sketch.



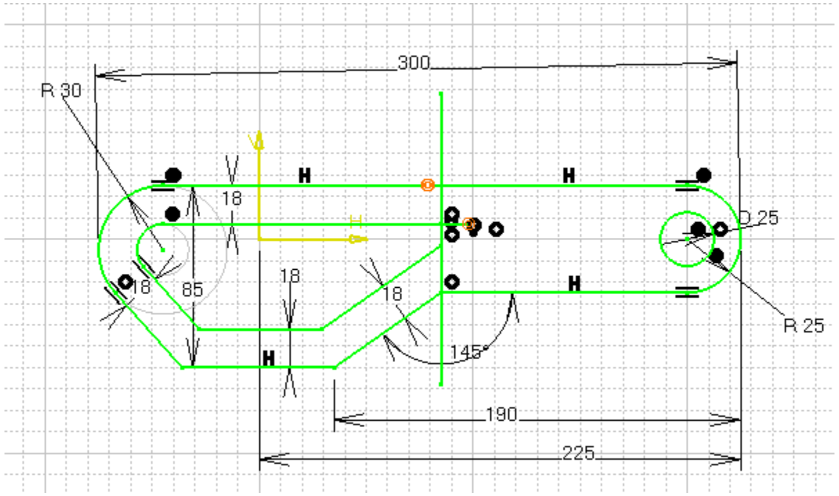


Do it Yourself (2/7)

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.



Student Notes:

Student Notes:

Do it Yourself (3/7)

6. Review the geometry.

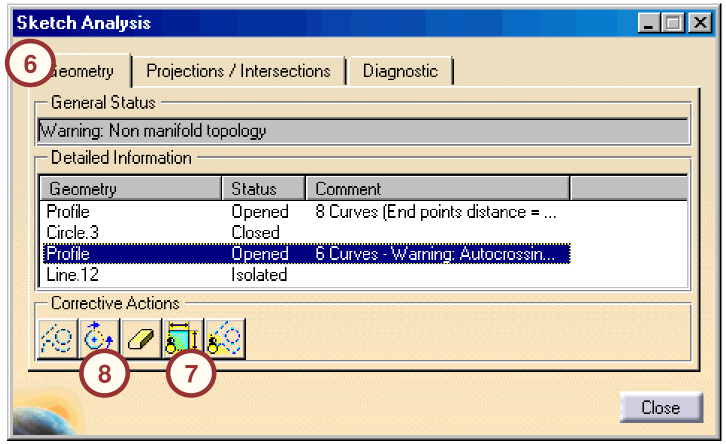
- Select the **Geometry** tab. The sketch contains two open profiles and an isolated line.

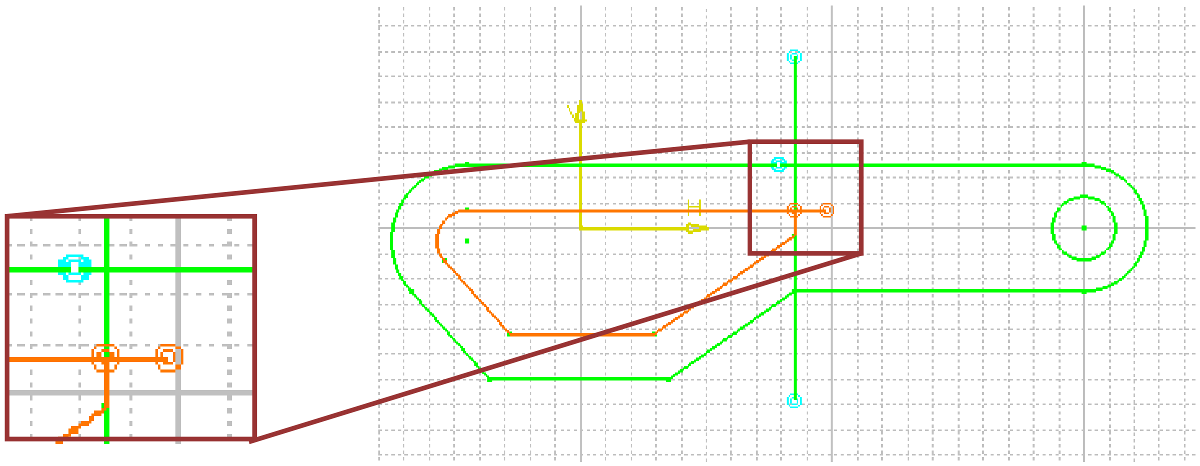
7. Remove the constraints from the display.

- 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.



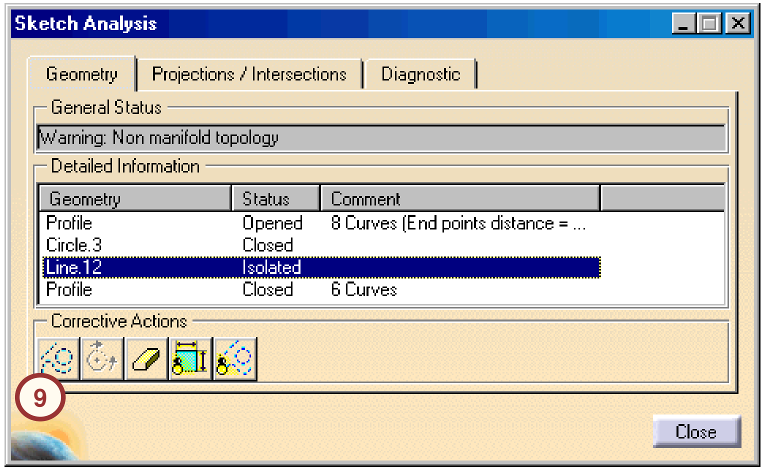


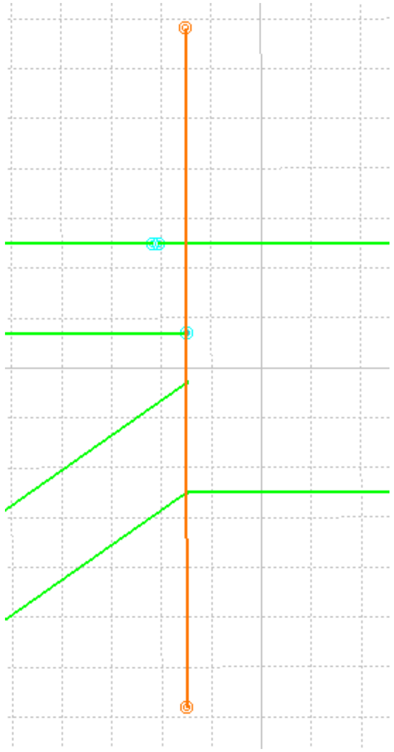
Student Notes:

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.



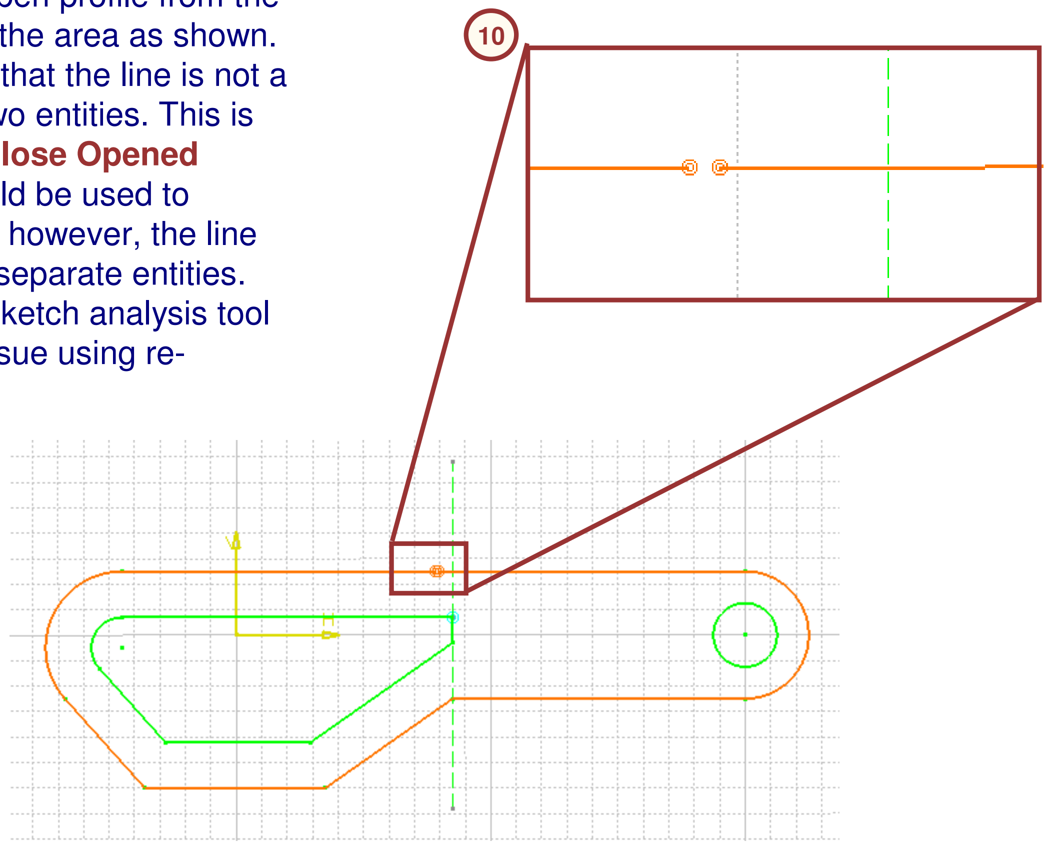


Student Notes:

Do it Yourself (5/7)

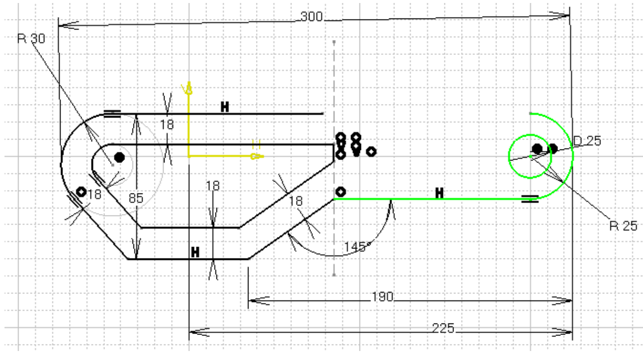
10. Review the second open profile.

- 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.

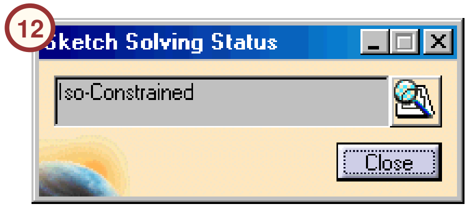


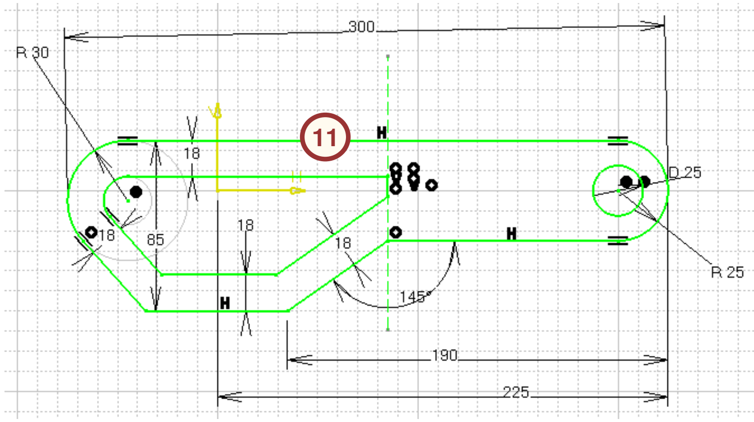
Do it Yourself (6/7)

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.





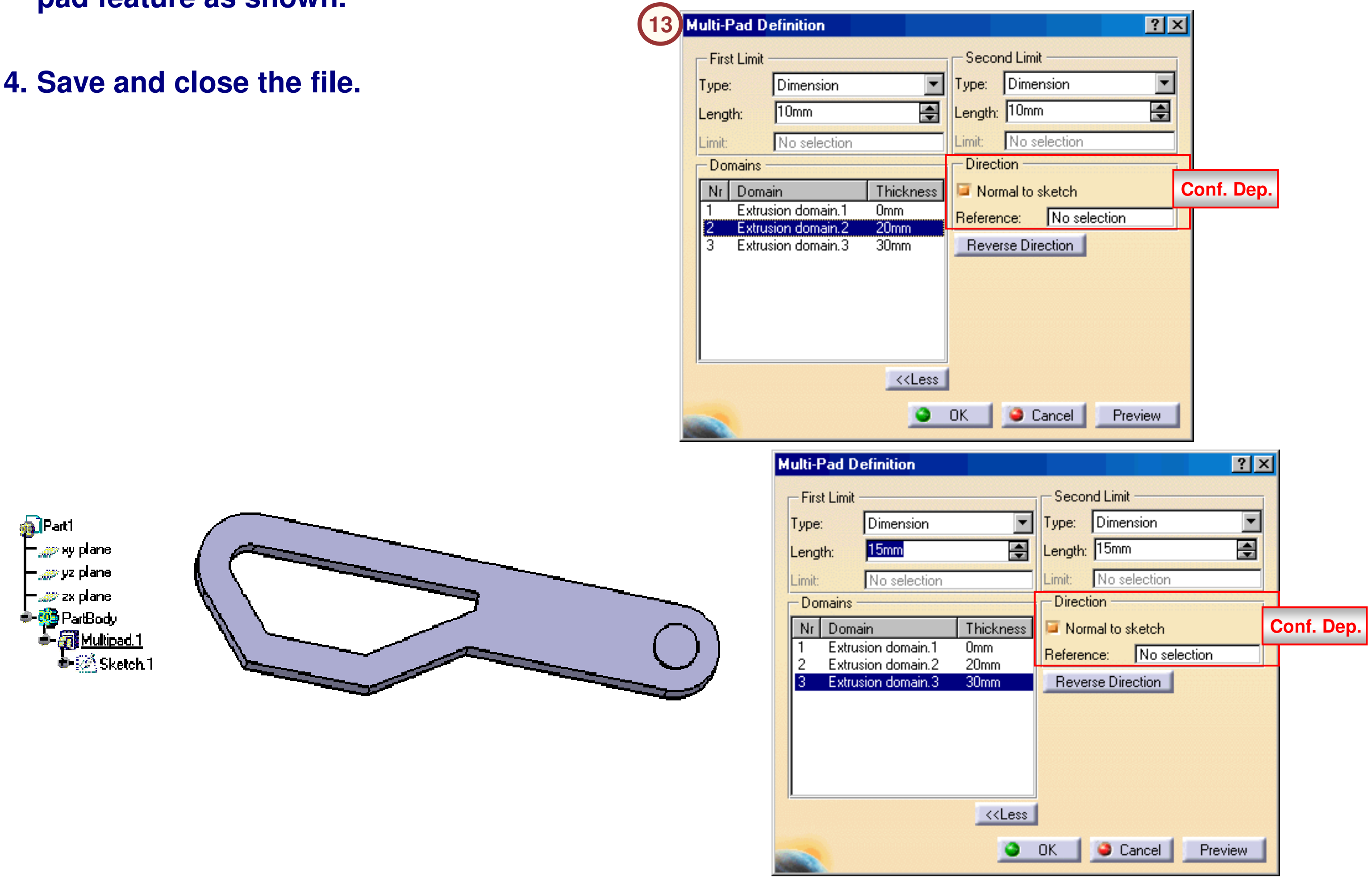
Student Notes:

Student Notes:

Do it Yourself (7/7)

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

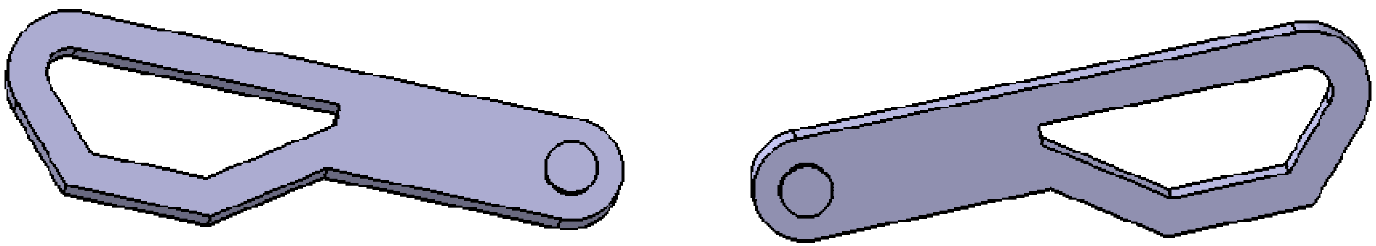
14. Save and close the file.



Student Notes:

Exercise Recap: Sketch Analysis and Pocket

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



Student Notes:

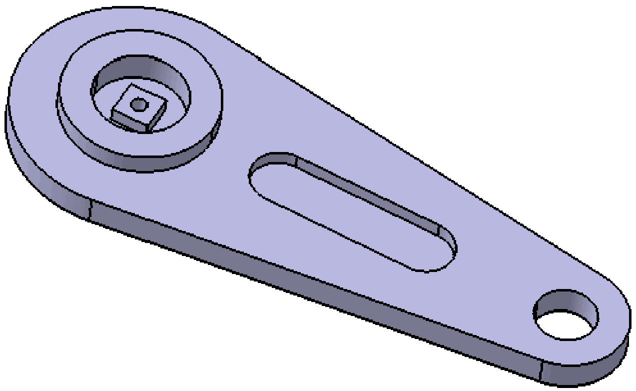
Exercise: Multiple Profile Sketch Features

Recap Exercise

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**

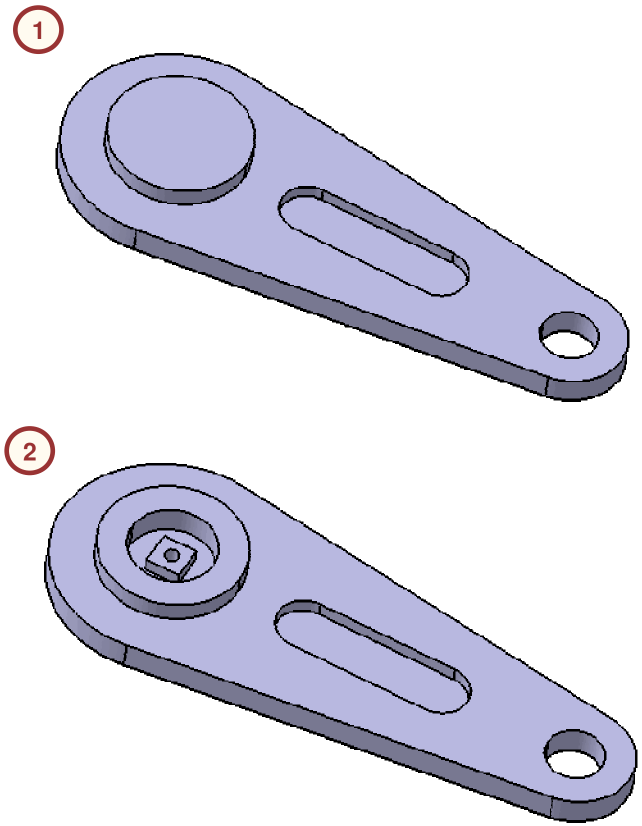


Student Notes:

Do it Yourself (1/2)

You need to create the following features:

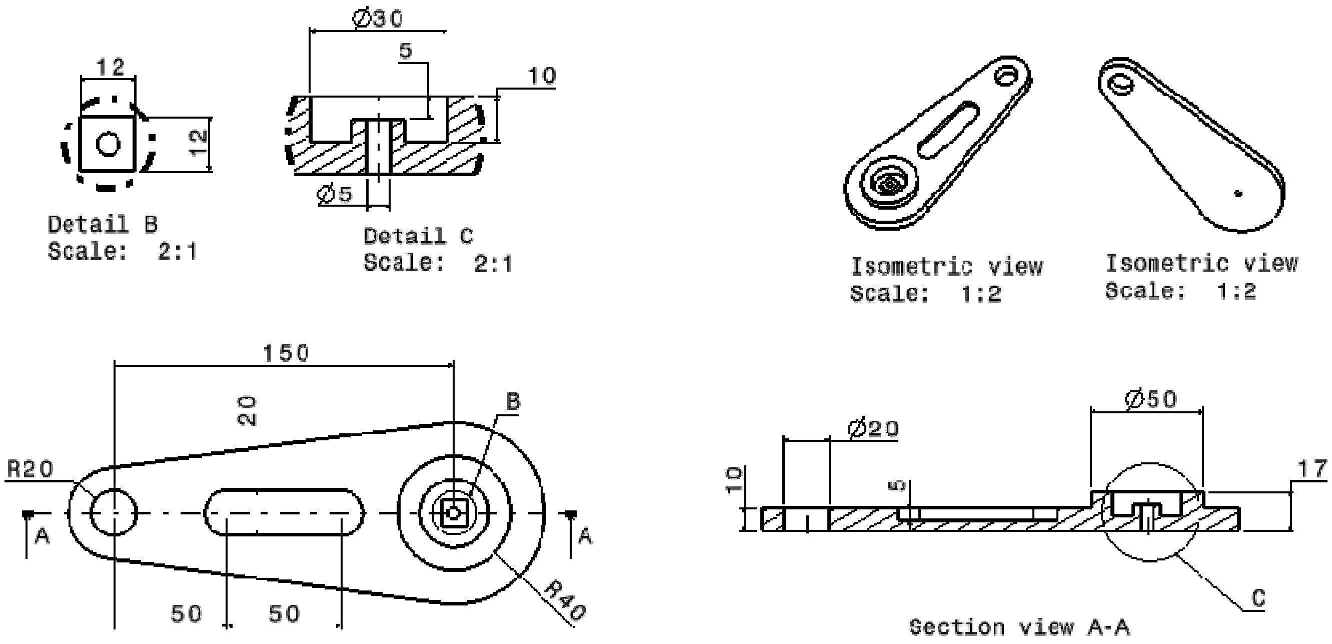
1. **Multi-pad**
2. **Multi-pocket**



Student Notes:

Do it Yourself (2/2)

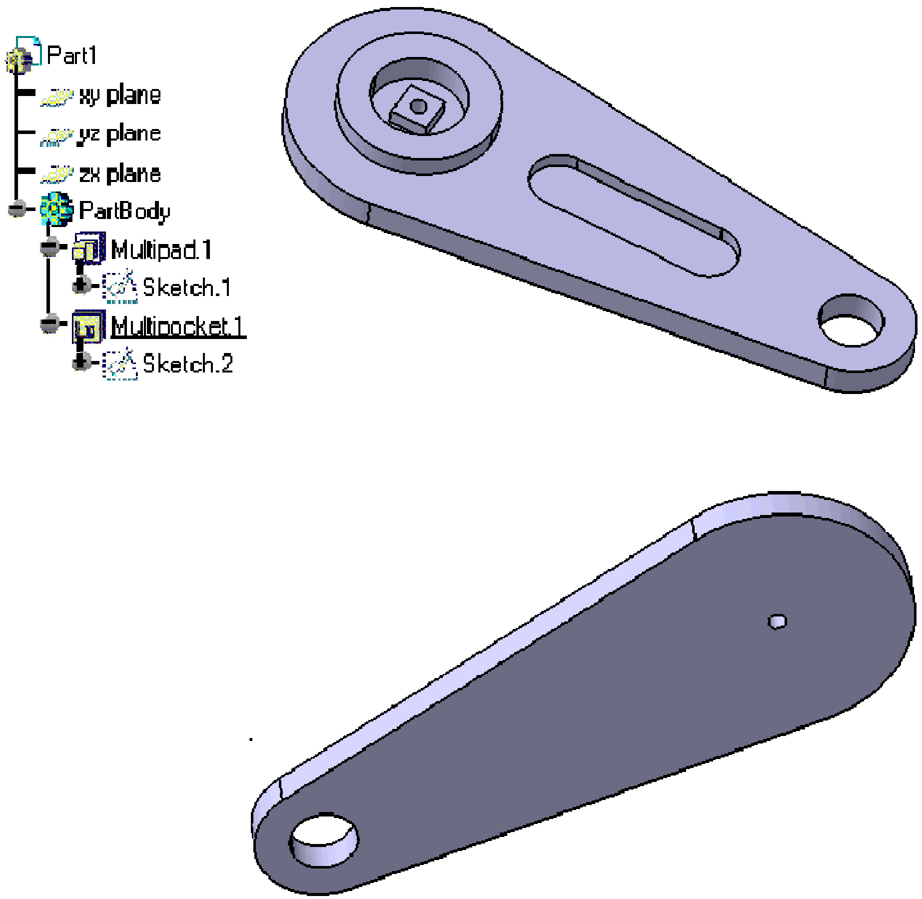
1. Create the following part.



Student Notes:

Exercise Recap: Multiple Profile Sketch Features

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



Student Notes:

Create Basic Wireframe Geometry

In this section, you will learn how to create wireframe elements (i.e., points, lines, and planes).

Use the following steps:

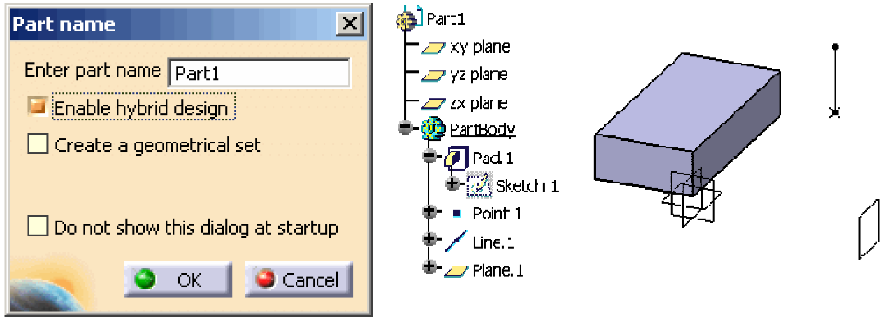
1. Create feature profiles and Axis system.
2. Create Multi-profile Sketch Feature
3. **Create basic wireframe geometry.**
4. Create shaft and groove features.
5. Shell the model.

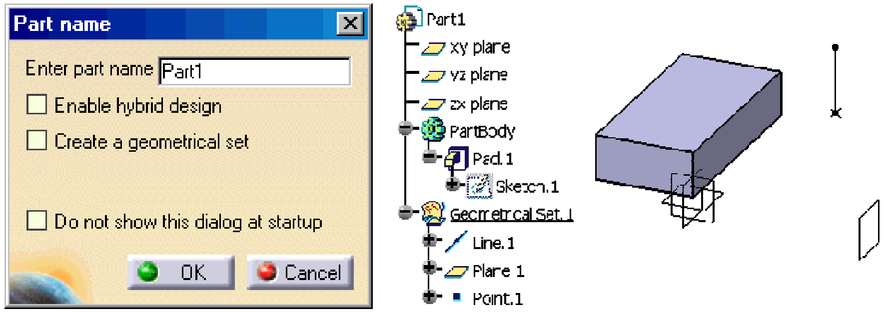
Student Notes:

Reference 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.



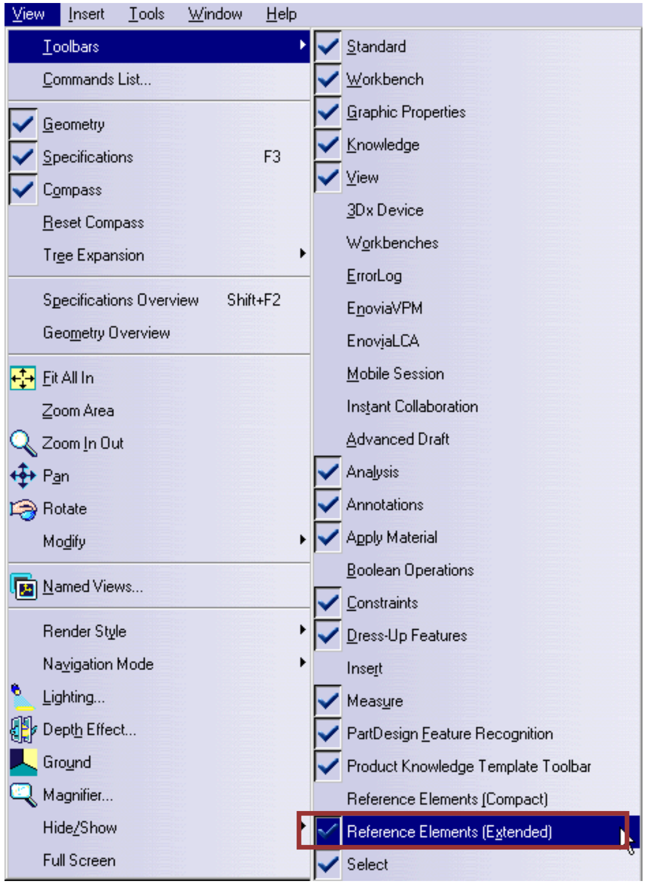


Student Notes:

Accessing the Reference Elements Toolbar

The toolbar is located at the bottom of the toolbars on the right-hand side of the screen. You may need to move other toolbars to view it.

If you cannot locate the toolbar, it may be turned off. To turn on the toolbar, click **View > Toolbars > Reference Elements (Extended)**.



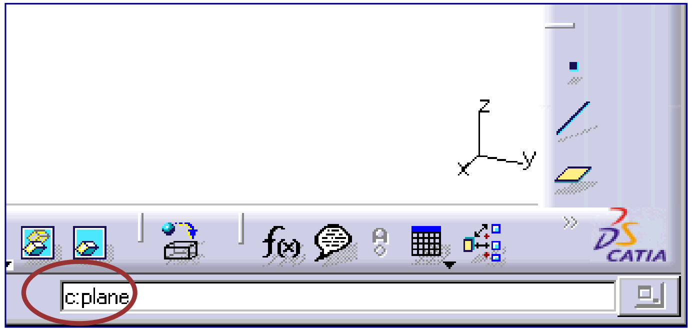
Student Notes:

Power Input Line

Instead of selecting the icons, you can use the power input line to access the 3D wireframe tools.

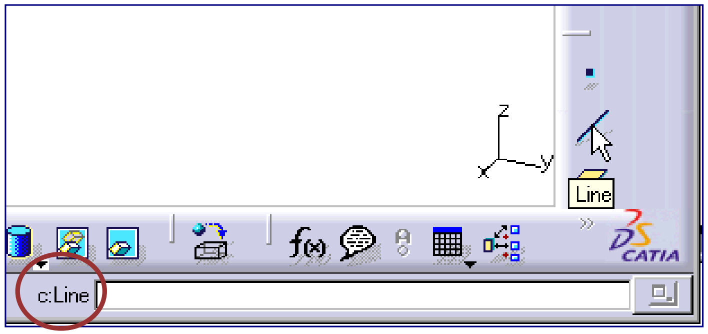
Type:

- [c:plane] to create a plane
- [c:point] to create a point
- [c:line] to create a line



The command can be used for many tools. It is a good way to launch functions when you cannot find the icon. To view the command, hover the mouse pointer over the icon.

For example, placing the pointer over the Line icon displays c:Line beside the power input line.

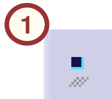


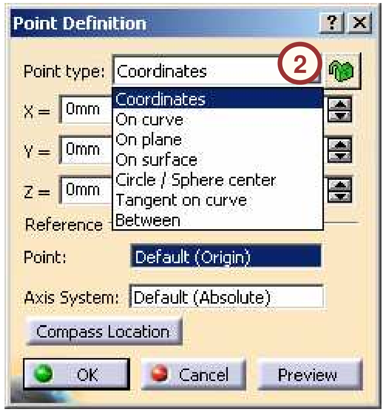
Student Notes:

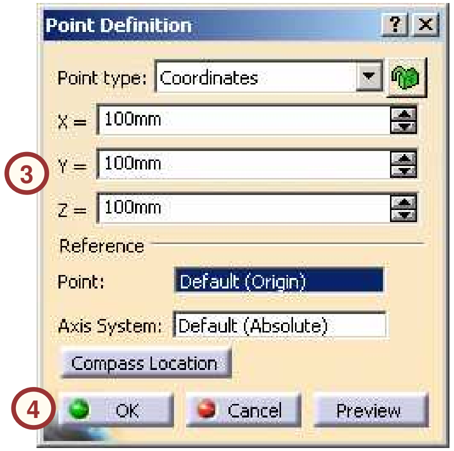
Points

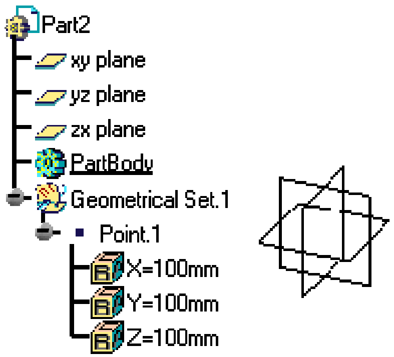
Points are used to mark a location on a model. They can be used as a basis for creating additional features. Use the following steps to create a point:

1. Click the **Point** icon.
2. Select the Point Type from the menu.
   - Many types of points can be created. The required fields vary depending on the selected type. In this example, you create a **Coordinates** point type.
3. Specify values as required. For a coordinate point, the X, Y, and Z distances from the reference point are required.
4. Click **OK** to create the point.
5. The point is added to the specification tree under the Geometrical set.









STUDENT GUIDE

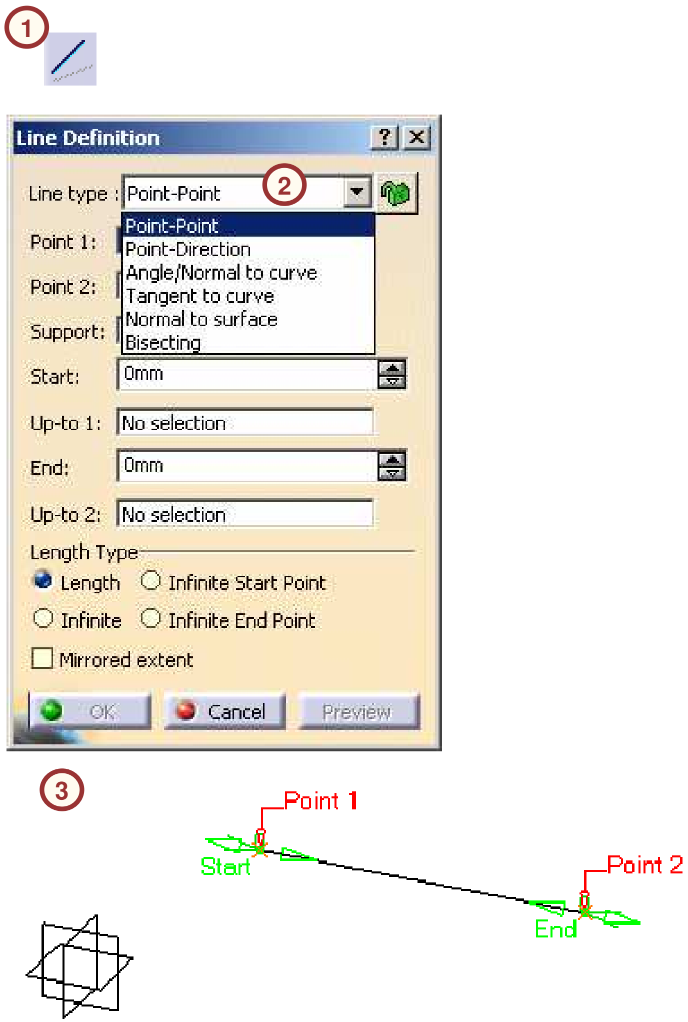
Student Notes:

Lines (1/2)

Lines are created for many purposes, they can be used to define the direction for additional geometry (solid and wireframe), or as an axis for a revolved feature.

Use the following steps to create a line:

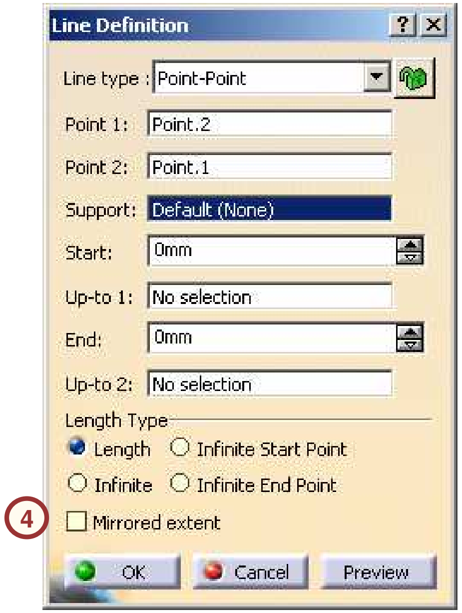
1. Click the **Line** icon.
2. Select the Line Type from the menu.
   - Many types of lines can be created. The required fields vary depending on the selected type. In this example, you create a Point-Point type line.
3. Specify values as required. For a Point-Point line, two points are required.

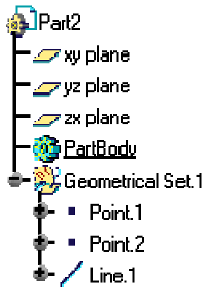


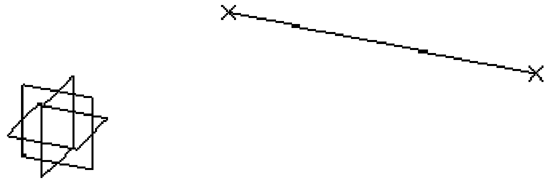
Lines (2/2)

Use the following steps to create a line (continued):

4. Click **OK** to create the line. The line is added to the specification tree under the Geometrical set.







Student Notes:

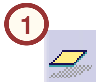
Student Notes:

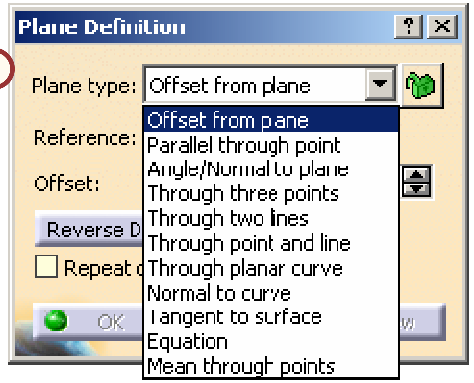
Planes (1/2)

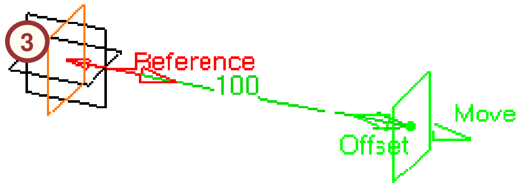
Planes are used to create a planar reference in a specific location. In the Part Design workbench, they are used as sketch supports.

Use the following steps to create a plane:

1. Select the **Plane** icon.
2. Select the Plane Type from the menu.
   - Many types of planes can be created. The required fields vary depending on the selected type. In this example, you will use the **Offset from plane** type.
3. Specify the values as required. For an Offset from plane type, a planar surface or an existing reference plane is required.



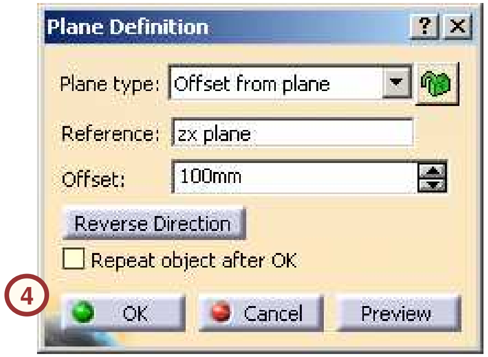


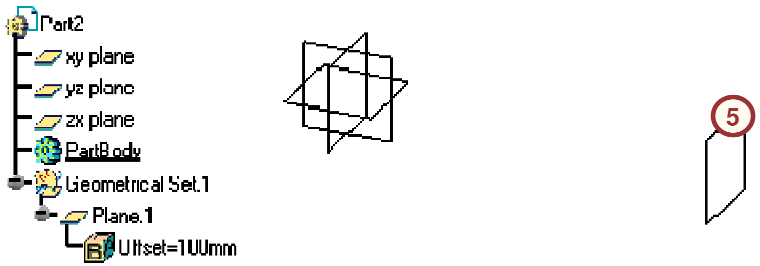


Planes (2/2)

Use the following steps to create a plane (continued…):

4. Click **OK** to create the plane.
5. The plane is added to the specification tree under the Geometrical Set.





Student Notes:

Student Notes:

Recommendations for Reference Elements

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

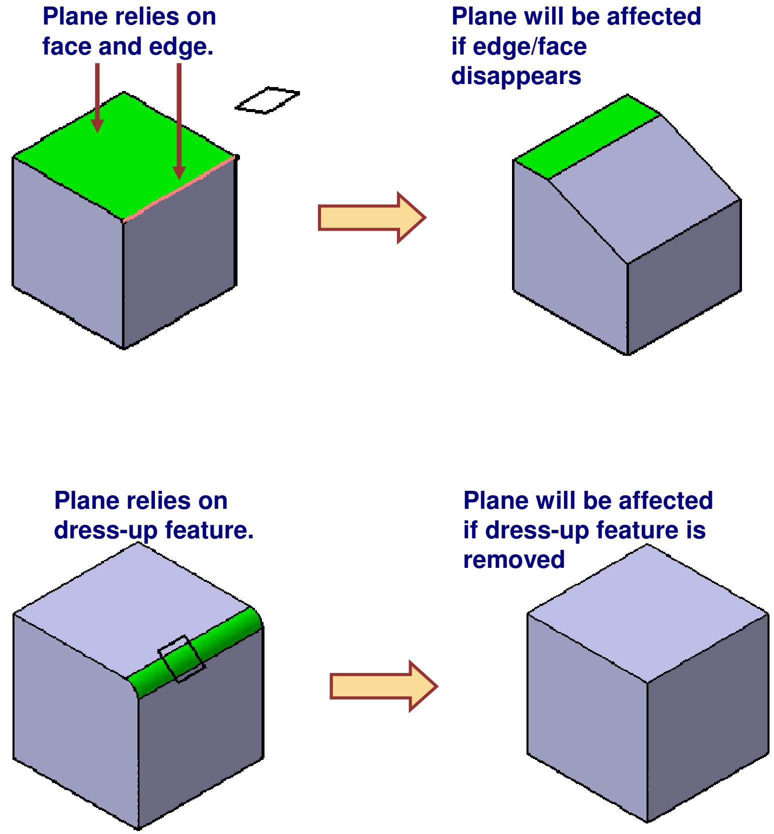
Student Notes:

No Reference Elements on Solid Face (1/3)

It is recommended not to create reference elements that rely on solid faces, edges or dress-up features.

During the design and development of a part:

a. The solid face or edge is subject to change and can disappear.
b. A planar face can later become a non-planar face.
c. Dress-up features can be removed for a downstream manufacturing processes.

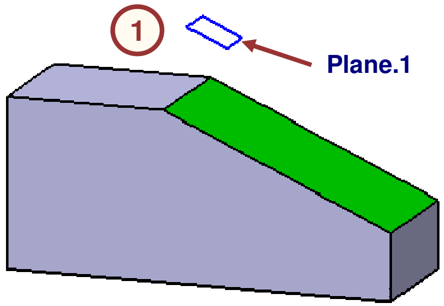


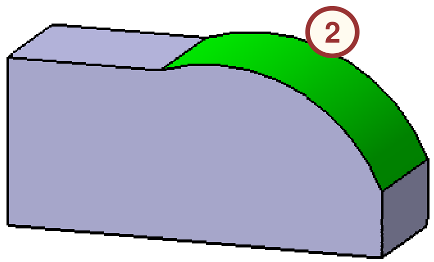
Student Notes:

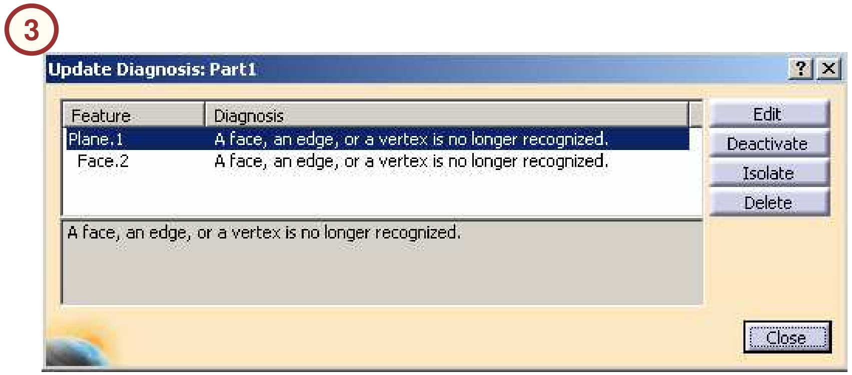
No Reference Elements on Solid Face (2/3)

In the example shown, reference element Plane.1 is offset from the highlighted solid face.

1. Plane.1 relies on solid face.
2. During further design and development of the part, the profile is changed such that the highlighted solid face becomes non-planar.
3. The reference element, Plane.1 will be affected and the design becomes unstable.





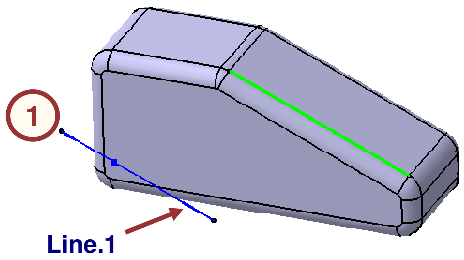


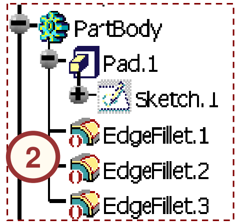
Student Notes:

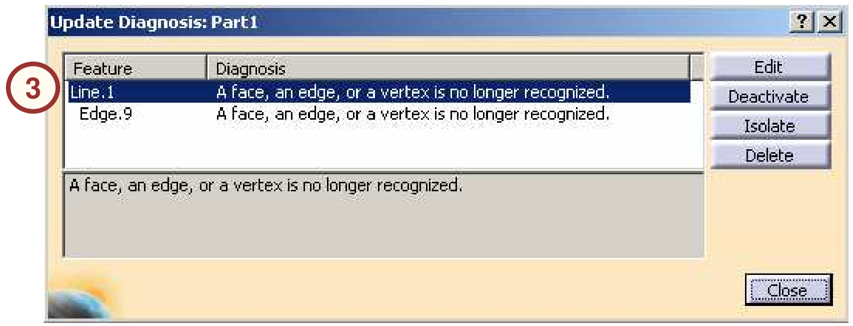
No Reference Elements on Solid Face (3/3)

In the example shown, reference element Line.1 is parallel to edge of fillet.

1. Line.1 relies on a dress-up feature.
2. Because of a downstream manufacturing process, the dress-up features are deactivated.
3. The reference element, Line.1 will be affected and the design becomes unstable.







Student Notes:

Create Shaft and Groove Features

In this section, you will learn how to create revolved features that add and remove material.

Use the following steps:

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

Student Notes:

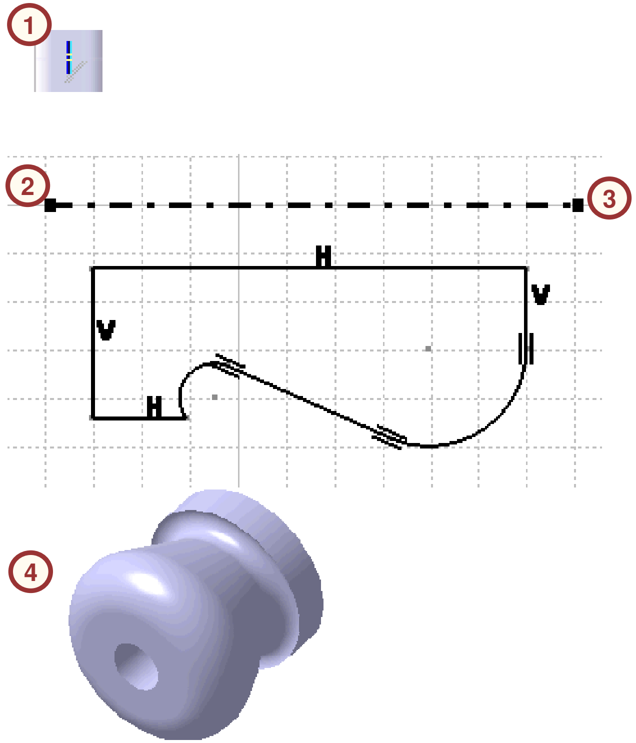
Creating an Axis

An axis can be used as a reference to create revolved features, such as shafts and grooves (discussed later in this lesson). The sketched profile is revolved about it.

An axis can also be used to create symmetrical sketched elements inside the Sketcher workbench.

Use the following steps to create an axis:

1. Click the **Axis** icon.
2. Click to create the start point for the axis.
3. Click again to create the endpoint.
4. Using the shaft command on the profile sketch, CATIA produces a shaft using the defined axis.



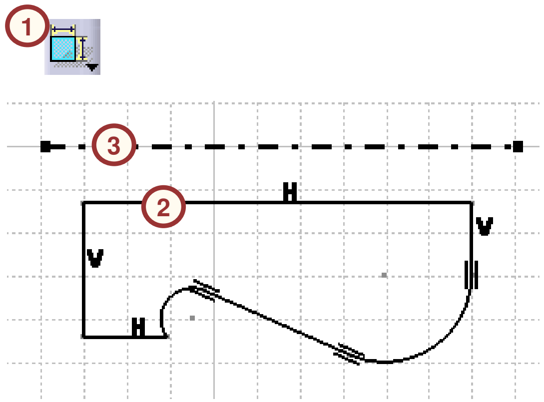
Student Notes:

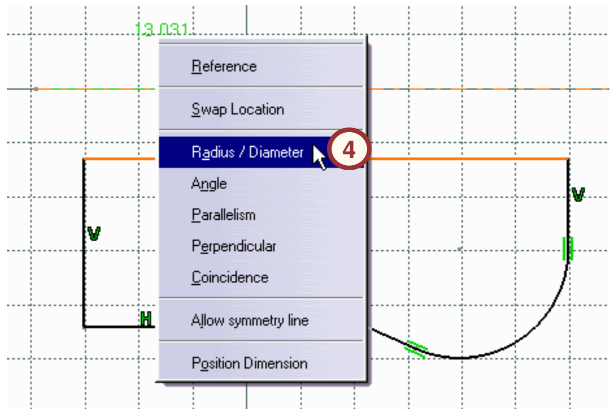
Dimensioning to an Axis

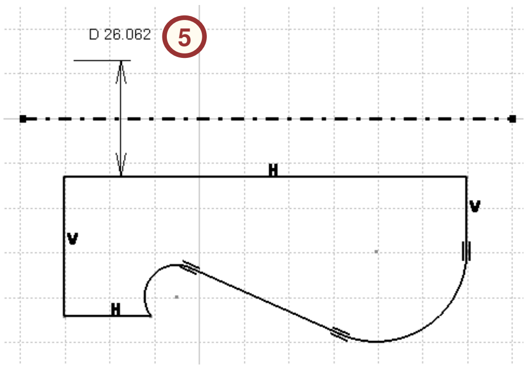
You can define diameter and radius dimensions to an axis. This is useful while creating the profile sketches for revolved features (discussed later in this lesson).

Use the following steps to create a Radius/Diameter dimension to an axis:

1. Click the **Constraint** icon.
2. Select the sketched element.
3. Select the axis.
4. Right-click and select **Radius/Diameter**.
5. Click to place the dimension.







Student Notes:

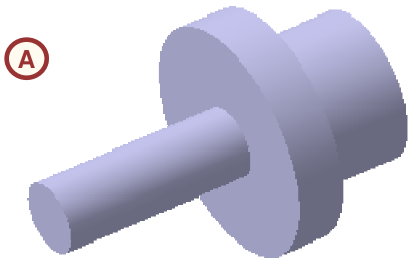
Revolved Features (1/2)

A revolved feature is created by revolving a 2D profile around an axis of revolution.

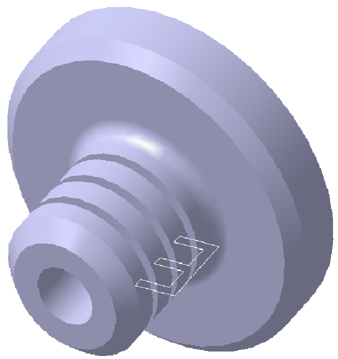
In the Part Design workbench, you can create two types of revolved features:

A. A shaft, which adds material.
B. A groove, which removes material.

A



B



Student Notes:

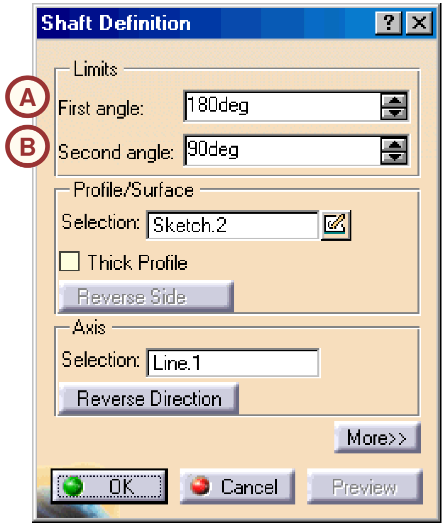
Revolved Features (2/2)

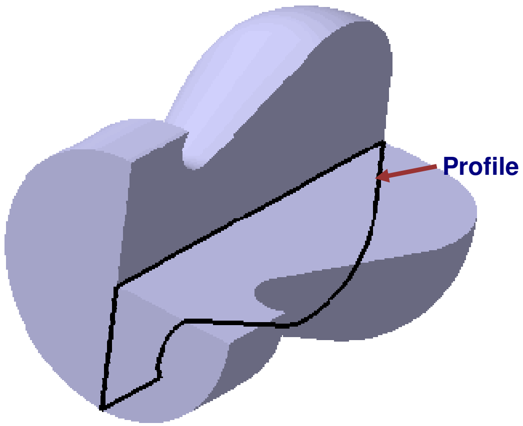
Revolved features can be revolved between 0° and 360°.

You can define the following limits:

A. The *First angle* limit defines the revolution angle of the profile around the axis, starting from the profile position and orientated in the clockwise direction.

B. The *Second angle* limit defines the revolution angle of the profile around the axis, starting from the profile position and oriented in the counterclockwise direction.



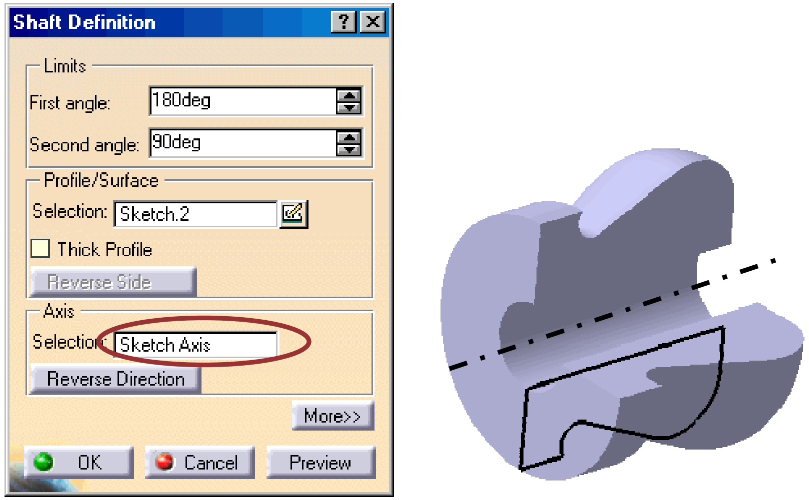


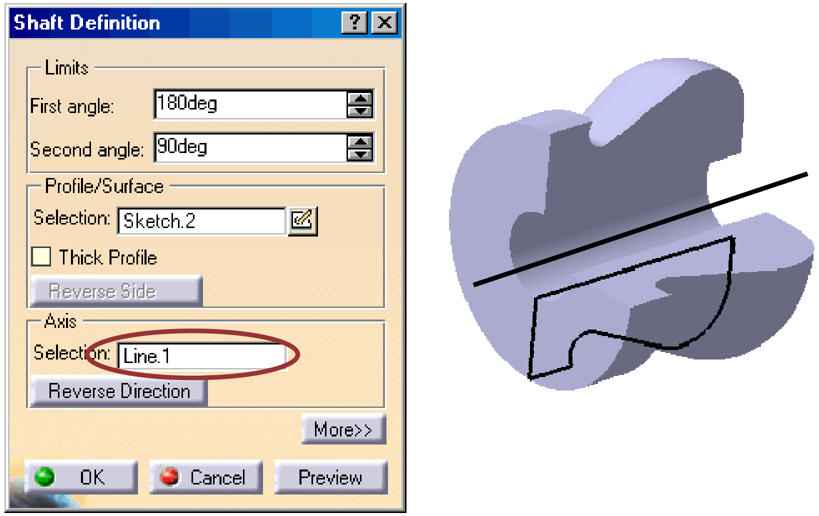
Student Notes:

Axis of Revolution

The axis of revolution for a revolved feature can be defined by two methods. The axis can be created inside the actual sketch containing the profile, using the **Axis** tool. If the axis is created inside the sketch, it will be detected automatically while defining the shaft or groove.

If you did not create an axis in the sketch, or want to use a different axis other than the one defined in the sketch, you can define it from the Shaft/Groove definition window in the Axis selection field. Any linear element in the model (e.g., an edge of existing geometry, a 3D wireframe line, a line created in a sketch) can be used.





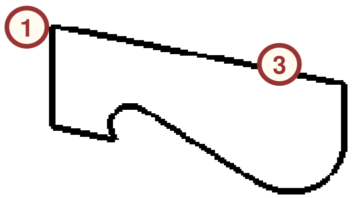
Student Notes:

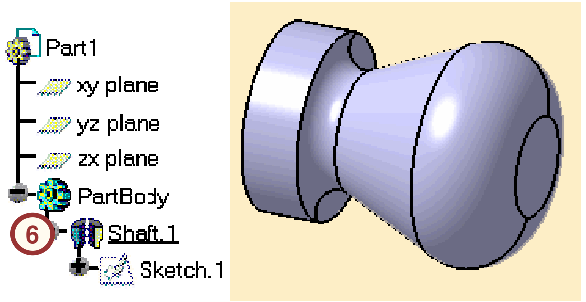
Shafts

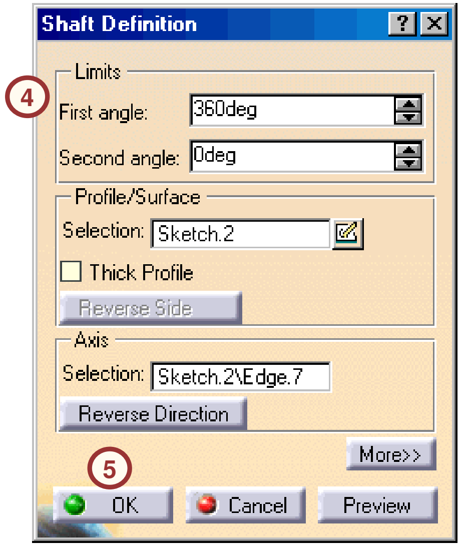
A shaft is a revolved sketched-based feature that adds material to the model.

Use the following steps to create a shaft:

1. Select the profile.
2. Click the **Shaft** icon.
3. If no axis is created inside the sketch, select an axis of revolution.
4. Define angle limits.
5. Click **OK** to complete the feature.
6. The shaft feature is added to the model.







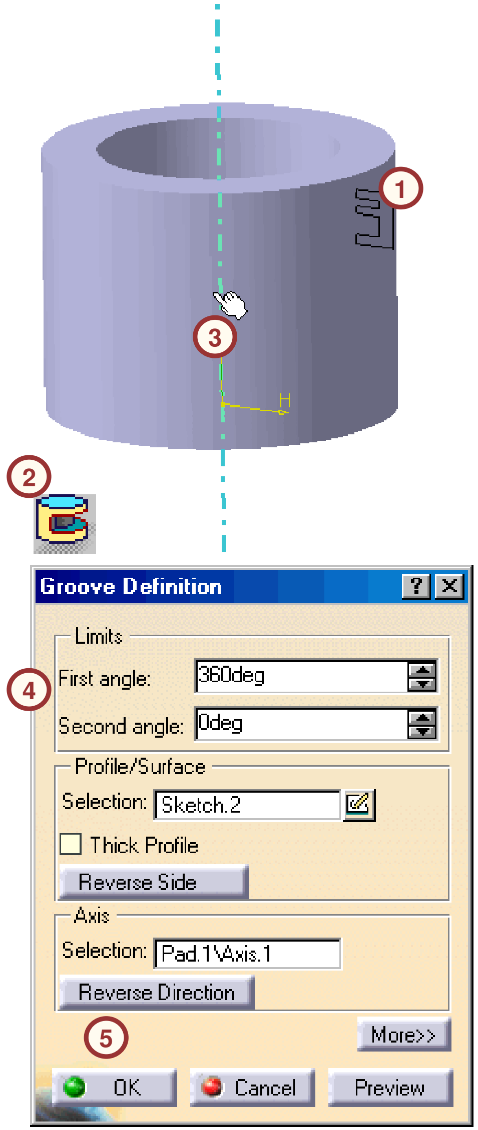
Student Notes:

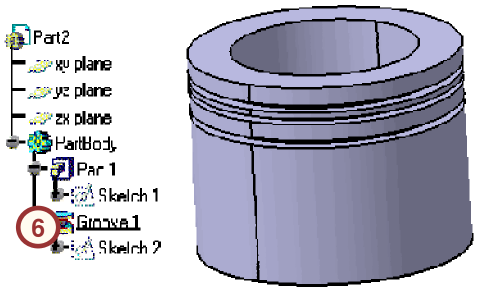
Creating Grooves

Grooves are revolved features that remove material from existing features by rotating a 2D profile around an axis. The axis and the profile can be created in the same sketch or the axis can reside outside of the sketch.

Use the following steps to create a Groove feature:

1. Select the Profile.
2. Click the **Groove** icon.
3. If no axis is created inside the profile sketch, select an axis of revolution. In this example, the implicit axis of the cylindrical feature is selected.
4. Define angle limits.
5. Click **OK** to complete the feature.
6. The Groove feature is added to the model.

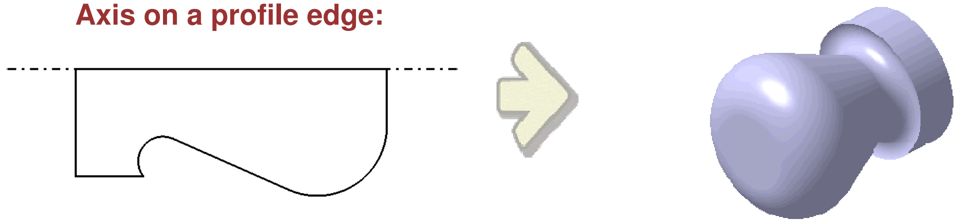




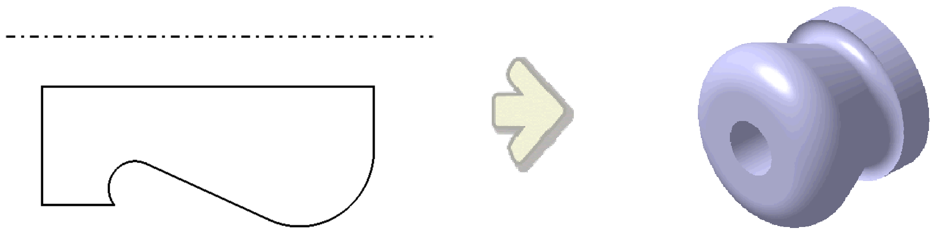
Restrictions for Revolved Features (1/2)

Not every sketch can be used to create a shaft base feature. The examples shown display various sketch solutions:

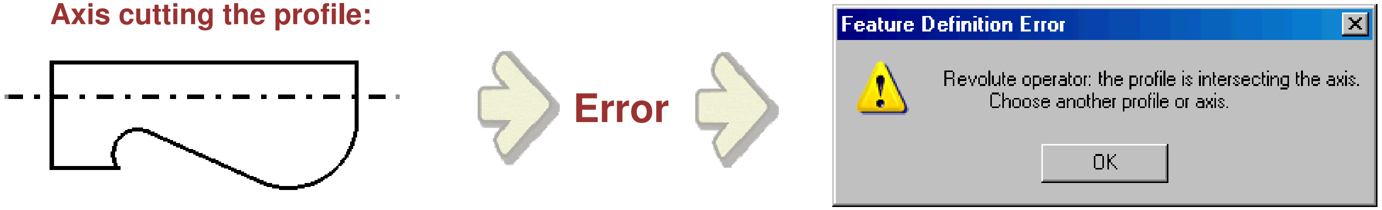
Axis on a profile edge:



Axis outside the profile:



Axis cutting the profile:



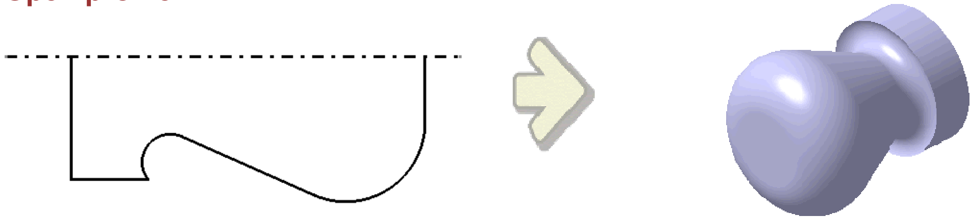
Student Notes:

Student Notes:

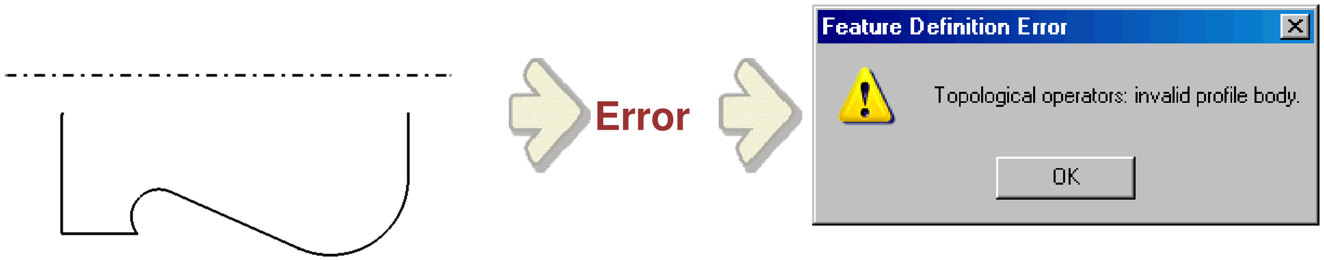
Restrictions for Revolved Features (2/2)

Not every sketch can be used to create a shaft base feature. Below are some examples showing various sketch solutions (continued):

Open profile:



Open profile and axis outside the profile:



Student Notes:

Exercise: Shaft and Groove

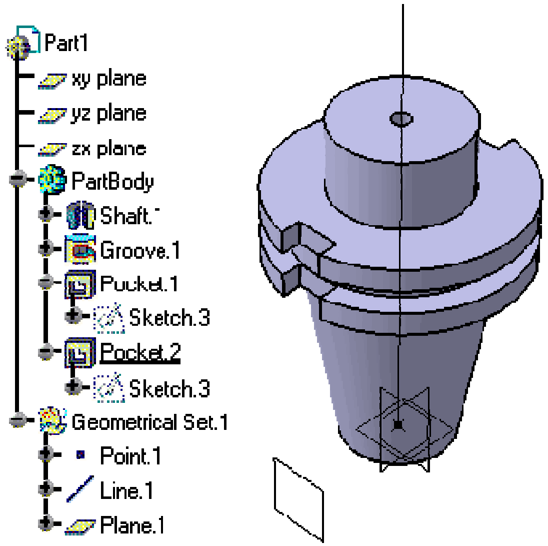
Recap Exercise

15 min

In this exercise you will create a new tool holder part by creating a revolved feature using a point, line, and sketch. A reference plane will then be used to create an additional feature.

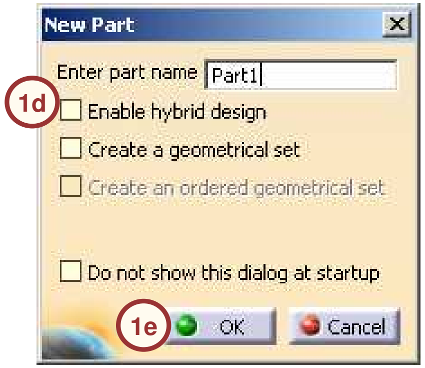
By the end of this exercise you will be able to:

- **Create reference geometry**
- **Create a shaft feature**
- **Create a groove feature**
- **Use reference geometry to create new features**

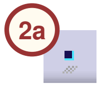


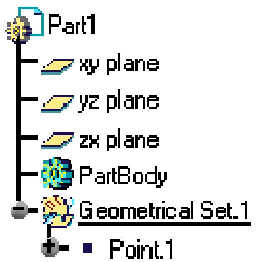
Do it Yourself (1/10)

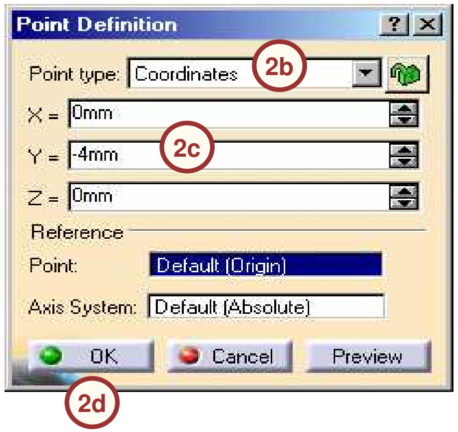
1. **Create a new part.**
   - Create a new part file.
     a. Click **File > New.**
     b. Select **Part** from the list of document types.
     c. Click **OK**.
     d. Leave the default name, clear the **Enable hybrid design** option.
     e. Click **OK**.



2. **Create a point.**
   - Create a point by specifying coordinates. This point will be used as a reference to create a line The line is then used as the axis of revolution for a shaft feature.
     a. Click the **Point** icon. If you can't find the icon, specify [c:point] in the power input line.
     b. Change the point type to **Coordinates**.
     c. Type **[–4]** for the Y value and leave all other inputs as default.
     d. Click **OK** to create the point.







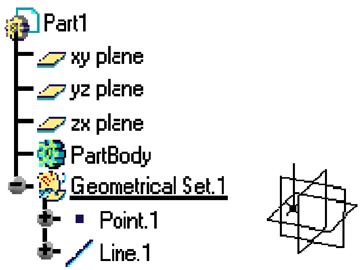
Student Notes:

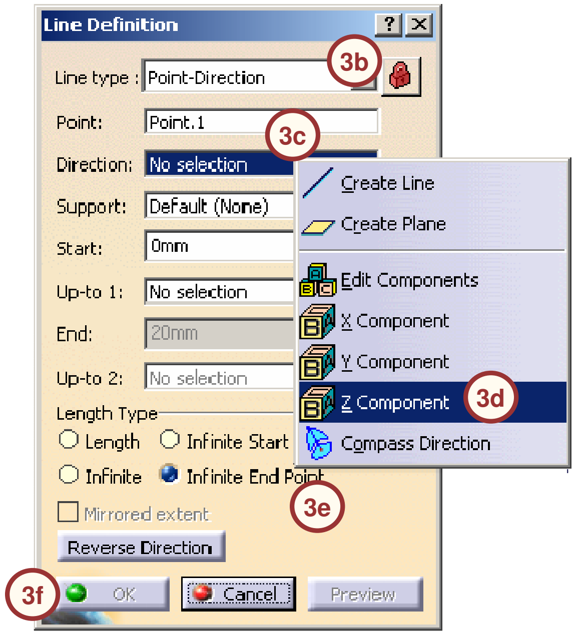
Student Notes:

Do it Yourself (2/10)

3. Create a line.

- Create a line in the Z axis direction using the created point.
  a. Click the **Line** icon. If you cannot find the icon, specify [c:line] in the power input line.
  b. Change the Line type to **Point-Direction**.
  c. Select the **Point.1** that was created previously.
  d. Right-click on Direction and click **Z Axis**.
  e. Select the **Infinite End Point** option for the Length Type.
  f. Click **OK** to complete the line.



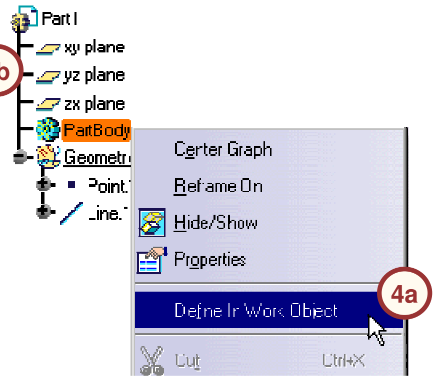


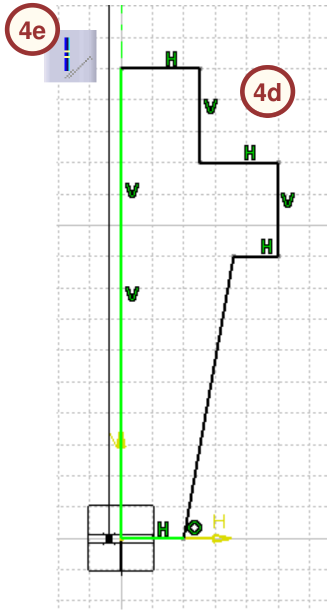
Student Notes:

Do it Yourself (3/10)

4. Create a sketch.

- Create a sketch that will represent the profile for a tool holder.
  a. Right-click on the PartBody and select **Define in Work Object**. This ensures that features that are created are added to the PartBody and not the Geometrical Set.
  b. Click the **Sketcher** icon.
  c. Select the YZ plane as the sketch support.
  d. Use the **Profile** icon in sketcher to create the lines.
  e. Create an axis vertically along the V axis.





Student Notes:

Do it Yourself (4/10)

4. Create a sketch (continued).

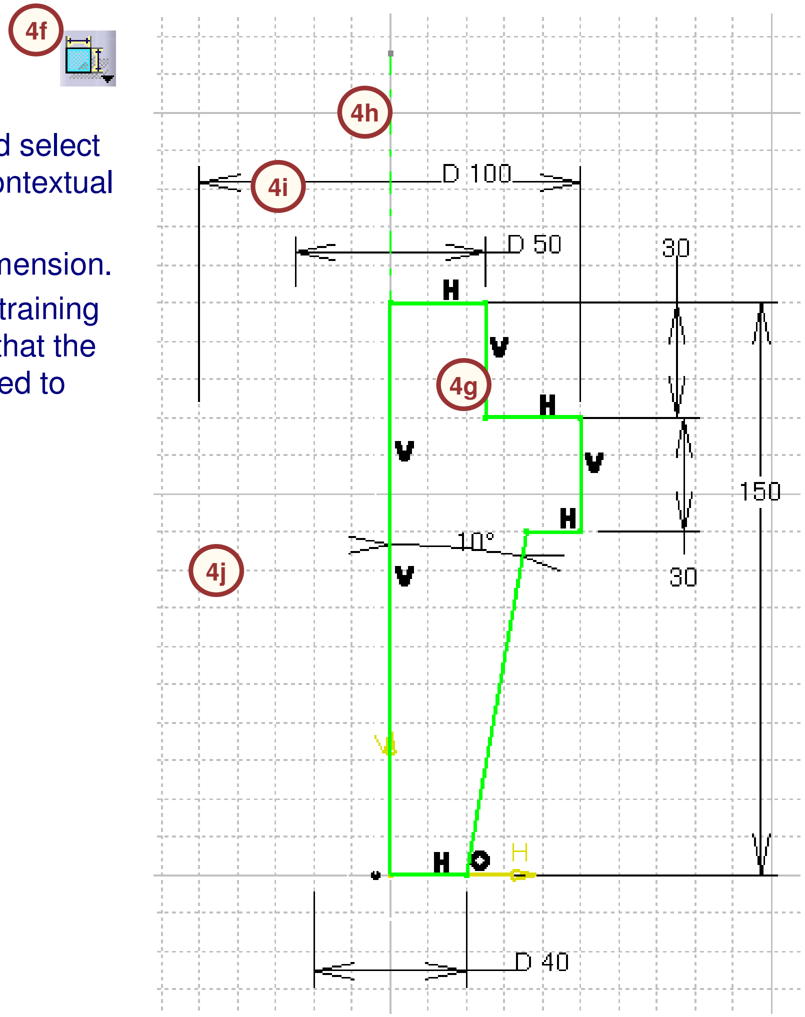
f. Click the **Constraint** icon.

g. Select the top vertical line.

h. Select the axis. Right-click and select **Radius/ Diameter** from the contextual menu.

i. Click to place the diameter dimension.

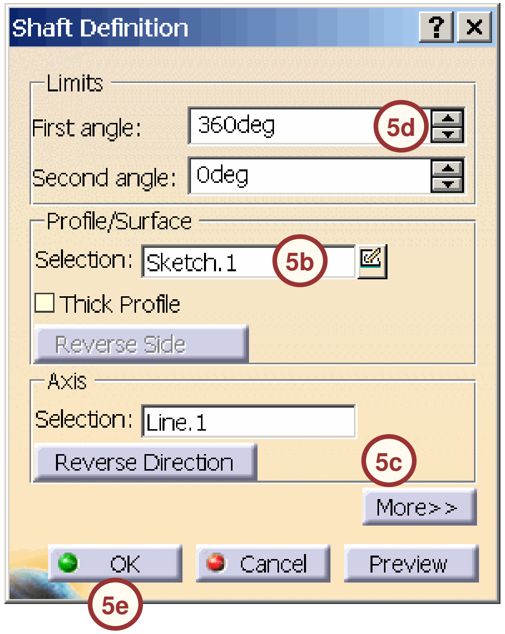
j. Finish dimensioning and constraining the sketch as shown. Ensure that the left edge of the sketch is aligned to the V axis.

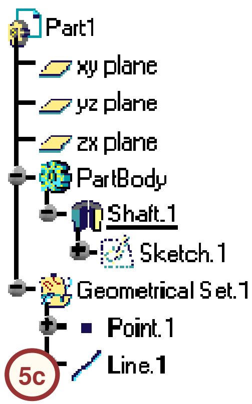


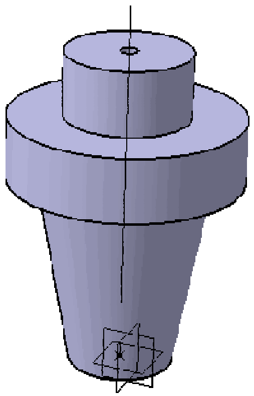
Student Notes:

Do it Yourself (5/10)

5. **Create a shaft feature.**
   - Create a revolved feature using the sketch and line created previously.
     a. Click the **Shaft** icon.
     b. Select Sketch.1 previously created as the profile selection.
     c. Click on the Axis Selection field and select Line.1.
     d. Make sure the first angle is [360deg] and the second angle is [0deg].
     e. Click **OK** to complete the shaft.



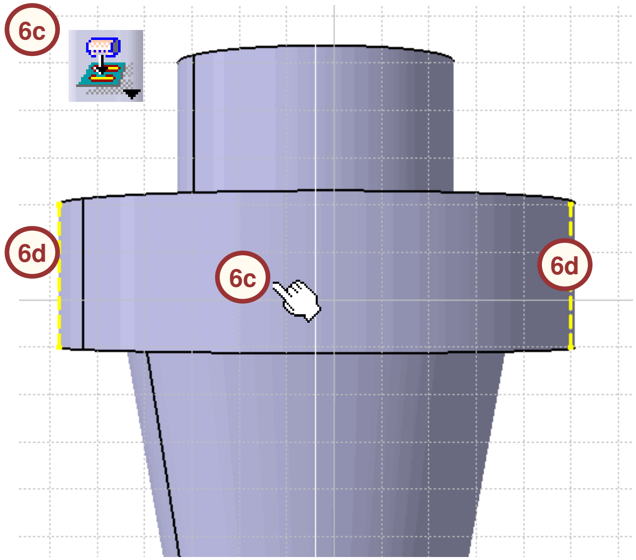




Student Notes:

Do it Yourself (6/10)

6. **Create a groove feature.**
   - Create a sketch that will be used as the profile for a groove on the tool holder.
     a. Click the **Sketcher** icon.
     b. Select YZ plane as the sketch support.
     c. Use the **Project Silhouette Edges** tool to project the side surface of the shaft.
     d. Select both projected edges and convert them to construction entities.



Student Notes:

Do it Yourself (7/10)

6. Create a groove feature (continued).

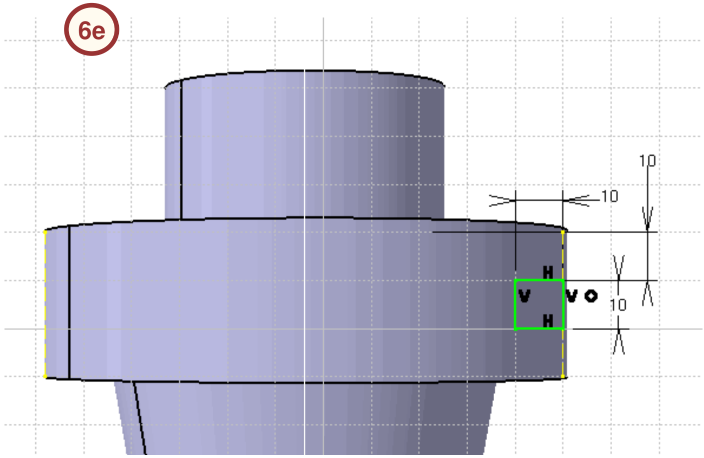
e. Sketch and dimension a square as shown.

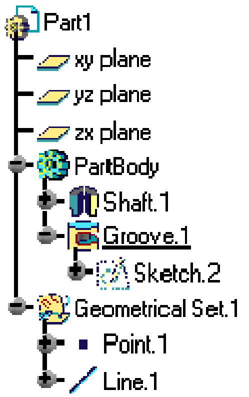
f. Exit the sketch.

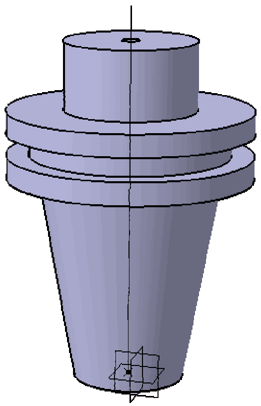
g. Click the **Groove** icon.

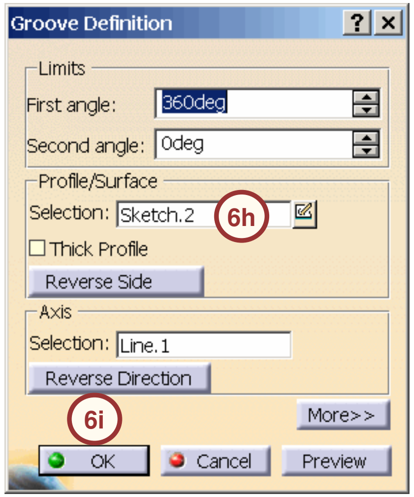
h. Select Sketch.2 as the profile and Line.1 as the axis.

i. Click **OK** to complete the groove.



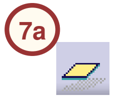


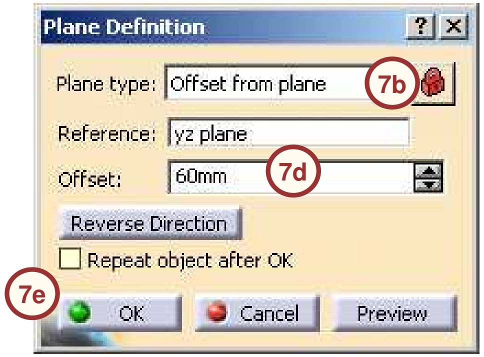




Student Notes:

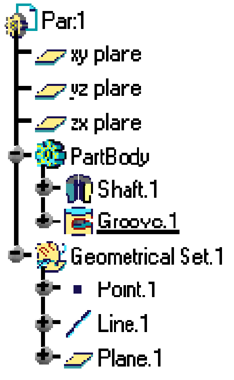
Do it Yourself (8/10)

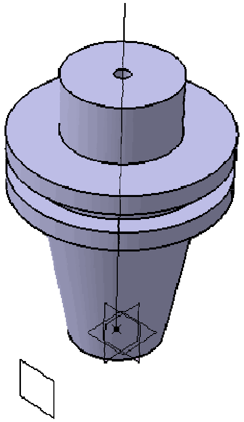




7. Create a reference plane.

- Create an offset reference plane that will be used as the sketch support for a new sketch.
  a. Click the **Plane** icon. If you cannot find the icon, type [c:plane] in the power input line.
  b. Select **Offset from plane** as the Plane type.
  c. Select the YZ plane as the reference plane.
  d. Specify [60mm] as the offset.
  e. Click **OK** to complete the plane.



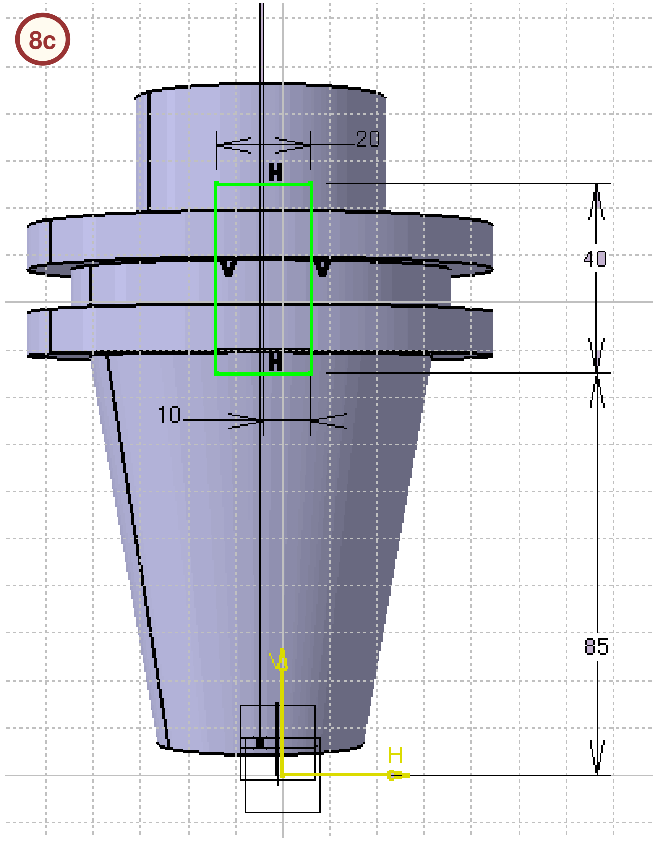


Student Notes:

Do it Yourself (9/10)

8. Create two pocket features.

- Create two pocket features using Plane.1 as the sketch support.
  a. Click the **Sketcher** icon.
  b. Select the Plane.1 as the sketch support.
  c. Sketch and constrain a rectangle as shown.
  d. Exit the Sketcher workbench.

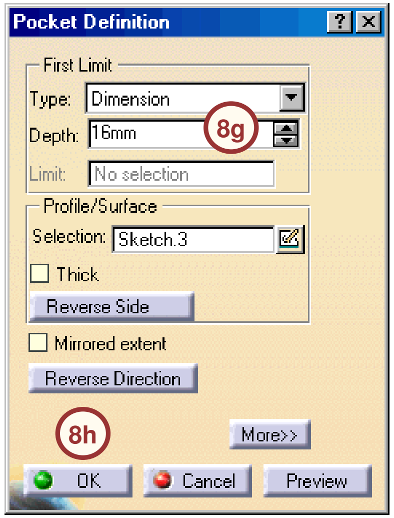


Student Notes:

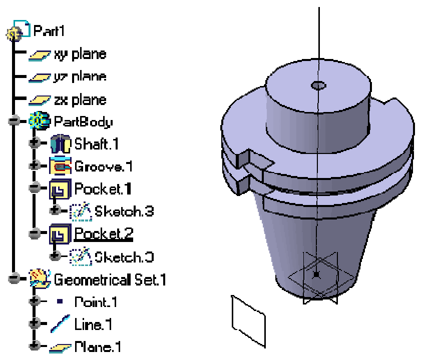
Do it Yourself (10/10)

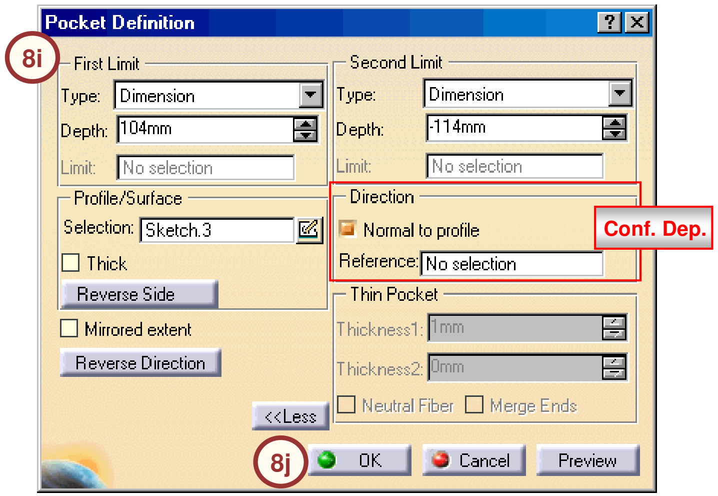
8. Create two pocket features (Continued).

e. Click the **Pocket** icon.
f. Select the sketch just created as the profile.
g. Specify [16mm] as the pocket depth.
h. Click **OK** to complete the pocket.
i. Create another pocket with the same Sketch. Specify the First Limit Depth as [104mm] and the Second Limit as [-114mm] for the second pocket.
j. Click **OK** to complete the second pocket.



9. Save and close the part.

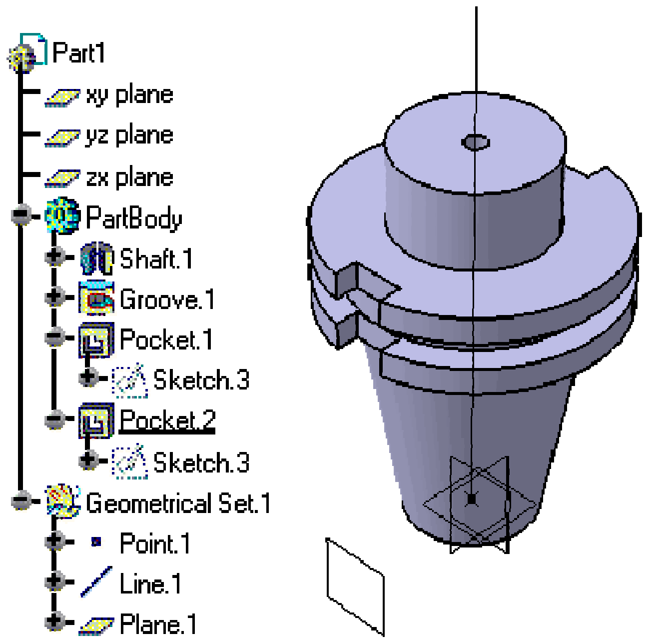




Student Notes:

Exercise Recap: Shaft and Groove

- ✓ Create a reference point and line
- ✓ Create a shaft feature
- ✓ Create a groove feature
- ✓ Create a reference plane
- ✓ Create features on reference plane



Student Notes:

Exercise: Shaft and Groove

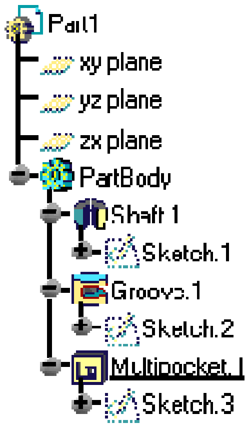
Recap Exercise

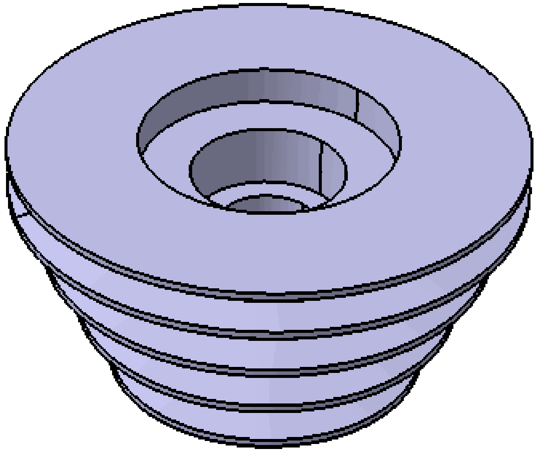
15 min

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





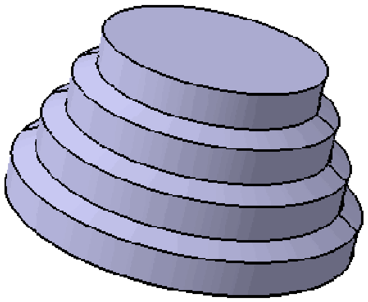
Do it Yourself (1/3)

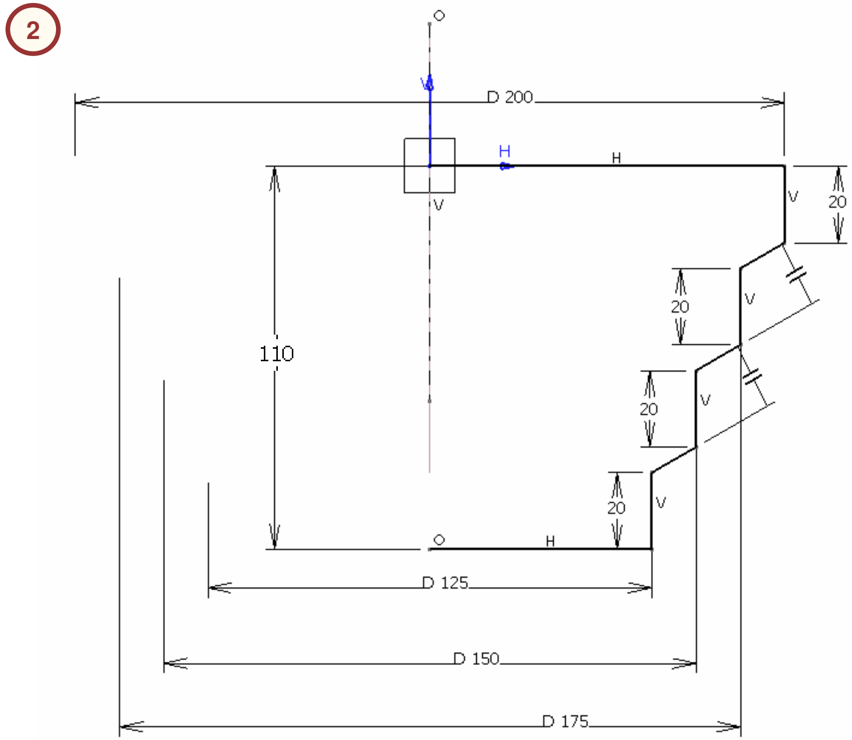
1. Create a new part file.

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

2. Create a shaft feature.

- Use the dimension shown to construct a shaft feature.





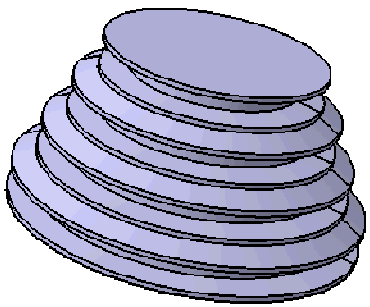
Student Notes:

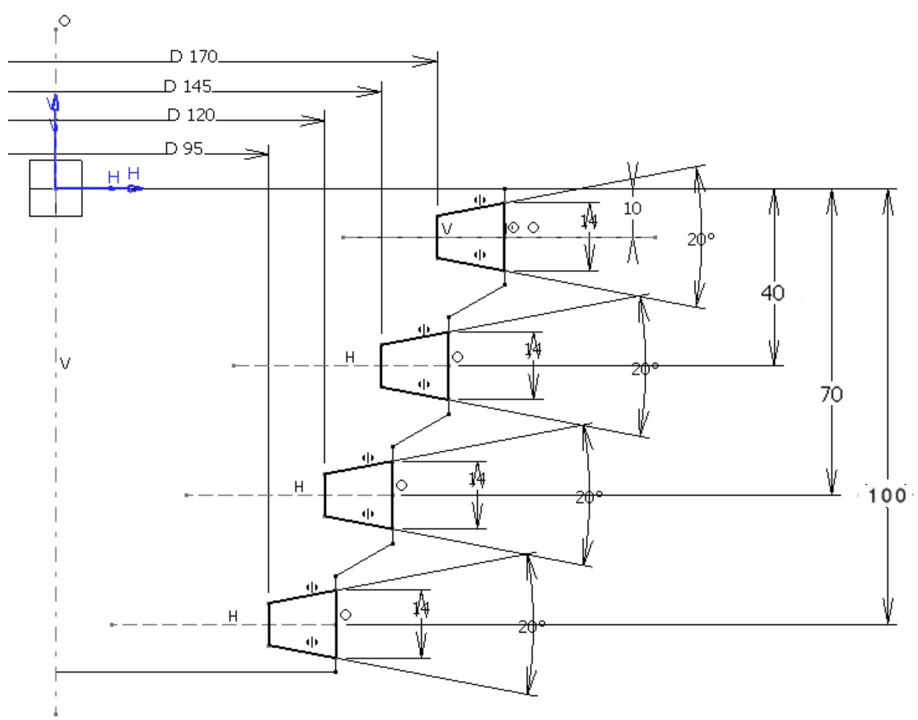
Student Notes:

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.





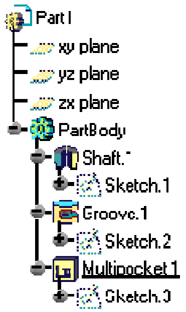
Student Notes:

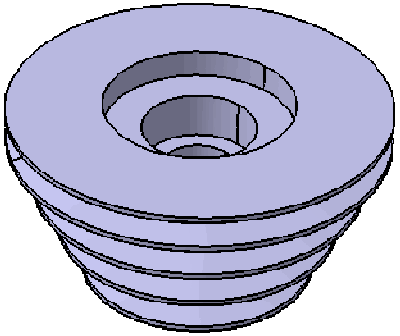
Do it Yourself (3/3)

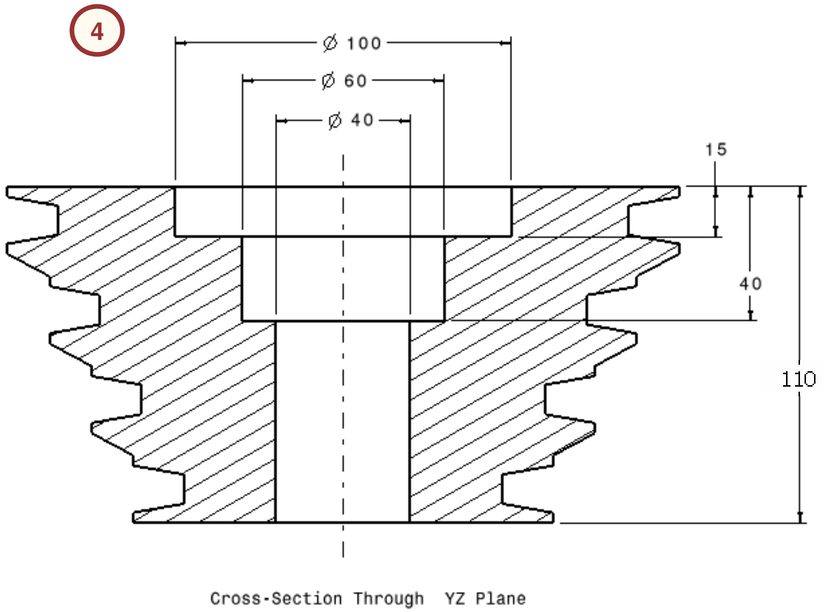
4. Create a multi-pocket feature.

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

5. Save and close the model.



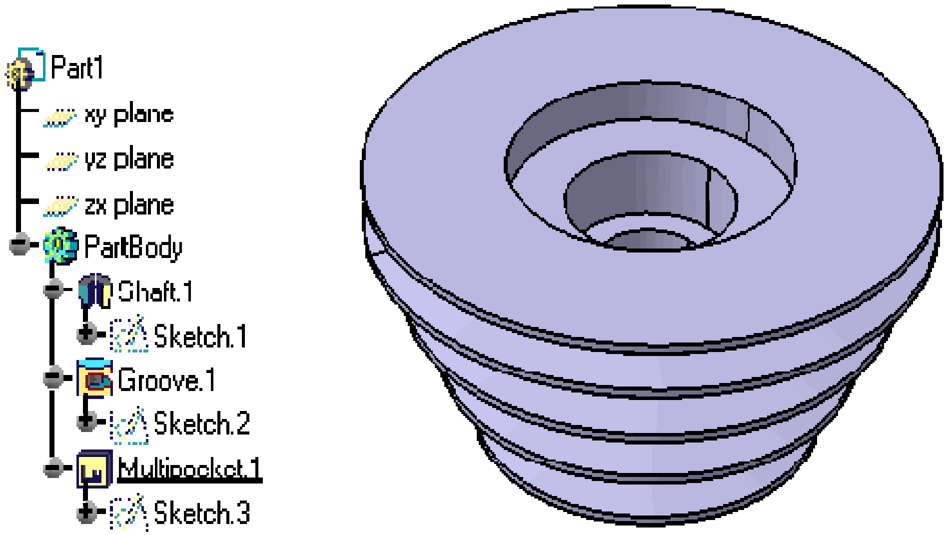




Student Notes:

Exercise Recap: Shaft and Groove

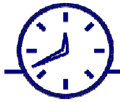
- ✓ Create a shaft feature
- ✓ Create a groove feature
- ✓ Create a multi-pocket feature



Student Notes:

Exercise: Shaft and Groove

Recap Exercise

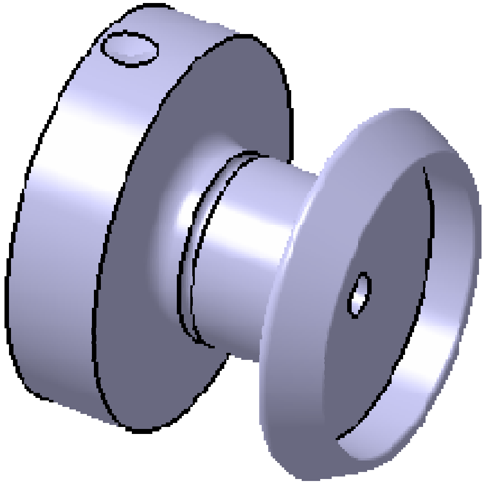


15 min

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:

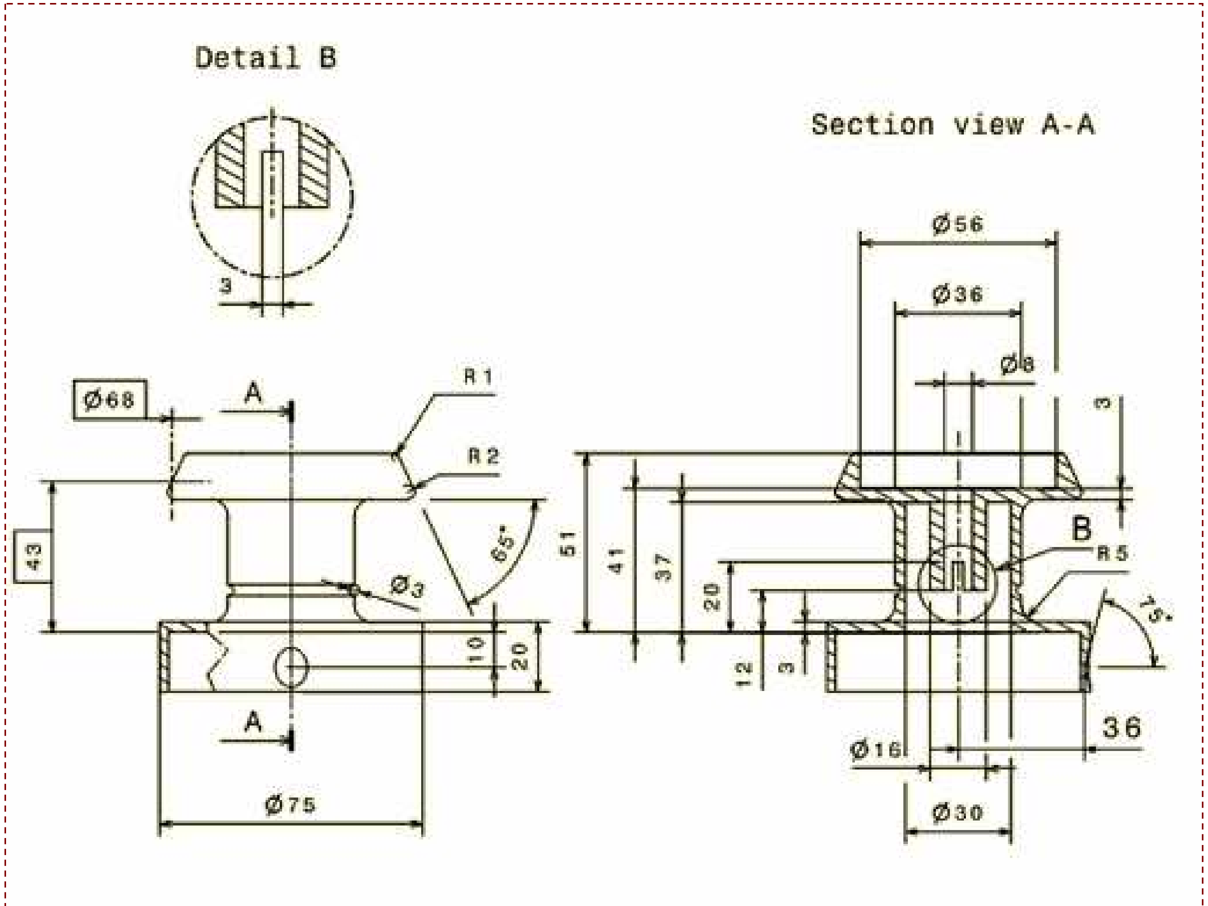
- Create a shaft feature
- Create edge fillets
- Create internal and external groove features
- Create a pocket feature
- Create a reference point and line
- Create a cone-shaped groove feature

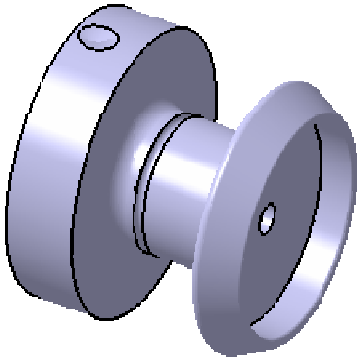


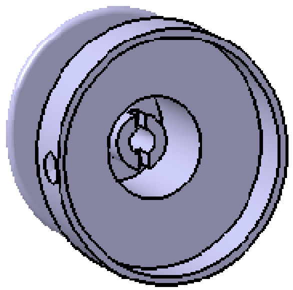
Student Notes:

Do it Yourself

1. Create the following spool part.



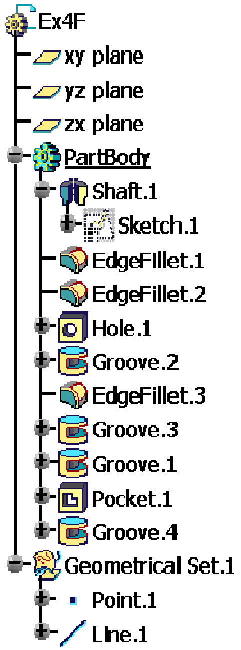


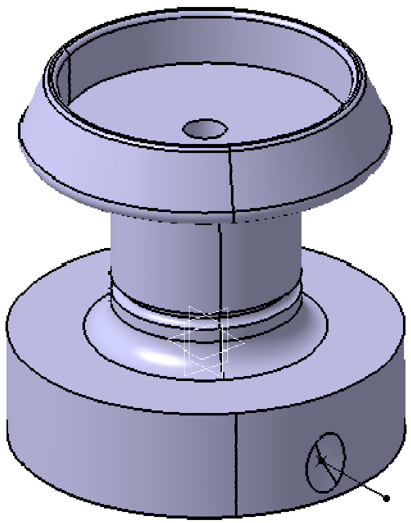


Student Notes:

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





Student Notes:

Shell the Model

In this section, you will learn how to create hollow models using the Shell operation.

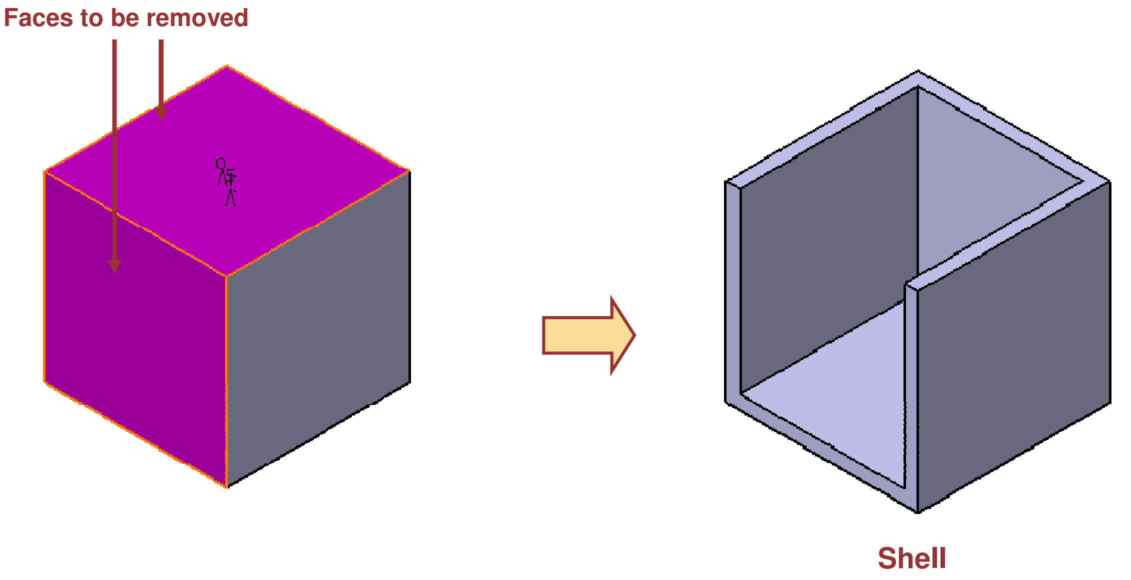
Use the following steps:

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

Student Notes:

Shelling

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.

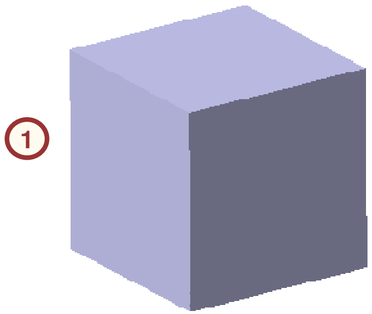


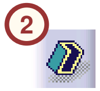
Student Notes:

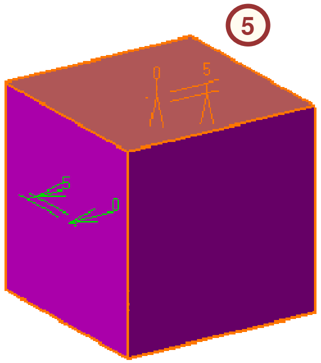
Shelling a Part (1/2)

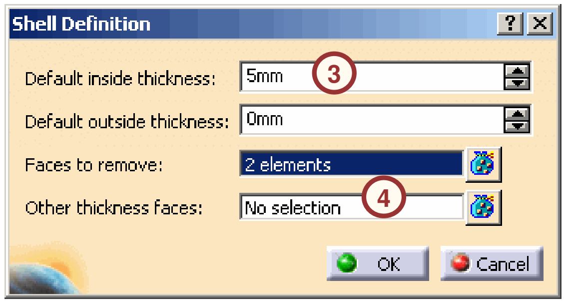
Use the following steps to shell a model where the remaining faces have a different thickness:

1. Select the face(s) to be removed.
2. Click the **Shell** icon.
3. Specify a wall thickness.
4. Select on the **Other Thickness Faces** field.
5. Select the wall(s) that will have a different thickness.







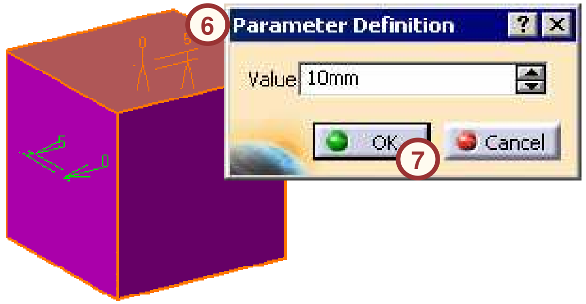


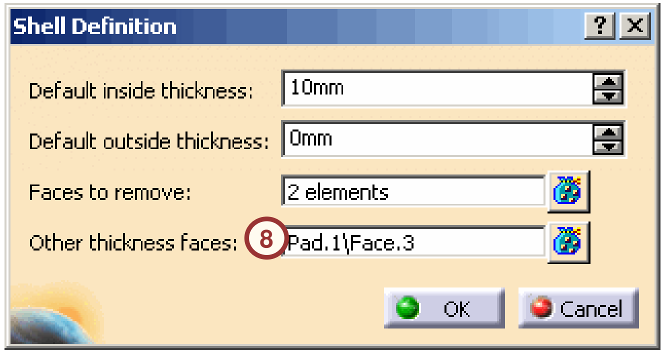
Student Notes:

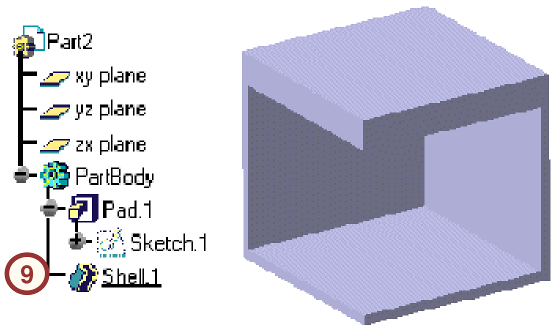
Shelling a Part (2/2)

Use the following steps to shell a model where the remaining faces have a different thickness (continued):

6. To change the thickness of the Other Thickness faces, double-click the dimension directly on the model, and specify the value. Take care to select the dimension associated with the correct direction.
7. Click **OK** to the **Parameter definition** dialog box.
8. Click **OK** to the **Shell Definition** dialog box.
9. The shell feature is added to the model.







Student Notes:

Recommendations for Shelling

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

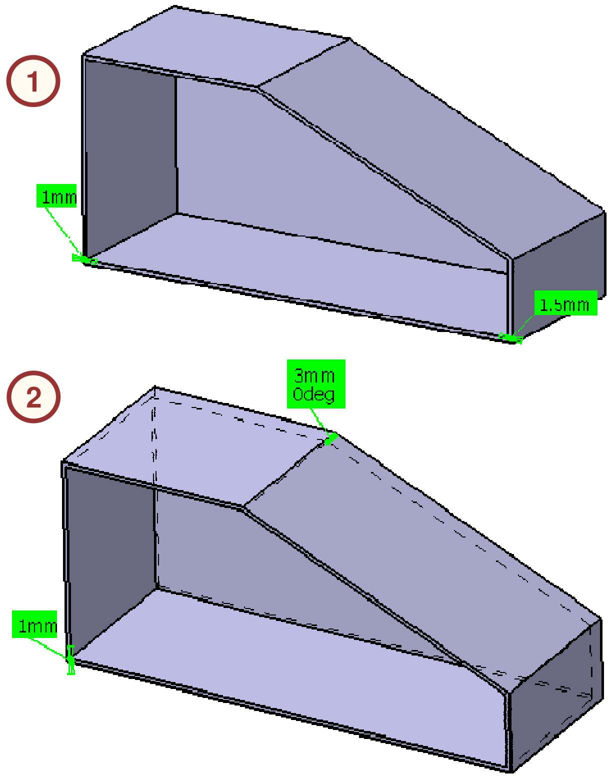
Student Notes:

Avoid Shell with Multiple Thickness

It is recommended that you avoid using shells having multiple thickness value.

It is not easy to visually distinguish the shell thickness values if:

1. The part is large and the shell thickness values are relatively small.
2. If the bottom face is assigned another thickness value, it is not possible to distinguish it unless the hidden lines are displayed or the distance is measured.

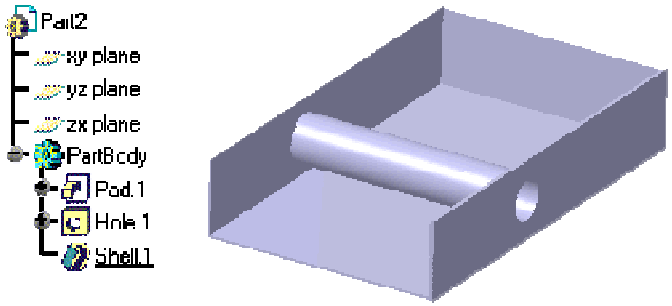


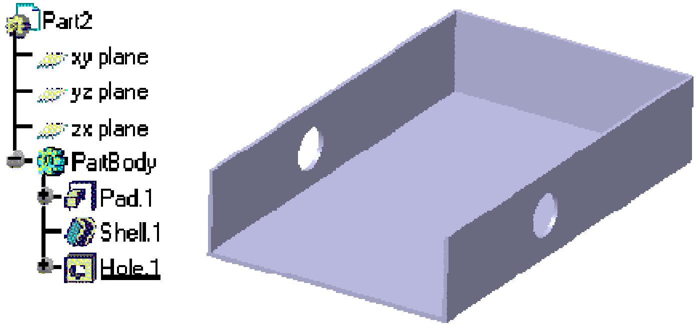
Student Notes:

Importance of Feature Order

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.

For example, when a feature containing a hole is shelled, a pipe is created. If the design intent requires a hole, the shell feature needs to be created before the hole.



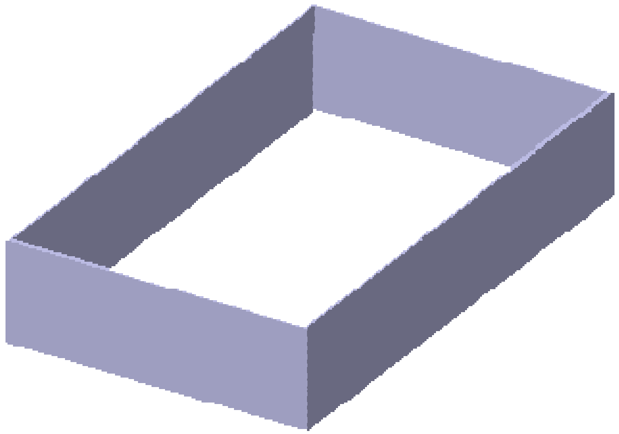


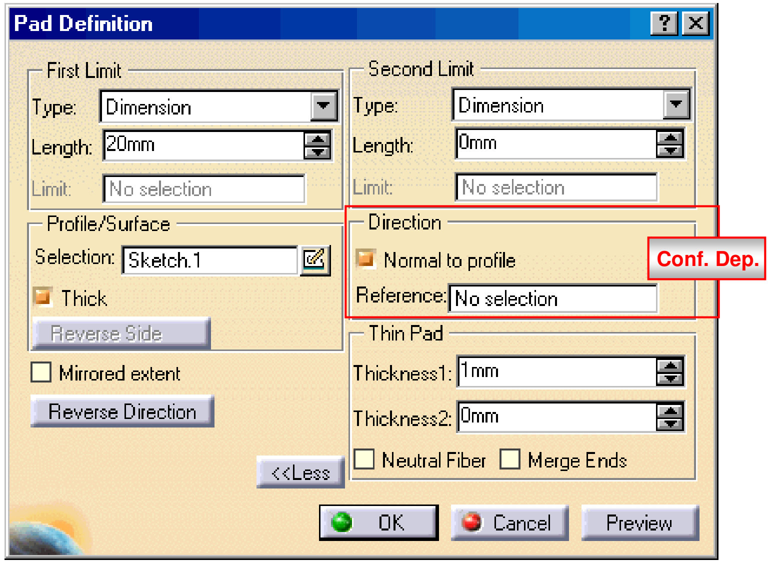
Student Notes:

Thin Features (1/2)

A thin feature is created by applying a constant thickness to a profile. Pads, pockets, shafts, and grooves can all be created as a thin feature. Use the **Pad Definition** dialog box to define its properties:

- 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.



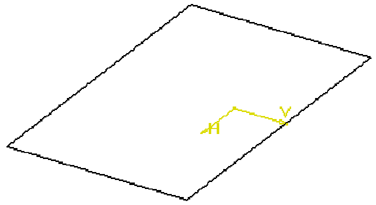


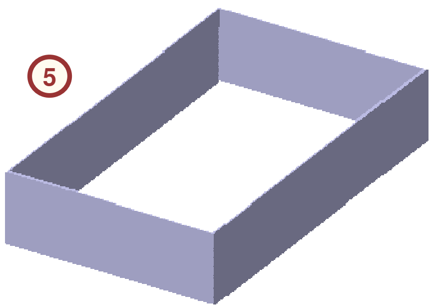
Student Notes:

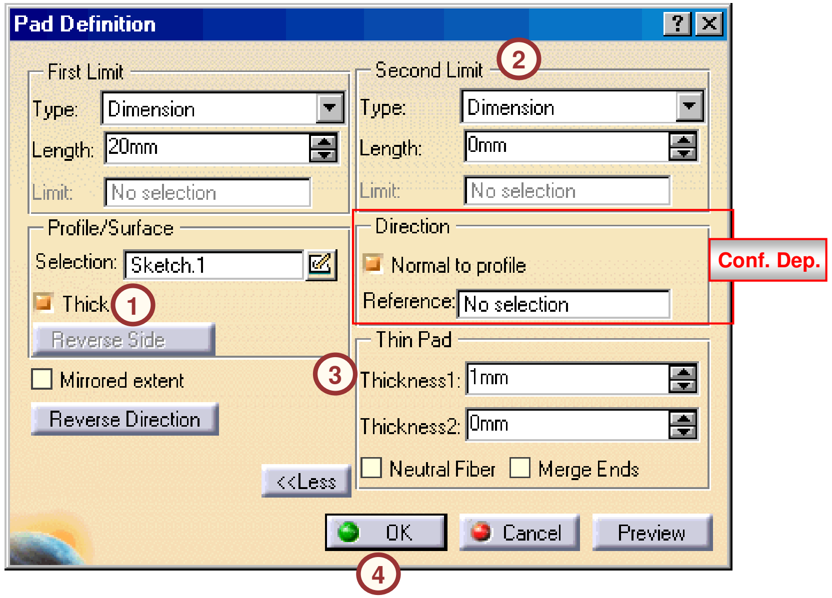
Thin Features (2/2)

The definition dialog boxes for pads, pockets, shafts, and grooves contain a section for defining a thin feature. Use the following steps to create a thin pad:

1. Select the **Thick** option.
2. The dialog box expands to display additional options.
3. Specify the thickness values. Thickness 1 defines the inside thickness, and Thickness 2 defines the outside thickness.
4. Click **OK** to complete the feature.
5. The feature is added to the model.

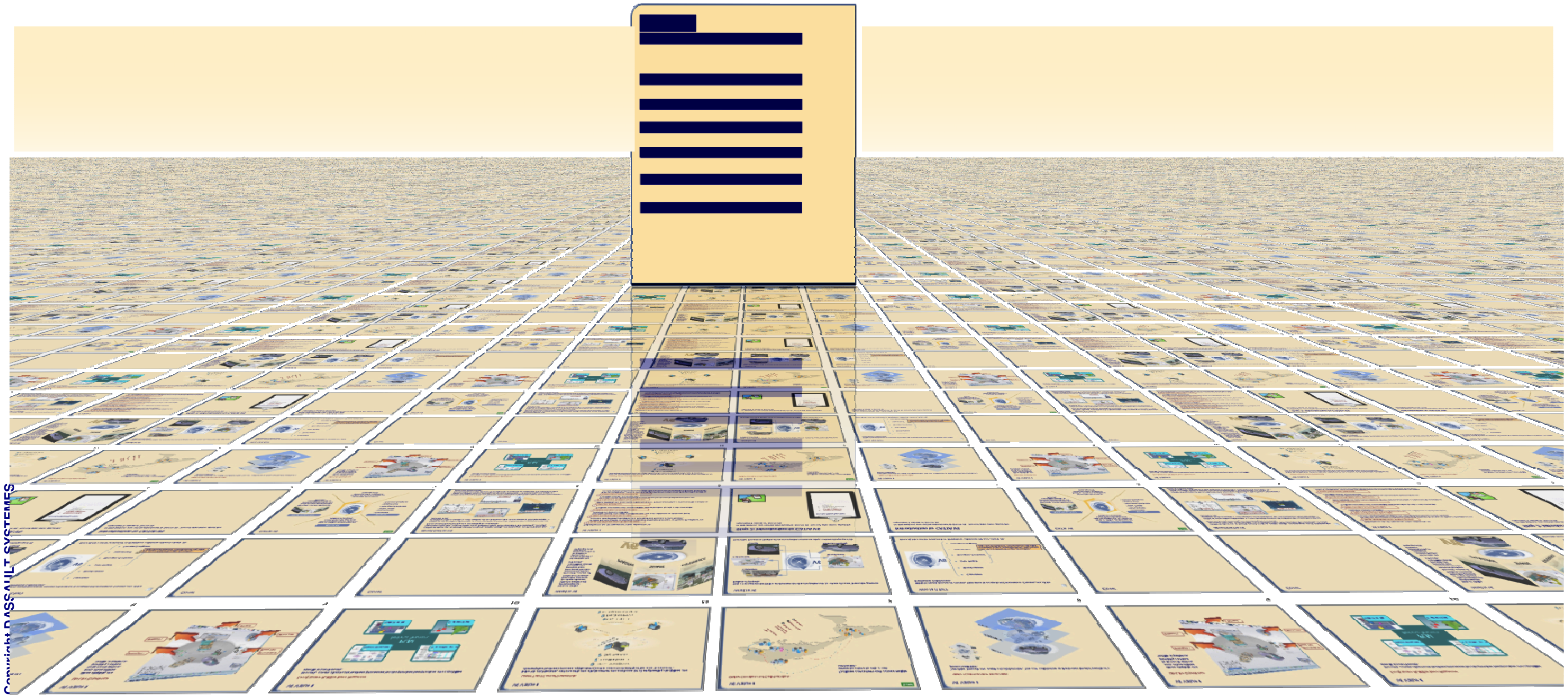






To Sum Up

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



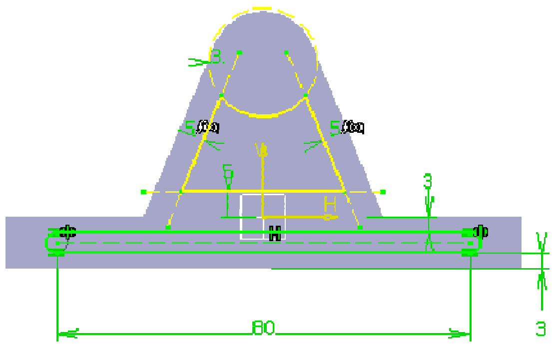
Student Notes:

Student Notes:

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.

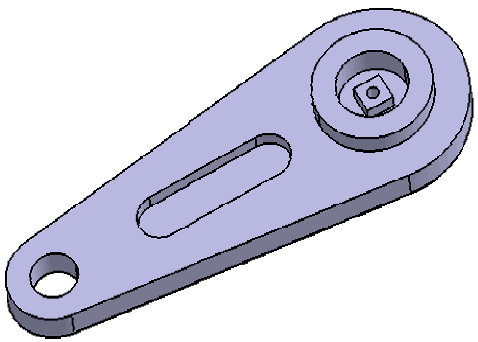
Sketcher includes the following additional tools:



- ✓ 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



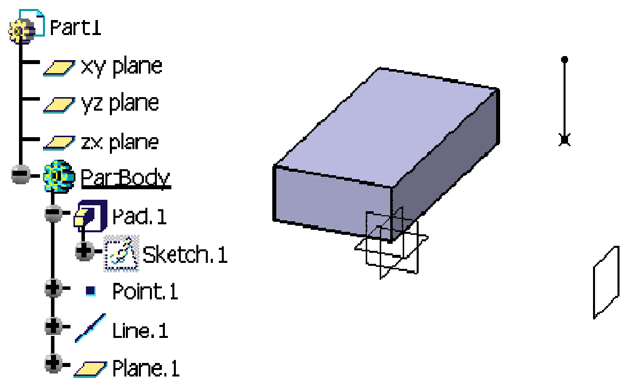
Careful thought must be given to the profiles created in the sketch when they are used to define a Multi-Pad/Pocket. The profiles cannot intersect, they must form a closed loop to avoid feature definition error.

Student Notes:

Create Basic Wireframe Geometry

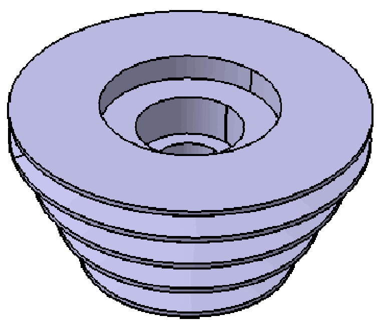
In the Part Design workbench, you have the ability to create points, lines, and planes outside of the Sketcher environment. These elements are called reference or 3D wireframe geometry.

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.

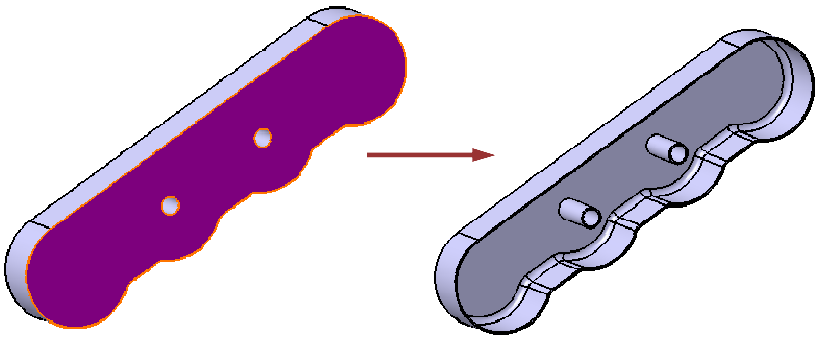
If you did not create an axis in the sketch you can define it from the Shaft/Groove definition window in the Axis selection field. Any linear element in the model can be used.

Student Notes:

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.

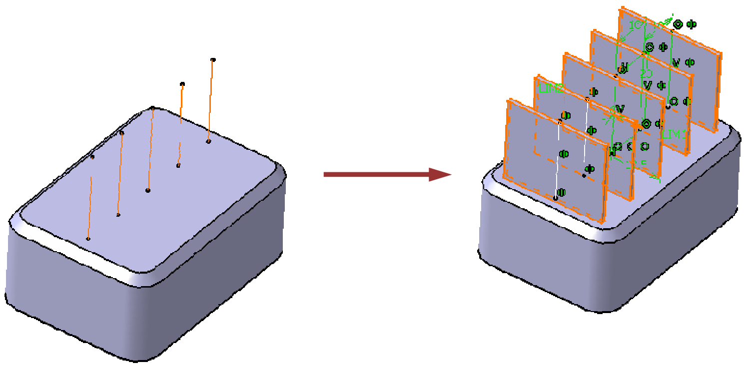
While shelling a model, it is important to consider the feature order. The Shell operation hollows all solid features in a model. If you do not want a feature to be shelled, it must be created after the shell operation.



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.



Student Notes:

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

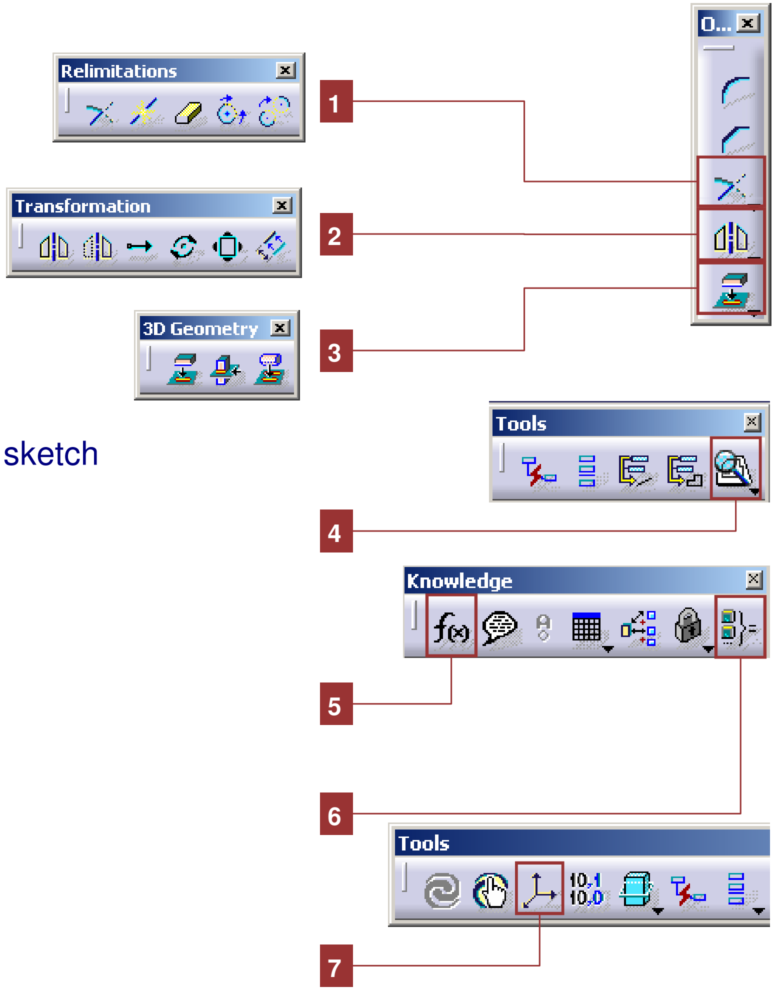
Knowledge

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



Student Notes:

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

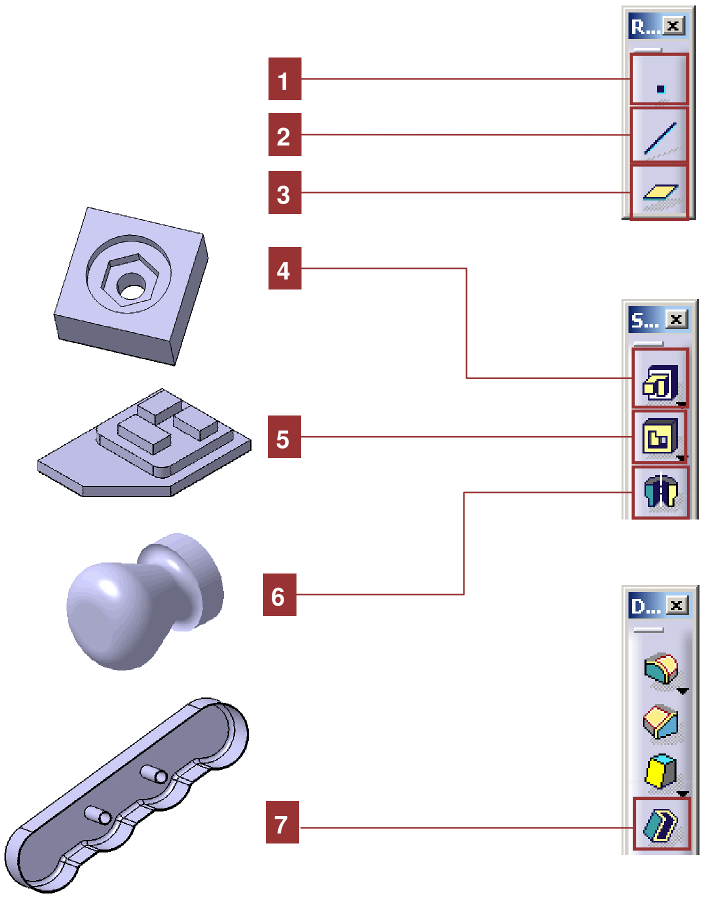
4 **Multi-pad:** creates several pads in one operation

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



Student Notes:

Exercise: Thin Pad and Shell

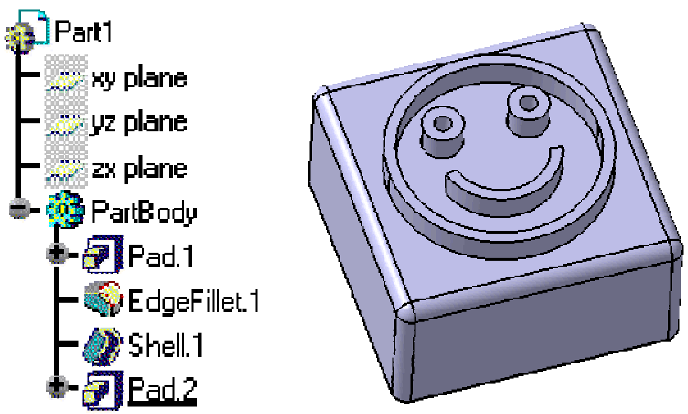
Recap Exercise

15 min

In this exercise you will create a happy face stamp from a new part. You will use the tools learned in this lesson to create a pad, a fillet, shell and thin feature. Detailed instructions for this exercise are provided.

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

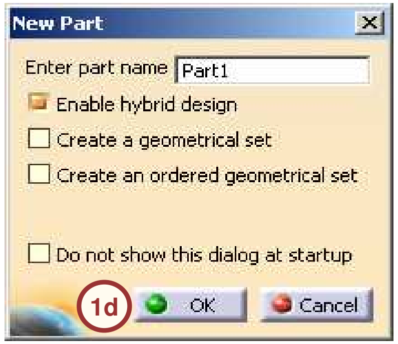
- Create a shell feature
- Create a thin pad



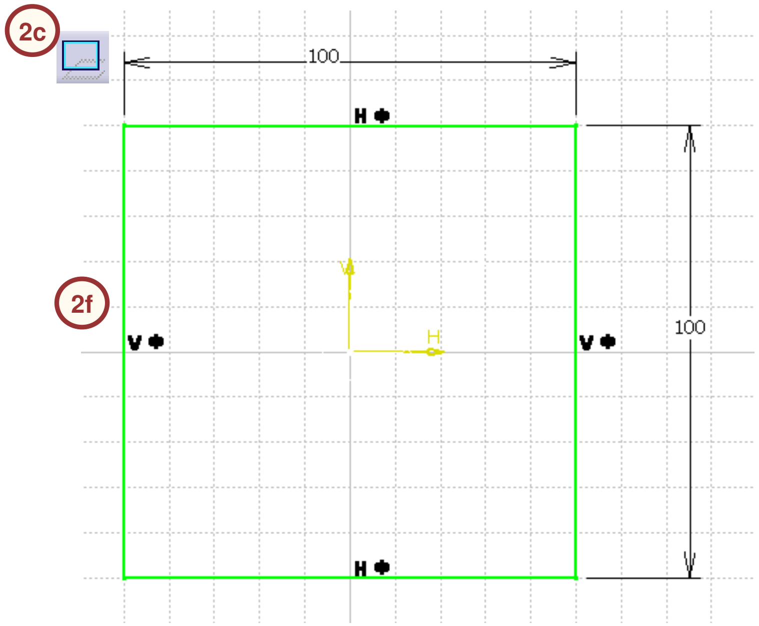
Student Notes:

Do it Yourself (1/6)

1. **Create a new part.**
   - Create a new part file.
     a. Click **File > New**.
     b. Select **Part** from the list of document types.
     c. Click **OK**.
     d. Accept the default name and click **OK**.



2. **Create a sketch.**
   - Create a square profile.
     a. Click the **Sketch** icon.
     b. Select the XY plane to place the sketch.
     c. Click the **Rectangle** icon and sketch an approximate square as shown.
     d. Dimension the square to [100mm] as shown.

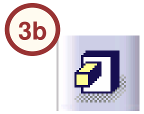


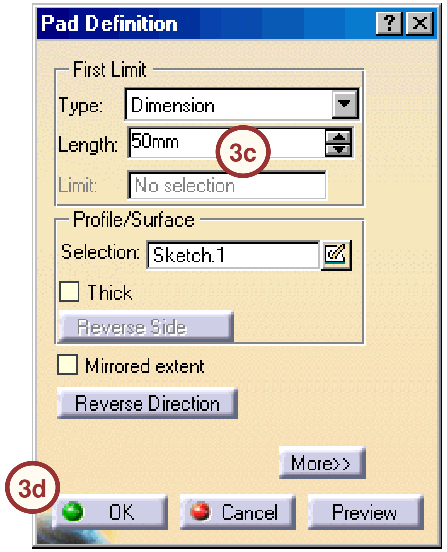
Student Notes:

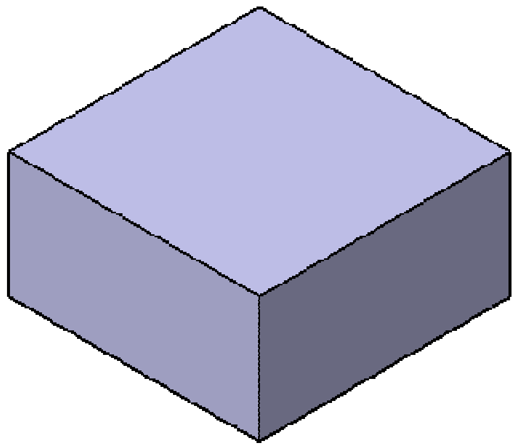
Do it Yourself (2/6)

3. Create a pad.

- Create a pad from the sketch.
  - a. Select Sketch.1.
  - b. Click the **Pad** icon.
  - c. Specify [50mm] as the pad length.
  - d. Click **OK** to complete the feature.







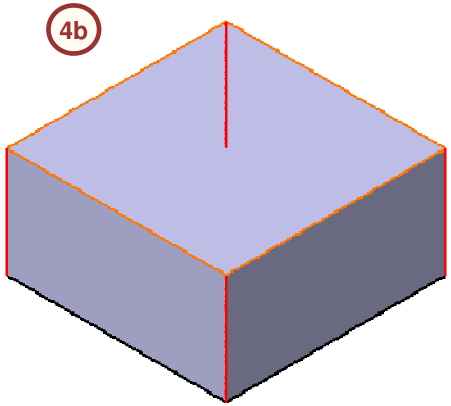
Student Notes:

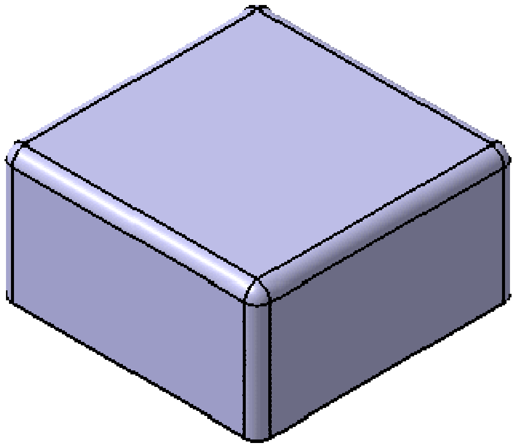
Do it Yourself (3/6)

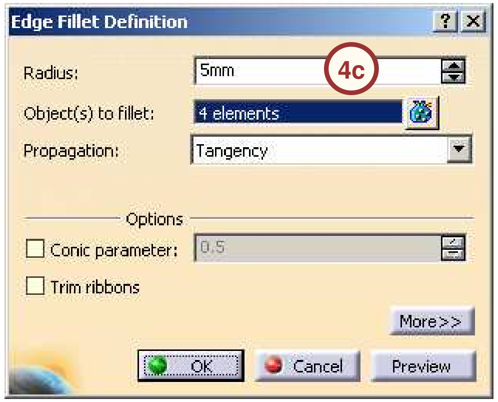
4. **Create an edge fillet.**
   - Create an edge fillet on all the edges of the pad except for the bottom surface edges.
     a. Click the **Edge Fillet** icon.
     b. Select the top surface of the pad and the four vertical edges.
     c. Specify [5mm] as the radius.

4a

4b





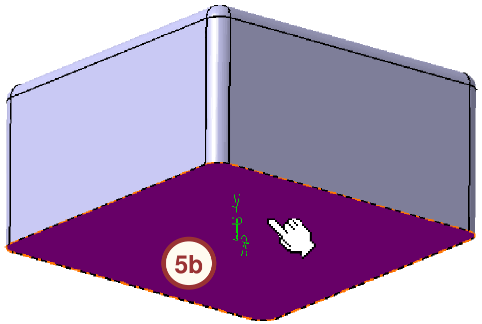


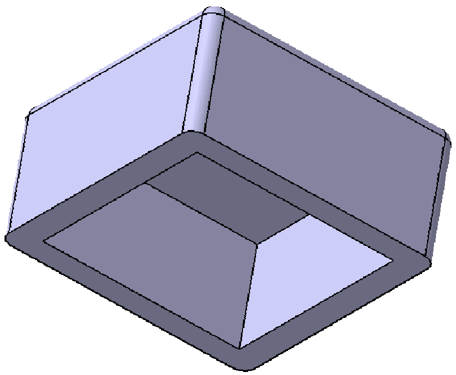
Student Notes:

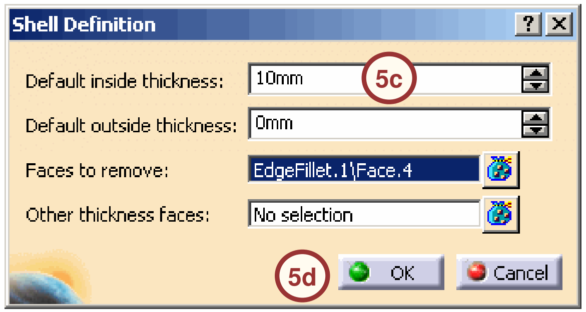
Do it Yourself (4/6)

5. Create a shell feature.

- The filleted pad will be shelled by removing the bottom surface and specifying a thickness to the rest of the surfaces.
  a. Click the **Shell** icon.
  b. Select the bottom surface of the filleted pad.
  c. Specify [10mm] as the **Default inside thickness**.
  d. Click **OK** to complete the shell.





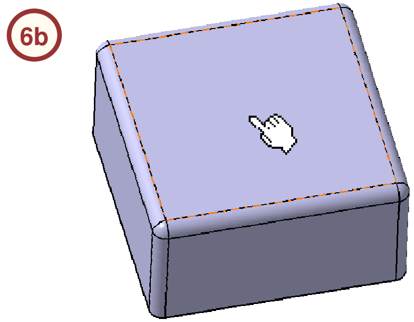


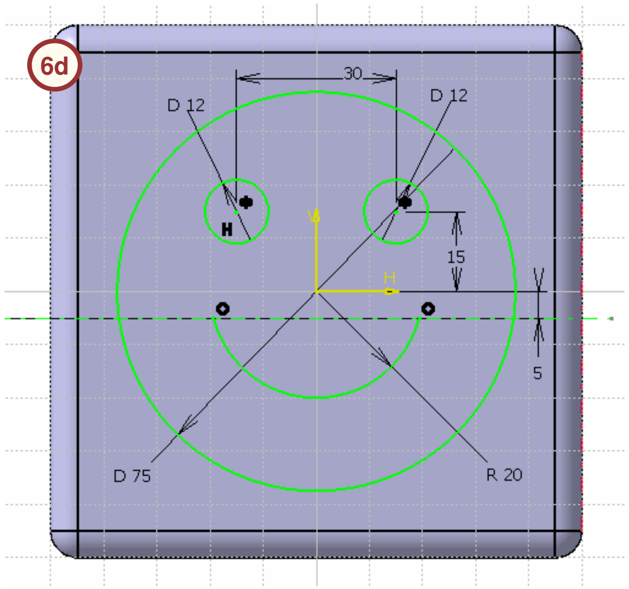
Student Notes:

Do it Yourself (5/6)

6. Create a happy face sketch.

- Create a sketch on the top surface of the pad with circles and arcs that resemble a happy face.
  a. Click the **Sketch** icon.
  b. Select the top surface of the pad as the sketch support.
  c. Sketch three circles and an arc to resemble a happy face.
  d. Dimension the sketch as shown.
  e. Exit the sketcher.



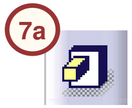


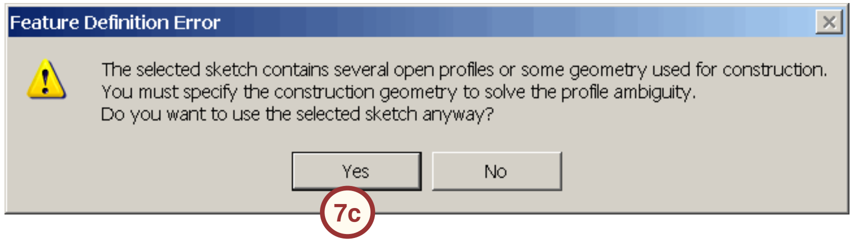
Student Notes:

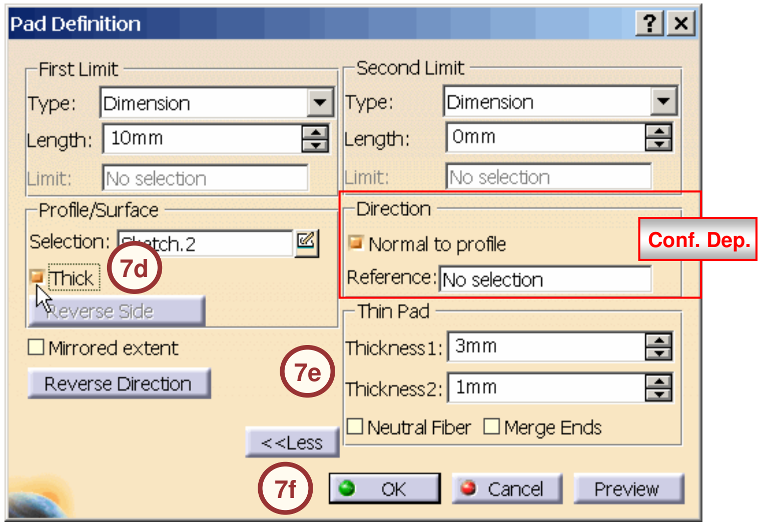
Do it Yourself (6/6)

7. Create a thin pad.

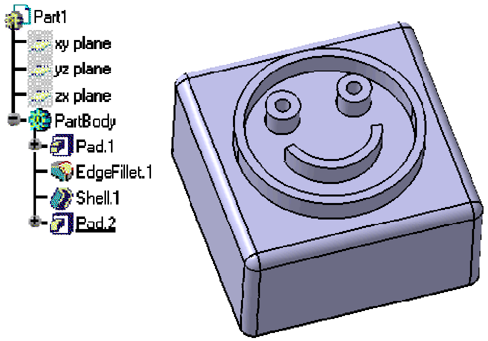
- Use the happy face sketch created in the previous step to create a thin pad.
  - a. Click the **Pad** icon.
  - b. Select the happy face sketch from the specification tree.
  - c. A feature definition error dialog box appears. Click **Yes** to continue.
  - d. When the Pad Definition dialog box opens, select the **Thick** option. The dialog box expands.
  - e. Specify [3mm] for Thickness1, [1mm] for Thickness2, and [10mm] for the Length.
  - f. Click **OK** to complete the pad.







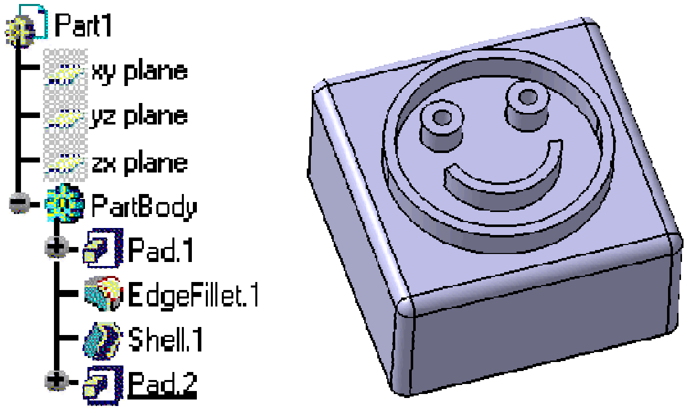
8. Save and close the model.



Student Notes:

Exercise Recap: Thin Pad and Shell

- ✓ Create a shell feature
- ✓ Create a thin pad



Student Notes:

Exercise: Pad, Fillet, Hole and Shell

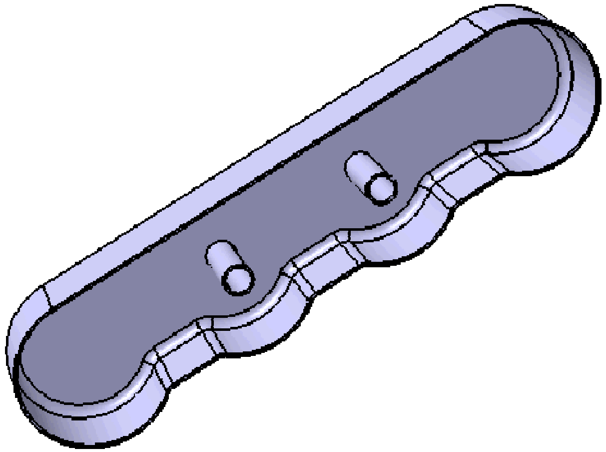
Recap Exercise

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

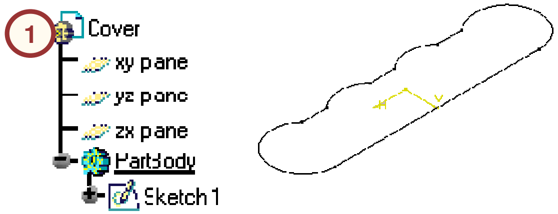


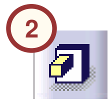
Student Notes:

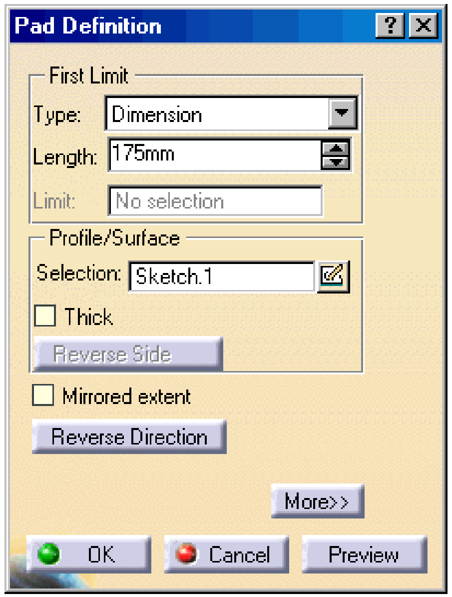
Do it Yourself (1/4)

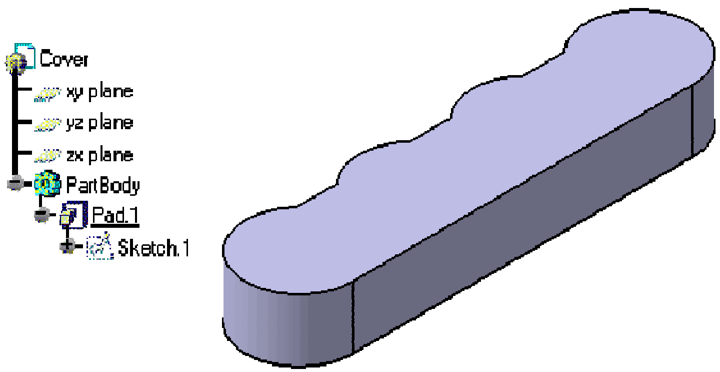
1. **Open up the part Ex4H.CATPart.**
   - Open an existing part file using the Open tool. The part file contains a sketch.

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







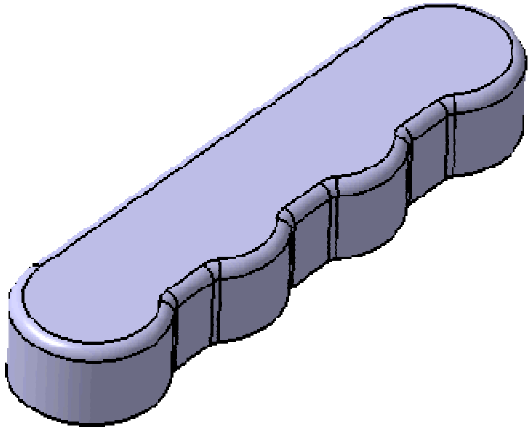


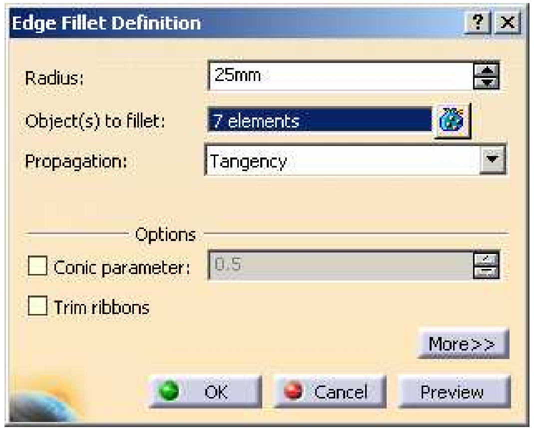
Student Notes:

Do it Yourself (2/4)

3. Create edge fillets.

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



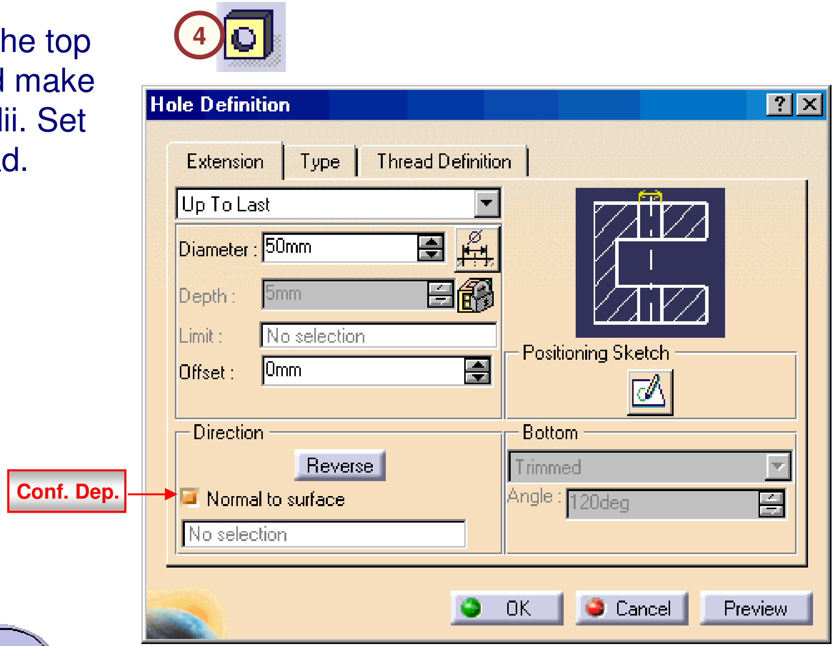


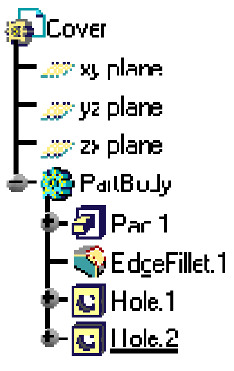
Student Notes:

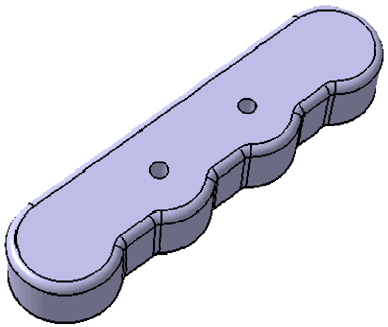
Do it Yourself (3/4)

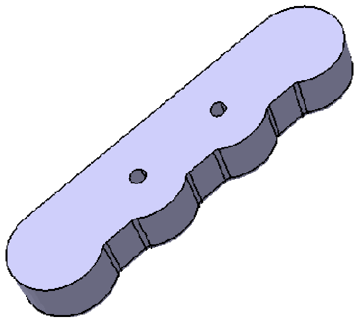
4. Create holes.

- 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.







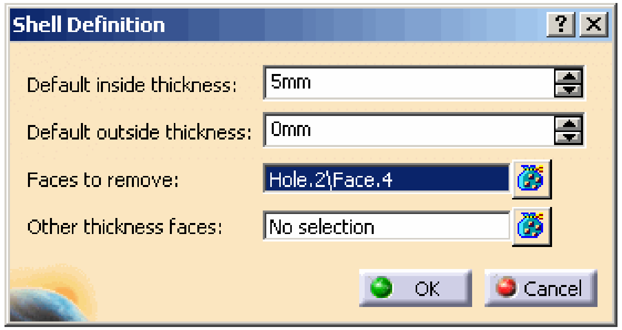


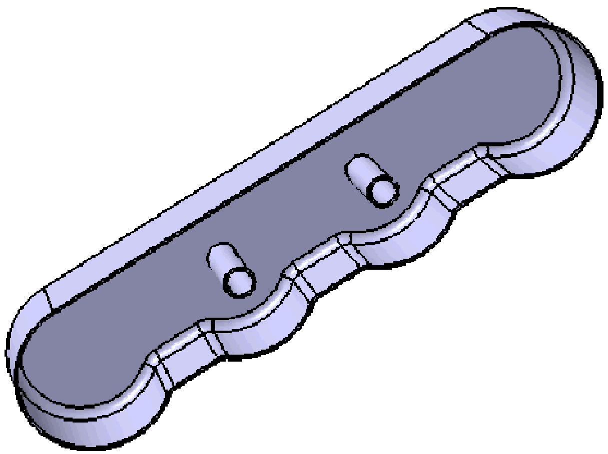
Student Notes:

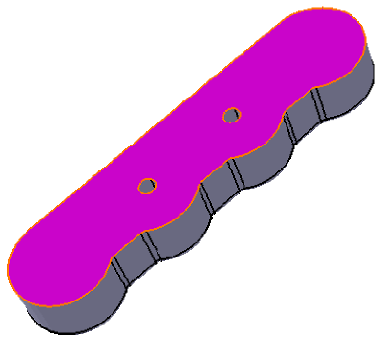
Do it Yourself (4/4)

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.



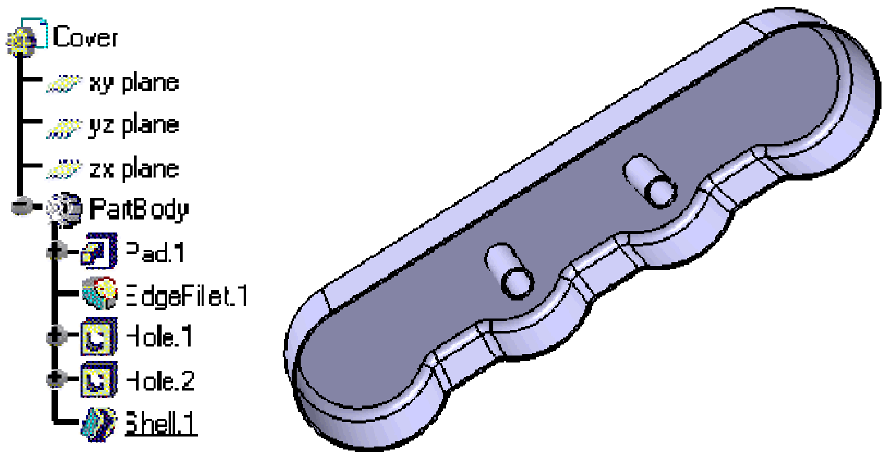




Student Notes:

Exercise Recap: Pad, Fillet, Hole and Shell

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



Student Notes:

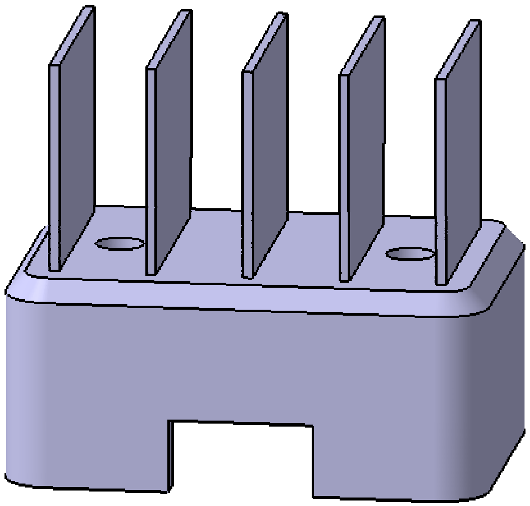
Exercise: Thin Pad, Shell and Holes

Recap Exercise

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:

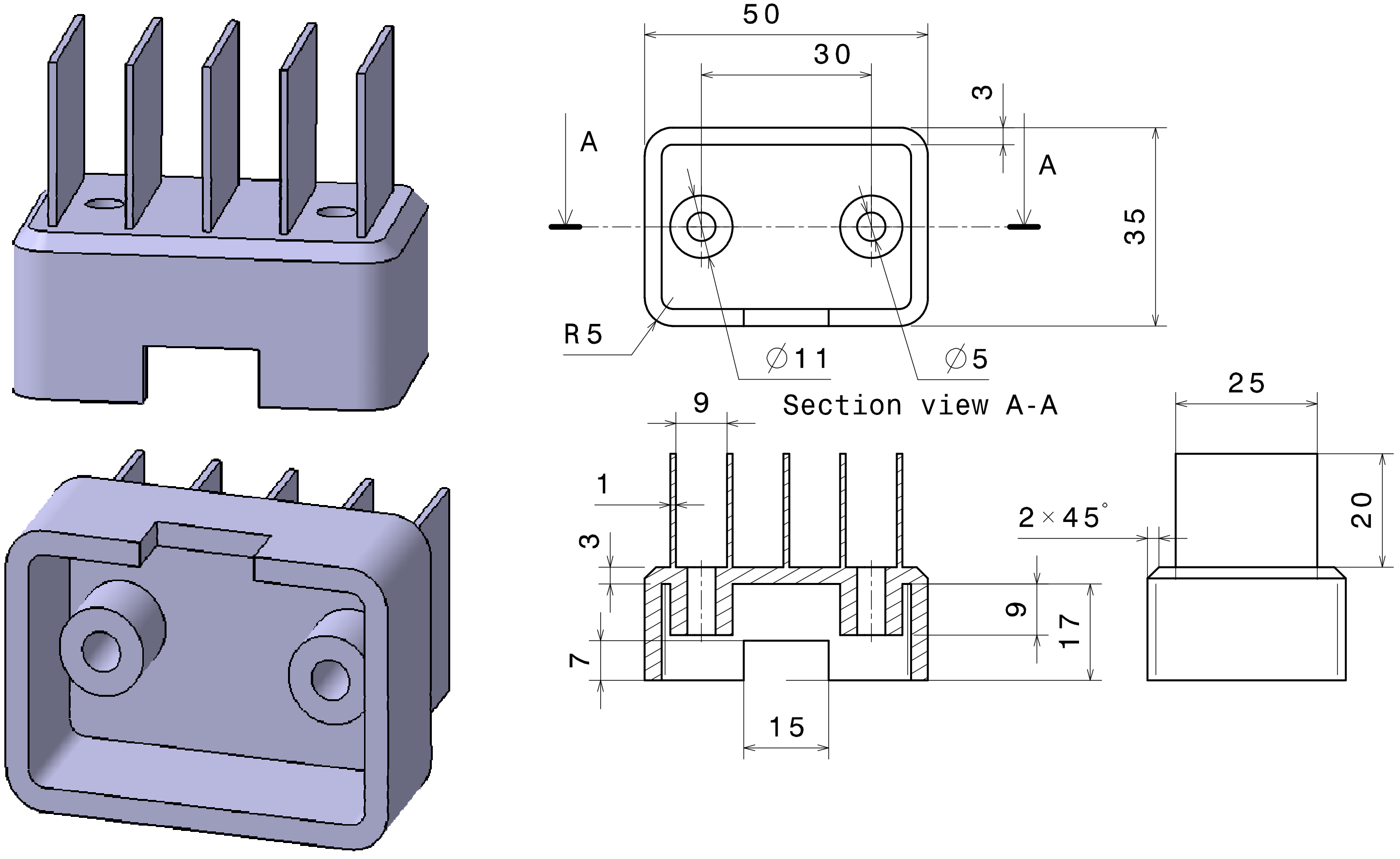
- Create pads
- Create a shell
- Create a thick pad
- Create holes
- Create a chamfer



Student Notes:

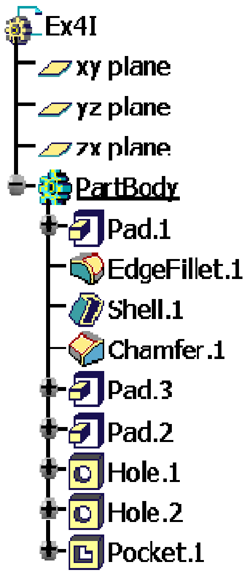
Do it Yourself

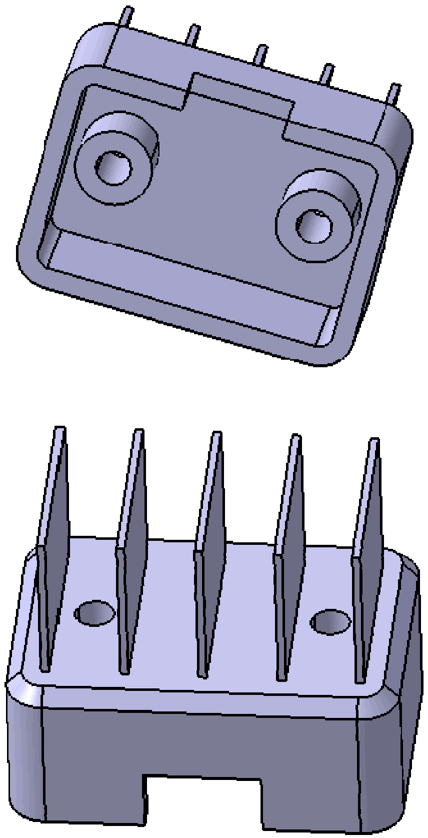
Create the following part.



Student Notes:

Exercise Recap: Thin Pad, Shell and Holes





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

Student Notes:

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.

Student Notes:

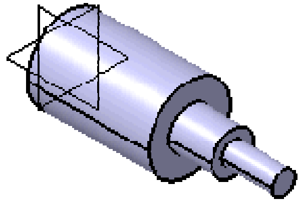
Do It Yourself: Drawing of the Handle Block (1/4)

You will need to create the following features:

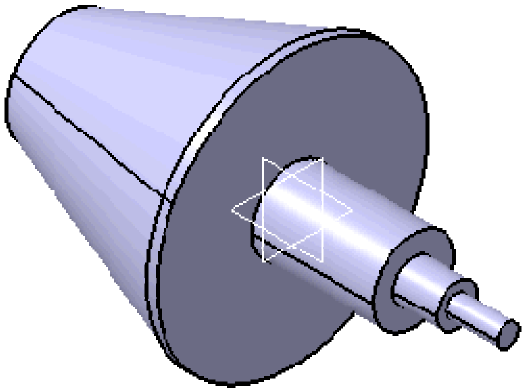
1. **Create a multi-pad feature.**
   - Create the bottom section of the model with the **Multi-pad** tool.

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

1



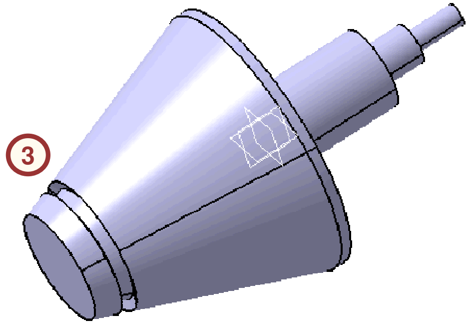
2



Student Notes:

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

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

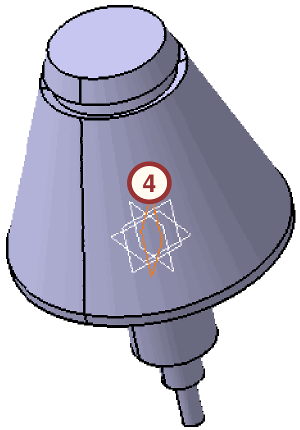


3. Create a groove.

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

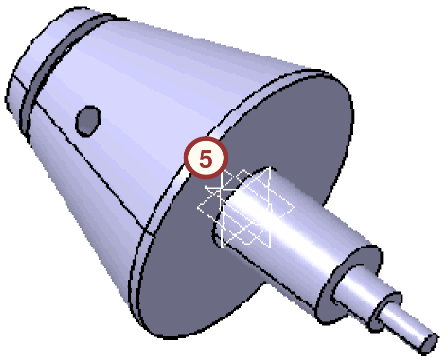
4. Create a plane.

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



5. Create holes.

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



Student Notes:

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

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

6. Create a pocket.

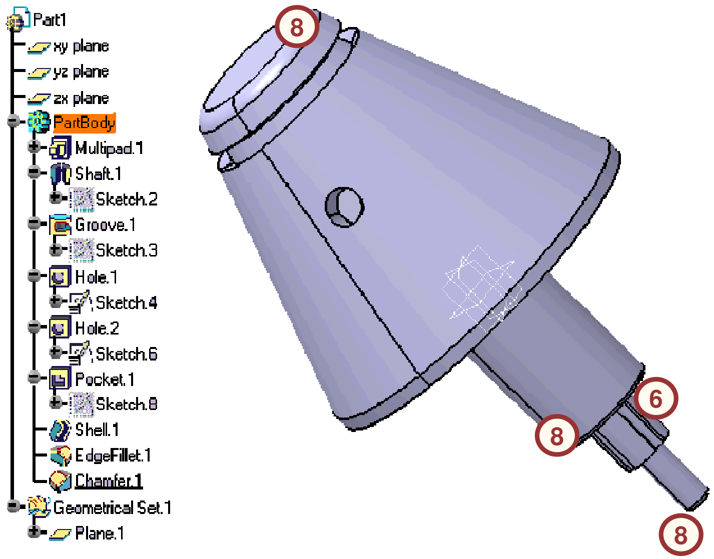
- Create the sketch for the pocket by creating one profile, use the **Rotate** tool to create the remaining three profiles.

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.

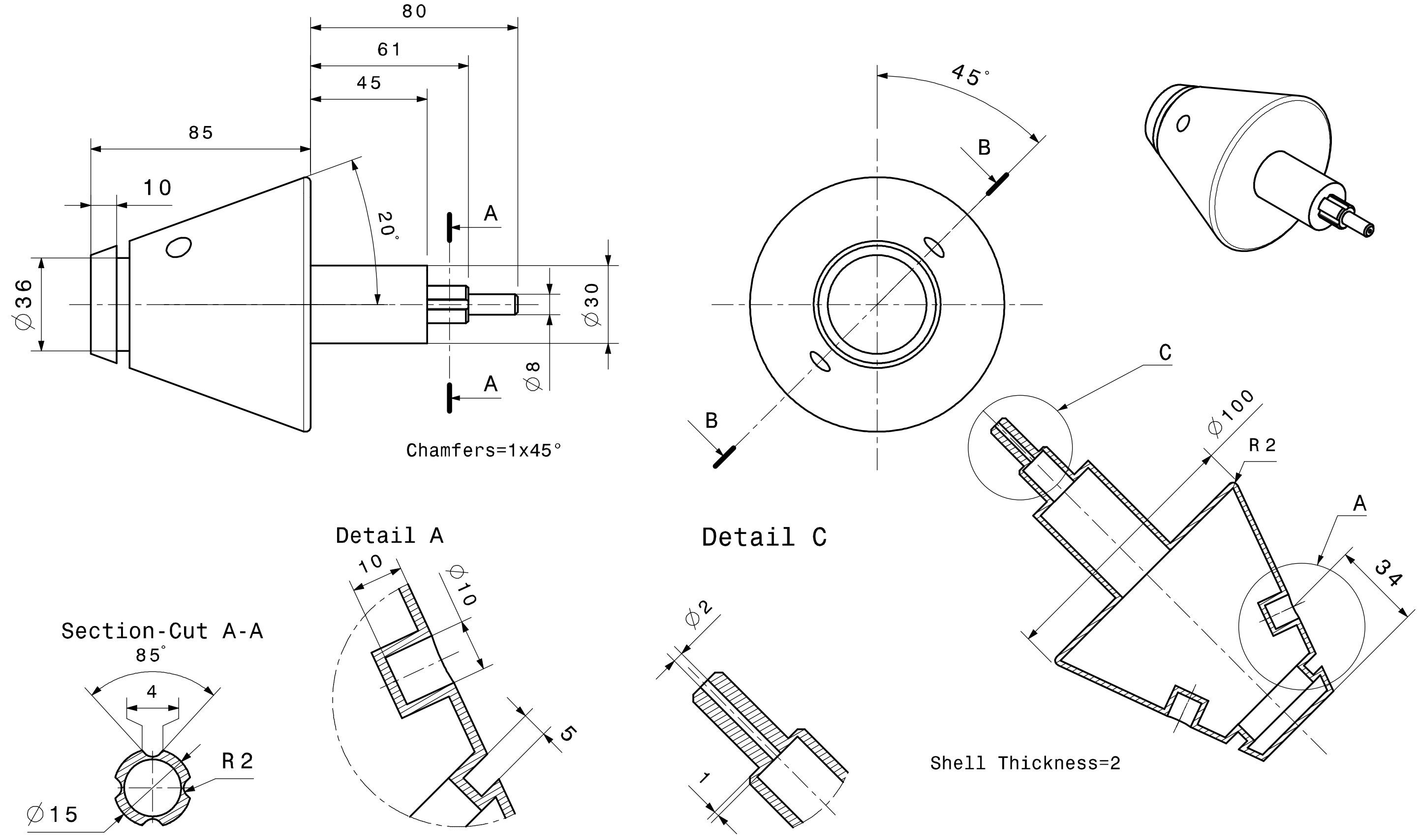
- Complete the model by adding a 5mm fillet to the top edge and a 1mm x 45 degree chamfer to the two edges shown.



Student Notes:

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

Use the dimensions shown to complete the Handle Block part.



Student Notes:

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