EXERCISE BOOK

Student Notes:

CATIA V5 Training

Exercises

Part Design Expert

Version 5 Release 19
September 2008

EDU_CAT_EN_PDG_AX_V5R19

Student Notes:

Table of Contents (1/5)

Student Notes:

Table of Contents (2/5)

Student Notes:

Table of Contents (3/5)

Student Notes:

Table of Contents (4/5)

Student Notes:

Table of Contents (5/5)

Student Notes:

Connector Housing

You will Apply the Concepts learned throughout the Part Design Course, by building the Master Exercise and following the Recommended process.

- Connector Housing Presentation
- Step 1: Get Familiar with the Design intent
- Step 2: Design the part with its original specifications
- Step 3: Receive and get familiar with the modification request
- Step 4: Modify the design following the modification request

Student Notes:

Connector Housing

You will practice the topics, learned throughout the course, by building the Connector Housing. To complete the exercise, you will follow the recommended steps.

Connector Housing

Master Exercise Presentation

90 min

In this exercise you will build a first version of the "Connector Housing".

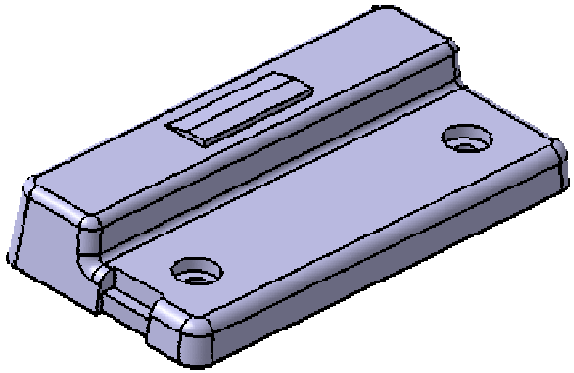
Step 1: You will understand the design Intent of the Part.

Step 2: You will Design the Housing with its initial specifications of dimensions provided in the drawing of the part.

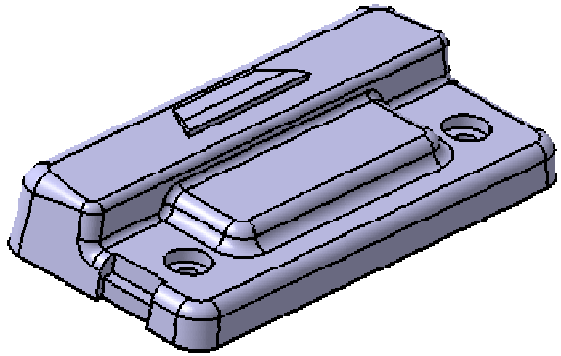
Step 3: Then, you will receive a modification request, that will make you change the part. You need to thoroughly understand the modification request.

Step 4: You will take into account the modification request and change the design accordingly to design the second version of the Connector Housing.

By following the recommended design process you will be able to design the two versions of the part.



First Version according to specifications



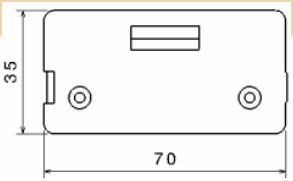
Second Version after modficaions

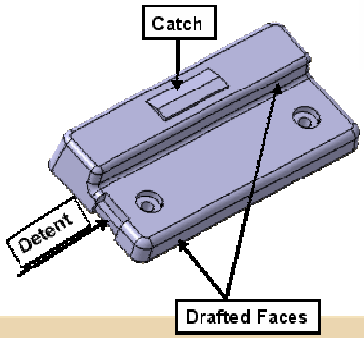
Student Notes:

Exercise Scenario: Connector Housing

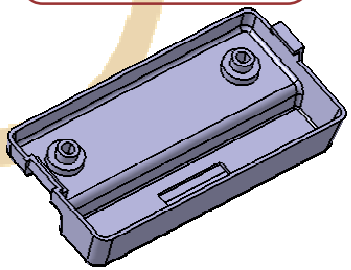
Student Notes:

1 Get familiar with the design intent

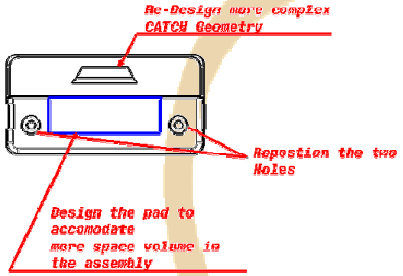




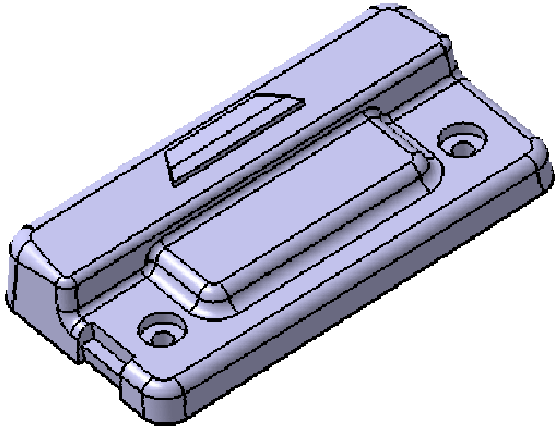
2 Design the part with its original specifications



3 Receive and get familiar with the modification request



4 Modify the Design following the modification request



Student Notes:

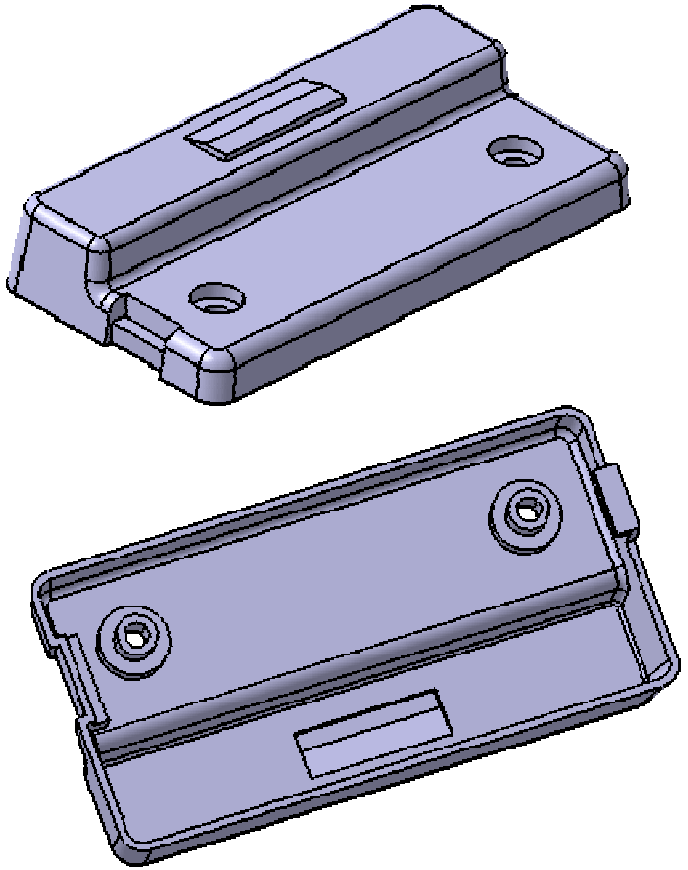
Connector Housing

Step 1: Get Familiar with the Design intent

20 min

In this step you will understand the specifications and design intent required to design the first version of the “Connector Housing”.

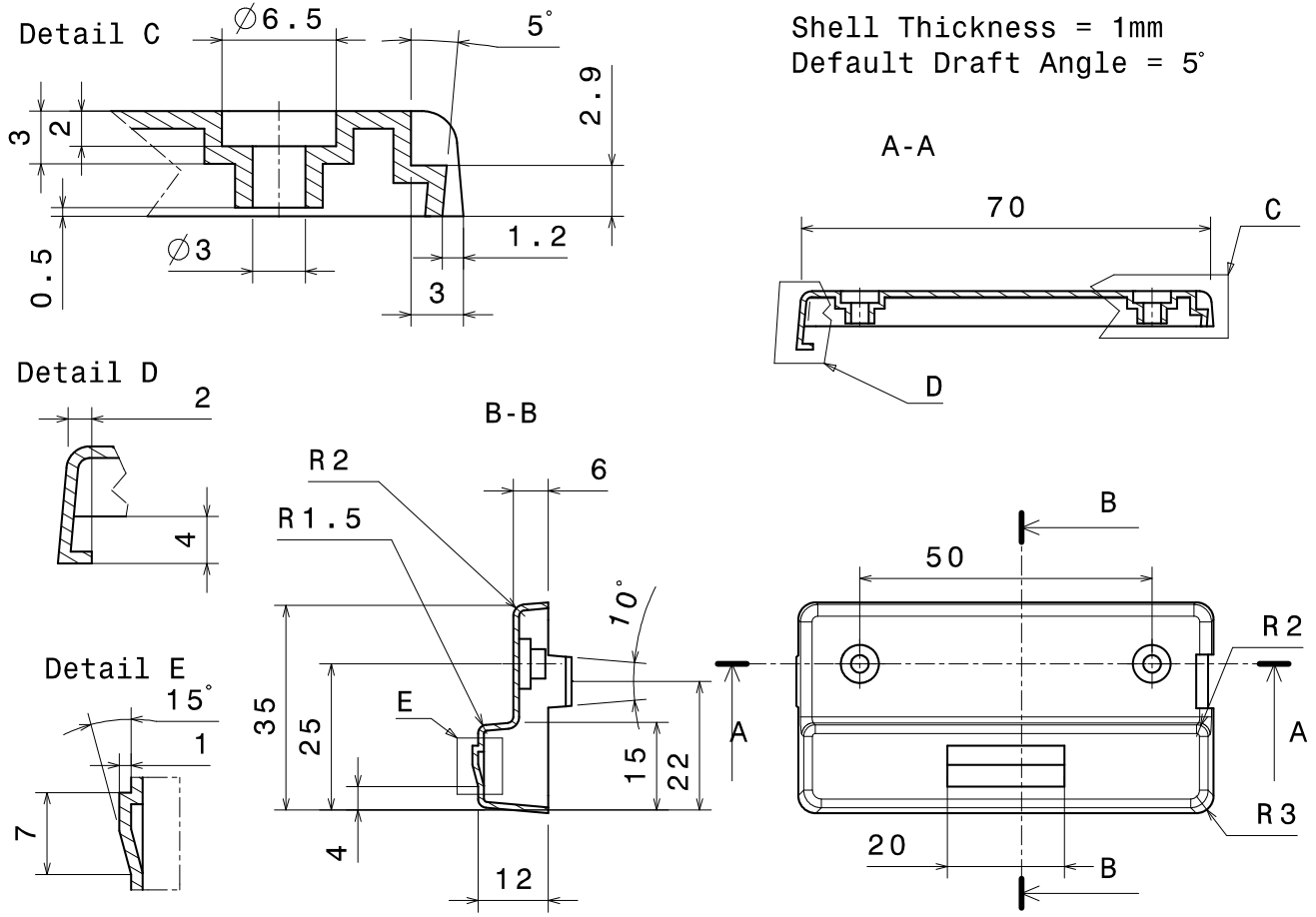
- You will understand the functional parts of the housing.
- You will study the drawing of the part thoroughly.
- You will answer a few questions in order to understand the way manufacturing features (such as draft and fillets) interact with functional features.



Student Notes:

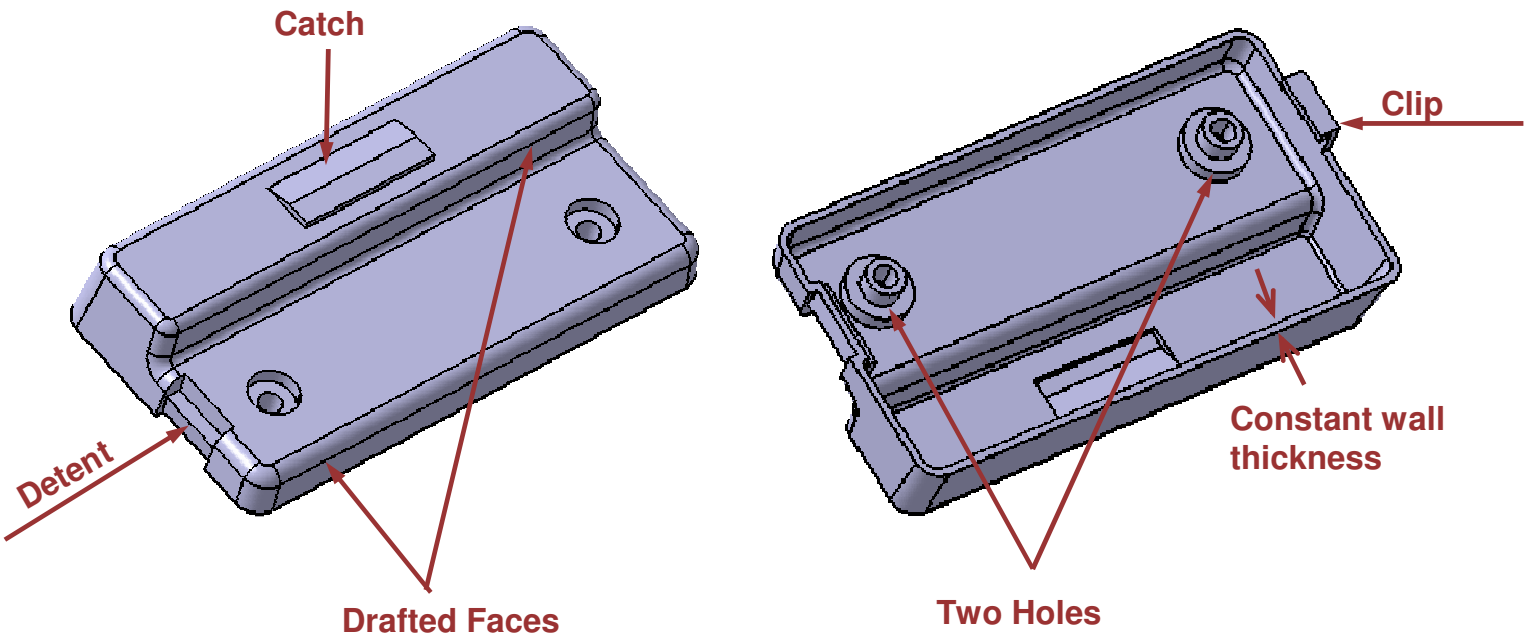
Connector Housing Drawing

- Understand the drawing thoroughly to design the part according to the specifications



Student Notes:

Design Intent: Connector Housing



- The connector housing is a molded part that is used in an assembly.
- The catch is centered on the base.
- Two counter bored through holes have same radius.
- The detent pocket and the clip are equally positioned.
- Surfaces need to be drafted to allow removal from the mold.
- Wall thickness is required to be constant.

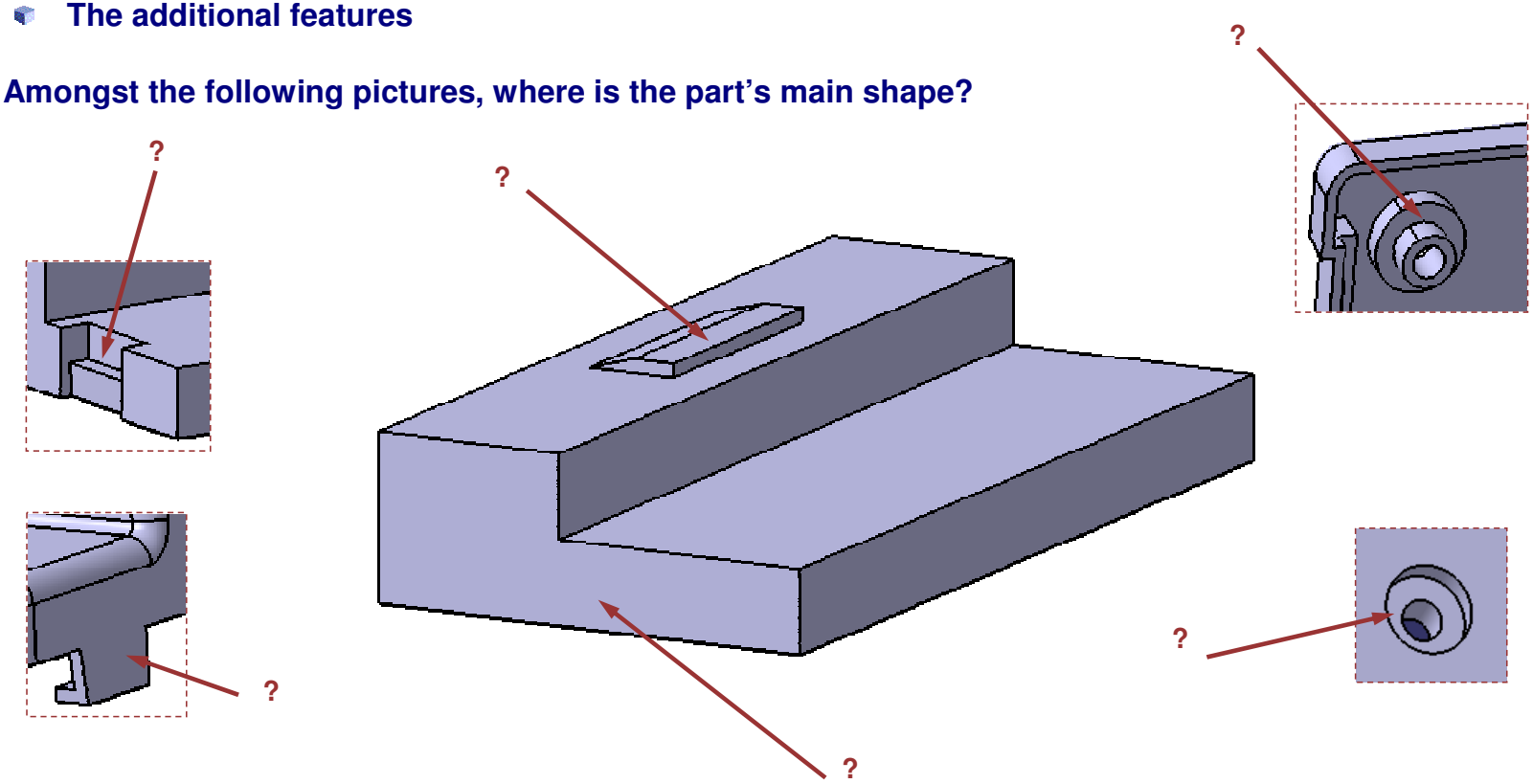
Student Notes:

Getting familiar with the Part's main function

The first thing to do is to identify the functional features of the part

- **The main shape**
- **The additional features**

Amongst the following pictures, where is the part's main shape?



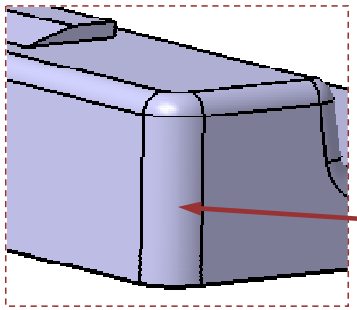
The Part is a housing. So, what should be the main function of the part?

Student Notes:

Few Questions:

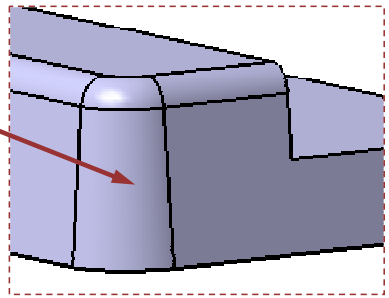
The part has a constant wall thickness. Some edges are filleted and few faces are drafted.
Each feature has a design intent and impacts the way the part should be designed.

- What is the easiest way to create a part with a constant thickness?
- What is the CATIA standard tool that will allow you to manage the draft angle?
- How is it possible to manage drafted faces and constant fillets?



This face has a constant radius

This face could also have a variable radius



Should you create your draft after your fillets or Vice-versa?

During the design, when is the best moment to apply a constant thickness?

- Before or after the fillets?
- Before or after the holes?

Student Notes:

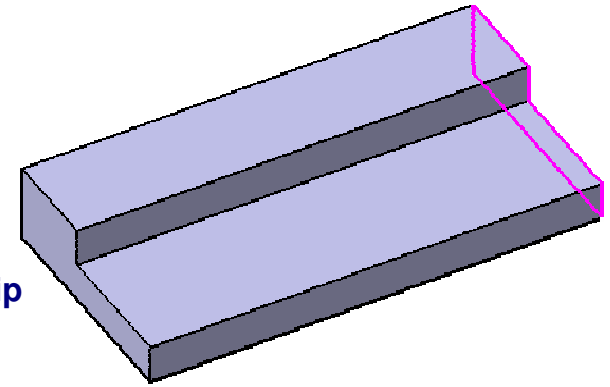
Connector Housing

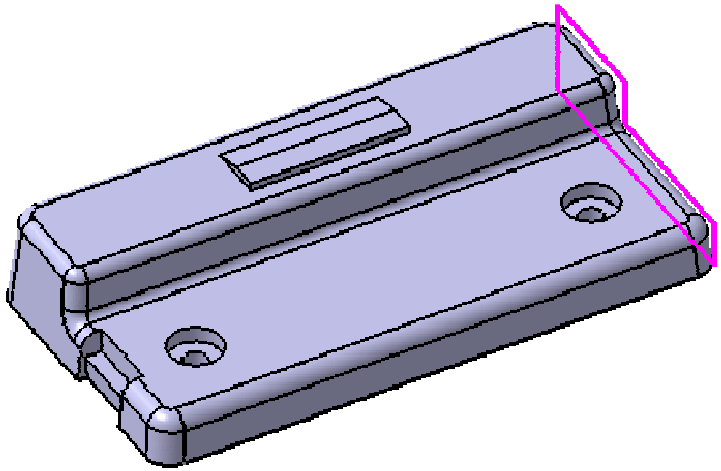
Step 2: Design the part with its original specifications

30 min

In this step you will create the Part corresponding to the dimensions and specifications provided. You will:

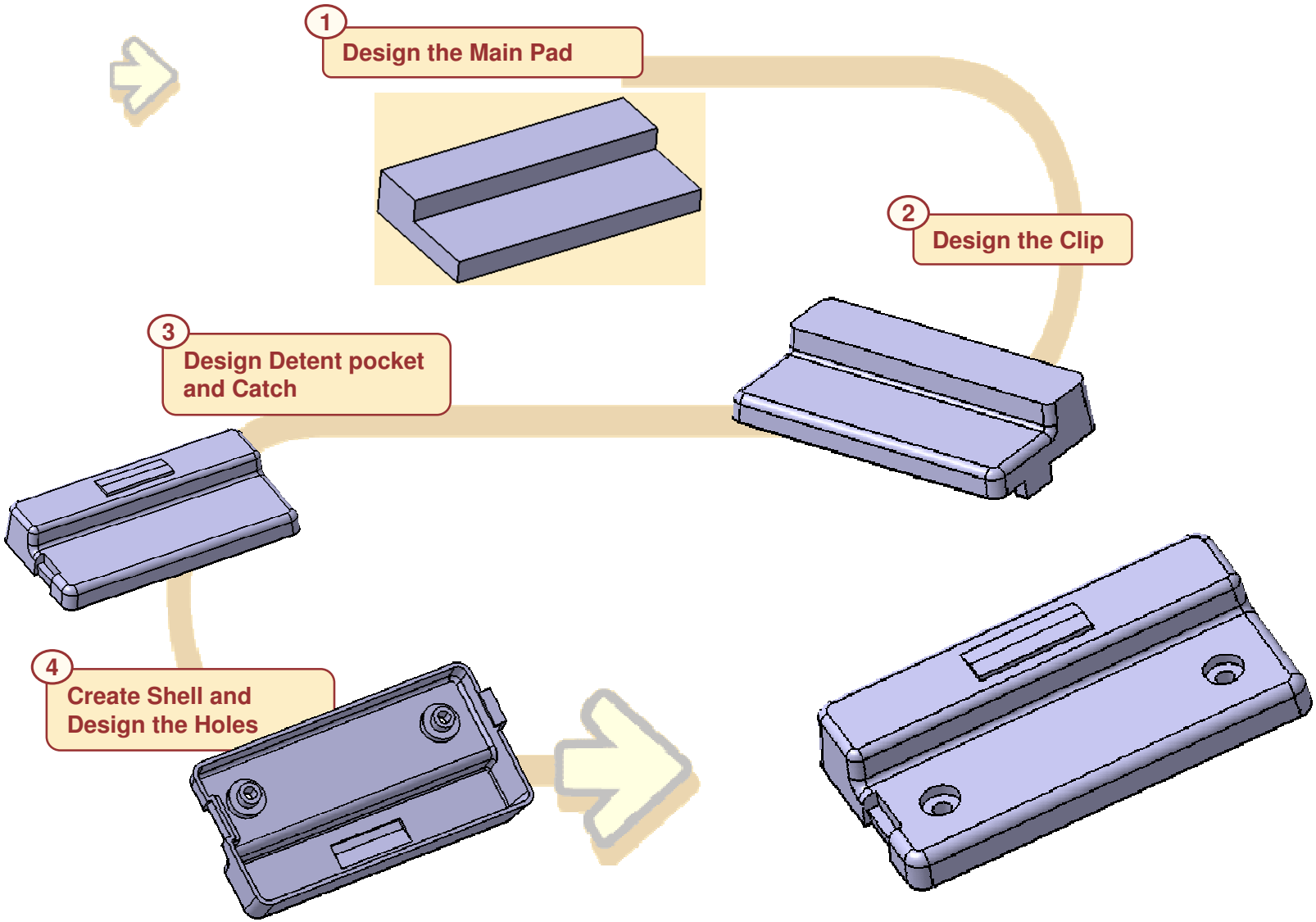
- **Create the Main Pad**
- **Design the various features like Detent Pocket, Catch, Clip**
- **Apply Dress-up features**
- **Create Holes**





Student Notes:

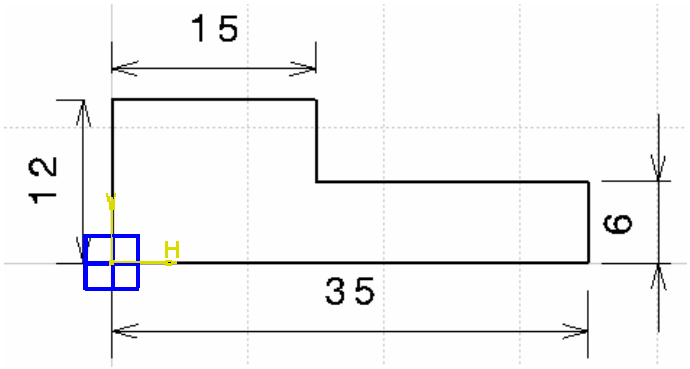
Step 2: Design Process

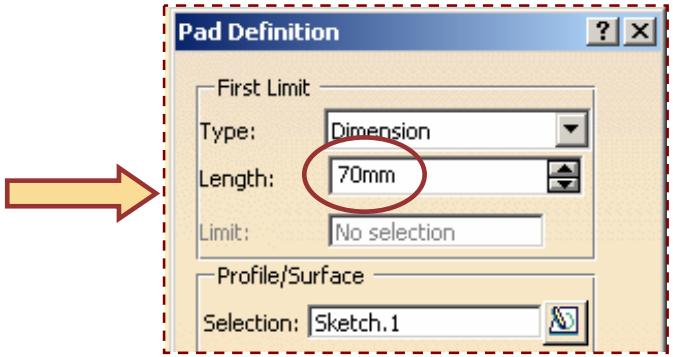


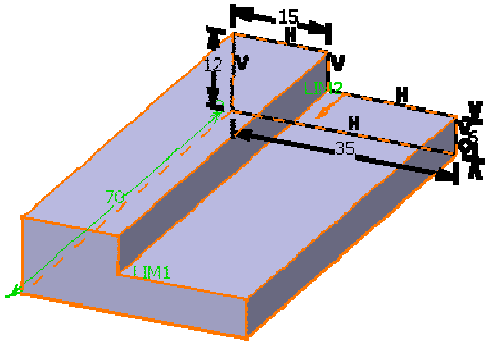
Student Notes:

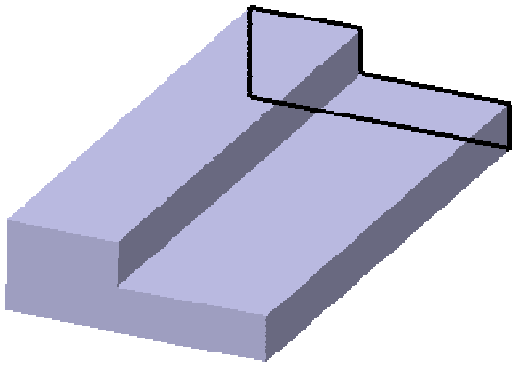
Design the Main Pad

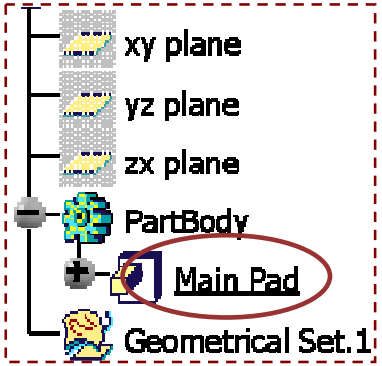
- Create the following sketch on YZ Plane.
- Extrude the sketch by 70 mm using Pad feature from Sketch Based Feature toolbar.
- Rename this Pad to 'Main Pad' by accessing the properties using the contextual menu.







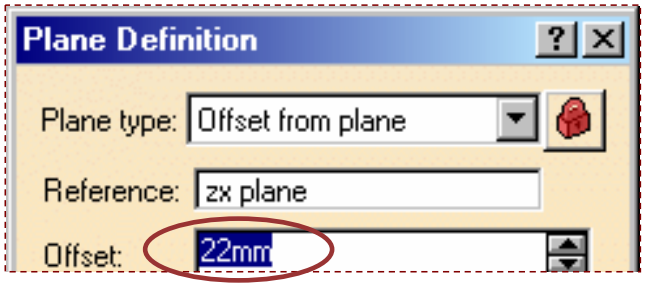


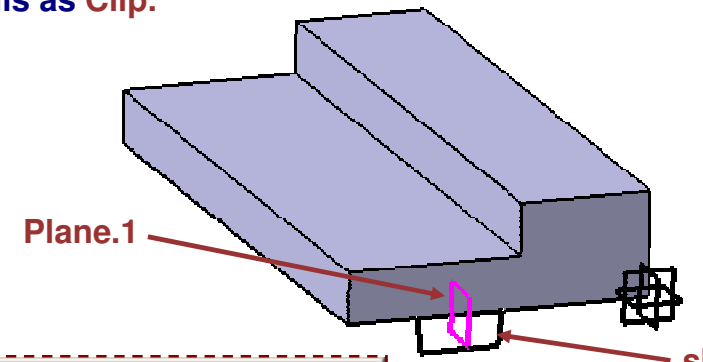


Student Notes:

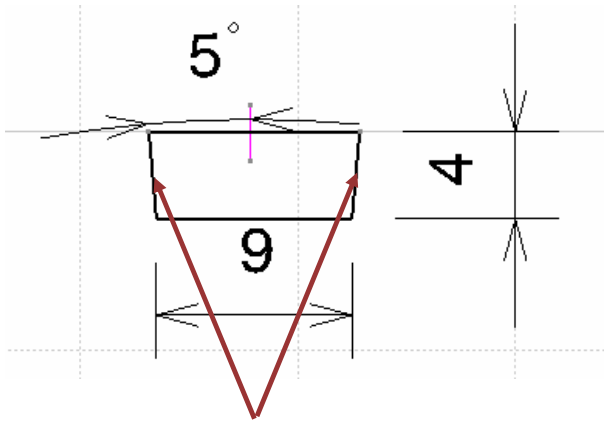
Design the Clip (1/3)

- Create a Plane(Plane.1) offset from ZX plane at 22 mm. Create it in Geometrical set.
- On YZ plane create the following sketch. Use recently created plane to constrain this sketch.
- Pad this sketch by 2 mm and rename this as Clip.

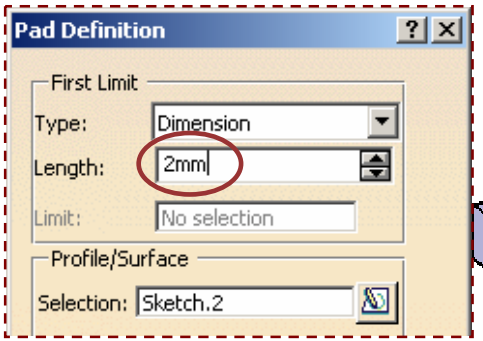


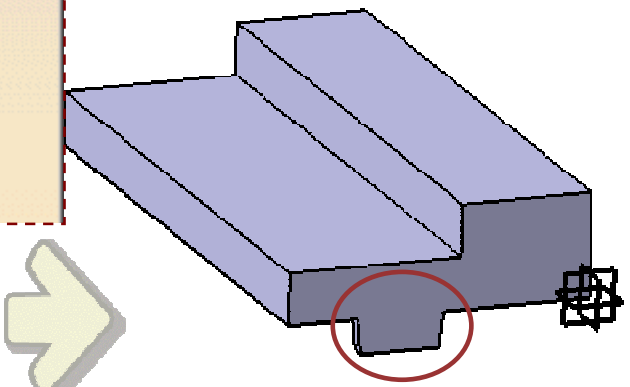


The details of the sketch are:



These lines are symmetrical about plane.1

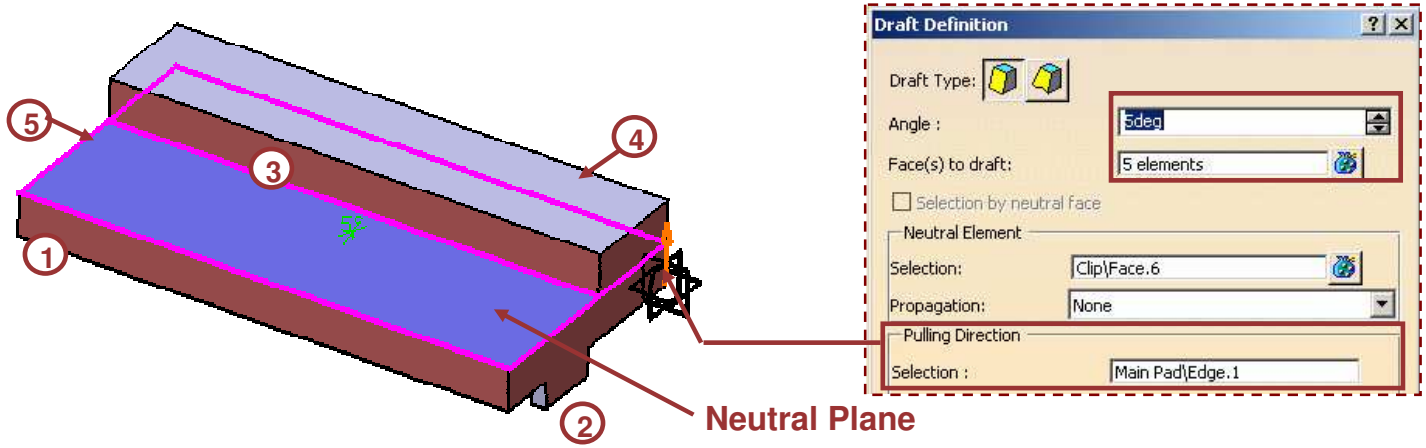




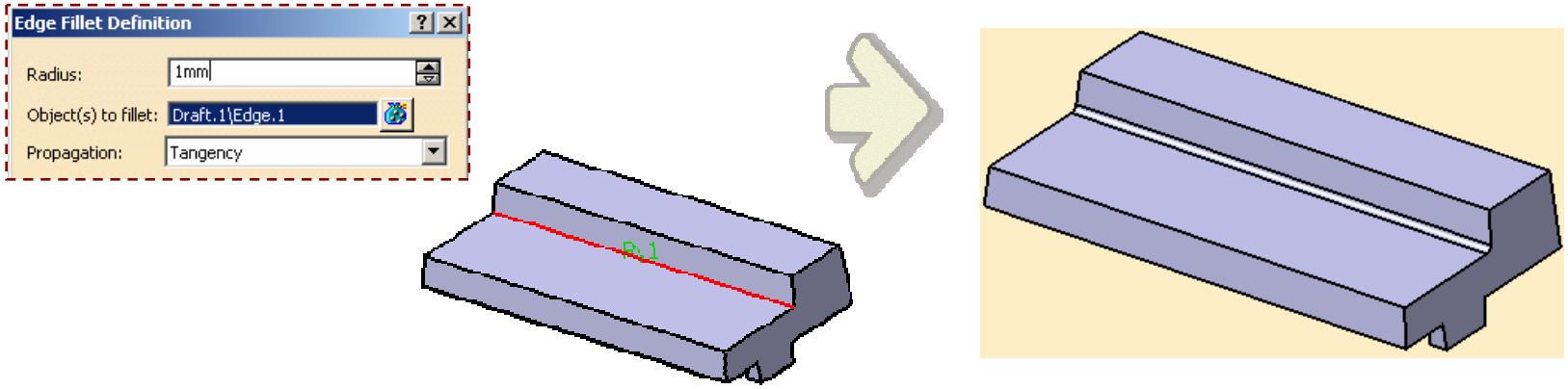
Student Notes:

Design the Clip (2/3)

- Apply Draft of 5 deg to five faces shown using Dress-Up features toolbar.



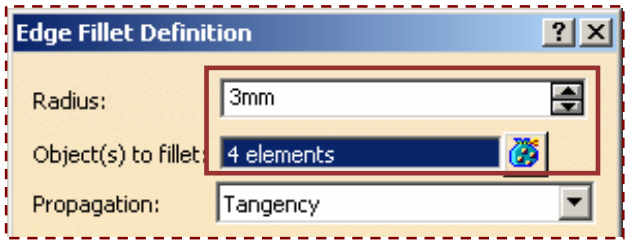
- Apply Edge fillet of 1 mm to the selected edge.

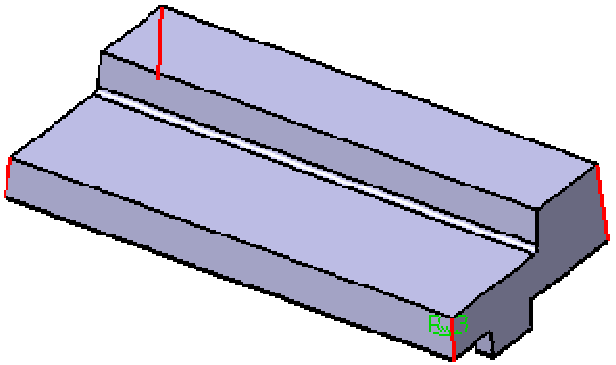


Student Notes:

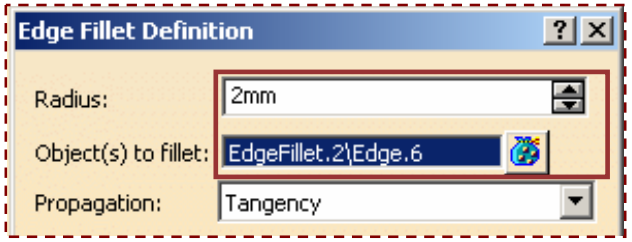
Design the Clip (3/3)

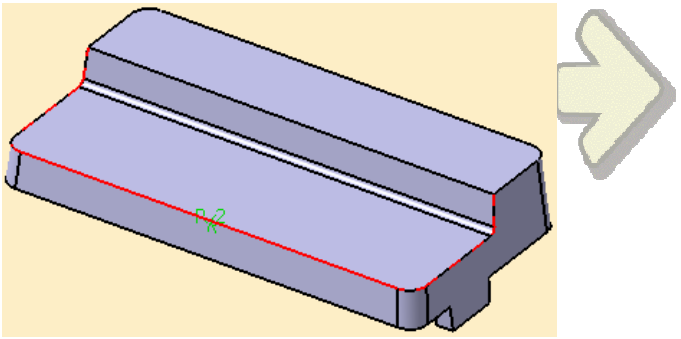
- Apply edge Fillet of 3 mm to the four edges.

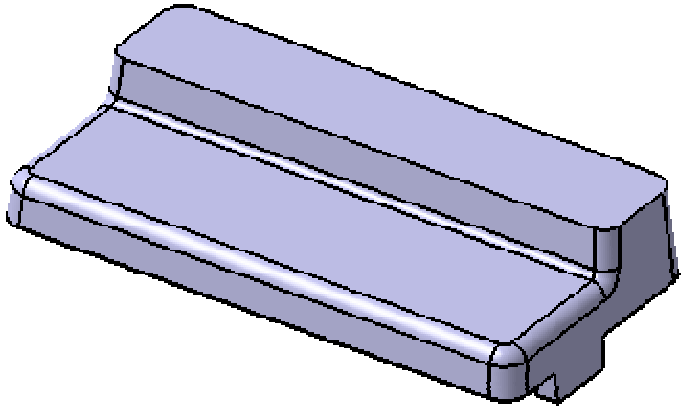




- Apply fillet of 2 mm to the edge shown.





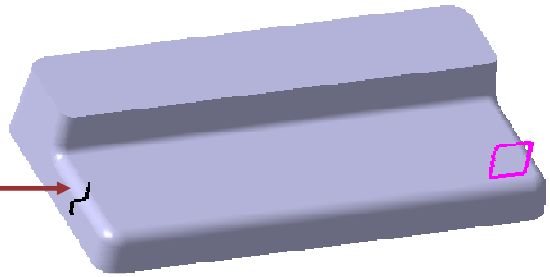


Student Notes:

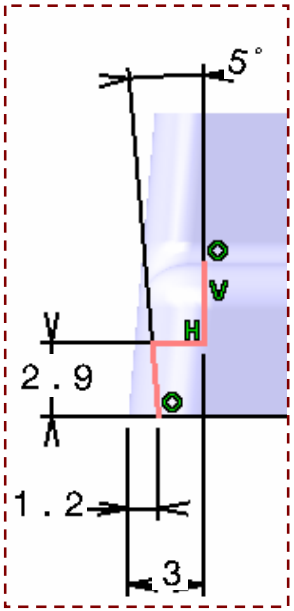
Design the Detent Pocket and Catch (1/3)

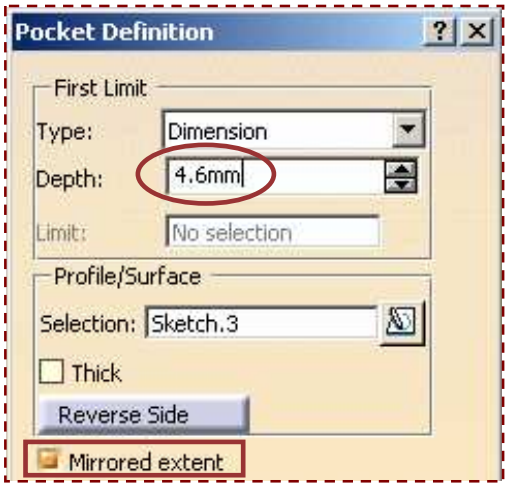
- Create a sketch to design the Detent pocket on plane.1.
- Create a Pocket of length 4.6 mm (Use Mirrored Extent).
- Rename this pocket as Detent Pocket.

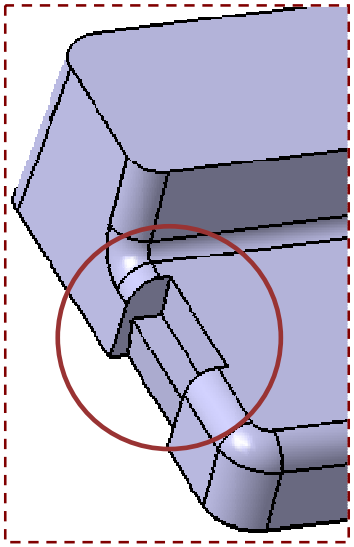
Sketch for Detent



Sketch Details



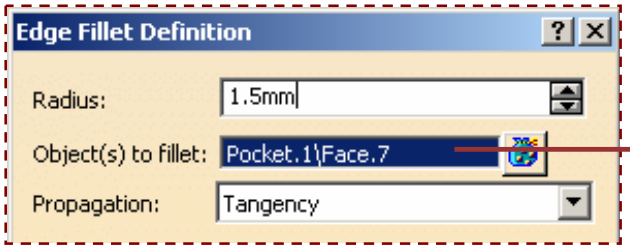


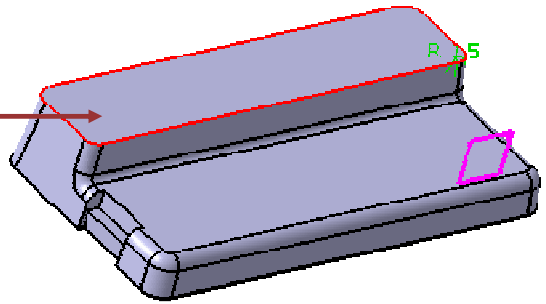


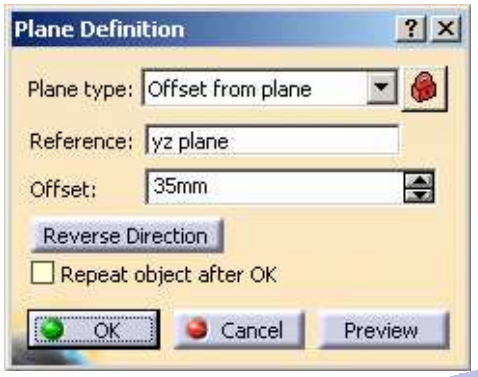
Student Notes:

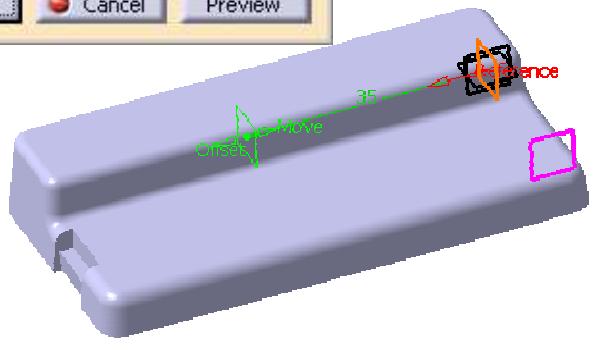
Design the Detent Pocket and Catch (2/3)

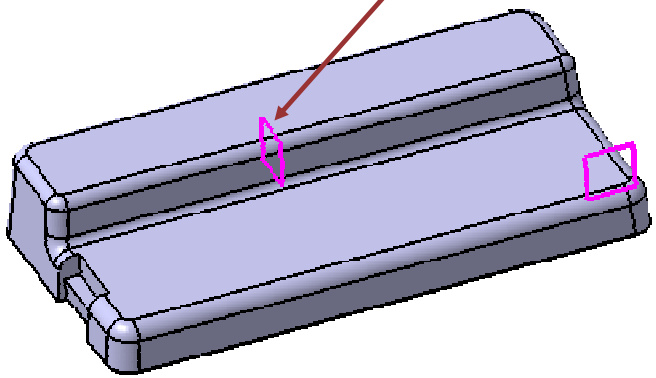
- Apply a Edge Fillet of 1.5 mm on the Top face of the housing.
- Offset a Plane from YZ plane at a distance of 35 mm. This is Plane.2.







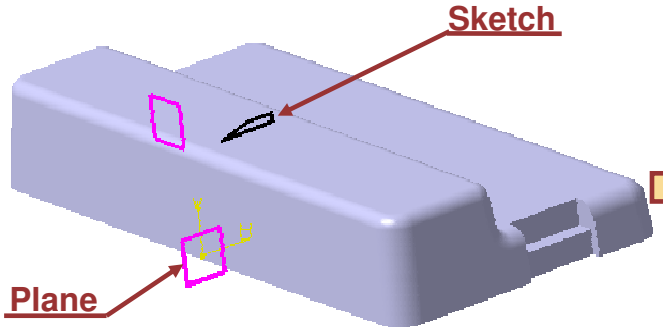




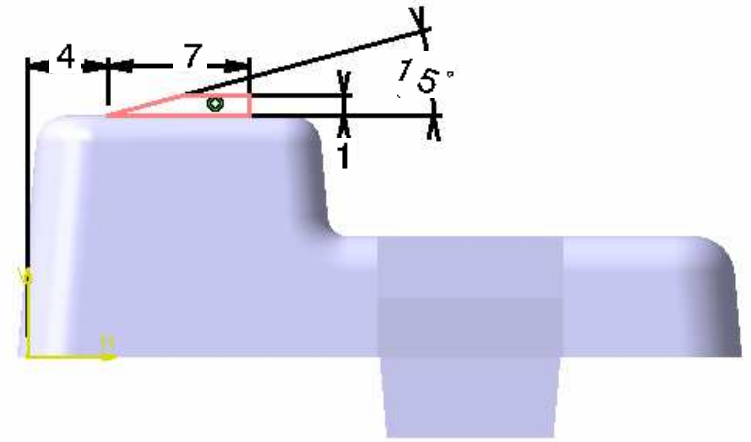
Student Notes:

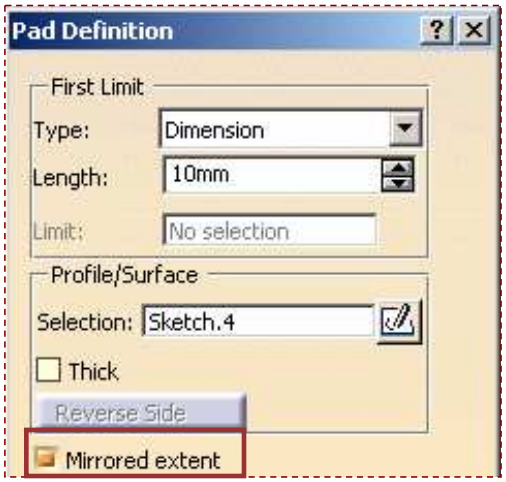
Design the Detent Pocket and Catch (3/3)

- On this plane, create a sketch for the catch.
- Pad the sketch by 10 mm (Mirror extent) to create the Catch. Rename the Pad to 'Catch'.

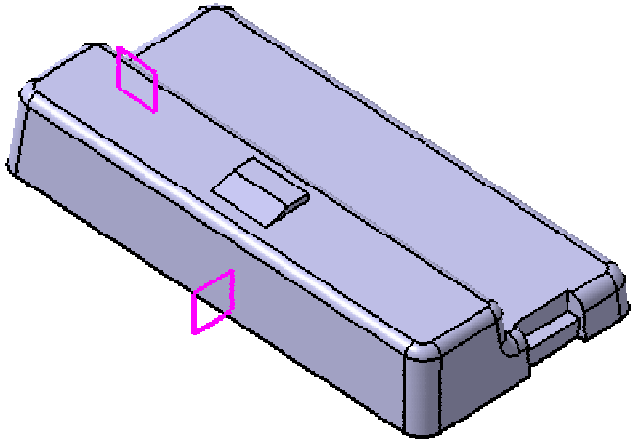


Details are





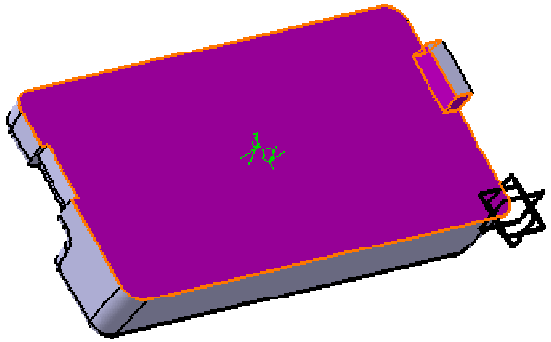
Pad the sketch using the following parameters

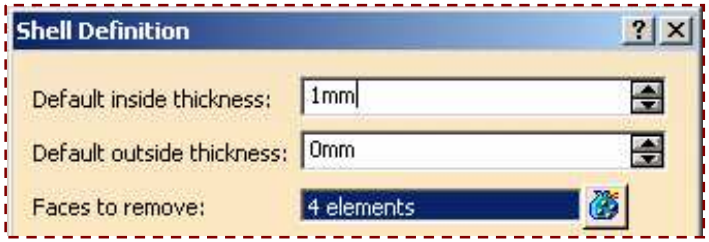


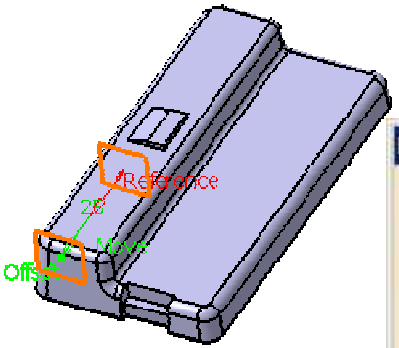
Student Notes:

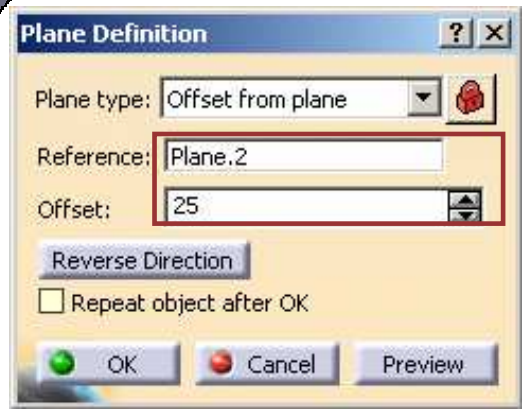
Create Shell and Design the Holes (1/4)

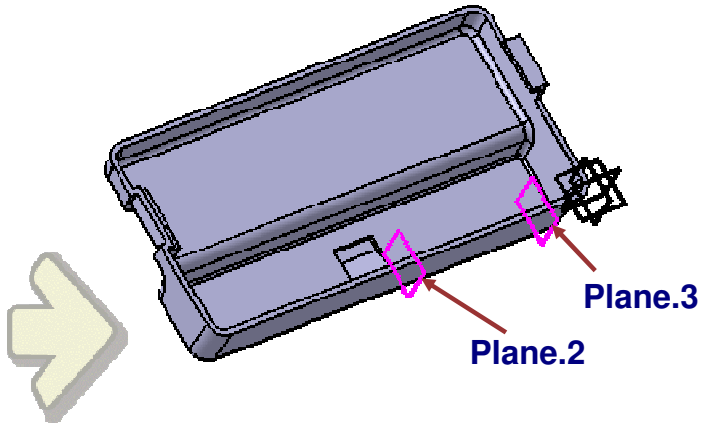
- Shell the bottom face of the part to get uniform wall thickness of 1 mm.
- Offset a Plane at 15 mm form Plane.2. This is Plane.3.







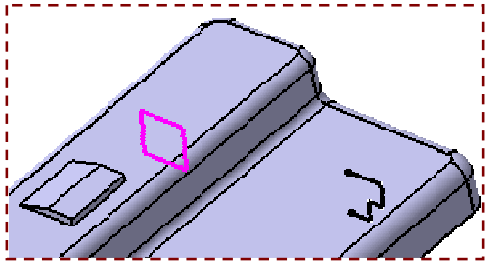




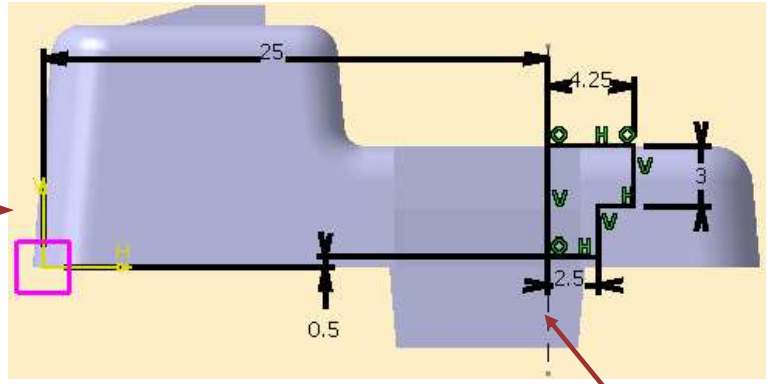
Student Notes:

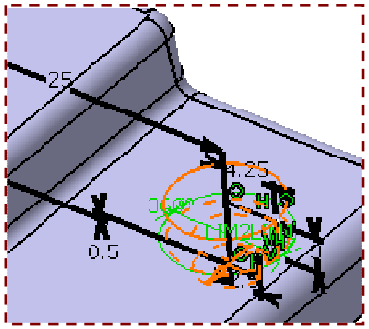
Create Shell and Design the Holes (2/4)

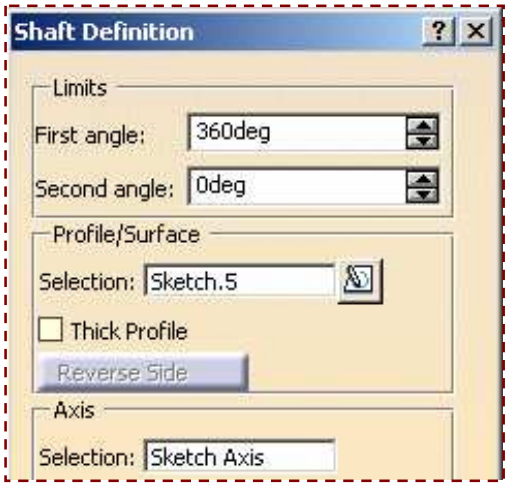
- Sketch on Plane.3 as shown.
- Revolve the sketch to create shaft.

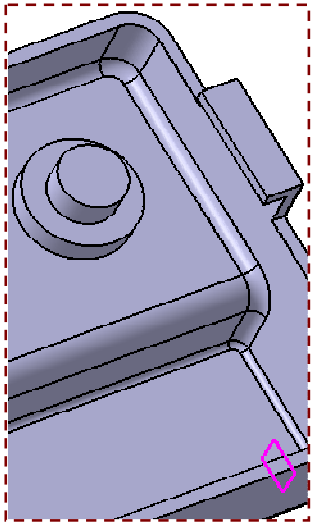


Sketch details





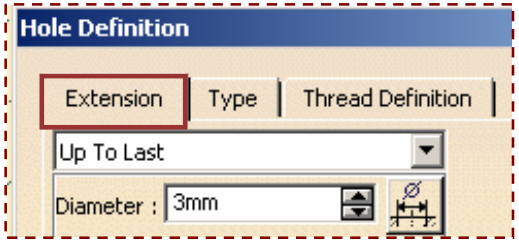


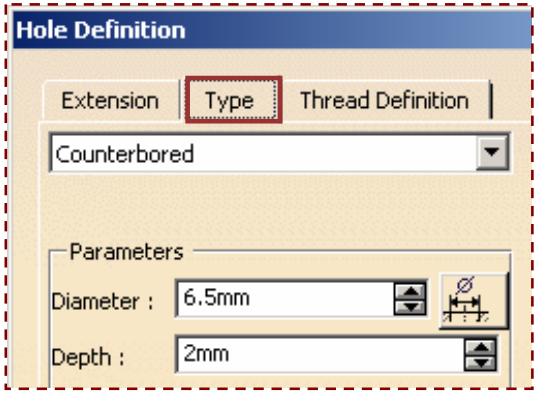


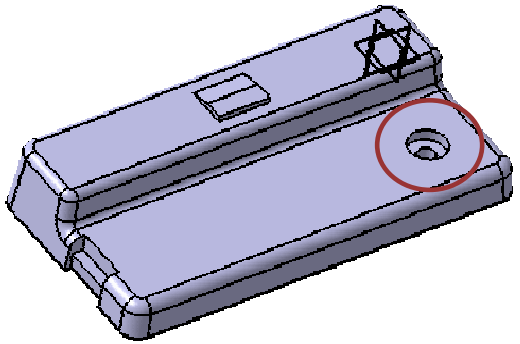
Student Notes:

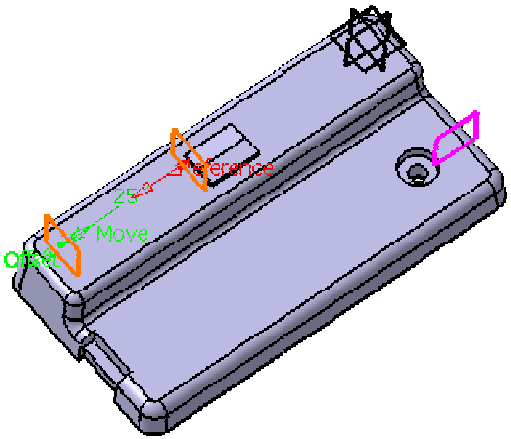
Create Shell and Design the Holes (3/4)

- Create a Counter bored hole of 3 mm diameter, concentric to the shaft.
- Create a Plane.4 offset from Plane.2 at a distance of 25 mm.





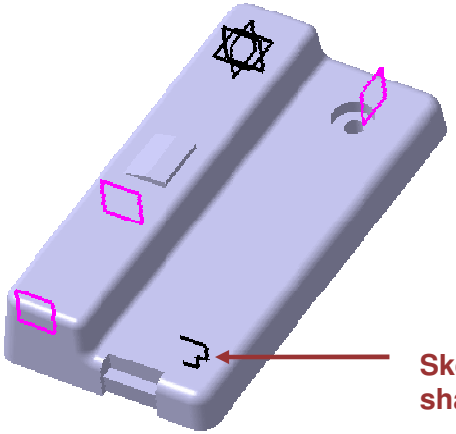




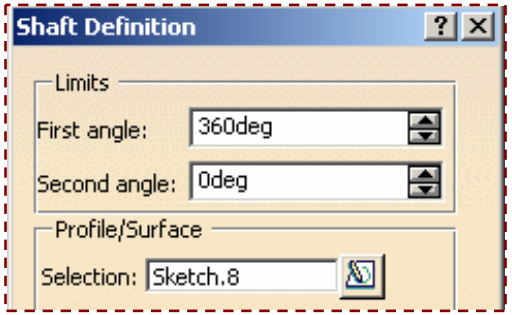
Student Notes:

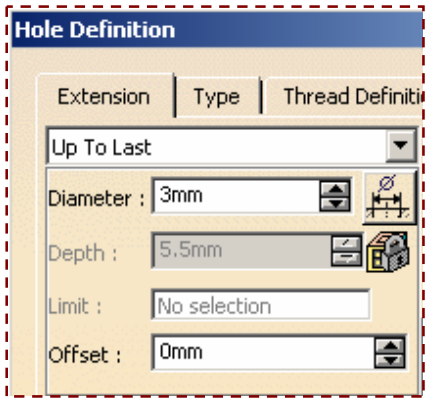
Create Shell and Design the Holes (4/4)

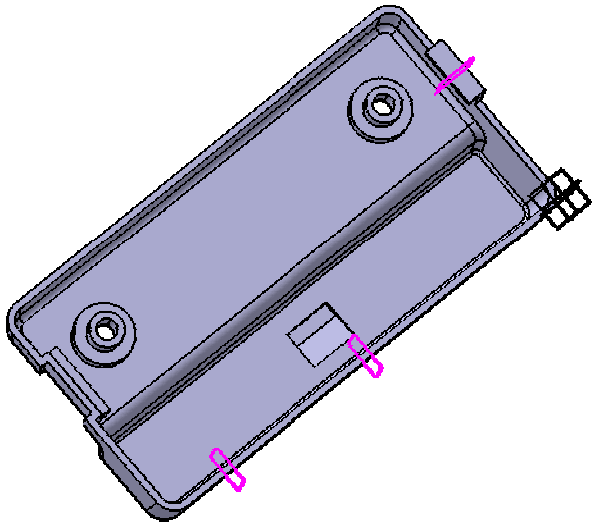
- Create a sketch similar to that of shaft.1 to create shaft.2.
- Create hole.2 same specifications as in hole.1 on shaft.2.



Sketch for shaft.2







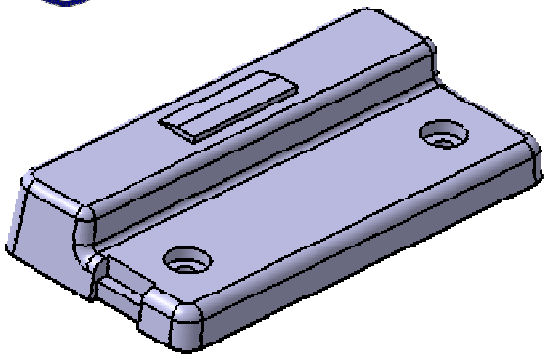
Result of step 2: PDG_Master_Connector_Housing_Step_2_end

Student Notes:

Connector Housing

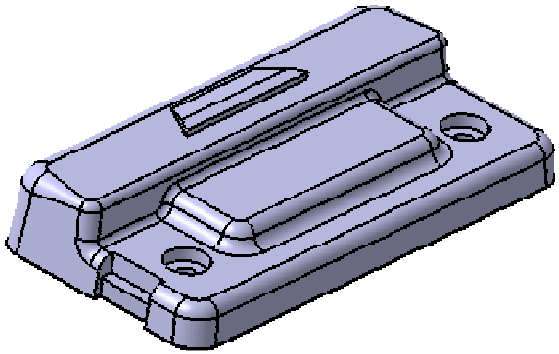
Step 3: Receive and get familiar with the modification request

20 min



Part Designed according to specifications

Modification request



Part expected after Modifications

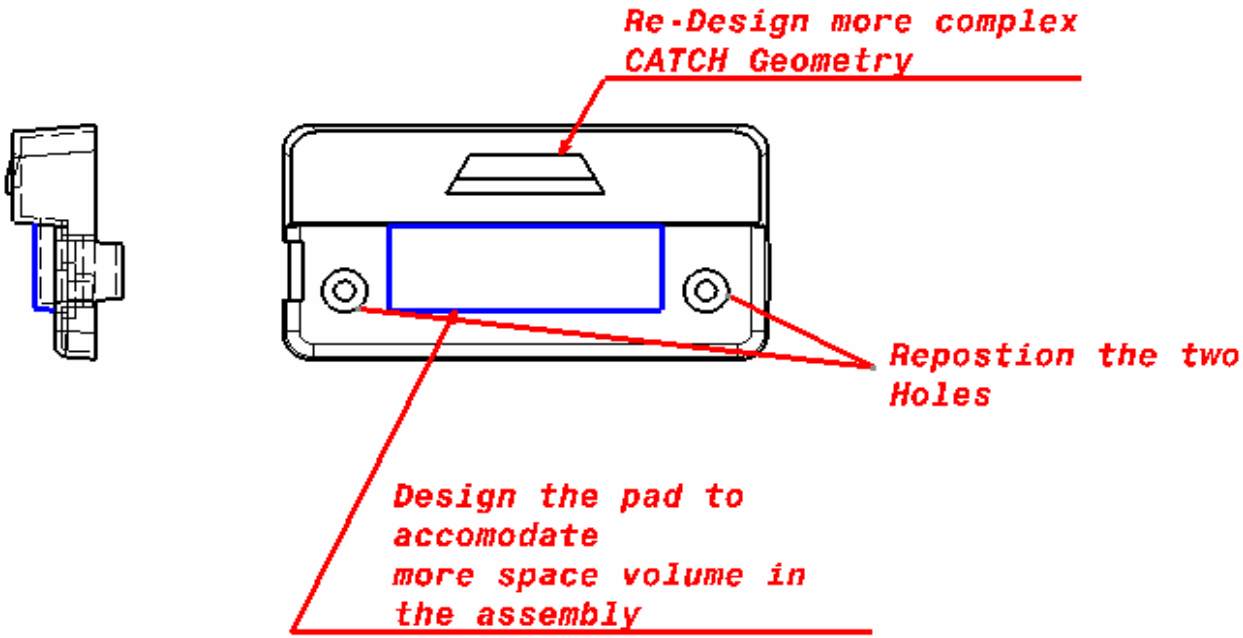
In this step you will understand what changes are required to be done in order to meet functional requirements.

Here you will :

- **Thoroughly understand the design changes by studying the drawing.**
- **Understand why these modifications are done.**
- **Study which features are affected due to above changes.**
- **Propose if any new modifications are required.**
- **Modify the parameters for dress up features.**

Student Notes:

Do It Yourself



- Study and understand the modifications required.
- The design for “Catch” is changed. It is now more complex.
- Design a completely new feature to allocate more space.
- Reposition the two holes to accommodate other features.

Student Notes:

Connector Housing

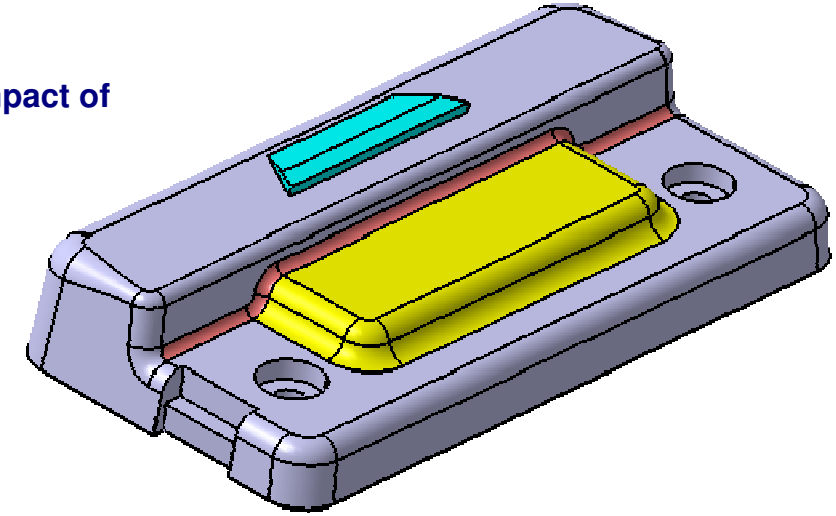
Step 4: Modify the design following the modification request

20 min

In this step you will create the Part in accordance with the modification request you have received.

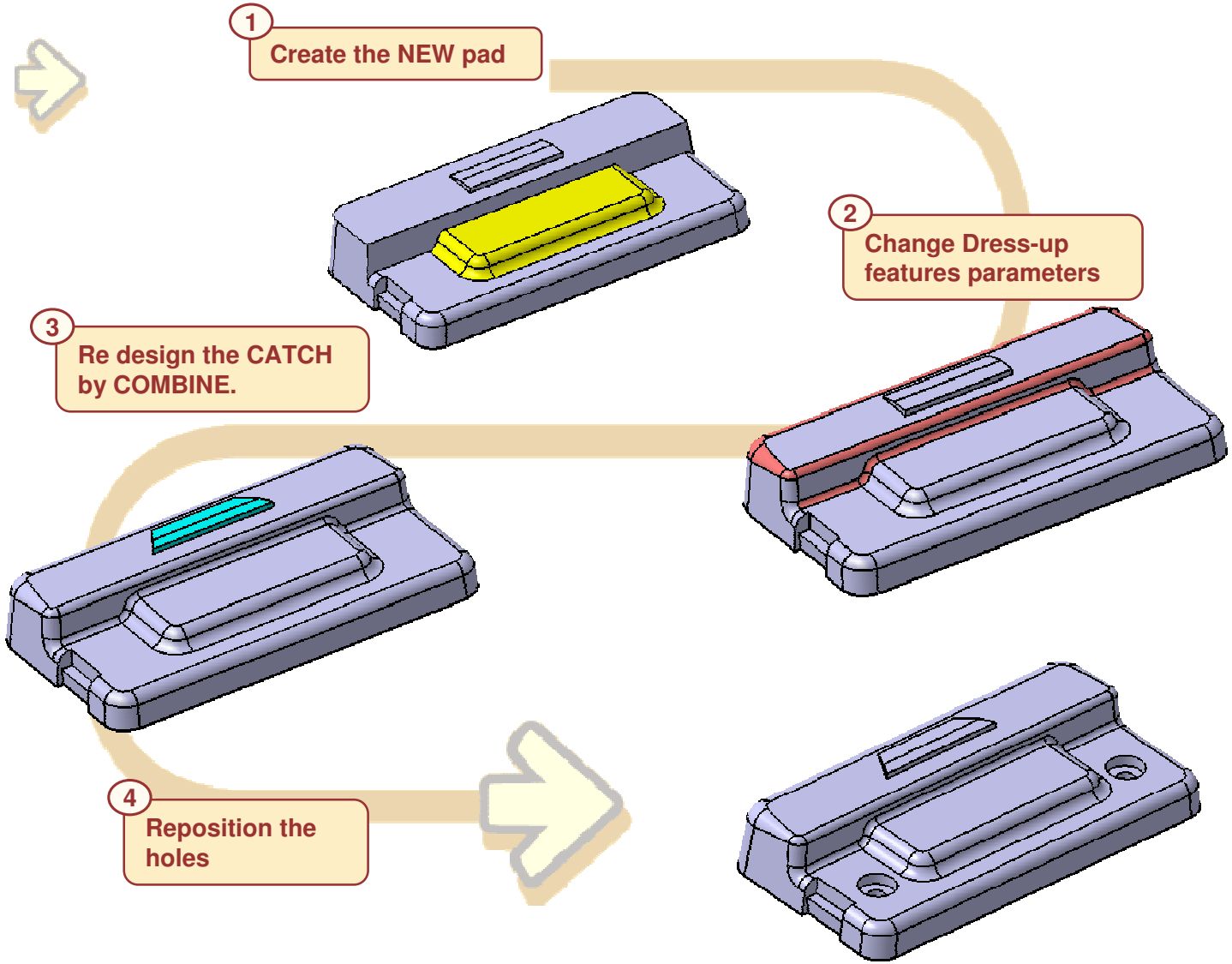
You will:

- **Design the Pad to allocate more space**
- **Re-design the CATCH geometry**
- **Reposition the holes, to account for the impact of above changes**
- **Modify the dress-up features**



Student Notes:

Step 4: Process

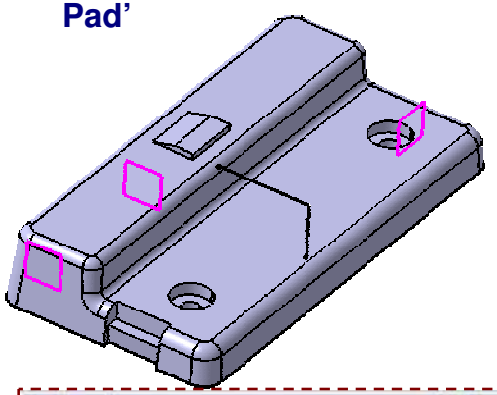


Student Notes:

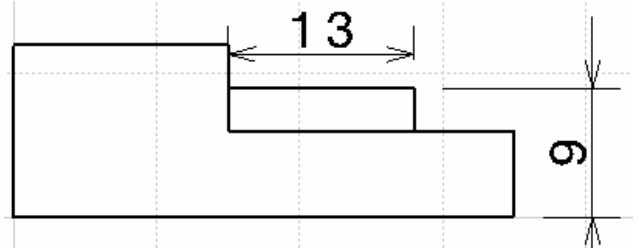
Design the New Pad

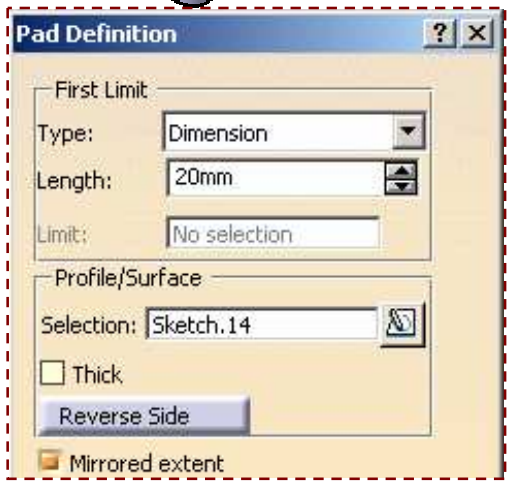
'PDG_Master_Connector_Housing_Step_4_Start.CATPart'

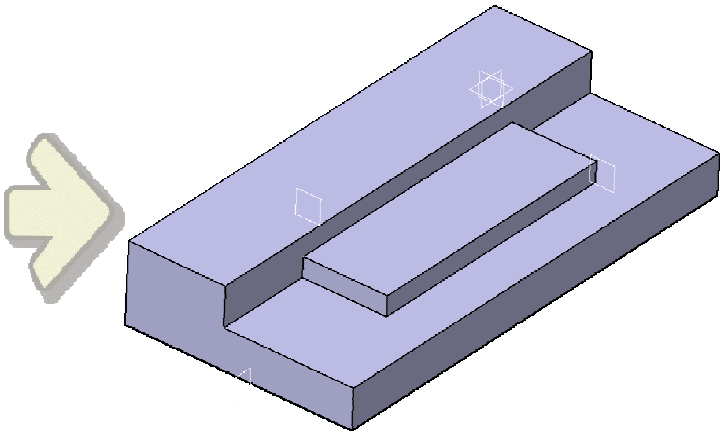
- Create a sketch to generate a Pad on Plane.2. Create it just after the Main Pad and before the Clip. Use 'Define in work object' functionality.
- Create a Pad of length 20mm. Select the 'Mirror extent' option. Name the result as 'New Pad'

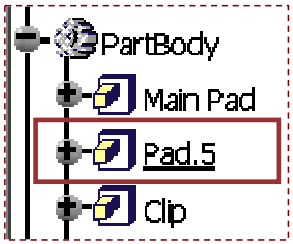


Details





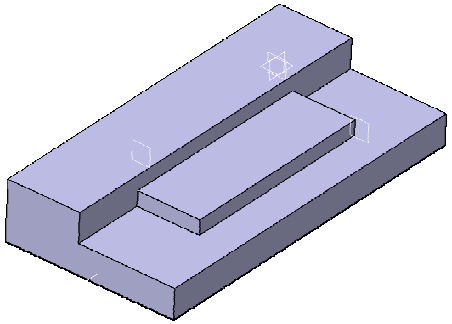


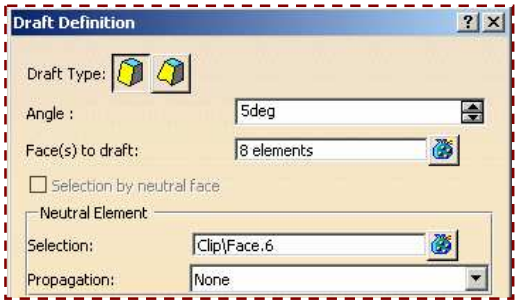


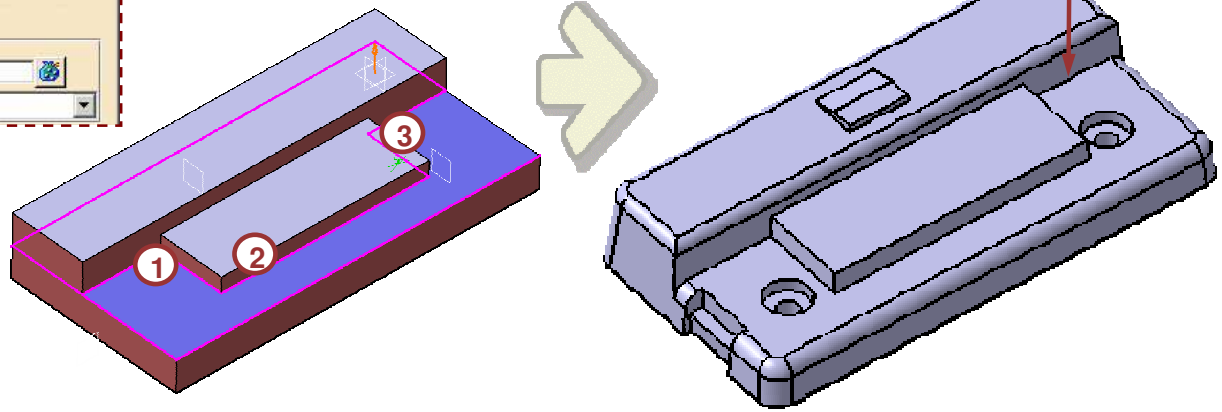
Student Notes:

Change the Dress-up feature Parameters (1/4)

- A warning is displayed, then you need to modify the indicated fillet. Delete it for the time being. You will recreate it taking into account the 'New Pad'.
- Edit the draft definition, and select three new faces of the 'New Pad'.



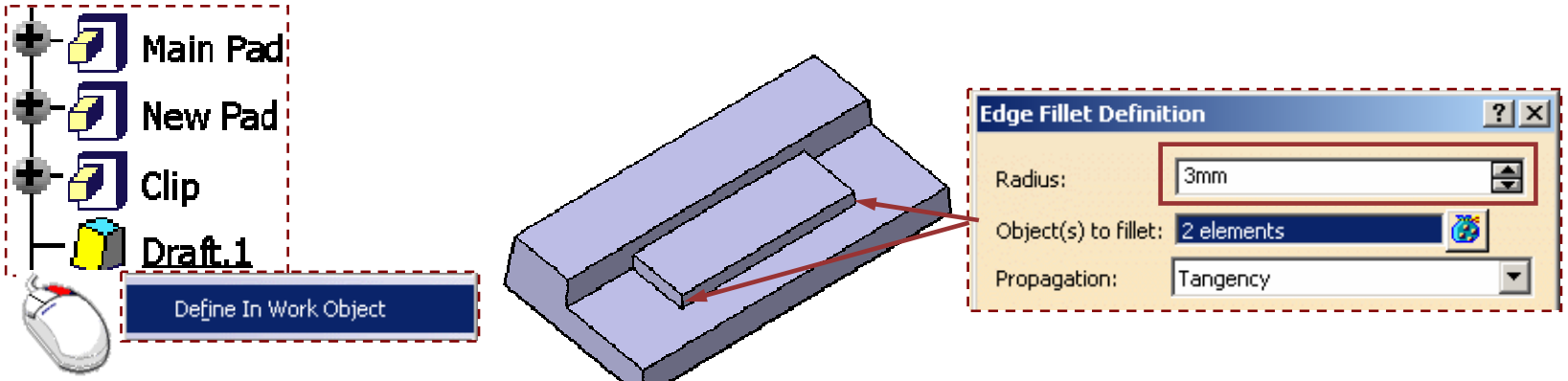




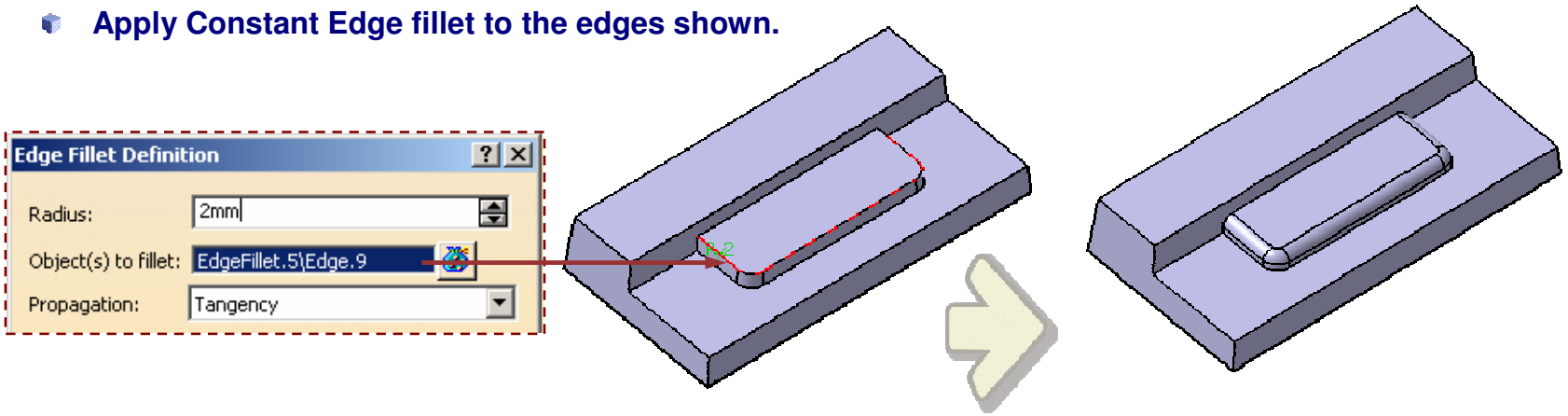
Student Notes:

Change the Dress-up feature Parameters (2/4)

- To apply fillets to the Newly created Pad, define in work object on Draft.1. So the fillets will be positioned after this Draft.



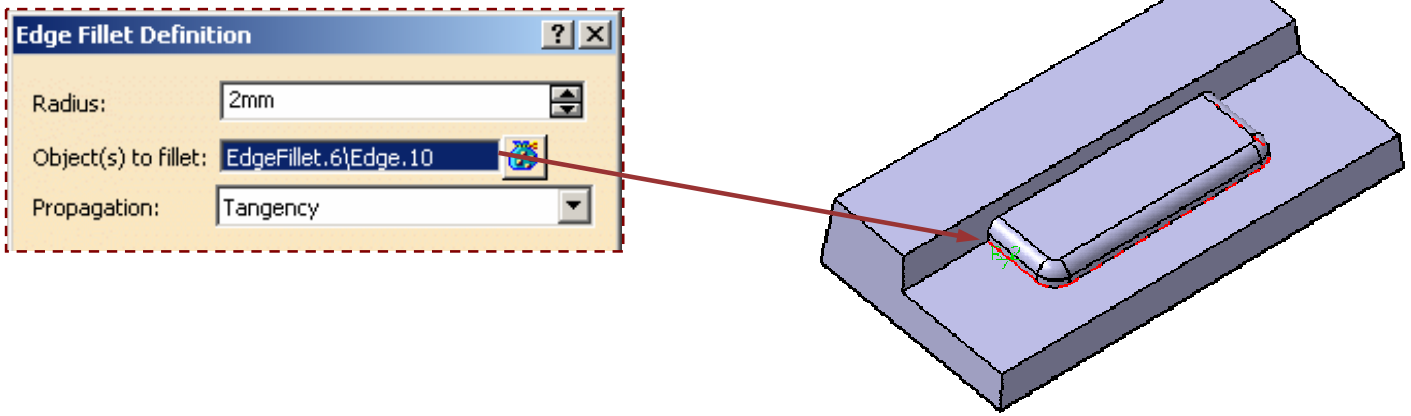
- Apply Constant Edge fillet to the edges shown.



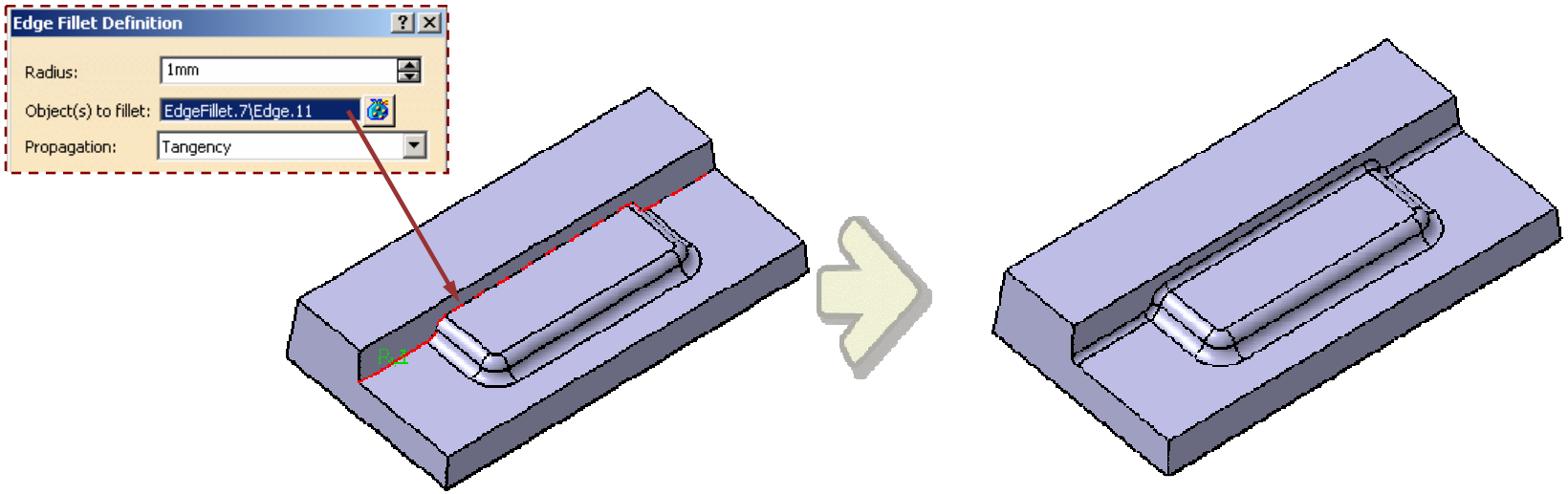
Student Notes:

Change the Dress-up feature Parameters (3/4)

- Apply fillet of 2 mm to the edge shown.



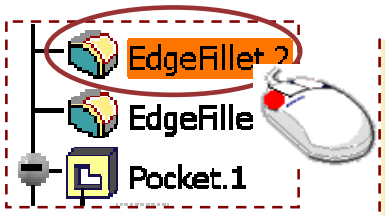
- Re-create the previously deleted fillet. Create a new fillet of 1 mm.

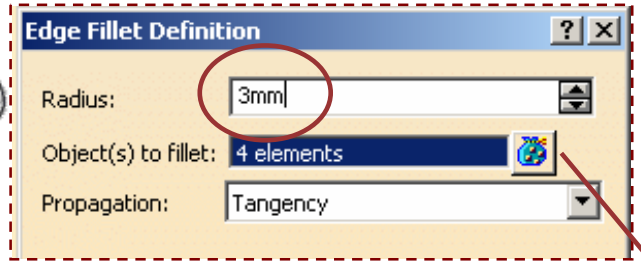


Student Notes:

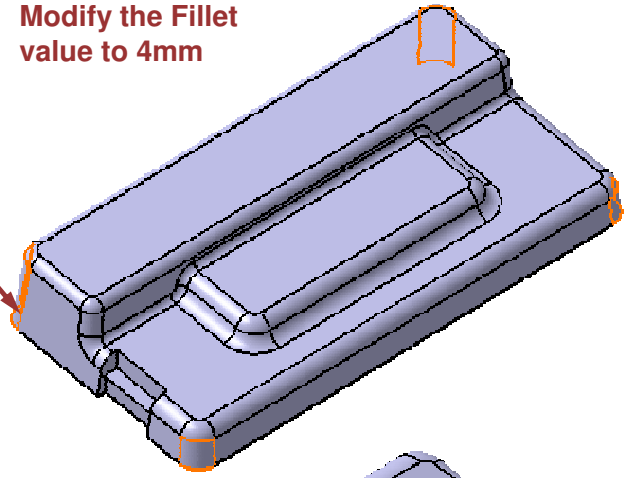
Change the Dress-up feature Parameters (4/4)

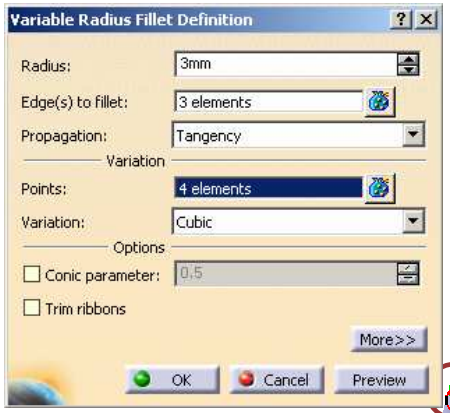
- Modify the value of previously created fillet from 3 mm to 4 mm.
- Change the Constant edge fillet on the top face to variable radius fillet with values varying from 1.2 mm to 3 mm.

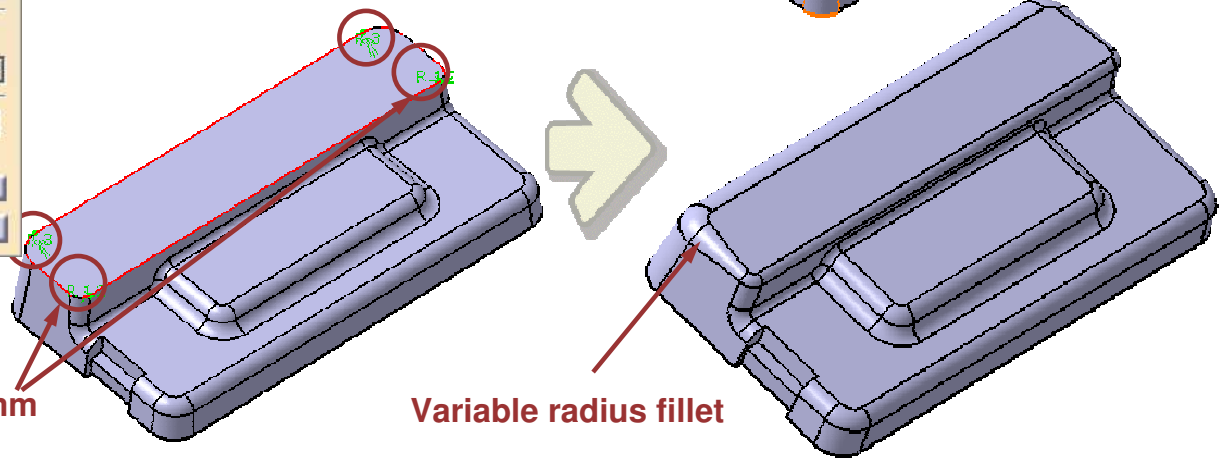




Modify the Fillet value to 4mm







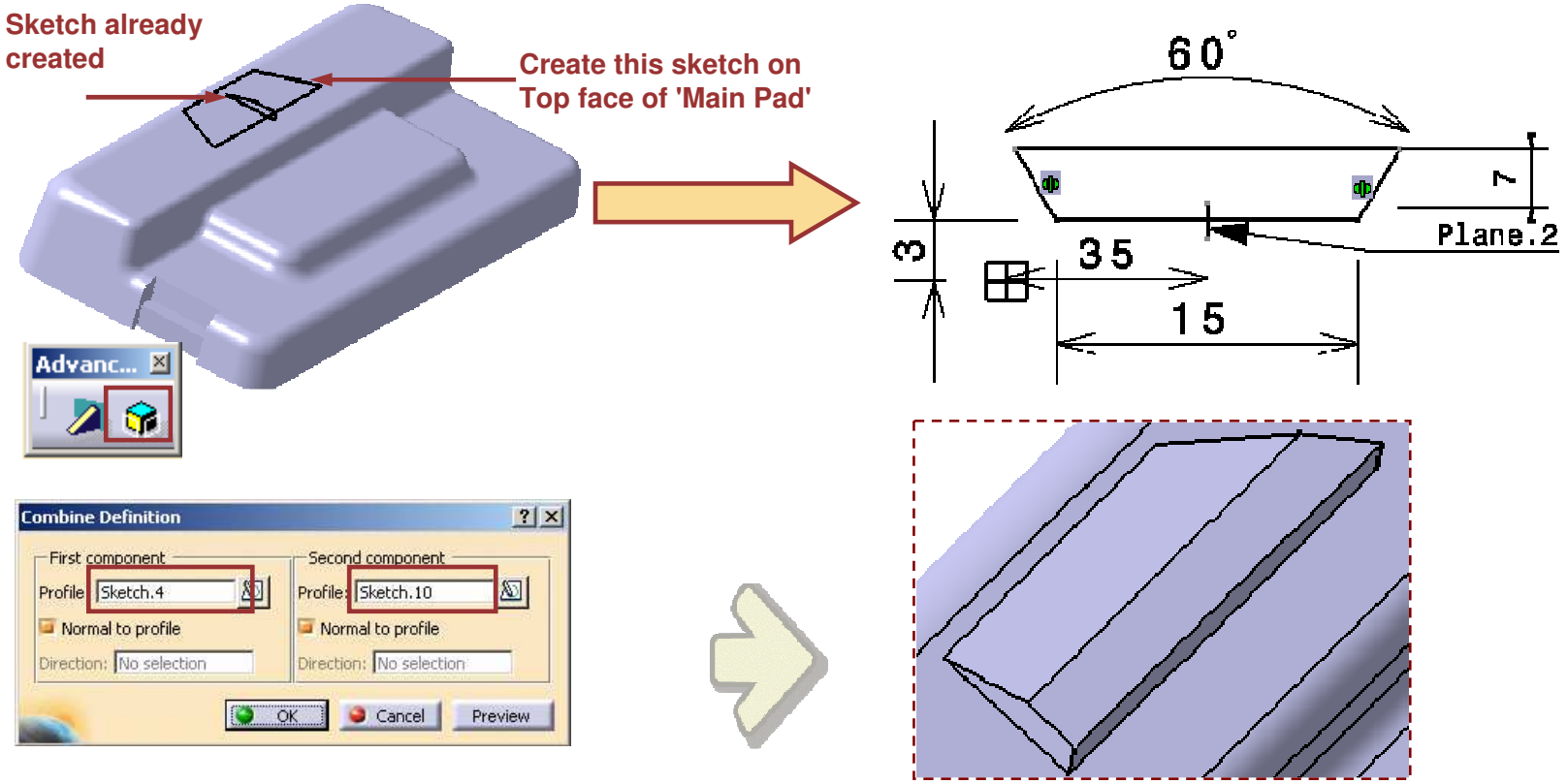
1.2 mm

Variable radius fillet

Student Notes:

Re-design the Catch by Combine

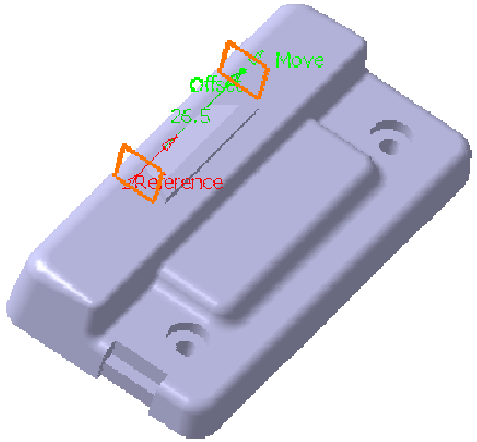
- Re-Design the Catch Geometry. Since the Design is modified, it is not easy to design the CATCH using Pad. So you will design it using Solid Combine.
- You will create it just after the Variable Radius Fillet. To create the Combine you need two sketches. Reuse the sketch (created before modification request). Create the second sketch on Top face of the 'Main Pad'.

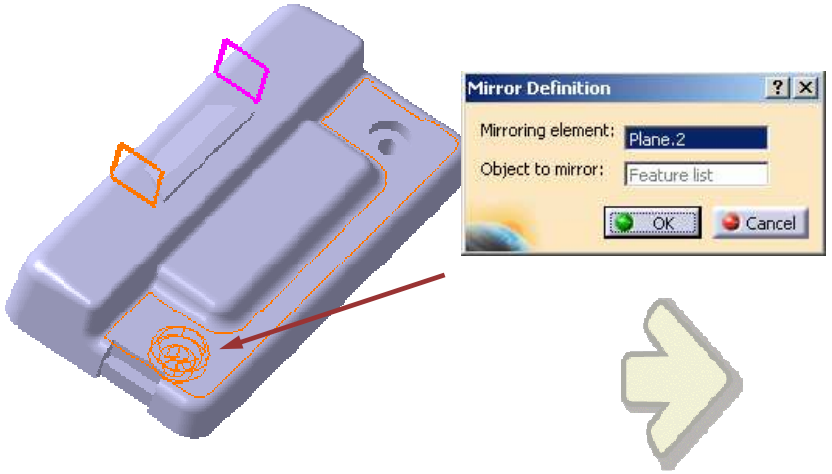


Student Notes:

Reposition the Holes

- Modify the position of Plane.3 from 25 mm to 26.5 mm.
- Delete Shaft.2 and Hole.2. Mirror Shaft.1 and Hole.1 about plane.2





Before Modification

After Modification

Load: 'PDG_Connector_Housing_Step_4_end.CATPart'

EXERCISE BOOK

Student Notes:

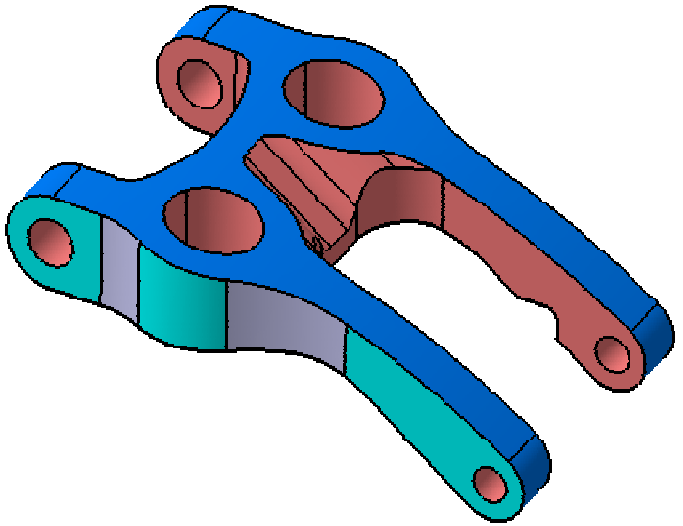
Bike Rear Lever

Part Design Fundamental Exercise

60 min

In this exercise you will :

- Use the dimensions given in the drawing
- Proceed in order to respect the specifications

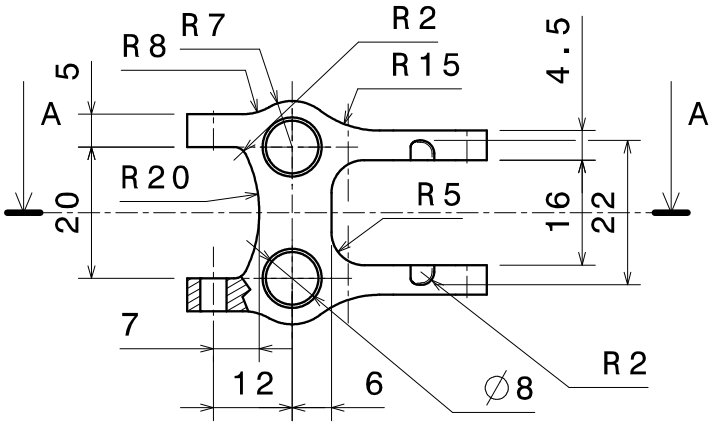


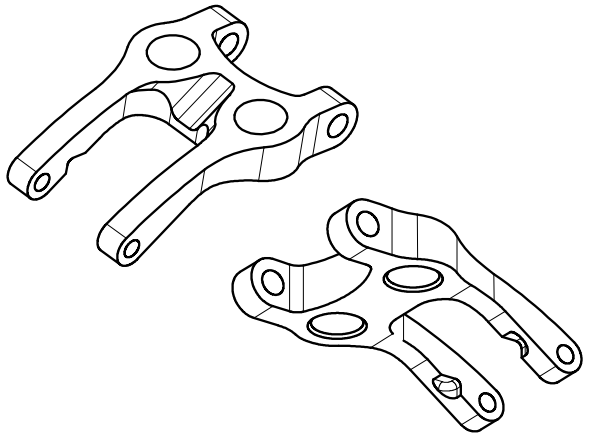
In order to understand the design intent, a 3D result without history is provided.

Bike_Rear_Lever_Result.CATPart

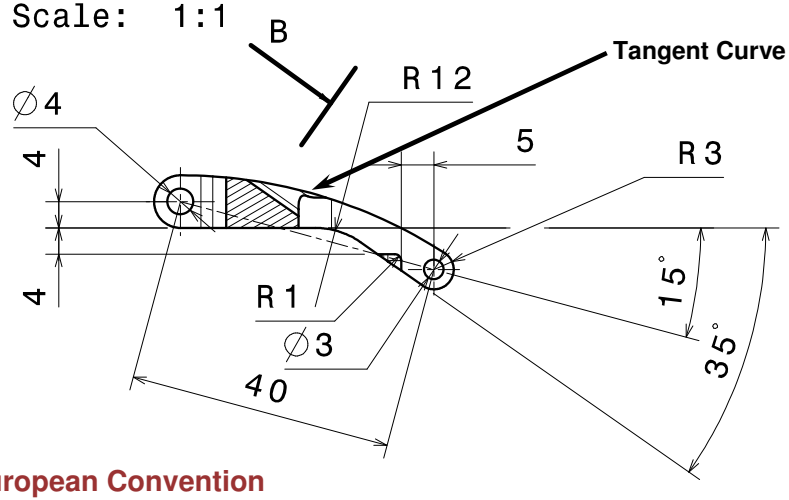
Student Notes:

Bike Rear Lever Drawing

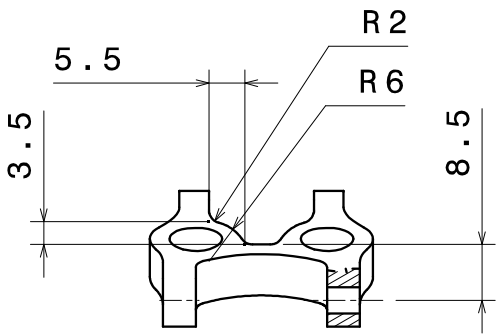




Section view A-A
Scale: 1:1



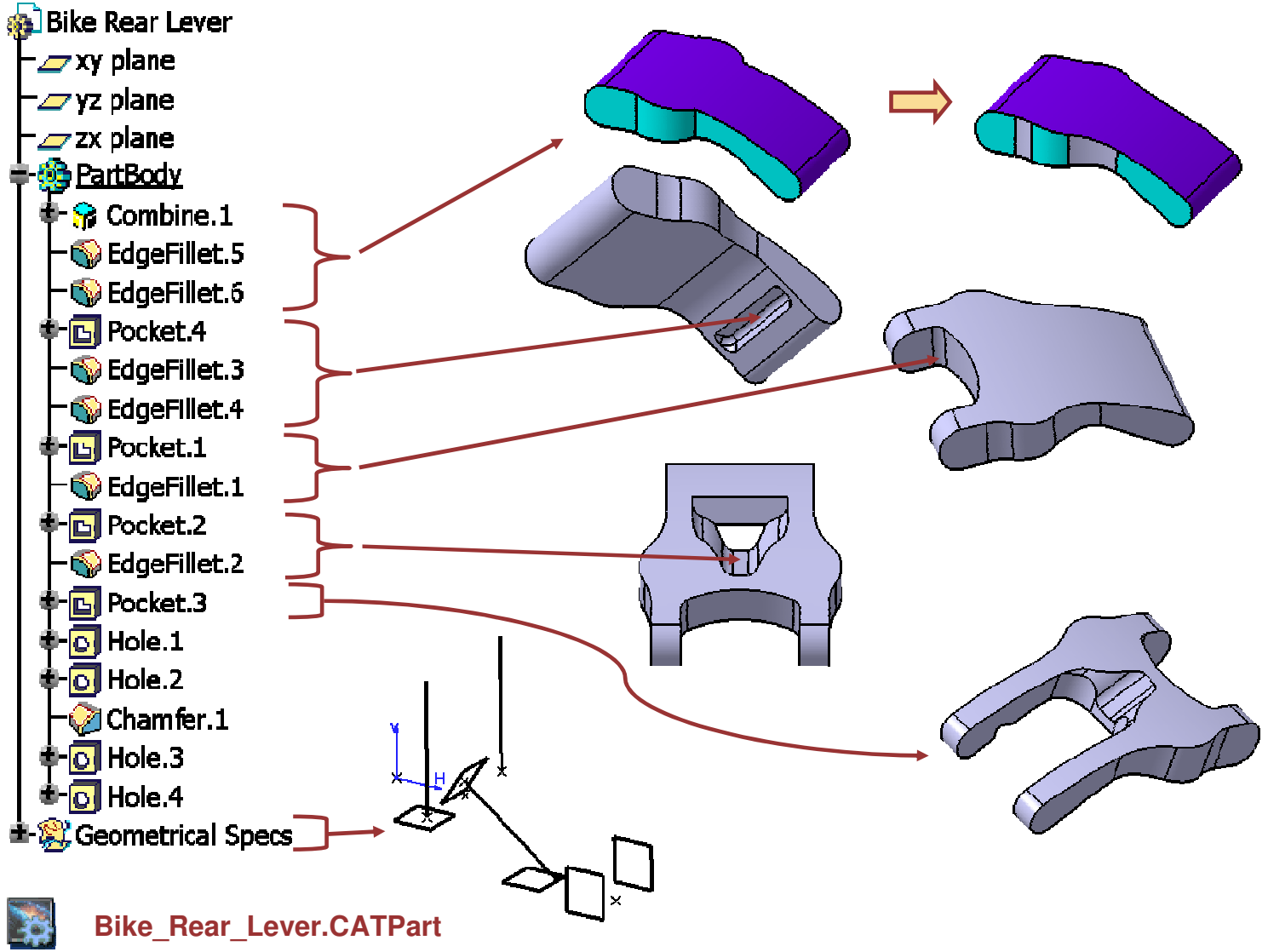
Auxiliary View B



European Convention
Chamfers = 0.5*45Deg

Student Notes:

Specifications: Bike Rear Lever



Bike_Rear_Lever.CATPart

EXERCISE BOOK

Student Notes:

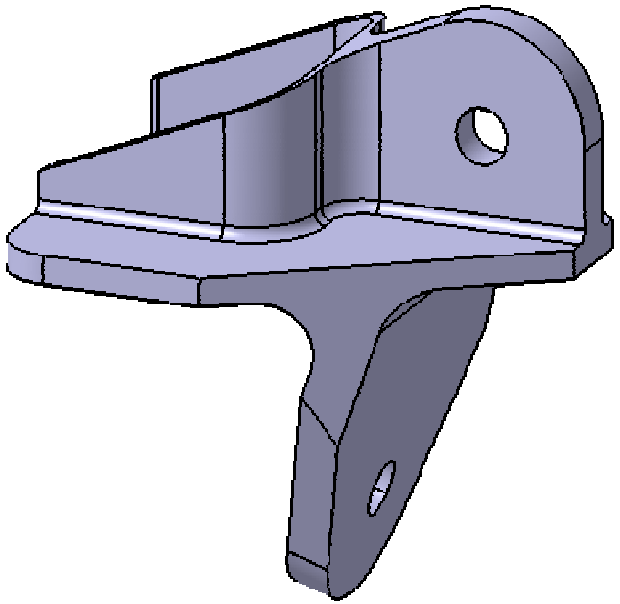
Fitting

Part Design Fundamental Exercise

60 min

In this exercise you will:

- Use the dimensions given in the drawing
- Proceed in order to respect the specifications

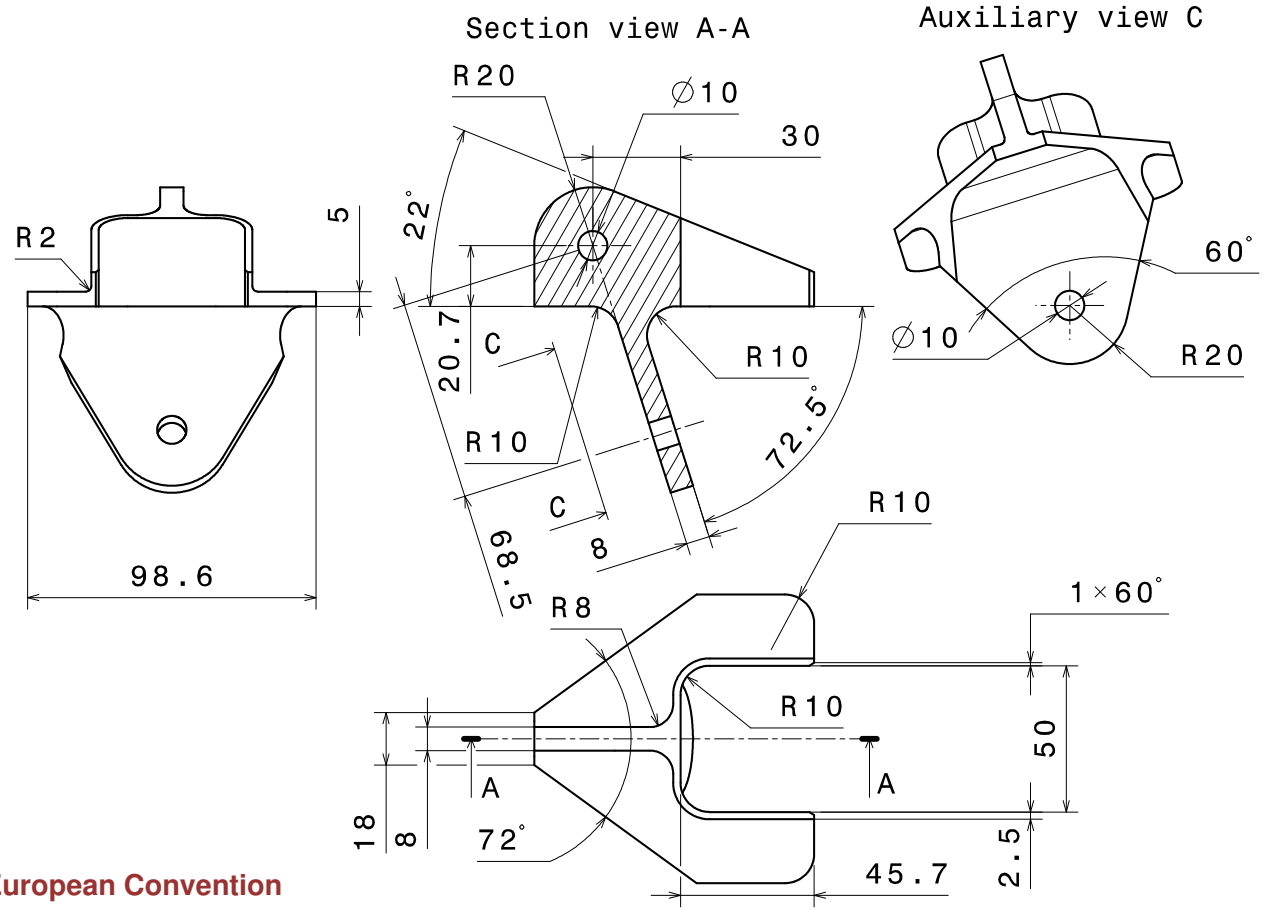


A 3D result without history is provided, in case you need the expected final geometry, in order to better understand the design intent by manipulating the part in CATIA:

Fitting_Result.CATPart

Student Notes:

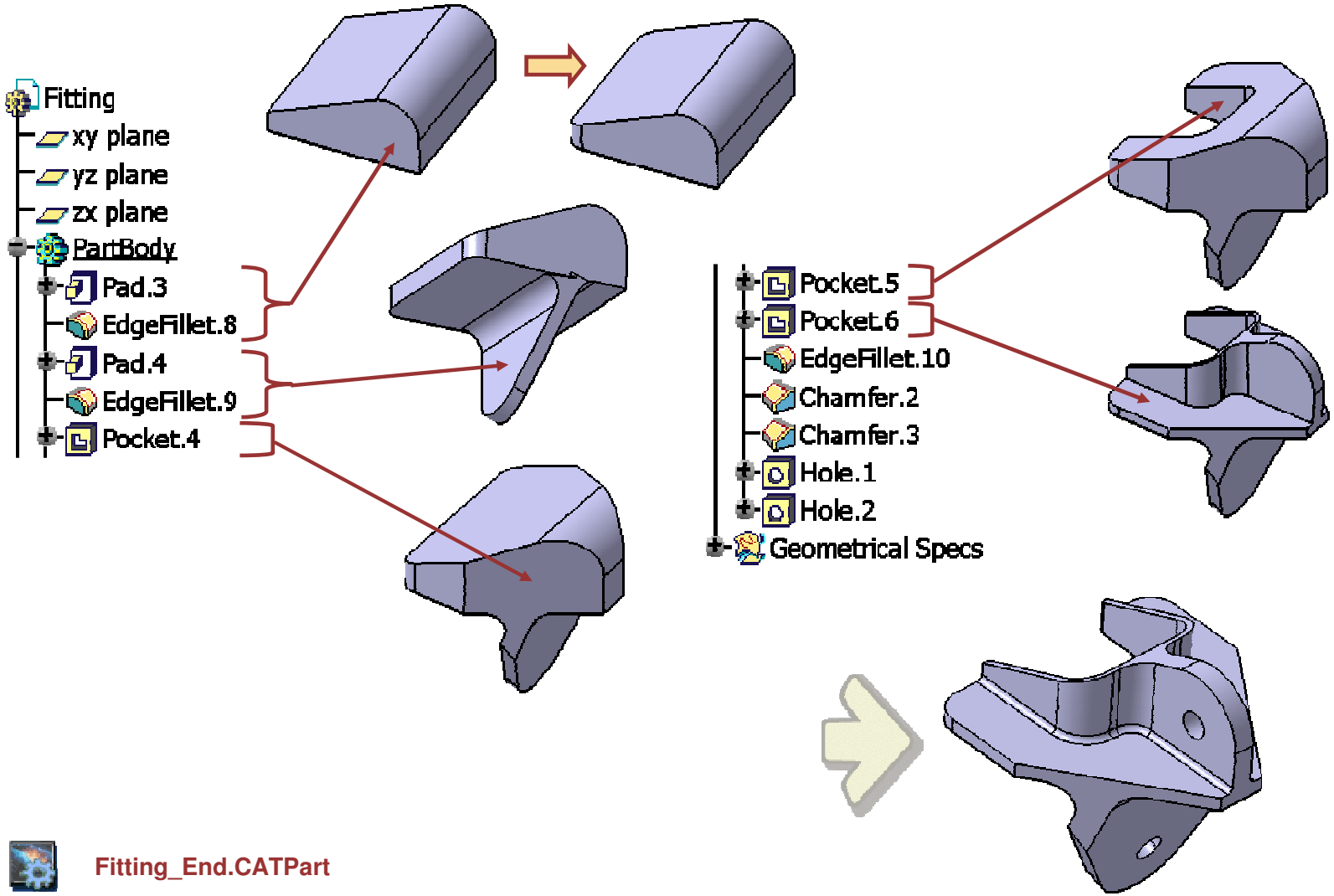
Fitting Drawing



European Convention

Student Notes:

Specifications: Fitting



Fitting_End.CATPart

Student Notes:

Master Exercise: Bottom Case

You will practice concepts learned throughout the course by building the master exercise and following the recommended process.

- Mobile Phone Bottom Case Presentation
- Mobile Phone Bottom Case (1): Design the Battery Space
- Mobile Phone Bottom Case (2): Design the Bottom Case
- Mobile Phone Bottom Case (3): Modifying the Design
- Mobile Phone Bottom Case (4): Analyze the Design

Student Notes:

Master Exercise: Bottom Case

In this exercise, you will design a complete part using the Part Design Workbench.

Mobile Phone Bottom Case

Master Exercise Presentation

90 min

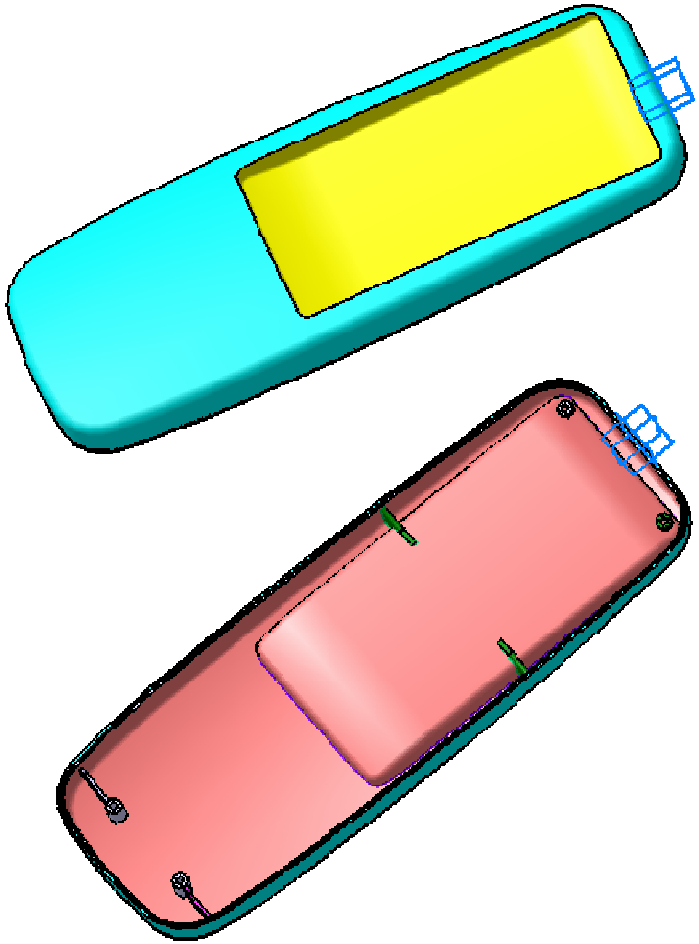
In this exercise you will build the bottom case of a mobile phone following the recommended design process:

Here you will first Design the battery reservation space

After that you will Design the Bottom case.

Then you will modify the design to study the impact of modification.

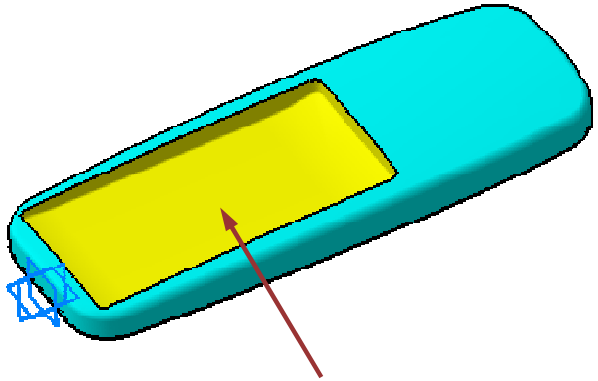
Finally, You will analyze the design and modify the part so that it can be manufactured.



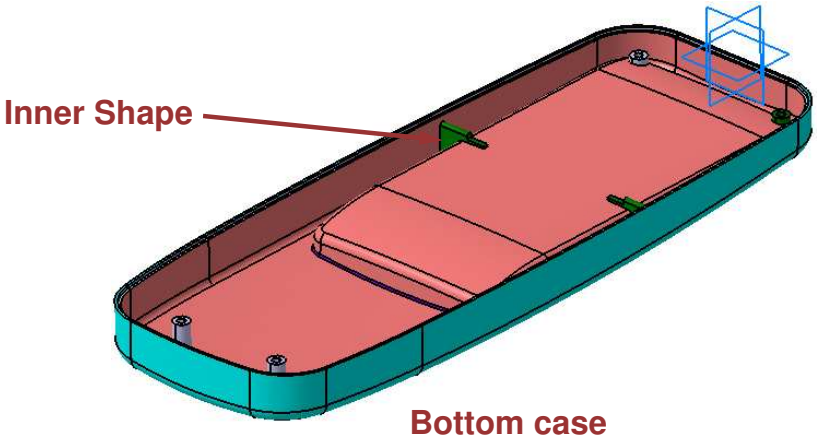
Student Notes:

Student Notes:

Design Intent:Mobile Phone Bottom case



Battery Reservation space



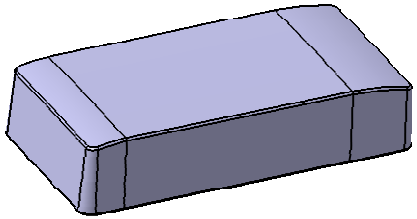
Inner Shape

Bottom case

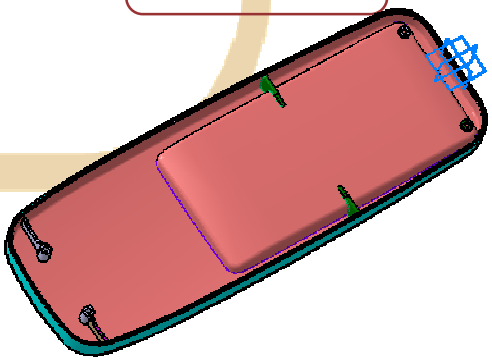
Student Notes:

Design process: Mobile Phone Bottom Case

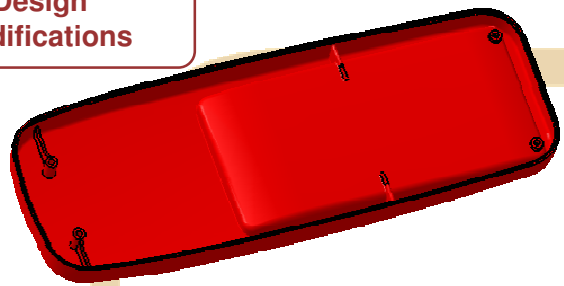
1 Design the battery Reservation Space



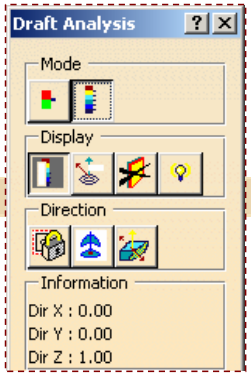
2 Design Of Bottom Case

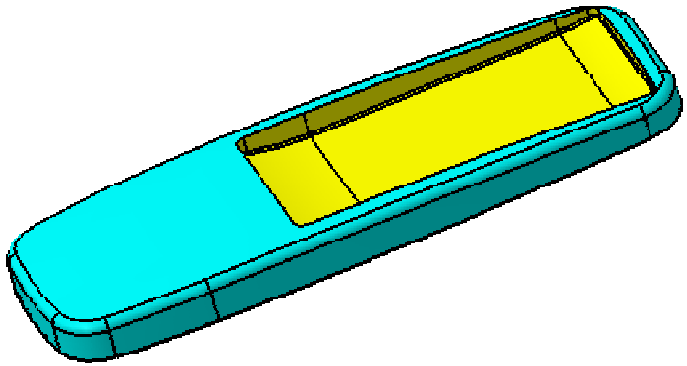


3 Design Modifications



4 Perform Draft analysis and Modify design





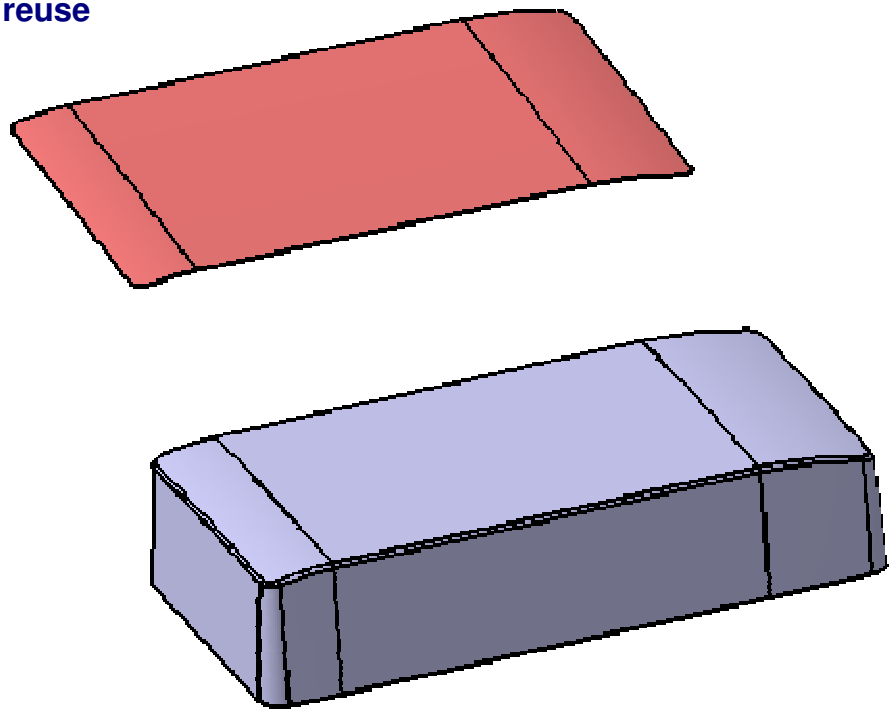
Mobile Phone Bottom Case

Step 1 – Design the Battery Reservation Space

10 min

In this step you will create the Part corresponding to the battery reservation volume in order to reuse this volume in the Part of the phone:

- **Create a Pad from input elements**
- **Apply Draft**
- **Apply Fillet**

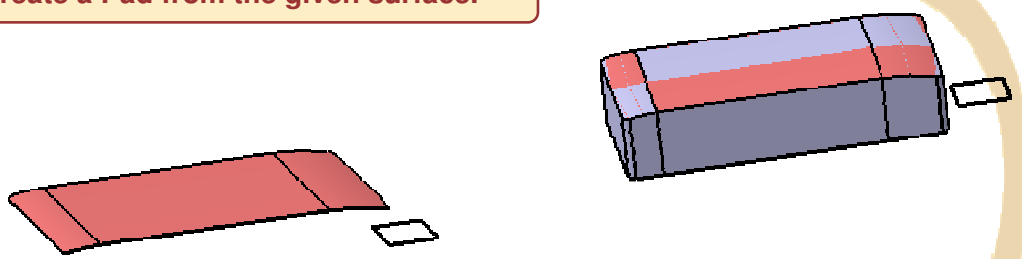


Student Notes:

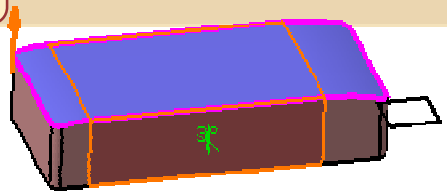
Student Notes:

Do It Yourself

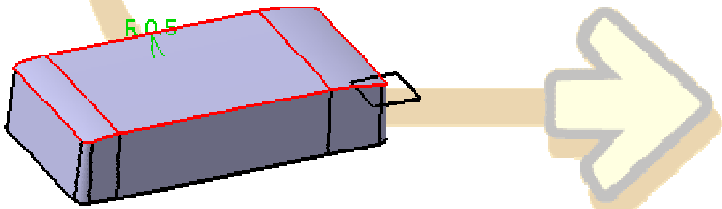
1. Create a Pad from the given surface.

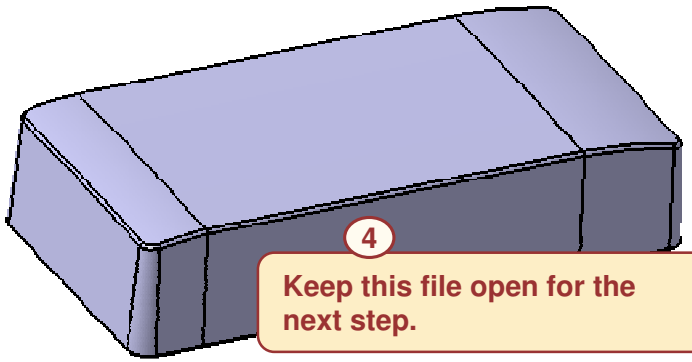


2. Apply draft to the faces.



3. Apply Edge Fillet.





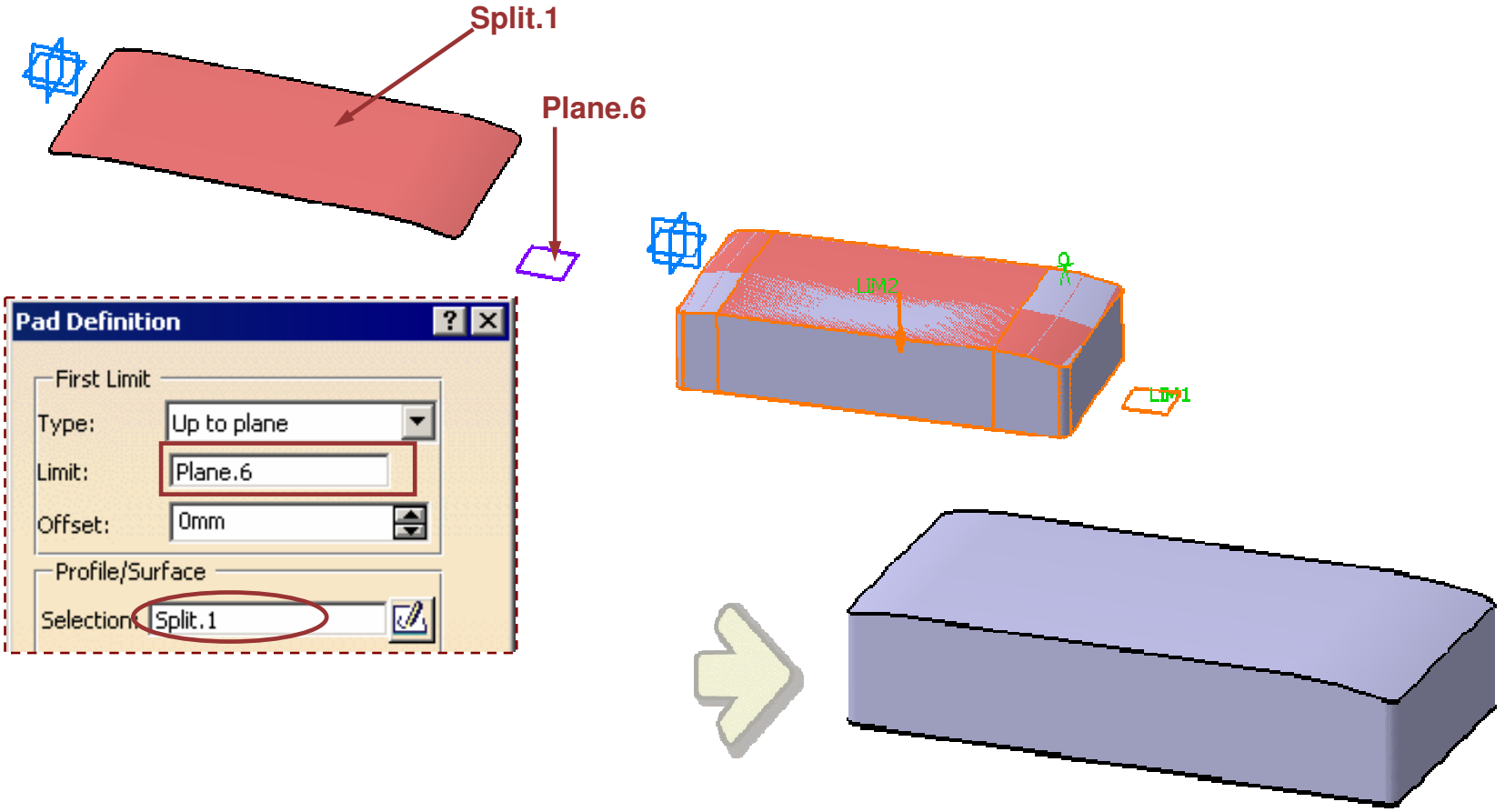
4. Keep this file open for the next step.

Student Notes:

Do It Yourself (1/3)

Load: 'Bottom_Case_Battery_Container_Shape_Step_1.CATPart'

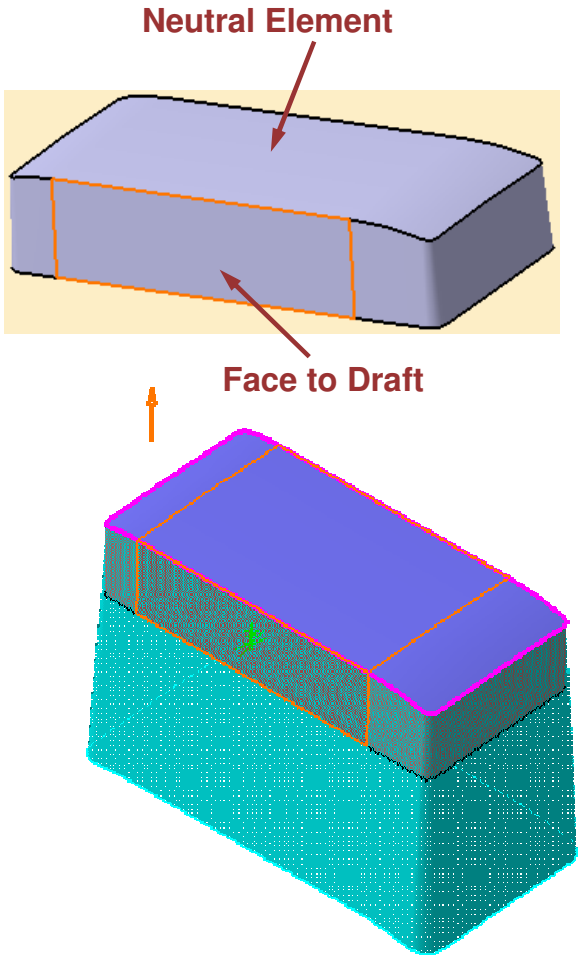
- Create the pad by extruding the surface ‘Split.1’ up to the plane ‘Plane.6’ in the PartBody as shown.

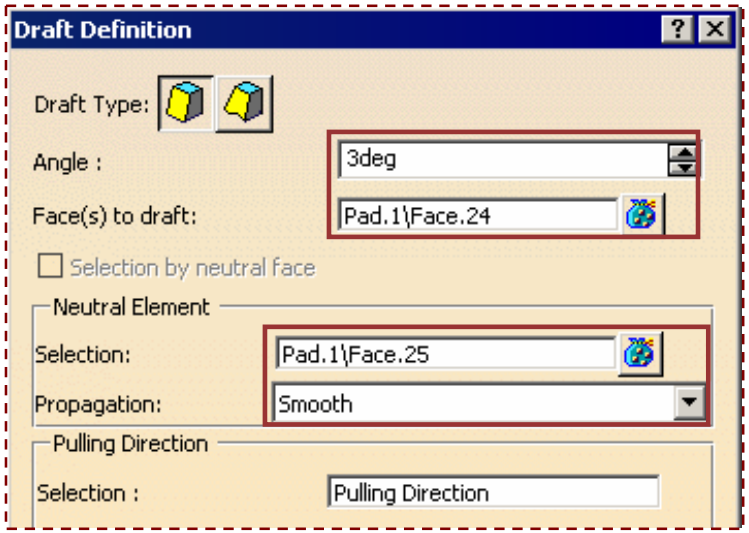


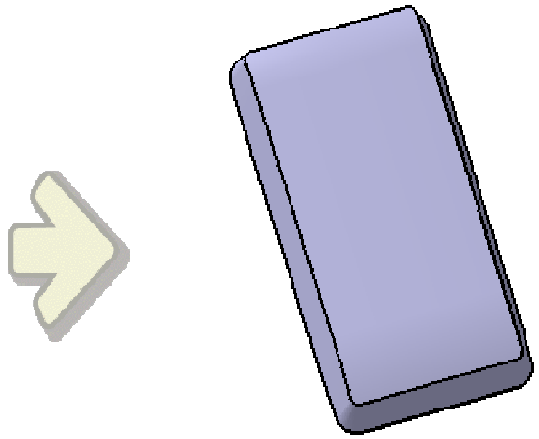
Student Notes:

Do It Yourself (2/3)

- Create a 3 degree draft on the face shown below, using the top face as the Neutral Element.



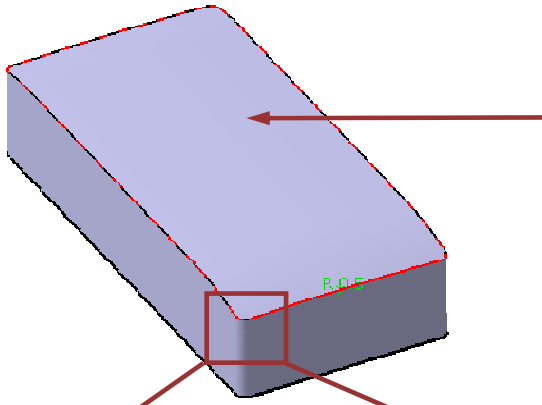


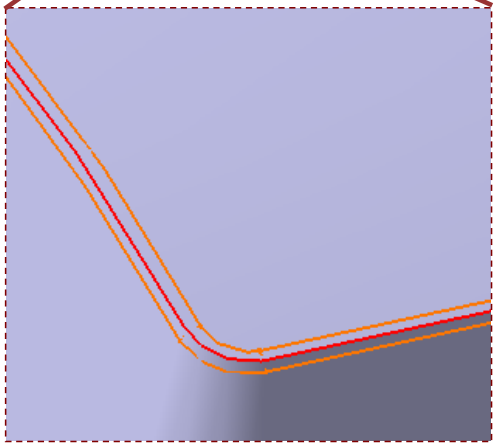


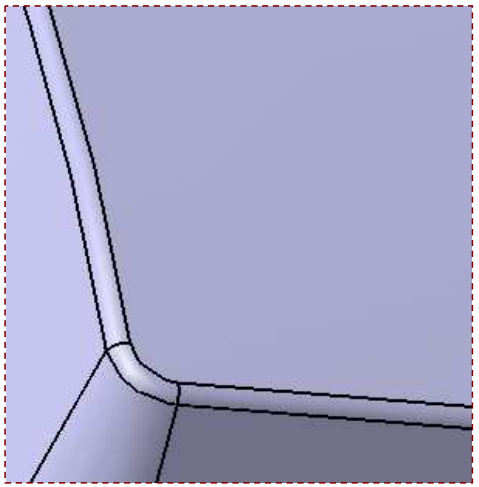
Student Notes:

Do It Yourself (3/3)

- Create a 1 mm fillet on the top face edges.







Keep this file open for the next step.

Student Notes:

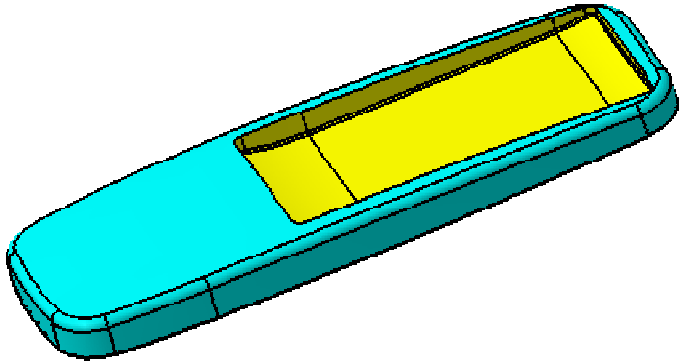
Mobile Phone Bottom Case

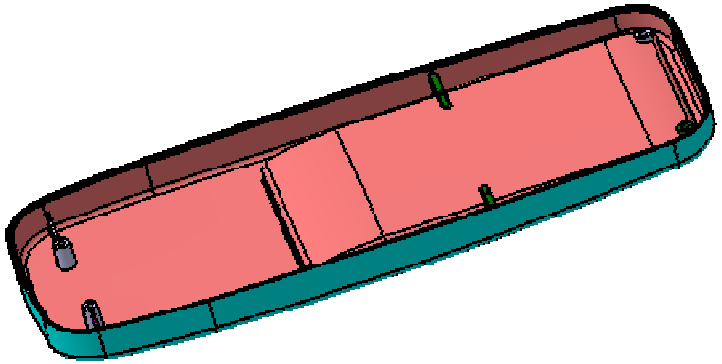
Step 2 – Design the Bottom Case

60 min

In this step you will create the main shape of the phone in a separate file from the battery reservation. By the way you:

- Design the External shape.
- Import Battery reservation space.
- Remove the battery space from the external shape.
- Design the Inner shape.
- Design the Interactive Board.
- Remove Interactive Board from Inner shape.
- Trim the above result from the External shape.
- Design the Lip.
- Assemble Lip to External shape.
- Assemble the External shape to the Part body.

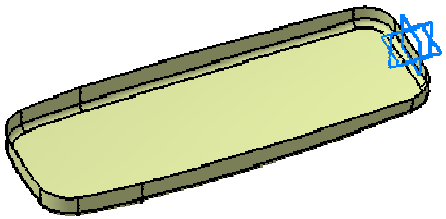


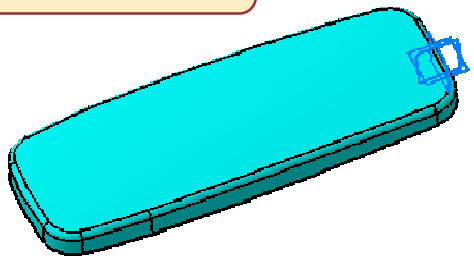


Student Notes:

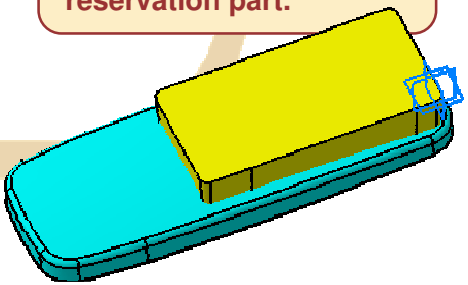
Design the External shape

1. Create the external shape using a surface as input.

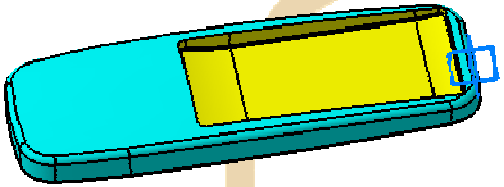




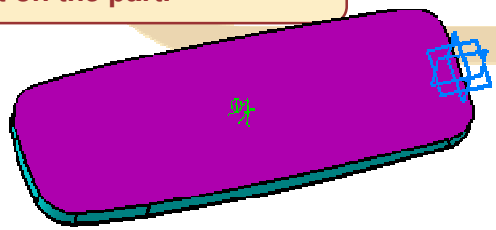
2. Import previously created Battery Space reservation part.

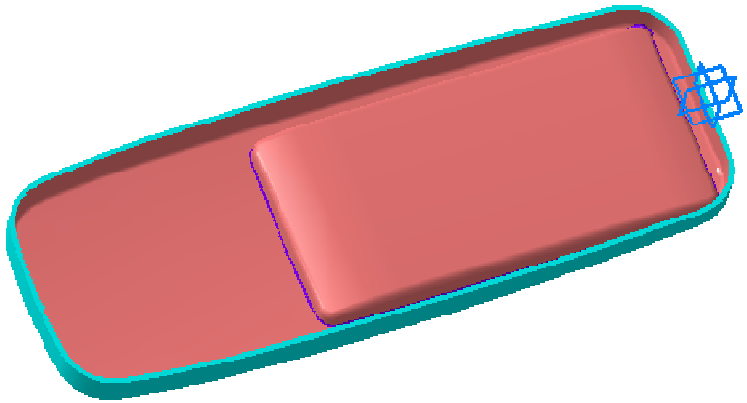


3. Remove the Battery space reservation part from the external shape.



4. Perform Shell and Edge fillet on the part.



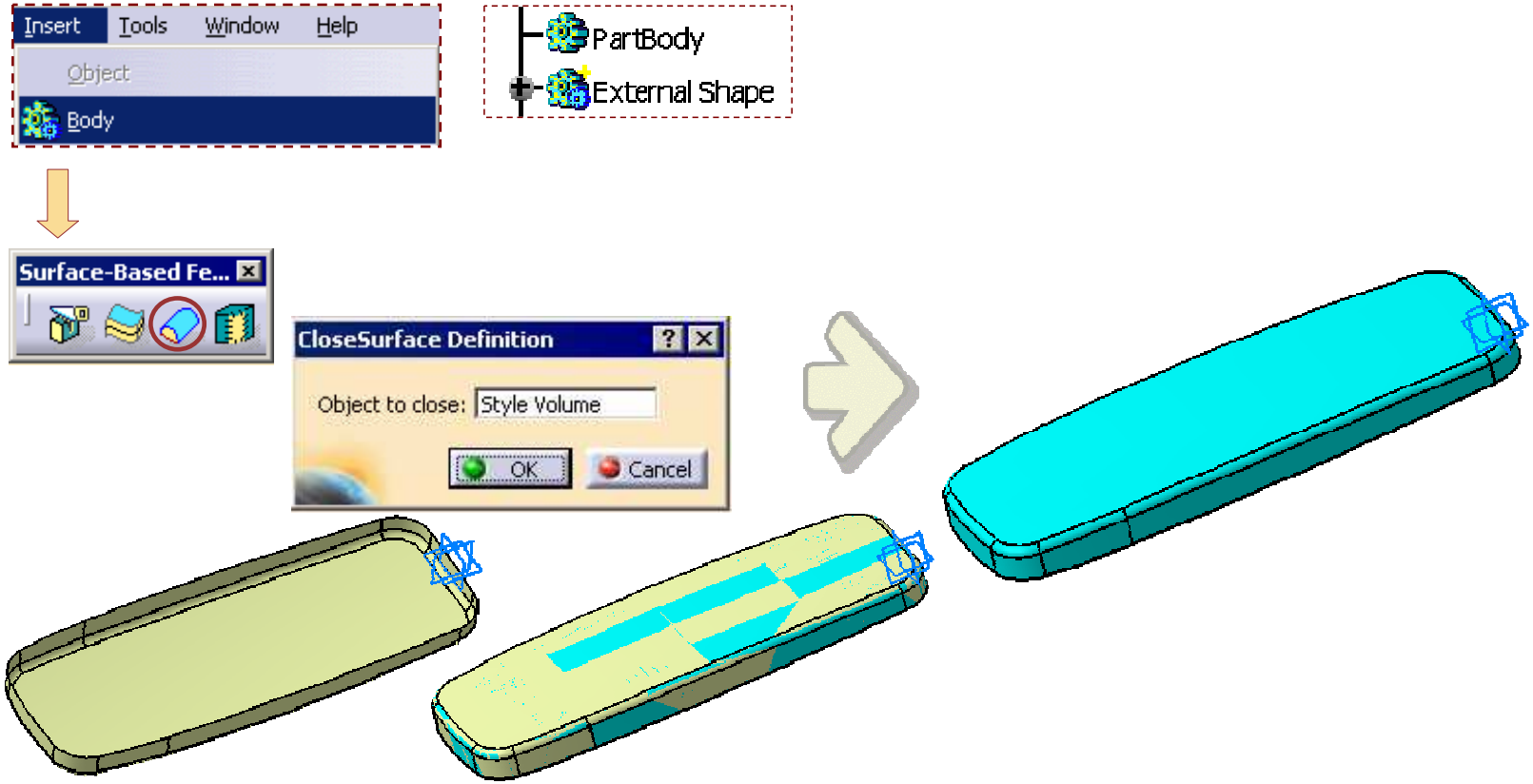


Student Notes:

Do It Yourself (1/4)

Load the part Bottom_Case_End_Step_2.1.CATPart
Result of Previous step:Bottom_Case_Battery_Container_Shape_Step_2.1

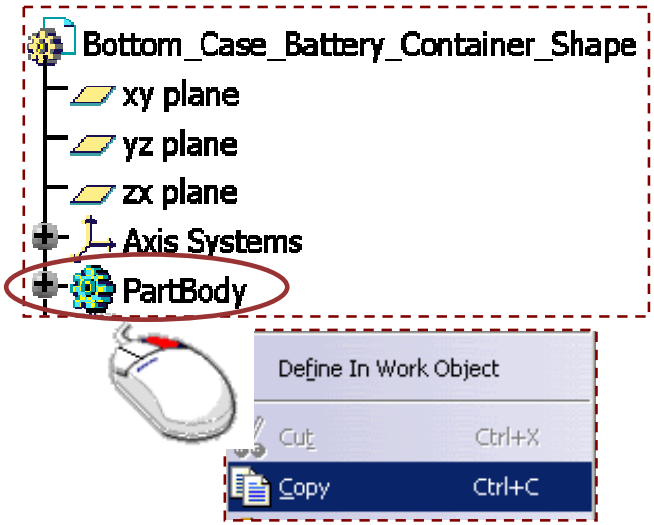
- Create the 'External Shape' Body in order to store the features that you will create.
- In this Body, create 'Close Surface' feature using the surface called 'Style Volume'.
- Hide the 'Style Volume' surface (you will not use it immediately.

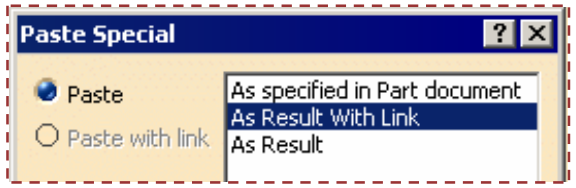


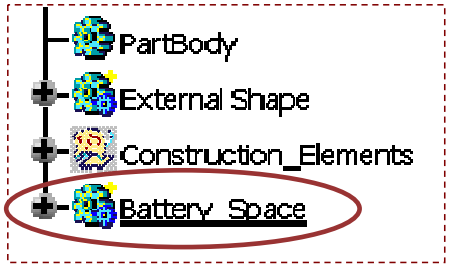
Student Notes:

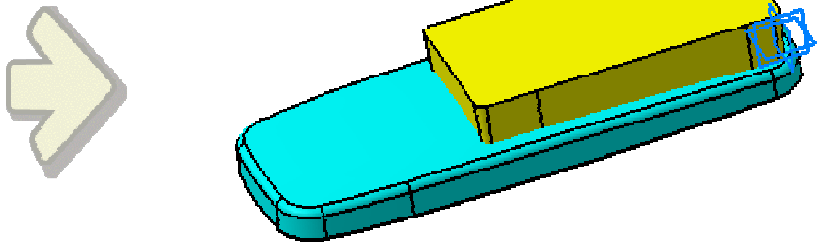
Do It Yourself (2/4)

- Copy the PartBody of the battery Volume.
- In the 'Bottom_Case_End_Step_2.1.CATPart', paste it 'as result with link' using paste special.
- Rename the resulting body as 'Battery_Space'.





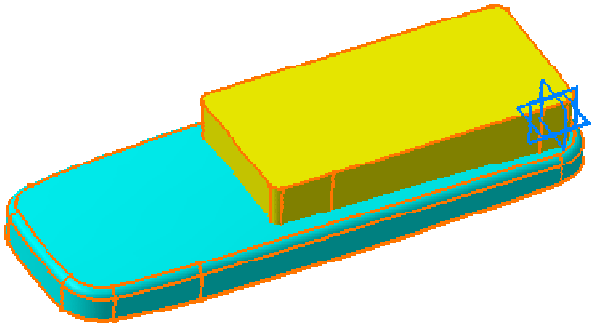


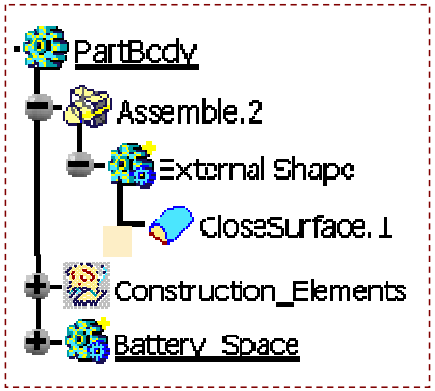


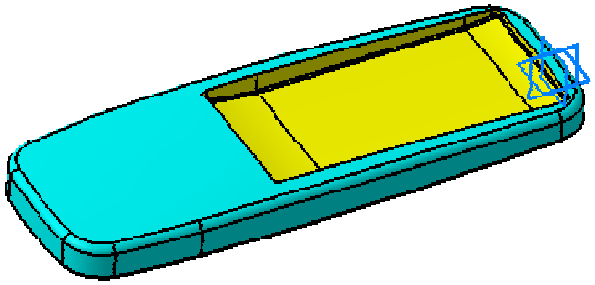
Student Notes:

Do It Yourself (3/4)

- Remove the 'Battery_Space' Body from the 'External Shape' Body using a Boolean operation.



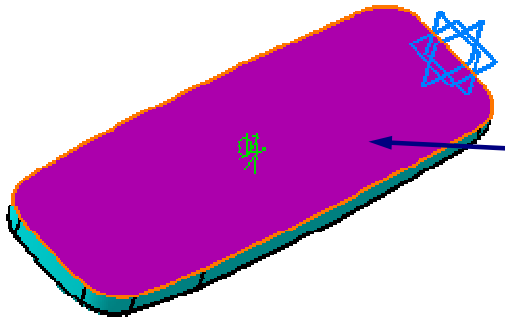


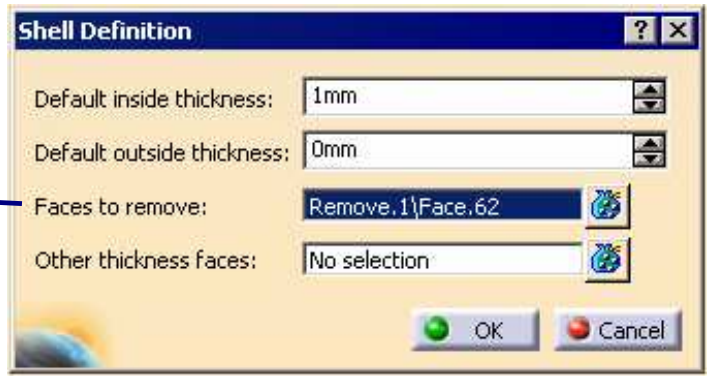


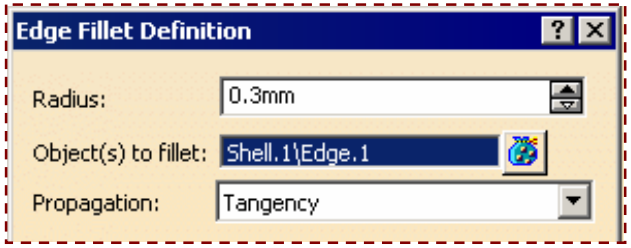
Student Notes:

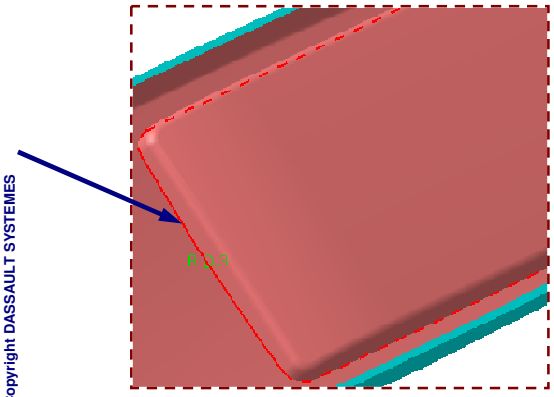
Do It Yourself (4/4)

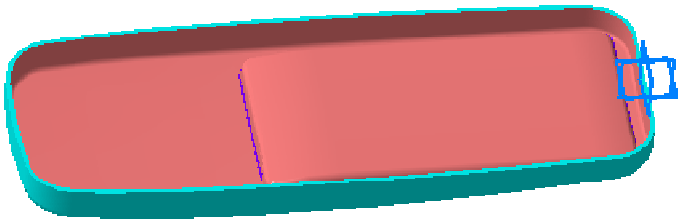
- Create a 1mm shell on the external shape.
- Create a 0.3mm fillet on the edge shown below.







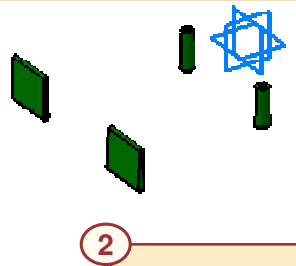




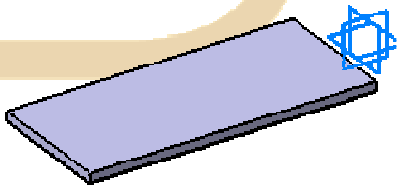
Student Notes:

Design the Inner shape and Interactive Board

1. Design the Inner shape.

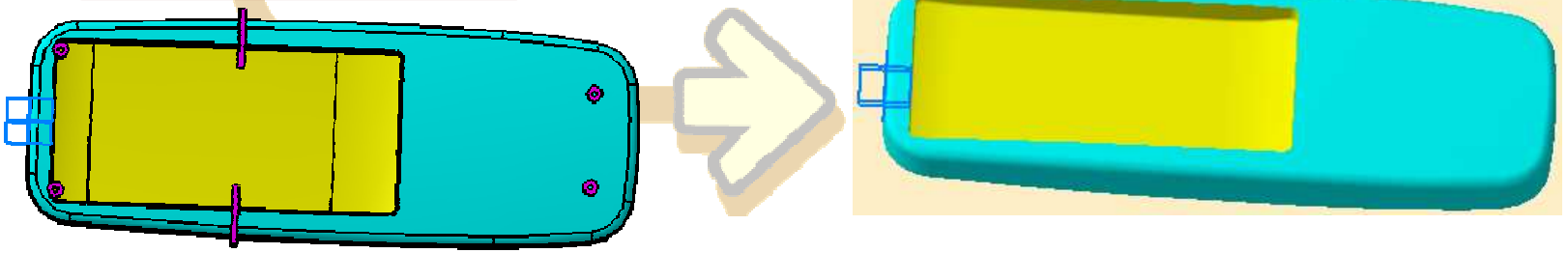


2. Design the interactive Board.



3. Remove Interactive Board from Inner shape using Boolean operations.

4. Trim the step 3 result with the External shape.

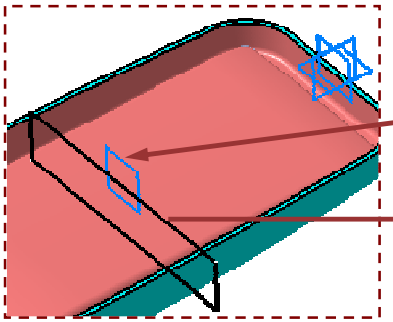


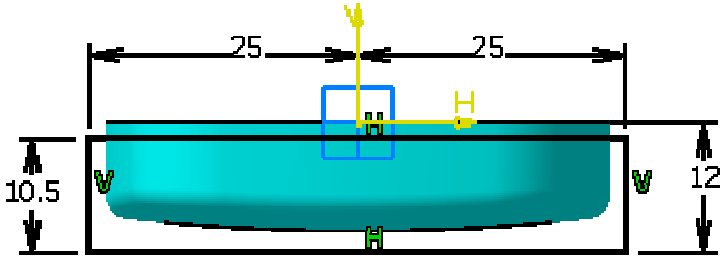
Student Notes:

Design the Inner Shape (1/6)

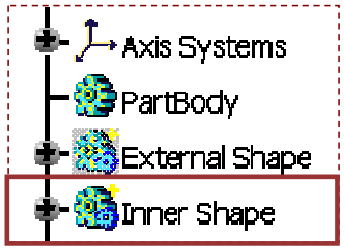
Load the part Bottom_Case_End_Step_2.2.CATPart
(Use EDIT > Links to Load and Open the Linked part.)

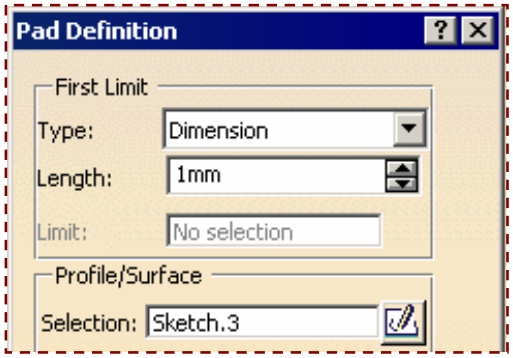
- In the 'Construction Elements' Geometrical Set, create a 44mm offset plane from YZ plane.
- On this plane, create the following sketch using the Y axis as the horizontal axis.

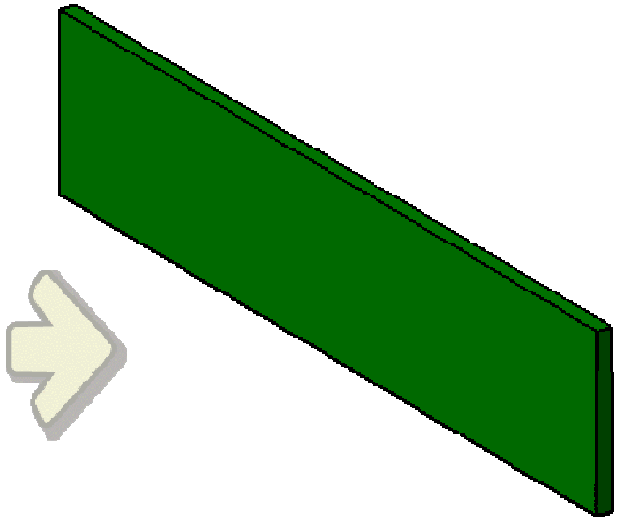




- Insert a new body called 'Inner Shape'.
- In this body, create a 1mm pad using the sketched profile shown above.





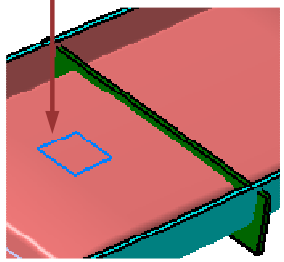


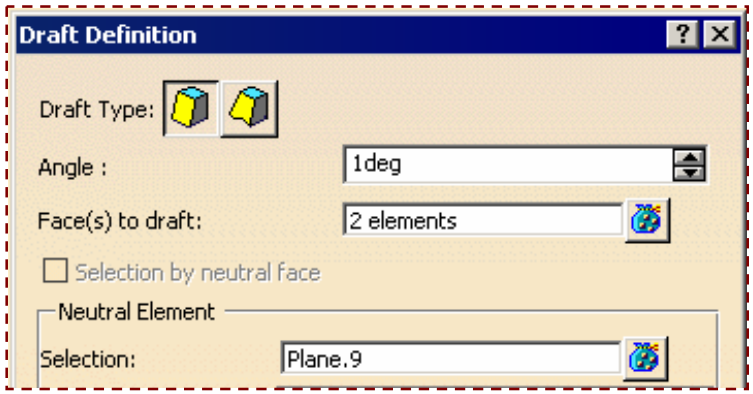
Student Notes:

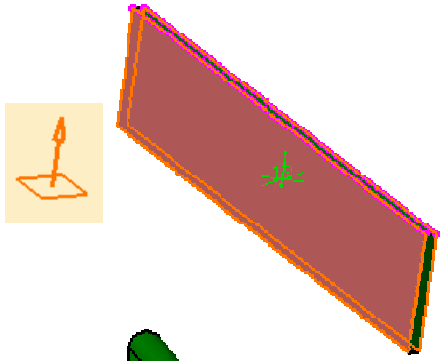
Design the Inner Shape (2/6)

- Create a plane offset from XY plane at a distance of 1.5 mm.
- Apply Draft of 1 deg. Use the plane as neutral element.

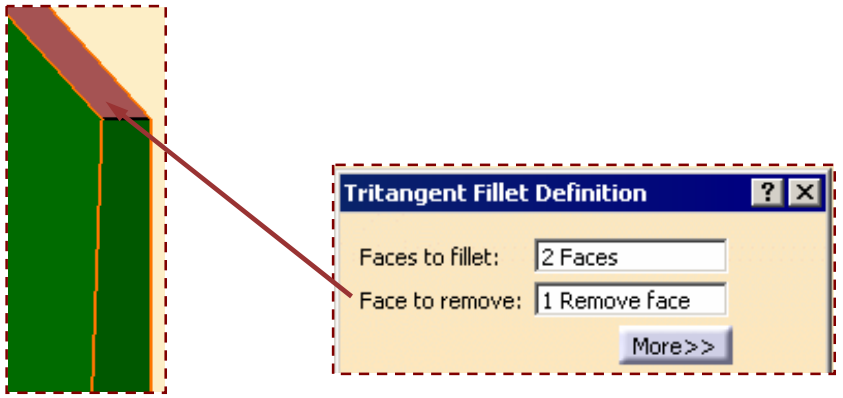
Plane

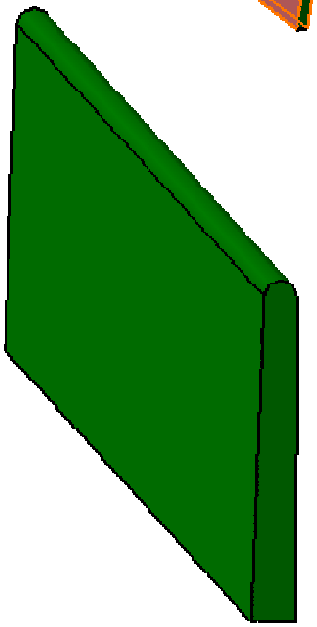






- Apply Tri-Tangent Fillet by removing the face shown.

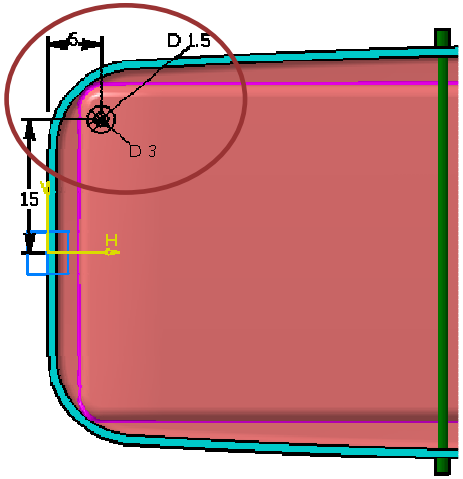




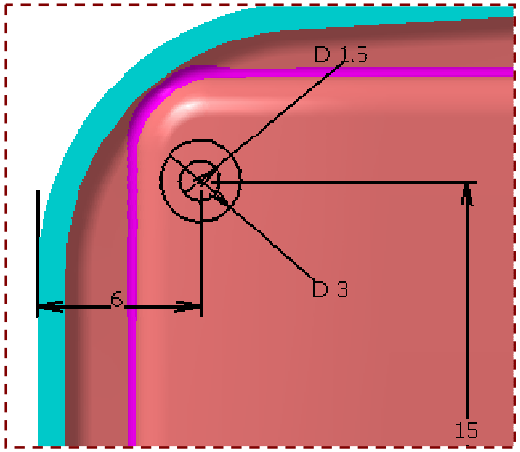
Student Notes:

Design the Inner Shape (3/6)

- Create a sketch on XY plane in construction elements body.

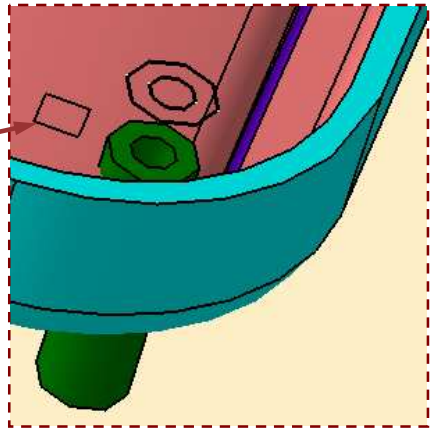


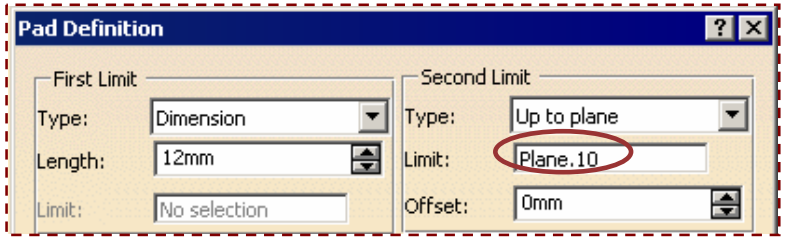
Details of sketch are



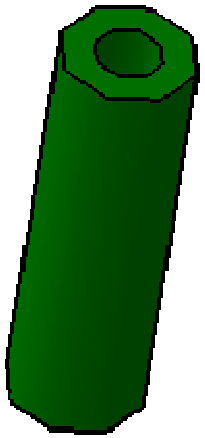
- Create a plane offset below XY plane at a distance of 2.5 mm in construction elements body.
- In Inner shape body create a pad of 12 mm(use above sketch). Use the Plane created above as one of the limit for the Pad.

Plane





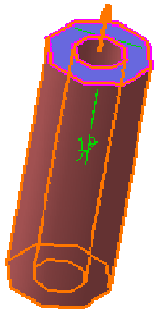
Optional: Click Reverse direction

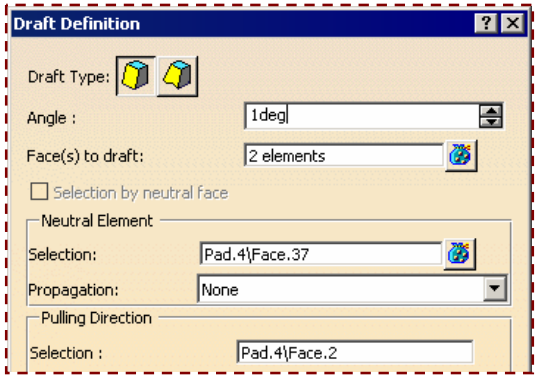


Student Notes:

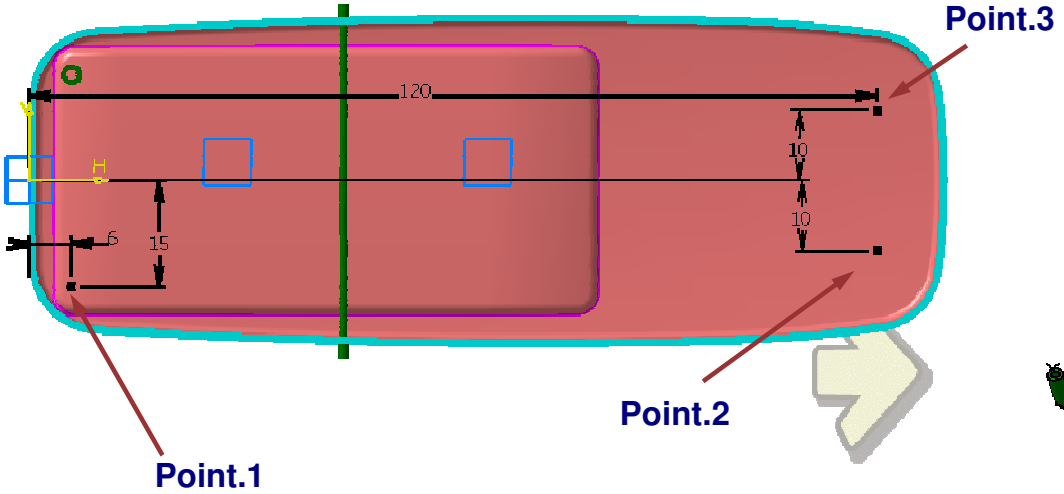
Design the Inner Shape (4/6)

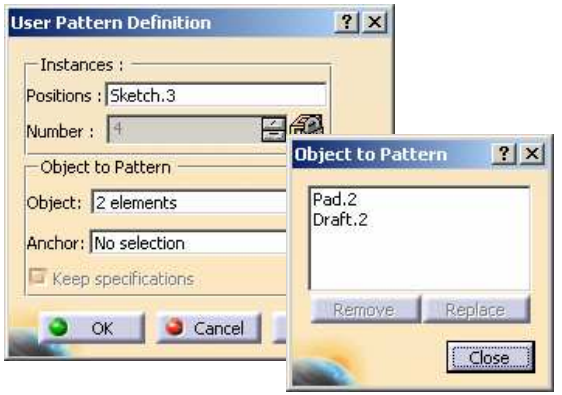
- Apply a Draft of 1 deg to the faces shown.





- Create a sketch(Consisting of points in construction elements body) on XY Plane.Create a User Pattern using the sketch(in inner shape body).

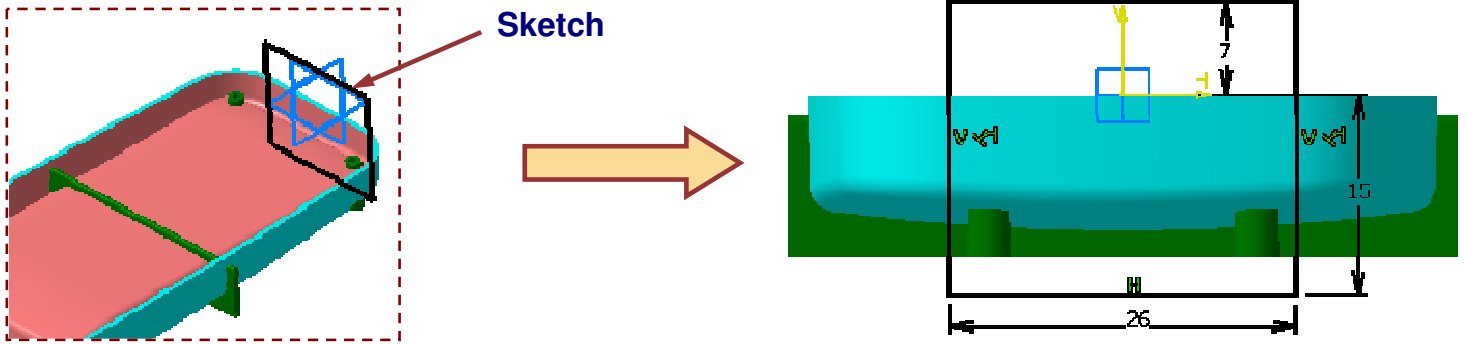




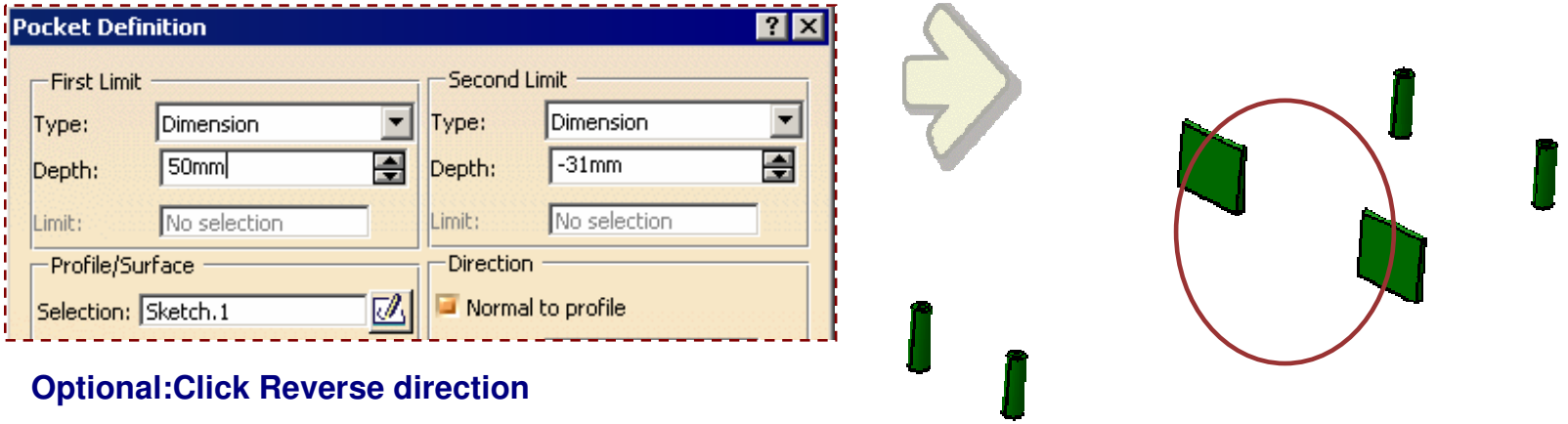
Student Notes:

Design the Inner Shape (5/6)

- Create a Sketch(in construction elements body) on YZ Plane.



- Create a pocket in inner shape body using the sketch.

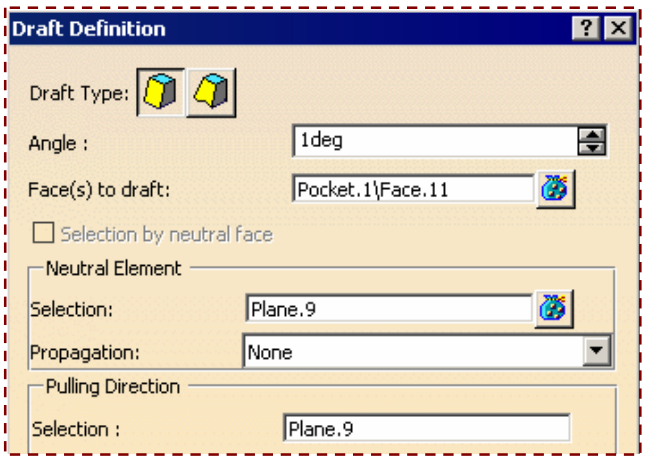


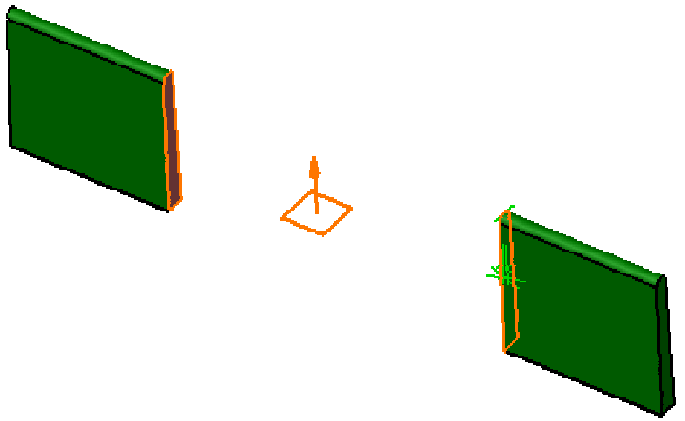
Optional:Click Reverse direction

Student Notes:

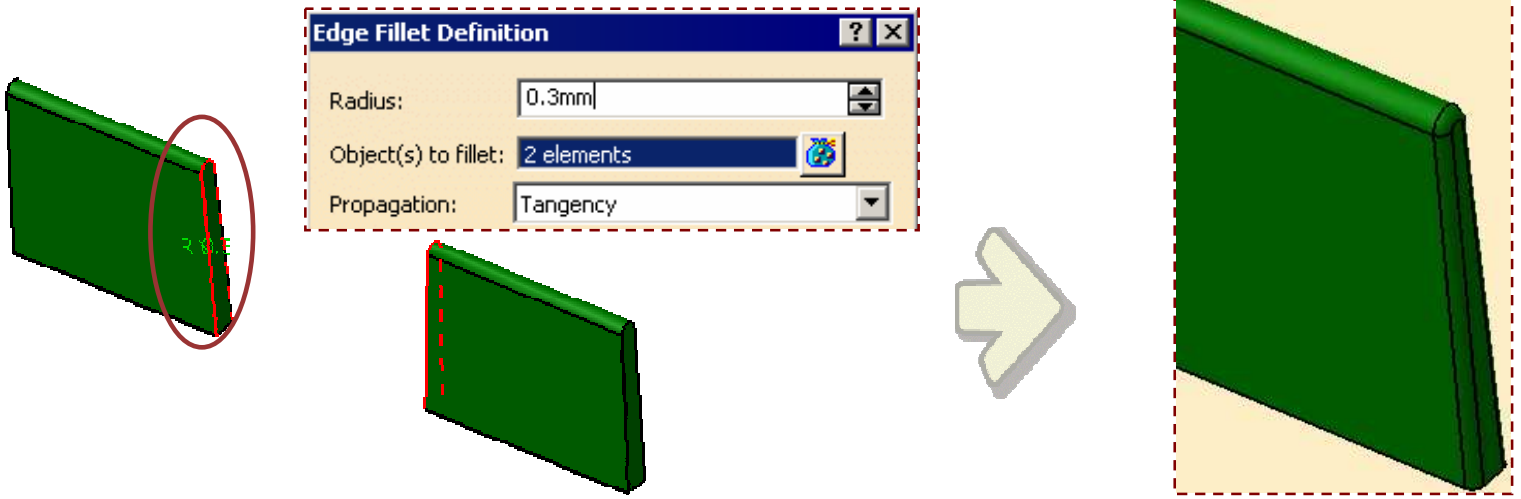
Design the Inner Shape (6/6)

- Create a Draft Of 1 Deg.Select previously created plane(offset from XY= 1.5 mm) as the neutral element





- Create a Edge Fillet of 0.3 mm

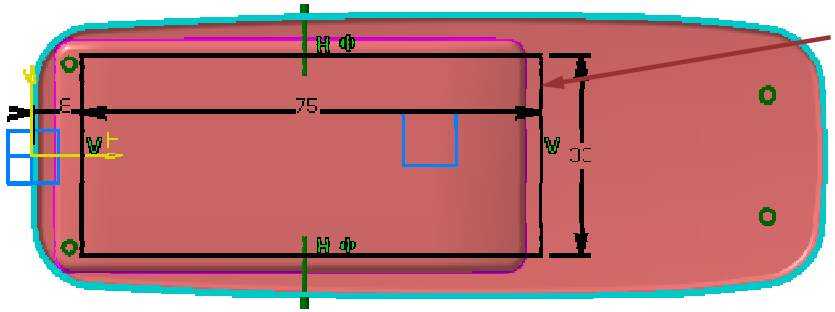


Student Notes:

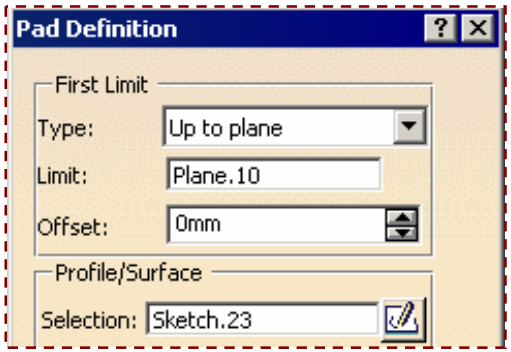
Design the Interactive Board

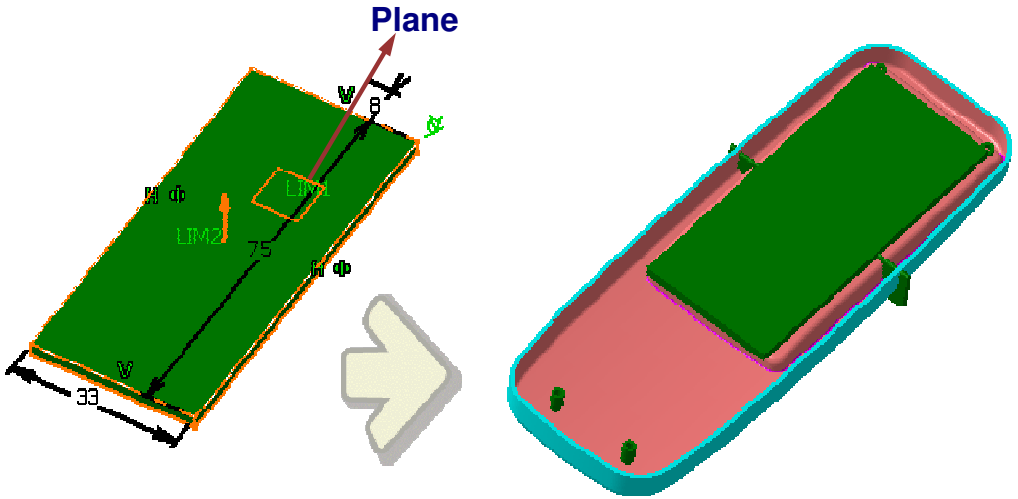
Load the part Bottom_Case_End_Step_2.3.CATPart
(Use EDIT > Links to Load and Open the Linked part.)

- Create a sketch in XY Plane in construction elements body.



- Insert a New Body Interactive_Board.
- Create a Pad using the above sketch up to Plane (Offset from XY by 2.5 mm)

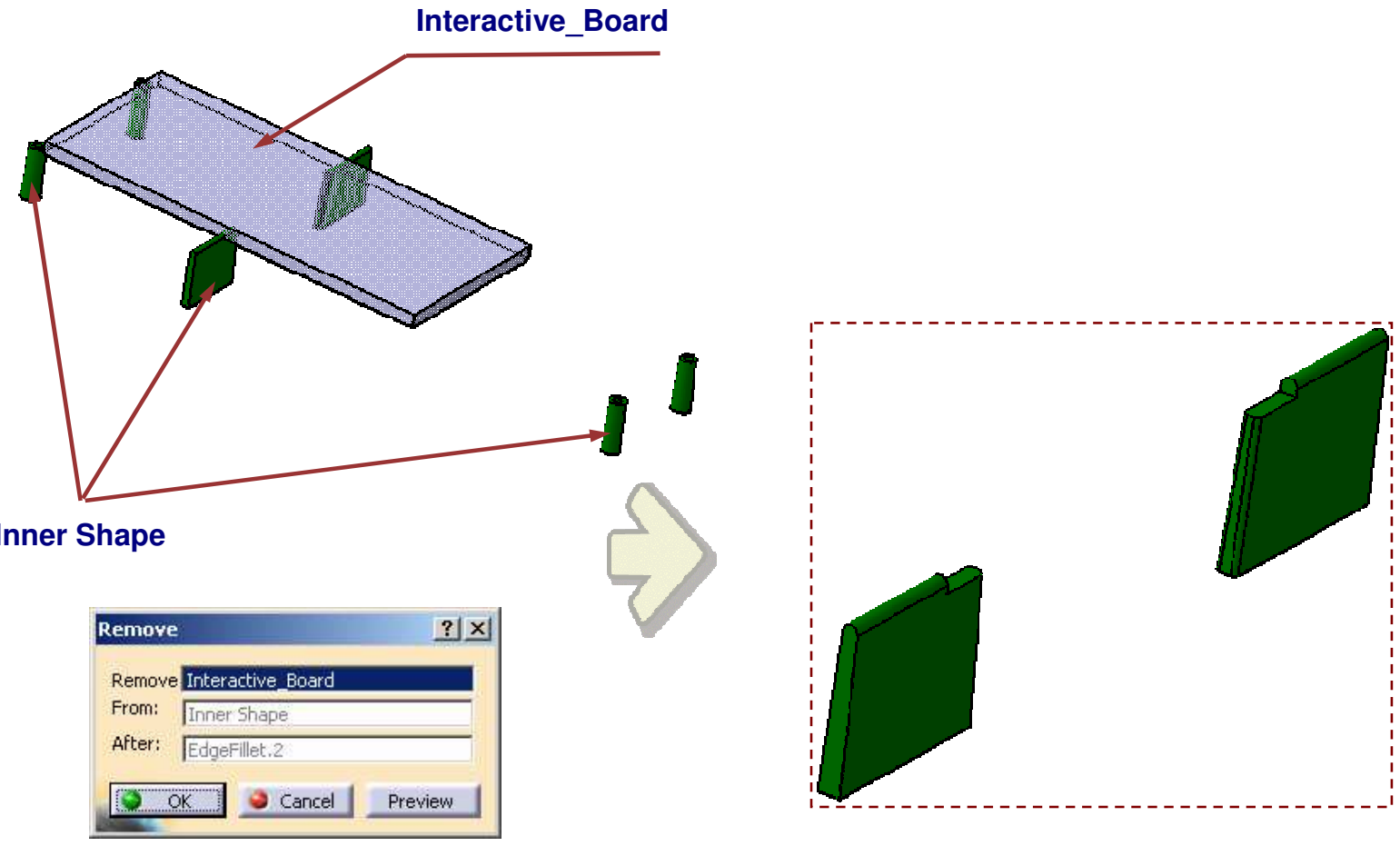




Student Notes:

Do It Yourself(1/3)

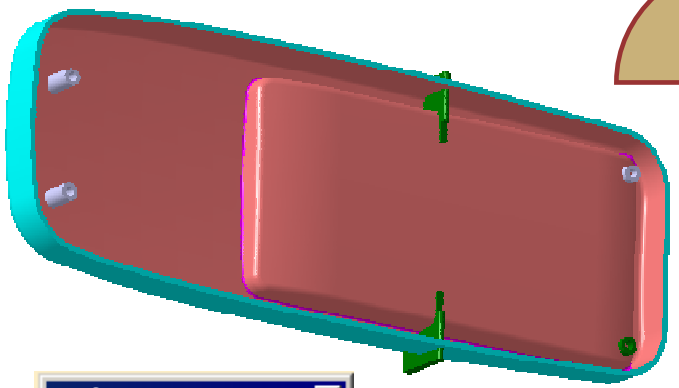
- Using Boolean REMOVE, remove the interactive_Board body from inner shape body.



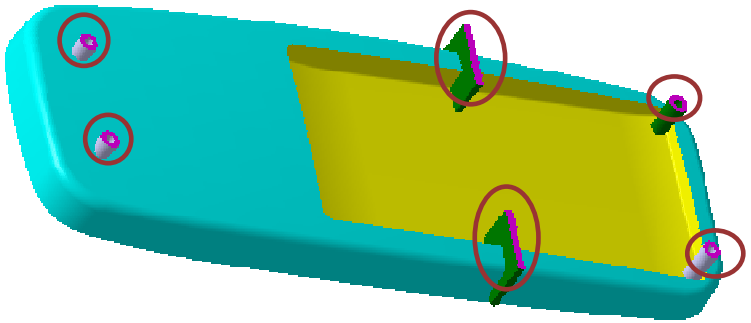
Student Notes:

Do It Yourself (2/3)

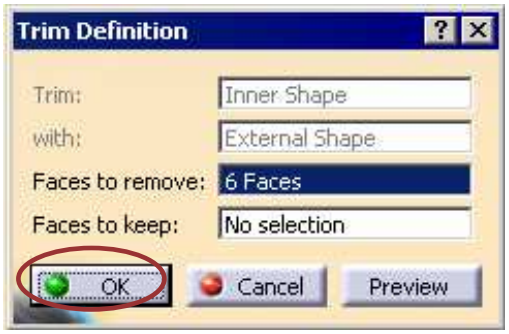
- Using Boolean TRIM, Trim the inner shape body with respect to External shape body.Select 6 faces to remove.

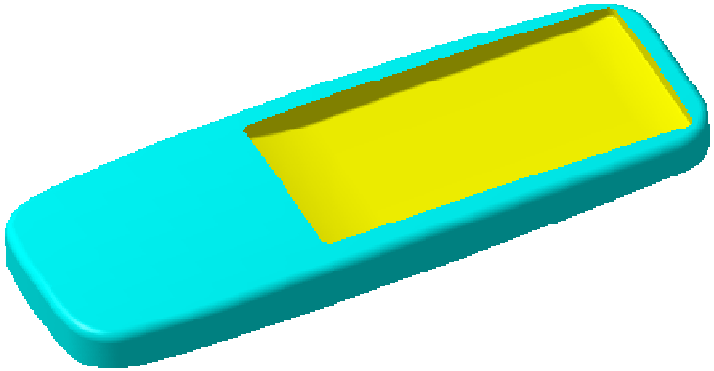


Rotate to see the other side of the part



Faces to be removed are highlighted in Purple

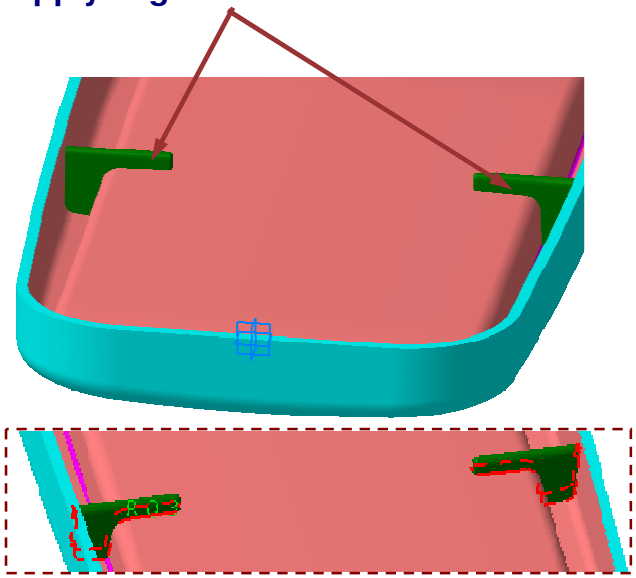


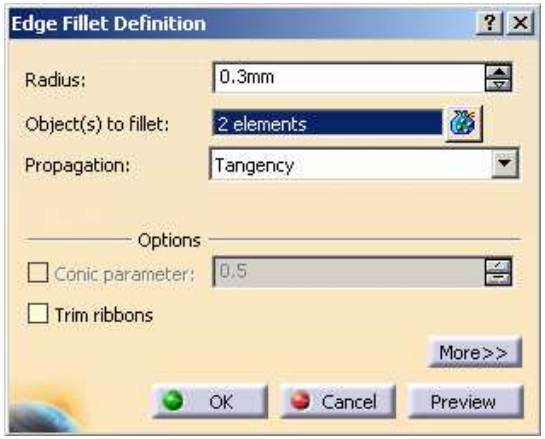


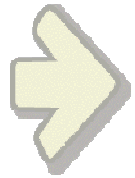
Student Notes:

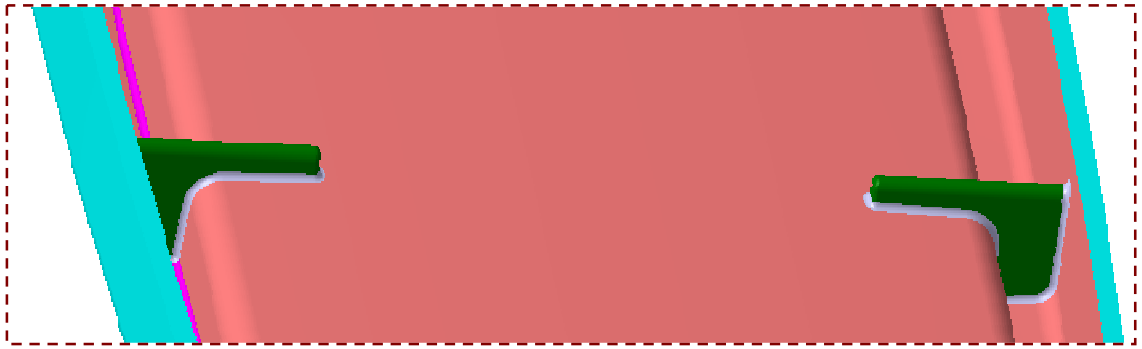
Do It Yourself (3/3)

- Apply edge fillet of 0.3 mm to the external shape part.





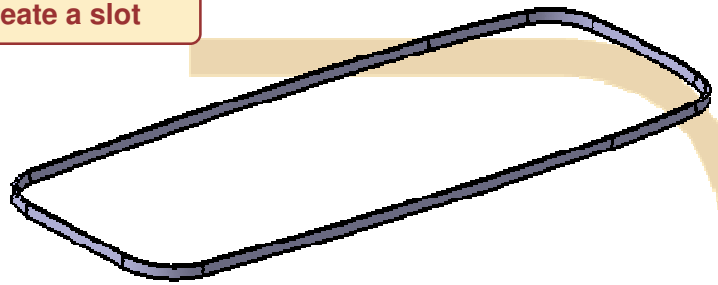




Student Notes:

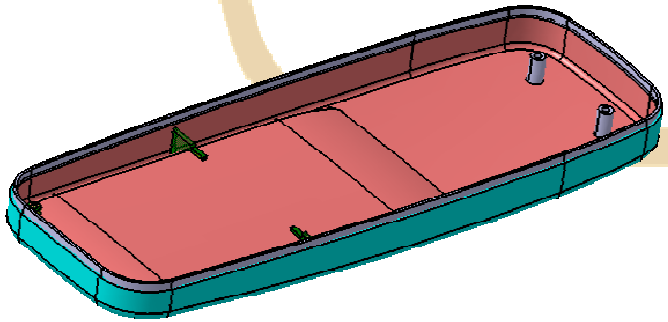
Design the LIP

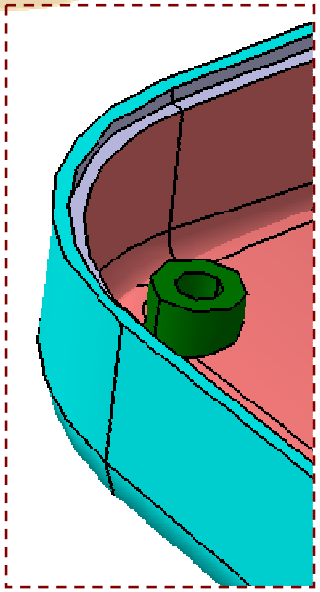
1 Create a slot



2 Apply draft to the slot

3 Assemble the LIP to the External shape body.



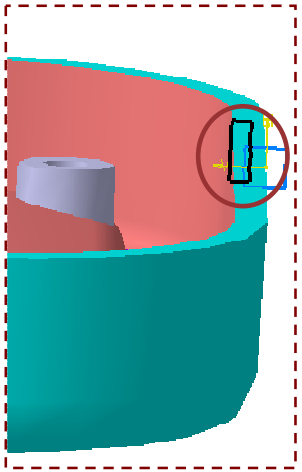


Student Notes:

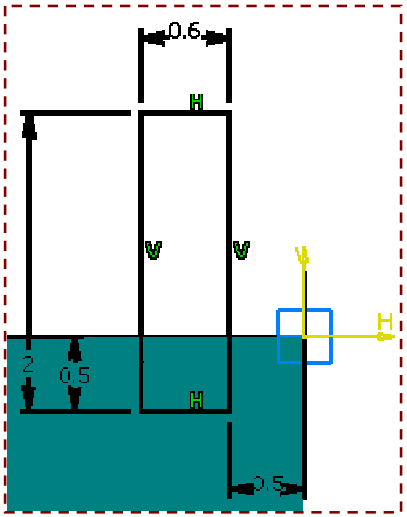
Do It Yourself (1/3)

Load the part Bottom_Case_Step_End_2.4.CATPart
(Use EDIT > Links to Load and Open the Linked part.)

- Insert a body and name it as LIP.
- Sketch on ZX plane. This sketch is used as the Profile for the slot.

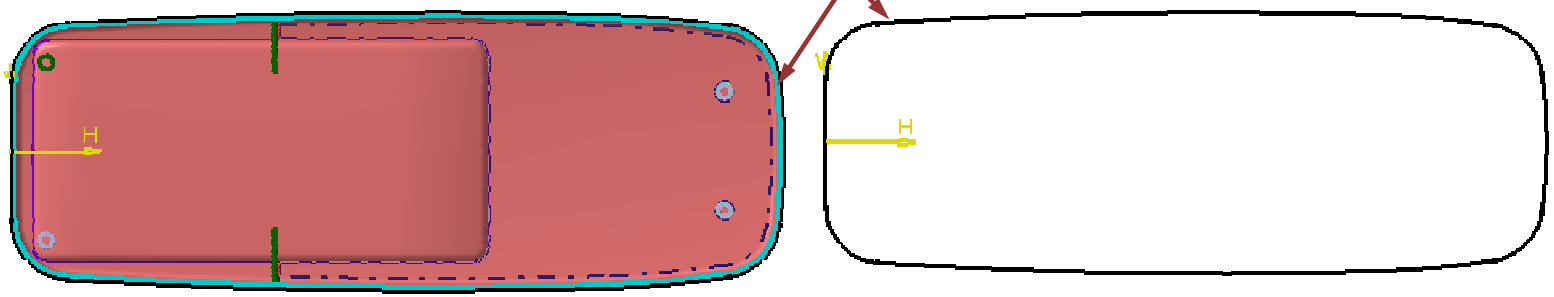


The Details of the sketch are:



- Create a sketch on XY plane Project the outer edge in it. It will be used as the center curve for the slot

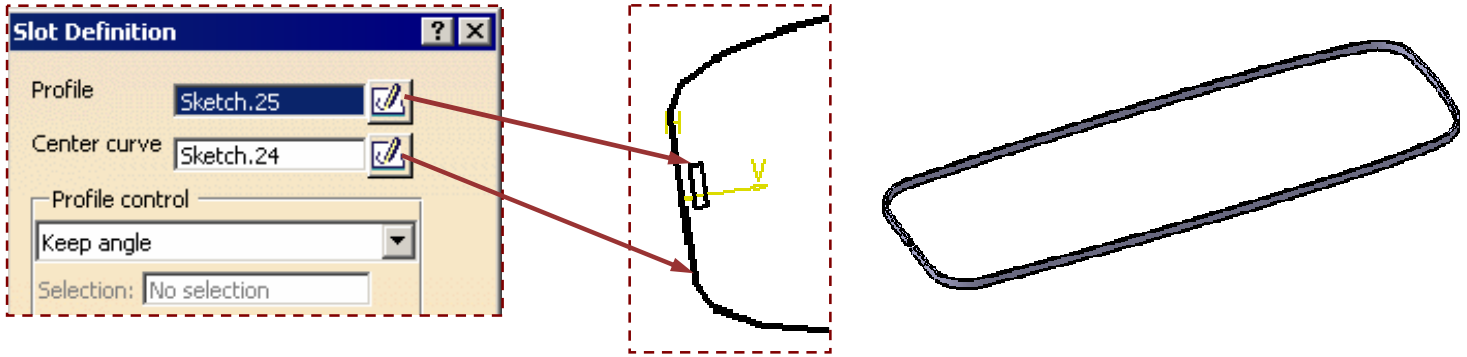
The Black edge is projected edge of the external shape



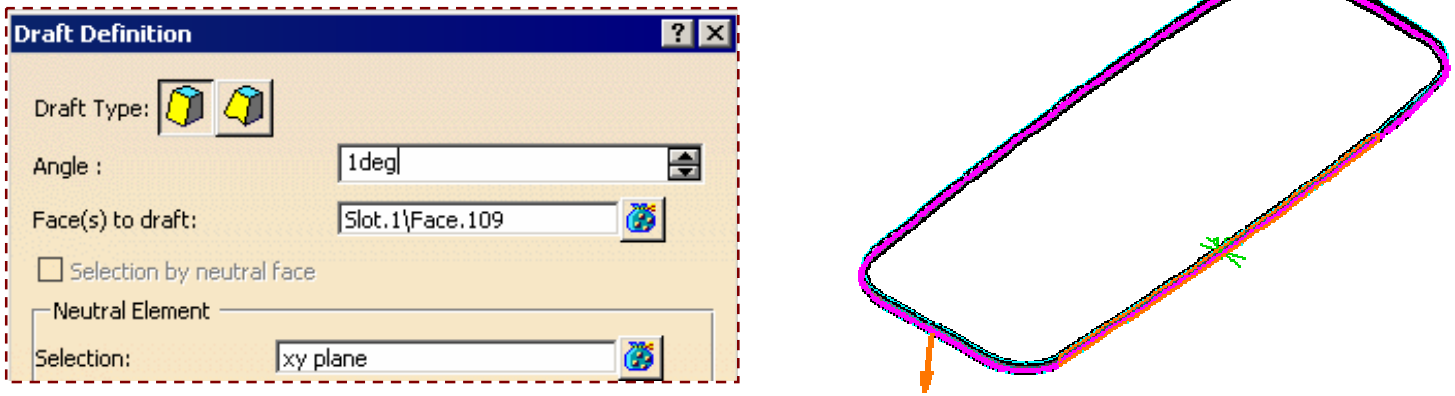
Student Notes:

Do It Yourself (2/3)

- Create a slot using the sketches created.



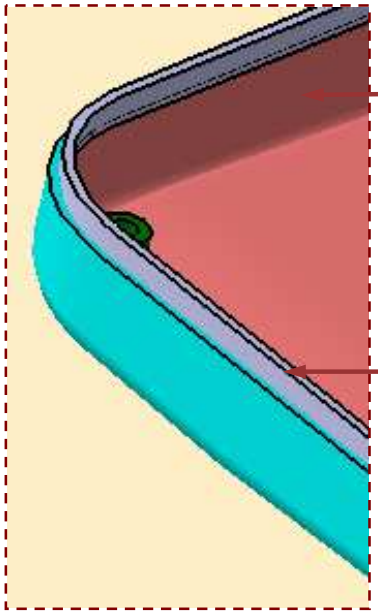
- Apply Draft of 1 deg to faces of the slot.

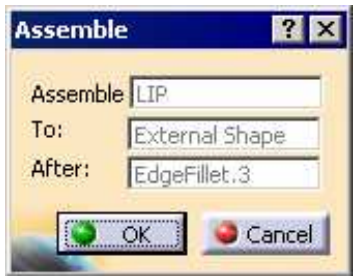


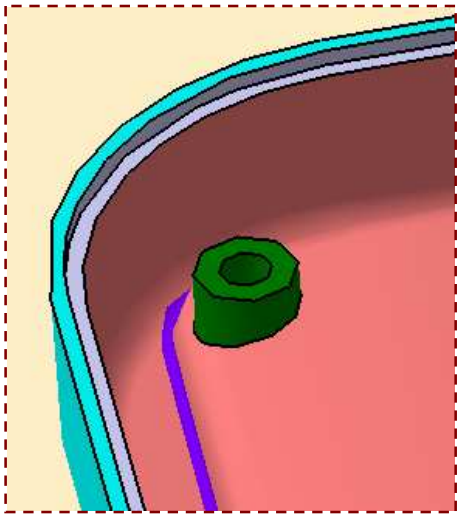
Student Notes:

Do It Yourself (3/3)

- Assemble the LIP body with the external shape body.



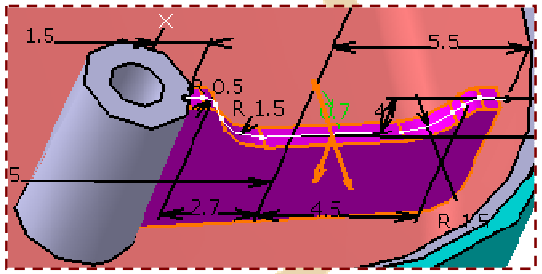




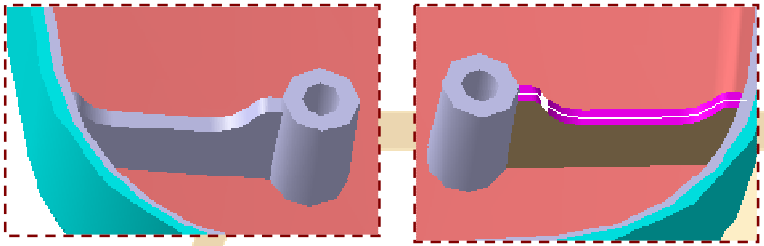
Student Notes:

Creating stiffeners and assembling External shape

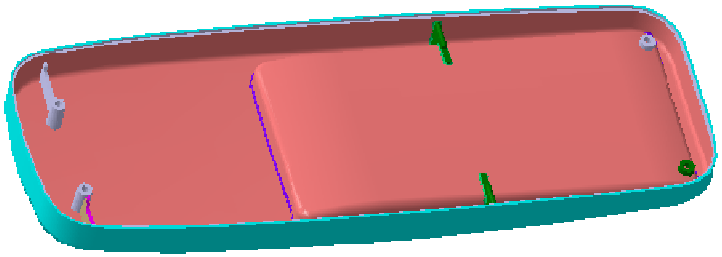
1 Create Stiffeners.Apply Draft to them



2 Mirror the stiffener



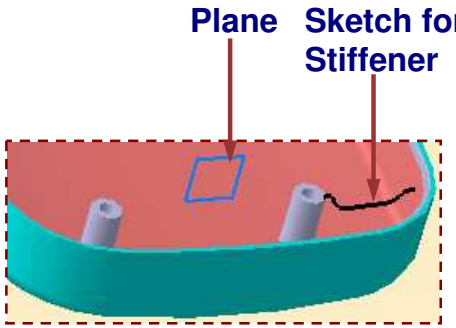
4 Assemble the external shape body with the part body.



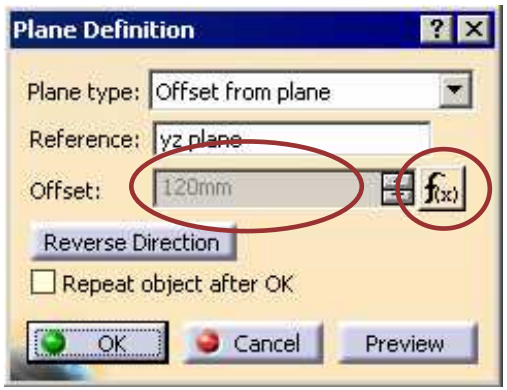
Student Notes:

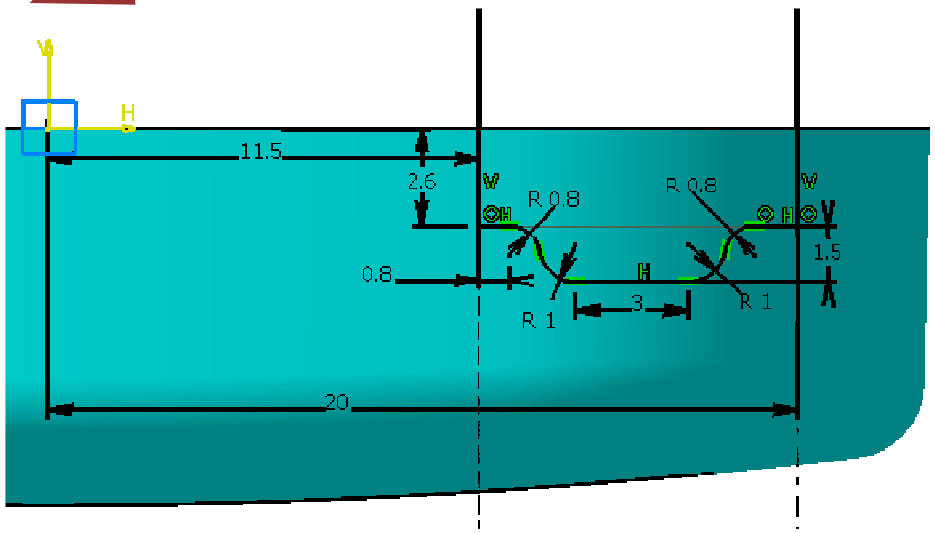
Do It Yourself (1/3)

- Create plane offset of 120 mm from YZ plane. Apply formula between the plane and sketch length for user pattern.
- Create a sketch for stiffener on it.



The Details of the sketch are:

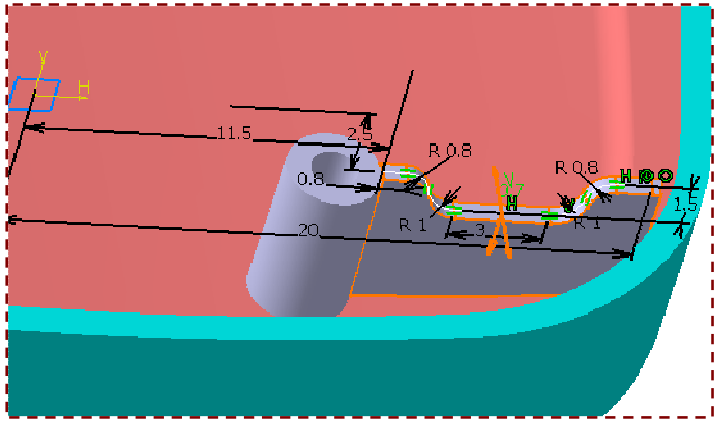


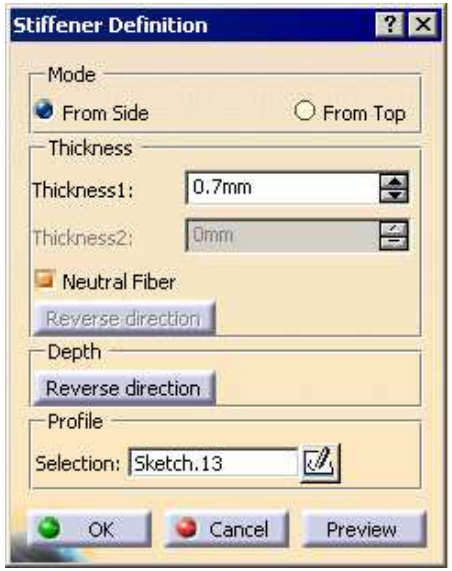


Student Notes:

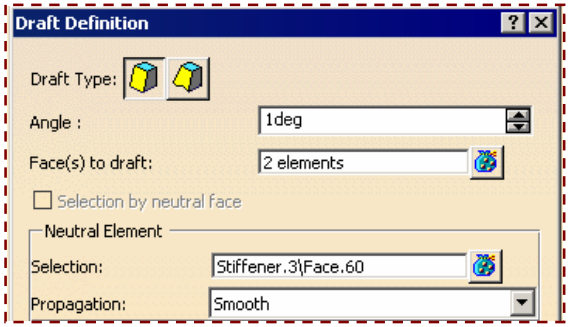
Do It Yourself (2/3)

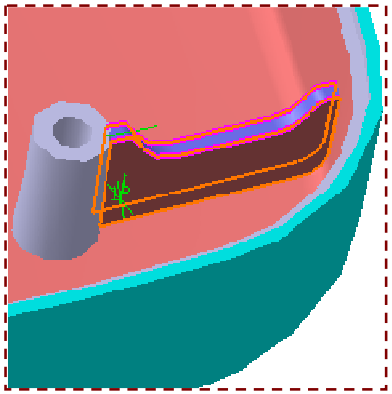
- Create a stiffener with thickness of 0.7 mm from the sketch.





- Apply Draft of 1 deg to the stiffener.

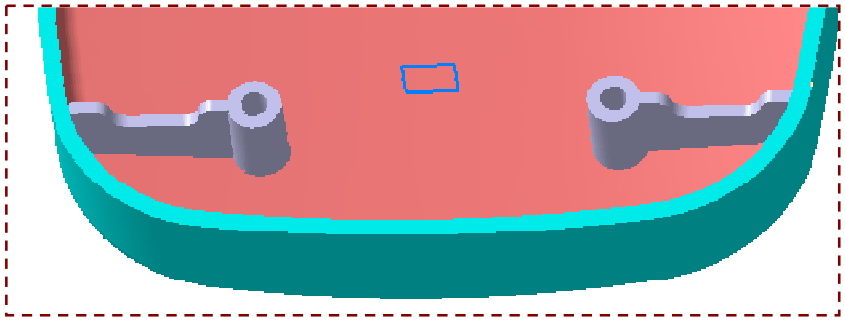




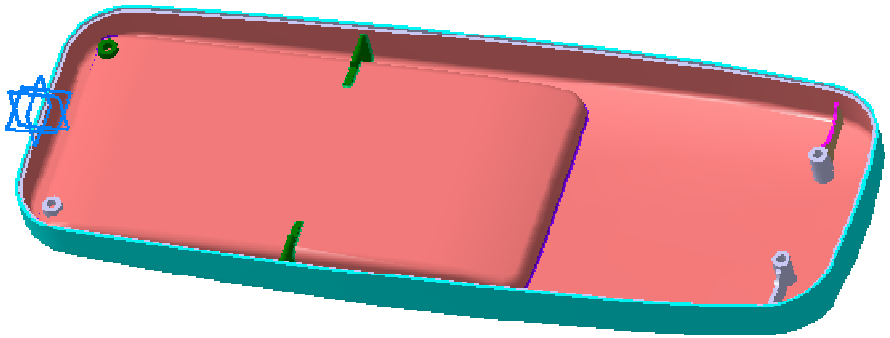
Student Notes:

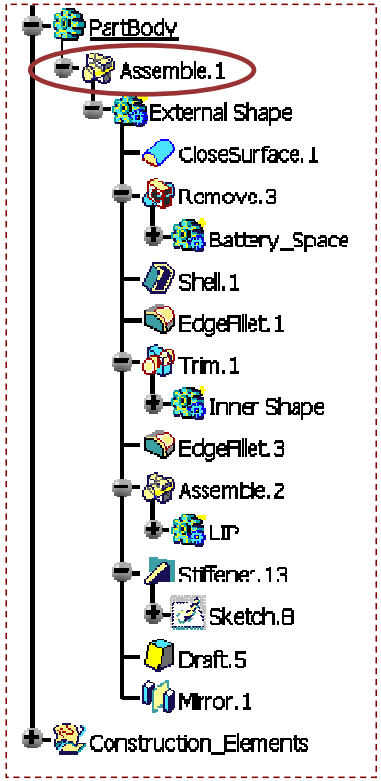
Do It Yourself (3/3)

- Create a Mirror of stiffener and draft about ZX plane.



- Assemble the External shape body with the Part body using Boolean operations.





Load: Bottom_Case_End_Step_Final.CATPart
(Use EDIT > Links to Load and Open the Linked part.)

Student Notes:

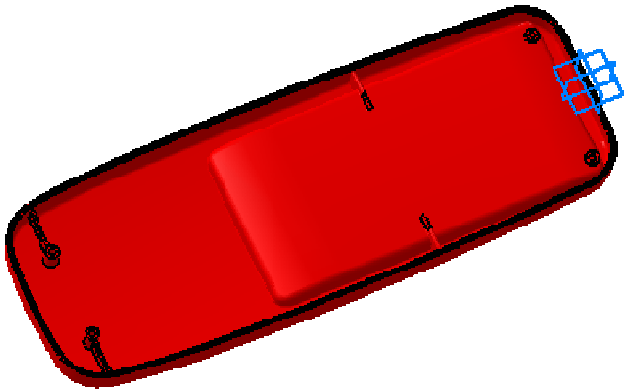
Mobile Phone Bottom Case

Step 3 – Modifying the Design

20 min

In this step you will modify the design of the part:

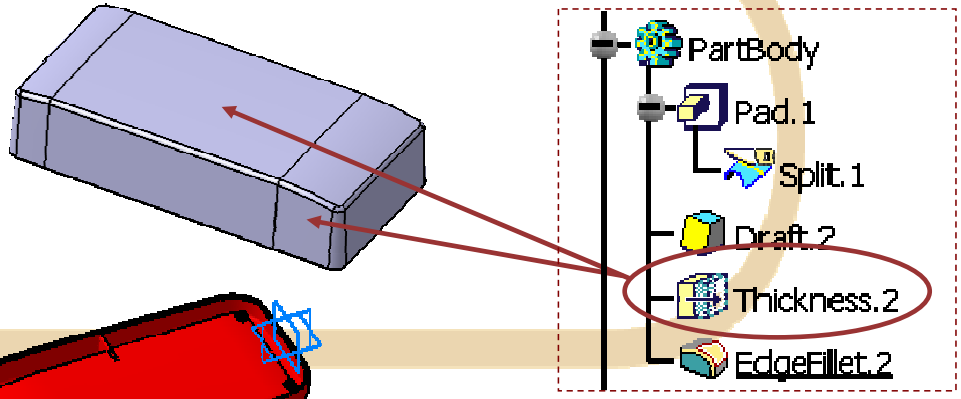
- **Modify the Dress-up features on battery reservation space.**
- **Apply the modifications to the Bottom case**
- **Modify the position of the pad**
- **Modify the position of stiffeners**



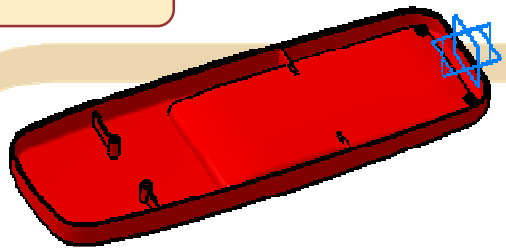
Student Notes:

Do It Yourself

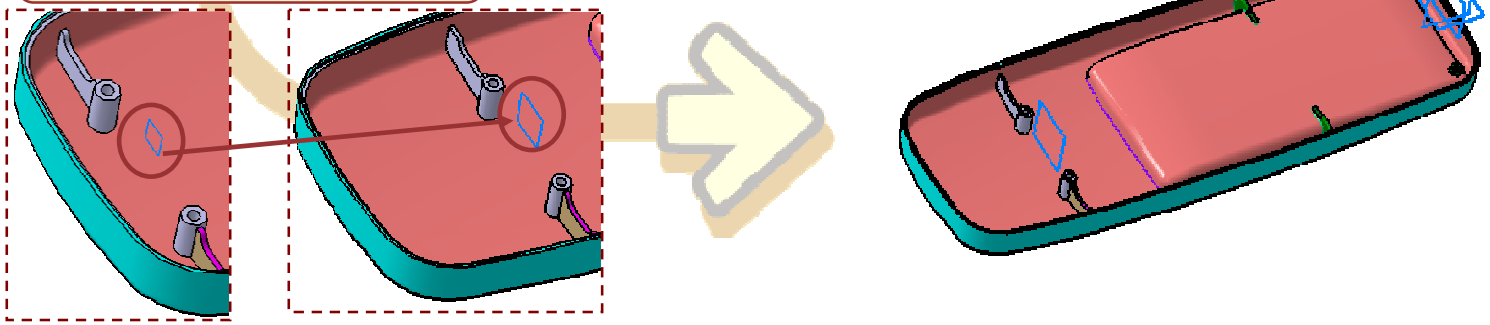
1. Apply Thickness to Reservation space.Edit the edge fillet value.



2. See the effect of modifications in Bottom Case part.



3. Move the position of pads and stiffeners

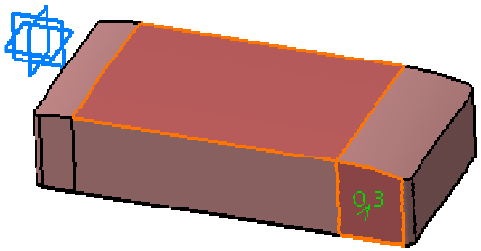


Student Notes:

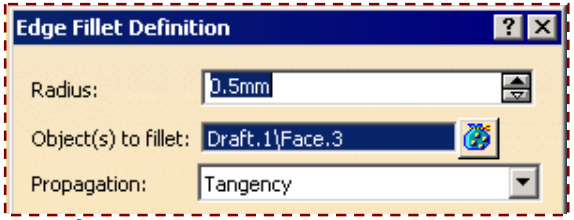
Do It Yourself (1/4)

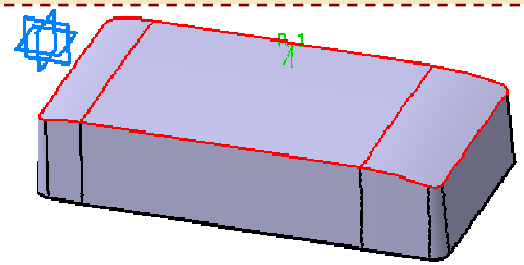
1.Load the part 'Bottom_Case_End_Step_3_Modify.CATPart'
1.Use Edit > Links > load and open
'Bottom_Case_Battery_Container_Shape_step_3_Modify.CATPart

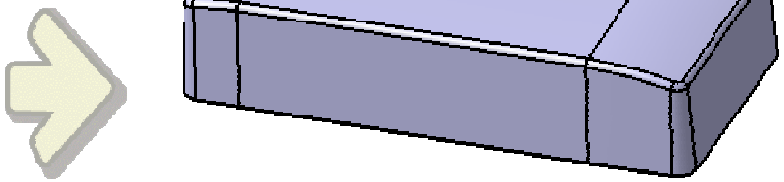
- Work in 'Bottom_Case_Battery_Container_Shape_step_3_Modify.CATPart'
- Apply thickness of 0.3 mm to the two faces shown.



- Modify the edge fillet from initial value of 1 mm to 0.5mm



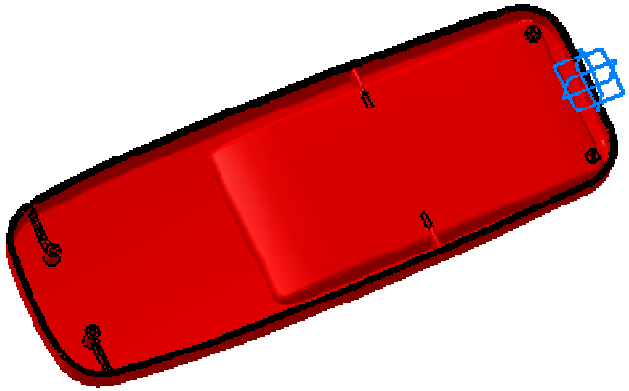




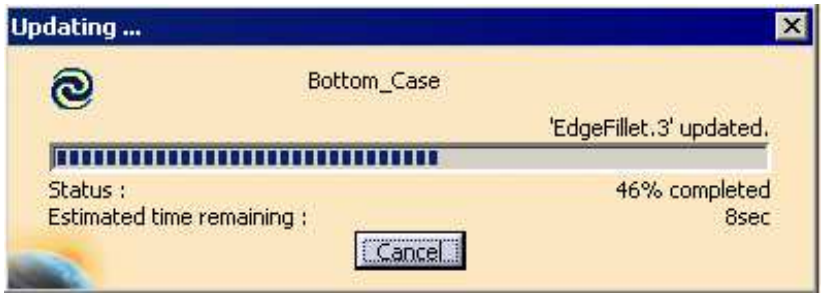
Student Notes:

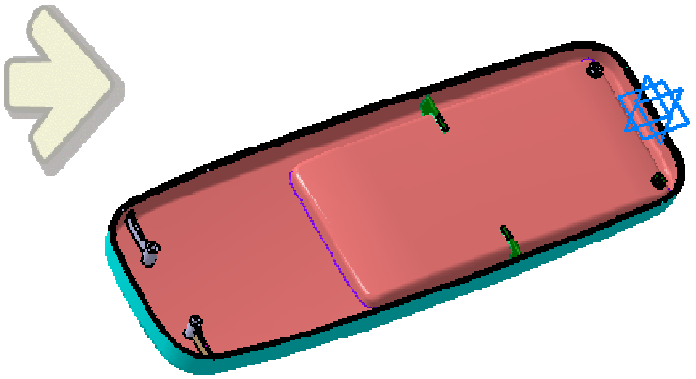
Do It Yourself (2/4)

- Work in 'Bottom_Case_End_Step_3_Modify.CATPart'.
- Update the part due to modification in the linked part.



- Update the part to take into account the modifications done in the linked part.

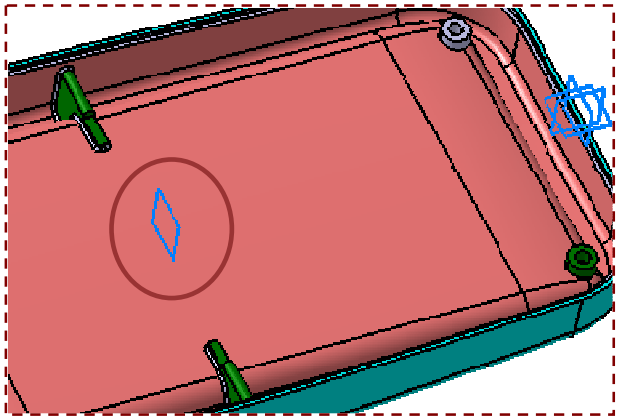


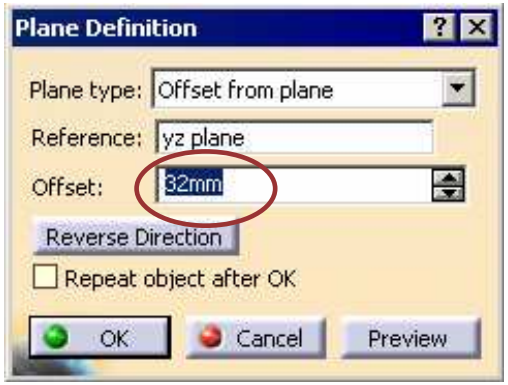


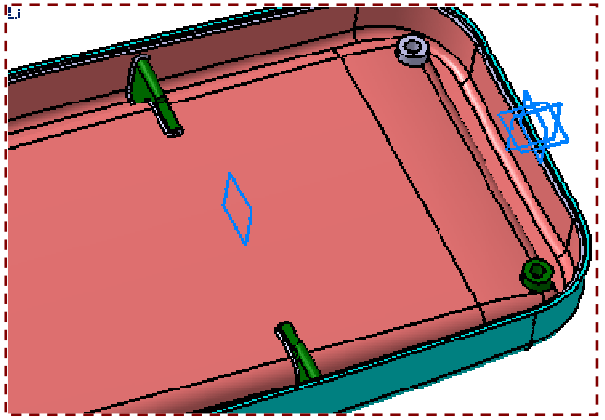
Student Notes:

Do It Yourself (3/4)

- Move Plane.1 which is in 'Construction_Elements' geometrical set from initial offset value of 44 mm to new value of 32 mm.



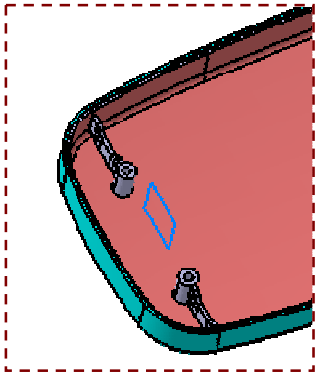


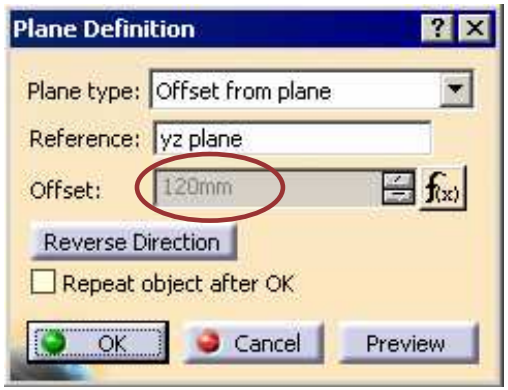


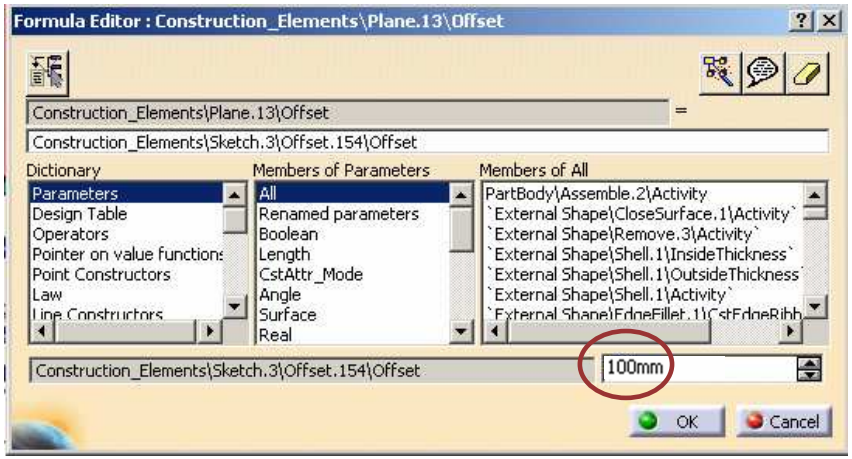
Student Notes:

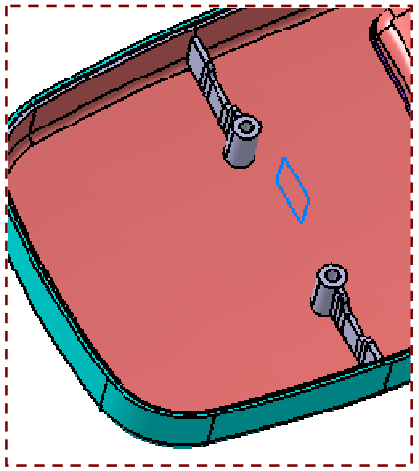
Do It Yourself (4/4)

- Similarly modify the plane on which Stiffeners are created.Move the plane from initial value of 120 mm to 100 mm by modifying the formula.









Result:'Bottom_Case_End_Step_3_Modify_End'. Edit links to open pointed part.

Student Notes:

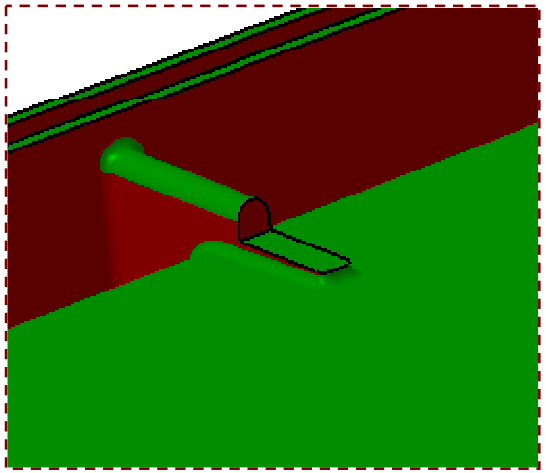
Mobile Phone Bottom Case

Step 4 – Analyze and Modify the Design

10 min

In this step you will analyze the part.

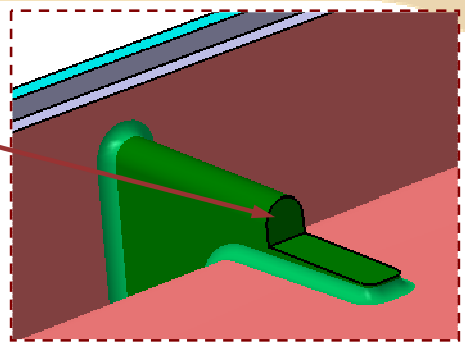
- Perform Draft analysis
- Check whether any draft is applied
- Apply Draft
- Modify the design according to draft analysis
- Re-perform the Draft analysis.



Student Notes:

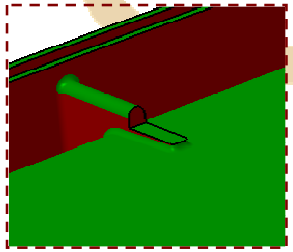
Do It Yourself

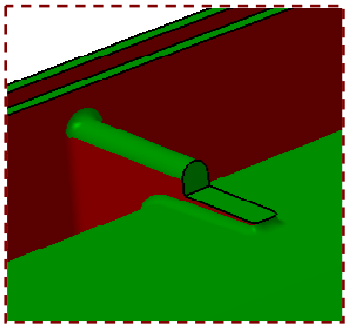
1. Perform draft analysis on the Face shown(Check whether Draft is applied or not)



2. Analyze the result.Apply draft and re-perform Draft analysis.

3. Modify the draft angle accordingly.



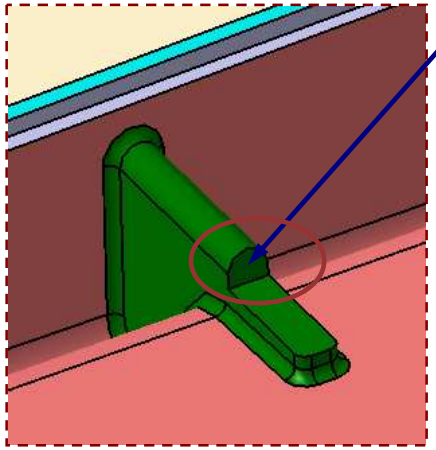


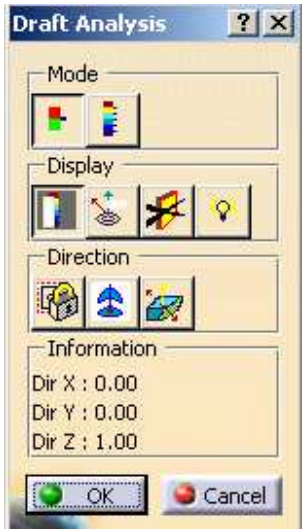
Student Notes:

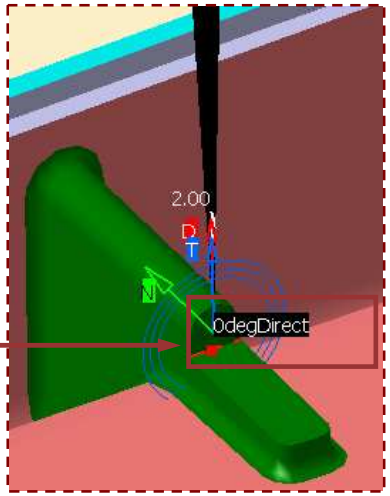
Do It Yourself (1/4)

Load the part 'Bottom_case_End_Step_4_Analyze.CATPart'.
Edit > Links to load linked part

- Select the Face shown to perform Draft Analysis
- Select the Draft analysis command from Analysis toolbar.(Make sure that Material mode is ON)





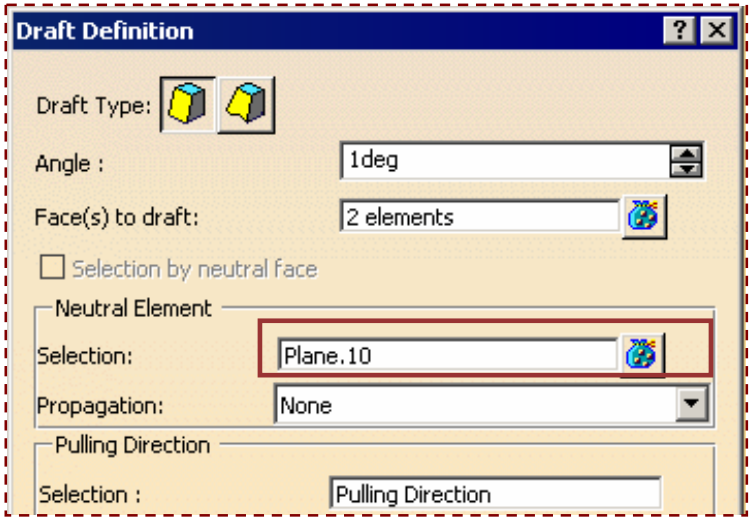


- You observe that 'No Draft' is applied on the Face

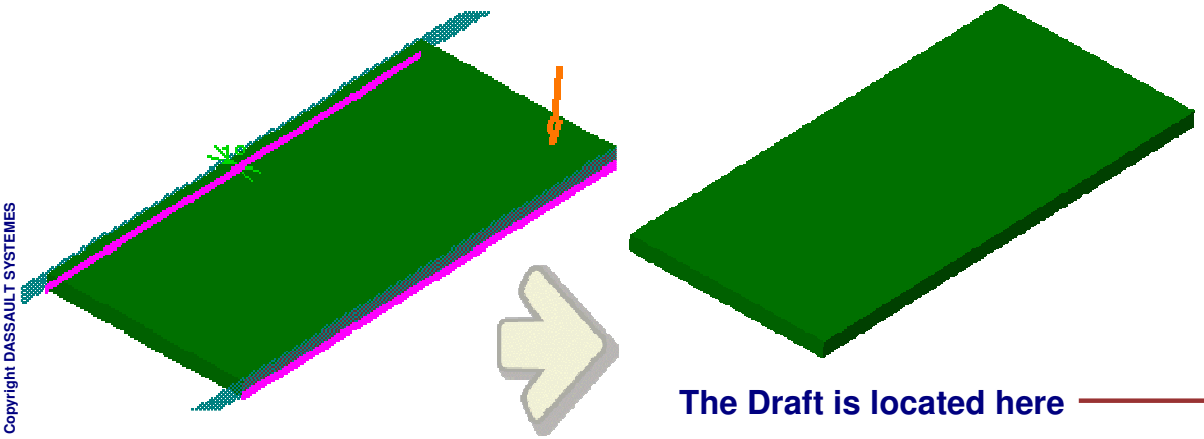
Student Notes:

Do it Yourself (2/4)

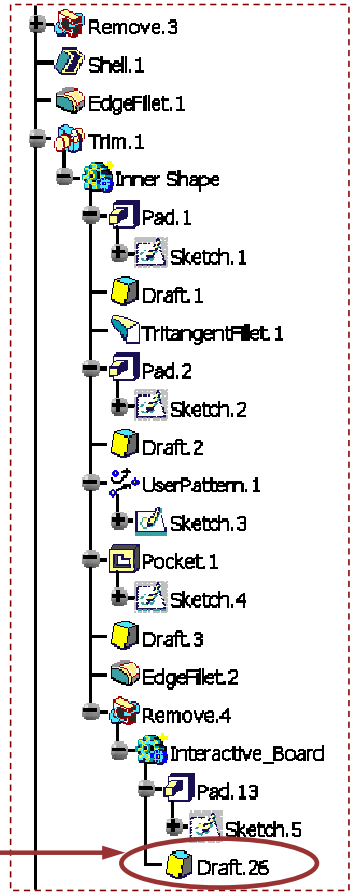
- Since no draft is applied,apply a Draft of 1 deg to the side faces of pad in Interactive Body.



Select neutral element as the plane previously created offset from XY Plane by 2.5 mm.



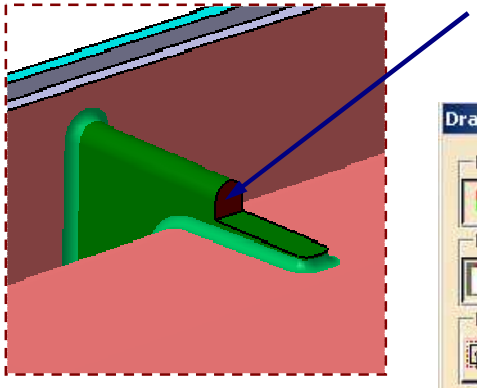
The Draft is located here

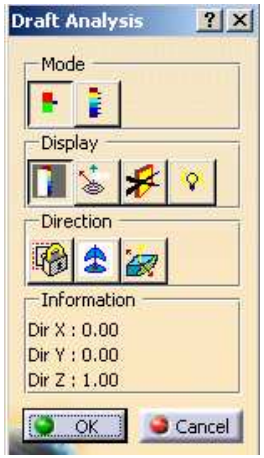


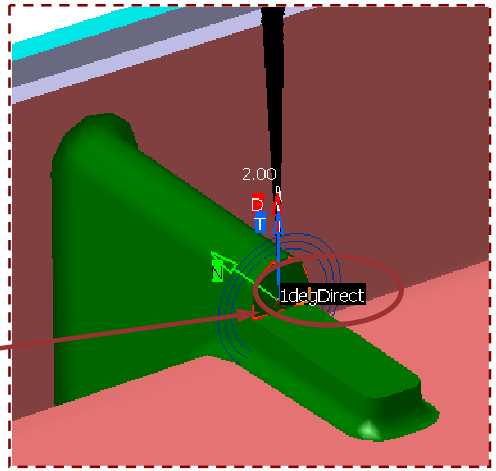
Student Notes:

Do It Yourself (3/4)

- Now you have applied a draft of 1 deg to this face.
- Re-perform the draft analysis.
- Select the Draft analysis command from Analysis toolbar.(Make sure that Material mode is ON)





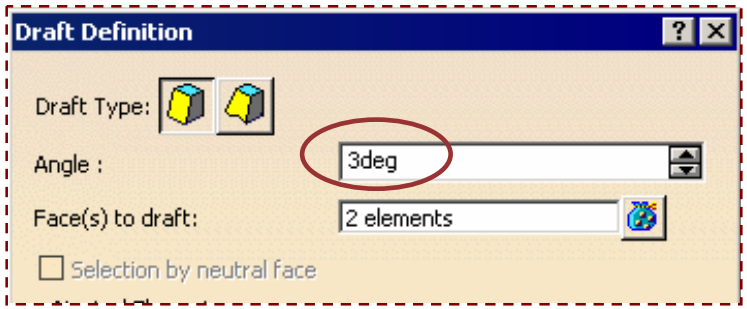


- This Draft of 1 deg is not sufficient to withdraw the part.

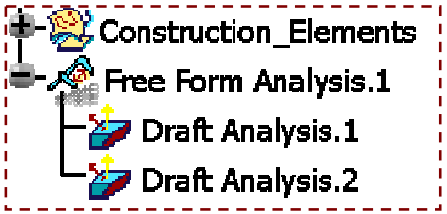
Student Notes:

Do It Yourself (4/4)

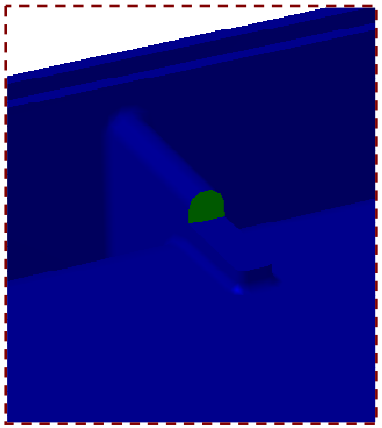
- Since the analysis shows value = 2 deg. To accommodate this value type a value higher than the result shown(I.e 3 deg)

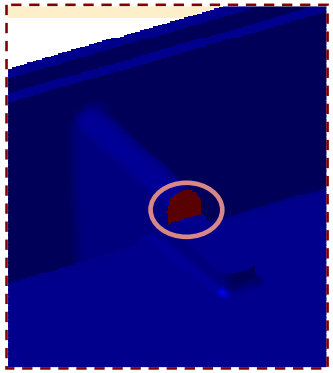


Hence modify the draft definition.
Type a new angle value of 3 deg.

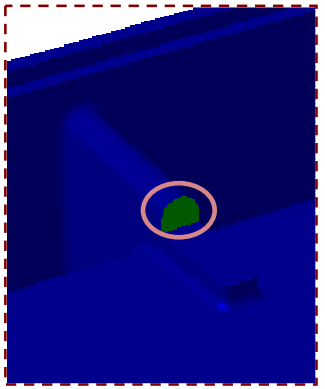


- Now re-perform the Draft analysis.Observe that there is color change in the face selected





Before Modification
(Draft angle =1 deg)



After Modification
(Draft angle =3 deg)

Result: Bottom_case_End_Step_4_Analyze_End.CATPart'. Edit links to open pointed Part.

Student Notes:

Meat Mincer Screw

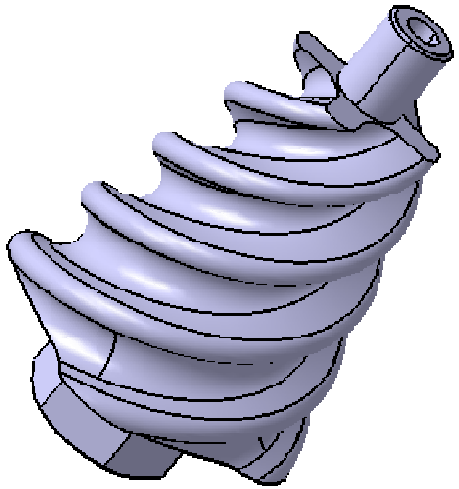
Part Design Advanced Exercise

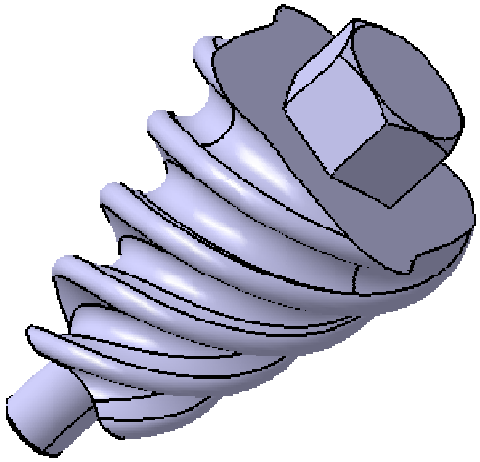
30 min

In this exercise you will build the Meat Mincer Screw by following a recommended process.

You will then study its Drawing in detail to understand the dimensions and specifications.

Finally,you will design Meat Mincer screw using Sketcher,Part Design, and Wireframe and Surface Design workbenches.

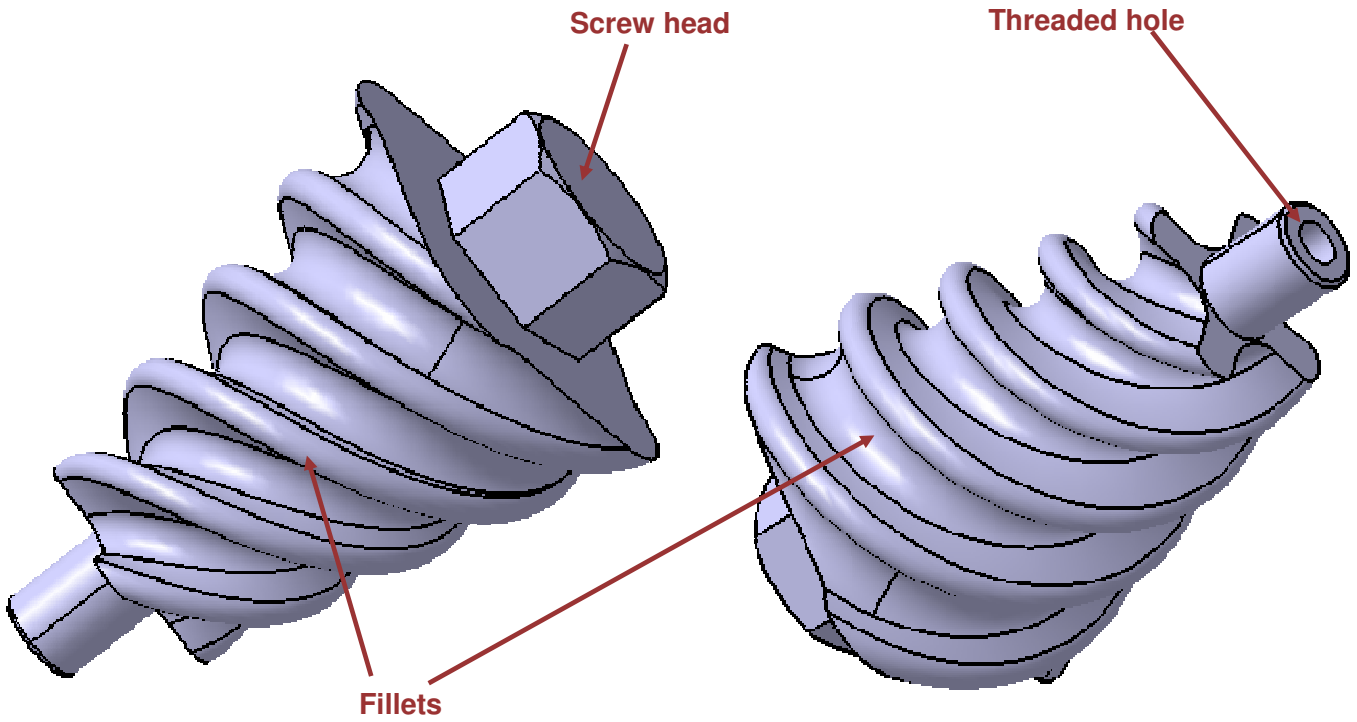




- Create Multi- sections solid along helixes.
- Apply Fillets.
- Design screw head.
- Create groove
- Apply chamfer

Student Notes:

Design intent: Meat Mincer Screw

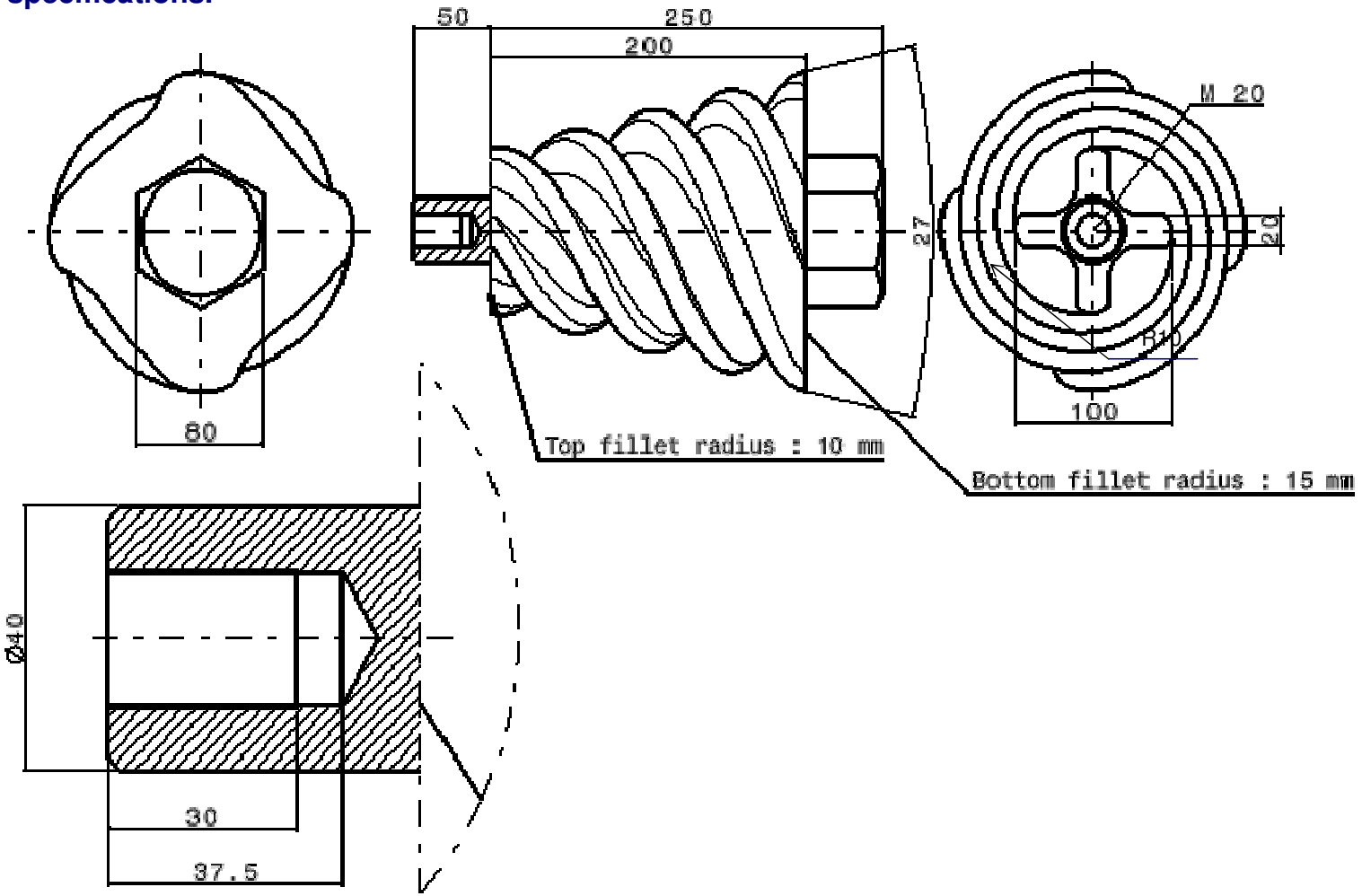


- Screw Head is used for clamping the screw using spanner.
- Threaded hole is used to clamp the screw to the Assembly.

Student Notes:

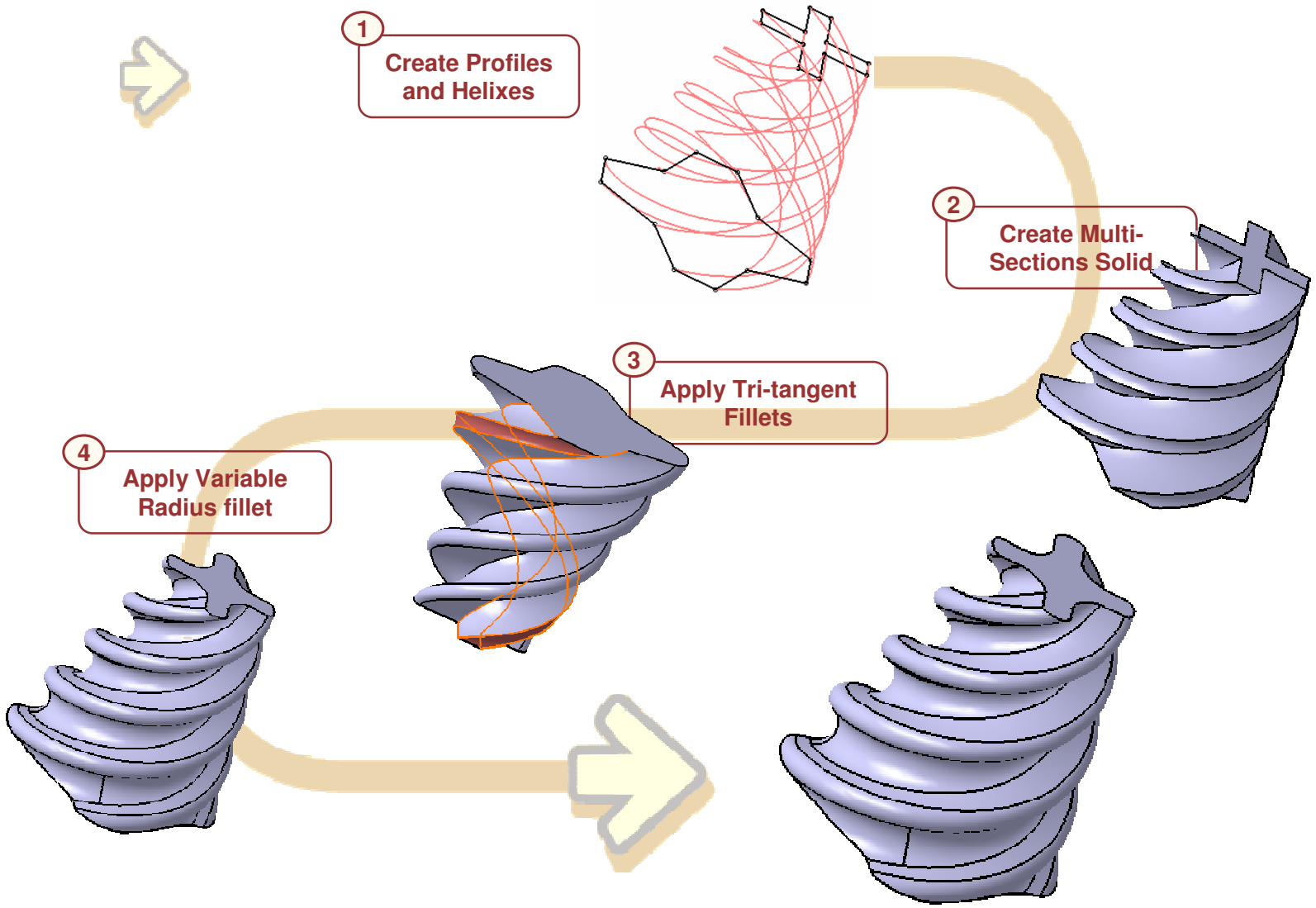
Meat Mincer Screw Drawing

- Understand the drawing thoroughly to design the part according to the specifications.



Student Notes:

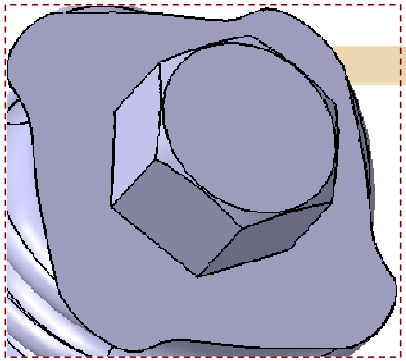
Design process: Meat Mincer Screw (1/2)



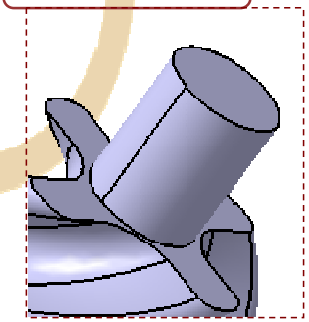
Student Notes:

Design process: Meat Mincer Screw (2/2)

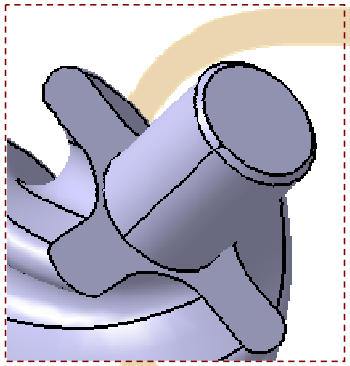
5 Design the Screw head and groove it.



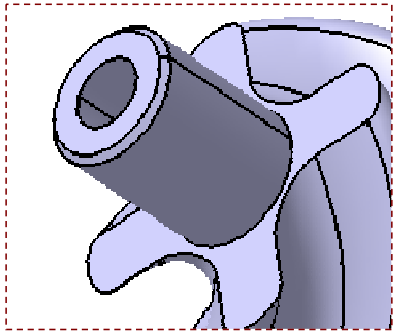
6 Create Pad



7 Apply Chamfer



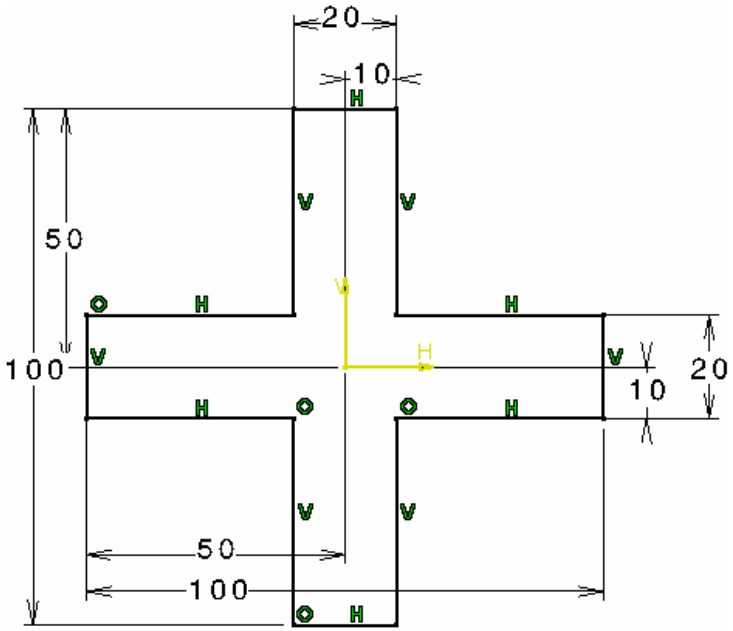
8 Create a threaded hole

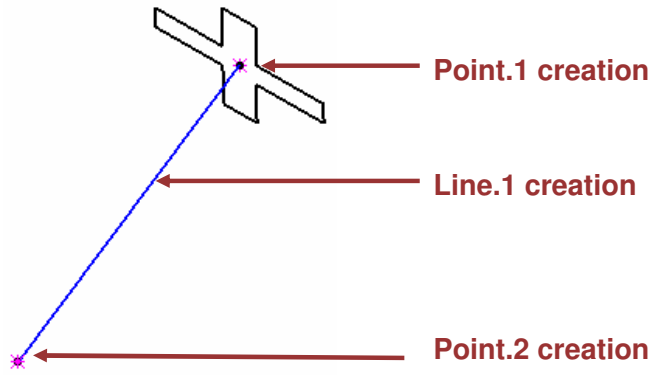


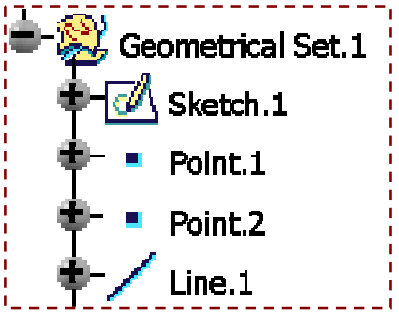
Student Notes:

Step 1: Create Profiles and Helixes (1/3)

- Create a sketch in XY plane in Geometrical set as shown.
- Create a wireframe point at origin.
- Create another wireframe point at (0,0,200).
- Create a wireframe line joining these two points



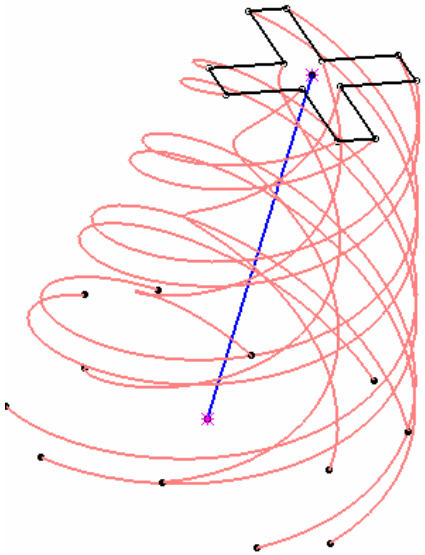


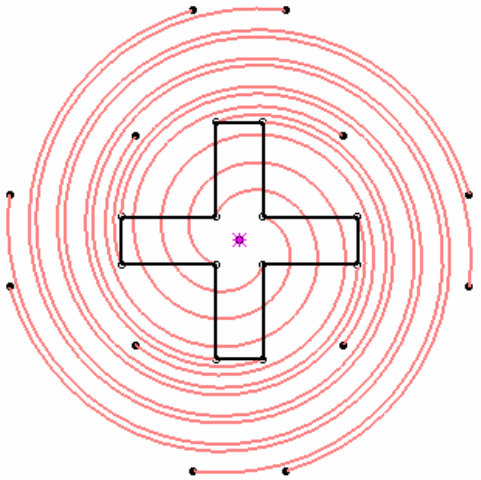


Student Notes:

Step 1: Create Profiles and Helixes (2/3)

- Create a helix in Generative Shape Design workbench.
- Create twelve helixes,each helix starting from different vertices of sketch.1
- Parameters for helix creation are:
  - Starting point: Twelve different vertices of Sketch.1 for twelve different Helixes.
  - Axis: Line.1
  - Pitch: 199.5 mm
  - Height: 200 mm
  - Orientation: Counter-Clockwise
  - Starting Angle: 0 deg
  - Taper Angle: -13.5 deg.
  - Way: Inward



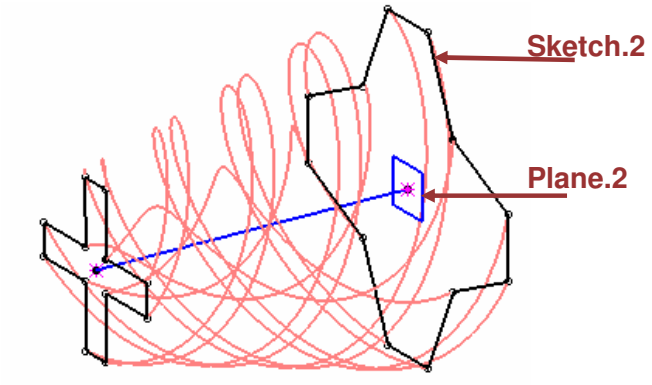


View Normal to sketch Plane

Student Notes:

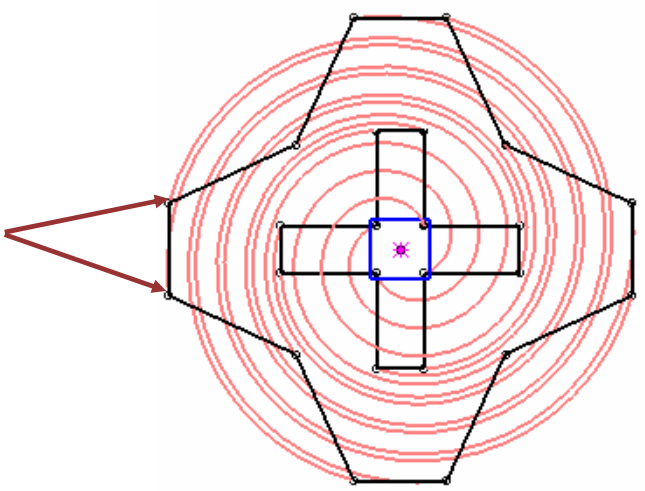
Step 1: Create Profiles and Helixes (3/3)

- Create a plane parallel to XY Plane and passing through Point.2.
- Create a sketch on this plane.
- Constrain the sketch with the endpoints of the helixes (use coincidence constraint).



Intersect the sketch endpoints with the sketch Plane.Then, create sketch.2 joining all these points.

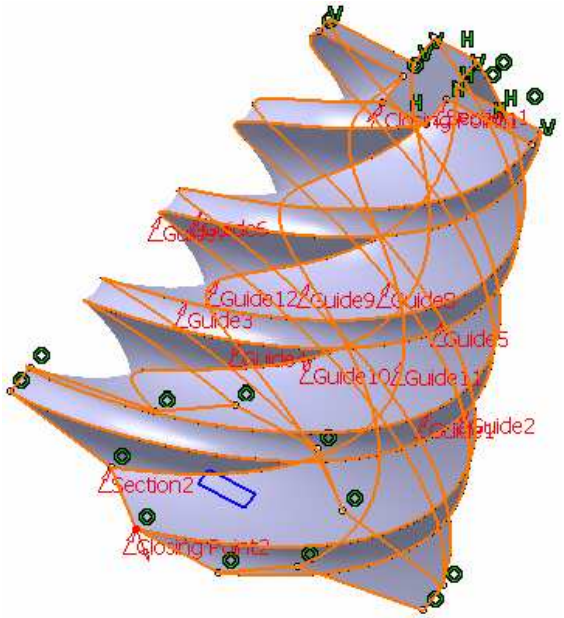
View Normal to Plane.1

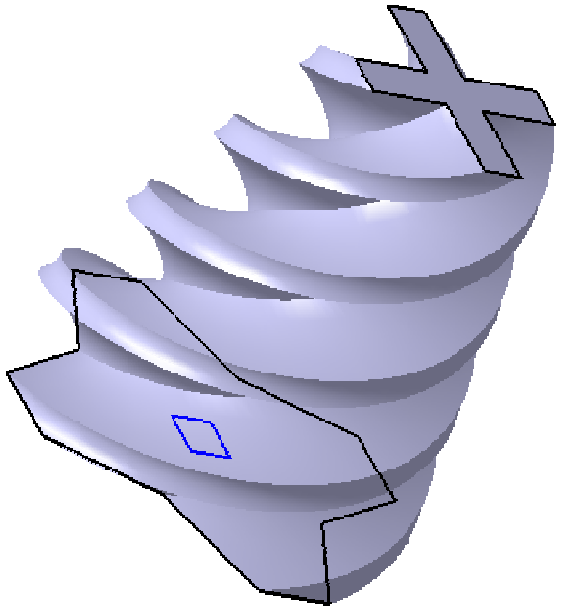


Student Notes:

Step 2: Create Multi-Sections Solid

- Create a Multi-Sections Solid, Using Sketch.1 and Sketch.2 as sections and twelve helixes as guiding curves
- Check for closing points and their orientations.

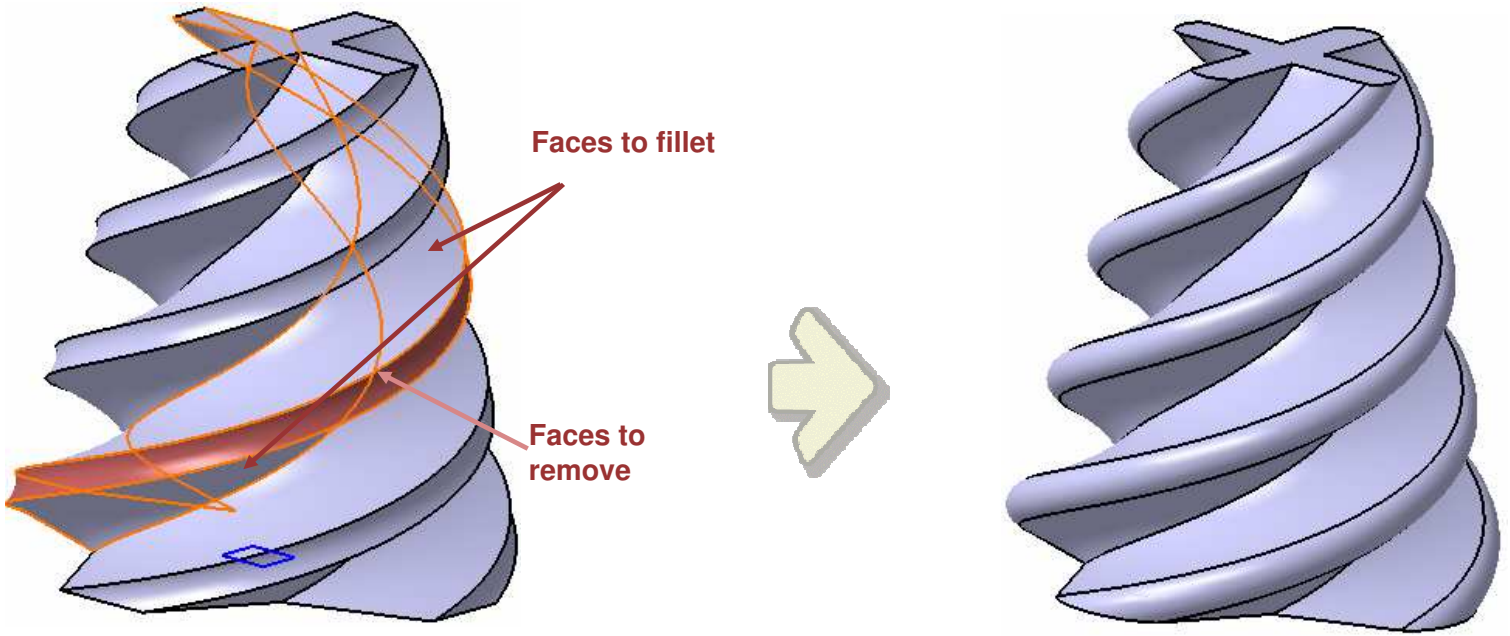




Student Notes:

Step 3: Apply Tri-Tangent Fillets

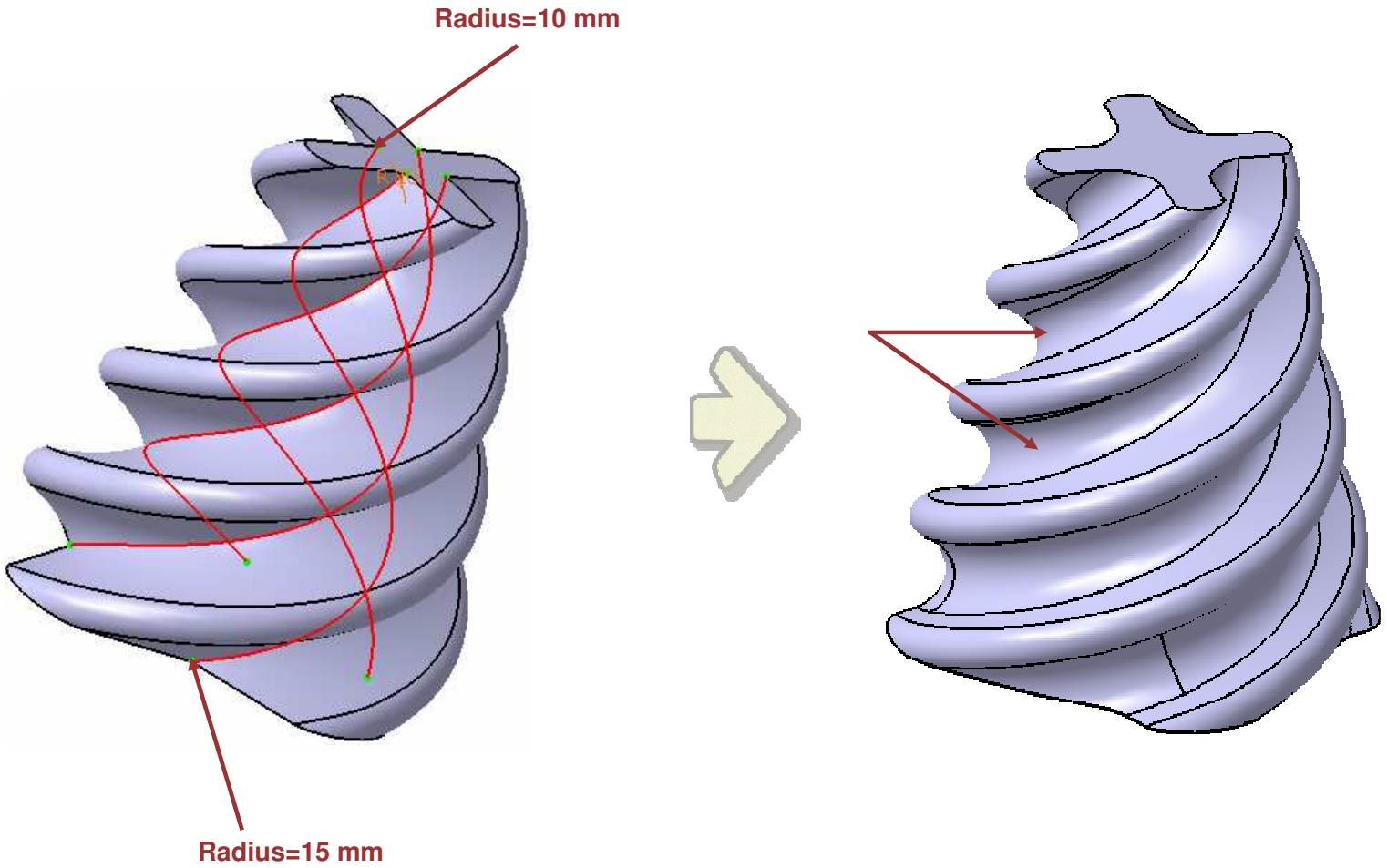
- Apply Tri-Tangent Fillet.



Student Notes:

Step 4: Apply Variable Radius Fillet

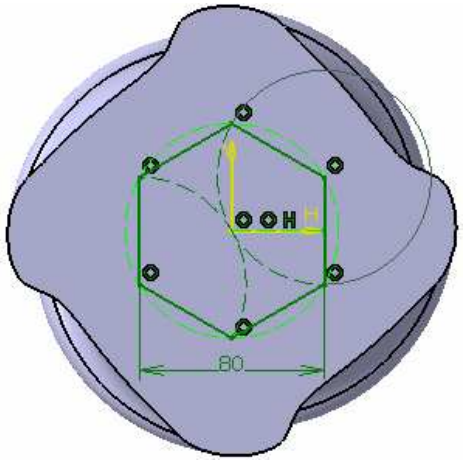
- Apply a variable radius fillet to the four edges. Radius at top =10 mm and radius at bottom = 15 mm.

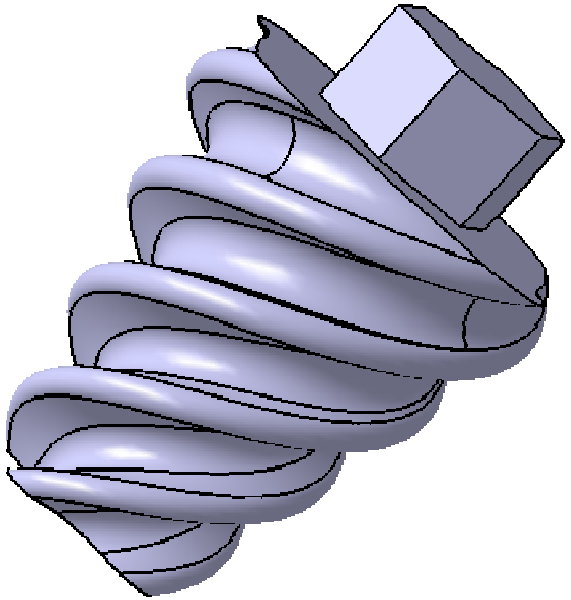


Student Notes:

Step 5: Design the Screw Head (1/2)

- Create a Hexagon on the Multi sections solids face as shown.
- Pad it by 50 mm.

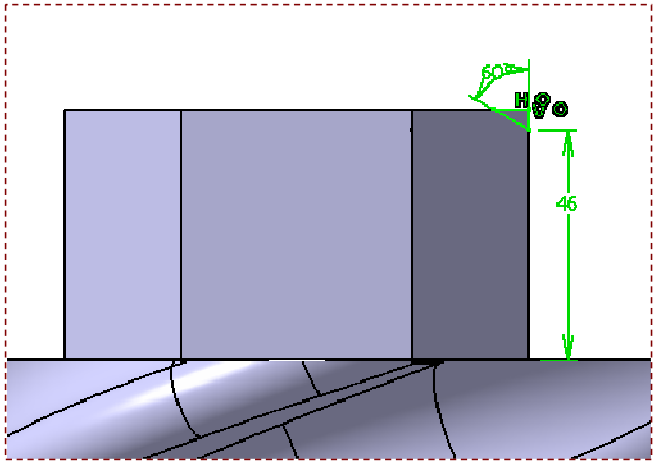


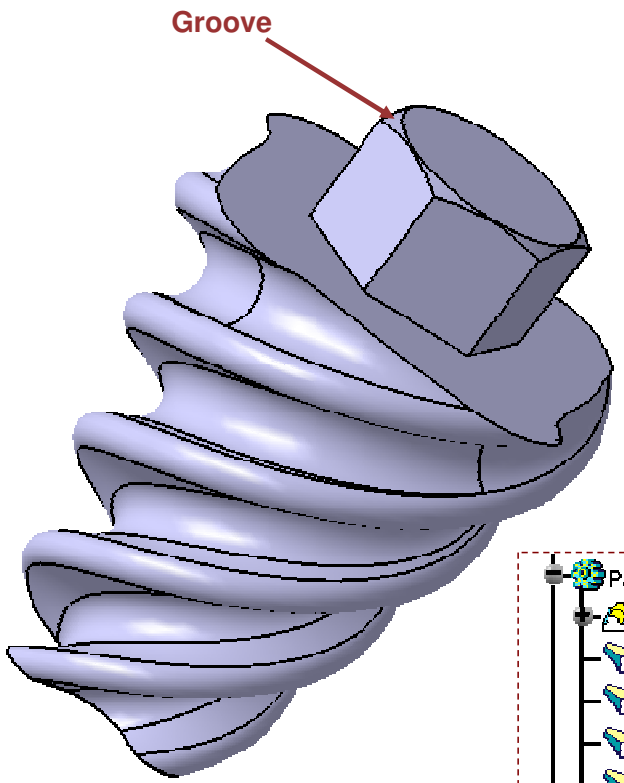


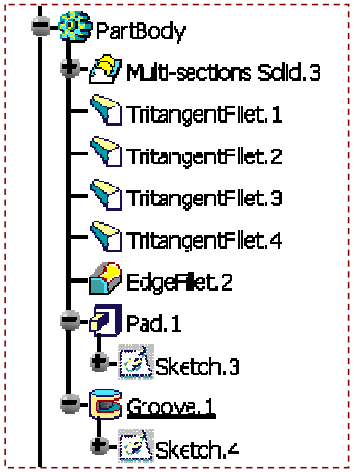
Student Notes:

Step 5: Design the Screw Head (2/2)

- Create a sketch as shown on YZ Plane
- Groove it around the axis as Line.1.



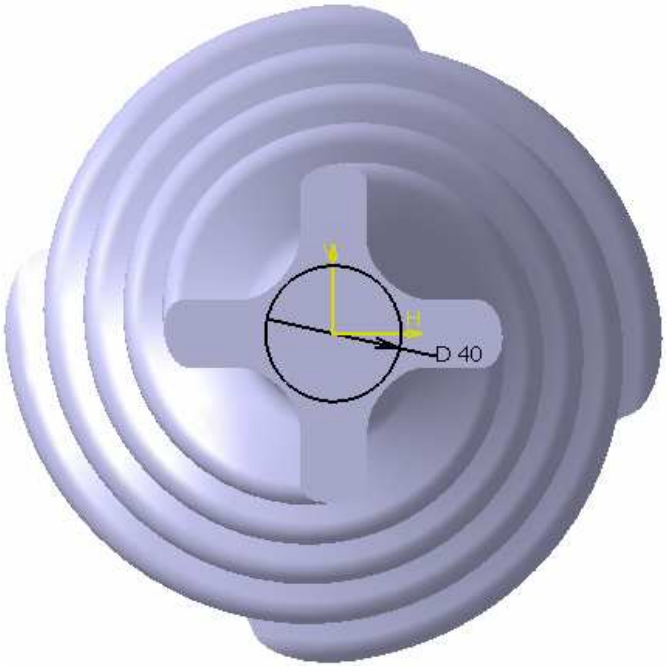


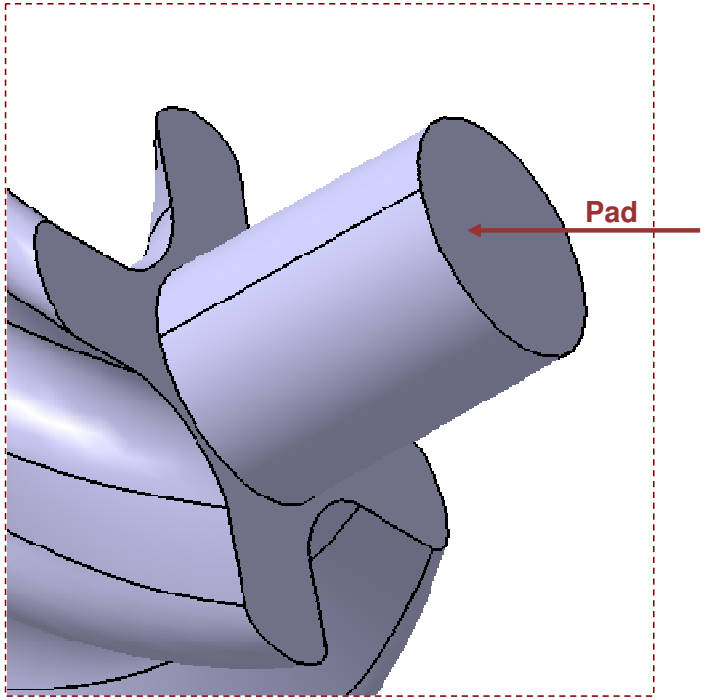


Student Notes:

Step 6: Create Pad

- Create a sketch as shown on Multi-section Solids face
- Pad it by 50 mm.

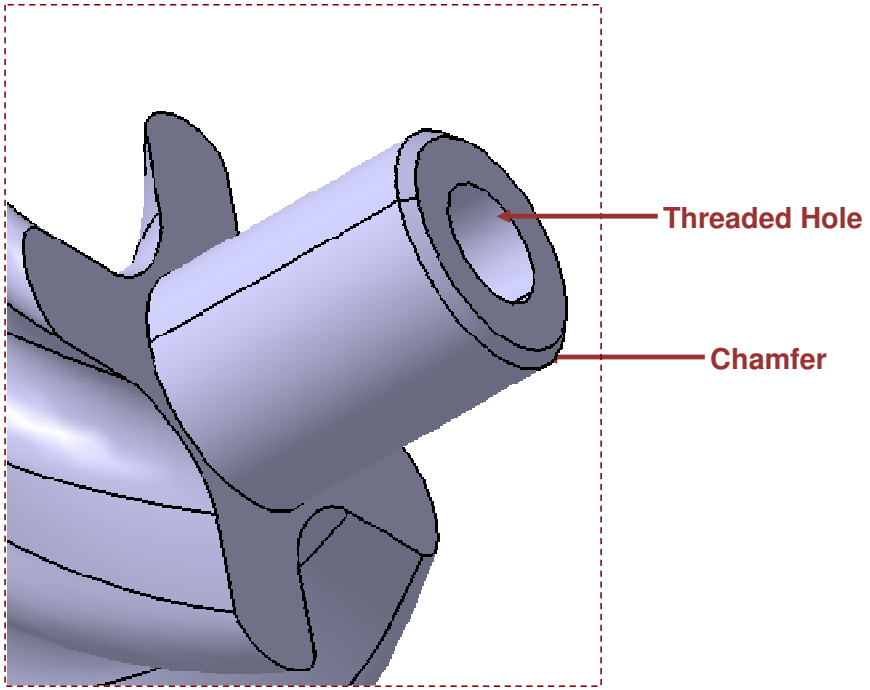




Student Notes:

Step 7 & 8: Applying a Chamfer and Creating a Threaded Hole

- Apply a Chamfer of 2 x 45 deg on edge shown.
- Create a Simple Blind Hole concentric to Pad.2 with following parameters:
  - Threaded Hole with type: Metric Thin Pitch
  - Thread Description: M22
  - Hole depth: 30 mm
  - Thread Depth: 10 mm
  - Right Threaded
  - V lower with angle of 120 deg.



Result: PDG_Meat_Mincer_Screw.CATPart

Student Notes:

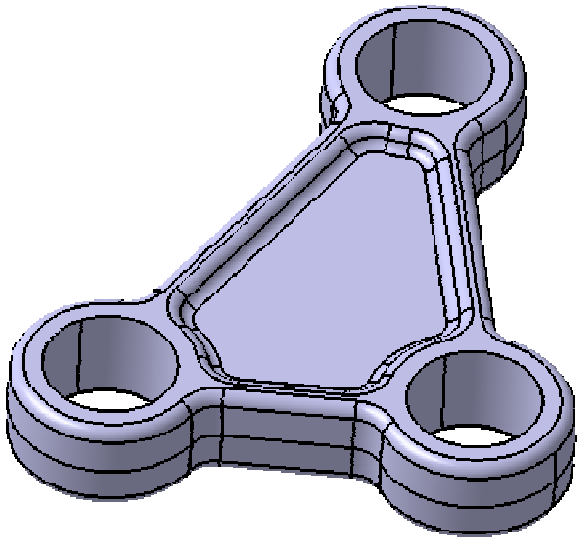
Hinge

Part Design Advanced Exercise

50 min

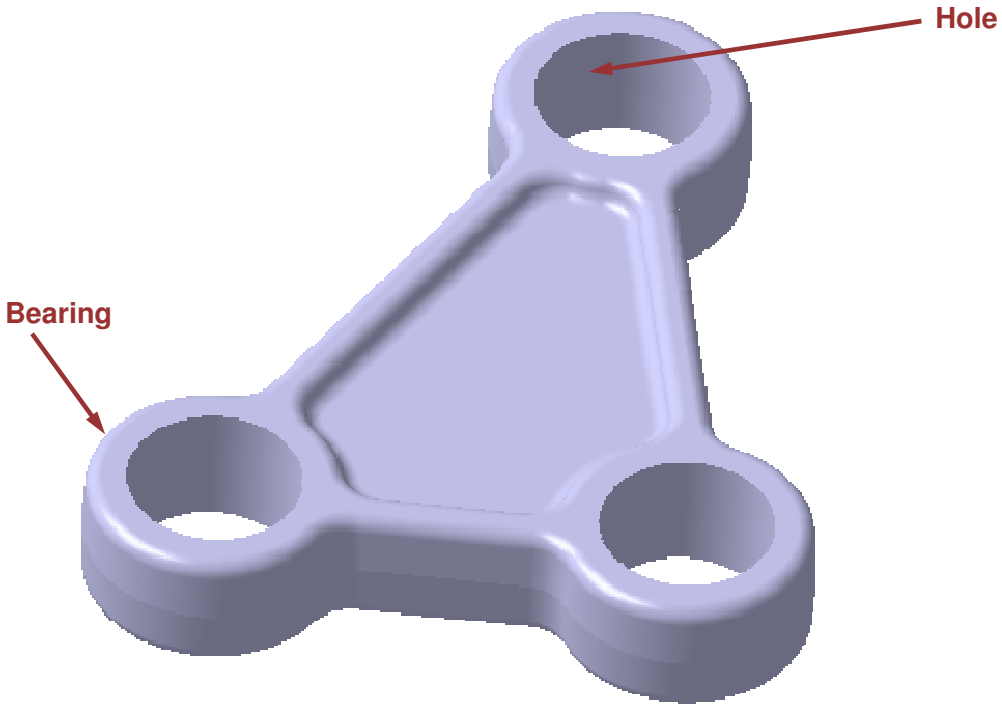
In this exercise you will build the Angle Bracket by following a recommended process.

- **You will first understand the design intent of the Hinge and identify its functional features.**
- **You will then study its Drawing in detail to understand the dimensions and specifications.**
- **Finally, you will design the various functional features of the Hinge according to specifications and by making use of wireframe elements.**



Student Notes:

Design intent: Hinge

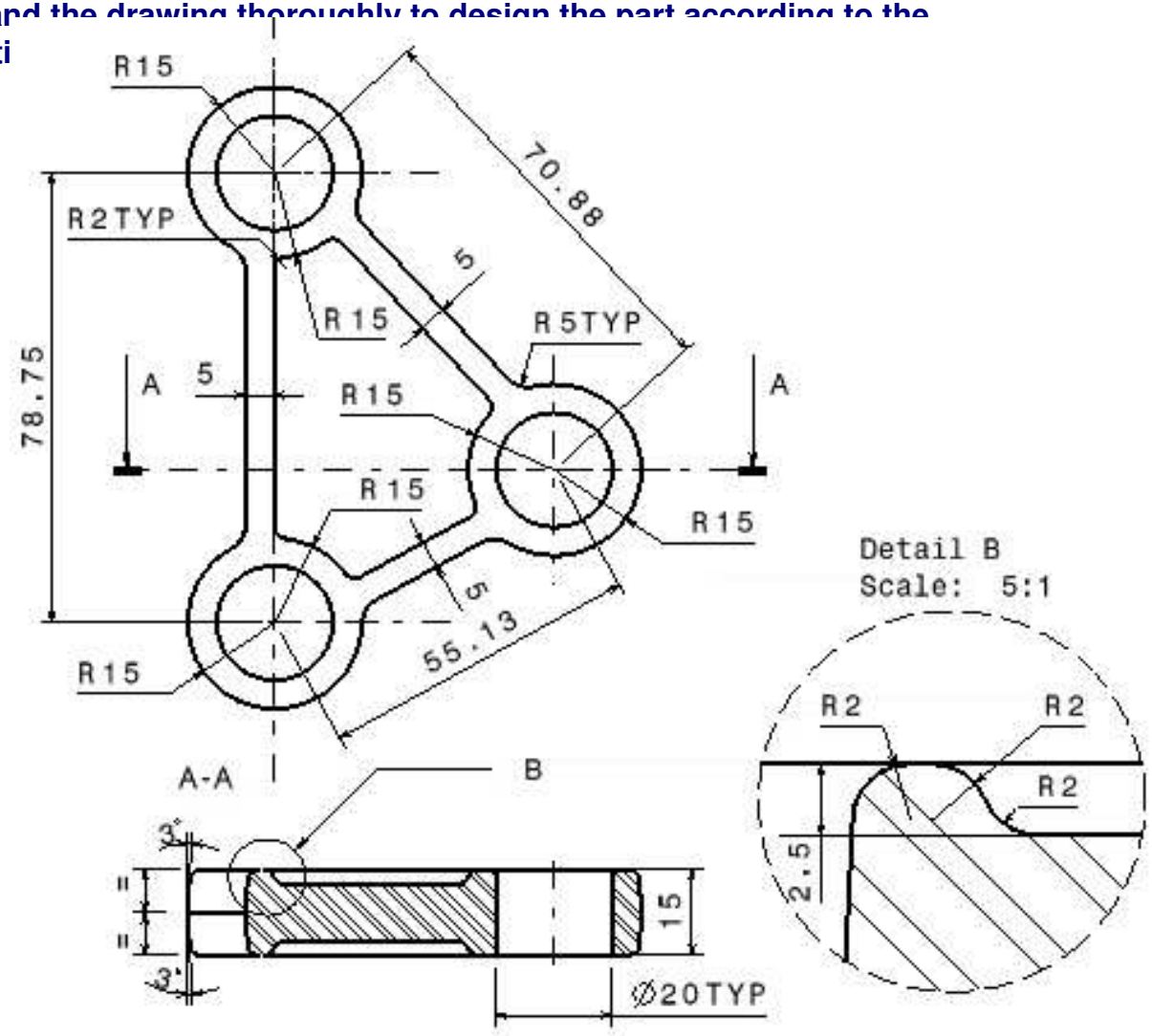


- The Hinge is a molded part that is used in an assembly
- The part is symmetrical
- The Holes are centered on the bearings

Student Notes:

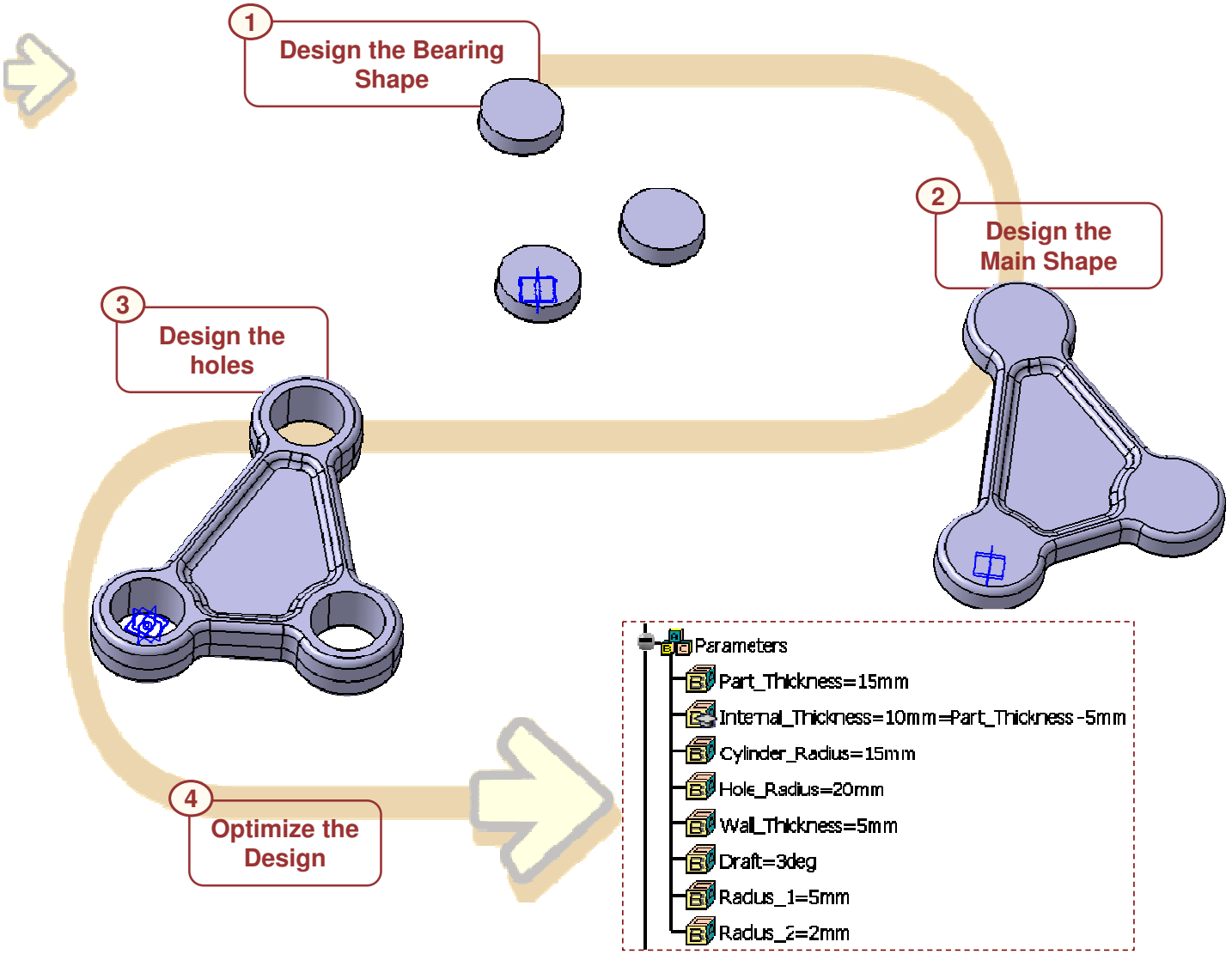
Hinge Drawing

- Understand the drawing thoroughly to design the part according to the specificati



Student Notes:

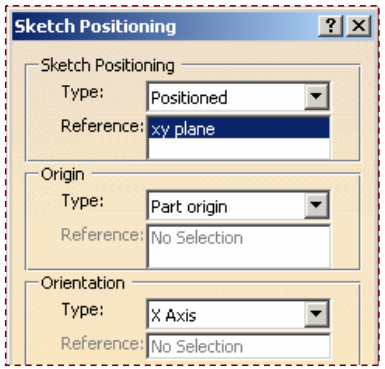
Design process: Hinge

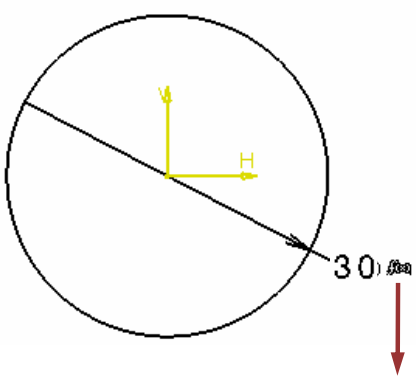


Student Notes:

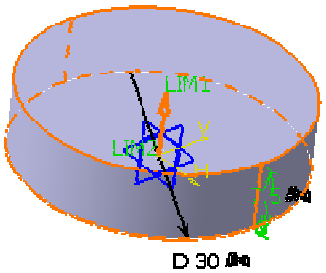
Step 1: Design the Bearing Shape (1/3)

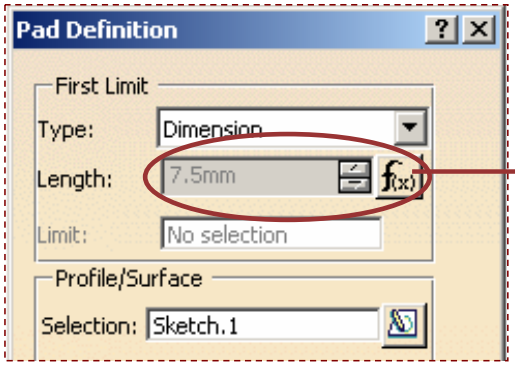
- Insert a 'Body.2'
- Create parameters as shown below
- In the Geometrical set create a positioned sketch on XY plane as shown.
- Pad it by 7.5 mm





The Diameter value is driven by Cylinder_Radius parameter.Value of this parameter is initially set to 15 mm.



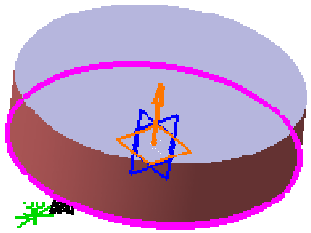


This value is half the 'Part_Thickness'Parameter.

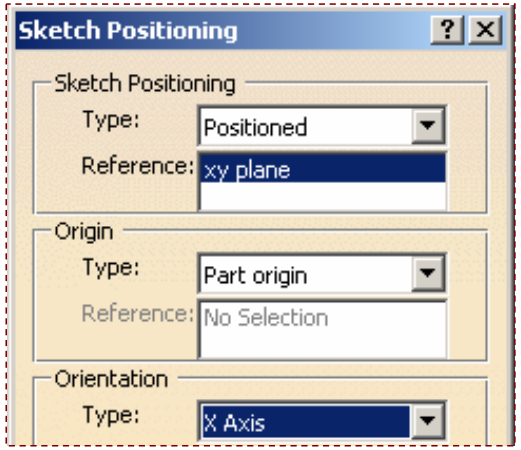
Student Notes:

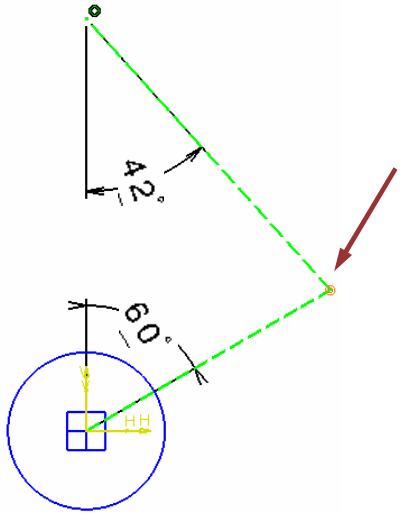
Step 1: Design the Bearing Shape (2/3)

- Apply a draft of 3 deg
- In the Geometrical set, create a point (0,78.75,0). This is Point.1
- Create a Sketch in Geometrical set. This is Sketch.2. It will be used to create Sketch.3



This value is equal to the value specified in 'Draft'Parameter.



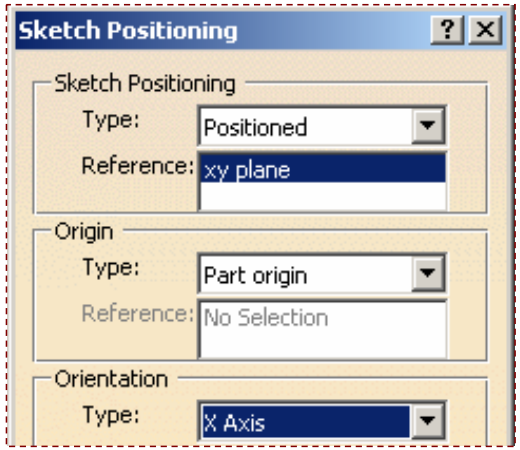


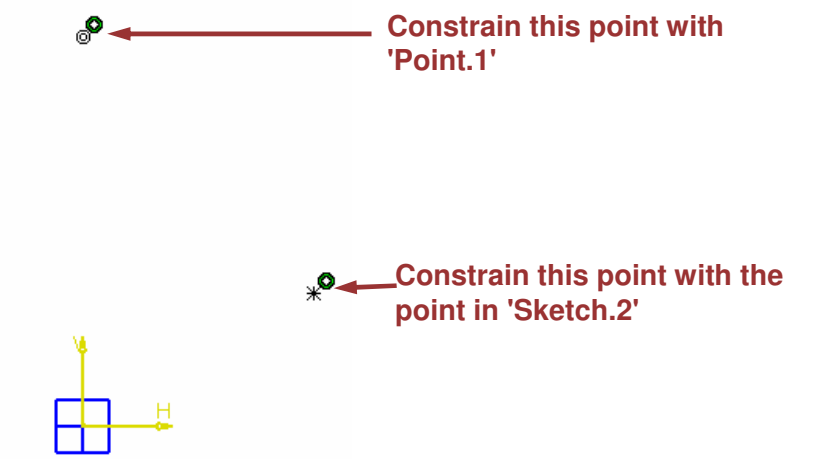
This sketch consists of only one Point. Create the construction lines to create this point.

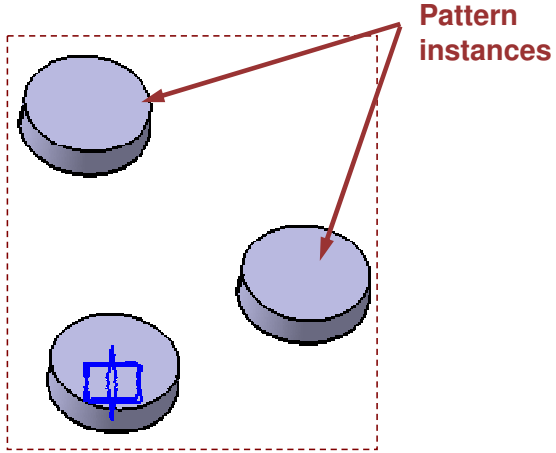
Student Notes:

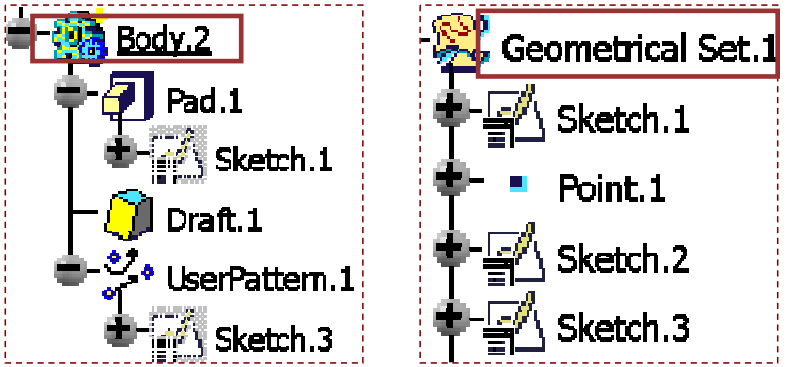
Step 1: Design the Bearing Shape (3/3)

- Create sketch.3.This Sketch consists two points. Using Point.1 and Sketch.2 to constrain this sketch.
- Create a User pattern from Pad.1 and sketch.3







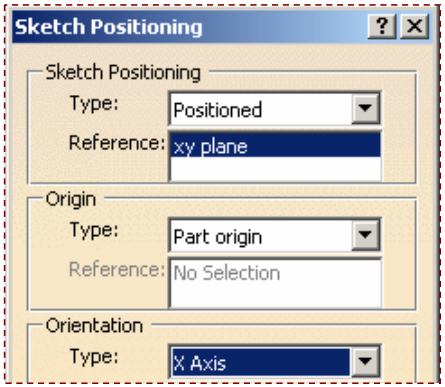


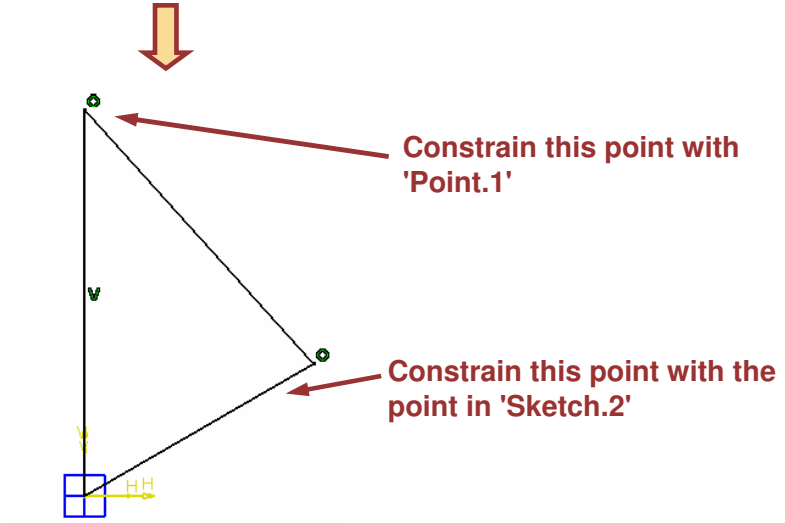
Student Notes:

Step 2: Design the Main shape (1/3)

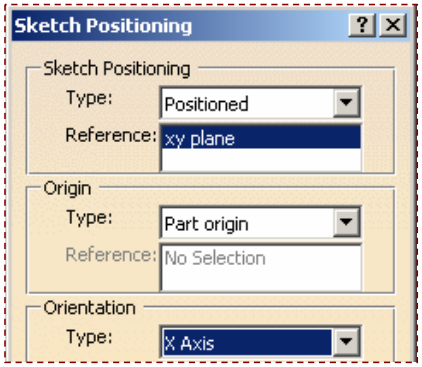
- Create a Positioned sketch on XY plane in geometrical set. This is sketch.4
- Create another sketch as shown.This is sketch.5.Use Sketch.4 to constrain this sketch.5

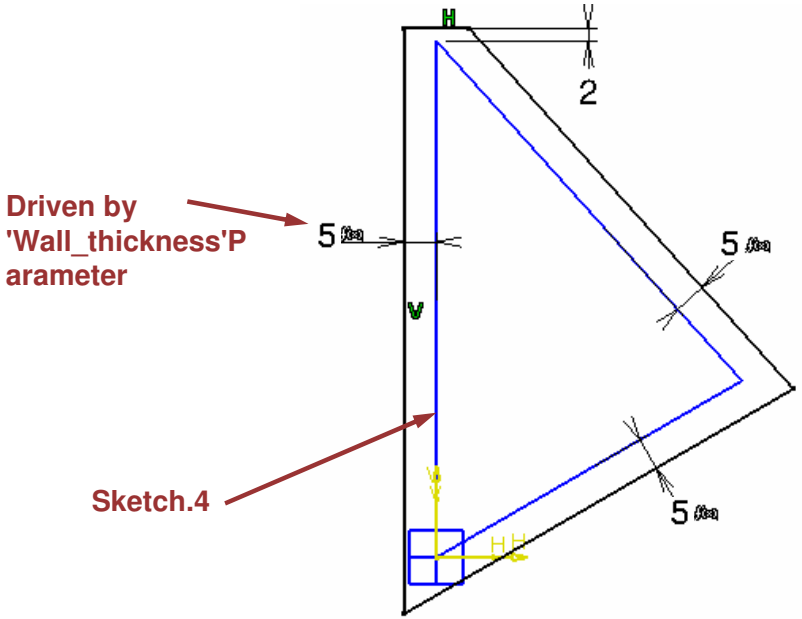
Sketch.4





Sketch.5

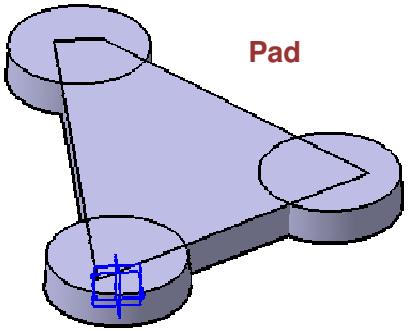


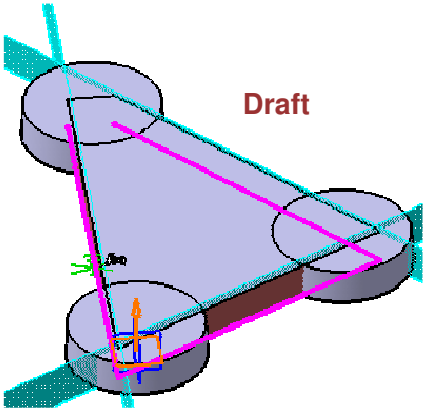


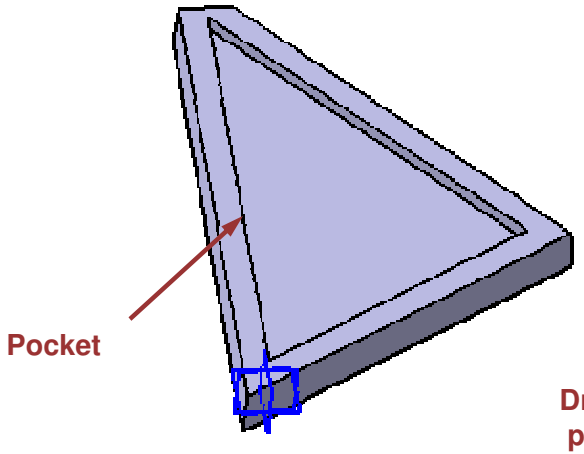
Student Notes:

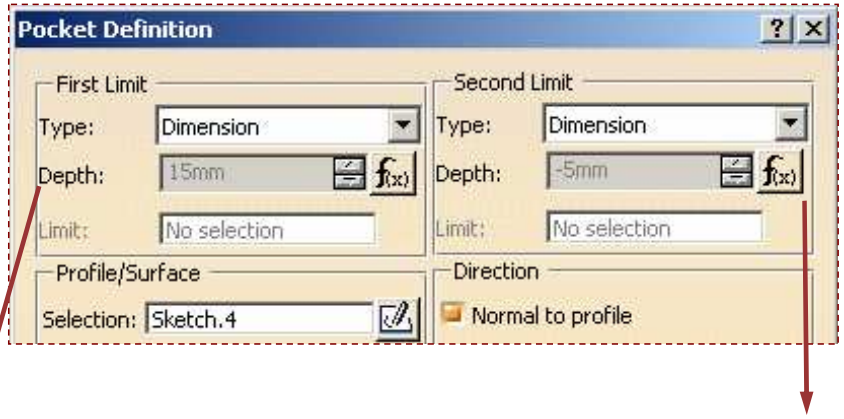
Step 2: Design the Main shape (2/3)

- Insert 'Body.3'
- Pad Sketch.5. Pad length is half of 'Part_Thickness' parameter. This is pad.2
- Apply draft of 3 deg driven by 'Draft Parameter'
- Create Pocket using Sketch.4









Driven by 'Part_Thickness' parameter

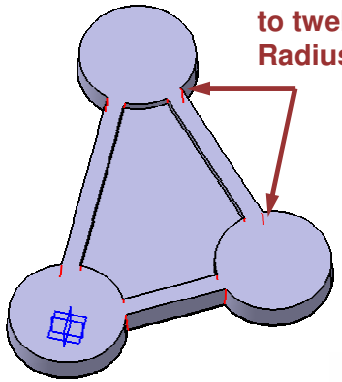
Driven by relation: 'Internal_Thickness' - 'Part_Thickness'

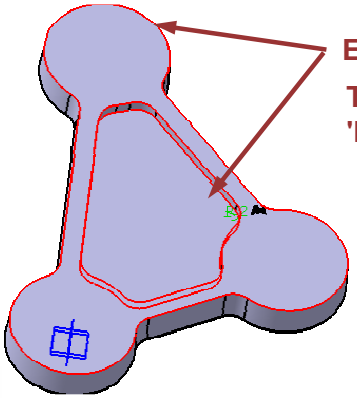
Student Notes:

Step 2: Design the Main shape (3/3)

- Insert 'Body.4'
- Assemble 'Body.2' and 'Body.2' into it
- Assemble 'Body.4' to Part Body
- Apply Edge fillets

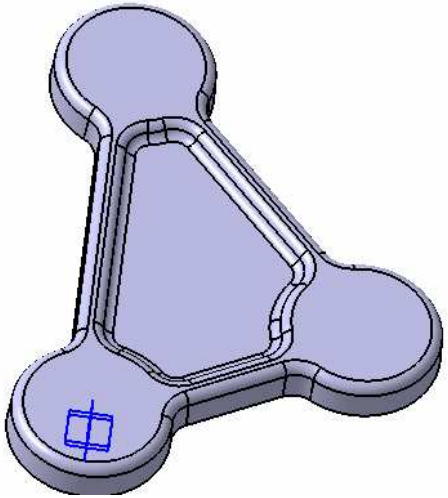
Edge fillet of 5 mm
to twelve edges. Link it with
Radius_1 Parameter



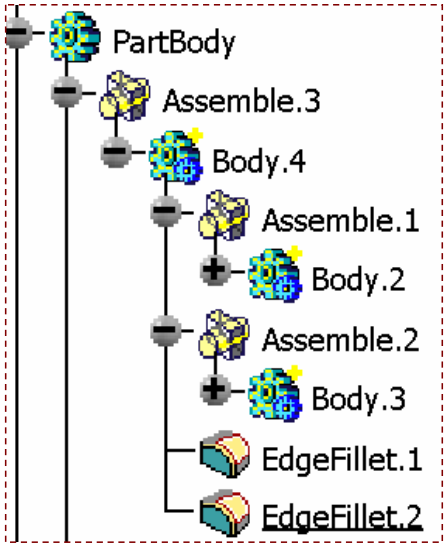


Edge fillet of 2 mm to two faces

This fillet value is driven by
'Radius_2' parameter



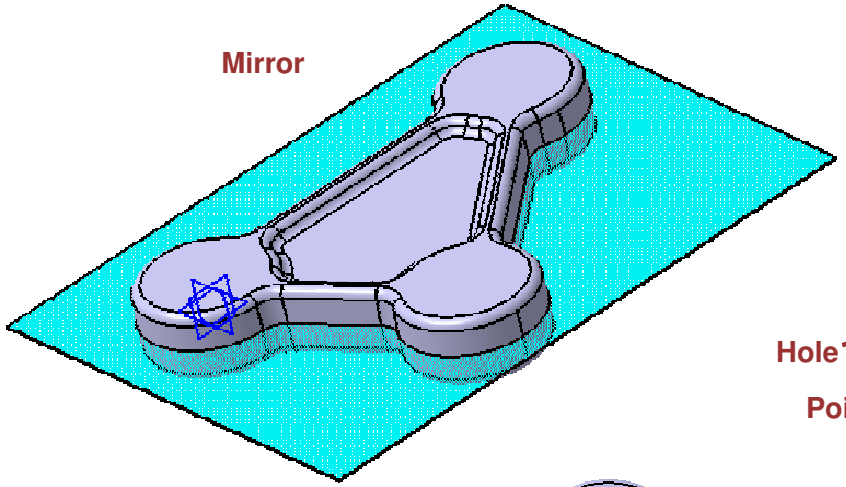
The Result

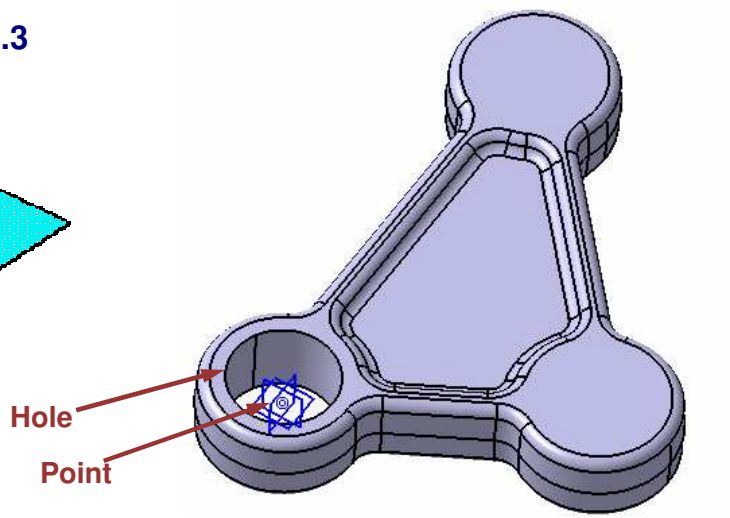


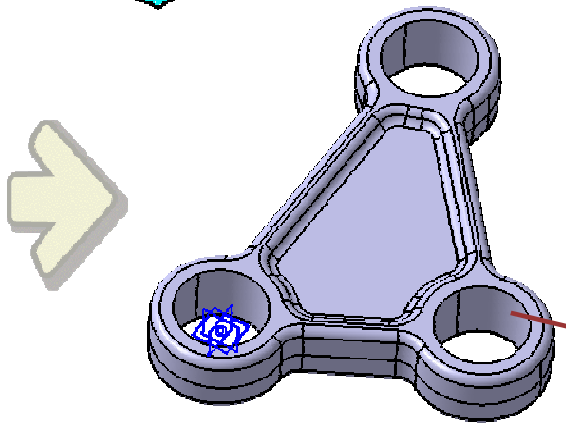
Student Notes:

Step 3: Design the Holes

- Mirror the whole part about XY Plane
- Create a up to last Hole of 20mm diameter concentric with point at origin (create a Point at origin). This is Point.2.
- Create User Pattern of the hole using sketch.3





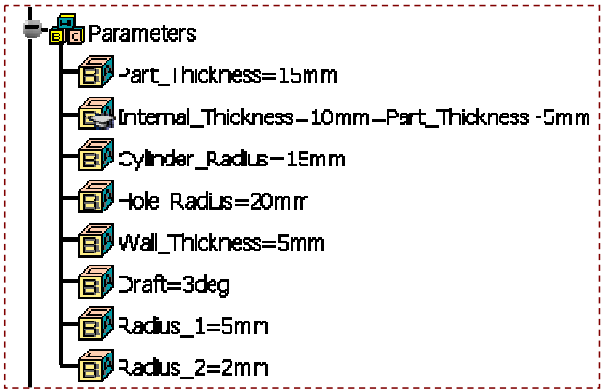


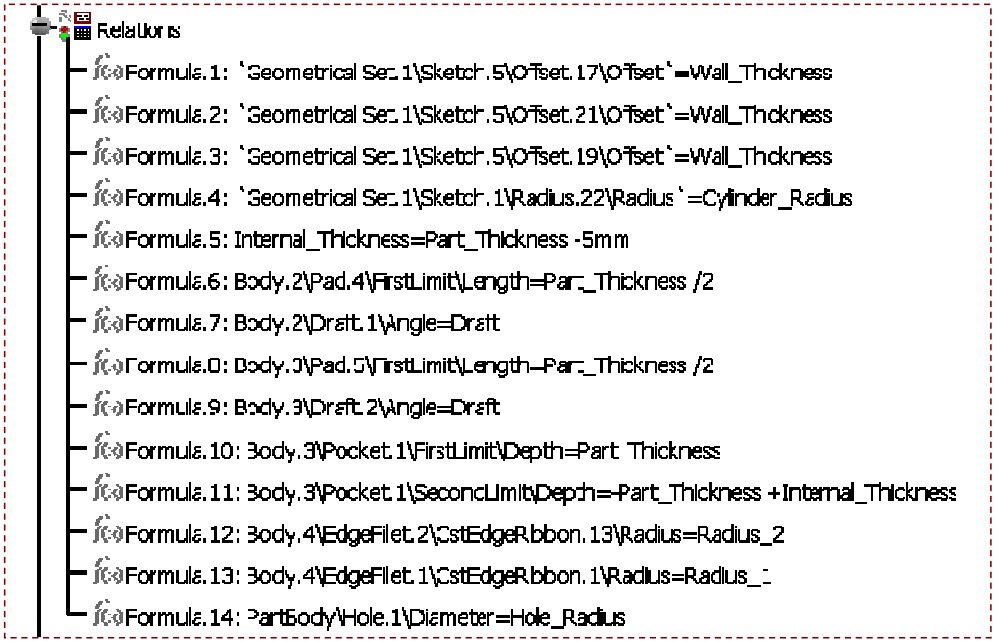
Hole Diameter Value Driven By Parameter 'Hole_Radius'. Its initial Value is set to 20 mm.

Student Notes:

Step 4: Optimize the Design

- You can modify your Design by manipulating the values of the parameters





Result: PDG_Hinge.CATPart

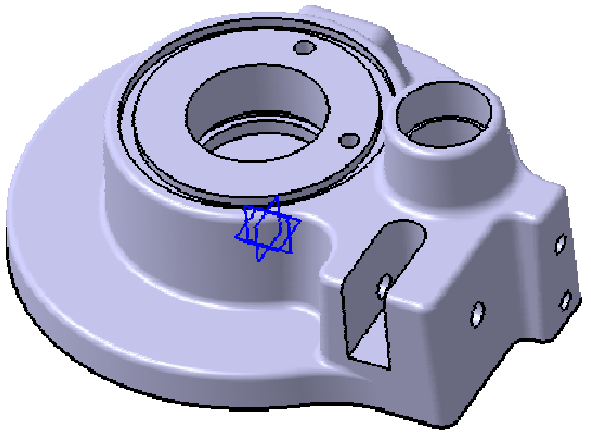
Car Jack Support

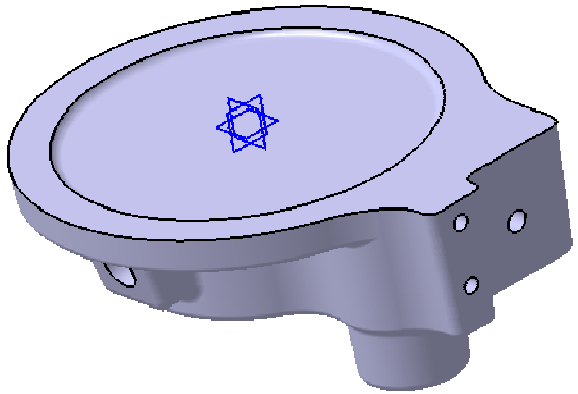
Part Design Fundamental Exercise

60 min

In this exercise you will build the Car Jack Support by following a recommended process.

- You will first understand the design intent of the Car Jack Support and identify its functional features.
- You will then study its Drawing in detail to understand the dimensions and specifications.
- Finally,you will design the various functional features of the Car Jack Support according to specifications and by making use of wireframe elements.

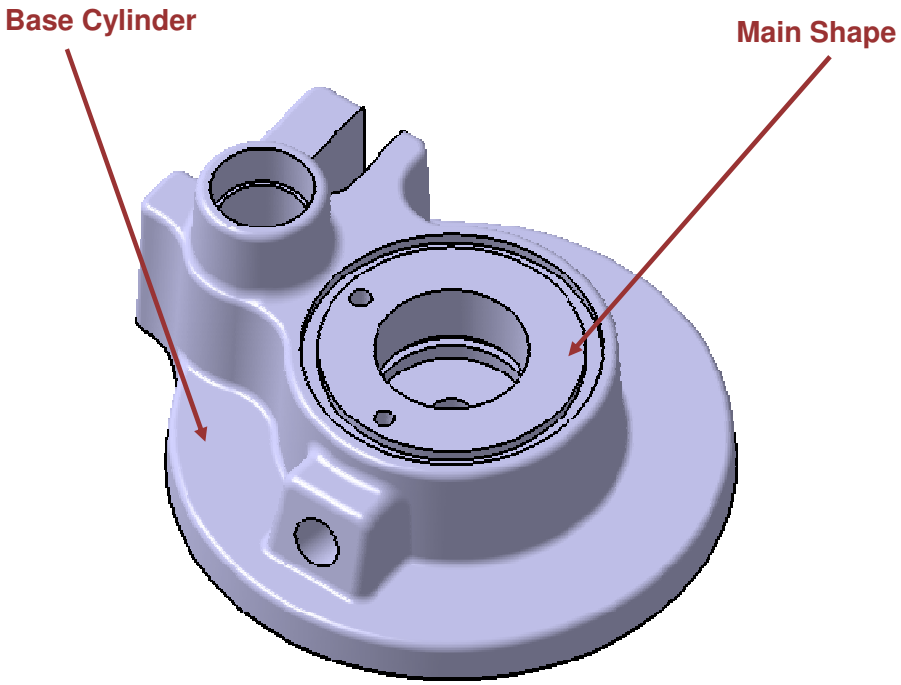




Student Notes:

Student Notes:

Design intent: Car Jack Support

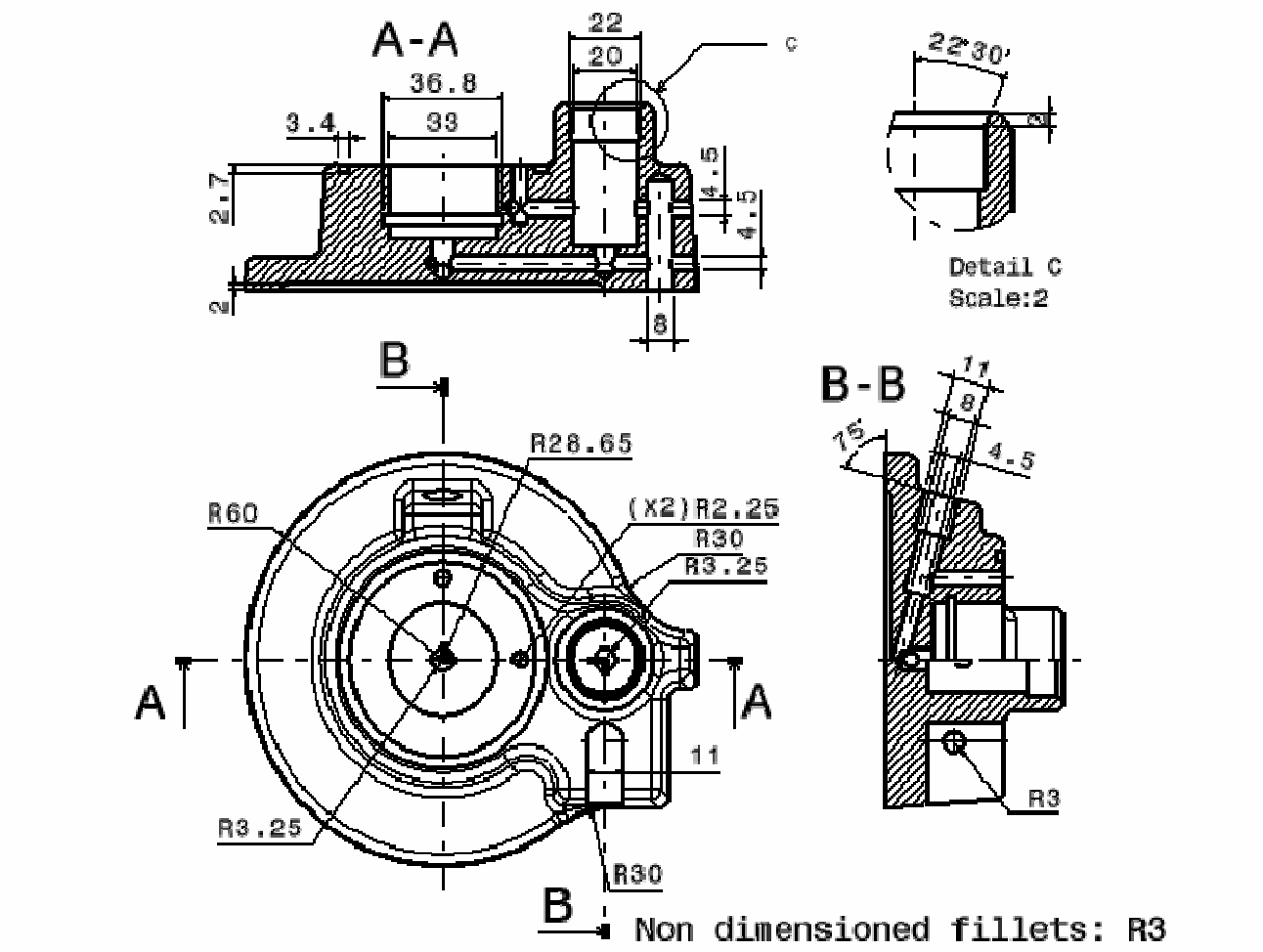


- Car Jack Support is a casting part used in an assembly.
- There is a network of holes in the part.

Student Notes:

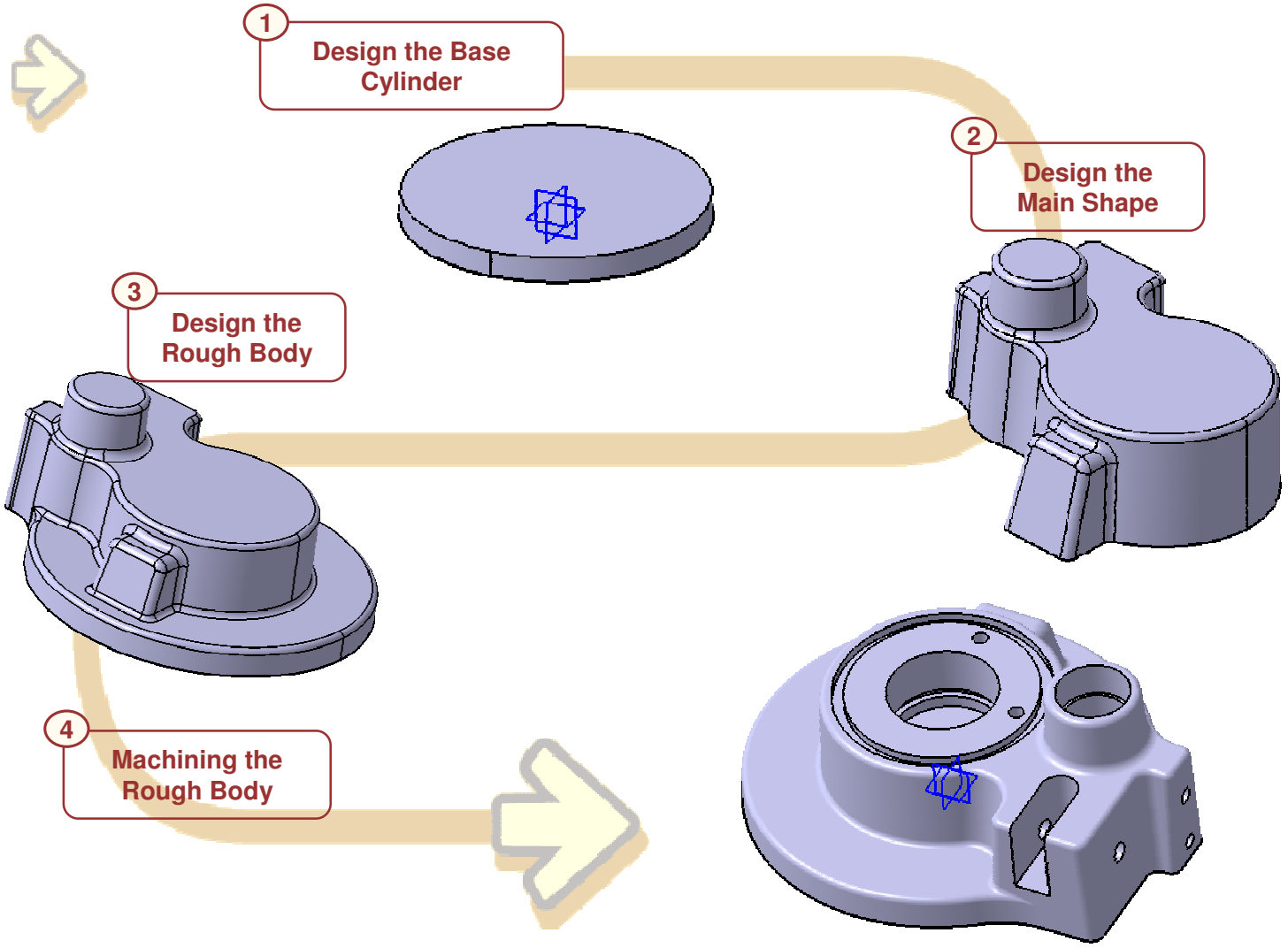
Car Jack Support drawing

- Understand the drawing thoroughly to design the part according to the specifications.



Student Notes:

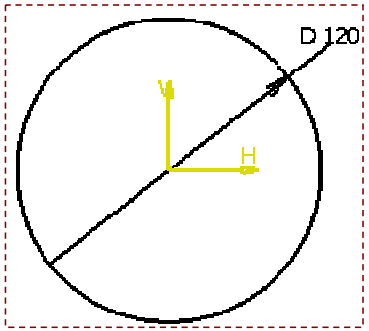
Design process: Car Jack Support

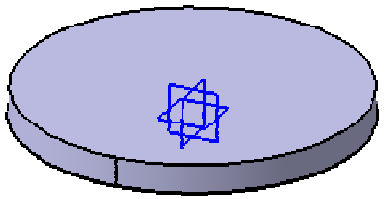


Student Notes:

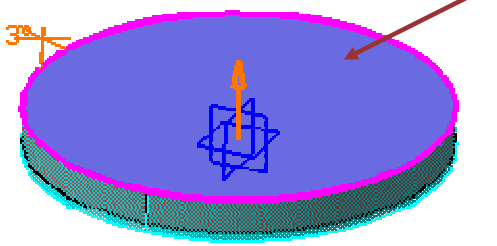
Step 1: Design the Base Cylinder

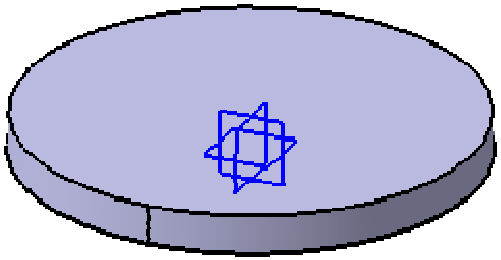
- Start a New Part.
- Insert body 'Base_Cylinder'.
- Create a positioned sketch on XY plane in geometrical set.This is sketch.1
- Pad it by 10 mm. Apply a draft of 3 deg as shown.





Neutral element and Pulling Direction

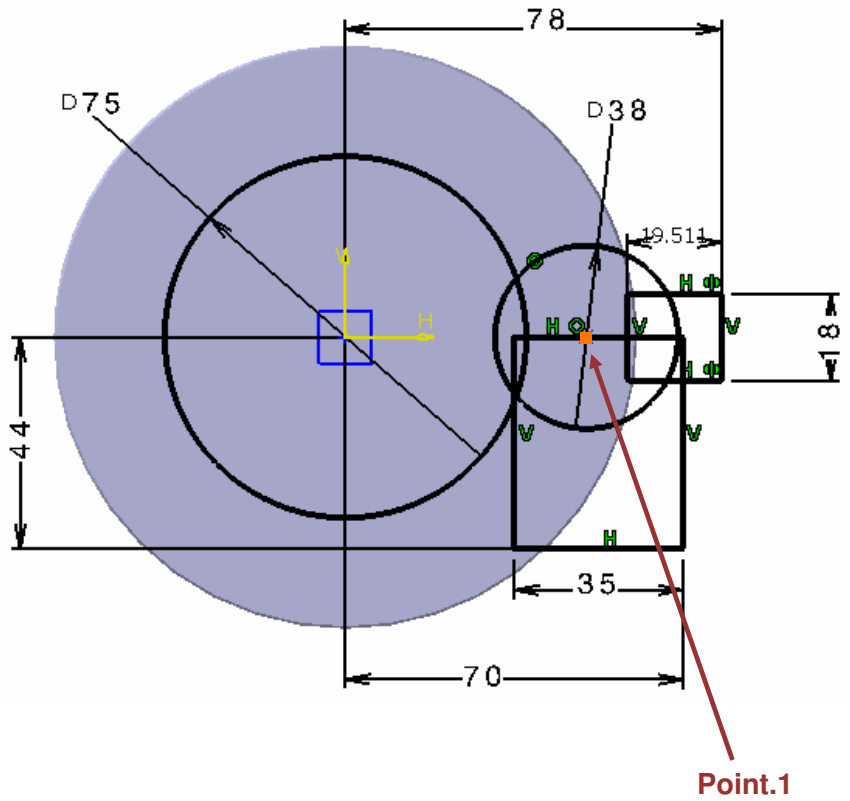




Student Notes:

Step 2: Design the Main Shape (1/7)

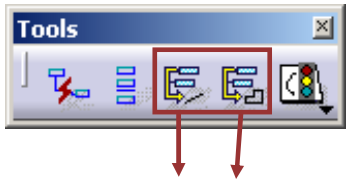
- In the Geometrical set Create a point (50,0,0).
- In the Geometrical set create positioned Sketch.2 on XY plane. Circle (diameter=38mm) is coincident with the above point.

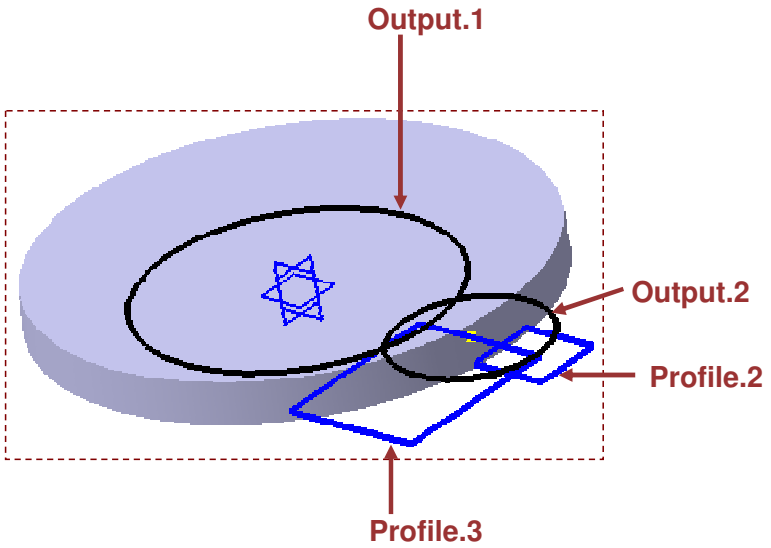


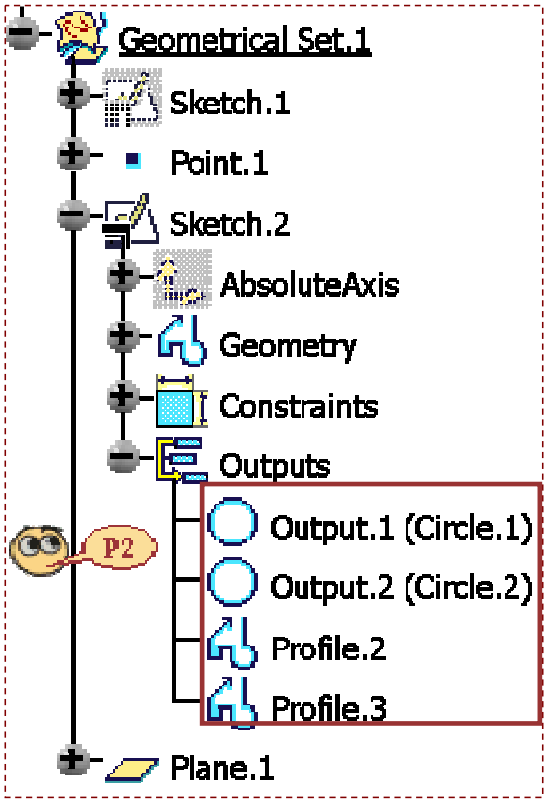
Student Notes:

Step 2: Design the Main Shape (2/7)

- Expose the elements of the sketch using 'Output feature' and 'Profile feature' tools in sketcher workbench so that you can use them individually.
- Create a Plane at a distance of 37 mm above the XY plane. This is Plane.1.



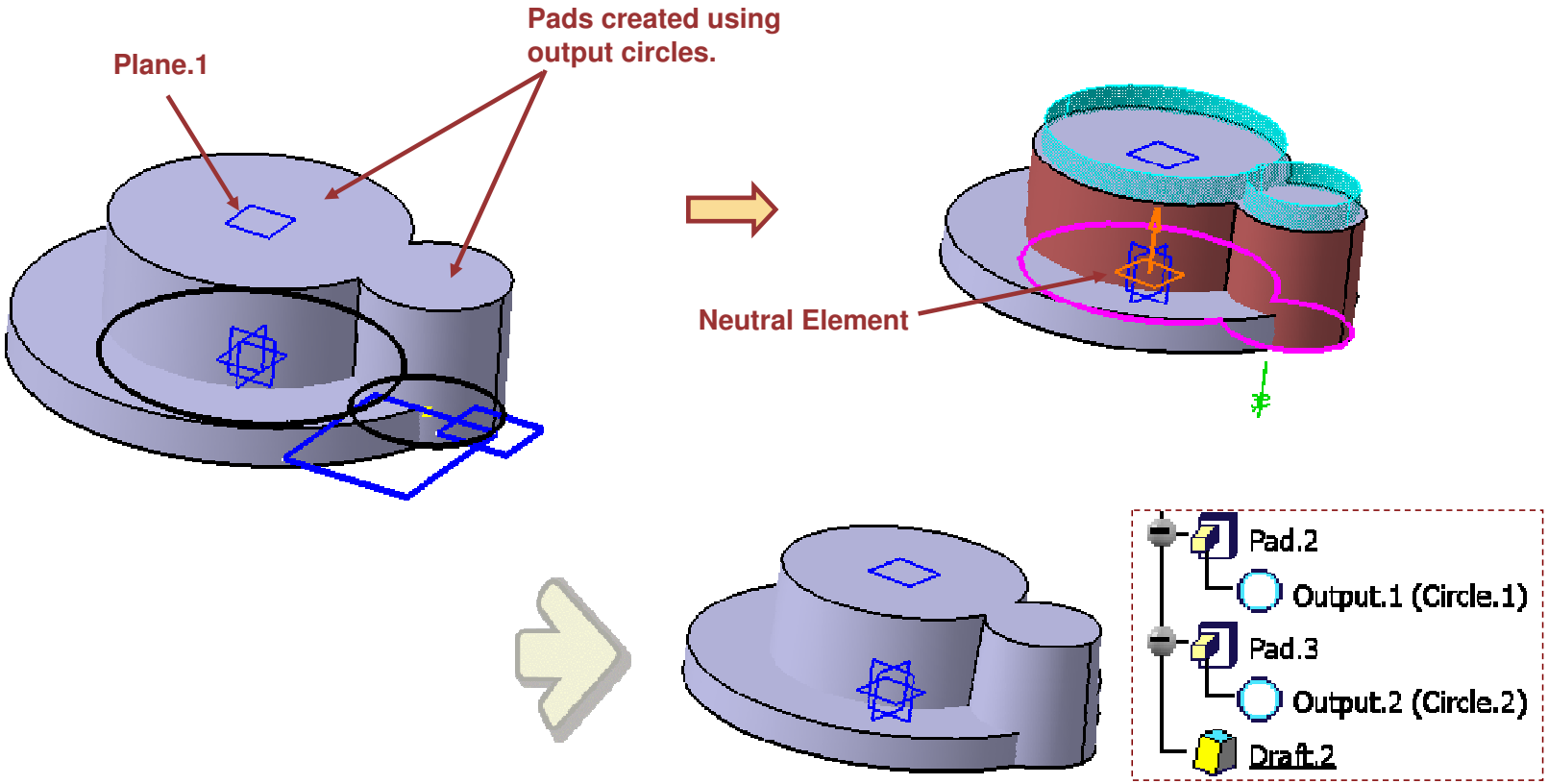




Student Notes:

Step 2: Design the Main Shape (3/7)

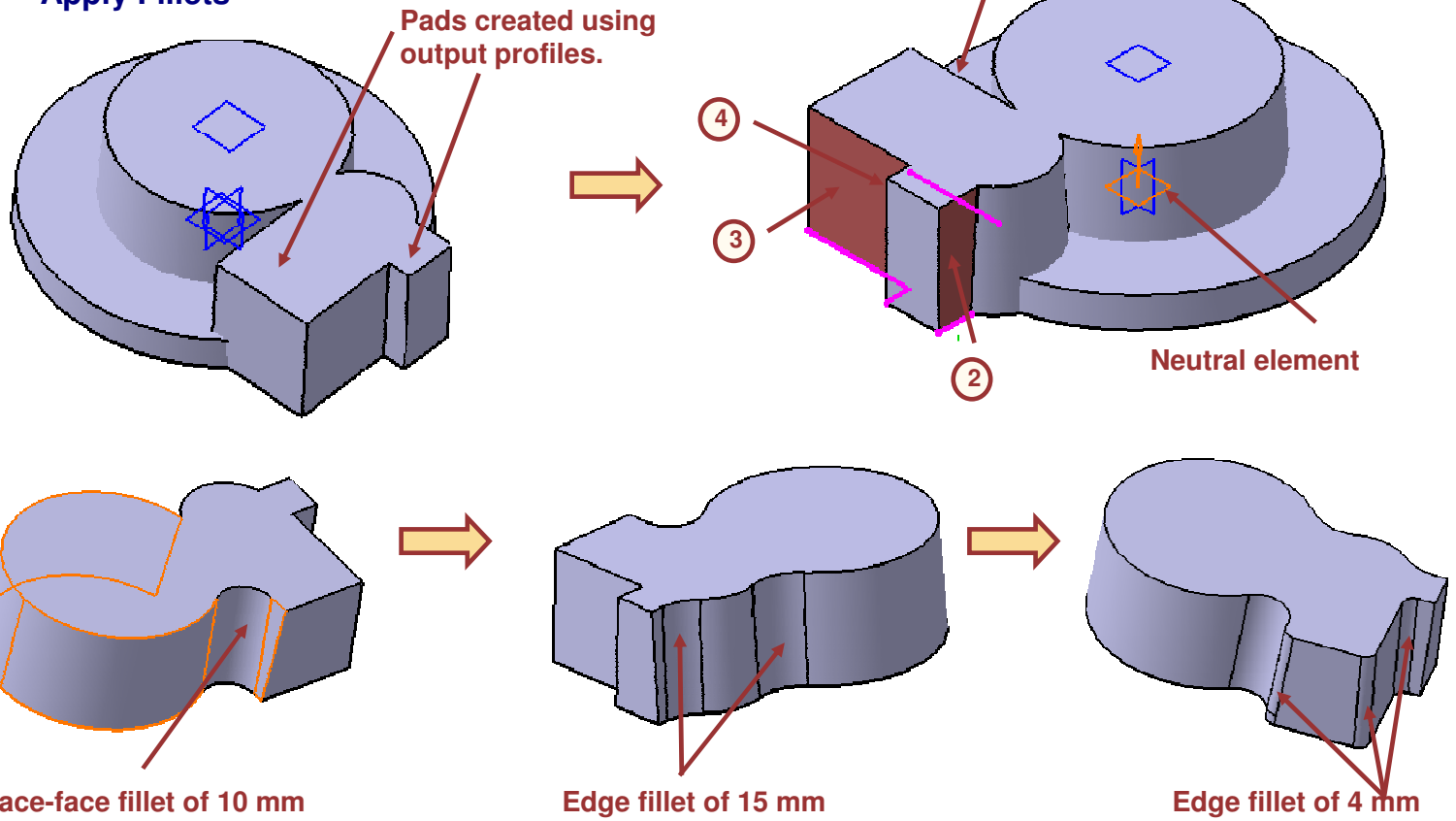
- Insert Body 'Main_Shape'.
- Pad 'Output.1(Circle.1)' upto plane.1 in this body.
- Pad 'Output.2(Circle.2)' upto plane.1 in this body.
- Apply a draft of 3 deg to these two pads as shown.



Student Notes:

Step 2: Design the Main Shape (4/7)

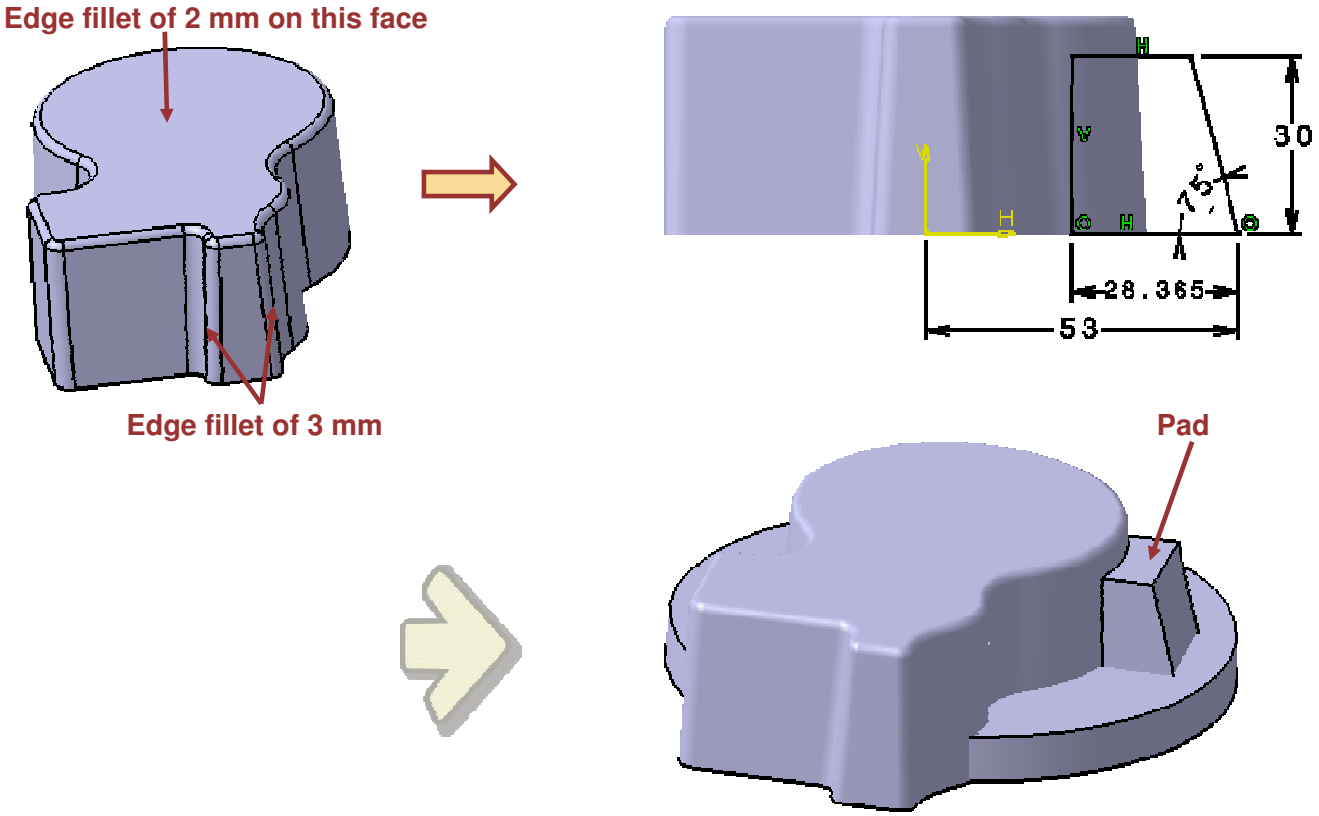
- Pad 'Profile.2' and 'Profile.3' upto plane.1
- Apply a draft of 3 deg to the four faces shown
- Apply Fillets



Student Notes:

Step 2: Design the Main Shape (5/7)

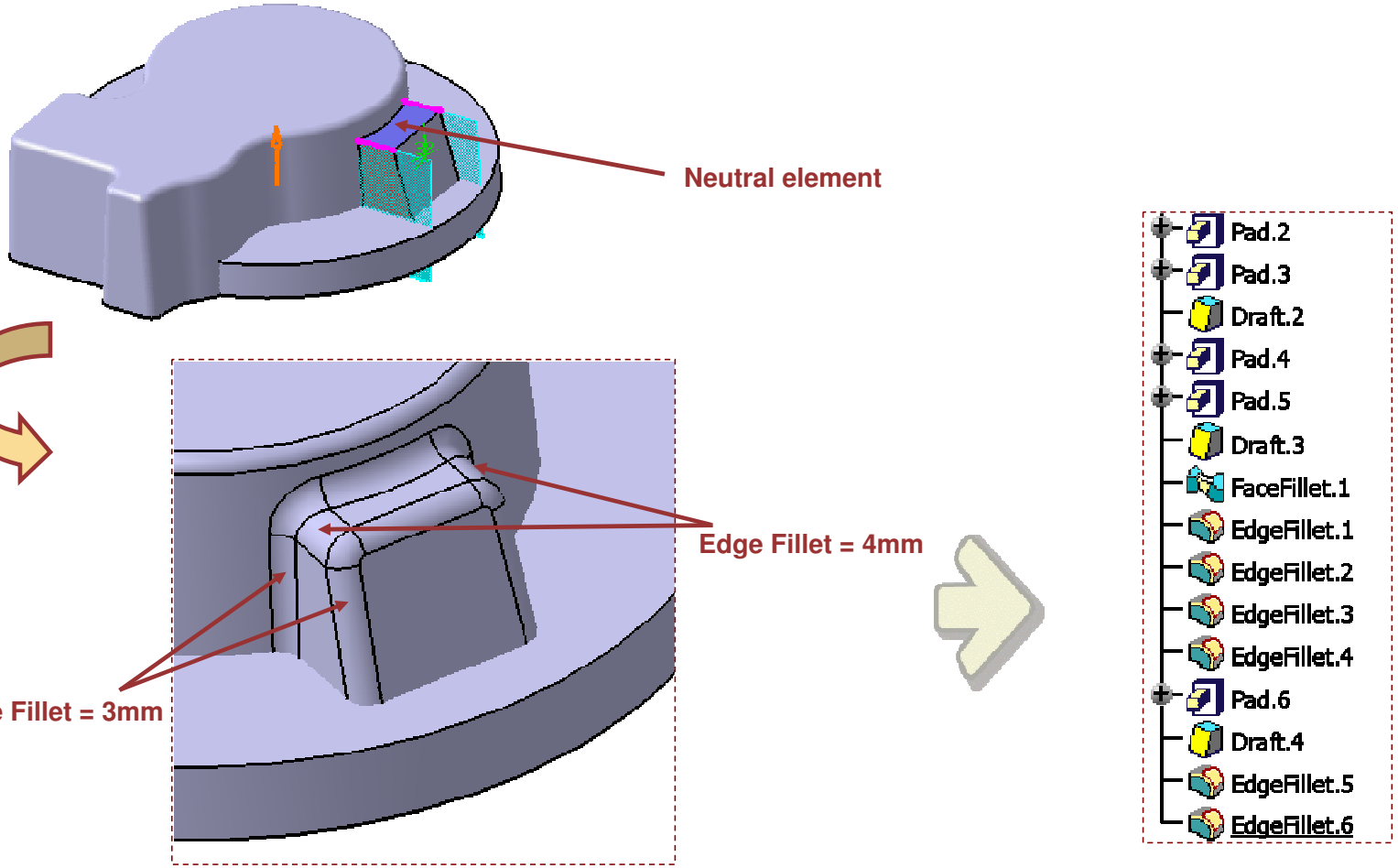
- Apply Fillets to the top face and edges shown.
- Create a positioned sketch on YZ plane and orient it with Y axis. Create it in geometrical set. This is sketch.3.
- Pad it by 12 mm (Mirrored Extent)



Student Notes:

Step 2: Design the Main Shape (6/7)

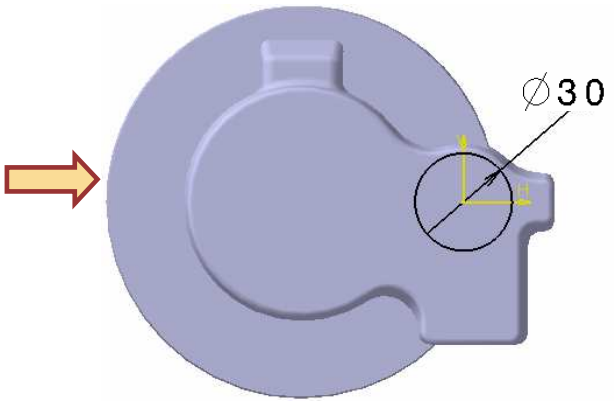
- Apply Draft of 3deg to the faces shown.
- Apply Edge Fillet to the edges shown.

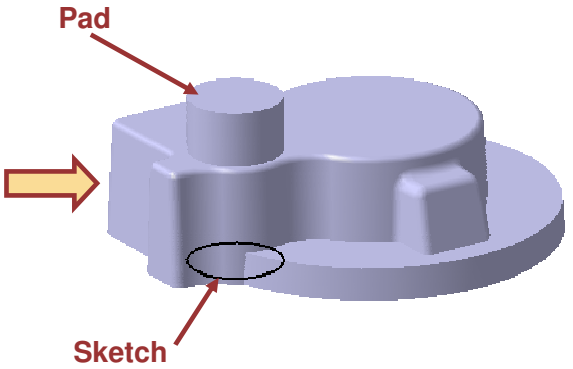


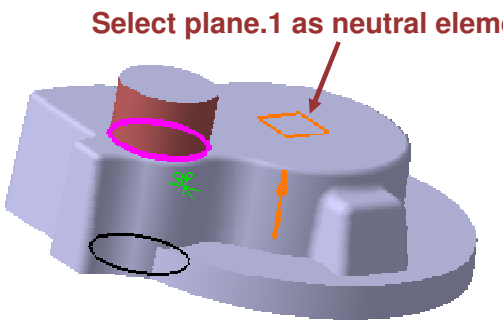
Student Notes:

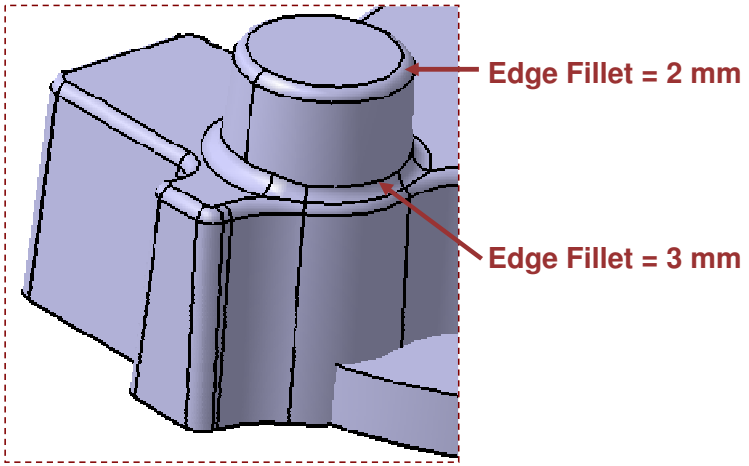
Step 2: Design the Main Shape (7/7)

- Create a Positioned sketch on XY plane. Use Point.1 as Projection Point to define origin.
- Pad it by 55 mm
- Apply draft of 3 deg and different Edge fillets





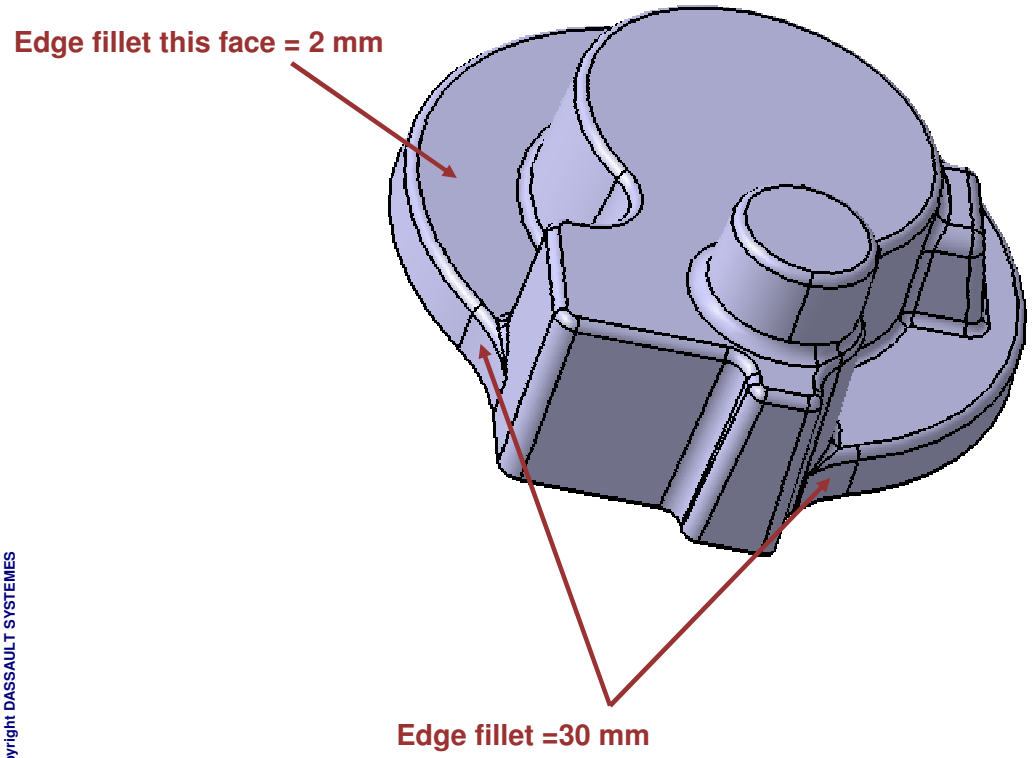


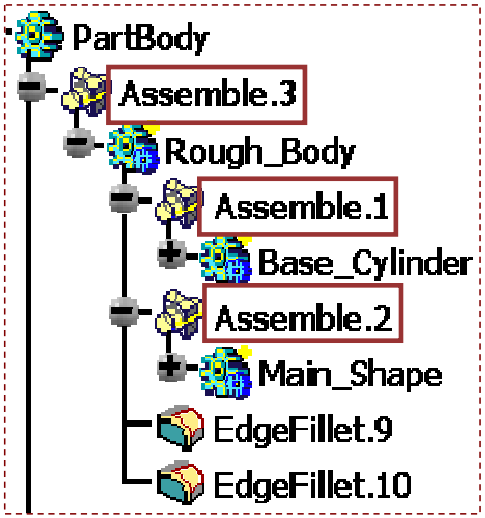


Student Notes:

Step 3: Design the Rough Body

- Insert 'Rough_Body'
- Assemble 'Base_Cylinder' to it
- Assemble 'Main_Shape' to it.
- Apply Fillets to this Rough Body.
- Assemble this 'Rough_Body' to Part Body.

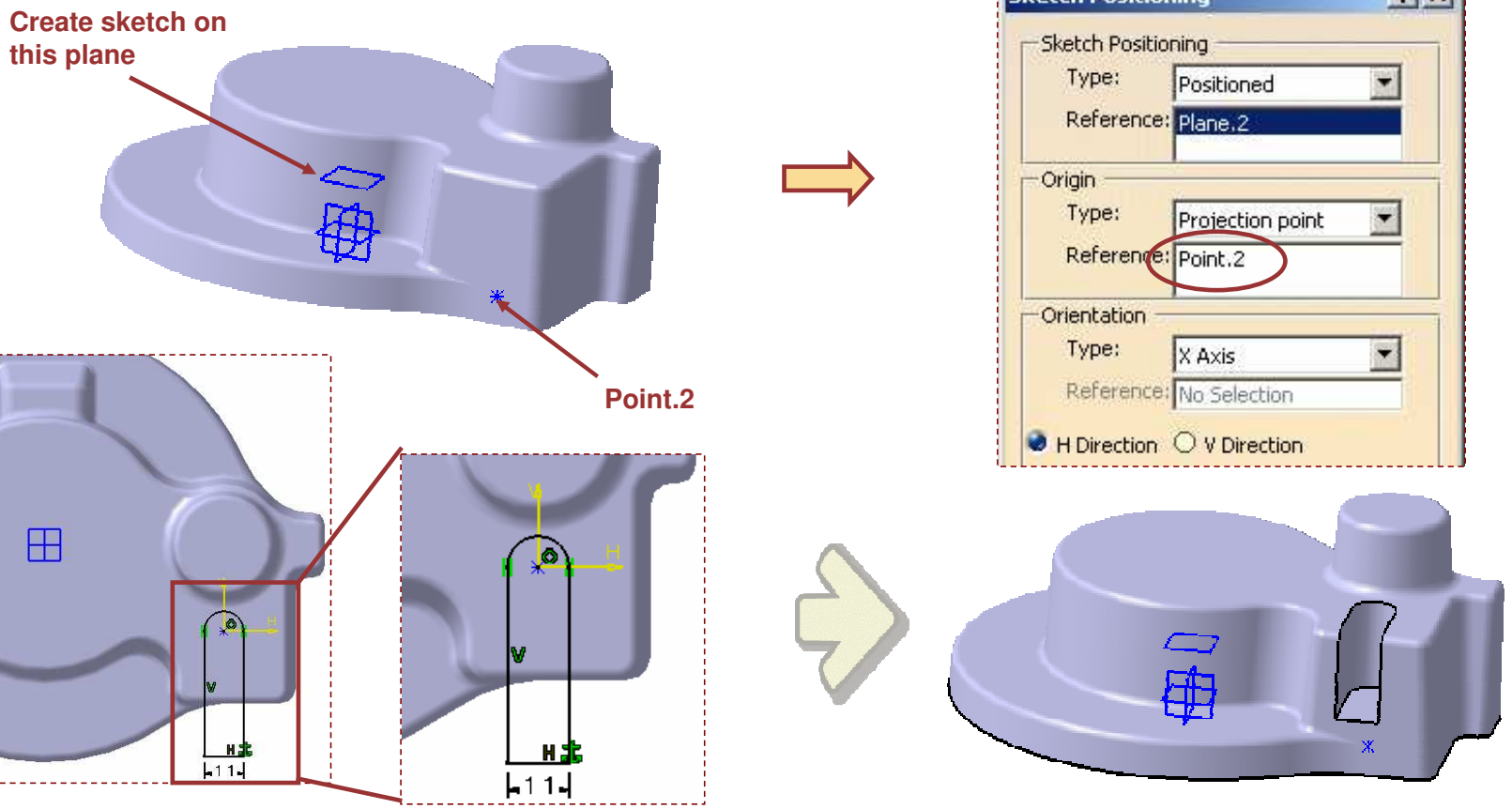




Student Notes:

Step 4: Machining the Rough Body (1/9)

- Create a Reference point on XY plane with H=50mm and V=-24mm. This is Point.2
- Create a Plane from XY plane offset of =14mm.
- Create a Positioned sketch on this plane and use point.2 as origin
- Create a pocket from it upto plane.1 with a offset distance of 5mm

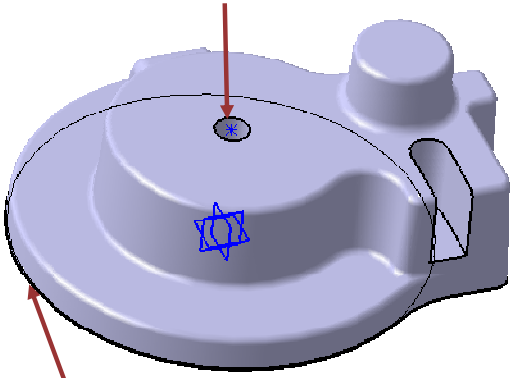


Student Notes:

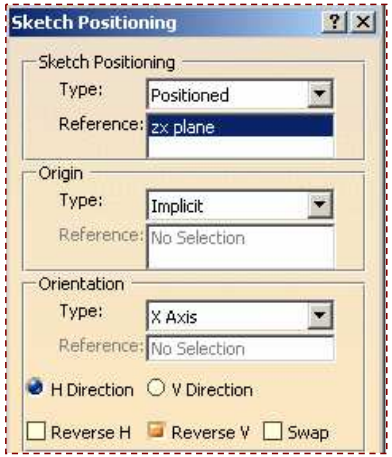
Step 4: Machining the Rough Body (2/9)

- Create a Hole with parameters shown. Make it concentric with the edge shown.
- Create a groove with the positioned sketch on ZX plane

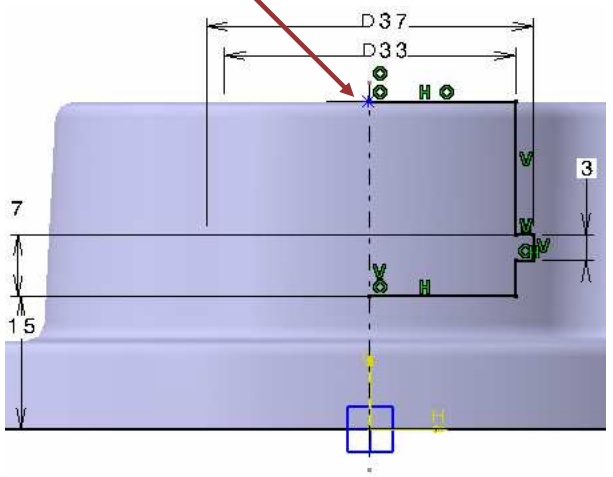
Diameter =10 mm
Depth = 30 mm
V-bottom angle =120 deg



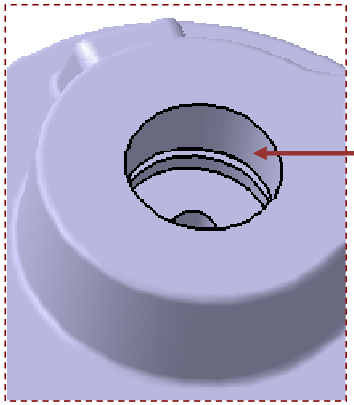
Make this Hole concentric with this edge



Use Plane.1 to constrain the groove sketch.



Sketch of Groove



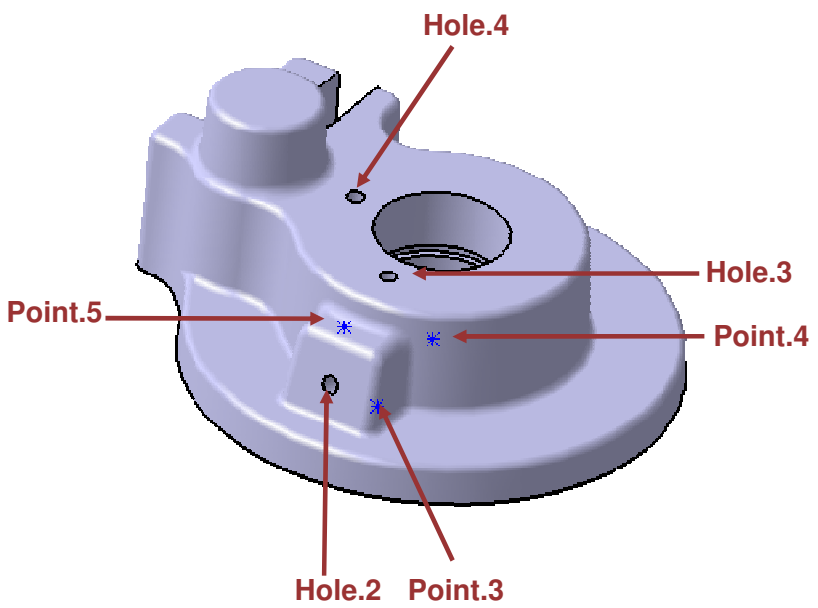
Groove

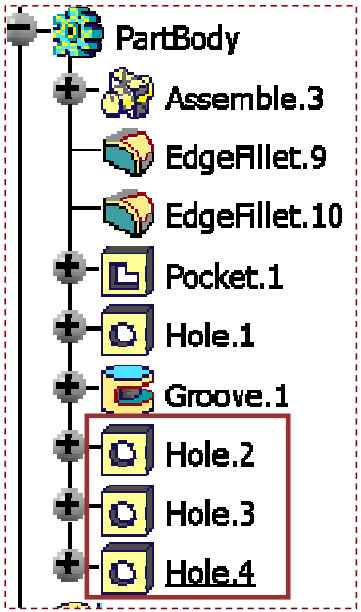
Student Notes:

Step 4: Machining the Rough Body (3/9)

- Create three points. Point.3 (0,0,6), Point.4 (0,24,0), Point.5 (24,0,0)
- Use these three points to create three holes

Hole	Diameter	Depth	Flat / V-Bottom
Hole.2 coincident with Point.3	4.5mm	50mm	V Bottom.Angle =120 deg
Hole.3 coincident with Point.4	4.5mm	25mm	Flat Bottom
Hole.4 coincident with Point.5	4.5mm	17mm	V Bottom.Angle =120 deg

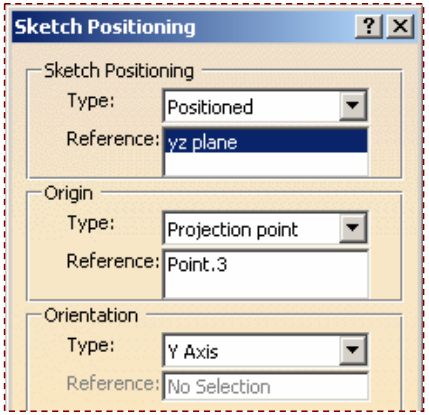


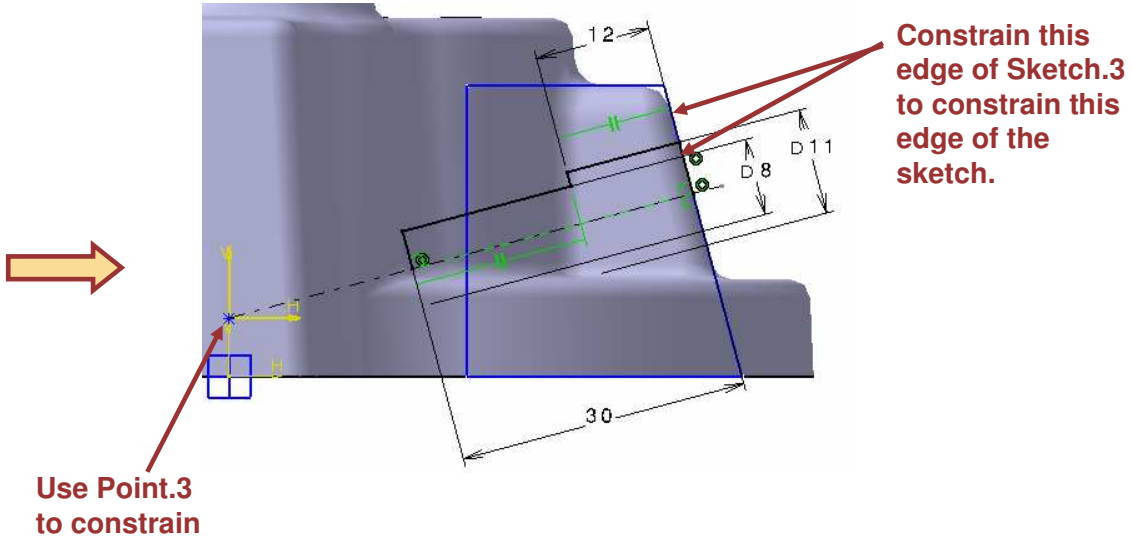


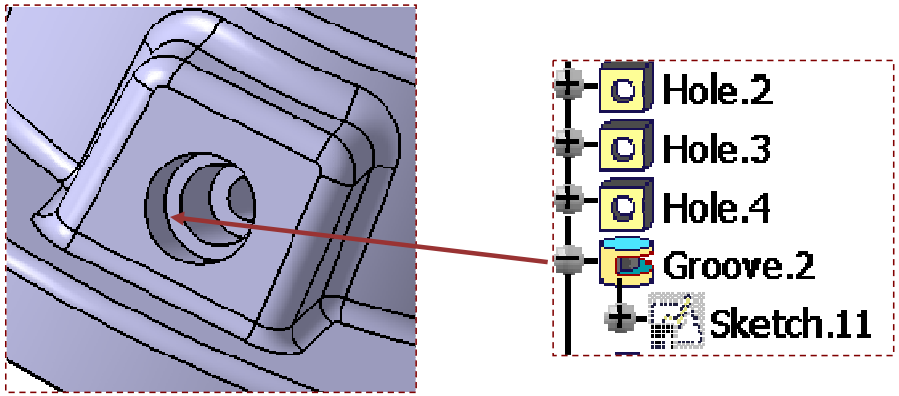
Student Notes:

Step 4: Machining the Rough Body (4/9)

- Create a positioned sketch as shown
- Groove it by 360deg



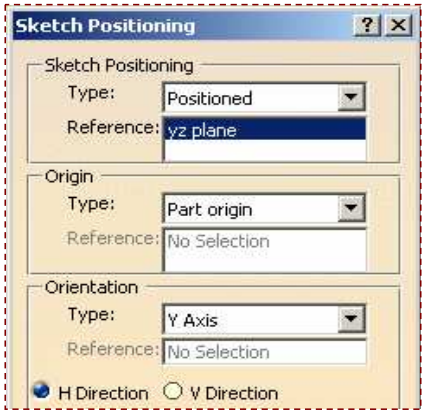


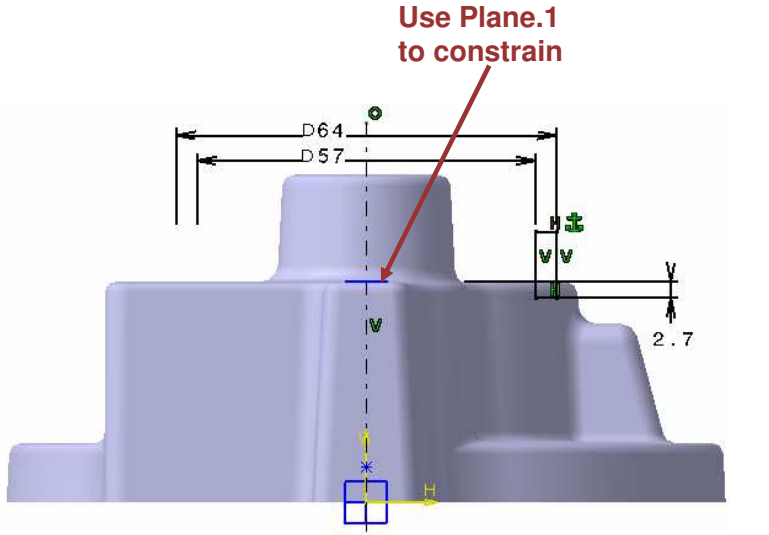


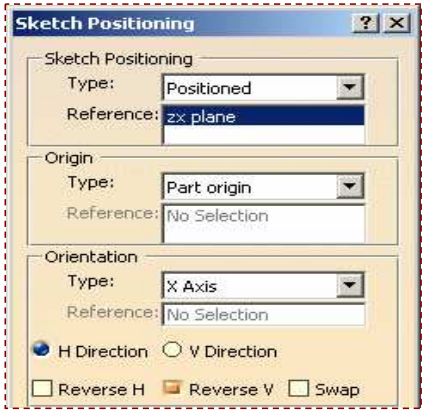
Student Notes:

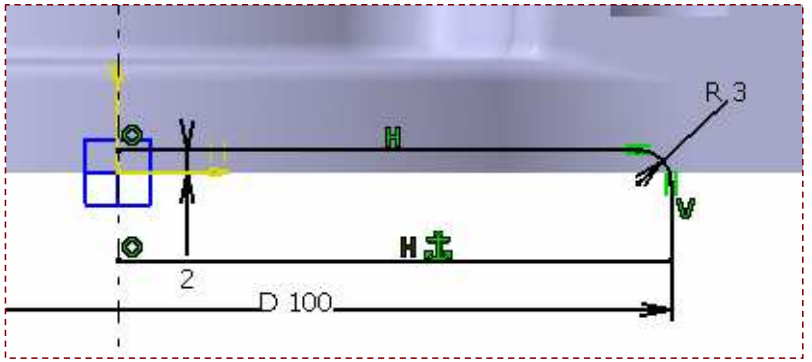
Step 4: Machining the Rough Body (5/9)

- Create two positioned sketches as shown





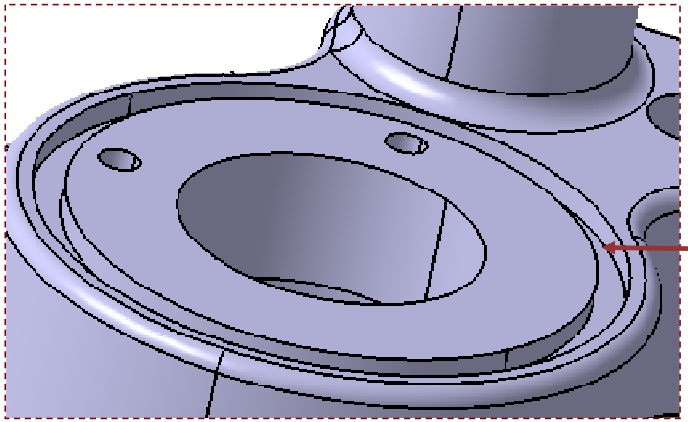


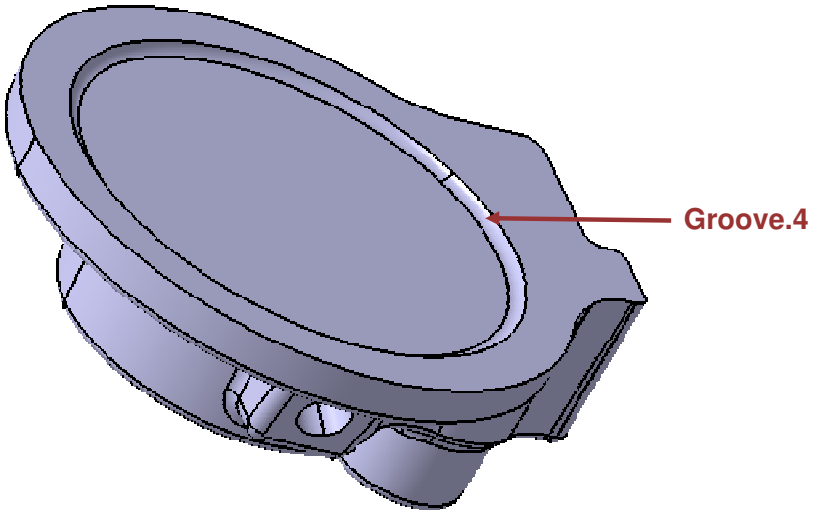


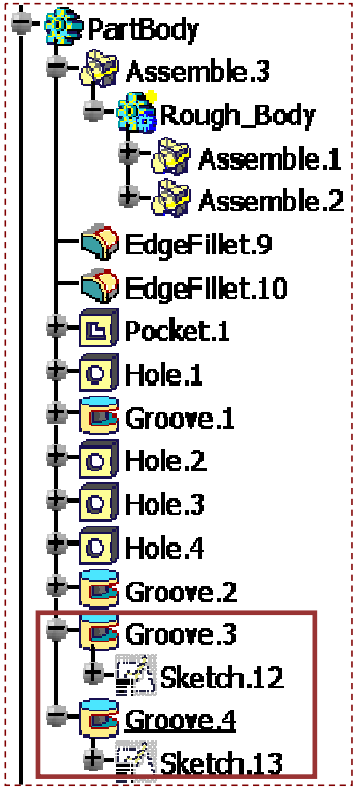
Student Notes:

Step 4: Machining the Rough Body (6/9)

- Create a 360 degree groove from two sketches



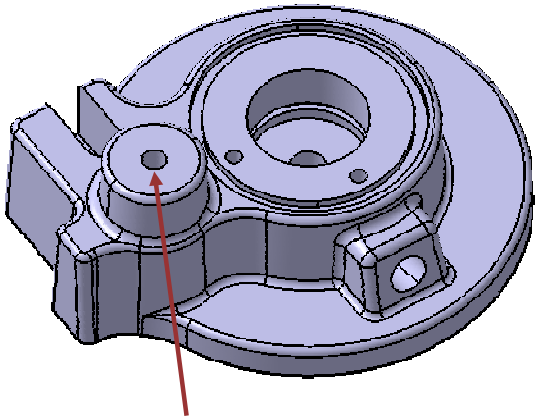




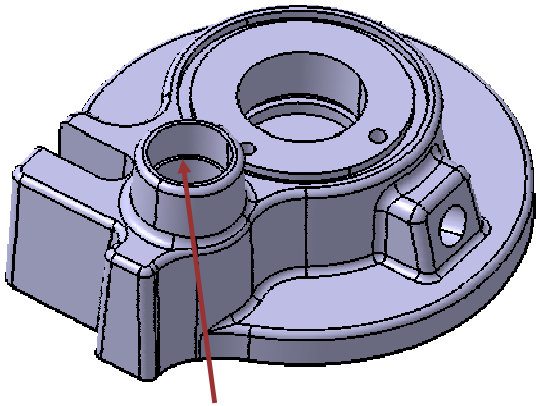
Student Notes:

Step 4: Machining the Rough Body (7/9)

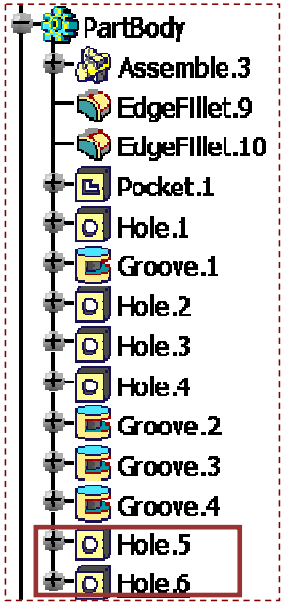
- Create a hole with following parameters:
  - Blind Hole. The Hole is concentric with Output.2.
  - Diameter = 6.5 mm
  - Depth = 50 mm
  - V-Bottom. Angle=120 deg
- Create another hole with following parameters:
  - Blind Hole. The Hole is concentric with Output.2.
  - Diameter = 20 mm
  - Depth =42 mm
  - Counter bored. Diameter = 22 mm. Depth =11 mm



Hole.5



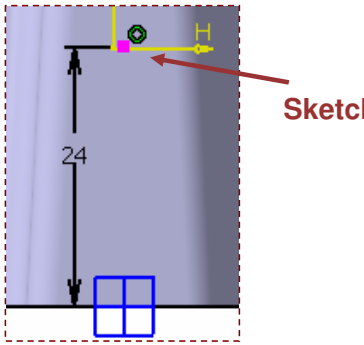
Hole.6



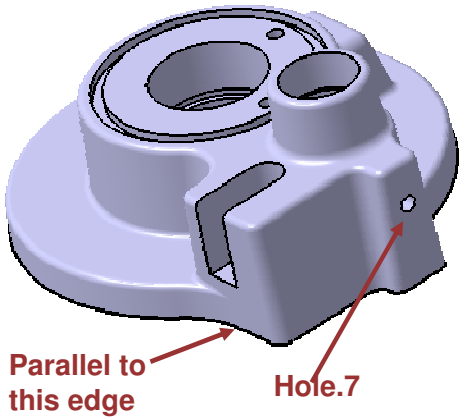
Student Notes:

Step 4: Machining the Rough Body (8/9)

- Create three more holes.
  - Blind Holes
  - Diameter of Hole.7 = 4.5 mm, Diameter of Hole.8 = 4.5 mm, Diameter of Hole.9 = 6 mm
  - Depth of Hole.7 = 56 mm, Depth of Hole.8 = 82 mm, Depth of Hole.9 = 32 mm
  - V-Bottom. Angle=120 deg.
  - All the holes are parallel to the direction shown.



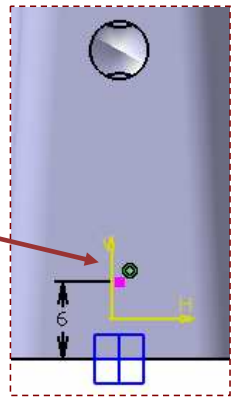
Sketch for Hole.7

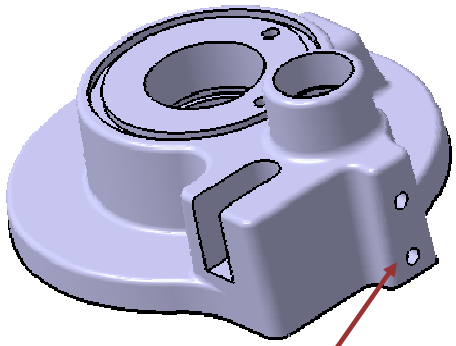


Parallel to this edge

Hole.7

Sketch for Hole.8





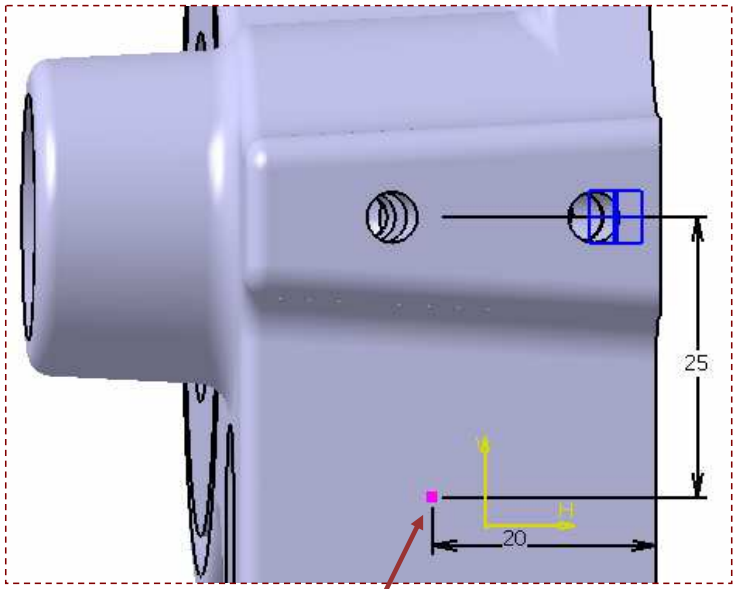
Hole.8

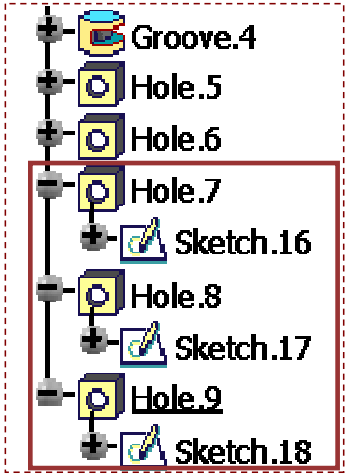
Student Notes:

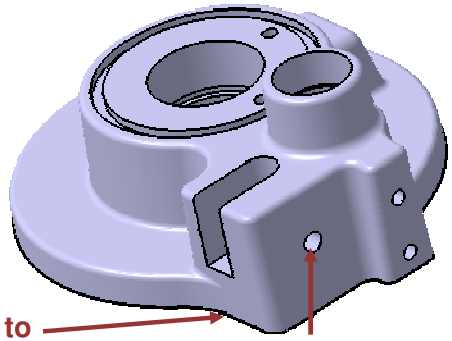
Step 4: Machining the Rough Body (9/9)

Result: 'PDG_Car_Jack_Support.CATPart'

- Create three more holes.
  - Blind Holes
  - Diameter of Hole.9 = 6 mm
  - Depth of Hole.9 = 32 mm
  - V-Bottom. Angle=120 deg.
  - All the holes are parallel to the direction shown.







Sketch for Hole.9

Parallel to this edge

Hole.9

EXERCISE BOOK

Student Notes:

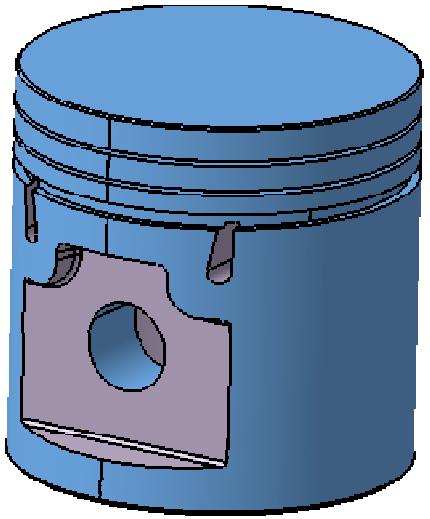
Piston

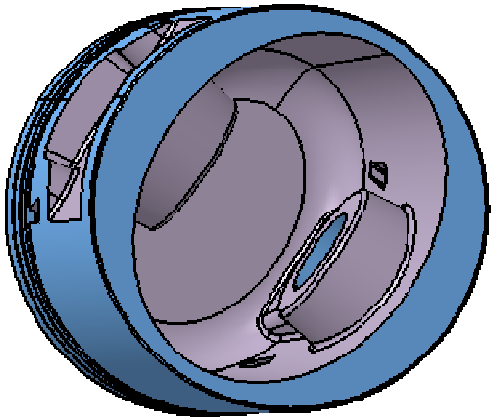
Part Design Advanced Exercise

95 min

In this exercise you will build the Piston by following a recommended process using Boolean approach.To create the Piston you will apply knowledge gained from Part design and knowledgeware fundamentals.

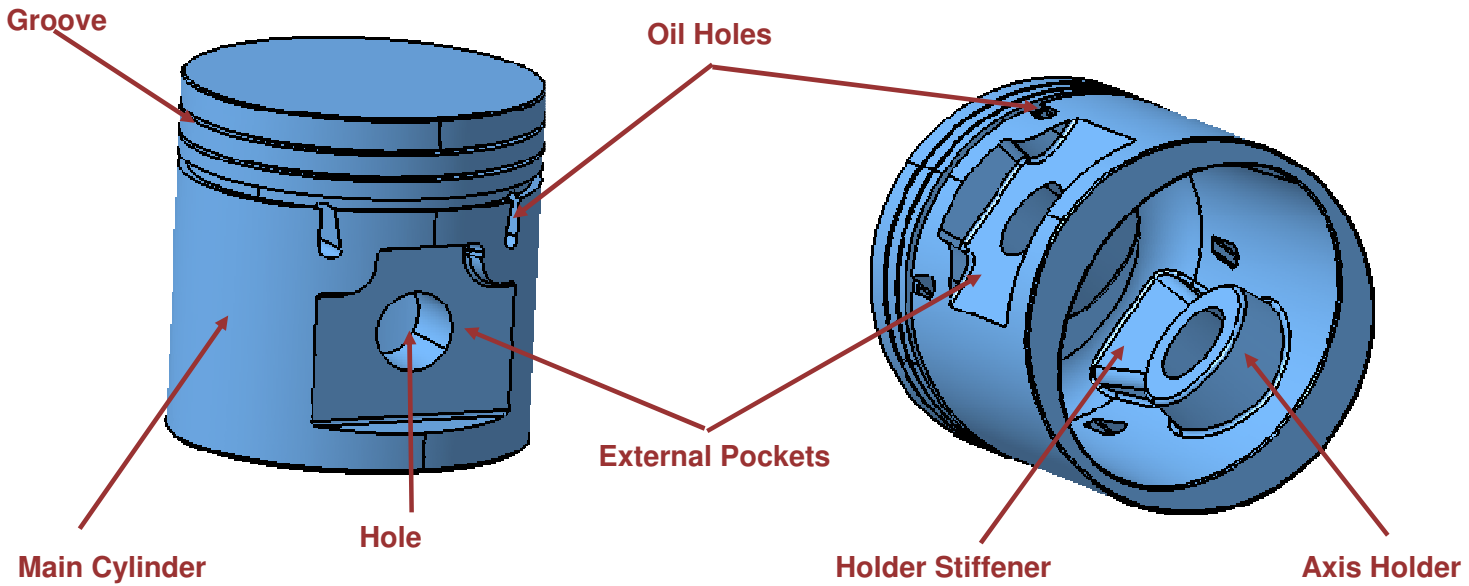
- You will first understand the purpose of various Wireframe elements used to design the different functional features of the Piston.
- After interpreting the wireframe elements, you will create them in the geometrical set. You will also define the various user parameters.
- Finally,you will design the various functional features of the piston using wireframe elements and optimize the design with user parameters.





Student Notes:

Design intent: Piston

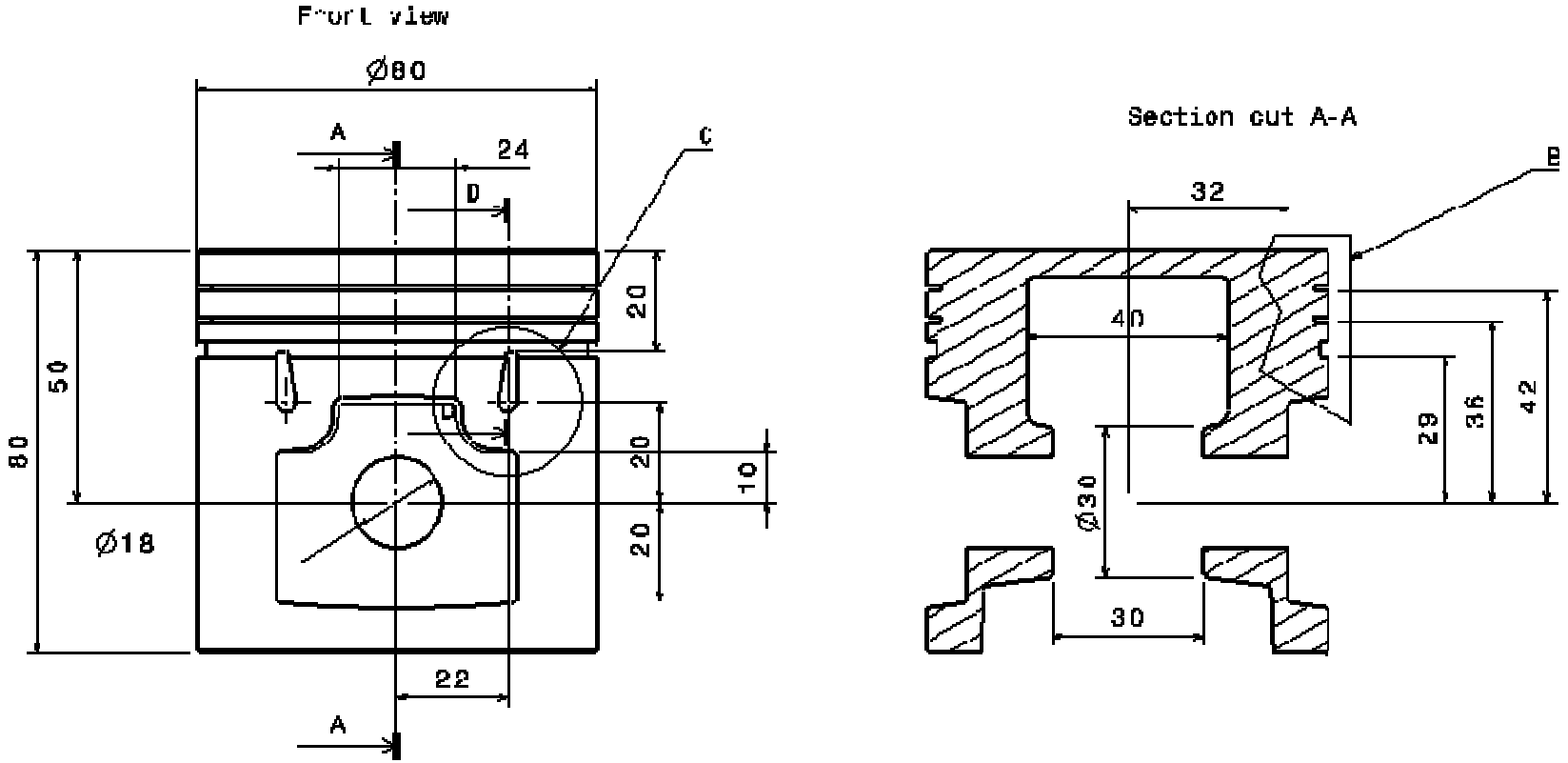


- The piston is a casted and machined part.
- “Holder stiffener” is used to provide support to the “Axis Holder”.
- A hole is created along “Hole Axis”.
- Oil holes are used to lubricate the Piston and other parts of the assembly.
- A groove is provided to accommodate Piston rings.

Student Notes:

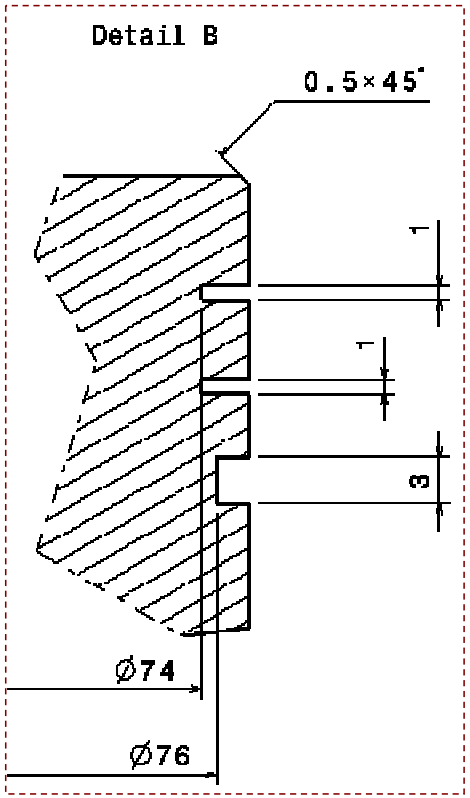
Piston Drawing (1/2)

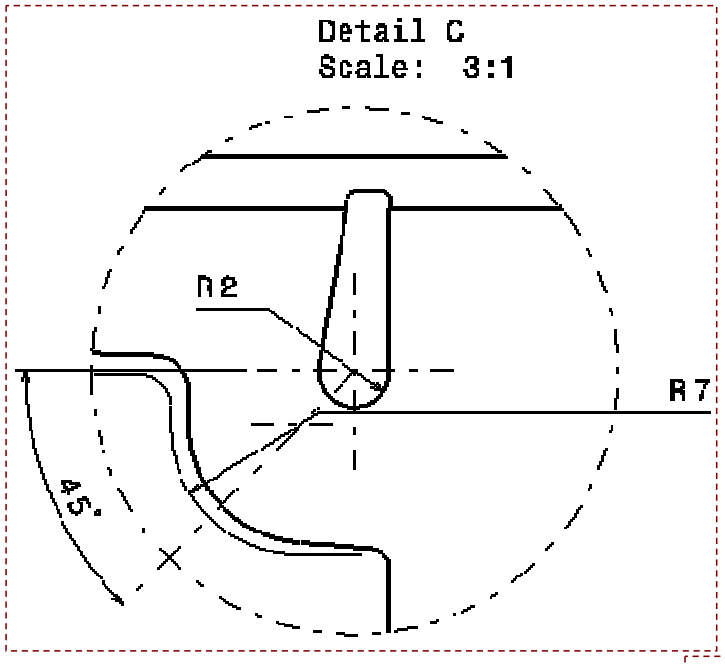
- Understand the drawing thoroughly to design the part according to the specifications.

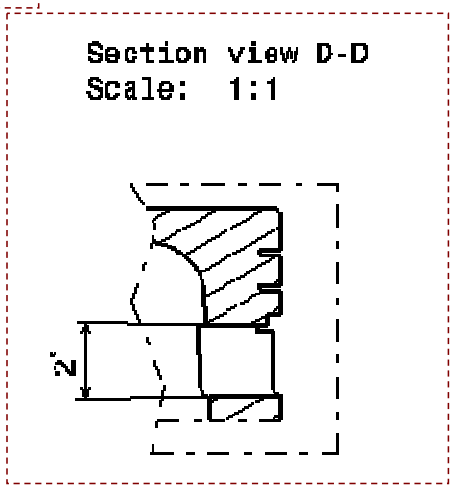


Student Notes:

Piston Drawing (2/2)





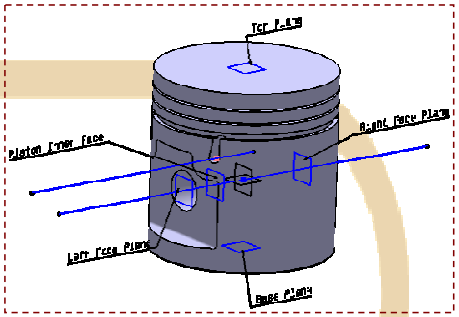


Student Notes:

Design process: Piston

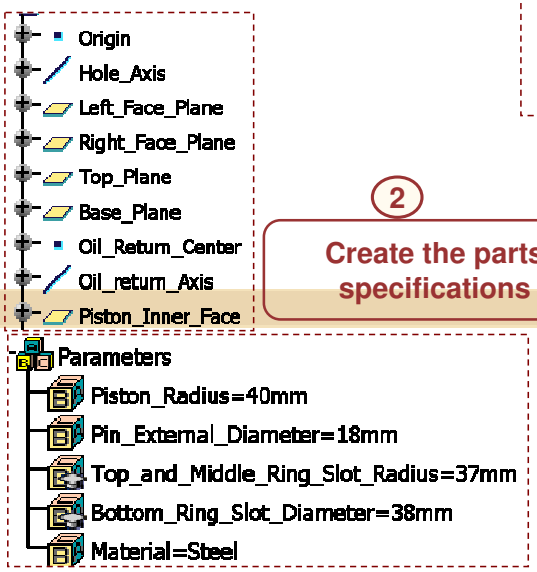
1

Understanding the design intent



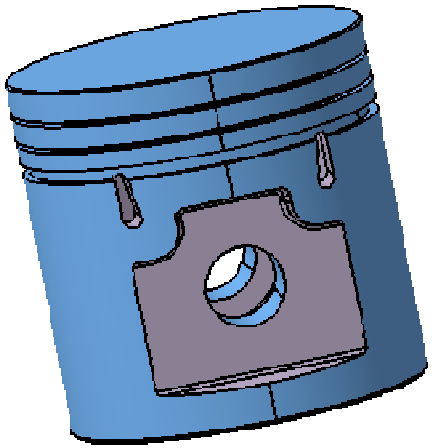
2

Create the parts specifications



3

Design the Piston with the given specifications



Student Notes:

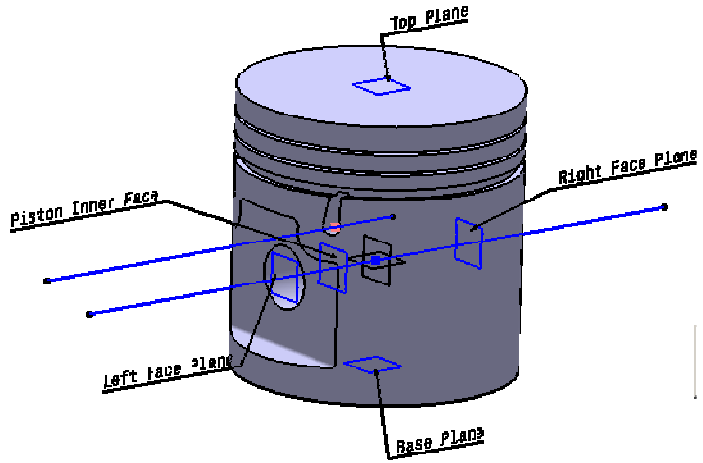
Piston

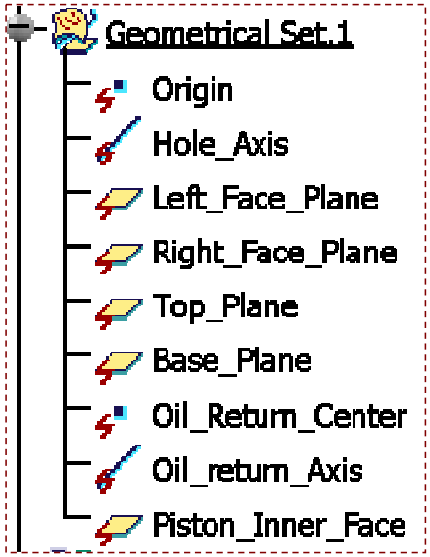
Step 1: Understanding the Design Intent

15 min

The purpose of this step is to understand the reason behind the creation of wireframe elements in relation with the solid. You will:

- **Study the solid.**
- **Study the Wireframe elements in relation with the solid.**
- **Determine the relation between the wireframe elements and corresponding part.**



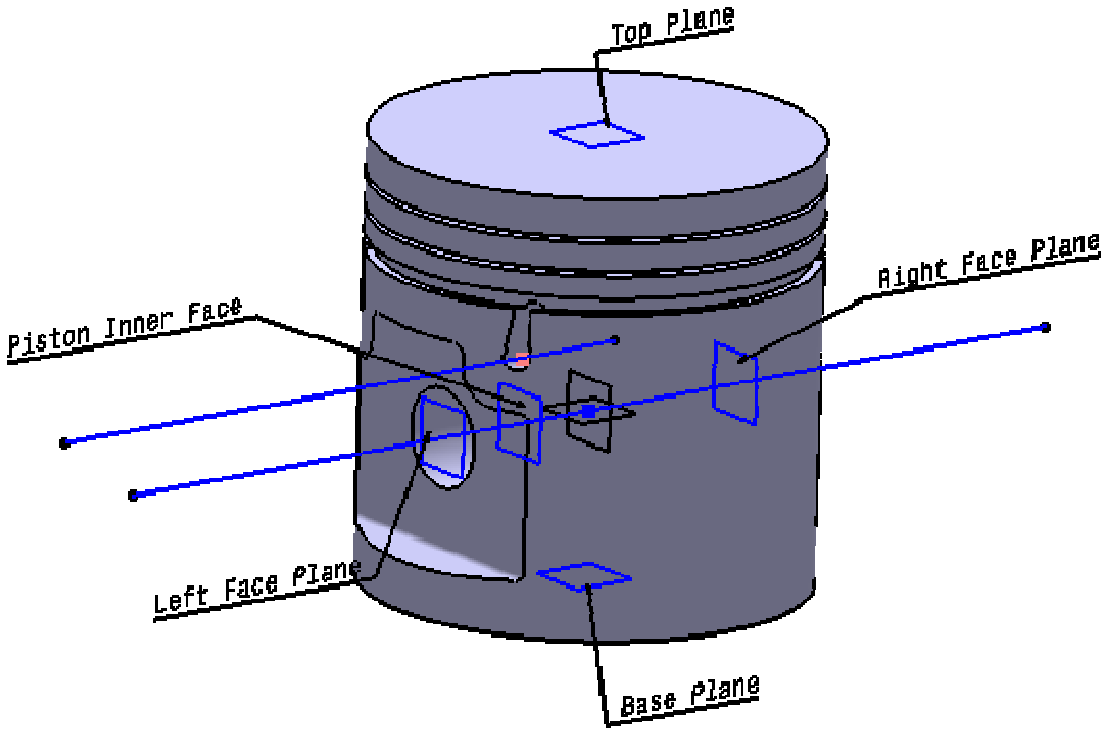


Student Notes:

Do It Yourself

'PDG_Piston_Understanding_Design_Intent.CATPart'

- Study the part without history.
- Observe the various Wireframe elements and interpret their relation with the solid.
- Try and answer questions such as:
  - Why 'Top Plane' was created?
  - What is the purpose of creating 'Piston Inner Face' plane

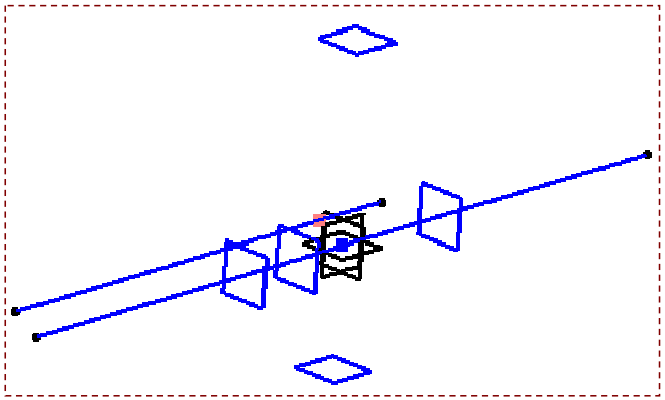


Student Notes:

Piston

Step 2: Creating Reference Geometry and Specifications

20 min



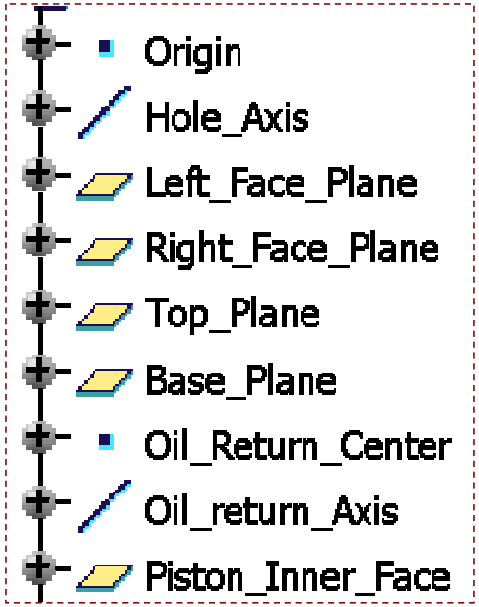
In this step you will create the user parameters and reference elements

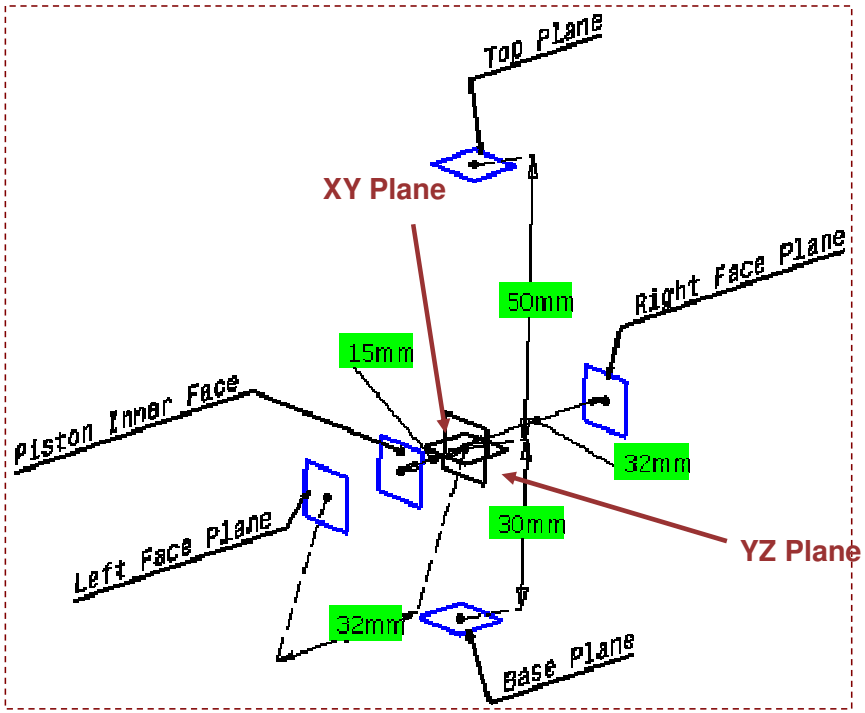
- **Create reference points,lines,planes.This forms the basis of the design.**
- **Create various user defined parameters.**

Student Notes:

Do It Yourself (1/3)

- Create one point at the origin.
- Create a line(Hole axis) passing through origin along X axis of length 100 mm(mirrored extent).
- Create four planes as: 'Left_Face_Plane', 'Right_Face_Plane', 'Top_Plane', 'Base_Plane'.
- Create another point on left face plane (H=22 & V=20). Rename this to 'Oil_Return_Center'.
- Create another line(Oil return axis) passing through 'Oil_Return_Center' point along X axis with end =100 mm and start = - 20 mm.
- Create fifth plane 'Piston_Inner_Face'





Student Notes:

Do It Yourself (2/3)

- Create five user-defined parameters.
  - 'Material' of type 'String'
  - 'Piston_Radius' of type 'Length'
  - 'Pin_External_Diameter' of type 'Length'
  - 'Top_and_Middle_Ring_Slot_Radius' of type 'Length'
  - 'Bottom Ring Slot Diameter' of type 'Length'
- 'Piston_Radius' controls the radius of the pad of Main cylinder
- You can modify the radius of piston by manipulating the value of parameter 'Piston_Radius'.
- Diameter for 'Pin' is controlled by 'Pin_External_Diameter' parameter.
- You will create relations between these parameters and features when you will design them.

Student Notes:

Do It Yourself (3/3)

- On completion of the design you should have created the following relations to drive your Piston externally.

Relations
- Formula.1: Rough_Body\Assemble.7\Main_Cylinder\Pad.1\Sketch.1\Radius.1\Radius=Piston_Radius
- Formula.2: `Core\Assemble.5\Axis Holder Features\Assemble.4\Inner_Additional_Shape\Assemble.1\Axis_Holder\Pad.2`
- Formula.3: Top_and_Middle_Ring_Slot_Radius=Piston_Radius - 3mm
- Formula.4: Bottom_Ring_Slot_Diameter=Piston_Radius -2mm
- Formula.5: Axis_Hole\Hole.1\Diameter=Pin_External_Diameter
- Formula.6: `Geometrical Set.1\Sketch.7\Radius.273\Radius`=Piston_Radius
- Formula.7: `Geometrical Set.1\Top Ring Slot Sketch\Offset.292\Radius`=Top_and_Middle_Ring_Slot_Radius
- Formula.8: `Geometrical Set.1\Middle Ring Slot Sketch\Offset.253\Radius`=Top_and_Middle_Ring_Slot_Radius
- Formula.9: `Geometrical Set.1\Bottom Ring Slot Sketch\Offset.250\Radius`=Bottom_Ring_Slot_Diameter
- Formula.10: `PartBody\Material`=`Material`

- You will design these relations when creating the feature i.e on the fly

Student Notes:

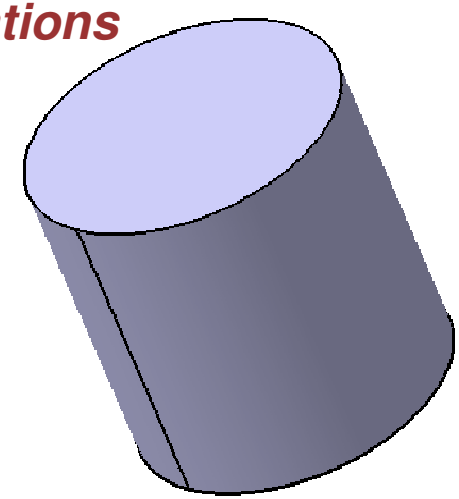
Piston

Step 3: Design the Piston with Specifications

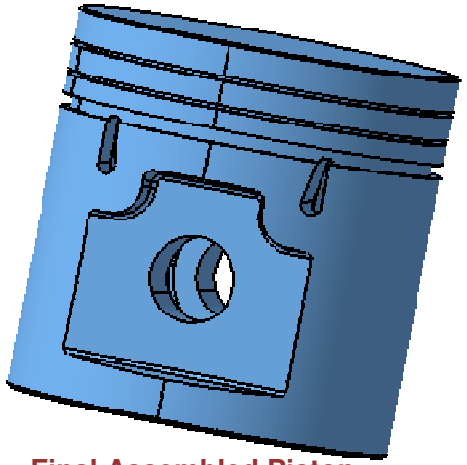
60 min

In this step you will create the part in accordance with the specifications provided.

- Create the main cylinder.
- Create various functional bodies.
- Apply Dress-up features.
- Assemble every body using the Boolean approach.



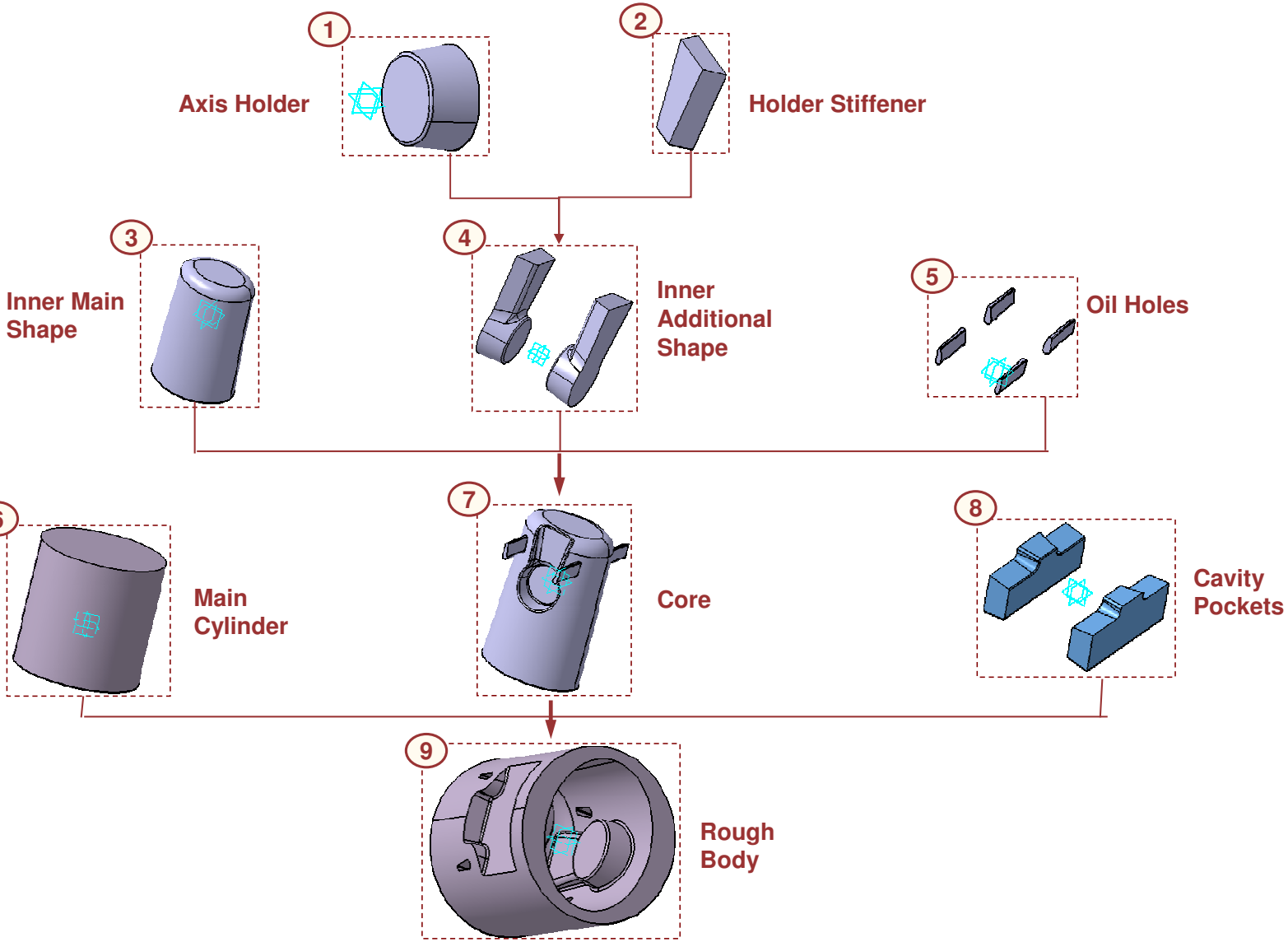
Main Cylinder



Final Assembled Piston

Student Notes:

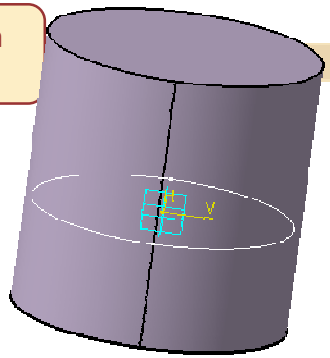
Structure of Rough Body in Piston



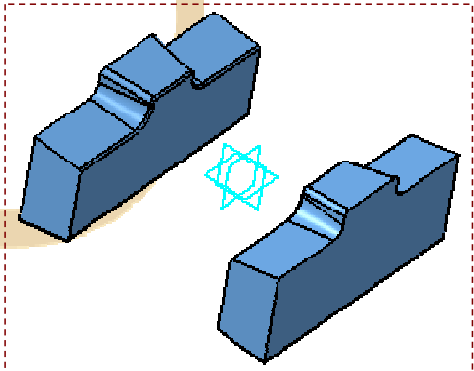
Student Notes:

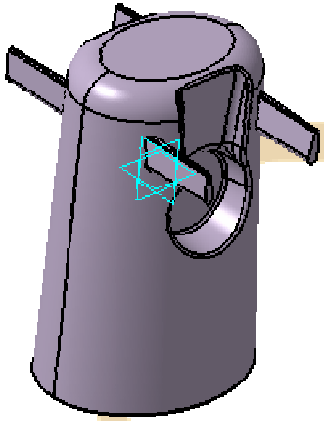
Design Process: Design the Piston with Specifications (1/2)

1 Design the Main Cylinder



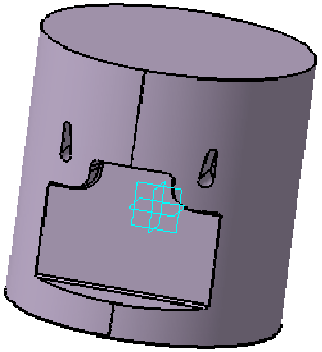
2 Design the Cavity Pockets

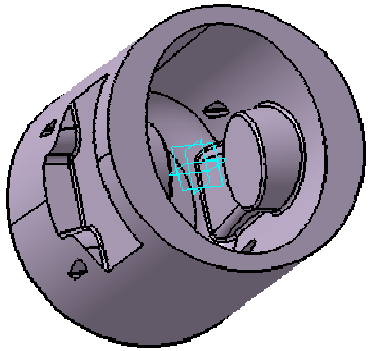




3 Design the Core
- Axis Holder Features
- Inner Main Shape
- Inner Additional Shape
- Axis Holder
- Holder Stiffener
- Oil Holes

4 Assemble the Main Cylinder, Cavity Pockets, Core in the Rough Body

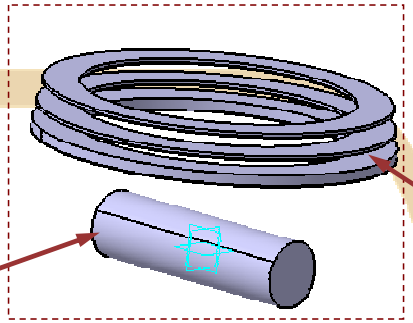




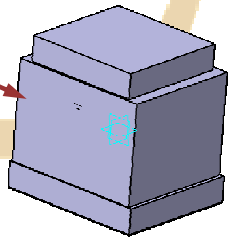
Student Notes:

Design Process: Design the Piston with Specifications (2/2)

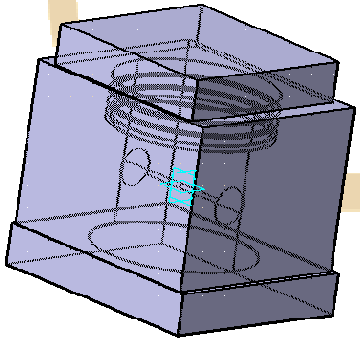
5 Design the features in the following bodies:
- External Machining
- Piston Ring Slots
- Axis Hole



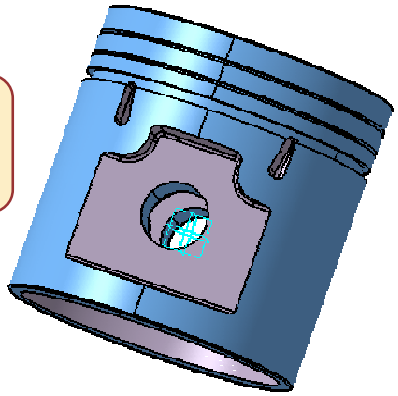
External Machining



6 Assemble the above bodies in'Machining' Body



7 Assemble the Rough body and Machining Body with the Part Body

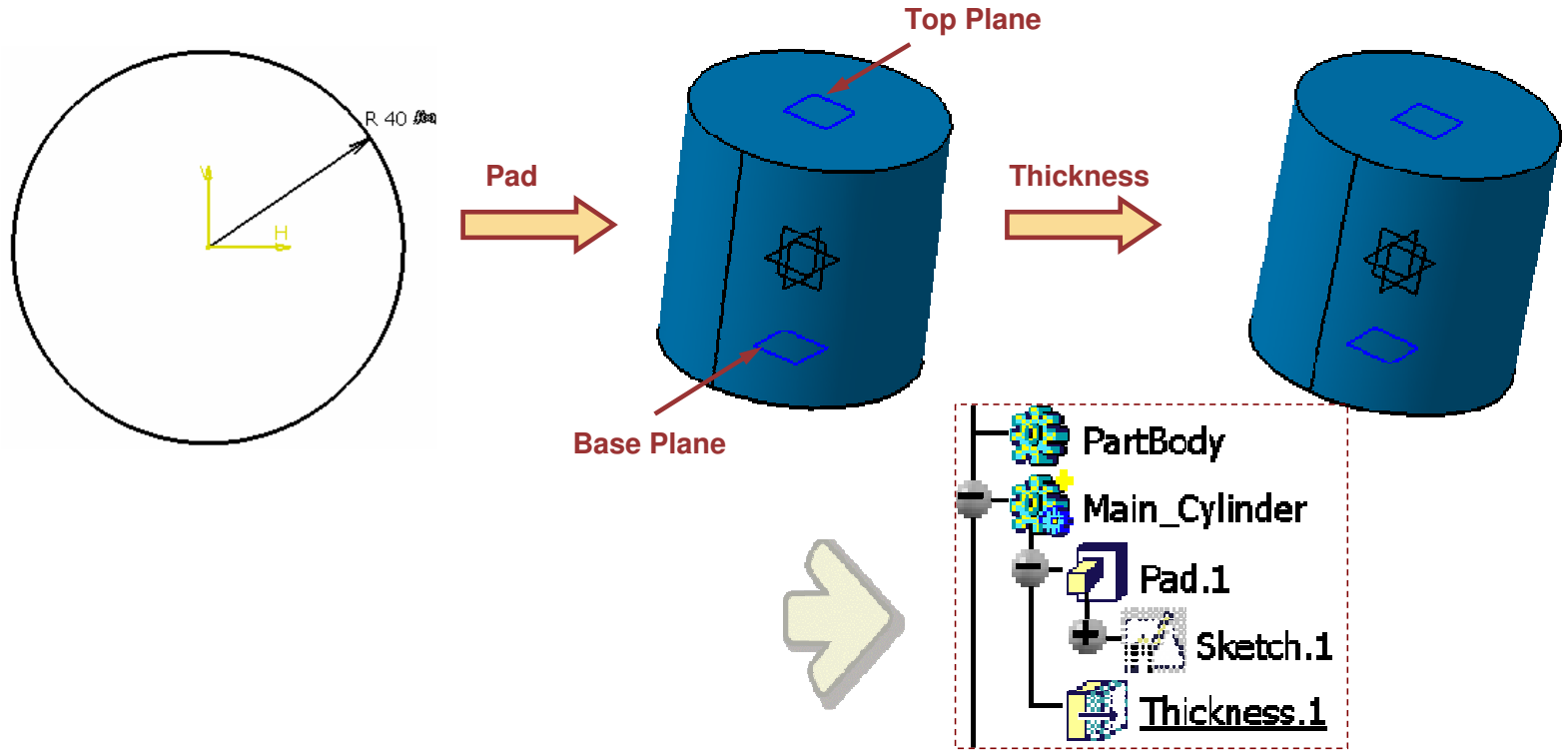


Student Notes:

Design the Main Cylinder

'PDG_Piston_step_3_start.CATPart'

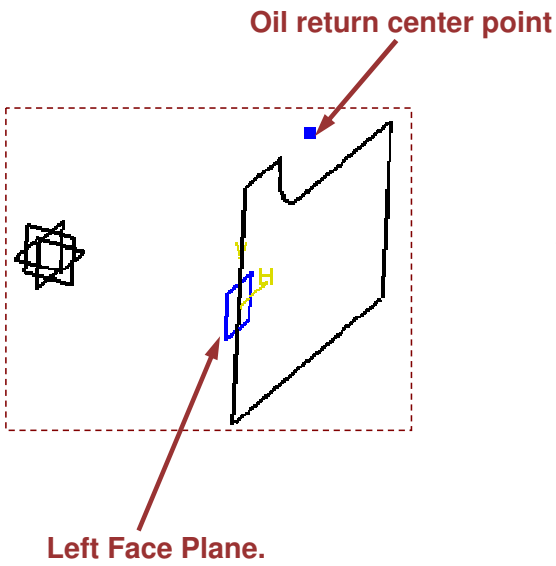
- Insert a Body 'Main_Cylinder'.
- Create the following sketch on XY plane. Apply formula to its radius. Assign Parameter 'Piston_Radius' to it. This is Sketch.1.
- Create a pad up to plane using first limit = 'Top_Plane' & second limit = 'Base_Plane'.
- Apply a Thickness of 2 mm to the all three faces.



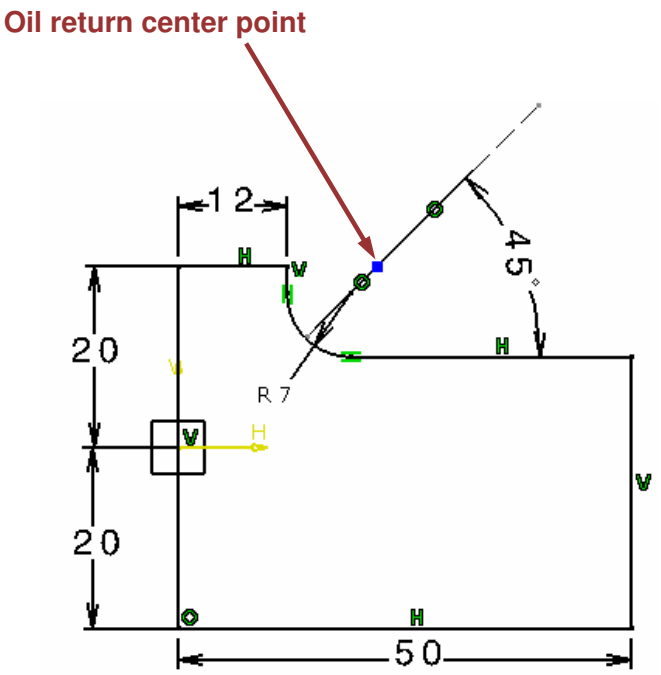
Student Notes:

Design the Cavity Pockets (1/3)

- Insert body 'Cavity Pockets'.
- Create a positioned sketch 'Cavity Pockets Sketch' in geometrical set on Left_Face_plane. Use Y axis as orientation.
- Use 'Oil_return_center'point to constrain the sketch.



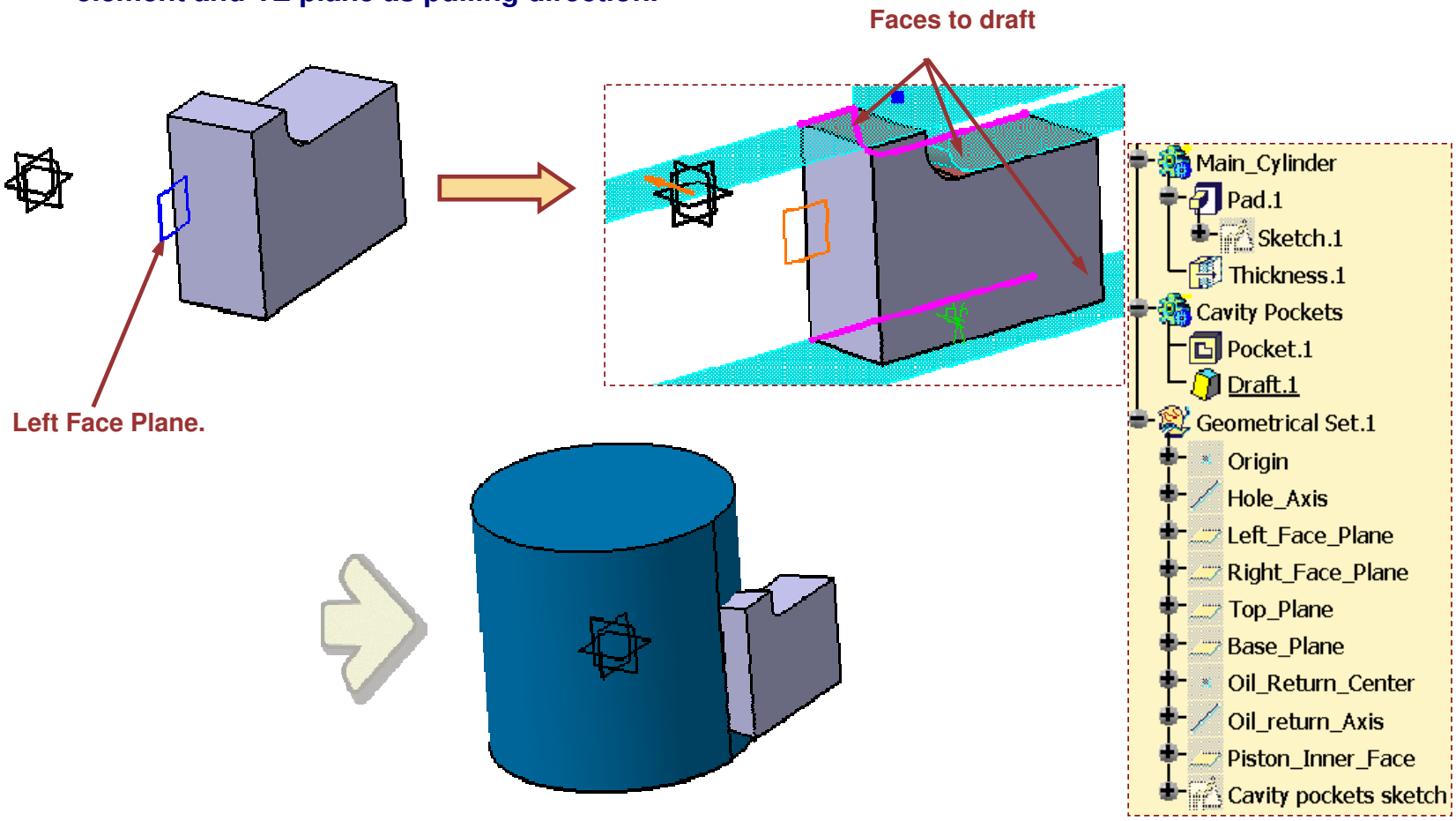
Sketch Details



Student Notes:

Design the Cavity Pockets (2/3)

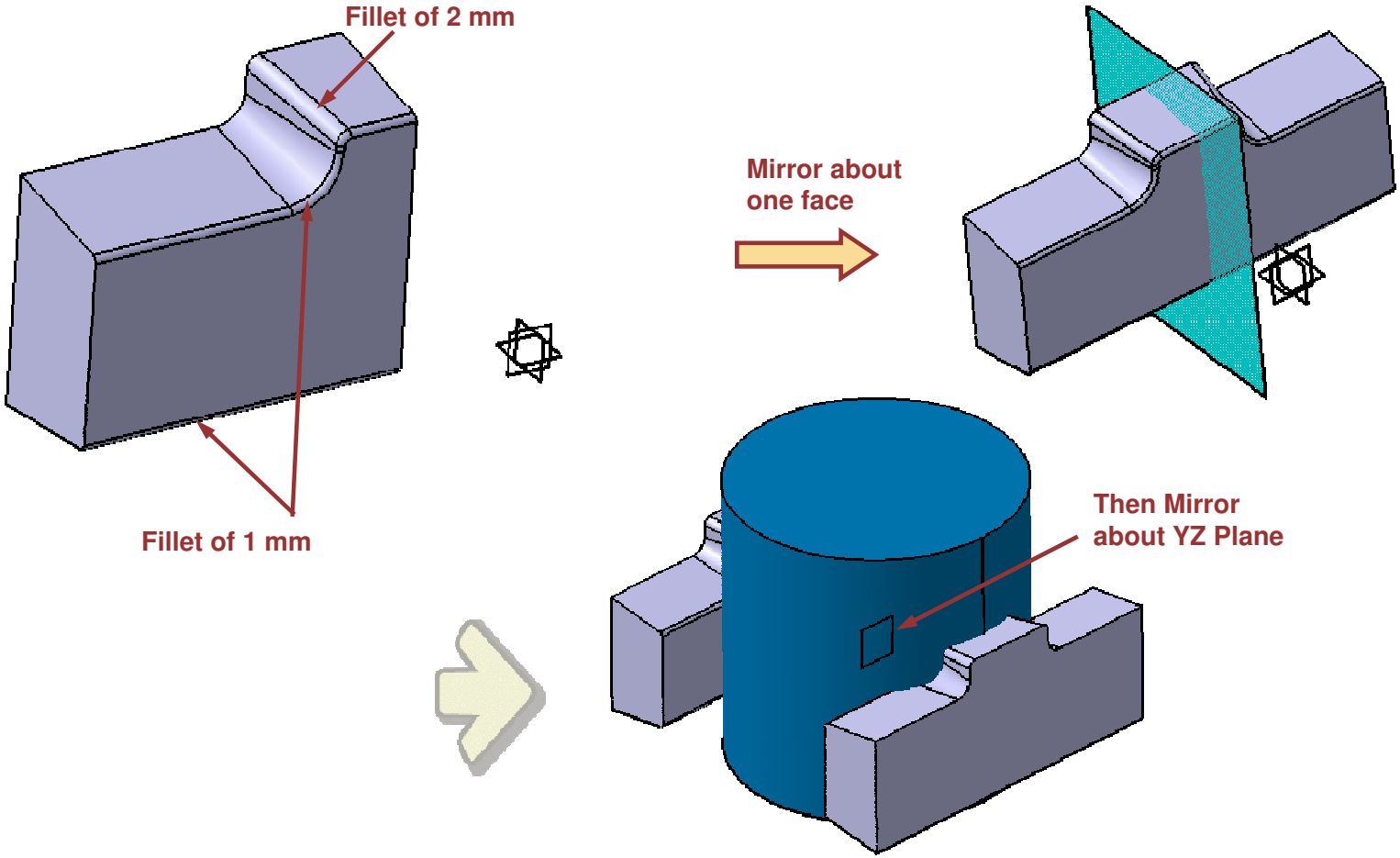
- In 'cavity pockets body' Pocket the sketch by 20 mm.
- Apply a draft of 8 deg to the faces shown.Use 'Left_Face_Plane' as neutral element and YZ plane as pulling direction.



Student Notes:

Design the Cavity Pockets (3/3)

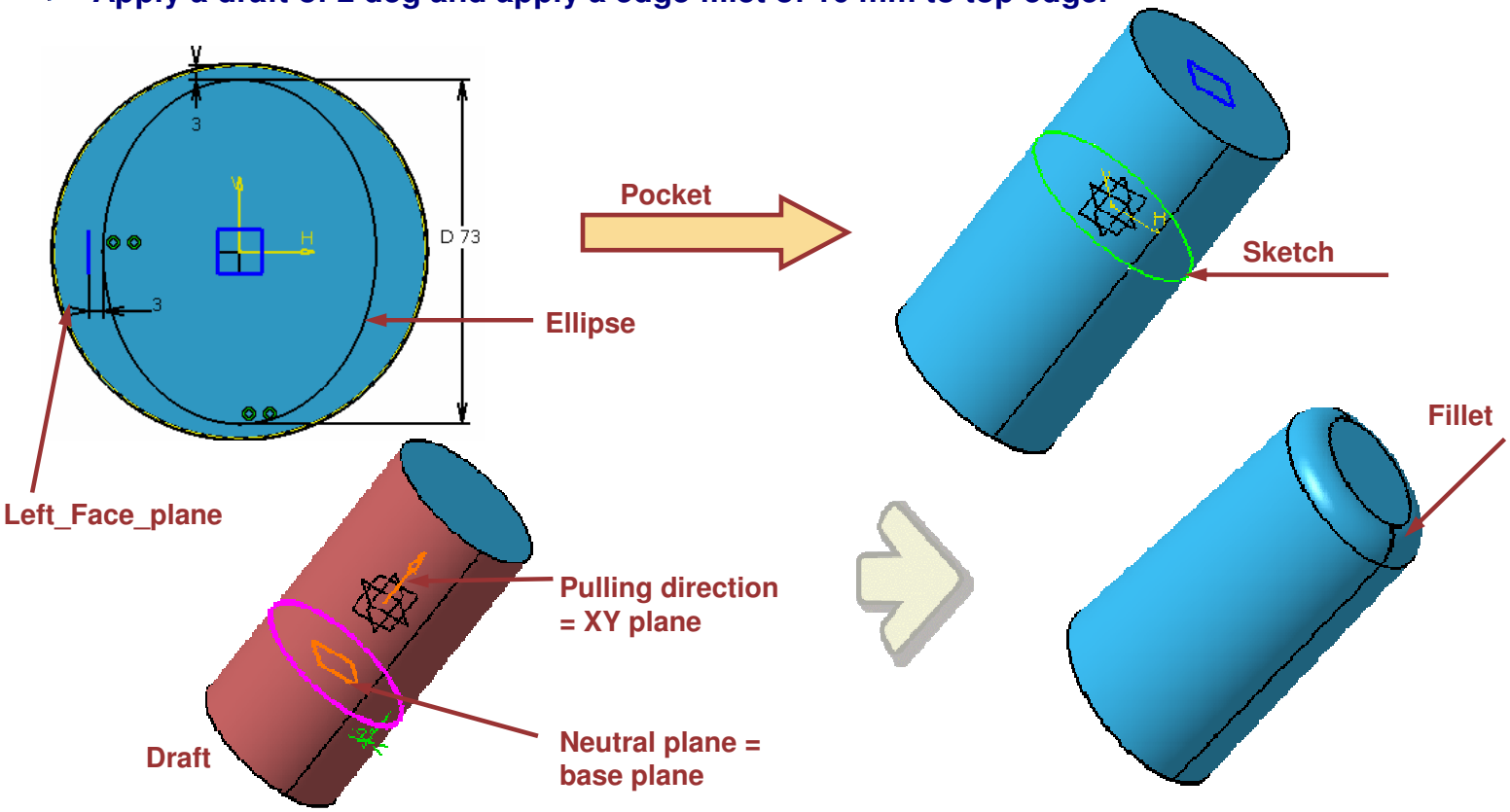
- Apply fillets as shown.
- Mirror it about pockets face and then mirror it about YZ plane



Student Notes:

Design the Inner Main Shape

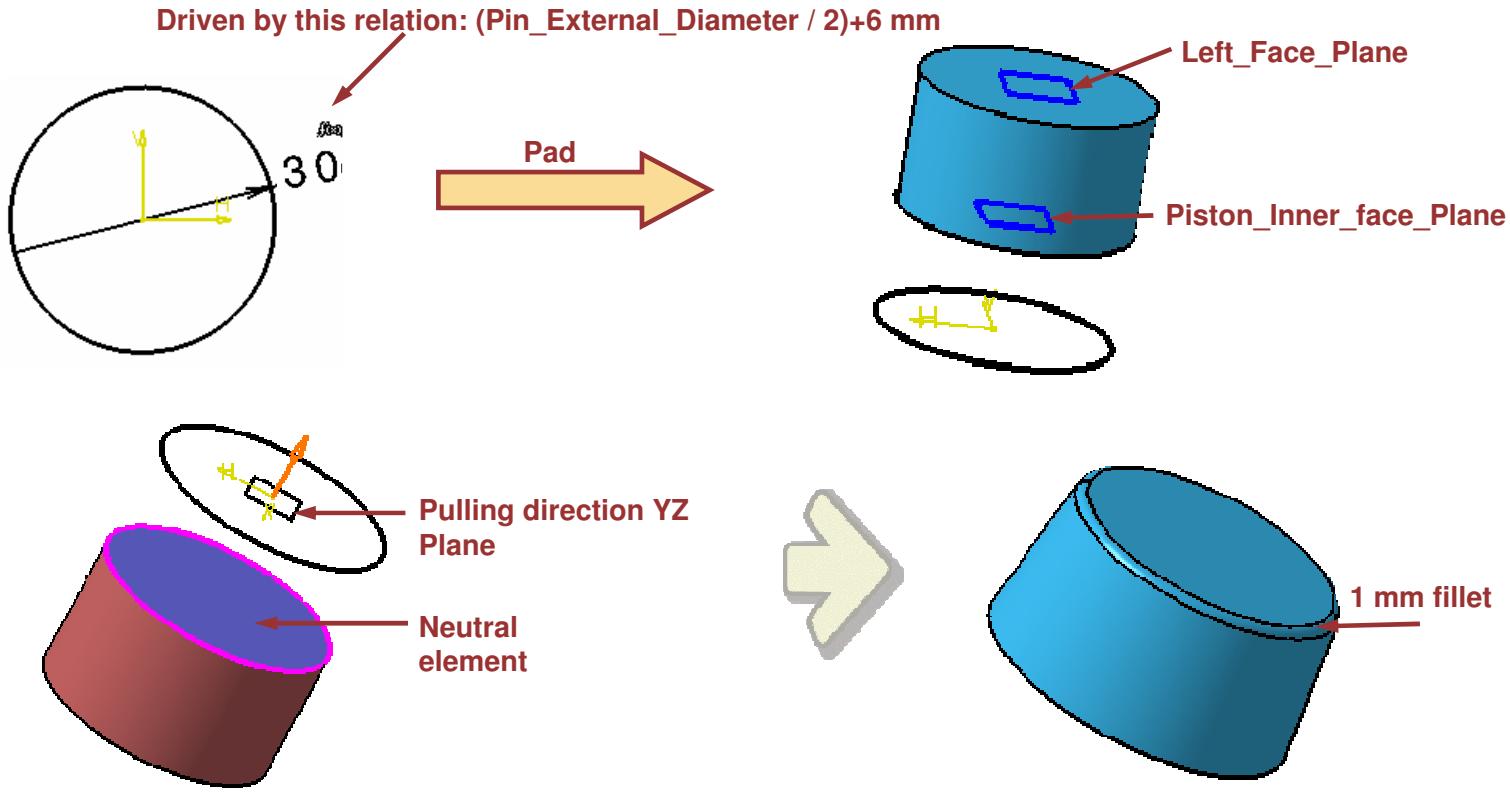
- Insert body 'Inner_Main_Shape'.
- Create a positioned sketch on XY plane. Use Left_Face_plane and sketch.1 to constrain it.
- In this body, create a Pocket from it, using first limit = Top_Plane with an offset = - 5 mm, and second limit as =70 mm.
- Apply a draft of 2 deg and apply a edge fillet of 10 mm to top edge.



Student Notes:

Design the Axis Holder

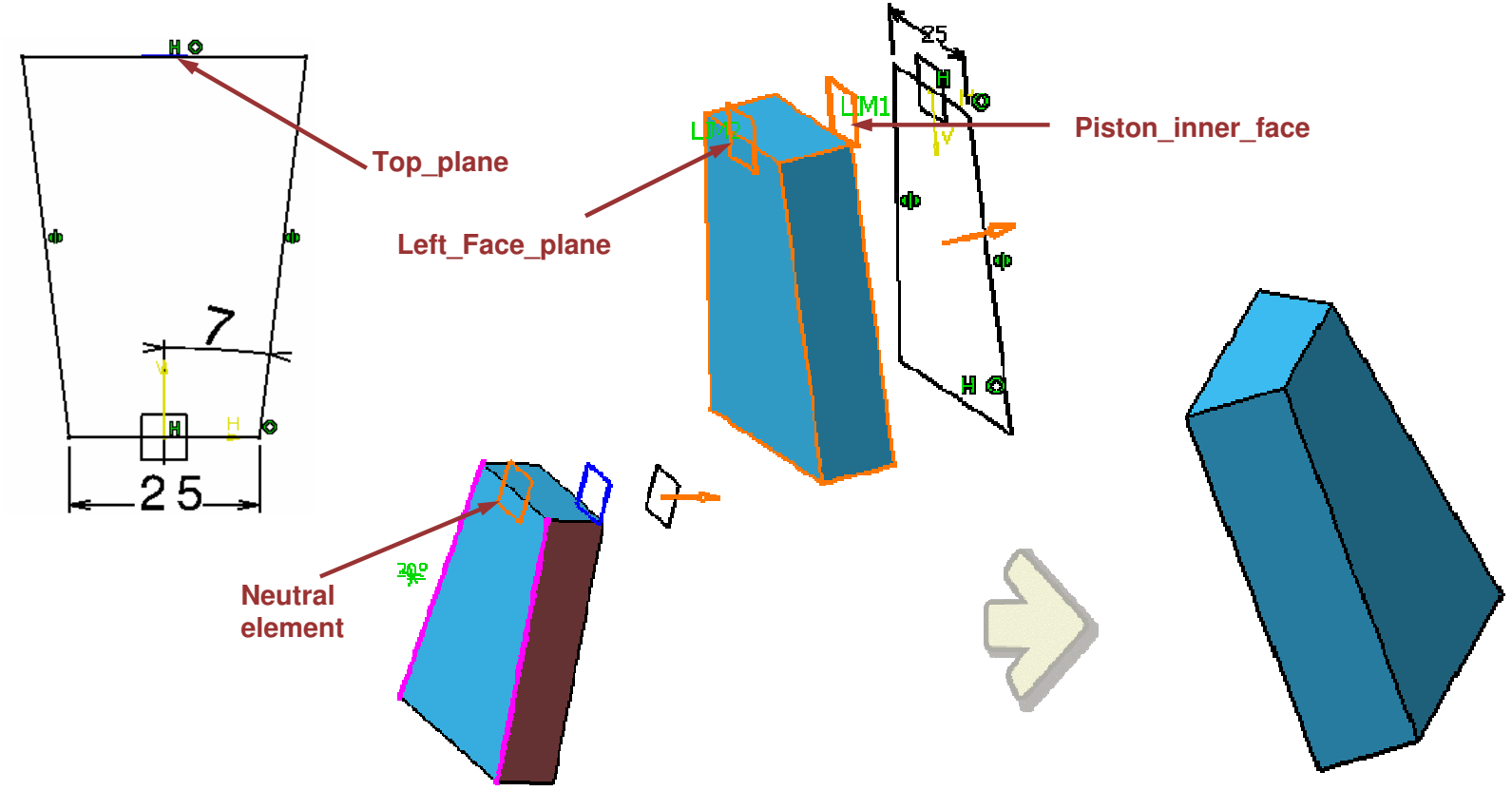
- Insert body 'Axis_Holder'.
- Create a positioned sketch on YZ plane with part origin.
- Create a Pad with 'Upto to Plane' type and use first limit 'Piston_Inner_face_plane'and second limit 'Left_face_plane'. This is Pad.2.
- Apply a draft of 7 deg.



Student Notes:

Design the Holder Stiffener

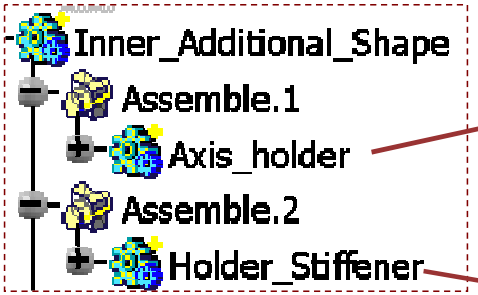
- Insert body 'Holder_Stiffener'.
- Create positioned sketch on YZ plane with part origin. Use Top_Plane to constrain it.
- Pad the sketch using first limit = Piston_inner_face with offset of – 5mm & second limit Left_Face_Plane.
- Apply draft of 20 deg to the two faces,with neutral element as Left_Face_Plane.

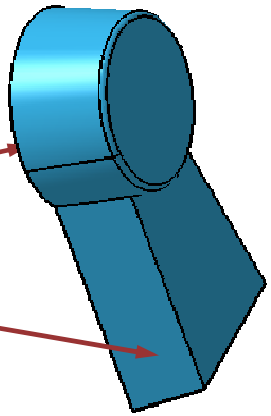


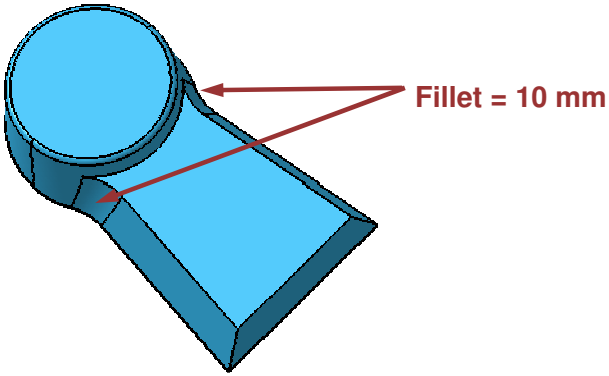
Student Notes:

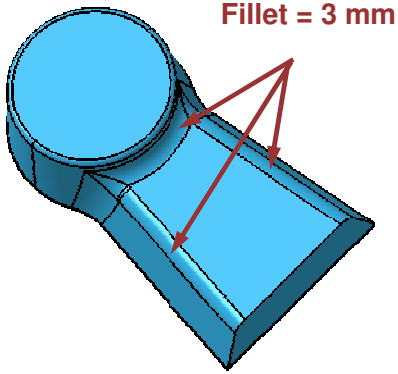
Design the Inner Additional Shape

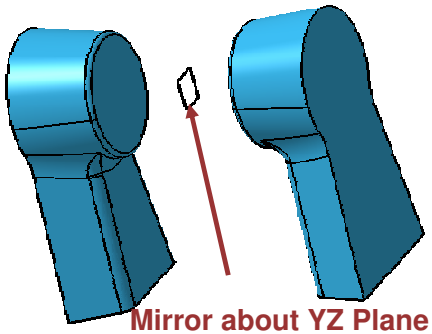
- Insert body 'Inner_Additional_Shape'.
- Assemble 'Axis_Holder' body with 'Inner_Additional_Shape' body.
- Assemble 'Holder_Stiffener' body with 'Inner_Additional_Shape' body after previous assemble operation.
- Apply edge fillets after assembling.
- Mirror about YZ plane.

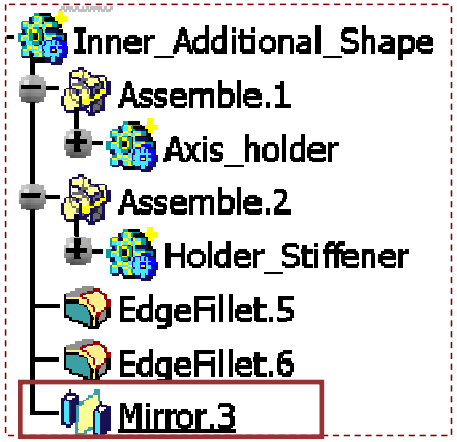








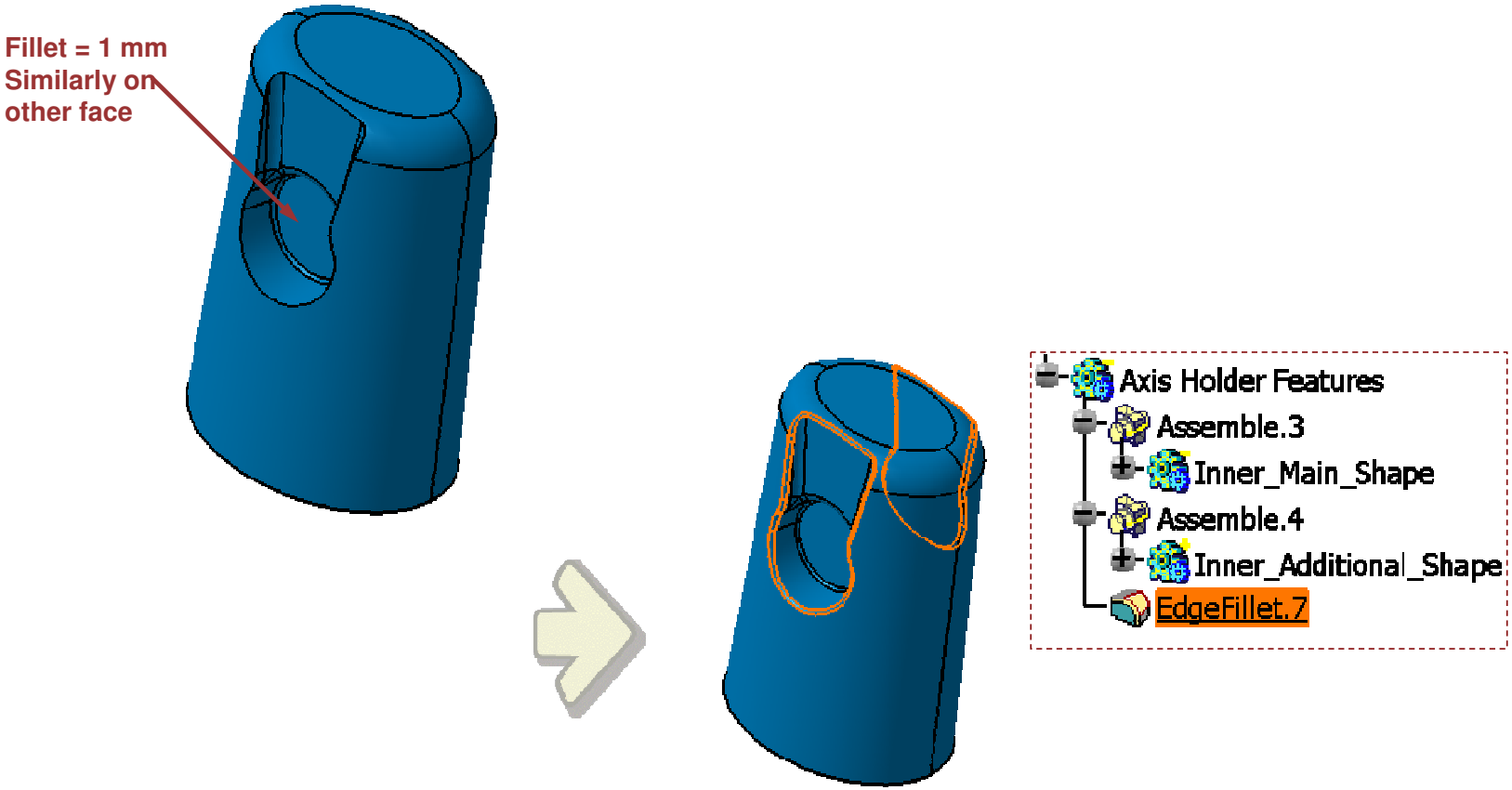




Student Notes:

Design the Axis Holder features

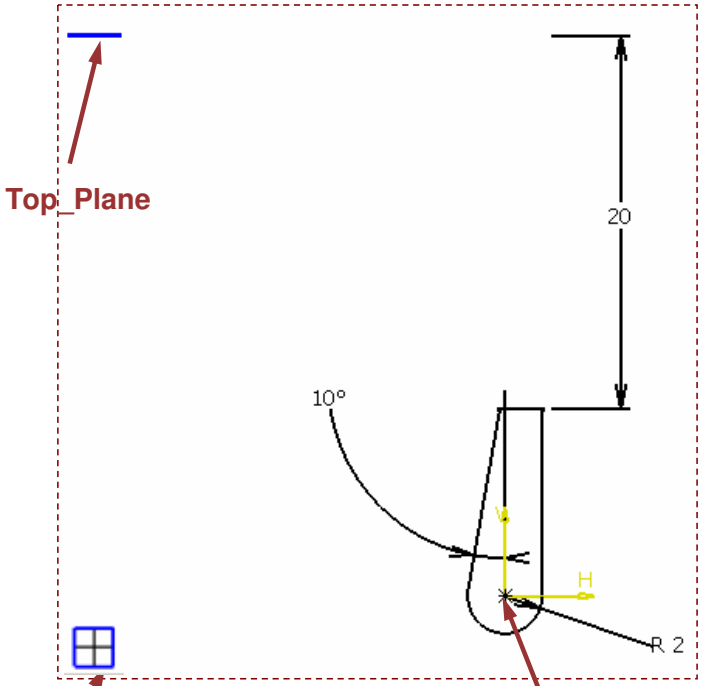
- Insert body 'Axis Holder Features'. Assemble 'Inner_Main_Shape' body & 'Inner_Additional_Shape' into it.
- Apply Edge fillet of 1 mm to two elements shown.

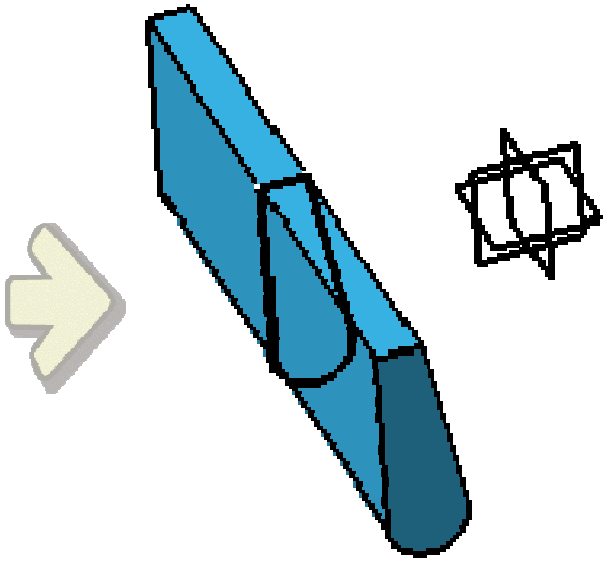


Student Notes:

Design the Oil Holes (1/2)

- Insert 'Oil_Holes' body.
- Create a Positioned sketch in geometrical set as shown on 'Left_Face_Plane'. Use 'Oil_Return_center' as projection point.
- Constrain the sketch using Top_Plane and
- Pocket it by 15mm (Mirrored extent).

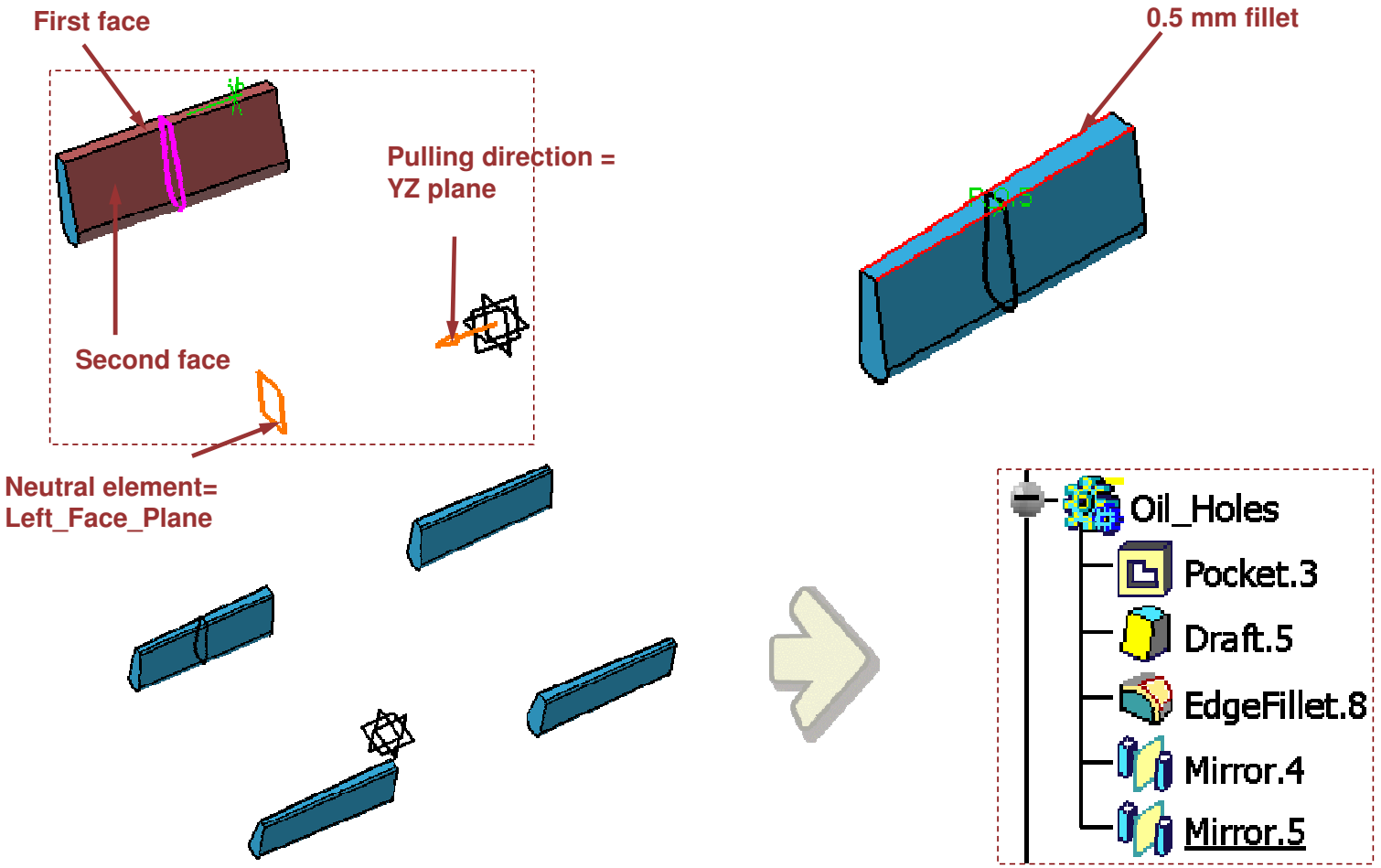




Student Notes:

Design the Oil Holes (2/2)

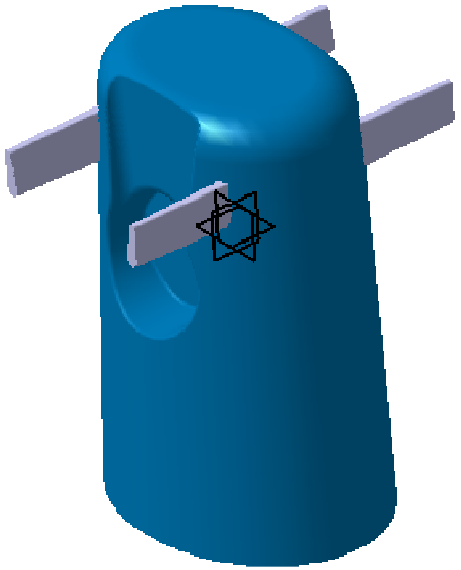
- Apply Draft of 1 deg to the two faces.
- Apply a fillet. Mirror the result about ZX and then about YZ plane



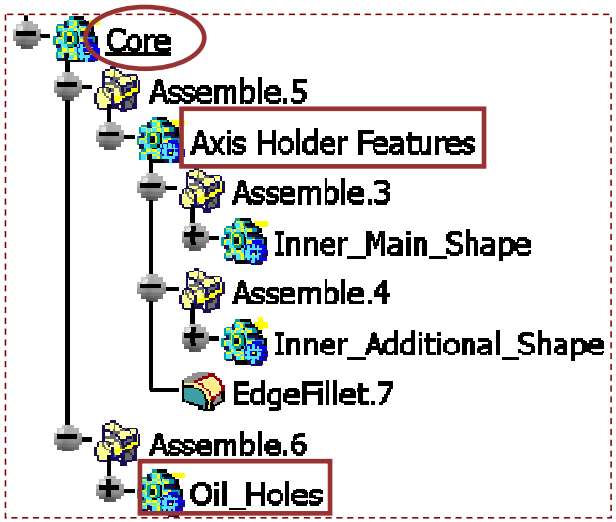
Student Notes:

Design the Core

- Insert body 'Core'.
- Assemble the 'Axis Holder features' body and 'Oil_Holes' body with the 'core' body.



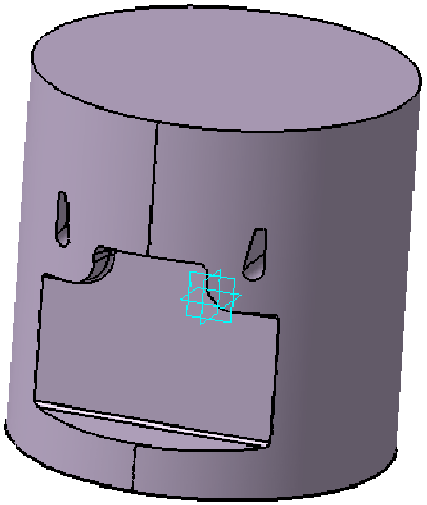
Core Body



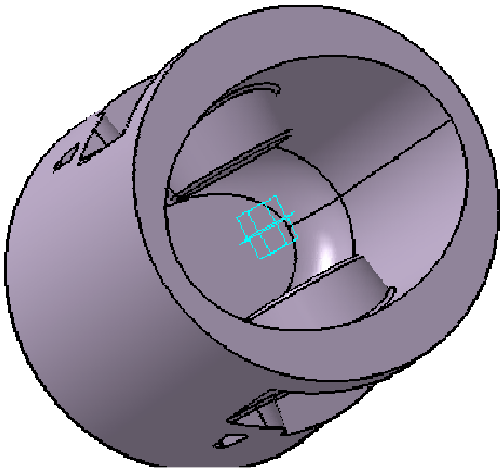
Student Notes:

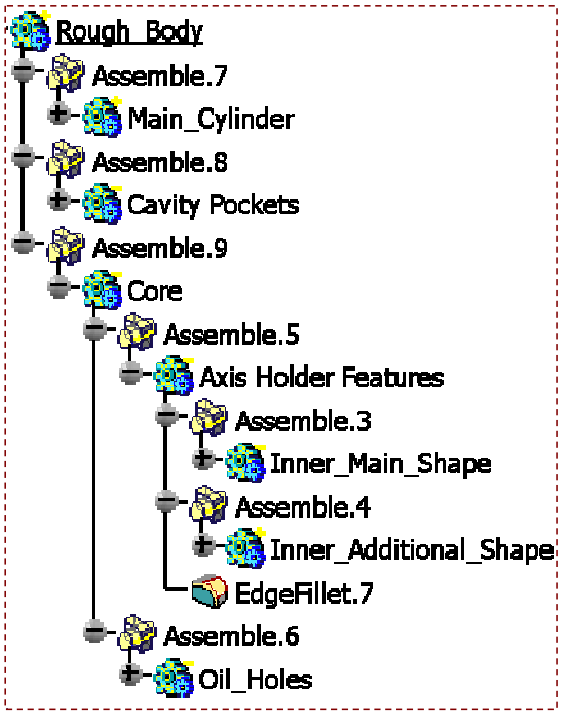
Design the Rough Body

- Insert body 'Rough_Body'.
- Assemble 'Main Cylinder', 'Core', 'Cavity pockets' bodies with it.



Rough Body





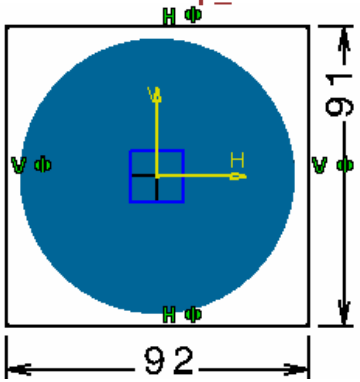
Student Notes:

Design the External Machining Feature

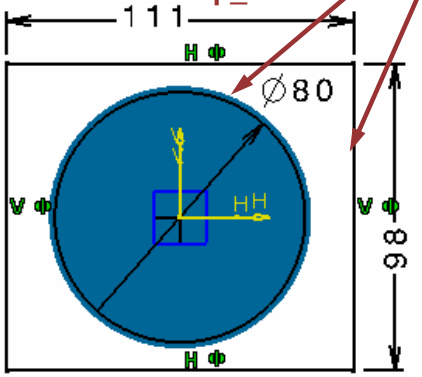
- Insert 'External Machining' body.
- Create 3 different sketches in Geometrical set as shown.
- Create 3 different pockets in 'External Machining' body.
- Apply a Chamfer of 1 mm x 45 deg to face of pocket.5.

Sketch contains a circle and a rectangle

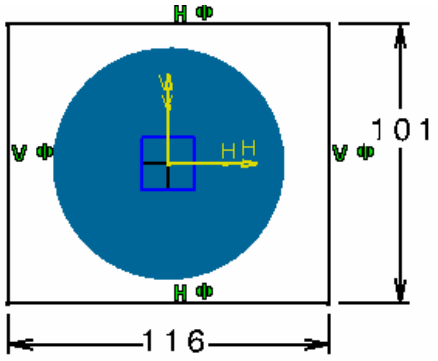
Sketch on Top_Plane

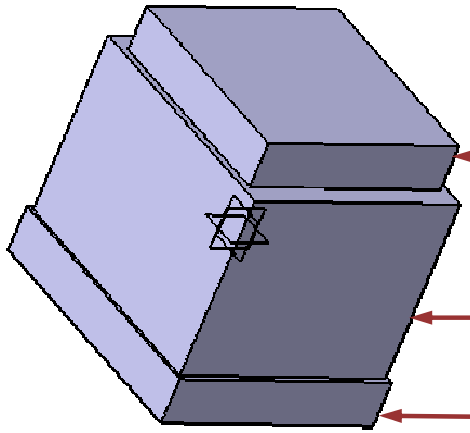


Sketch on Top_Plane



Sketch on Base_Plane



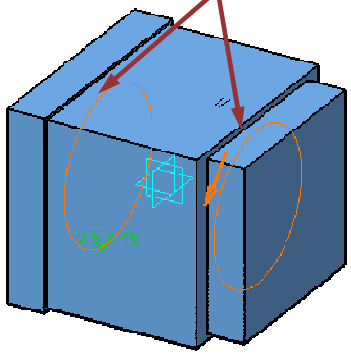


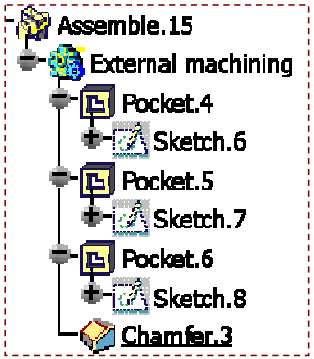
Pocket = 20 mm using I st sketch

Pocket upto Base Plane using 2nd sketch

Pocket =20 mm using 3 rd sketch.

Chamfer



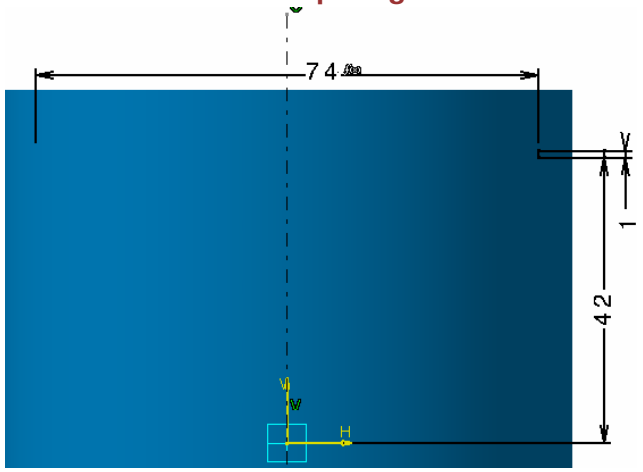


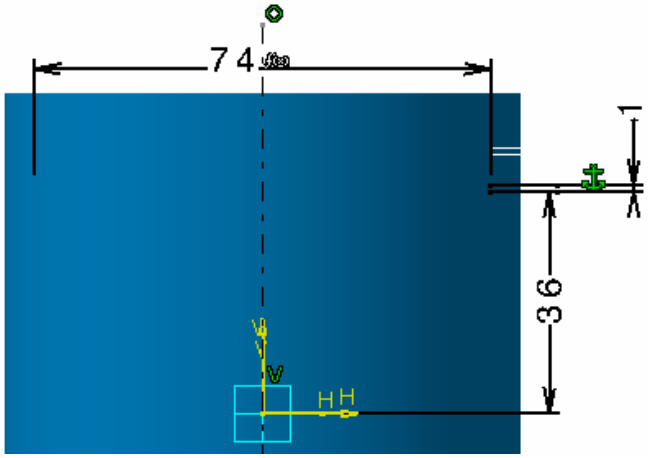
Student Notes:

Design the Piston Ring Slots (1/2)

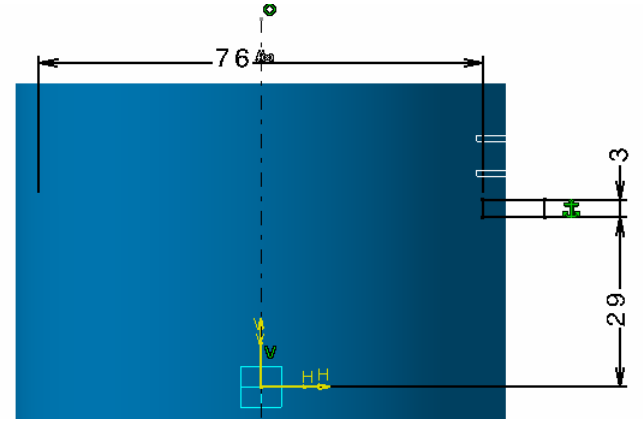
- Insert body'Piston Ring Slots'.
- Create 3 different positioned sketches in geometrical set on YZ plane.
- Apply Formulae by relating to'Top_and_Middle_Ring_Slot_Radius' 'Bottom_Ring_Slot_Diameter' parameters.

Top Ring Slot Sketch





Middle Ring Slot Sketch

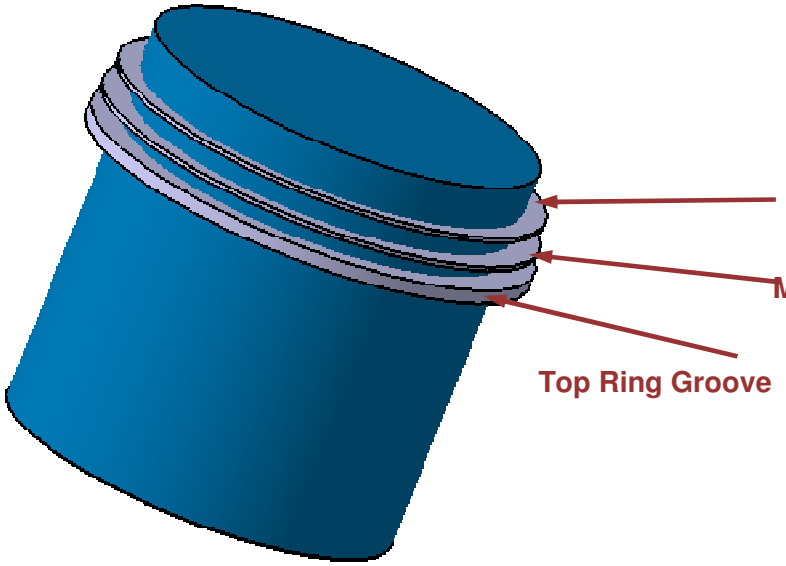


Bottom Ring Slot Sketch

Student Notes:

Design the Piston Ring Slots (2/2)

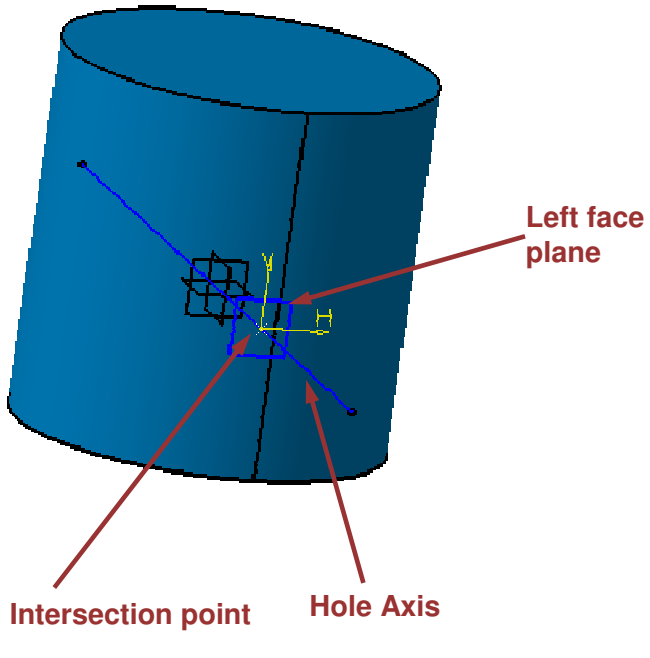
- Groove the three sketches.

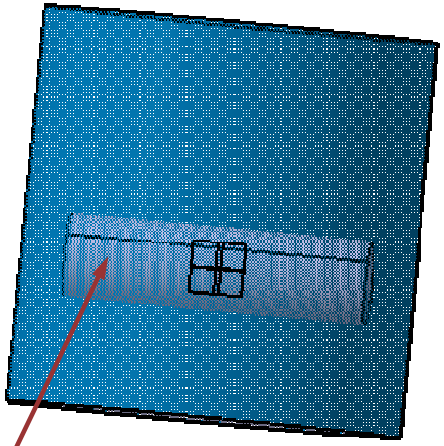


Student Notes:

Design the Axis Hole

- Insert body ' Axis Hole'.
- Create a sketch on left face plane.
- Intersect the left face plane with Hole axis. This point is the center point of axis hole.
- Create a hole using upto plane (Select Right Face plane) with a diameter of 18 mm. The diameter is driven by 'Pin external Diameter' parameter.

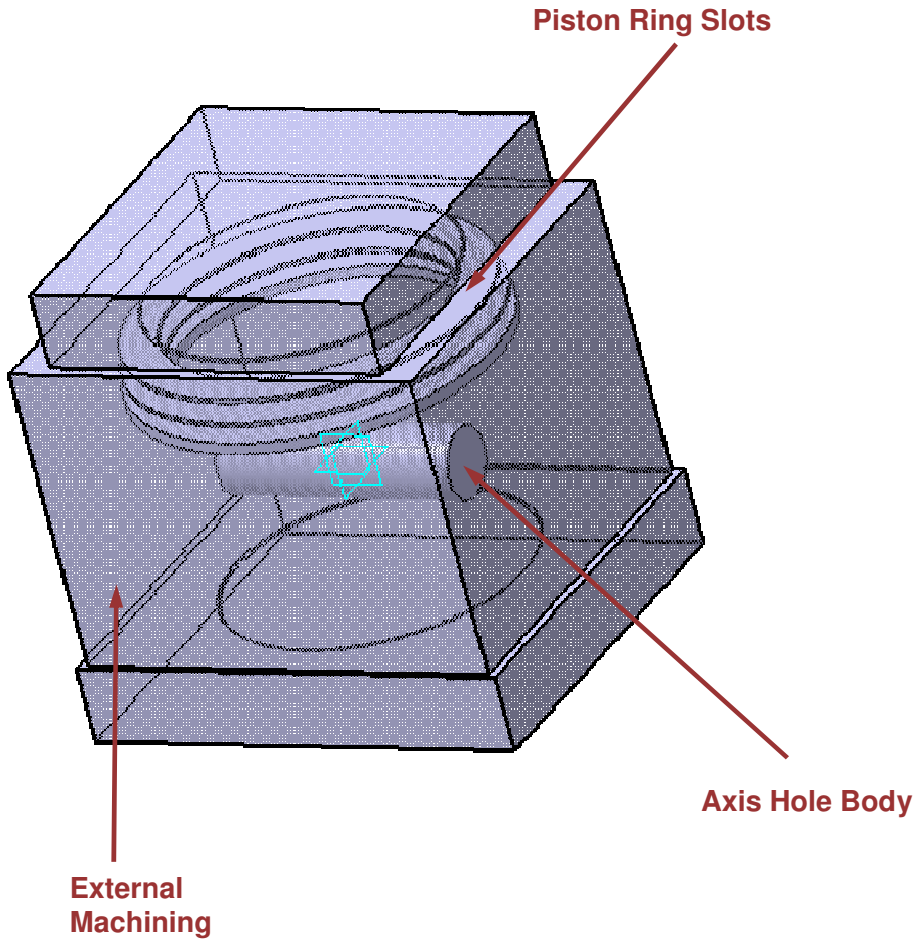


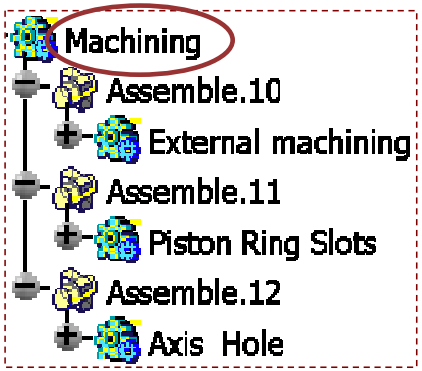


Student Notes:

Design the Machining features

- Insert 'Machining' body.
- Assemble 'External Machining', 'Piston Rings Slots', 'Axis_Hole' bodies with it.



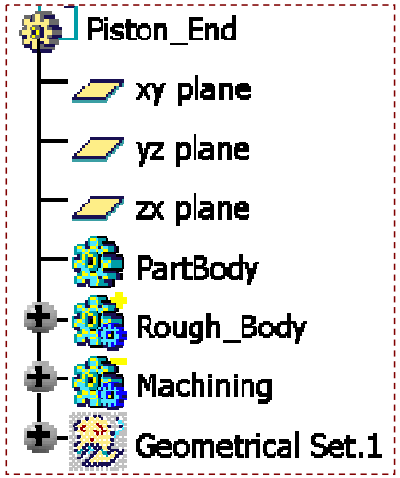


Contents of Machining Body

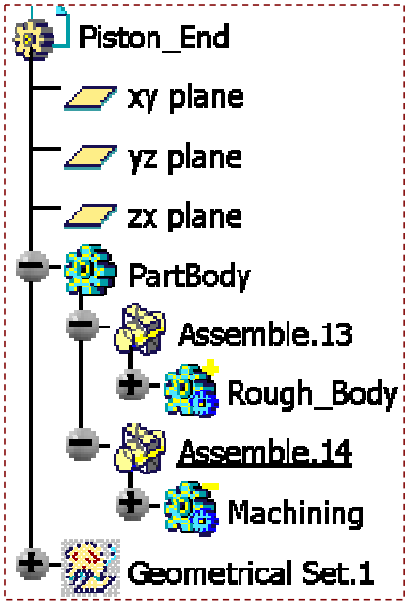
Student Notes:

Completing and Optimizing the Design

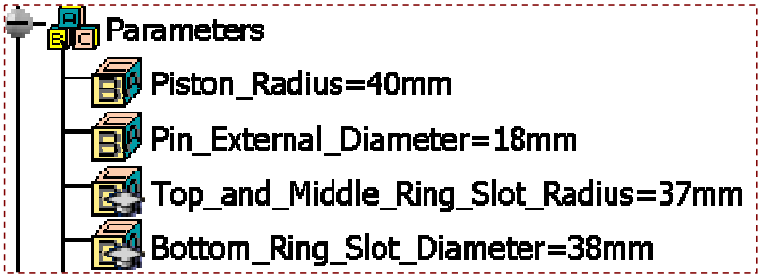
- Assemble 'Rough Body' and 'Machining Body' with PartBody.
- Optimize the design by changing parameter values.
- Enter different combinations of values using parameters and choose the best design.



Before assembling



After assembling



Modify to Optimize the Design

Result: 'PDG_Piston.CATPart'

Student Notes:

Side Toolhead

Part Design Advanced Exercise

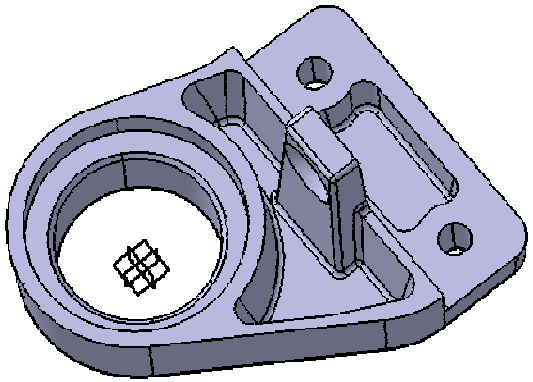
60 min

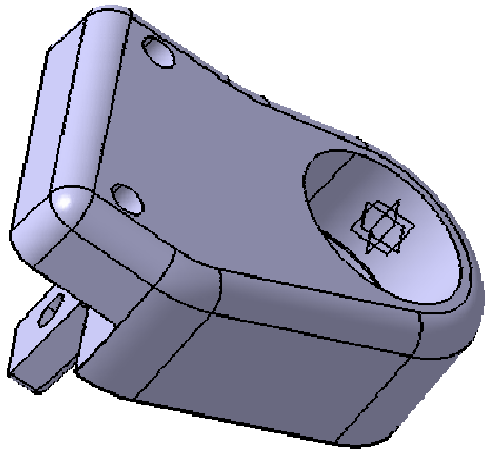
In this exercise you will build the Side Toolhead by following a recommended process.

- **You will first understand the design intent of the Side Toolhead and identify its functional features from the drawing.**
- **You will then study its structure to decide your own design process.**
- **Finally, you will design the various functional features according to specifications and by making use of wireframe elements.**

Here you will:

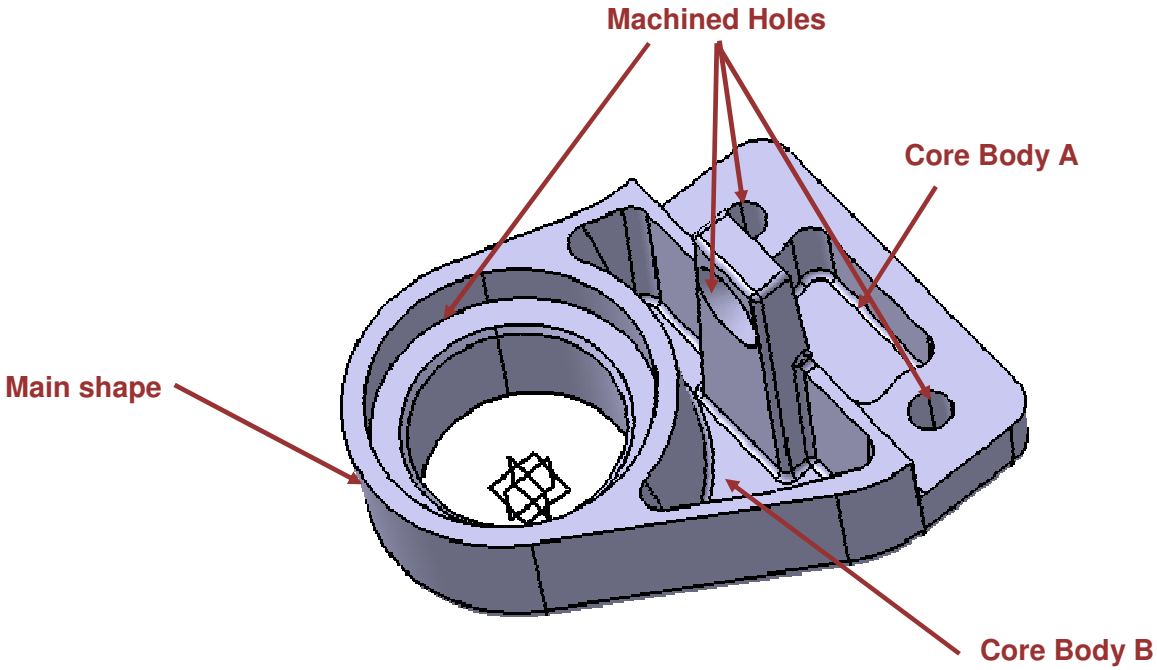
- **Design the Rough Body**
- **Design the Machined Body**
- **Assemble the Results**





Student Notes:

Design intent: Side Toolhead

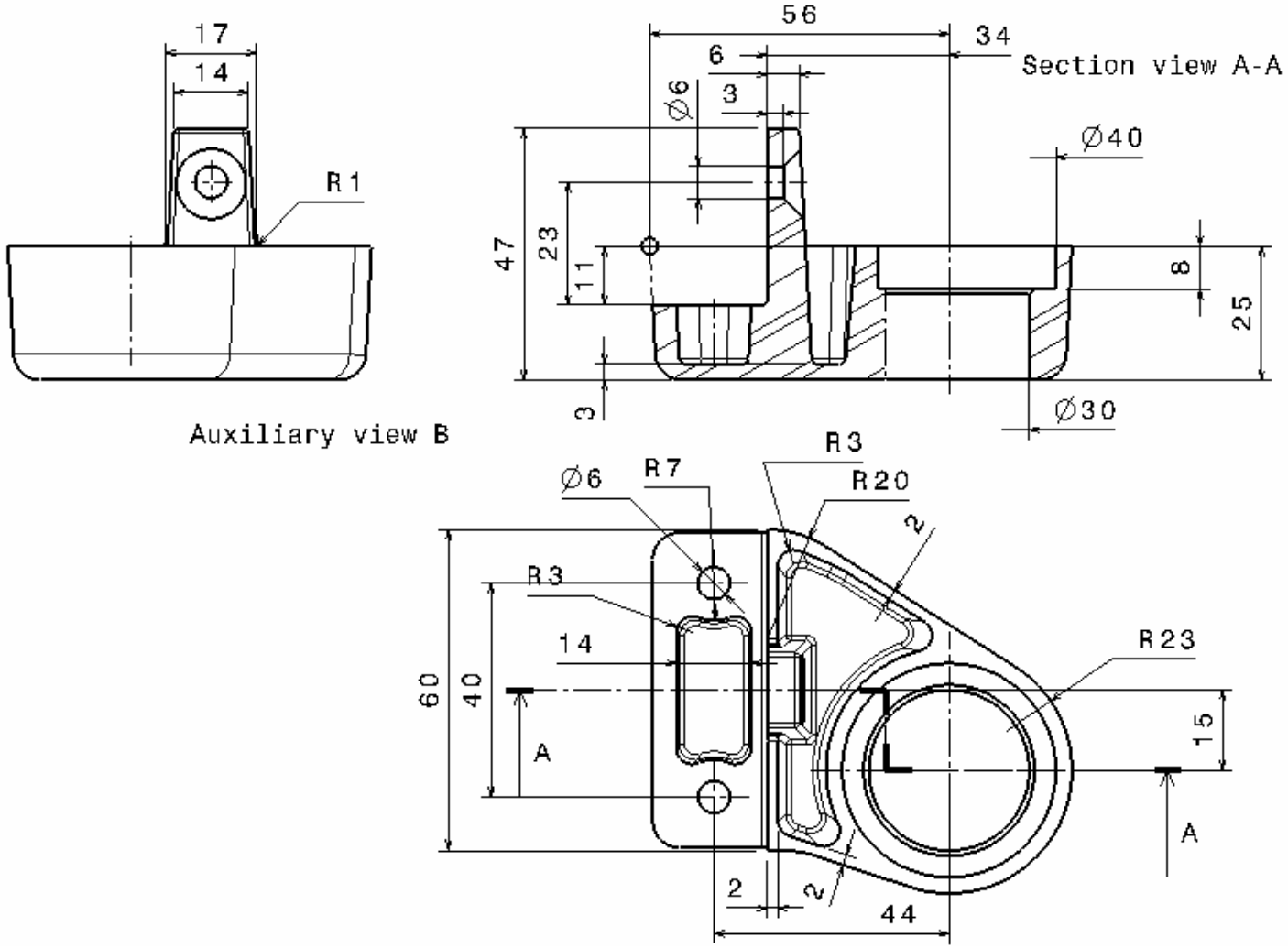


- Side Toolhead is a Casted Part.

Student Notes:

Side Toolhead Drawing

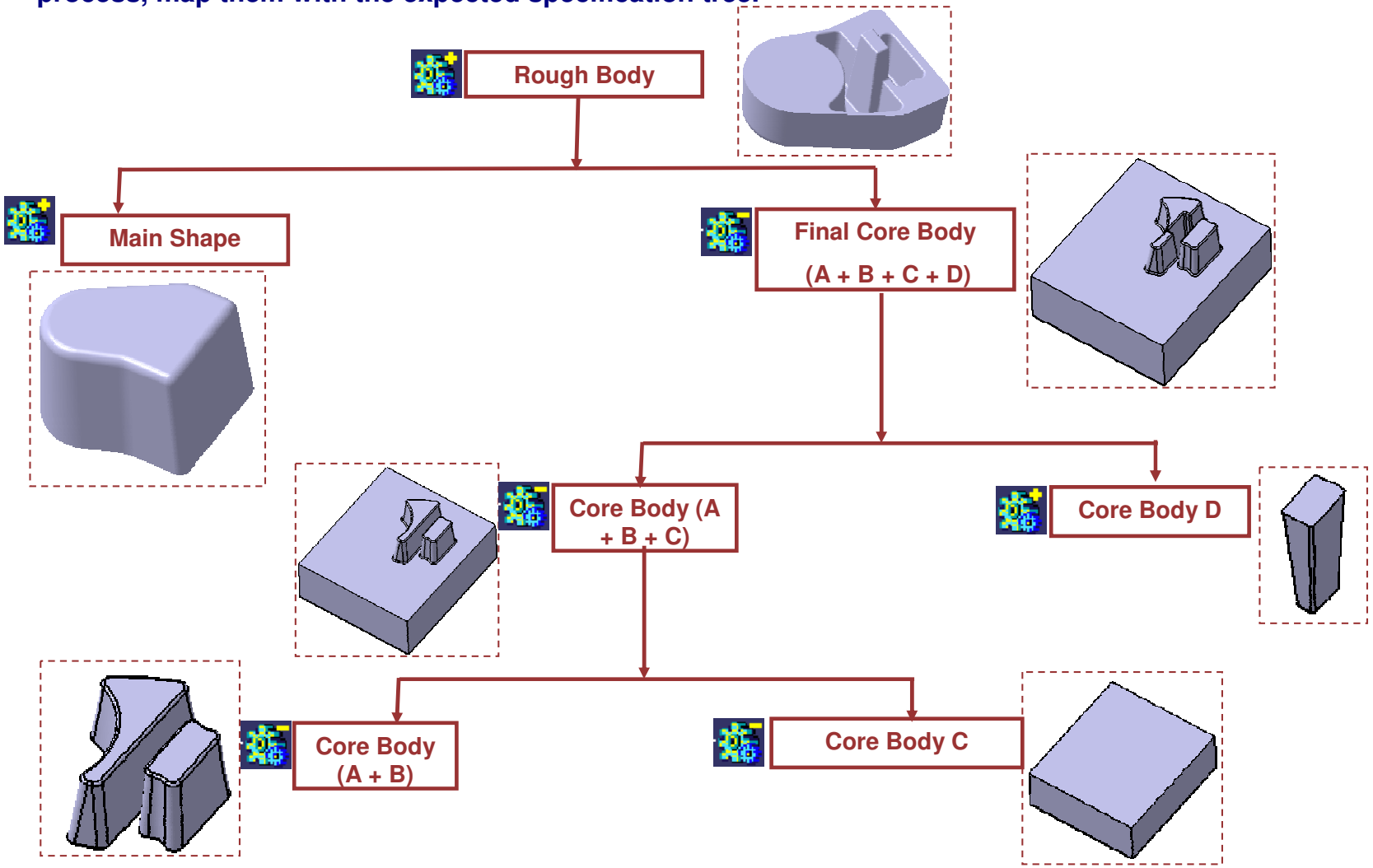
- Design the Side Toolhead as per the specifications in the Drawing.



Student Notes:

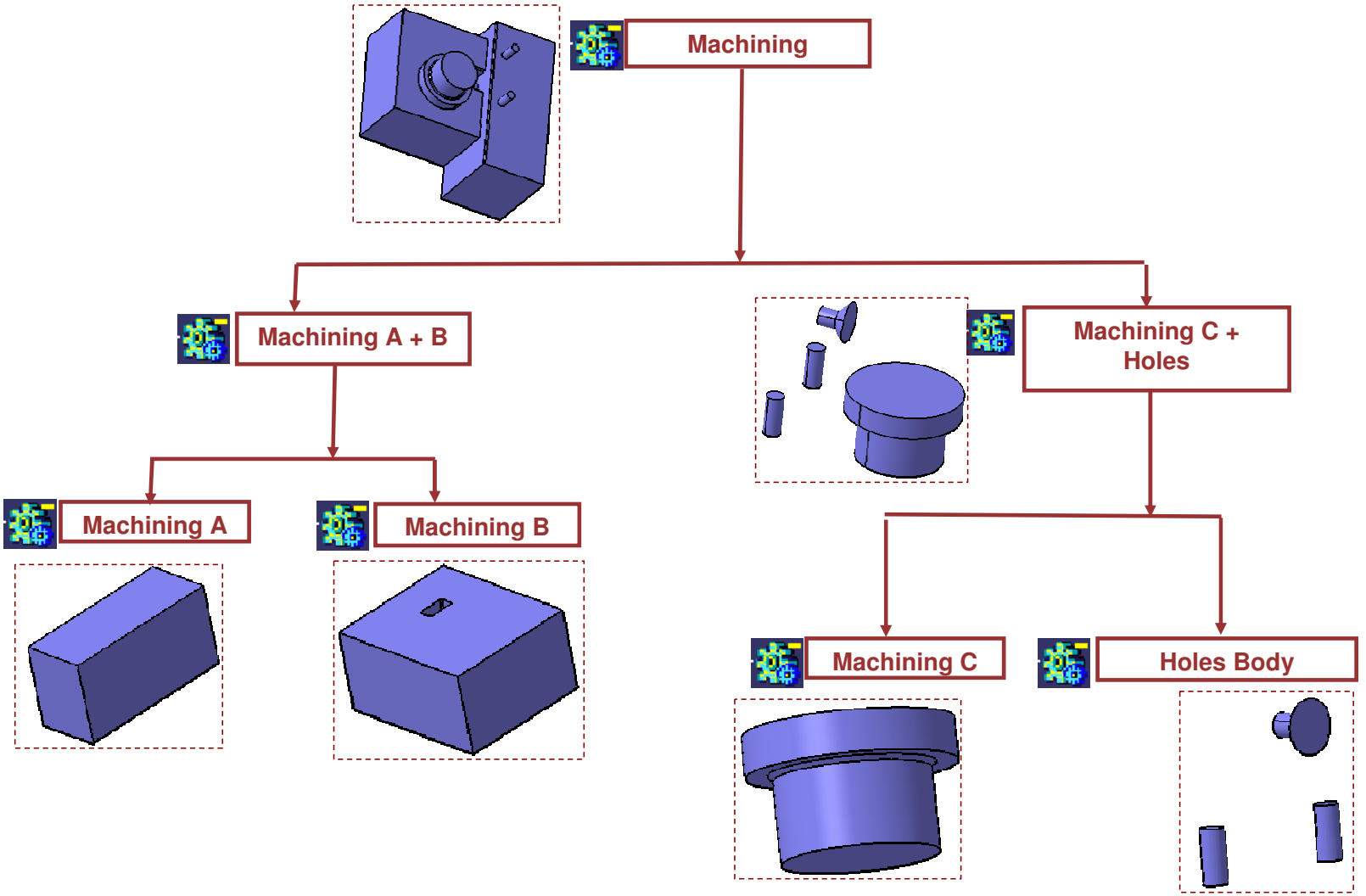
Structure of the Rough Body

- Identify the bodies and Boolean operations needed. Also decide your own design process, map them with the expected specification tree.



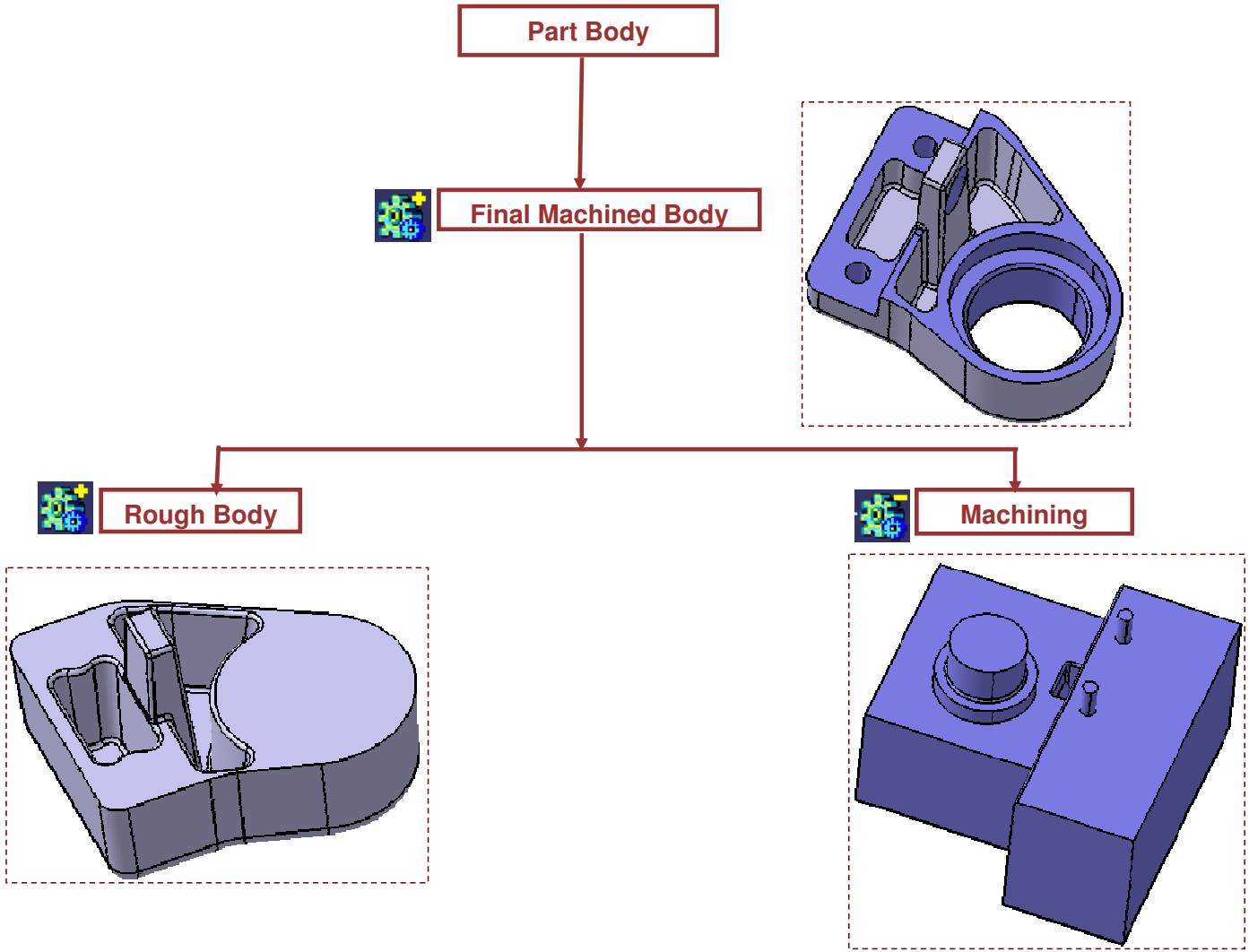
Student Notes:

Structure of the Machining



Student Notes:

Structure of the Part Body



Student Notes:

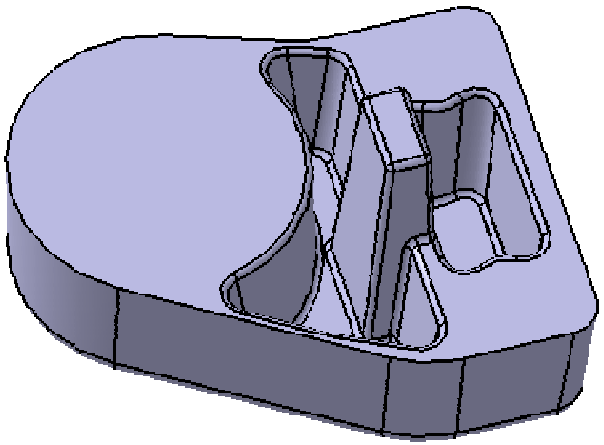
Side Toolhead

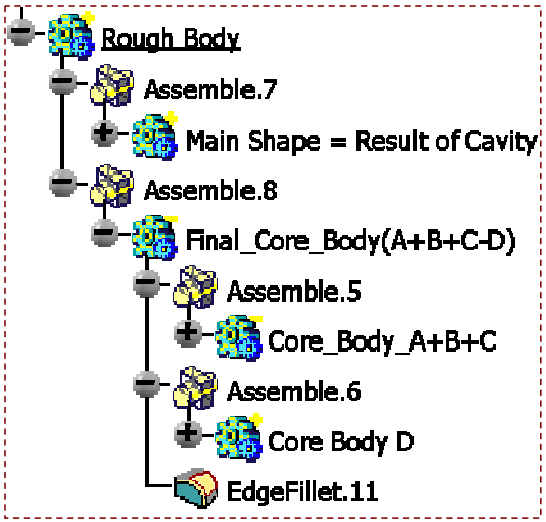
Step 1: Design the Rough Body

30 min

In this step you will start designing the Rough Body for the Side Toolhead according to specifications. In this step you will:

- Design the Wireframe Elements for Rough Body
- Design Core Bodies A, B, C
- Assemble these bodies



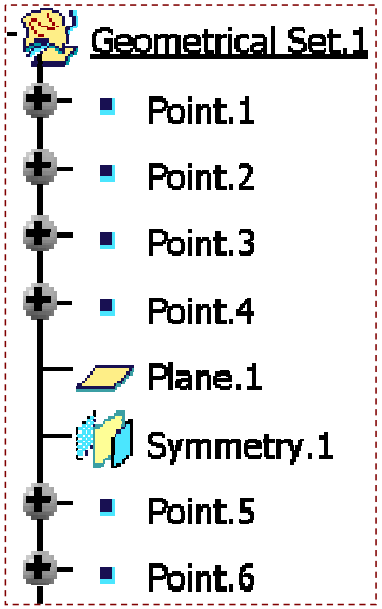


Student Notes:

Design the Wireframe Elements for Rough Body (1/3)

- Here you will create some primary wireframe elements to start the design of the Rough Body. You will create remaining wireframe elements on the fly.
- You will create 7 points, 2 lines, 3 planes, 2 symmetries.
- To create symmetries you need to access 'Generative Shape Design' workbench

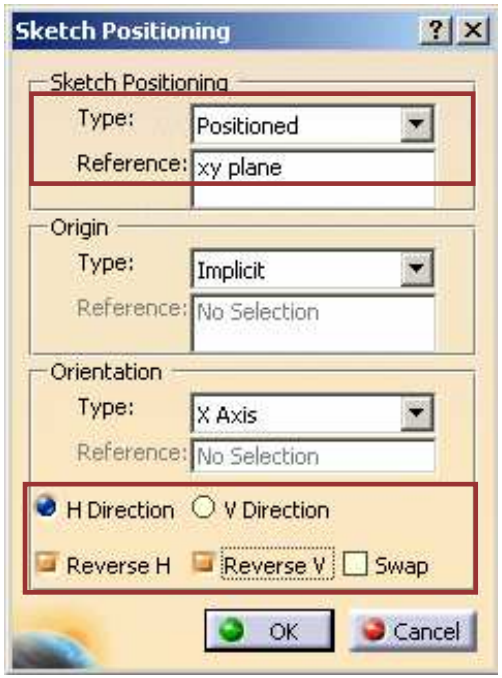
Wireframe Element	Parameters	Parent(s)
Point.1	(15, 34, 0)	Origin
Point.2	(20, 0, 0)	Point.1
Point.3	(10, -6, 0)	Point.2
Point.4	(0, 0, 37)	Point.1
Plane.1	Parallel through Point	YZ Plane, Point.4
Symmetry.1	---	Point.3 about Plane.1
Point.5	(0, 10, 0)	Point.2
Point.6	(0, 12, 0)	Point.5

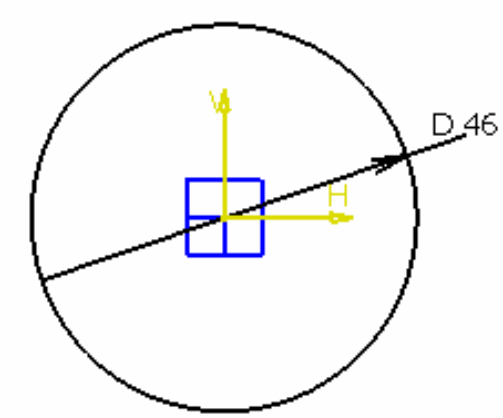


Student Notes:

Design the Wireframe Elements for Rough Body (2/3)

- Now, Create a Positioned Sketch on XY plane as shown.



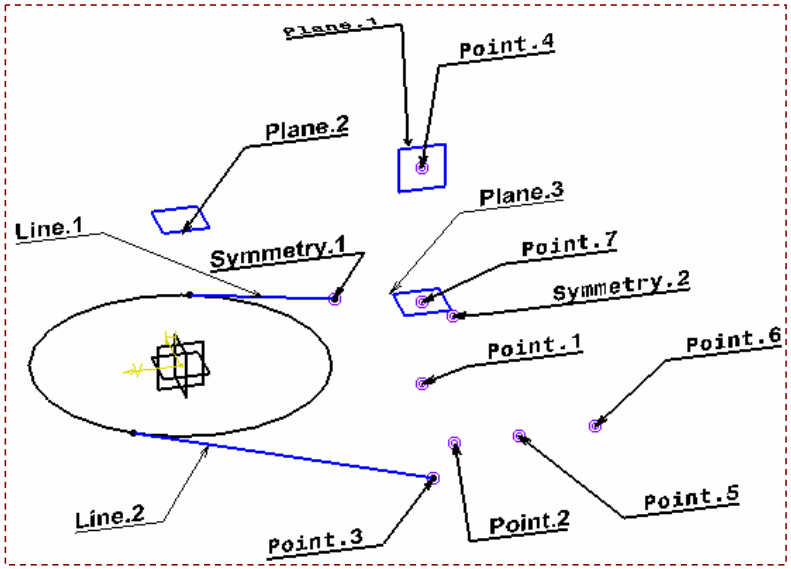


Student Notes:

Design the Wireframe Elements for Rough Body (3/3)

- Create the following wireframe elements

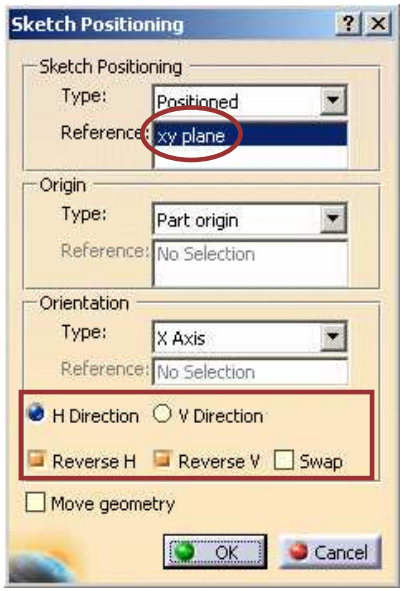
Wireframe Element	Parameters		Parent(s)
Line.1	Tangent to Curve, Bi-Tangent option	Use Next solution tab to get required result	Sketch.1, Symmetry.1
Line.2	Tangent to Curve, Bi-Tangent option		Sketch.1, Point.3
Plane.2	Offset by 25 mm		XY Plane
Symmetry.2	---		Point.5 about Plane.1
Point.7	(0, 0, 14)		Point.1
Plane.3	Parallel through Point		XY Plane, Point.7

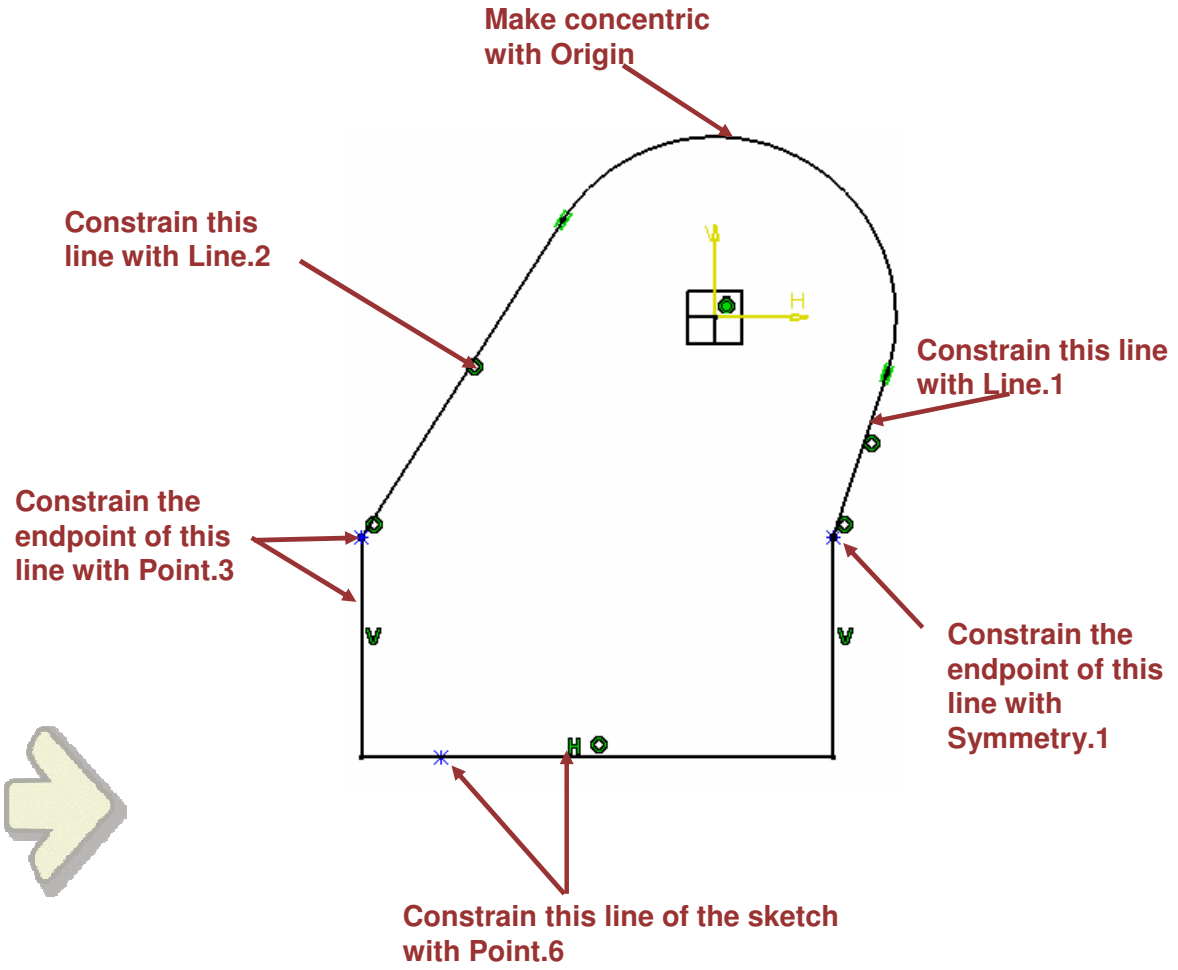


Student Notes:

Design the Main Shape (1/3)

- Create Positioned Sketch.2 in Geometrical Set on XY plane.

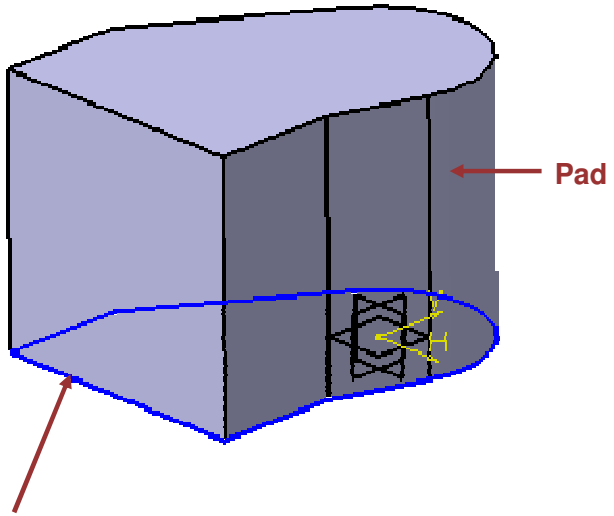


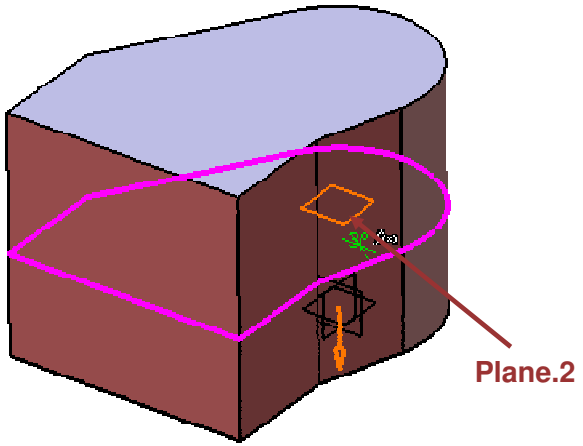


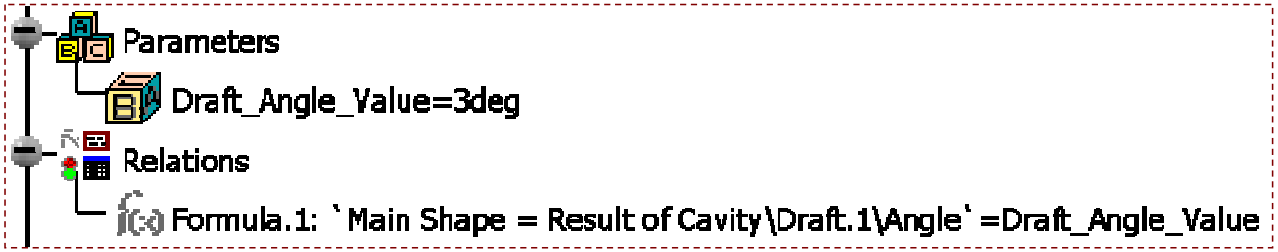
Student Notes:

Design the Main Shape (2/3)

- Insert Body 'Main Shape'. This Body is the result of the Cavity
- Create a Pad of 60 mm from Sketch.2.
- Apply a Draft of 3 deg to all the vertical four faces shown. This is driven by a parameter 'Draft_Angle_Value'
- Select Plane.2 as neutral element.



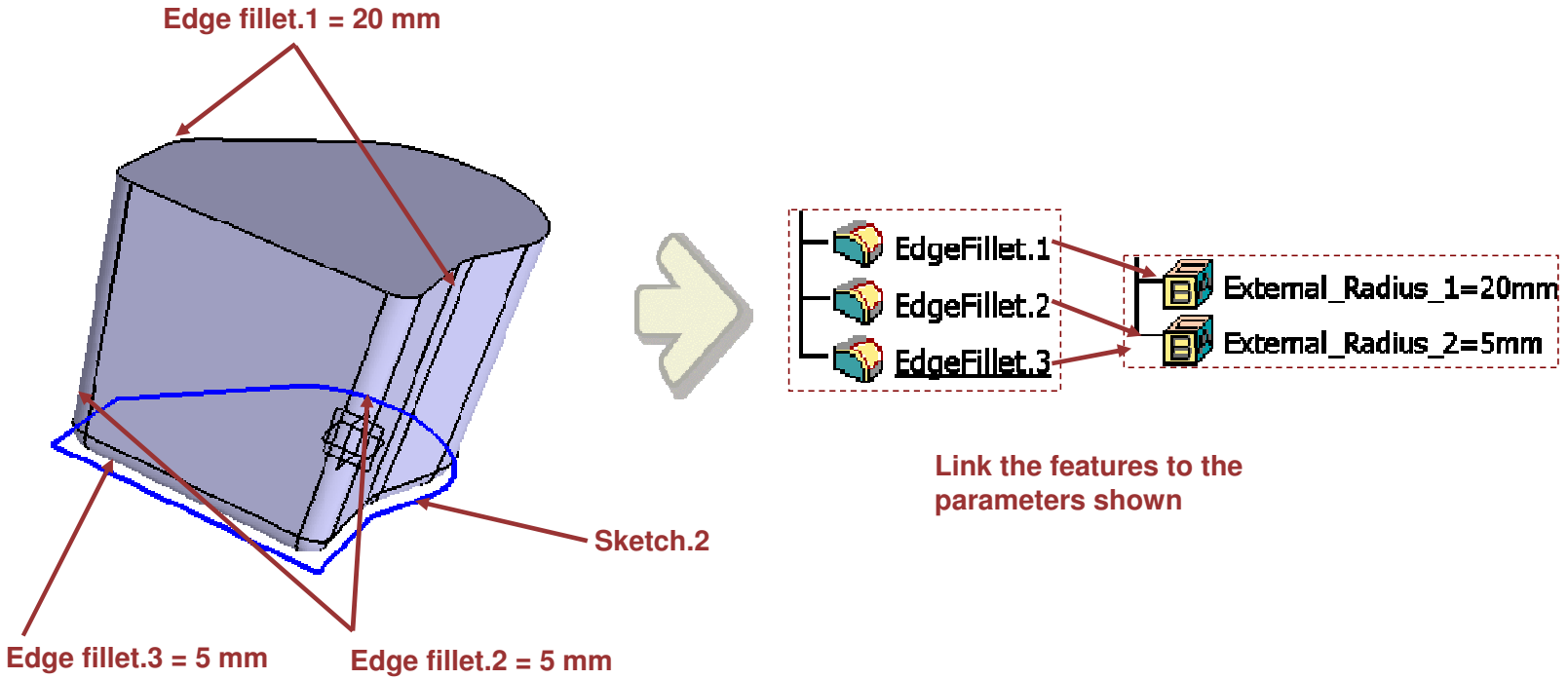




Student Notes:

Design the Main Shape (3/3)

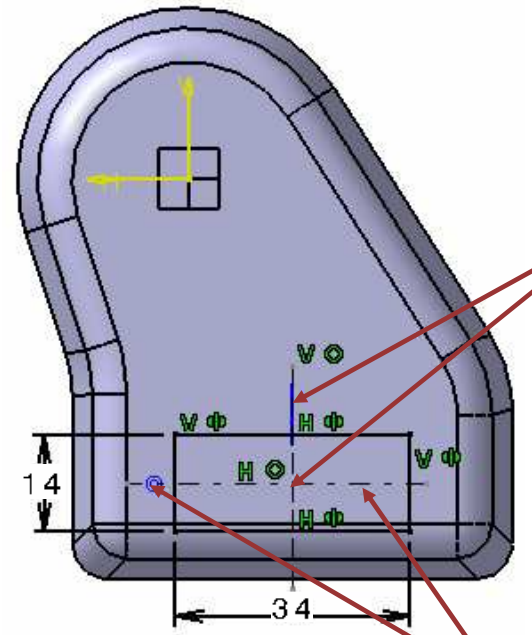
- Apply three different Edge Fillets. Two fillets having 5 mm radius and other having 20 mm radius.
- Create two parameters to drive the fillet values
  - External_Radius_1 = 20mm
  - External_Radius_2 = 5mm



Student Notes:

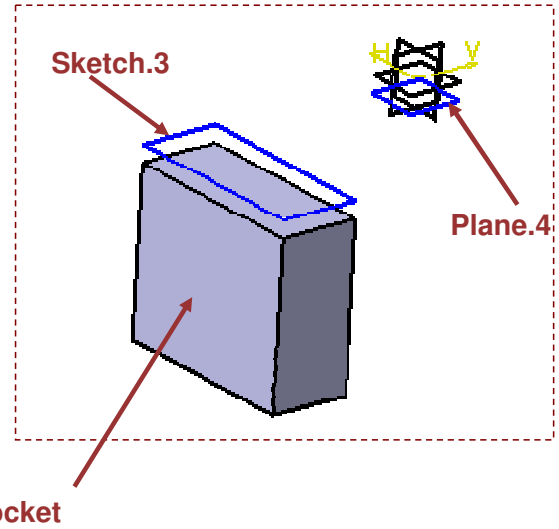
Design the Core Body A (1/3)

- Create a Positioned Sketch.3 in Geometrical set on XY plane as shown. Use Part Origin, orientation as X axis and reverse H and V directions
- Create a plane offset from XY plane by 3 mm. This is Plane.4
- Insert a body 'Core Body A'. In this body, create a pocket from this sketch:
  - Limit 1 = 32 mm
  - Limit 2 = Upto Plane.4



Constrain this line with 'Plane.1'

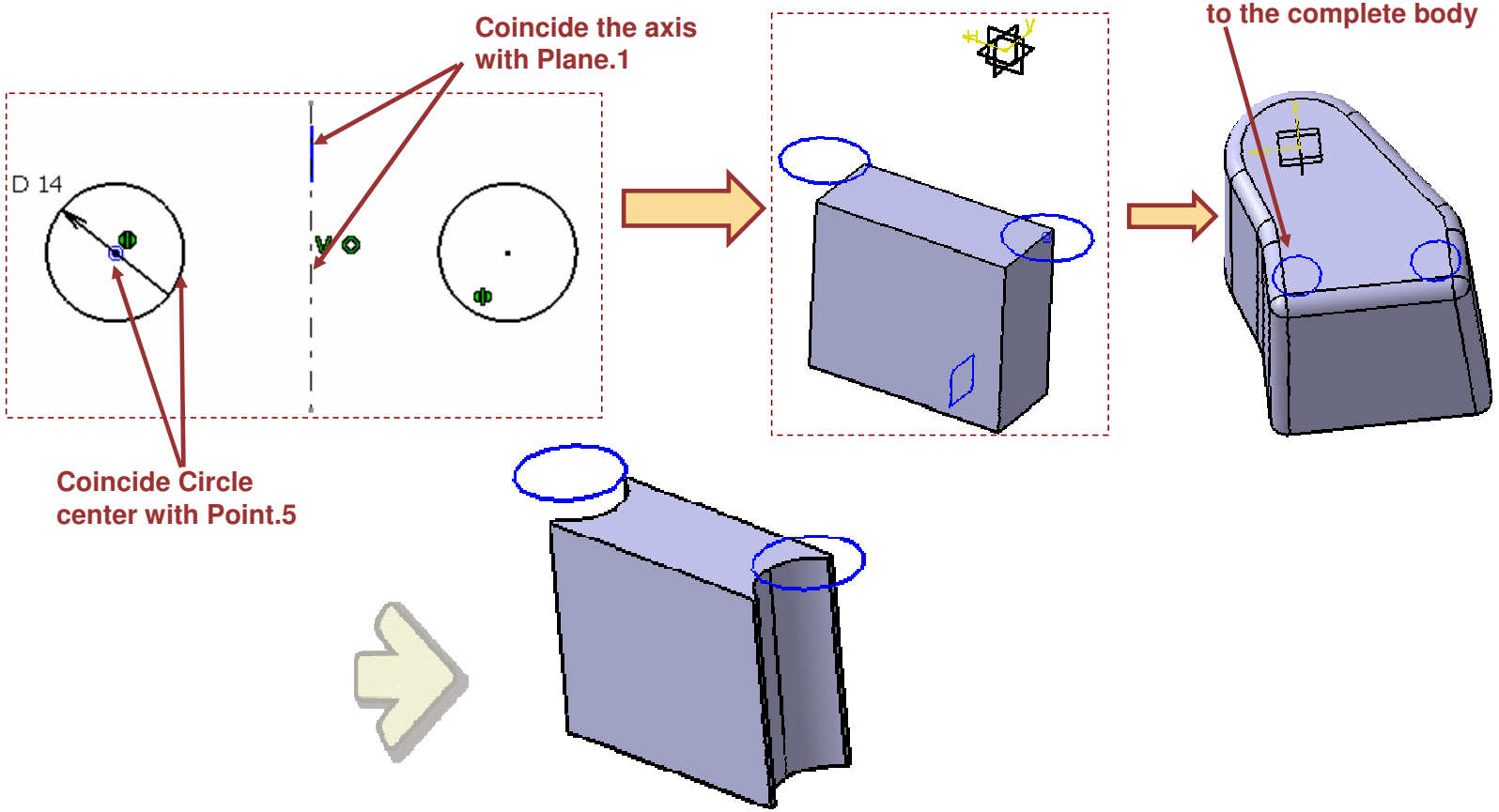
Constrain this line with 'Symmetry.2'



Student Notes:

Design the Core Body A (2/3)

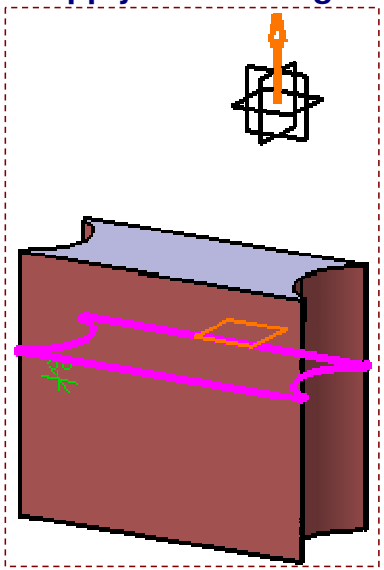
- Create a Positioned Sketch.4 in Geometrical set on XY plane as shown.
  Use Part Origin, orientation as X axis and reverse H and V directions
- Use Point.5 and Plane.1 to constrain this sketch.
- Create a pad of 35 mm from it in the 'Core Body A'.

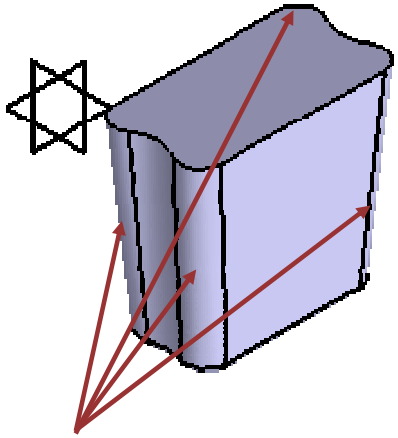


Student Notes:

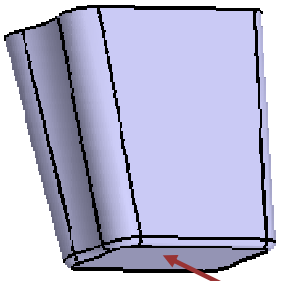
Design the Core Body A (3/3)

- Apply draft of 3 deg to the four faces as shown. The value is driven by 'Draft_Angle_Value' parameter. Select Neutral plane as plane.3.
- Apply a edge fillet of 3 mm to the four edges. This is driven by 'Internal_Radius_1' Parameter.
- Apply another edge fillet on one Face as shown. This is driven by 'Internal_radius_2' Parameter.



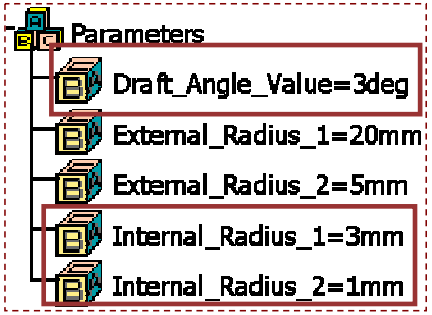


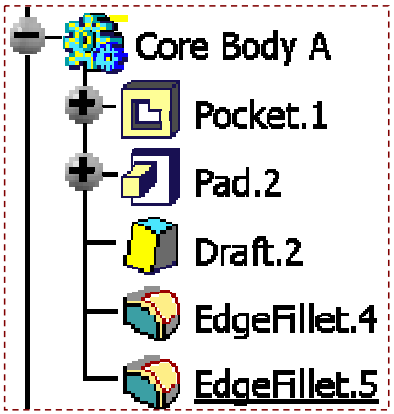
Edge fillet of 3 mm



Edge fillet of 1 mm to this face

The Parameters are as shown:

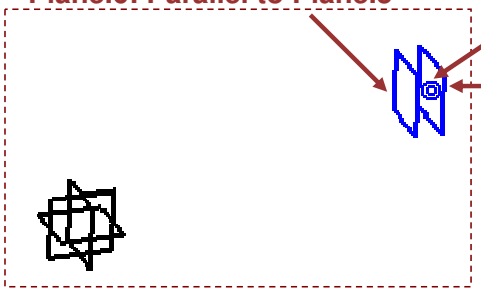


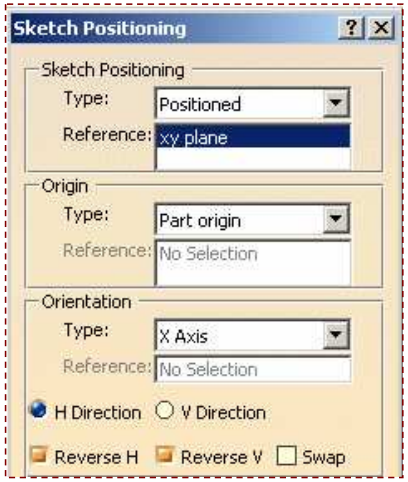


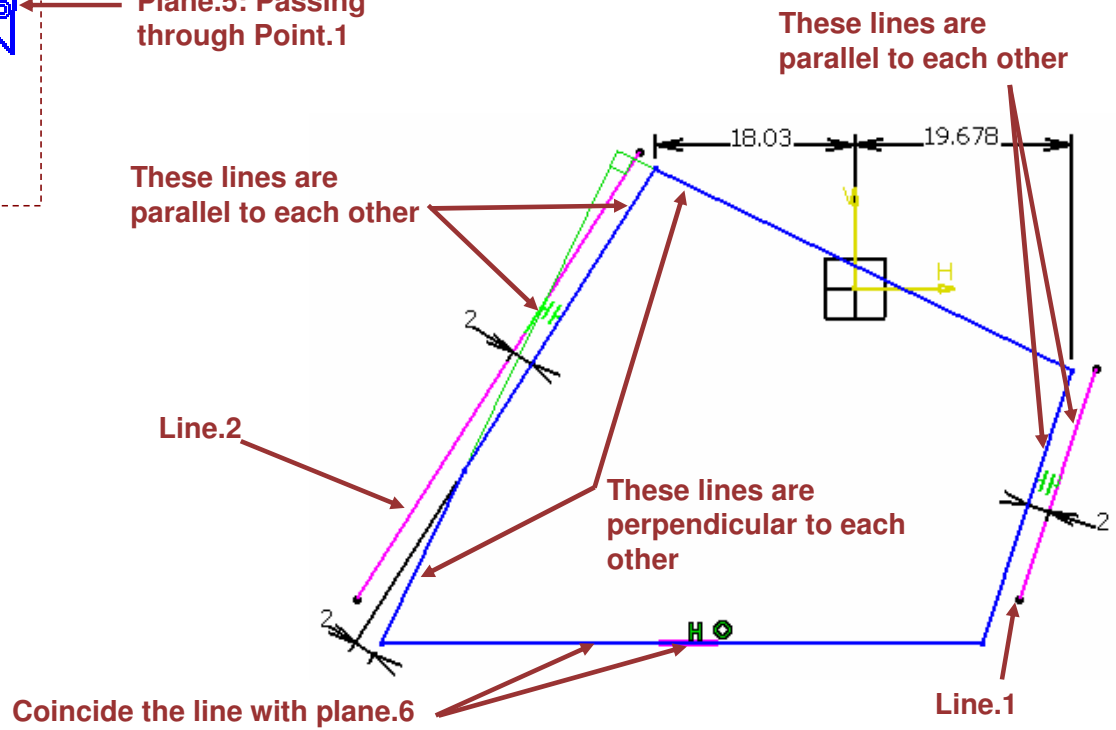
Student Notes:

Design the Core Body B (1/4)

- Create a plane parallel to ZX plane passing through Point.1.This is Plane.5
- Create another plane parallel to plane.5 at a distance of 2 mm. This plane is in between Plane.5 and standard Planes. This is Plane.6
- Create a Positioned Sketch on XY plane in Geometrical set.
- Use Plane.5, Line.1, Line.2 to constrain this sketch. This is Sketch.5.



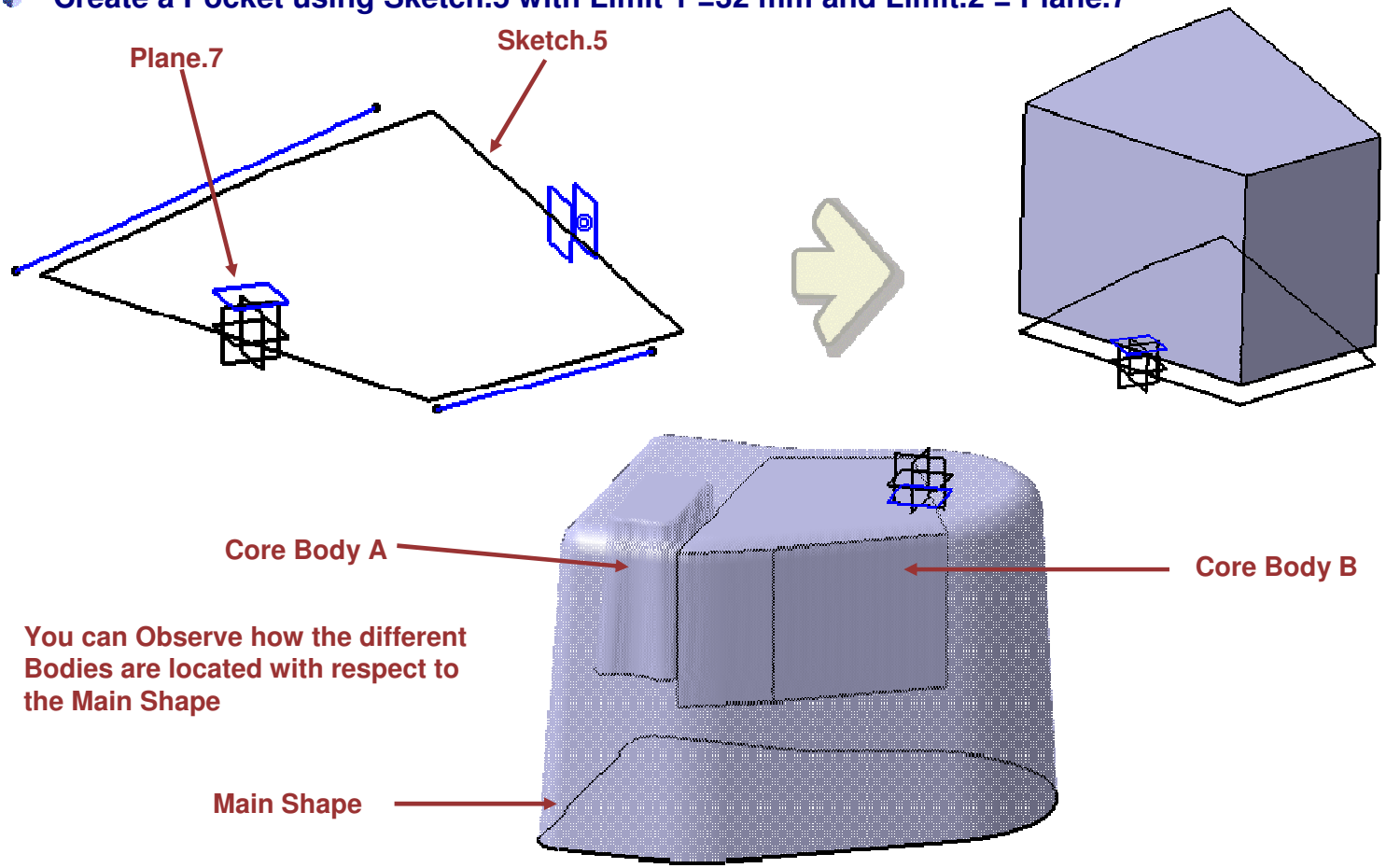




Student Notes:

Design the Core Body B (2/4)

- In Geometrical set create a Plane offset from XY plane at a distance of 3 mm upwards. This is Plane.7.
- Insert body 'Core Body B'.
- Create a Pocket using Sketch.5 with Limit 1 =32 mm and Limit.2 = Plane.7

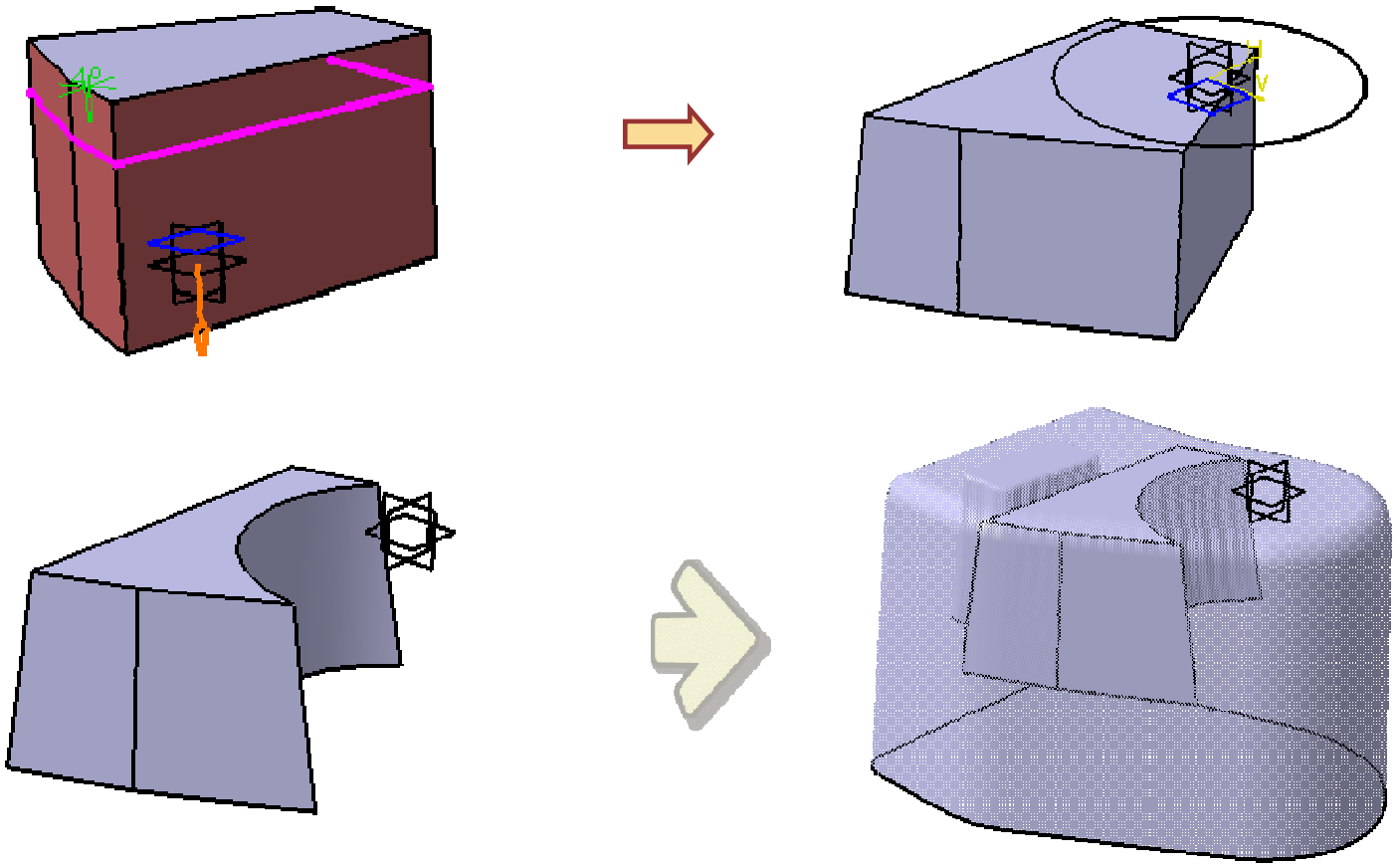


You can Observe how the different Bodies are located with respect to the Main Shape

Student Notes:

Design the Core Body B (3/4)

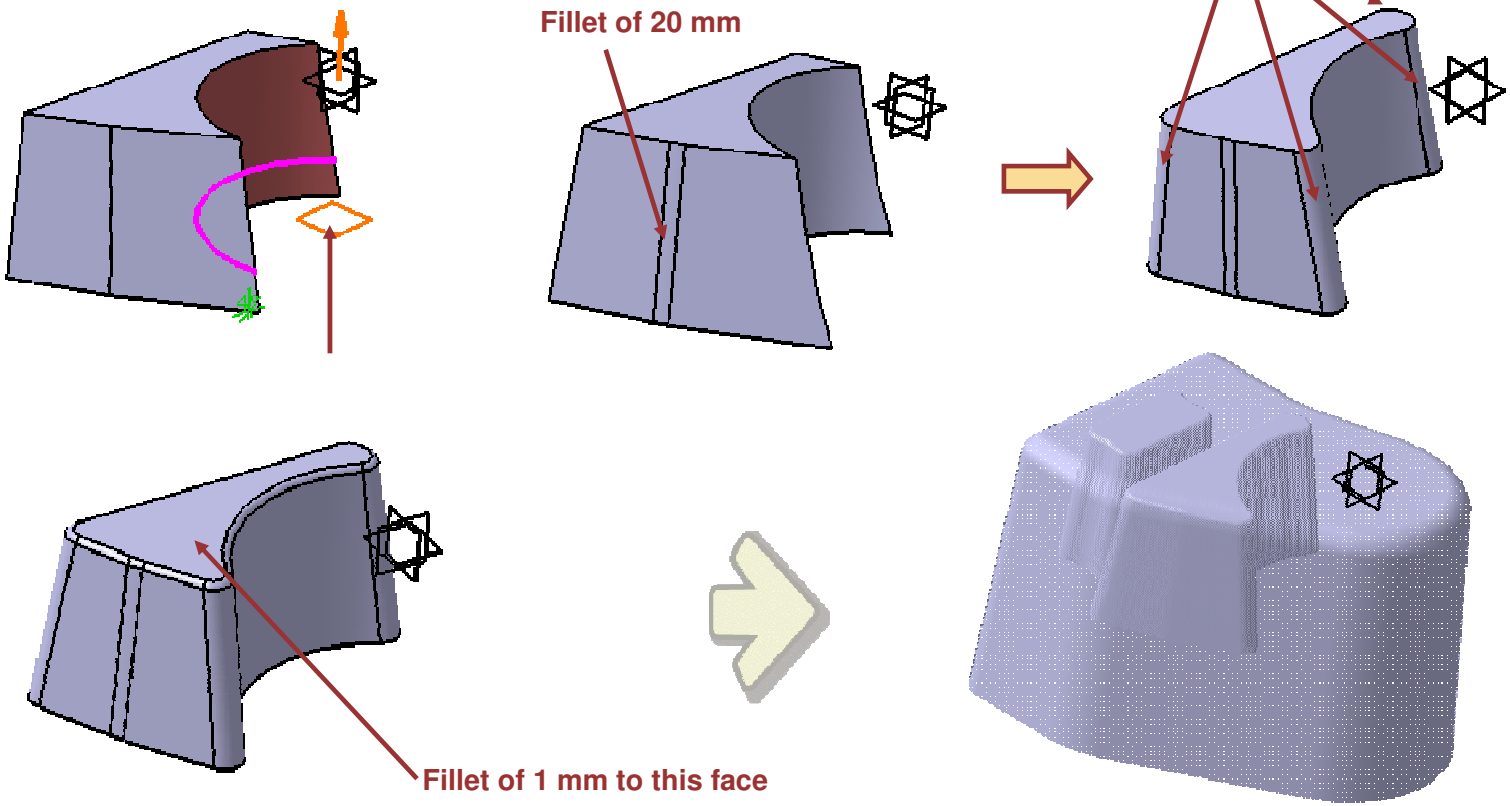
- Apply a draft of 4 deg to all five vertical faces of this pocket. Use Plane.2 as neutral element.
- Show Sketch.1 which you had created. Pad it by 60 mm.



Student Notes:

Design the Core Body B (4/4)

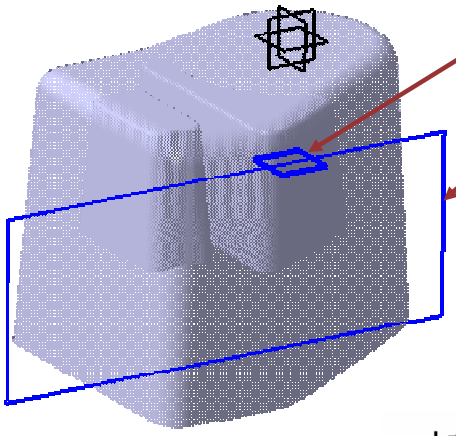
- Apply a draft of 4 deg to the face shown. Select Plane.2 as neutral element.
- Apply three Edge Fillets:
  - Value = 20 mm, Driven by 'External_Radius_1' parameter on one edge
  - Value = 3 mm, Driven by 'Internal_Radius_1' parameter
  - Value = 1 mm, Driven by 'Internal_Radius_2' parameter

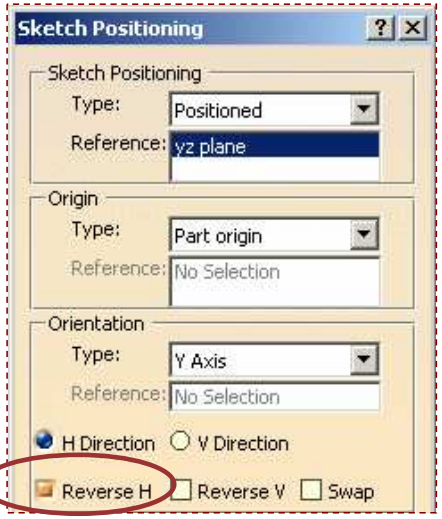


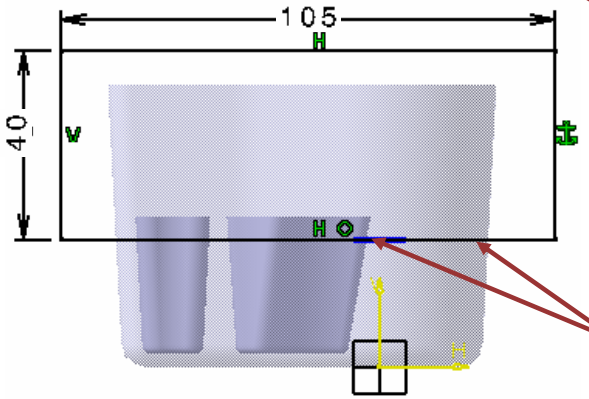
Student Notes:

Design the Core Body C (1/2)

- In the Geometrical set create a plane parallel to XY plane at a distance of 27 mm. This is Plane.8.
- In the Geometrical set create a positioned sketch on YZ plane. This is Sketch.6
- Use Plane.8 to constrain this sketch.





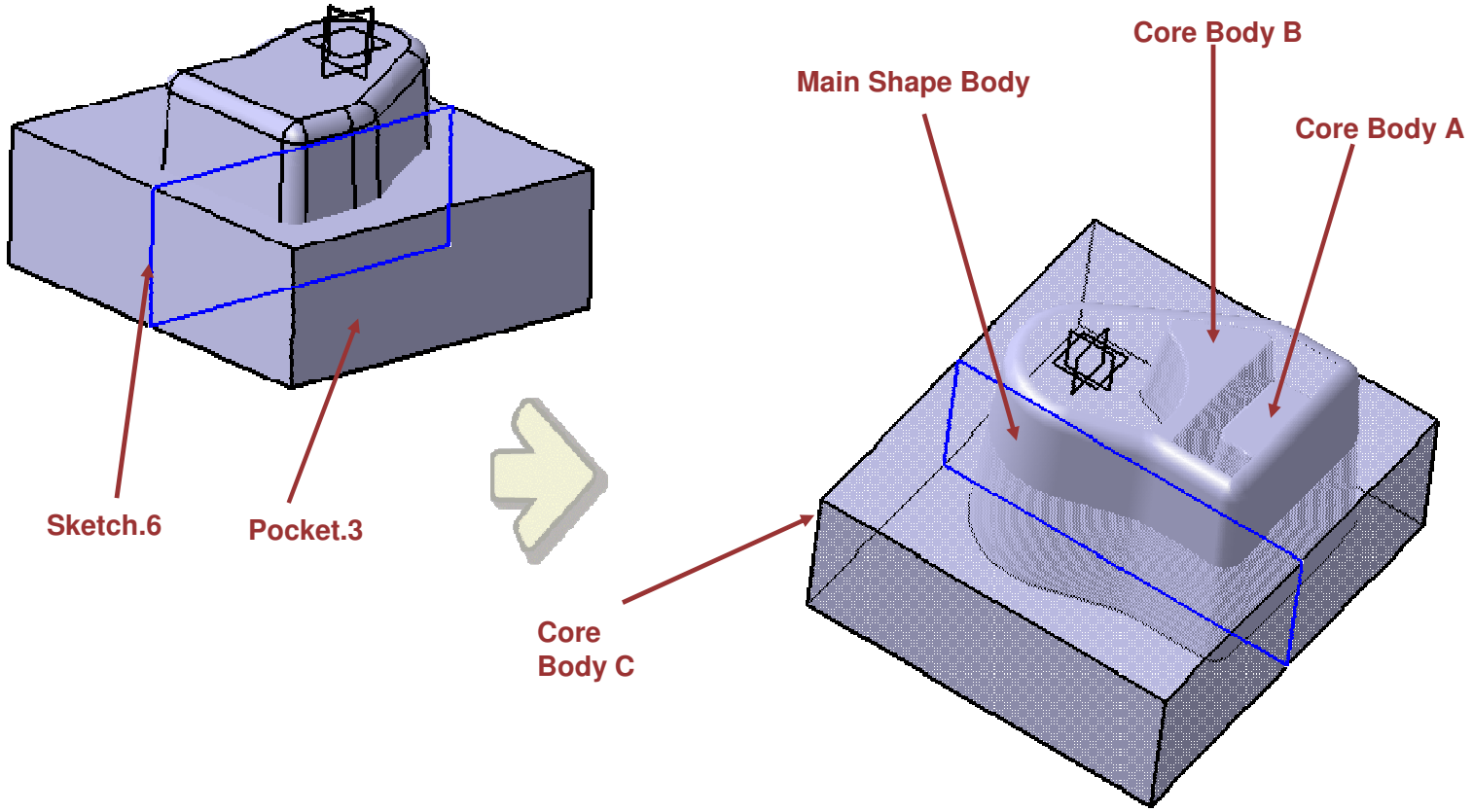


Coincide this line with Plane.8

Student Notes:

Design the Core Body C (2/2)

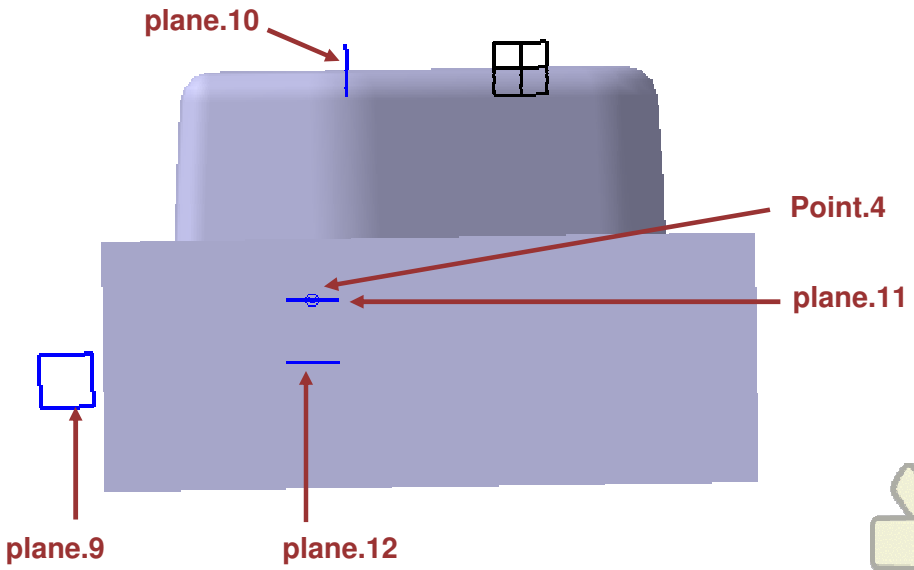
- Insert Body 'Core Body C'
- In this body, create a Pocket of 60 mm (Mirrored Extent). Using Sketch.6

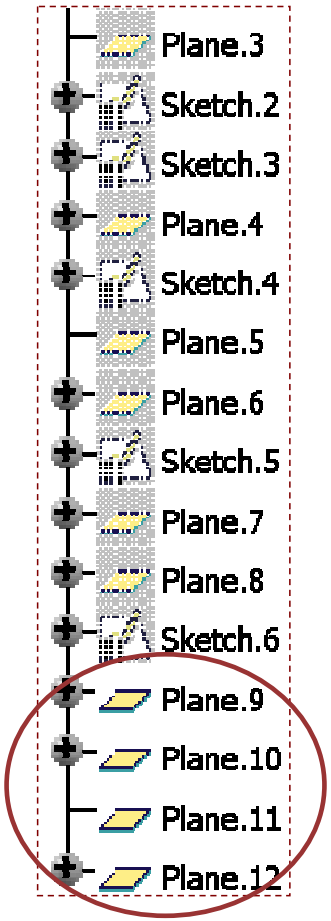


Student Notes:

Design the Core Body D (1/4)

- In the Geometrical set, create four planes as follows:
  - Plane.9: Offset from Plane.1 at a distance of 7 mm.
  - Plane.10: Offset from Plane.5 at a distance of 6 mm.
  - Plane.11: Parallel to XY plane passing through Point.4.
  - Plane.12: Offset from Plane.11 at a distance of 10 mm.

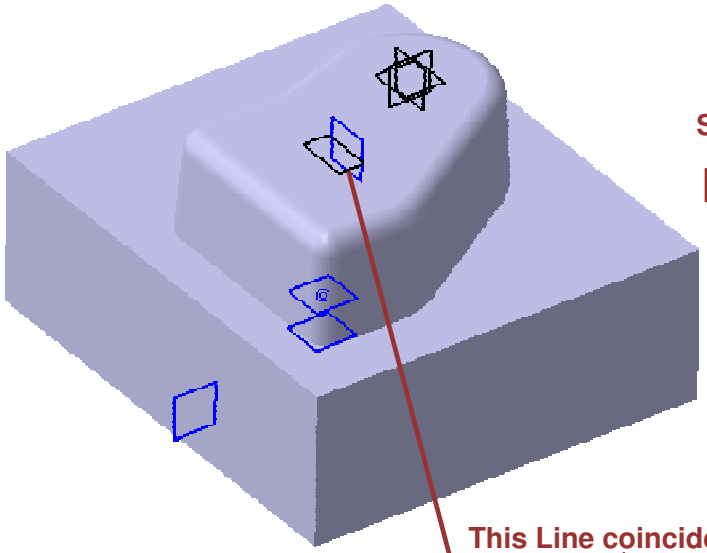




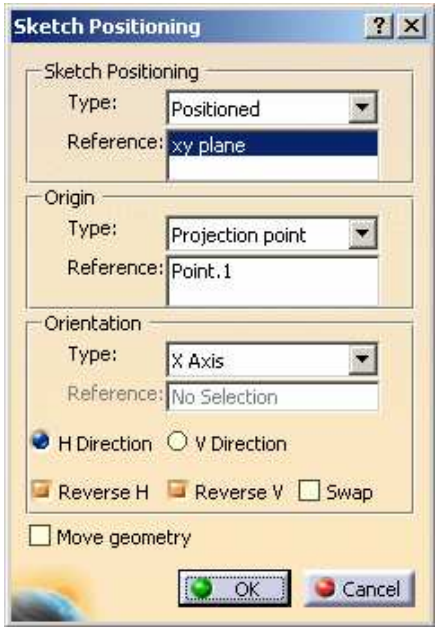
Student Notes:

Design the Core Body D (2/4)

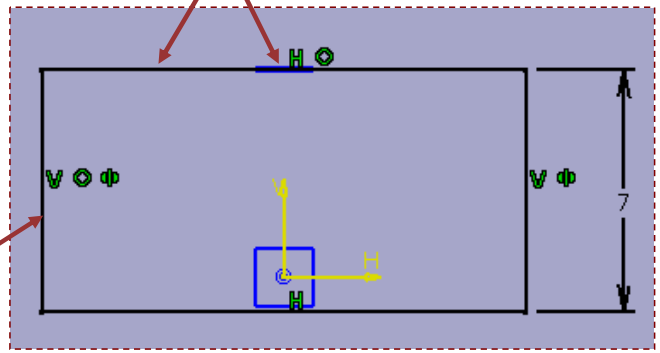
- Create a Positioned sketch on XY plane in Geometrical set. This is Sketch.7
- Insert body 'Core Body D'.



Sketch details



This Line coincident with Plane.10



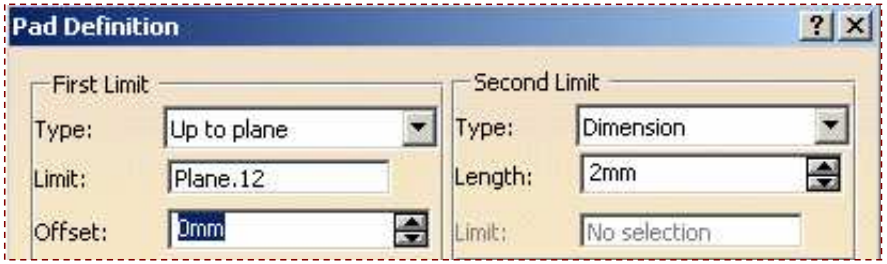
This line coincides with Plane.9

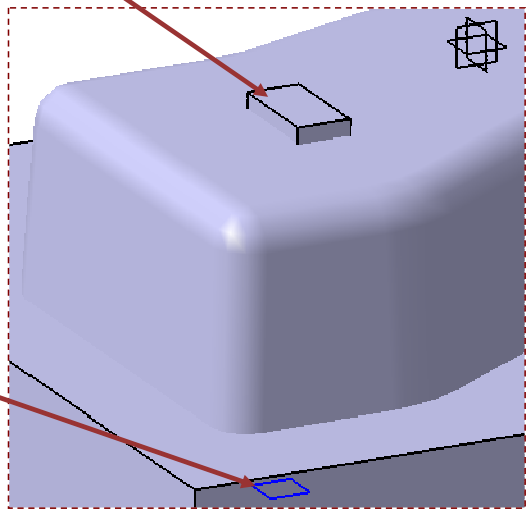
Note: The vertical lines in the sketch are symmetric about the vertical axis

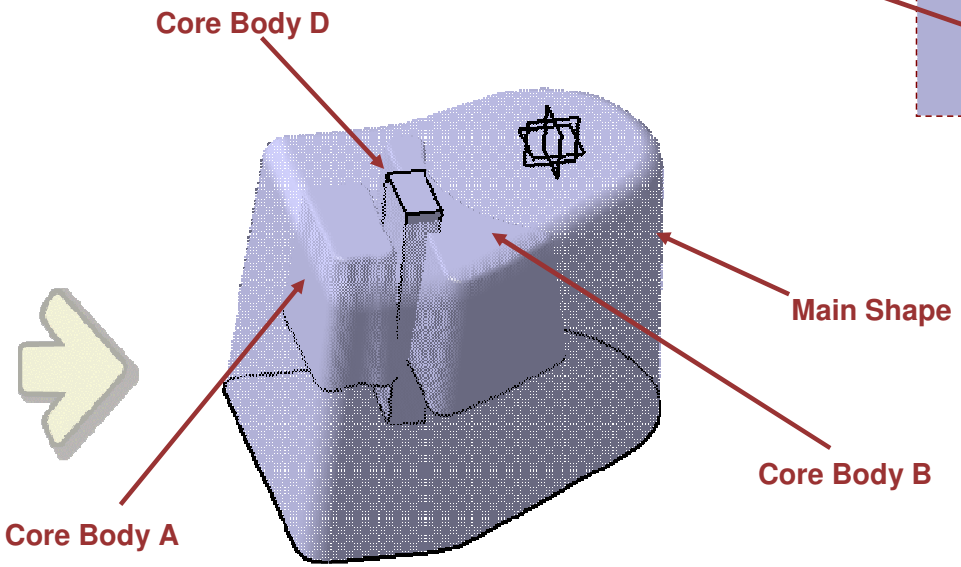
Student Notes:

Design the Core Body D (3/4)

- Pad this sketch as shown



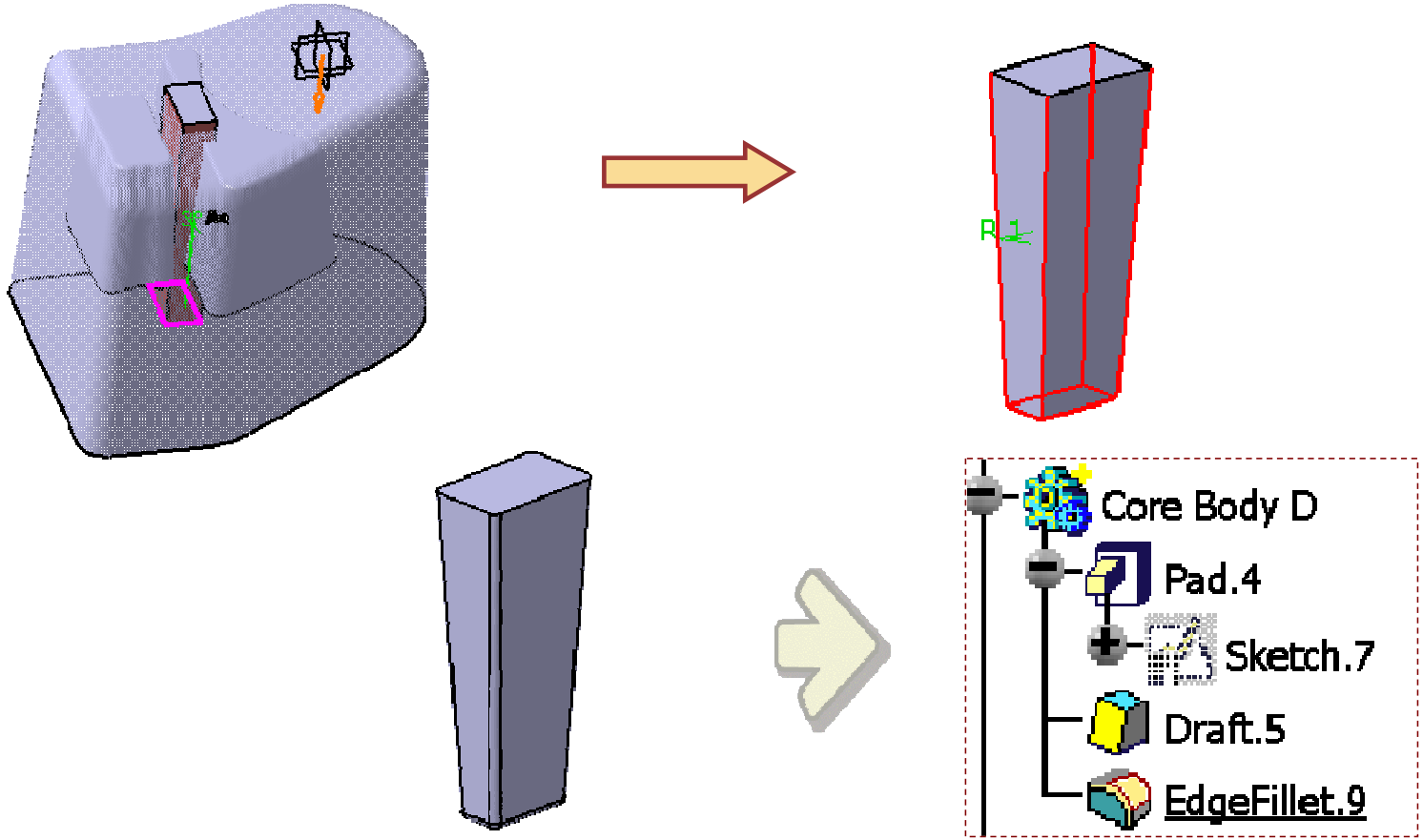




Student Notes:

Design the Core Body D (4/4)

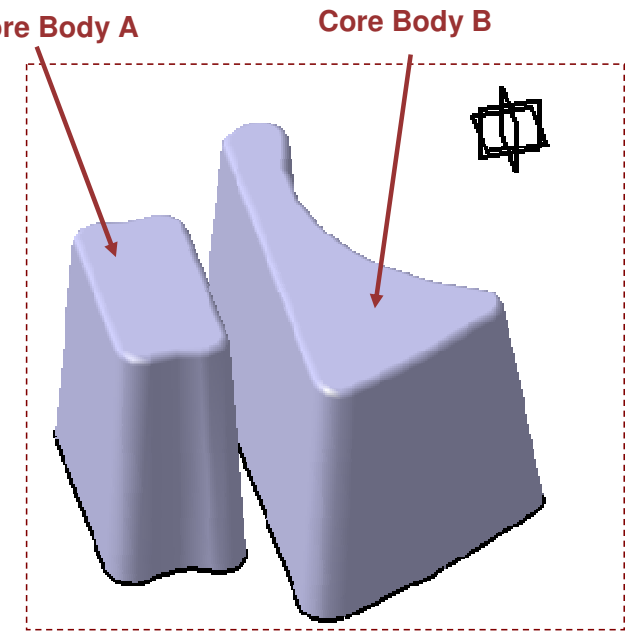
- Apply a draft of 3 deg to the four faces shown. This is driven by 'Draft_Angle_Value' parameter.
- Apply a fillet of 1 mm. This value is driven by 'Internal_Radius_2' parameter.

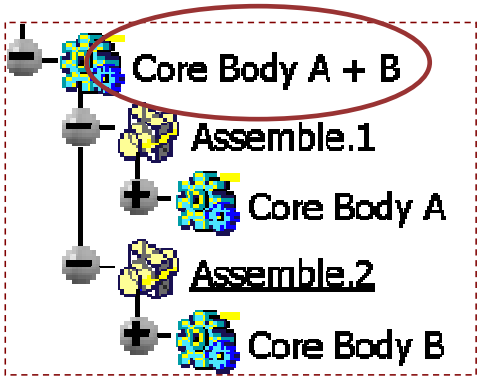


Student Notes:

Assemble Core Bodies A & B

- Insert body 'Core Body A + B'.
- Assemble 'Core Body A' and 'Core Body B' into it.

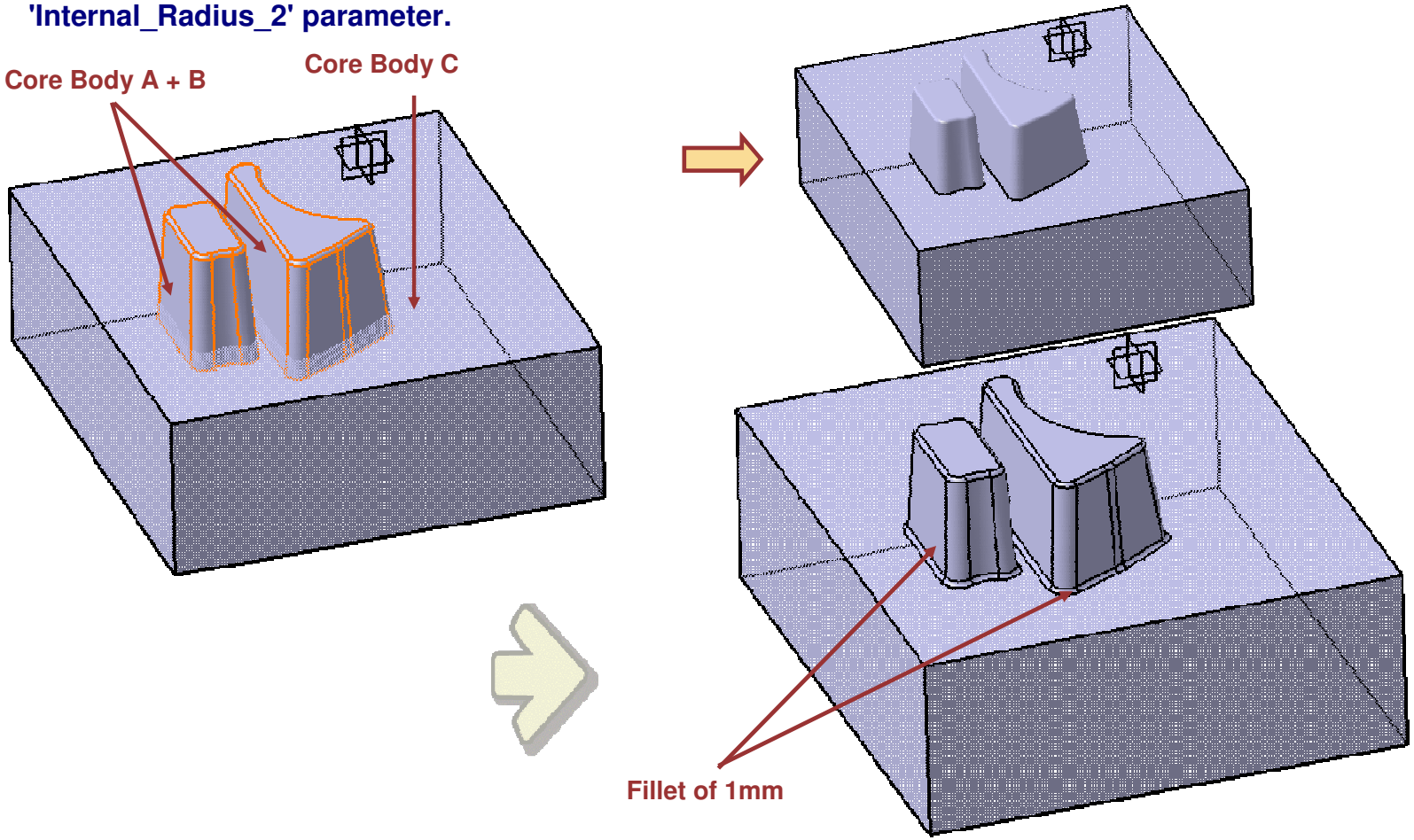




Student Notes:

Assemble Core Body C and Core Body (A + B)

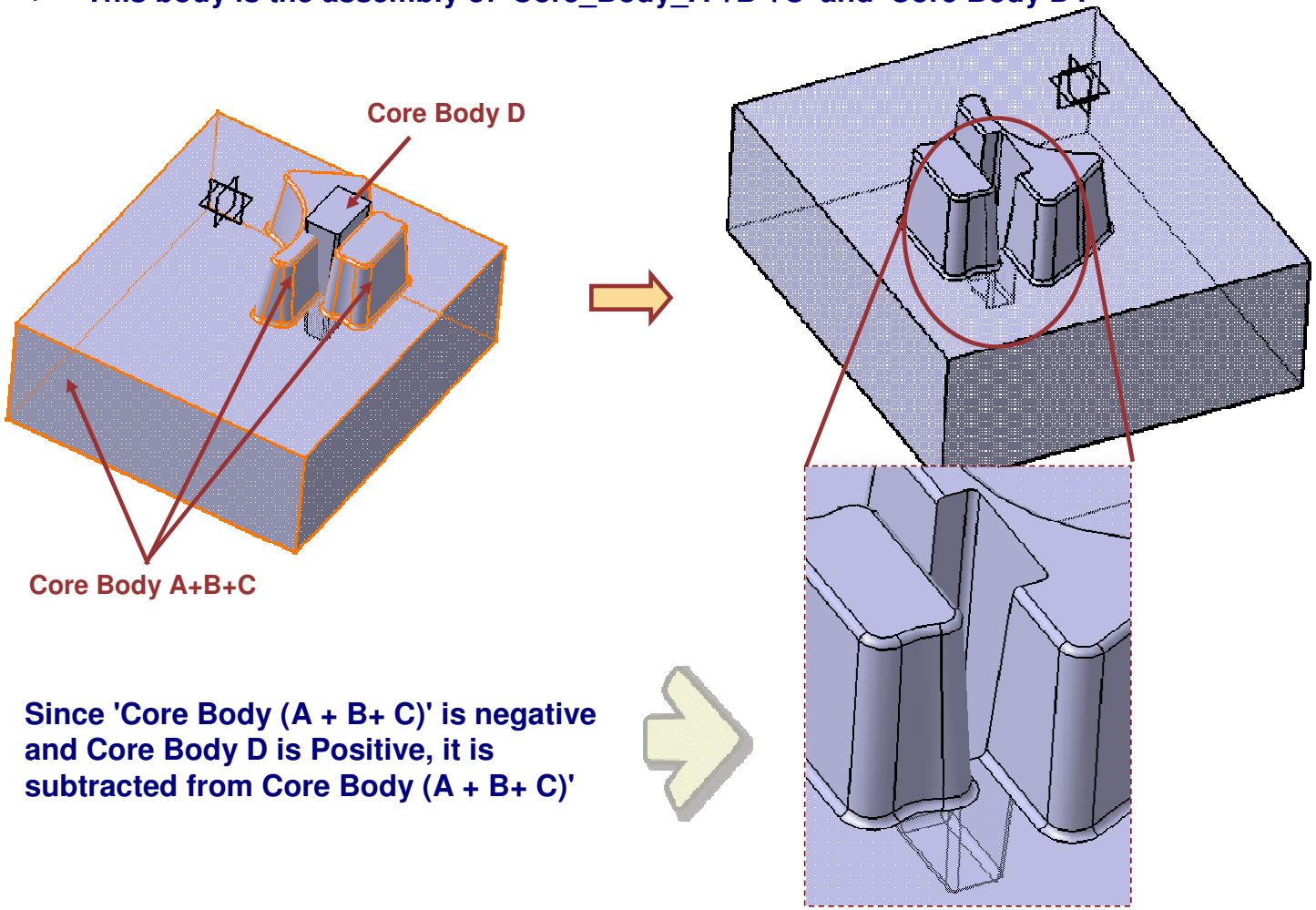
- Insert body 'Core Body A + B + C'.
- Assemble 'Core Body A + B' and 'Core Body C' into it.
- Apply a Edge Fillet of 1 mm to the newly formed edge. This is driven by 'Internal_Radius_2' parameter.



Student Notes:

Design the Final Core Body (1/2)

- Insert body 'Final Core Body'.
- This body is the assembly of 'Core_Body_A +B +C' and 'Core Body D'.

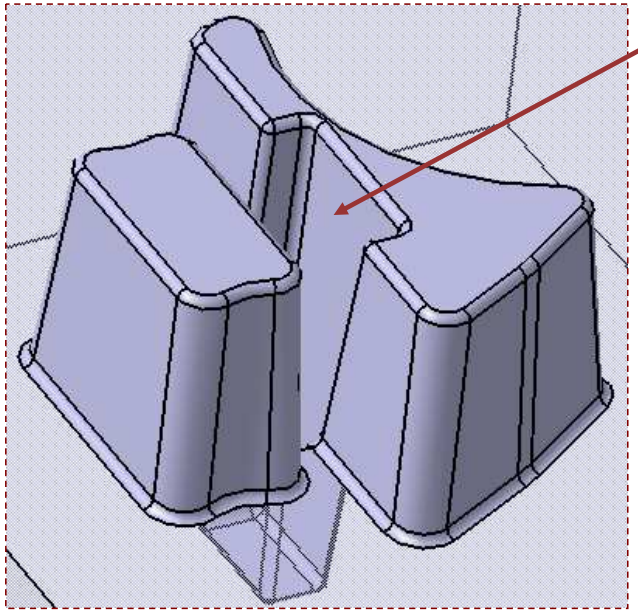


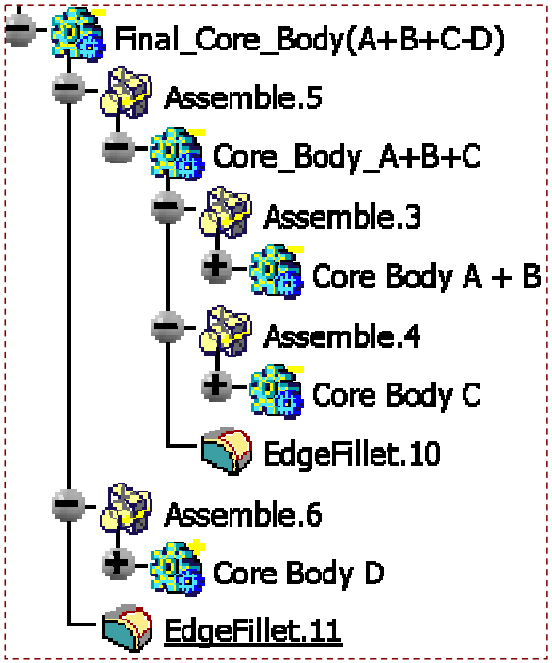
Since 'Core Body (A + B+ C)' is negative and Core Body D is Positive, it is subtracted from Core Body (A + B+ C)'

Student Notes:

Design the Final Core Body (2/2)

- Apply Edge Fillet to the face shown. This is driven by 'Internal_Radius_2' parameter.

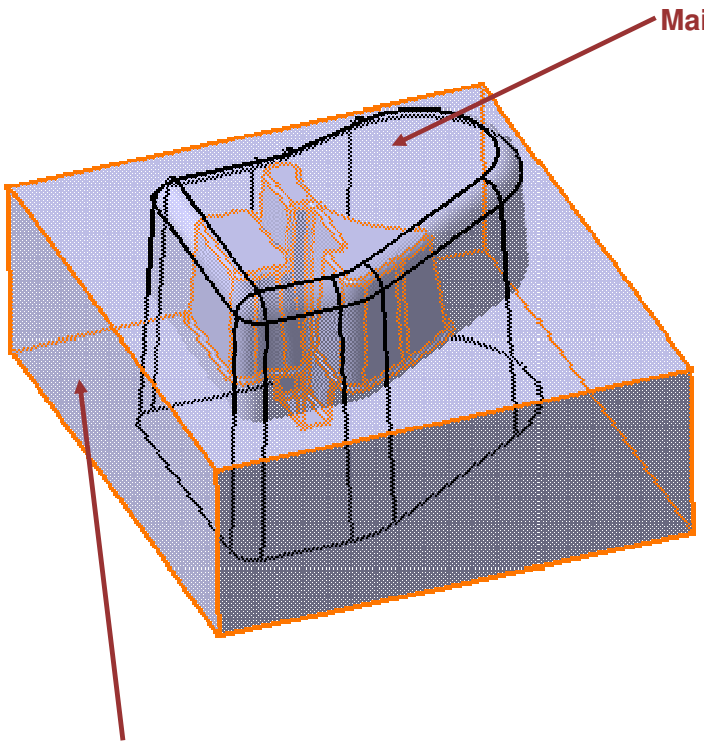


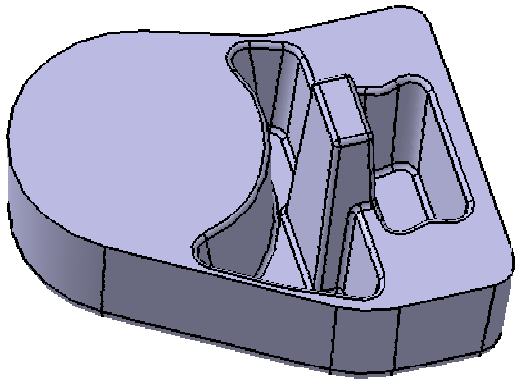


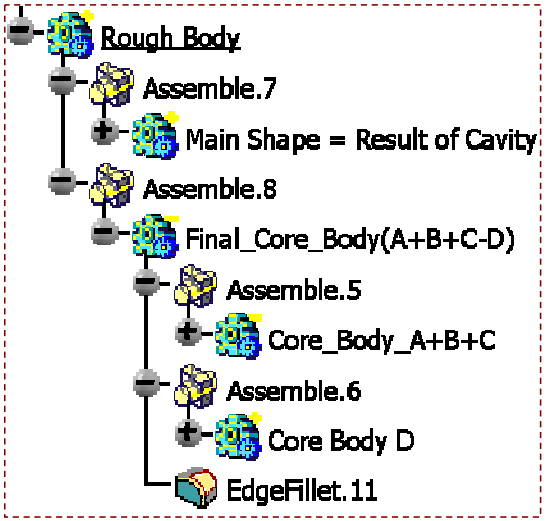
Student Notes:

Assemble into Rough Body

- Insert Body 'Rough Body'
- Assemble into it 'Main Shape' and 'Final Core Body'







Load: Side_Toolhead_Attach_Step_1_End.CATPart

Student Notes:

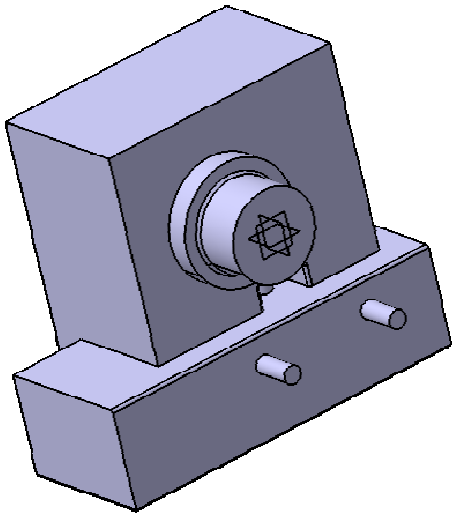
Side Toolhead

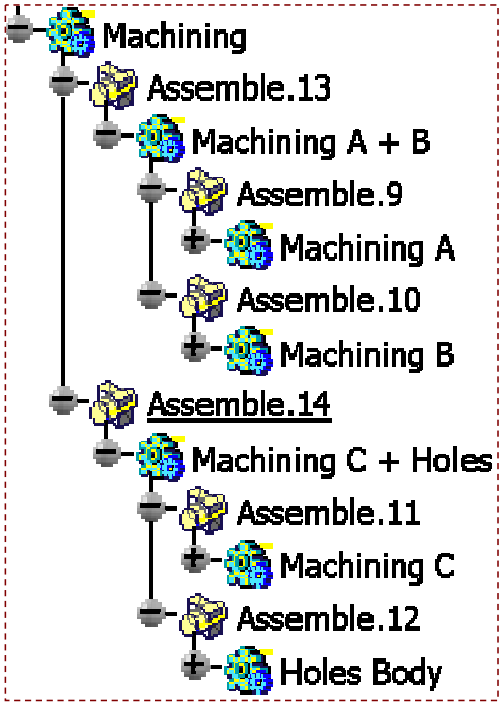
Step 2: Machine the Rough Body

20 min

In this step you will design the 'Machining elements' in order to produce final machined design. Here you will:

- **Design the Machining A element**
- **Design the Machining B element**
- **Design the Machining C element**
- **Assemble all the Machined Bodies**

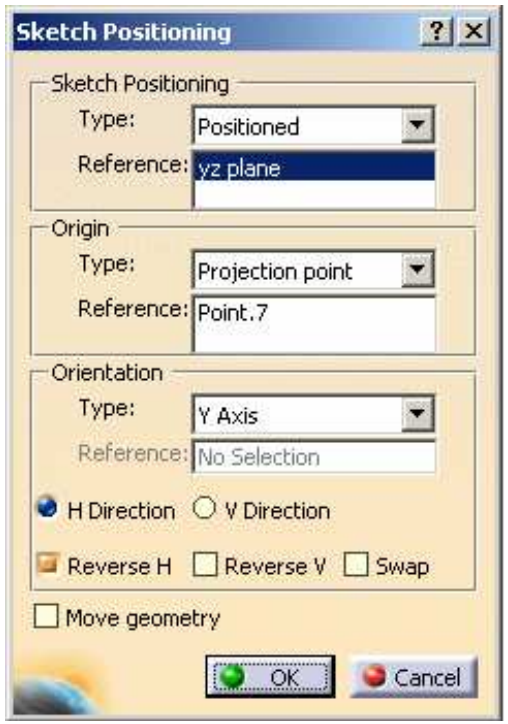


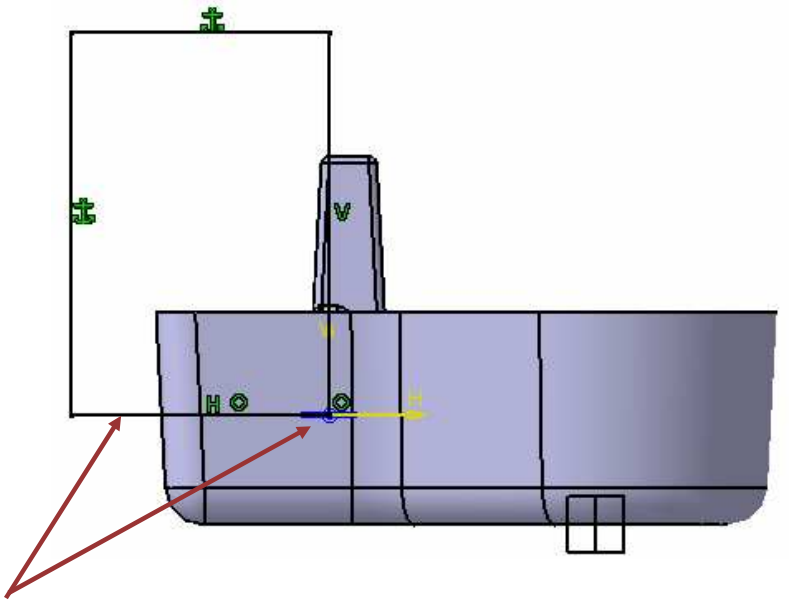


Student Notes:

Machining A (1/2)

- Create a Positioned sketch on YZ plane as shown. This is Sketch.8
- Use Plane.3 to constrain this sketch



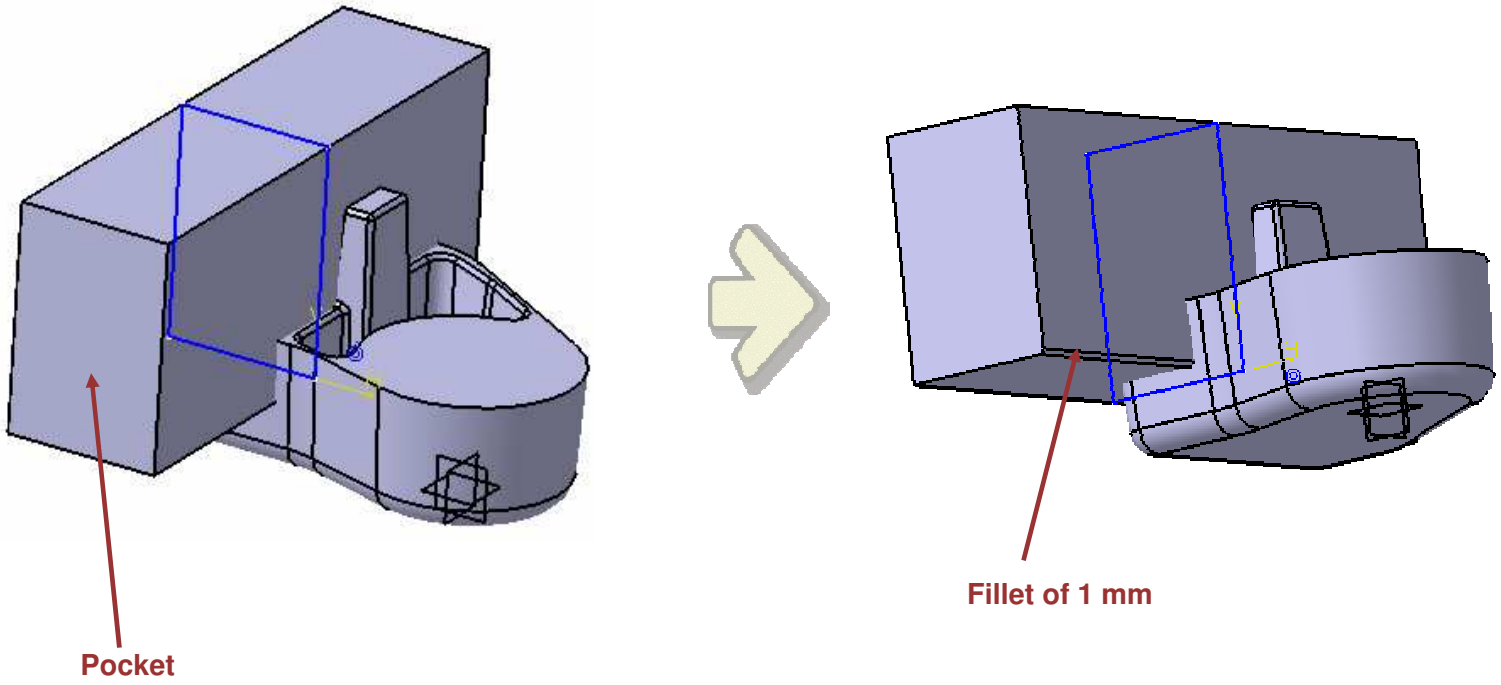


This Line is coincident with Plane.3

Student Notes:

Machining A (2/2)

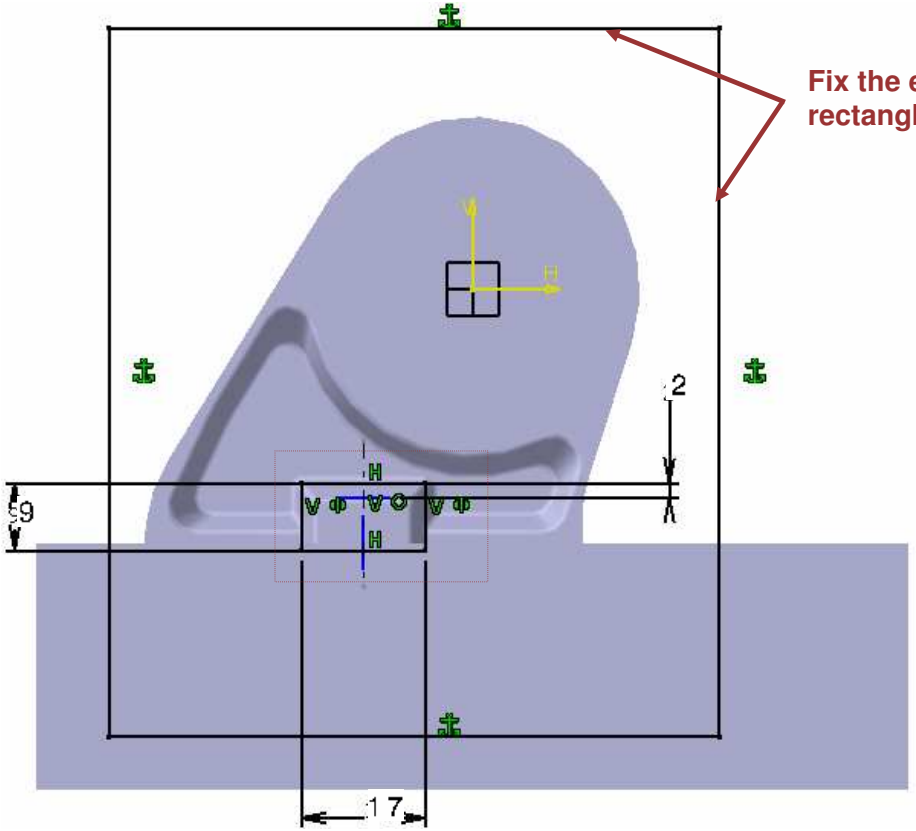
- Insert body 'Machine A'
- Create a pocket of 60 mm (Mirrored Extent) using this sketch.
- Apply a edge fillet of 1 mm to this pocket. The radius value is driven by 'Internal_radius_2' parameter.

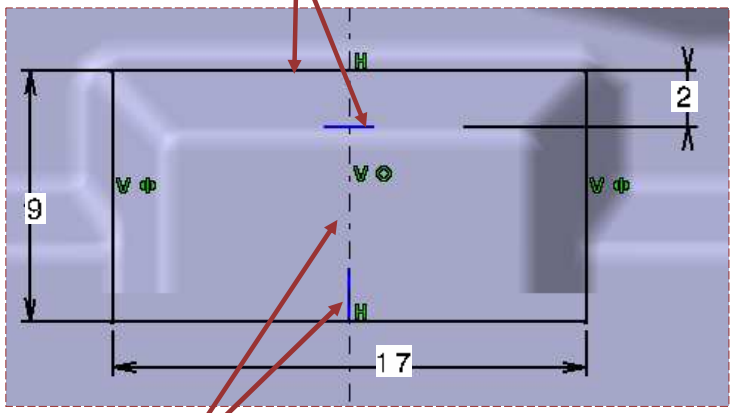


Student Notes:

Machining B (1/2)

- Create a positioned sketch on XY plane in Geometrical set as shown.
- Constrain the sketch using Plane.1 and Plane.10.

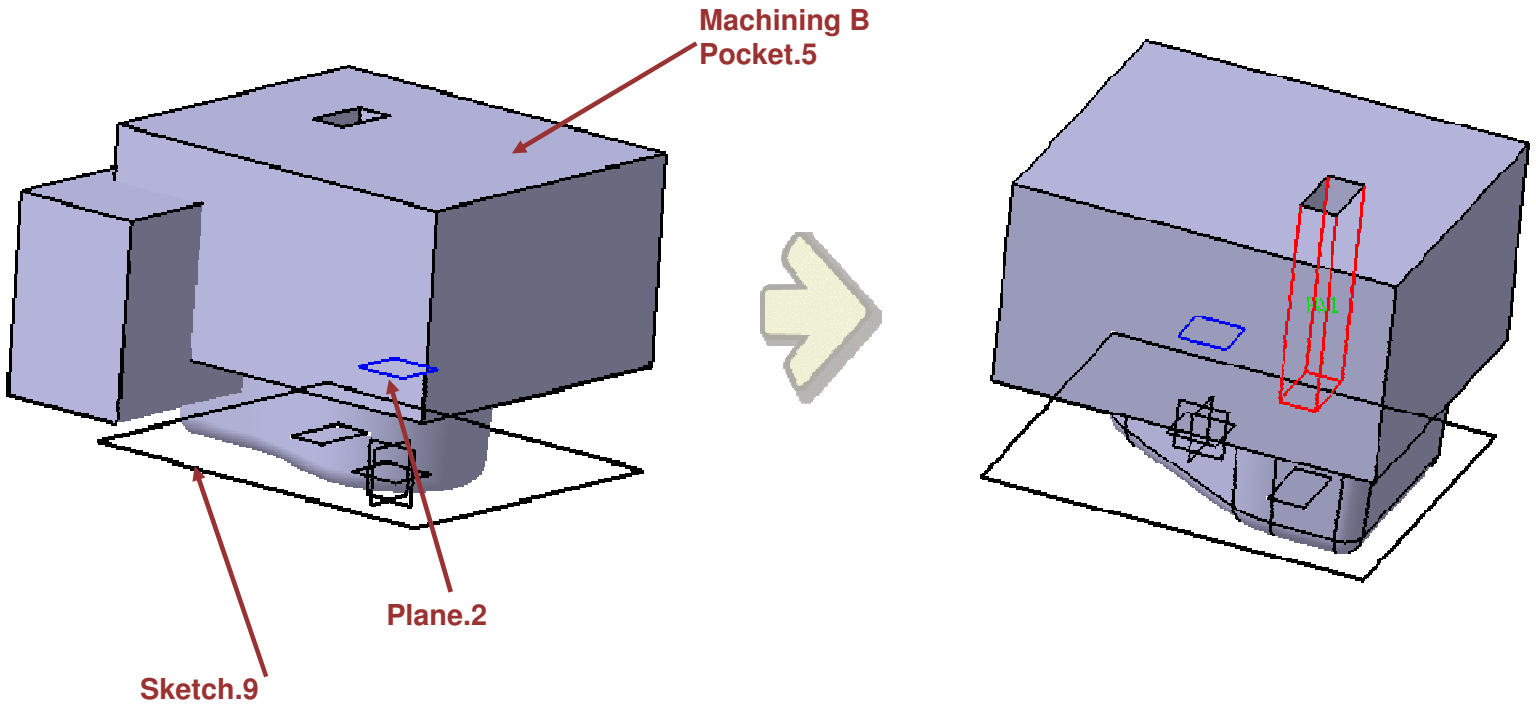




Student Notes:

Machining B (2/2)

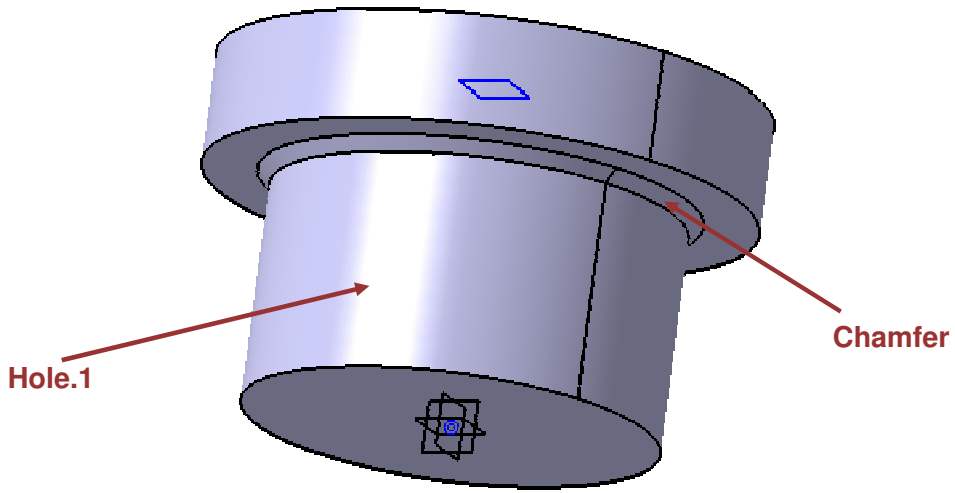
- Insert Body 'Machining B'
- Pocket this Sketch.9 with Limit 1 = 76 mm and Limit.2 = Plane.2
- Apply a fillet of 1 mm to the 8 edges shown



Student Notes:

Machining C

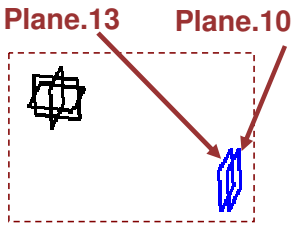
- Create a point at origin in geometrical set
- Insert body 'Machining C'
- Create a hole on Plane.2 and constrain it with this point at origin with following specifications:
  - Diameter of Hole = 30 mm
  - Create the Hole upto XY plane
  - Trimmed Bottom
  - Counter Bored Hole
    - Diameter: 40 mm
    - Depth: 8mm
- Apply a chamfer of 1mm x 45 deg

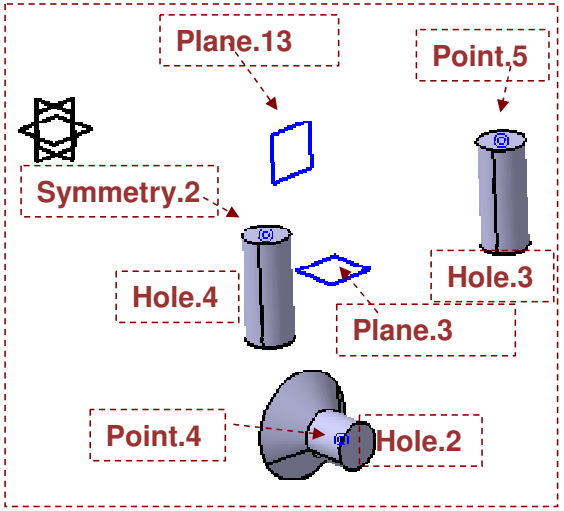


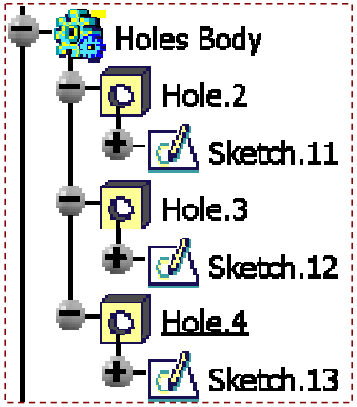
Student Notes:

Create the Holes

- Insert body 'Holes Body'
- Create plane.13 offset from plane.10 at a distance of 1mm.
- Create the first hole as per the specifications. This is Hole.2
  - Create this Hole on Plane.13 And use Point.4 to constrain it
  - Extension: Blind
  - Diameter: 6 mm, Depth: 10 mm, Flat Bottom
  - Type: Countersunk, Mode= Depth & Angle
    - Depth: 4 mm
    - Angle: 90 deg
- Create the second hole as per specifications. This is Hole.3
  - Create this Hole on Plane.3 And use Point.5 to constrain it
  - Extension: Upto Plane, Select XY Plane
  - Diameter: 6 mm, Trimmed Bottom
  - Type: Simple
- Create the Third hole as per specifications. This is Hole.4
  - Create this Hole on Plane.3 And use Symmetry.2 to constrain it
  - Extension: Upto Plane, Select XY Plane
  - Diameter: 6 mm, Trimmed Bottom
  - Type: Simple



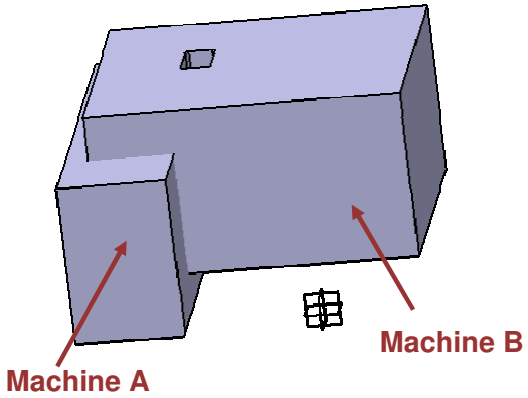


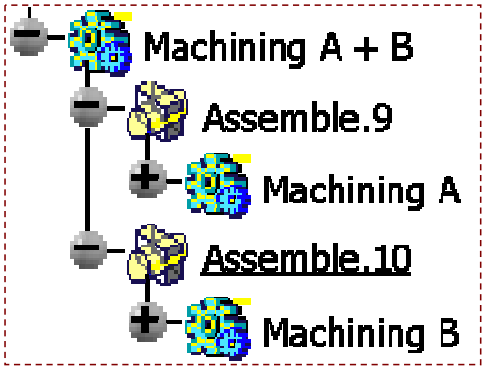


Student Notes:

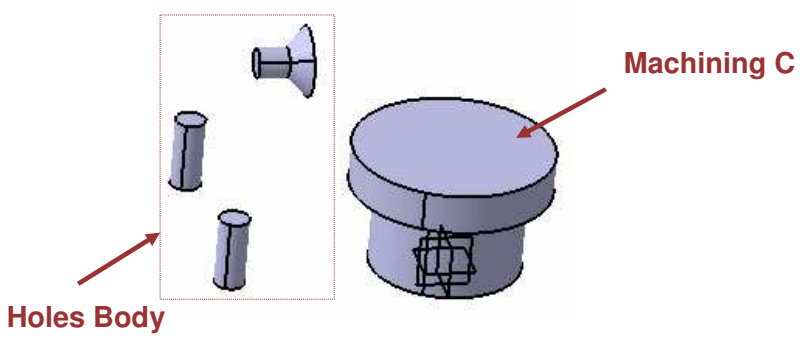
Assemble the Machined Bodies (1/2)

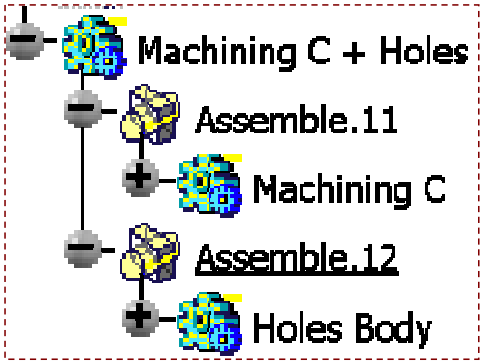
- Insert body 'Machining A + B'
- Assemble 'Machining A' and 'Machining B' Bodies into 'Machining A + B'





- Insert body 'Machining C + Holes'
- Assemble 'Machining C' and 'Holes Body' Bodies in 'Machining C + Holes' body

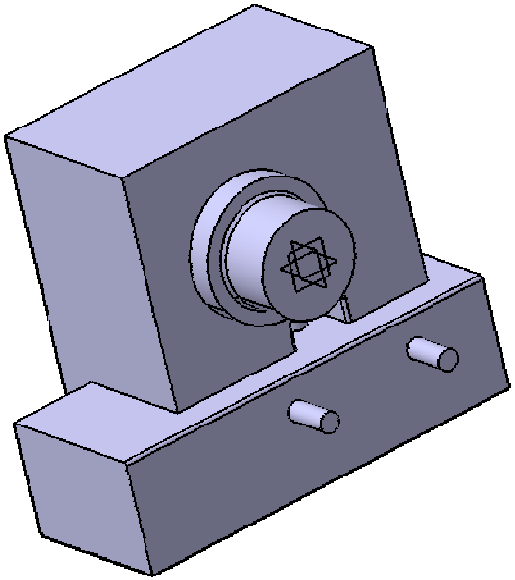


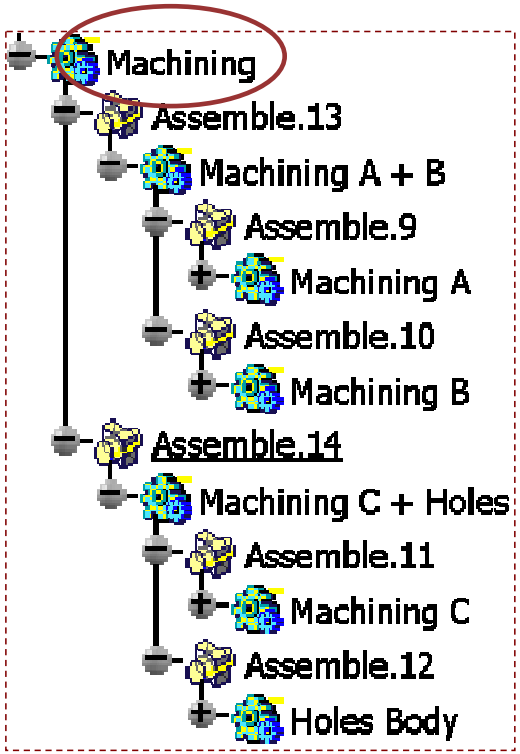


Student Notes:

Assemble the Machined Bodies (2/2)

- Insert body 'Machining'
- Assemble 'Machining A + B' and 'Machining C + Holes' Bodies in it.





The Entire Machining Body should look like this

Load: Side_Toolhead_Attach_Step_2_End.CATPart

Student Notes:

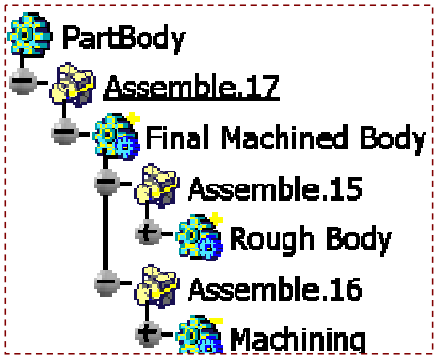
Side Toolhead

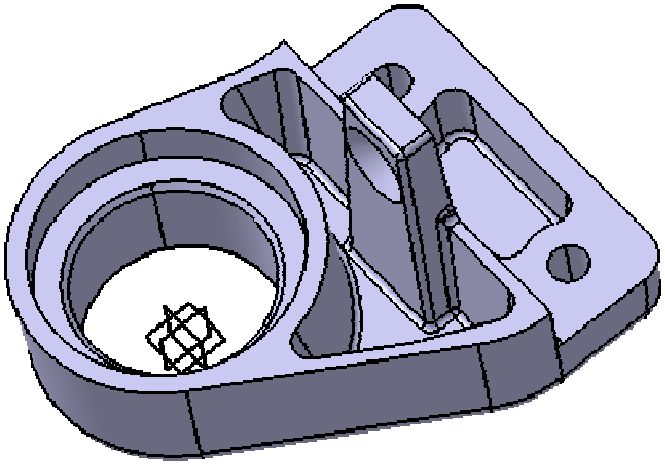
Step 3: Complete the Design

5 min

In this step you will Finalize the design of the Side Toolhead. You will

- **Assemble the Rough part and the Machined part with part body.**

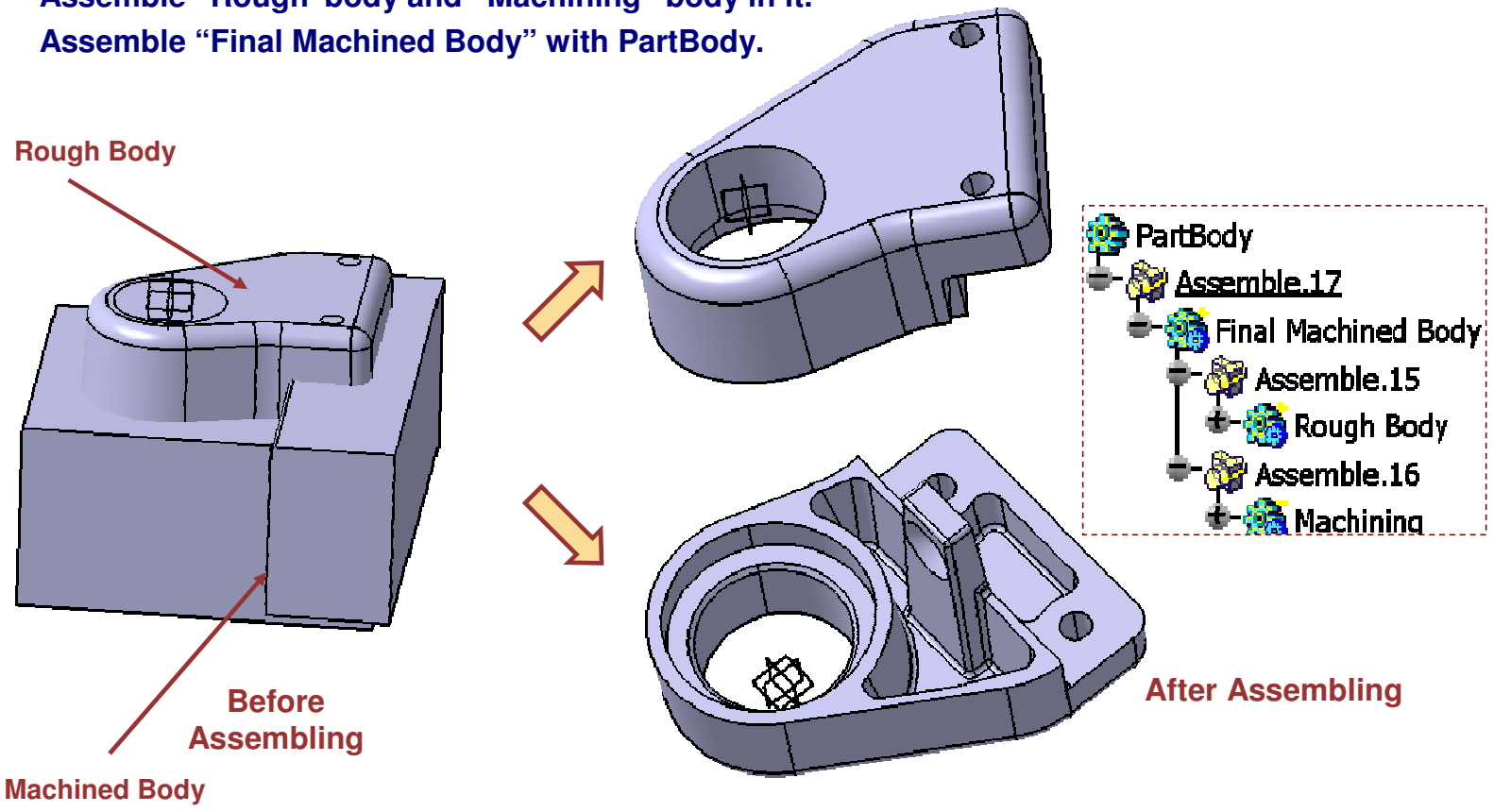




Student Notes:

Assemble the Rough and the Machined Bodies

- Insert Body “Final Machined” body
- Assemble “Rough”body and “Machining” body in it.
- Assemble “Final Machined Body” with PartBody.



Result Part: Side_Toolhead_Attach.CATPart

Student Notes:

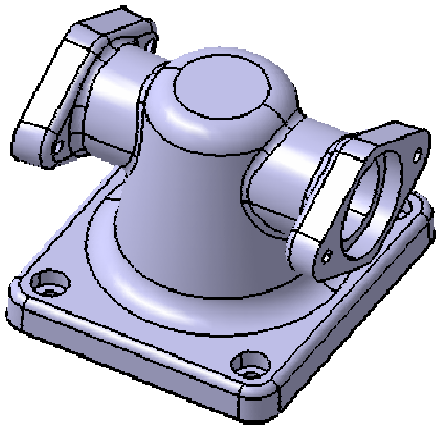
Tee Fitting

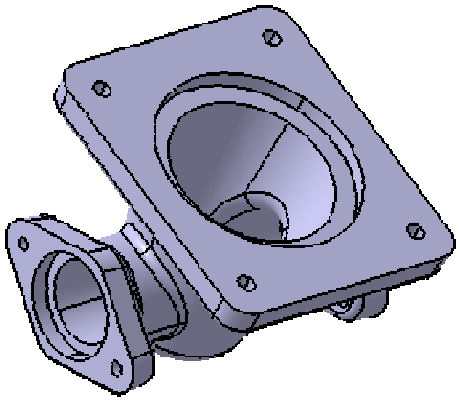
Part Design Advanced Exercise

60 min

In this exercise you will build the Tee Fitting by following a recommended process.

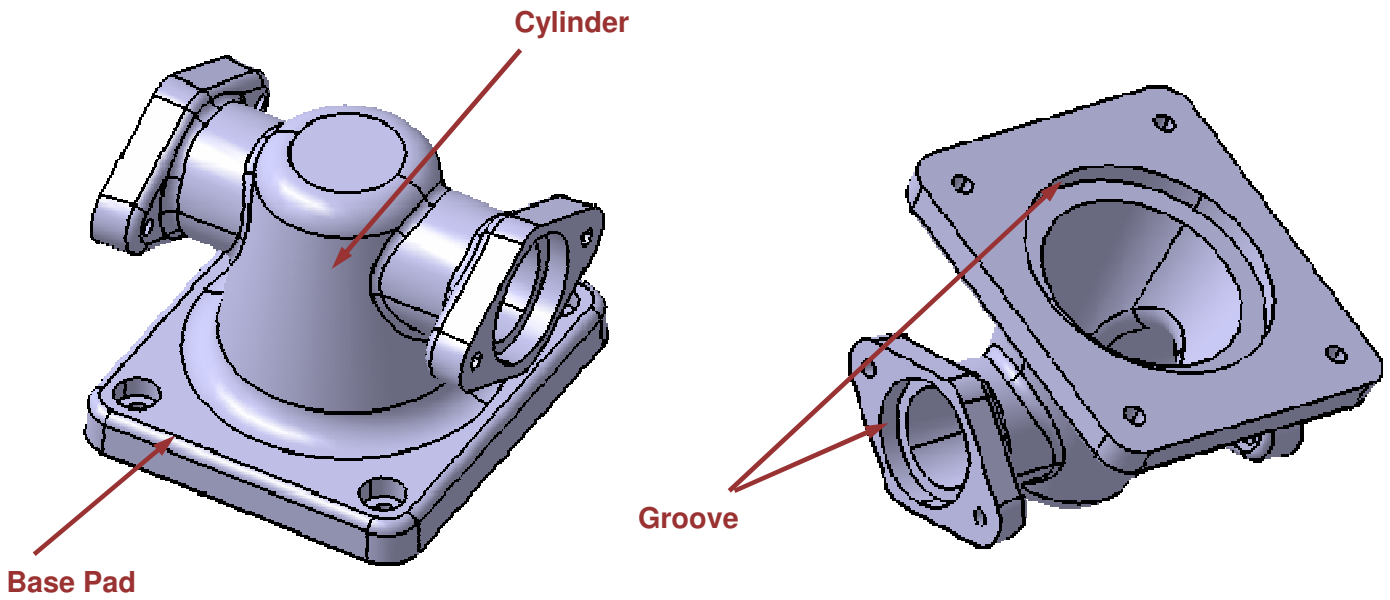
- **You will first understand the design intent of the Tee Fitting and identify its functional features.**
- **You will then study its drawing in detail to understand the dimensions and specifications.**
- **You will first design its outer rough shape and then the inner rough shape. You will remove this inner shape from the outer shape. To create the final part you will design the grooves and holes.**





Student Notes:

Design intent: Tee Fitting

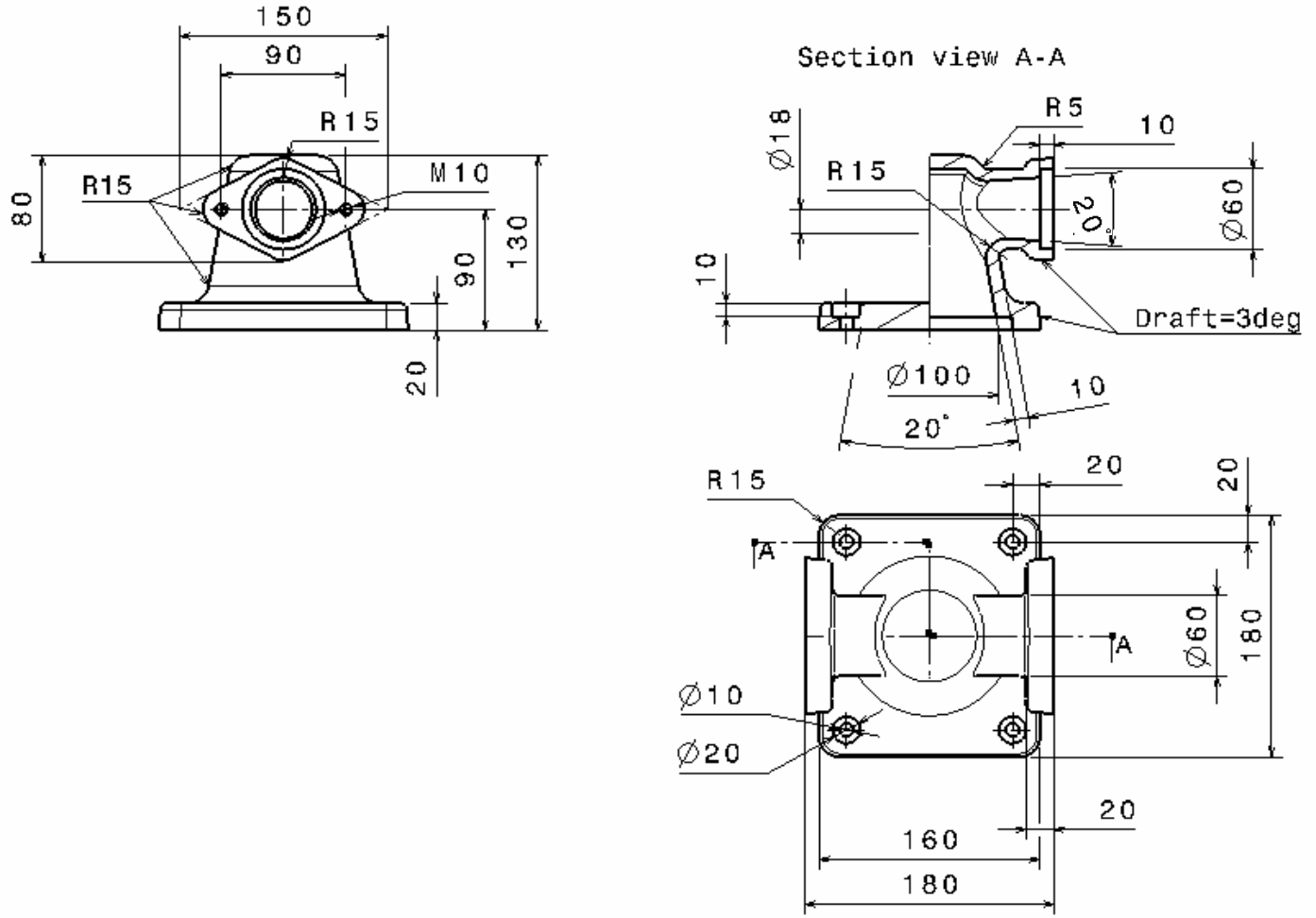


- The Tee Fitting is a casted part.
- Base pad is used for clamping and for providing support.

Student Notes:

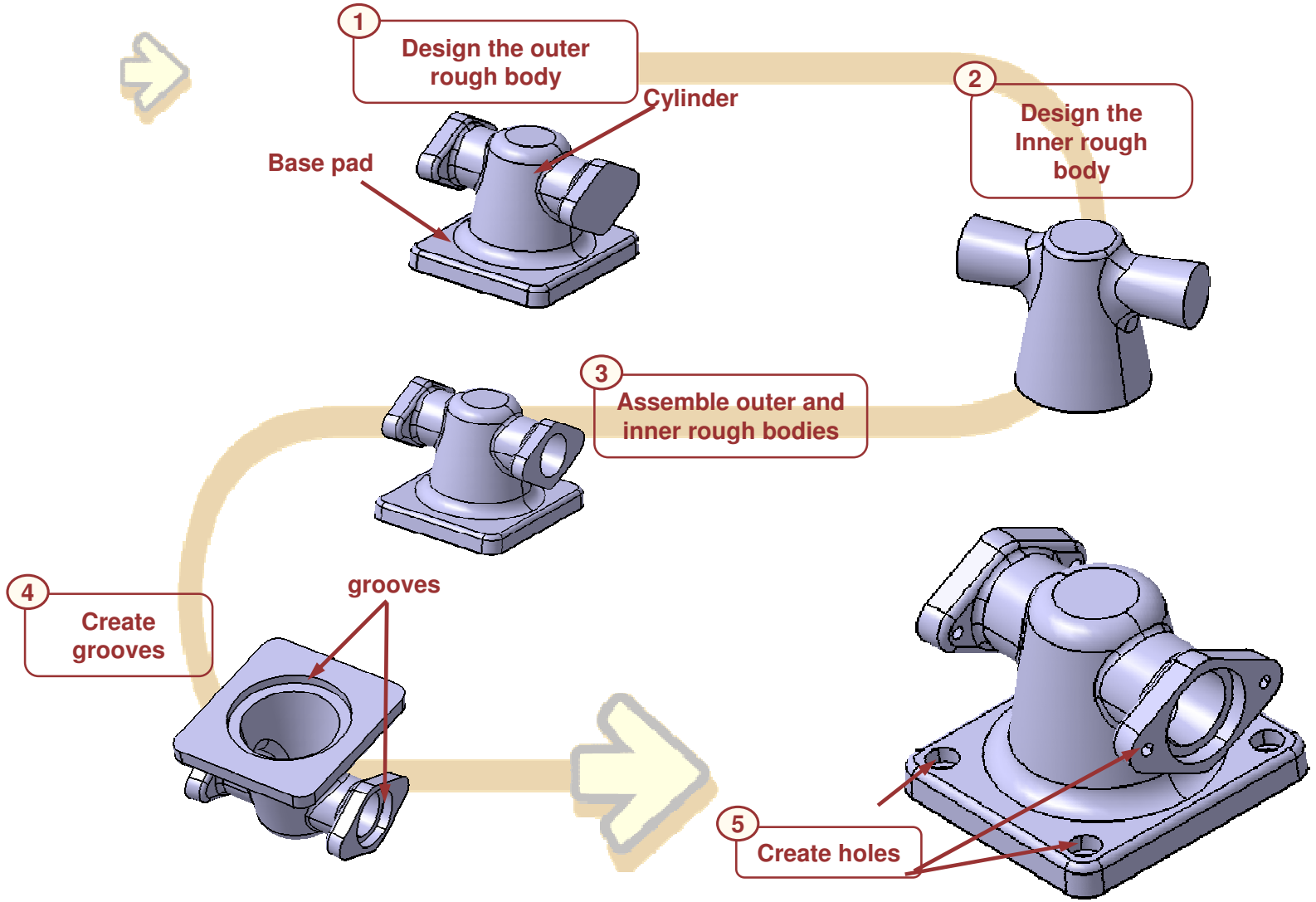
Tee Fitting Drawing

- Understand the drawing thoroughly to design the part according to the specifications.



Student Notes:

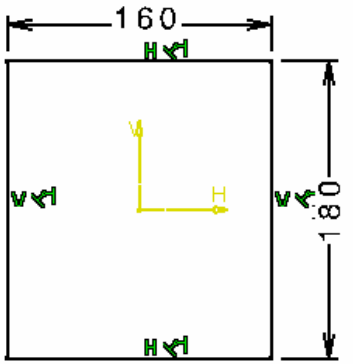
Design process: Tee Fitting

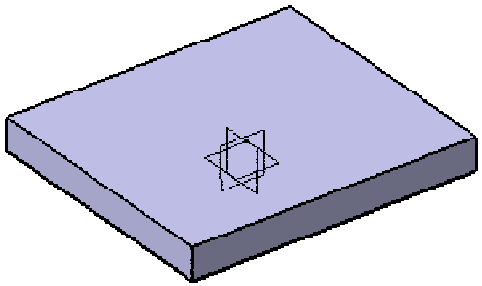


Student Notes:

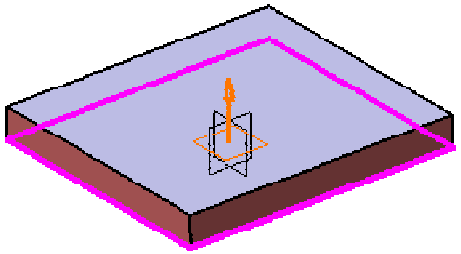
Step 1: Design the Outer Rough Body (1/7)

- Insert a body 'Body.2'
- Create a positioned sketch on XY plane oriented on X axis as shown.
- Pad it by 20 mm.
- Apply drafts and fillets

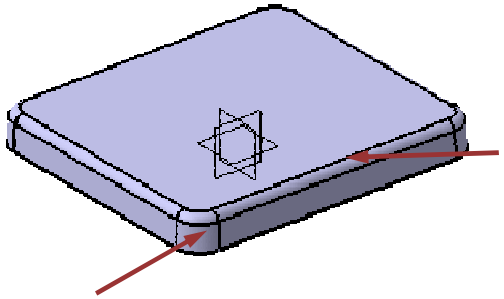




Pad length is governed by 'Thickness' parameter

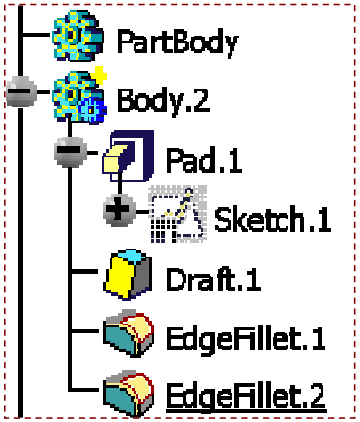


Draft Value = 3 deg. It is governed by 'Draft_Angle' parameter



Fillet of 15 mm on four edges. Driven by 'Radius_1' parameter.

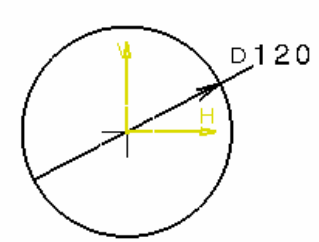
Fillet of 5 mm on top edge. Driven by 'Radius_2' parameter.

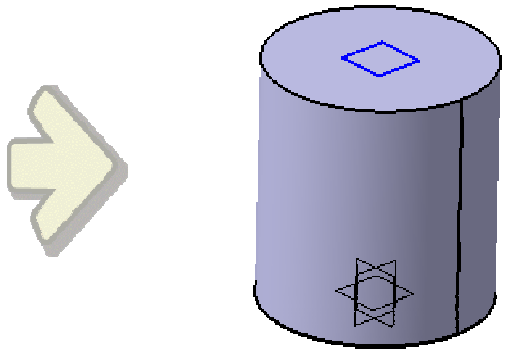


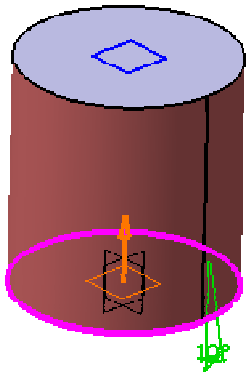
Student Notes:

Step 1: Design the Outer Rough Body (2/7)

- Insert 'Body.3'.
- In Geometrical set create a plane offset from XY plane at a distance of 130 mm. This distance is driven by 'Overall_Height' parameter.
- Create a positioned sketch on XY plane in body.3 with X axis orientation and origin as part origin.
- Pad this sketch upto Plane.1
- Apply drafts and fillets.

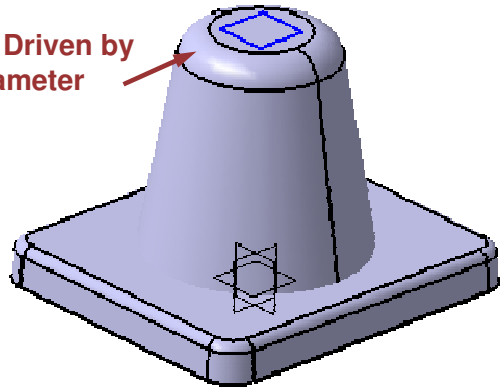


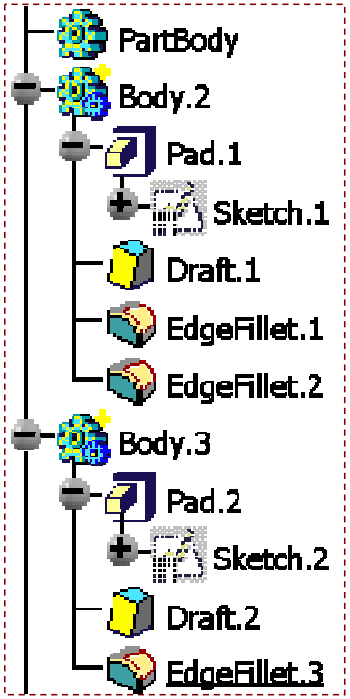




Draft Value = 10 deg

Fillet = 15 mm. Driven by 'Radius_1' parameter

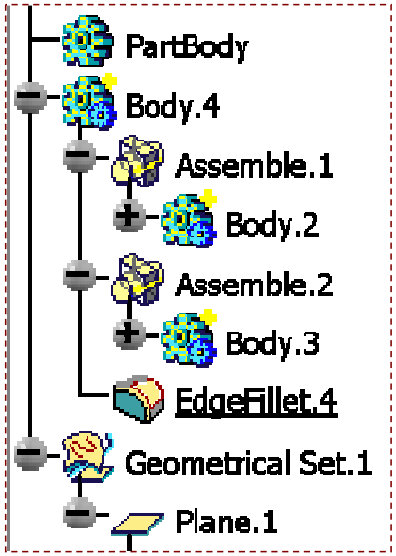


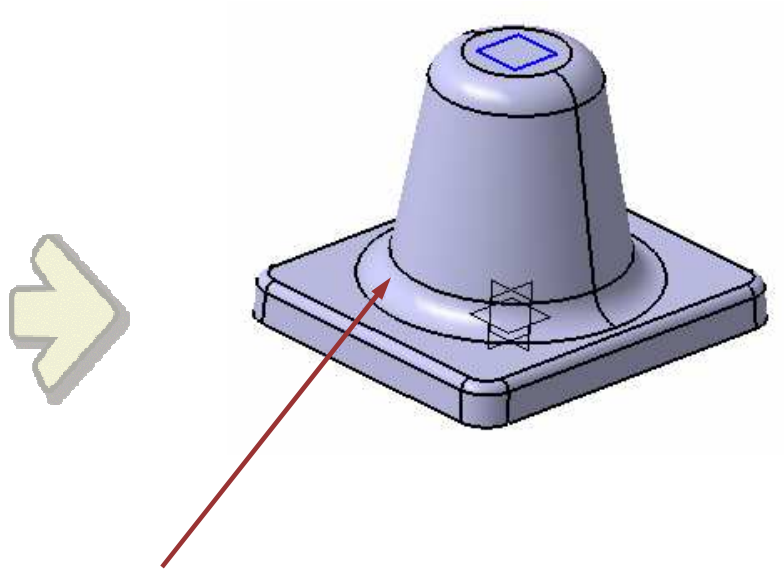


Student Notes:

Step 1: Design the Outer Rough Body (3/7)

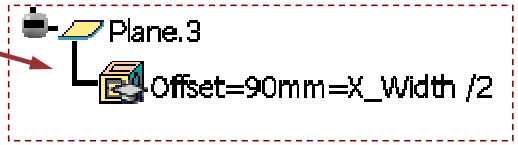
- Create 'Body.4'.
- Assemble Body.2 and Body.3 into a new body 'Body.4'.
- Apply Fillets.





Fillet = 15 mm. Driven by 'Radius_1' parameter.

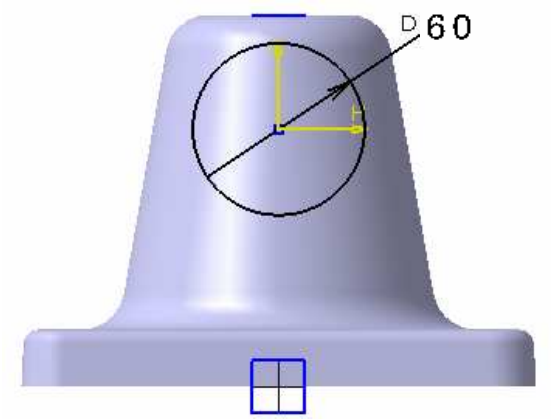
- Create Plane.3 by using parameters as shown.

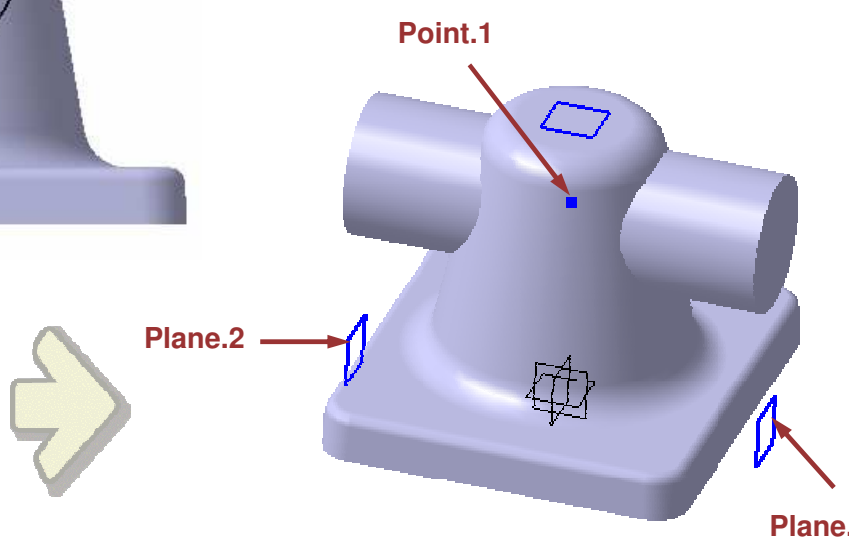


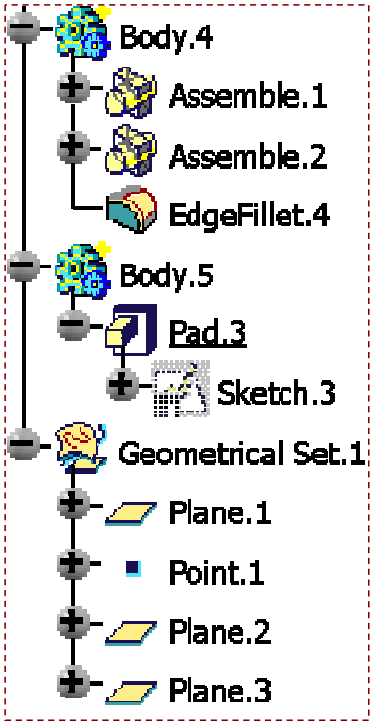
Student Notes:

Step 1: Design the Outer Rough Body (4/7)

- Insert 'Body.5'
- Create a point in Geometrical set (0,0,90). Also, create two planes on either side of YZ plane at 90 mm.These are Plane.2 and Plane.3.
- Create positioned sketch on YZ plane with projection point as Point.1
- Pad it using Lim 1 = plane.2 and lim 2 =plane.3.





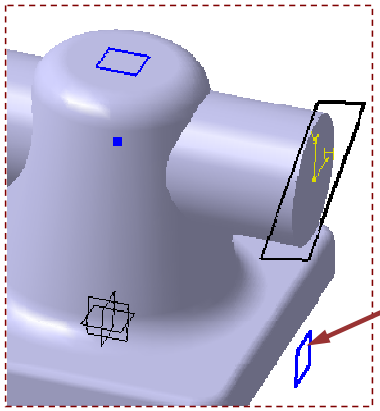


Plane.3 is Half the value of parameter 'X_Width'.

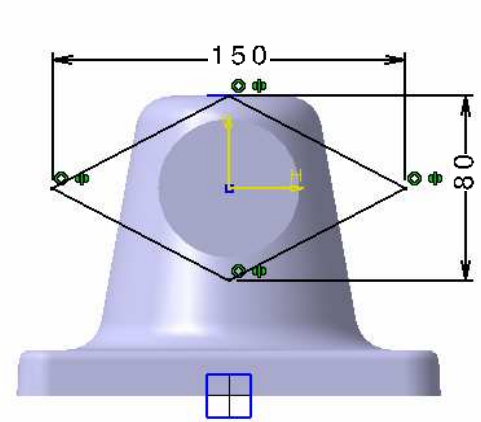
Student Notes:

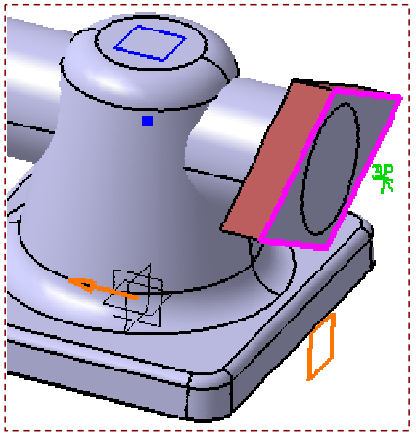
Step 1: Design the Outer Rough Body (5/7)

- Insert 'Body.6'
- Create a positioned sketch on Plane.3 and point.1 as projection point in Body.6.
- Pad it by 20 mm. This is driven by 'Thickness' parameter.
- Apply drafts and fillets. Mirror these features about YZ plane.



Plane.3

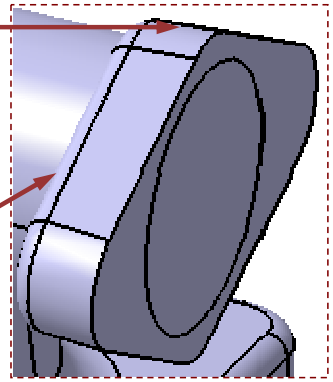


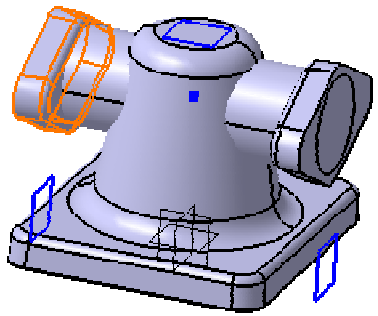


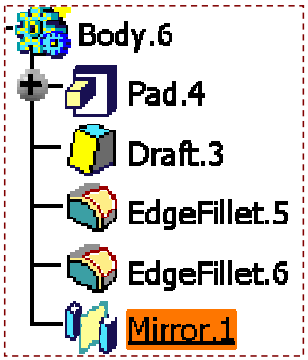
Draft = 3 deg. Driven by 'Draft_Angle' parameter.

Four Fillets driven by Radius_1 parameter.

Fillet driven by Radius_2 parameter.



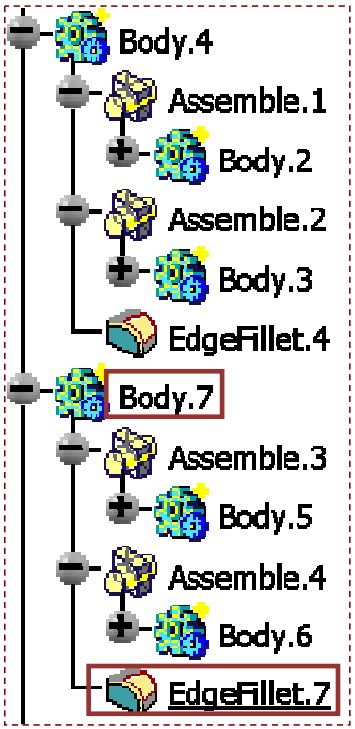


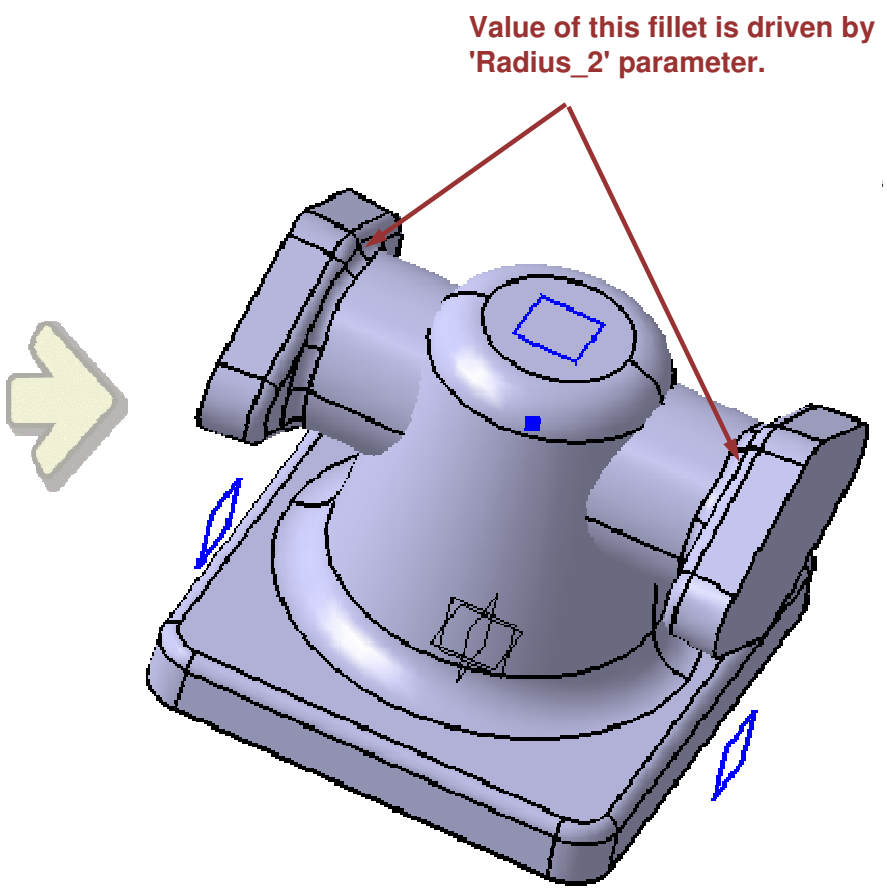


Student Notes:

Step 1: Design the Outer Rough Body (6/7)

- Insert 'Body.7' and assemble 'Body.5' and 'Body.6' with it.
- Apply edge fillet to this assembly.

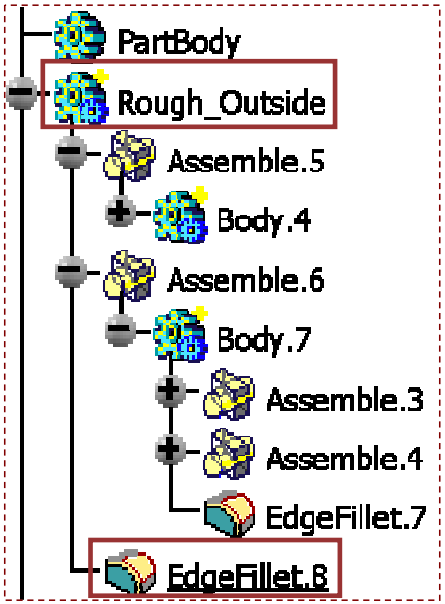


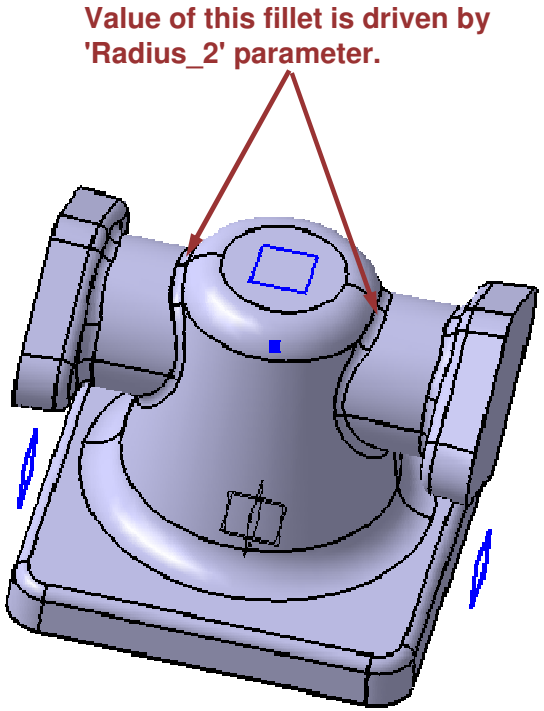


Student Notes:

Step 1: Design the Outer Rough Body (7/7)

- Insert 'Rough_Outside' and assemble previous assembly results of'Body.4' and 'Body.7' with it.
- Apply edge fillet to this assembly.





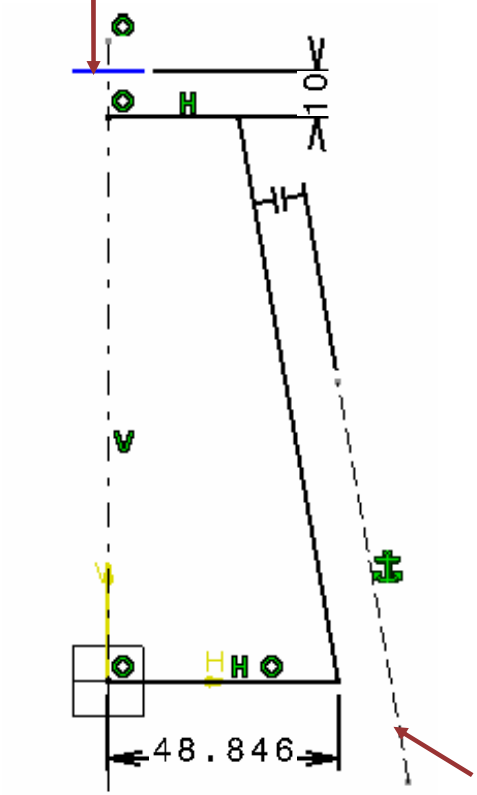
Load: PDG_Tee_Fitting_Step_1_End.CATPart

Student Notes:

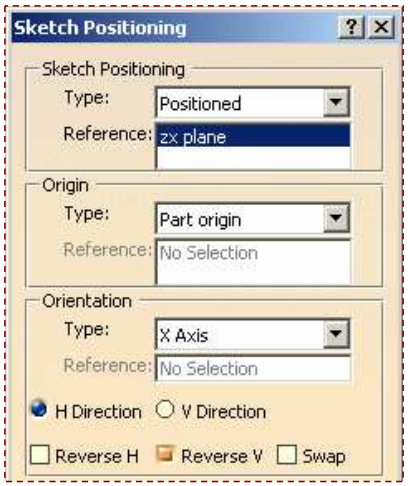
Step 2: Design the Inner Rough Body (1/2)

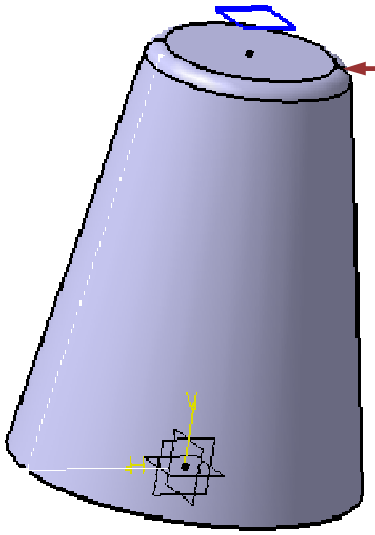
- Insert 'Rough_Inside' body.
- Create a positioned sketch on ZX plane.
- Create a groove from this sketch.
- Apply a edge fillet.

Plane.1



Draft.2 edge is made use of.



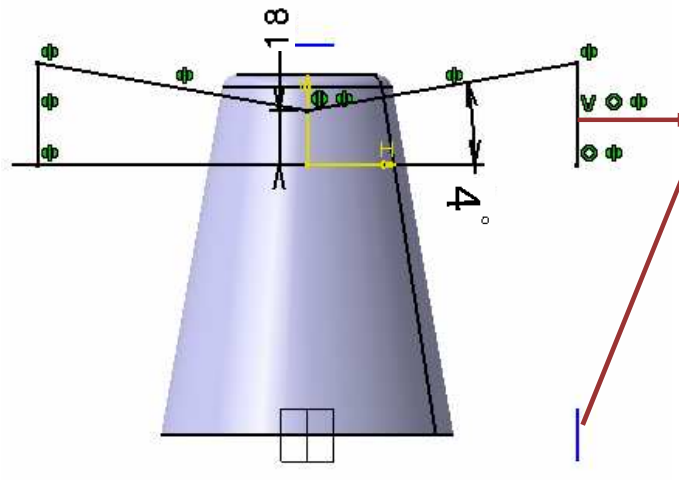


Fillet of 5 mm driven by 'Radius_2' parameter.

Student Notes:

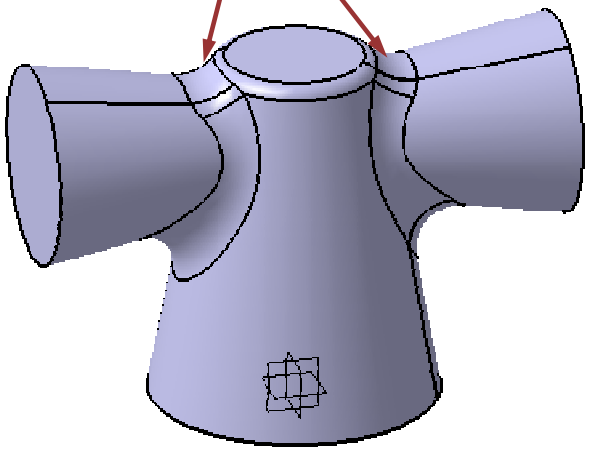
Step 2: Design the Inner Rough Body (2/2)

- Create a positioned sketch on ZX plane. Use projection point as point.1 and orient along X axis. Invert the V direction.
- Groove the sketch about H axis.
- Apply the fillet to the groove.



Constrain this with Plane.3

Fillet value driven by 'Radius_1' parameter.

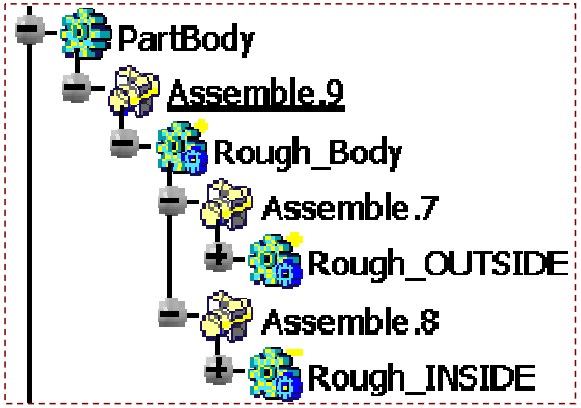


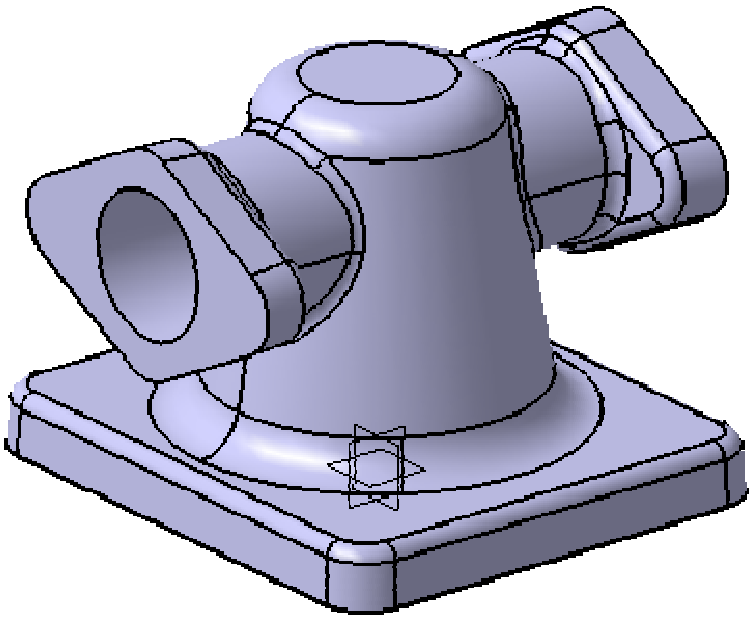
Load: PDG_Tee_Fitting_Step_2_End.CATPart

Student Notes:

Step 3: Assemble Outer and Inner Rough Bodies

- Insert 'Rough_body'
- Assemble 'Rough_OUTSIDE' and 'Rough_INSIDE' bodies to it
- Assemble this 'Rough_Body' to the PartBody



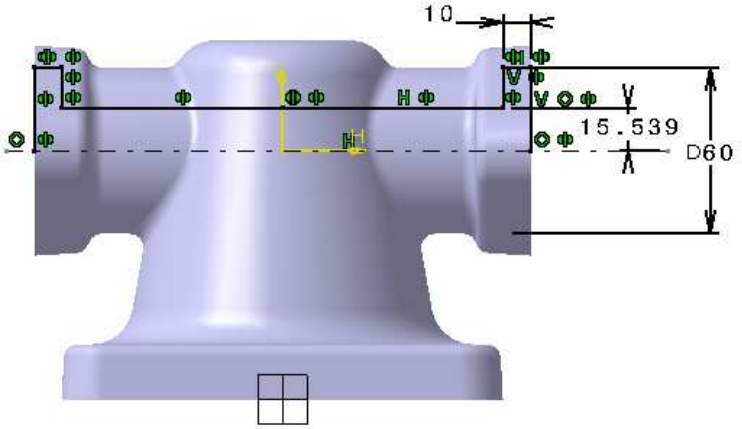


Load: PDG_Tee_Fitting_Step_3_End.CATPart

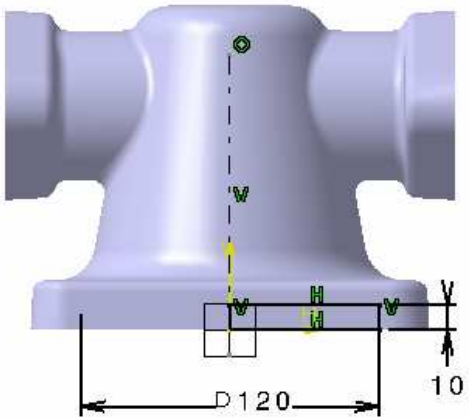
Student Notes:

Step 4: Create Grooves (1/2)

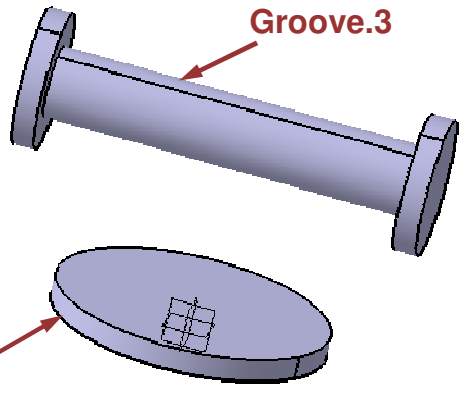
- Insert 'Body.8' to create grooves
- Create a positioned sketch on ZX plane with origin as Point.1.Use Plane.2 to constrain it. Create a groove using this sketch
- Create another positioned sketch on ZX Plane



Sketch for Groove.3



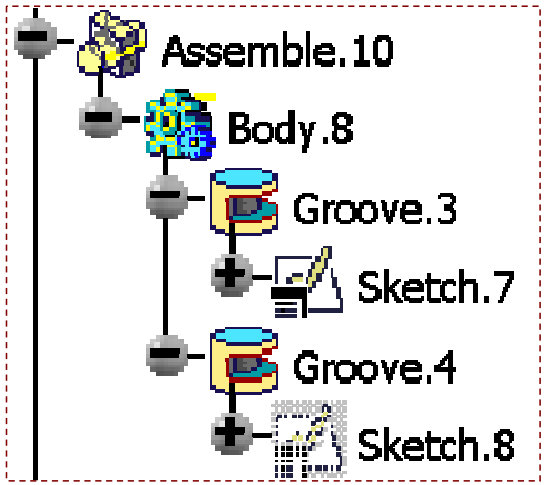
Sketch for Groove.4

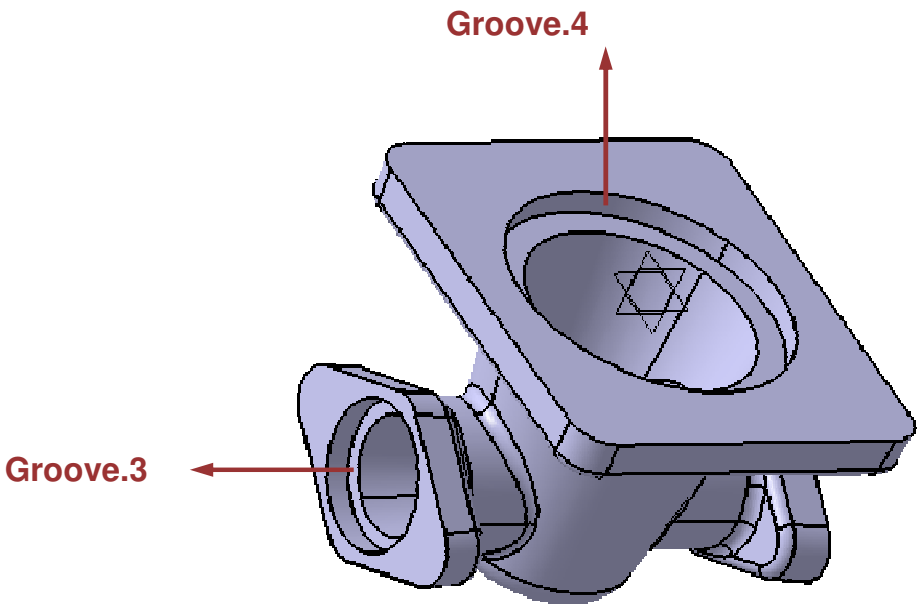


Student Notes:

Step 4: Create Grooves (2/2)

- Assemble 'Body.8' to the Part body



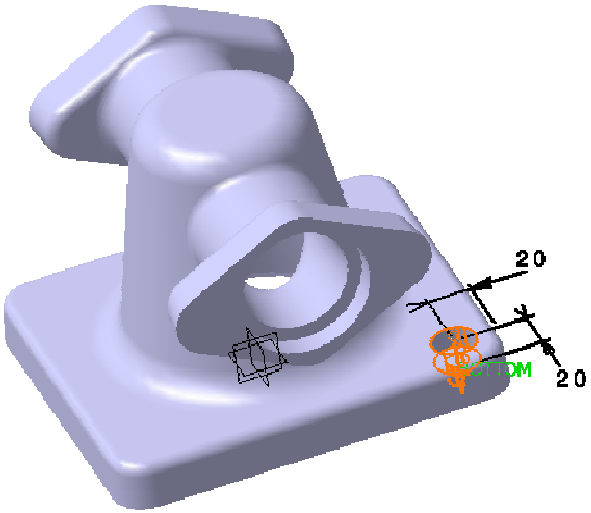


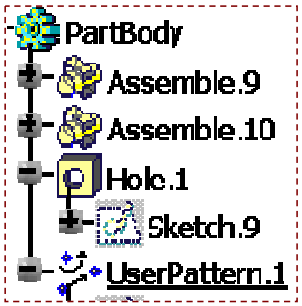
Load: PDG_Tee_Fitting_Step_4_End.CATPart

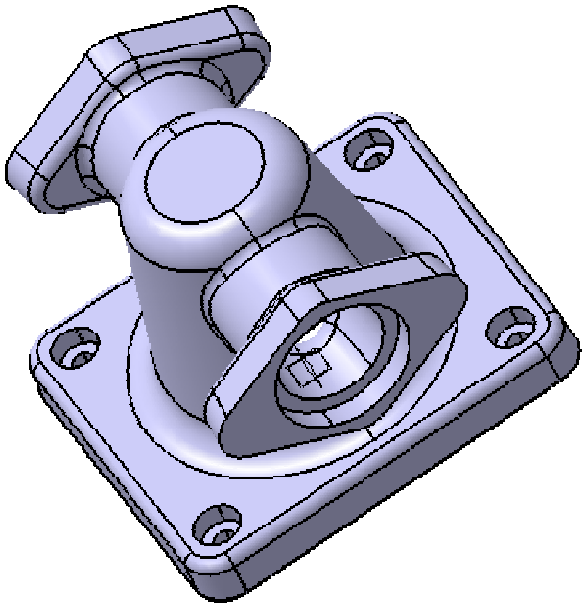
Student Notes:

Step 5: Create Holes (1/3)

- Design the hole with following attributes on the Base pad and position it as shown
  - Diameter =10 mm, up to next
  - Counter bored with Diameter = 20 mm and depth = 10 mm
  - Constrain it with respect to the sketch edge for base Pad
- Pattern the hole. Constrain the pattern sketch using the base pad sketch.



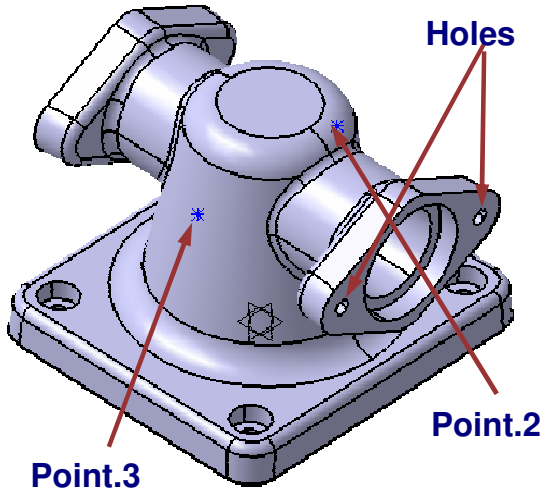




Student Notes:

Step 5: Create Holes (2/3)

- Design two threaded holes with following attributes on right face
  - Diameter = 8 mm, up to next
  - Thread Diameter = Thread depth =10 mm
- Create point.2 and point.3 in geometrical set.
- Use Point.1 as reference point to create them.
- For Point. 2: X=0, Z=0 and govern its Y coordinate by parameter 'Horizontal_Hole_Gap (initially set to 90 mm)'. The Y coordinate is half of this parameter value.
- For Point. 3: X=0, Z=0 and govern its Y coordinate by parameter 'Horizontal_Hole_Gap (initially set to 90 mm)'. The Y coordinate is half of this parameter value. This value is negative.
- Use these points to position the center points of the two holes.

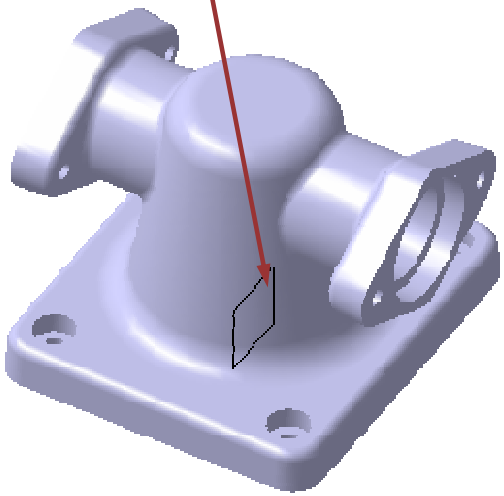


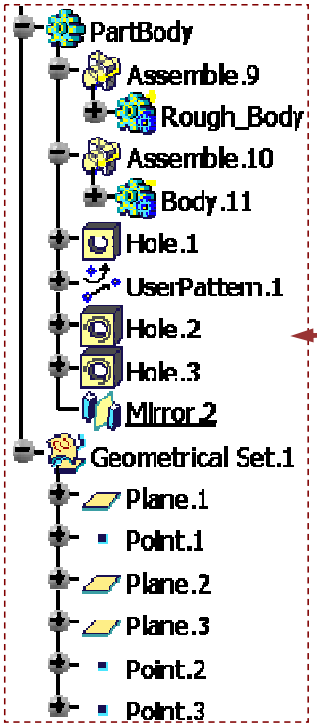
Student Notes:

Step 5: Create Holes (3/3)

- Mirror about YZ plane.
- You can optimize your design by manipulating the parameter values.

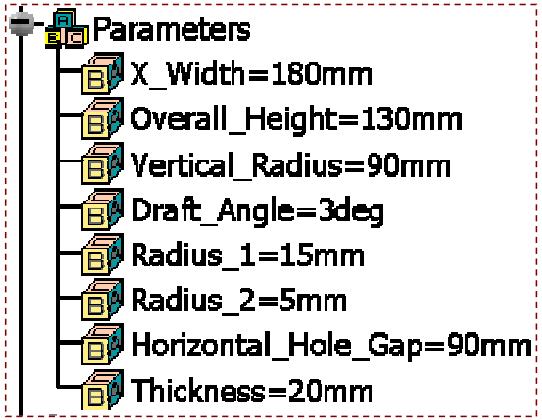
Mirror about this plane





Tree structure showing features

Tree structure showing parameters



Final Result: PDG_Tee_Fitting

Student Notes:

Pedal Crank

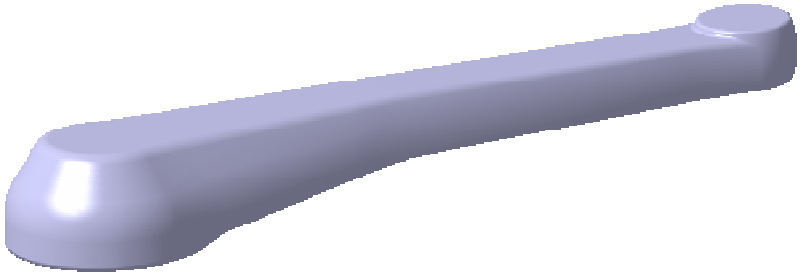
Part Design Advanced Exercise

60 min

In this exercise you will design a rough pedal crank.

To do so, you will use

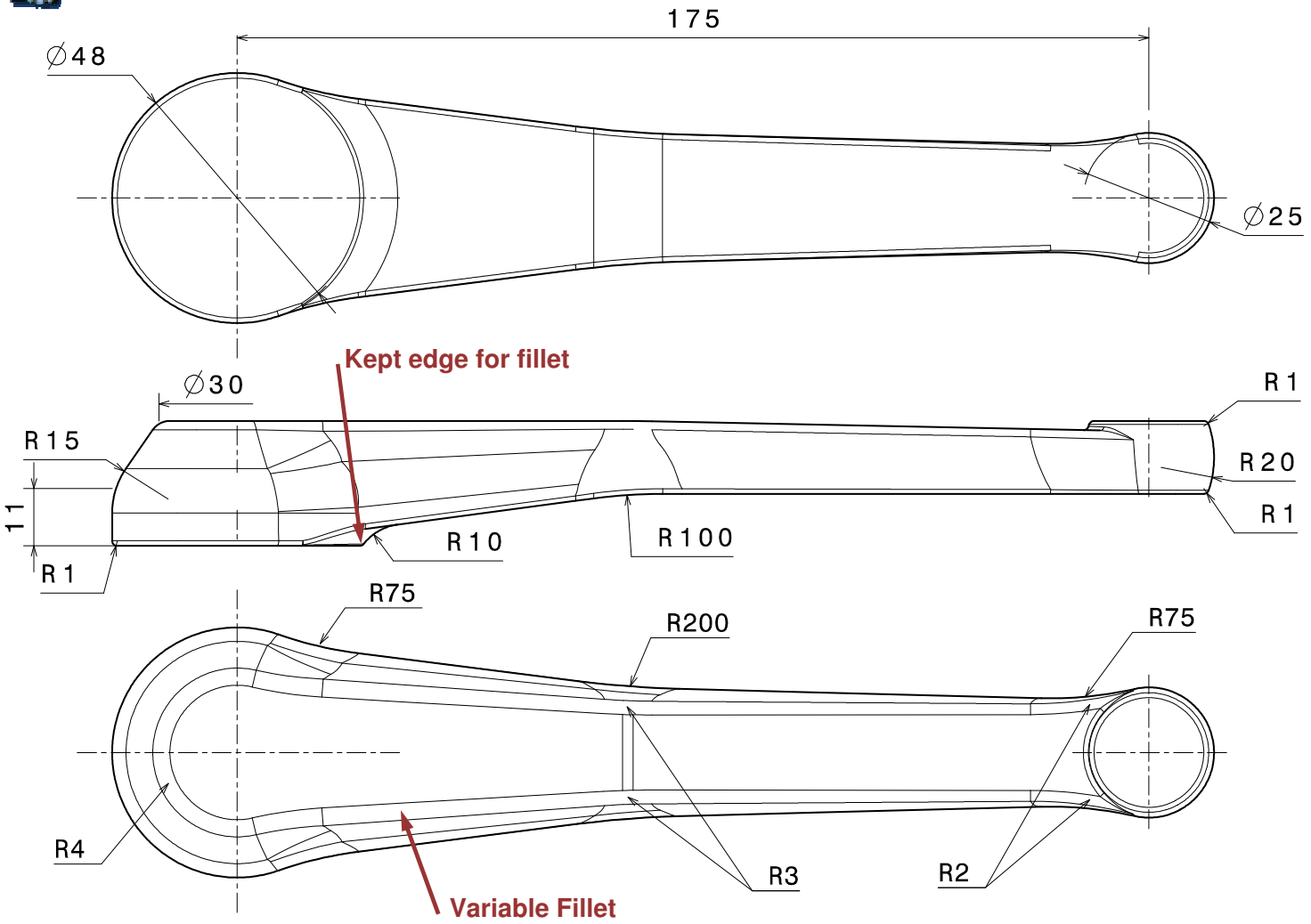
- **Multi-Sections Solids**
- **Shafts**
- **Boolean Operations**
- **Constant Fillets**
- **Variable Fillets**



Student Notes:

Pedal Crank Drawing

You can use 'Pedal_Crank_Right_Result.CATPart' to examine the expected part.



Student Notes:

Design process: Pedal Crank

1-2-3

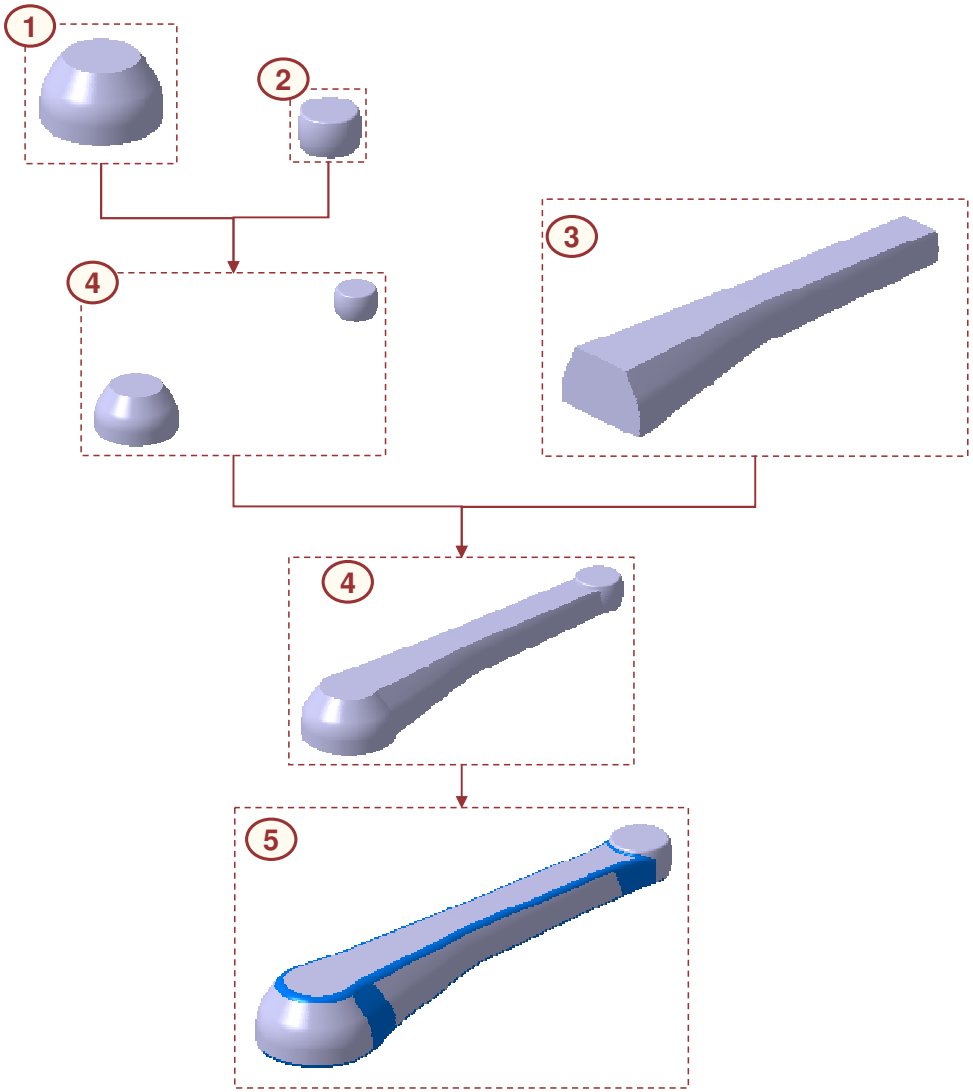
Create the individual bodies and their intrinsic fillets

4

Make Boolean Operations

5

Create the last Fillets



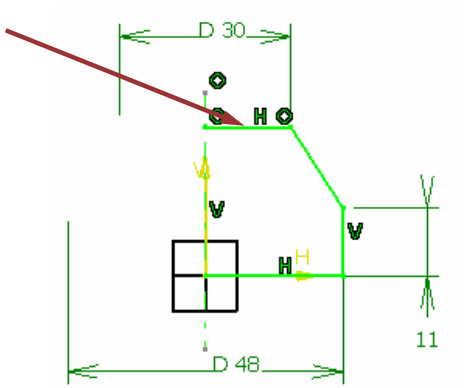
Student Notes:

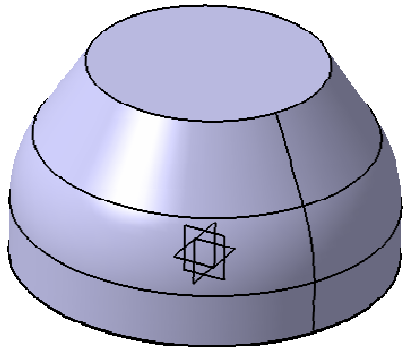
Step 1: Create the Crank Axis

Pedal_Crank_Right_Shape_Start.CATPart

- Create the Crank Axis Sketch in ‘Crank Link Specs’ geometrical set
  - On the YZ Plane
  - Using the following specifications
- Create a new body called ' Crank_Axis ' for the following part design features
  - Create a Shaft
  - Add the fillet of 15 mm

Coincident with ' Plane.5 '

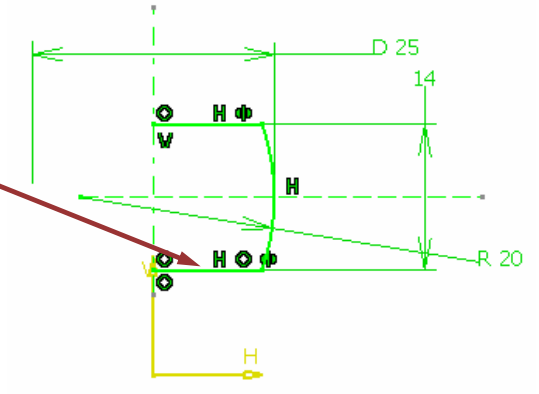




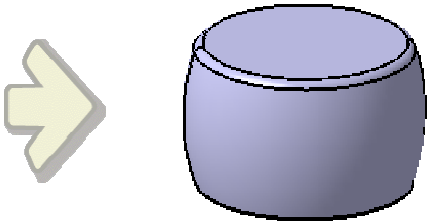
Student Notes:

Step 2: Create the Pedal Axis

- Create the Pedal Axis Sketch in 'Pedal Axis Specs' geometrical set
  - On the Plane.1
- Create a new body called ' Pedal_Axis ' for the following part design features
  - Create a Shaft
  - Add a fillet to the edge of the top face of 1mm as shown.



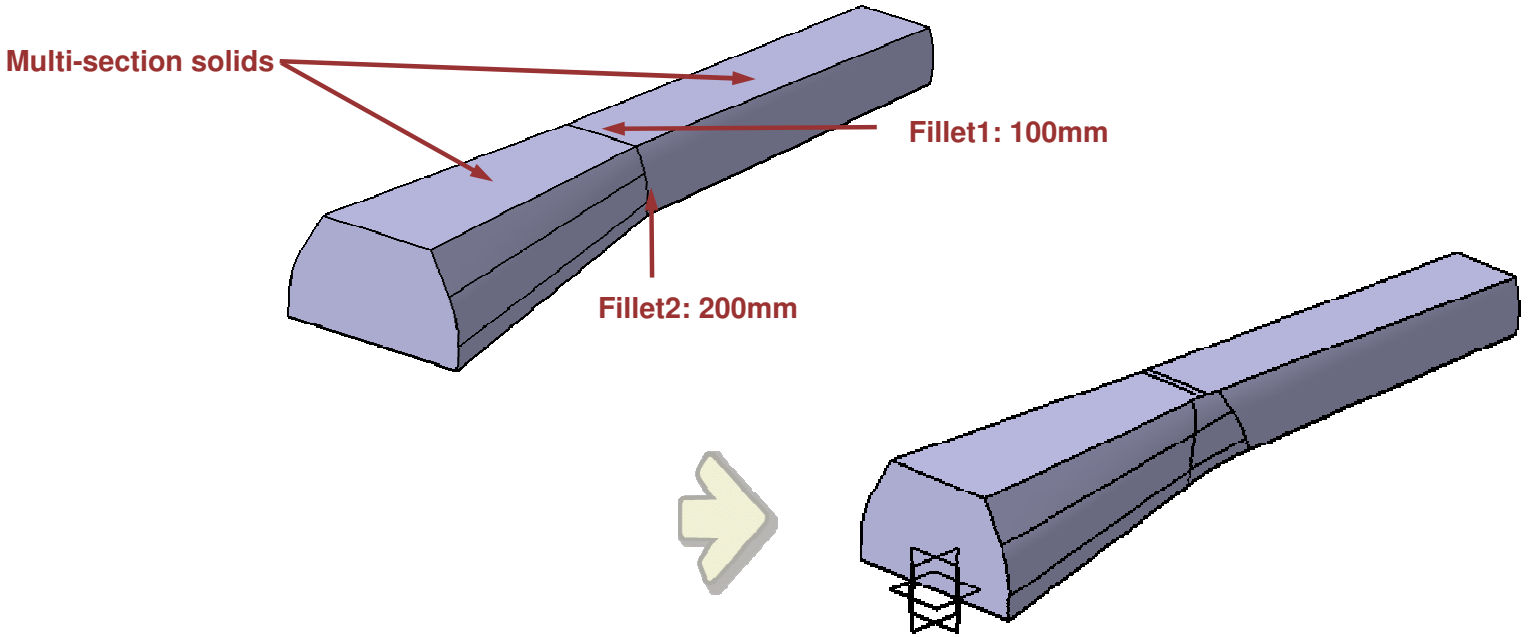
Coincident with ' Plane.3 '



Student Notes:

Step 3: Create the Crank Link

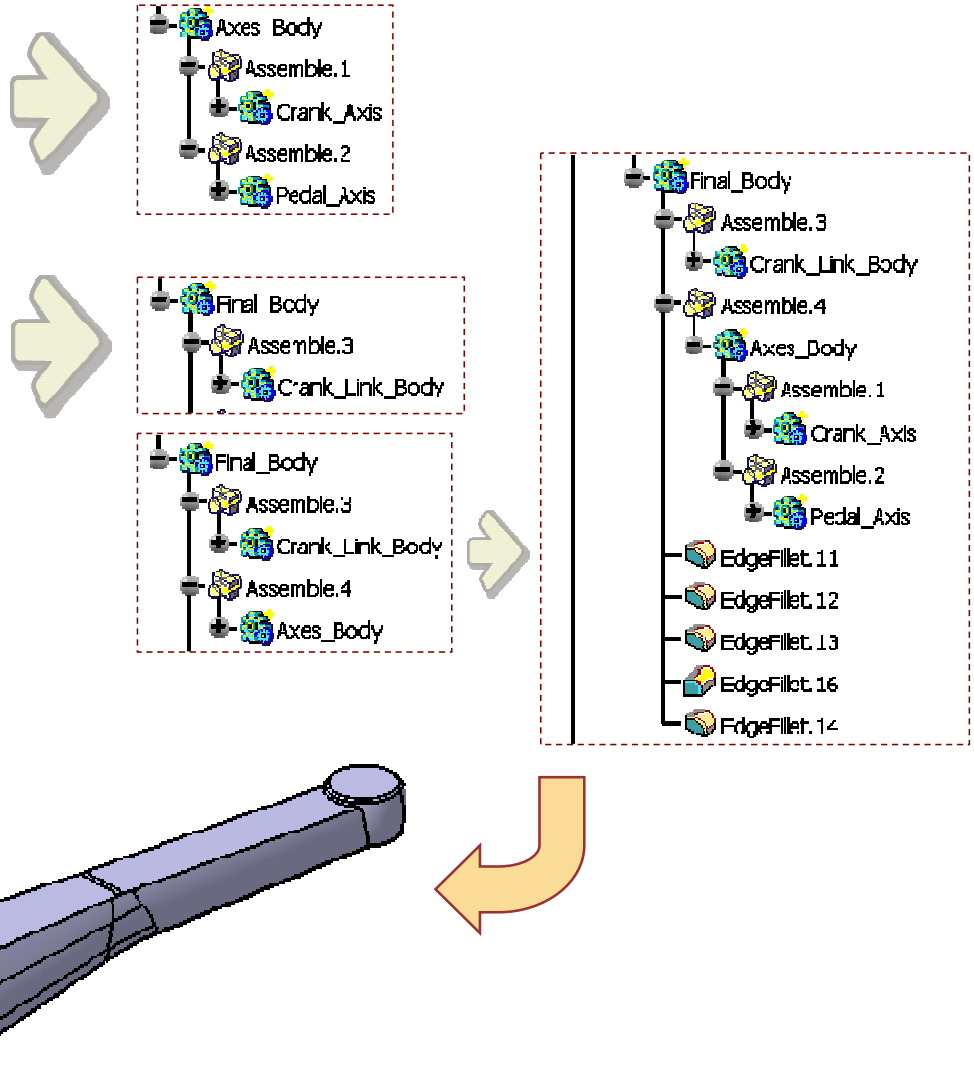
- Create a new body called ' Crank_Link_Body ' for the following part design features
  - Create a multi-section solid using the sketches ' section 1 ' and ' section 2 '
  - Create a multi-section solid using the sketches ' section 2 ' and ' section 3 '
  - Add the fillets as shown



Student Notes:

Step 4: Make the Boolean operations

- Create a new body called ' Axes_Body '
  - Assemble ' Crank_Axis ' Body and ' Axes_Body '
  - Assemble ' Pedal_Axis ' body
- Create a new body called ' Final_Body '
  - Assemble ' Crank_Link_Body ' and ' Final_Body '
  - Assemble ' Axes_Body ' and ' Final_Body '
- Add the fillets



Student Notes:

Step 5: Create the last Fillets

Result: 'Pedal_Crank_Right_Shape_End.CATPart'

- Create the fillets
  - In the following order
  - Using the dimensions in the drawing
- Assemble the ' final body ' with the ' PartBody '