



## CATIA V5 Training Exercises

# CATIA Generative Sheetmetal Design

Version 5 Release 19  
September 2008

EDU\_CAT\_EN\_SMD\_FX\_V5R19

Student Notes:

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Student Notes:

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Student Notes:

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# Sheet Metal Walls

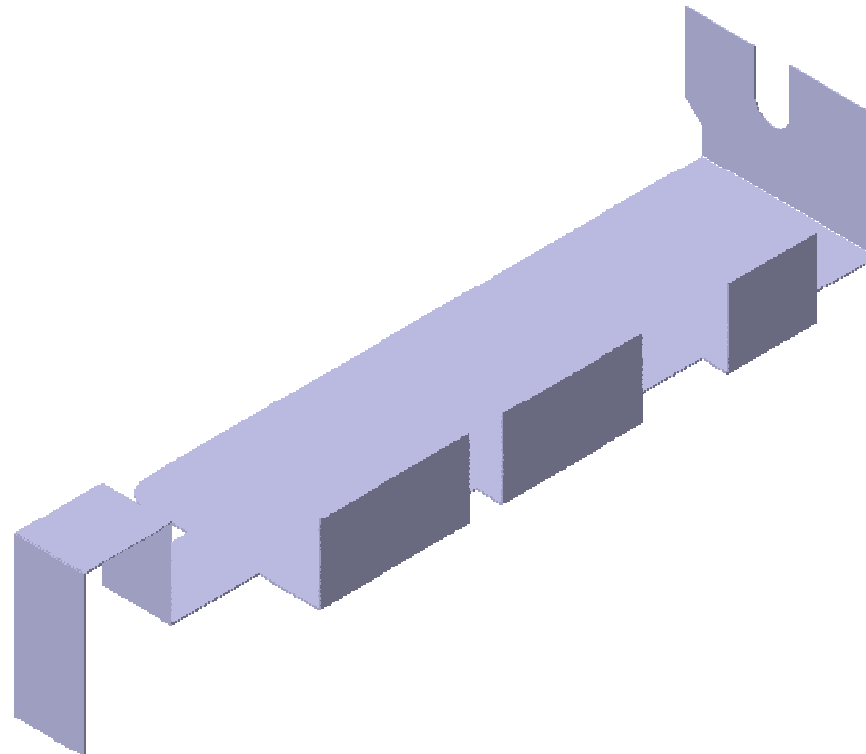
## Recap exercise



45 min

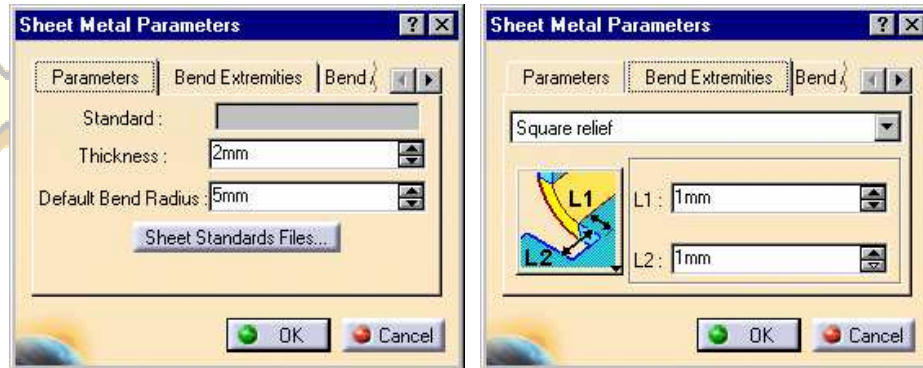
In this exercise you will create :

- First Wall
- Secondary Walls
- Tangent Walls
- Profile Walls
- Wall on Edge Walls
- Extruded Walls



Student Notes:

## Design Process: Wall Creation (1/3)

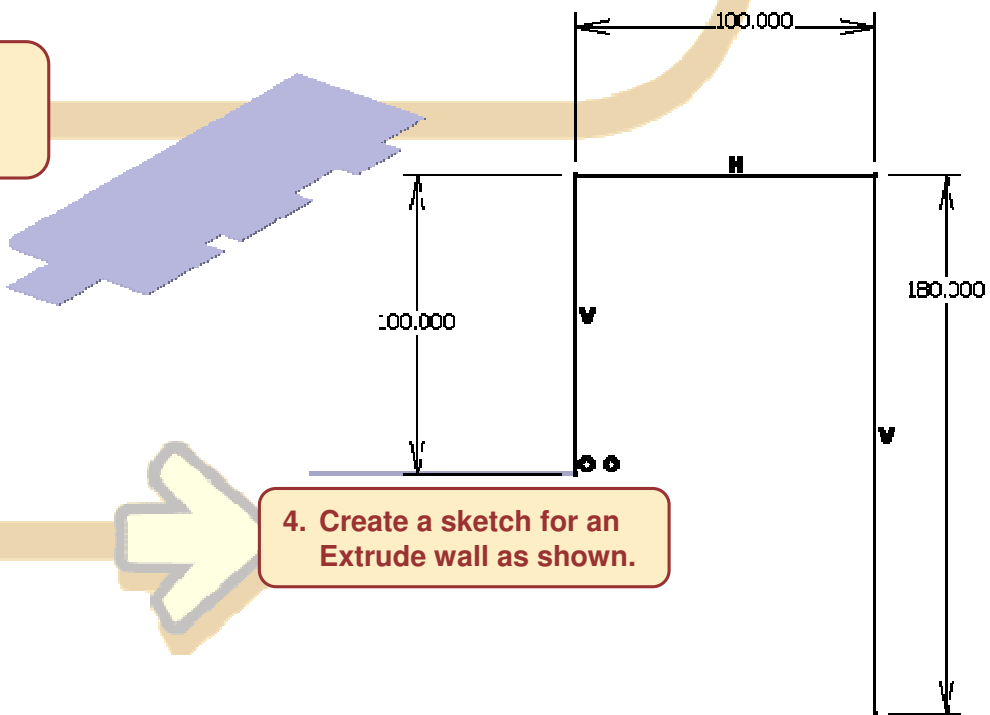


1. Open Bracket.CATPart. Set the default parameters as shown. Leave the other parameters at the defaults.

2. Create a Profile Wall from the existing sketch.

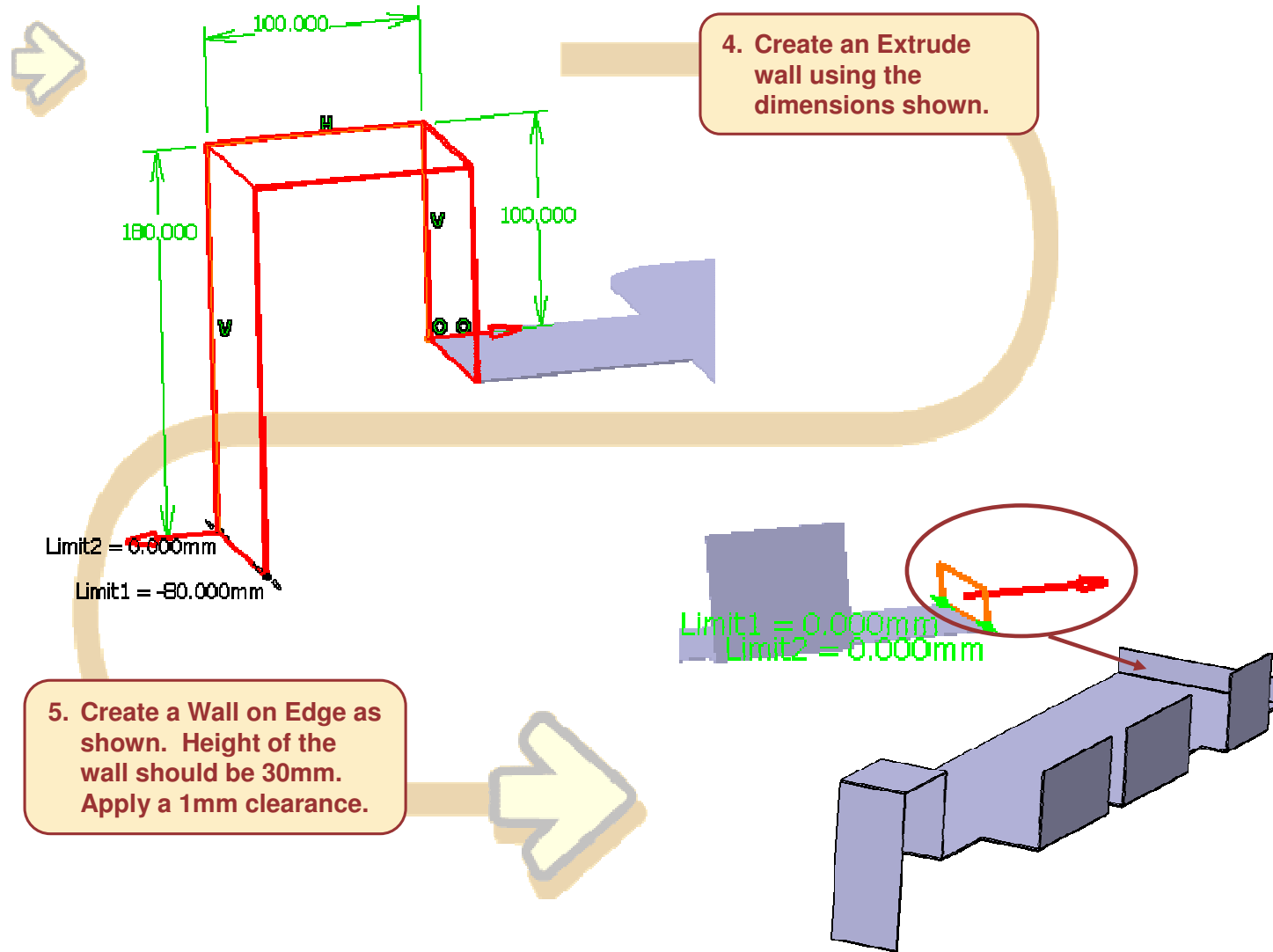
3. Create 3 walls using the Wall on Edge function as shown. All three walls will measure 90mm in Height.

4. Create a sketch for an Extrude wall as shown.



Student Notes:

## Design Process: Wall Creation (2/3)







Student Notes:

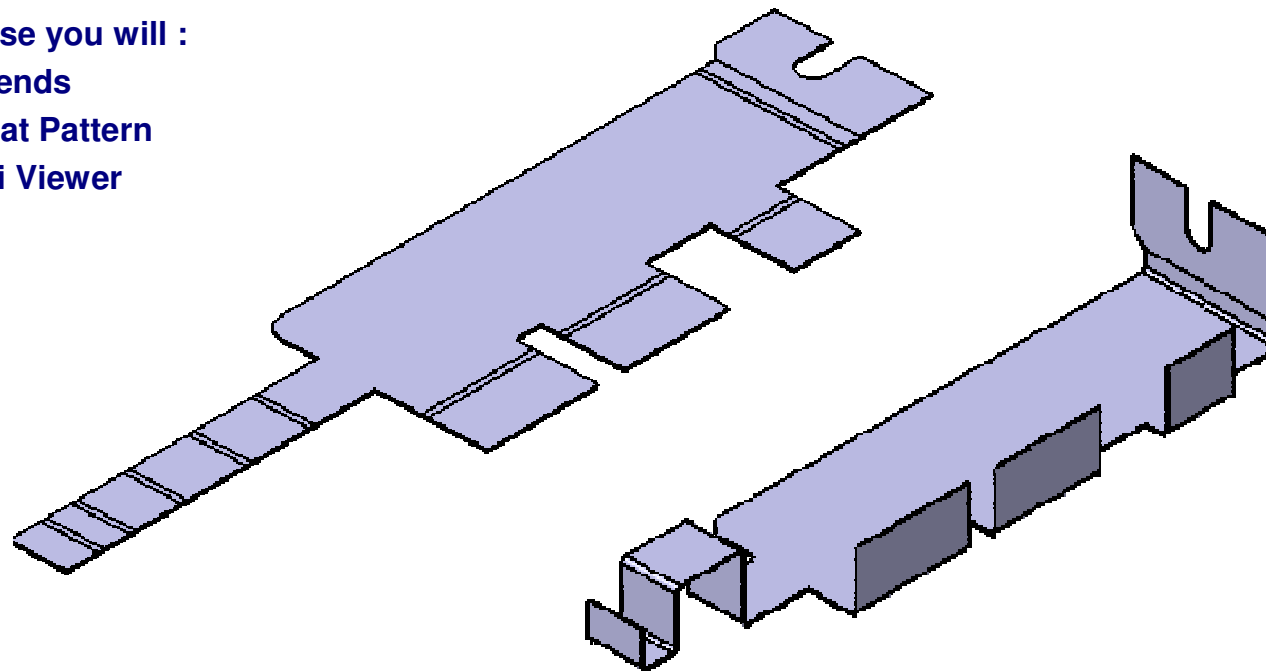
# Bends and Unfolded Mode

## Recap Exercise



In this exercise you will :

- Create Bends
- Create Flat Pattern
- Use Multi Viewer

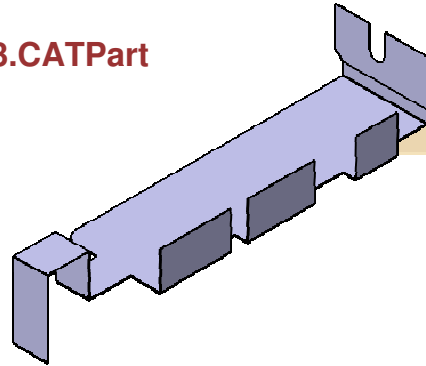


Student Notes:

## Design Process: Bend and Flat Pattern Creation (1/3)

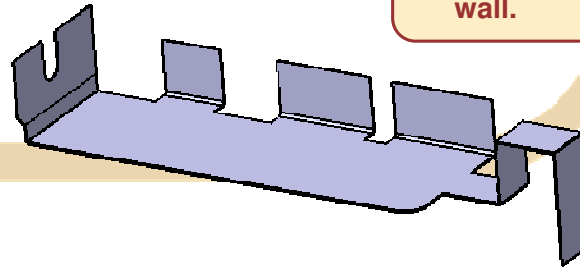


Part used: Bracket\_3.CATPart

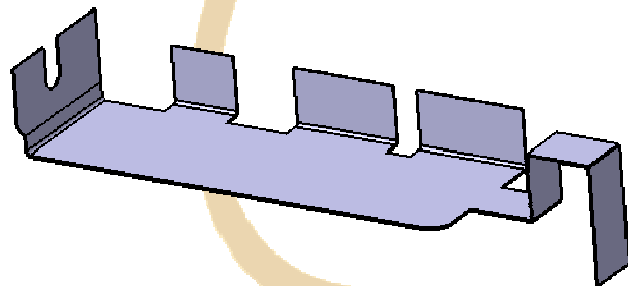


1. Open Bracket.CATPart. If you did not complete the exercise from lesson 2 you can use bracket\_3.CATPart instead.

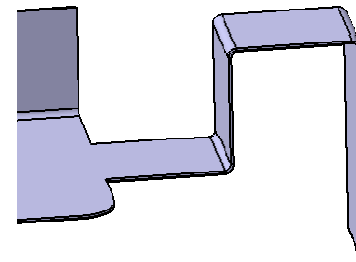
2. Create bends between the 3 Wall on Edges and the first wall.



3. Create bend between the first wall and the Wall on Edge with clearance.

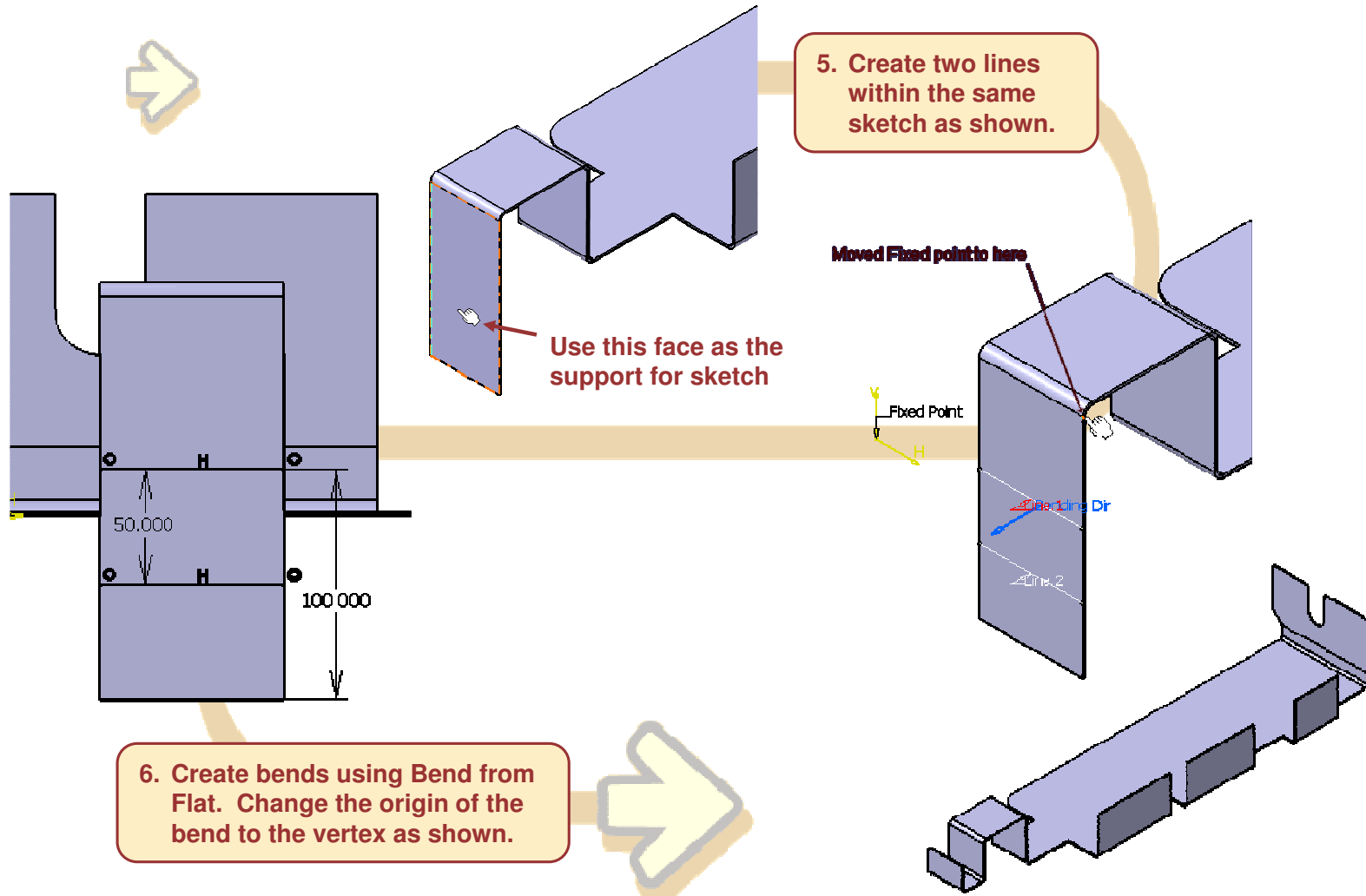


4. Create bends for the extruded wall.



Student Notes:

## Design Process: Bend and Flat Pattern Creation (2/3)



6. Create bends using Bend from Flat. Change the origin of the bend to the vertex as shown.

5. Create two lines within the same sketch as shown.

Use this face as the support for sketch

Moved Fixed point to here

Fixed Point

Fixed Point

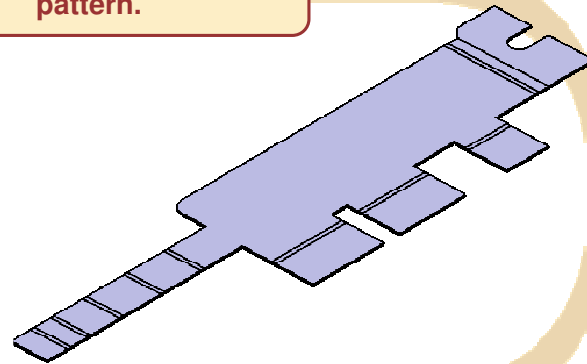
## Design Process: Bend and Flat Pattern Creation (3/3)



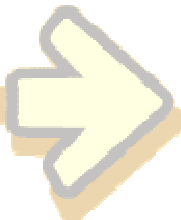
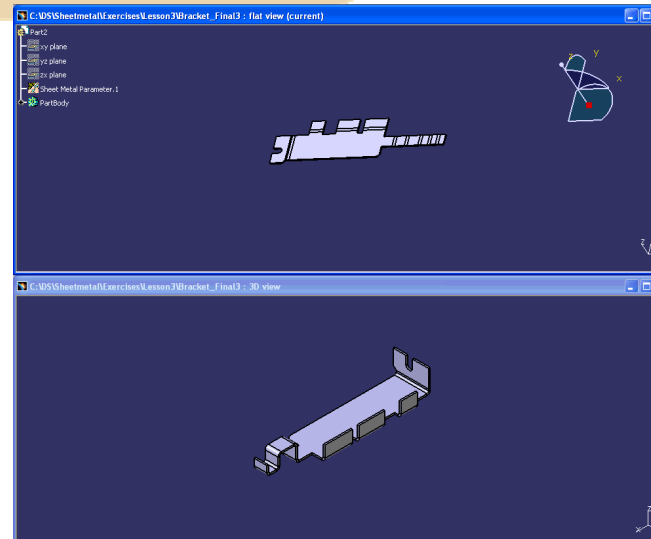
You can check your result with the Final Part : Bracket\_Final3.CATPart



7. Create a flat pattern.



8. Use the multi-Viewer to view both unfolded and folded views at the same time. Save and close the model.



# Flanges Exercises

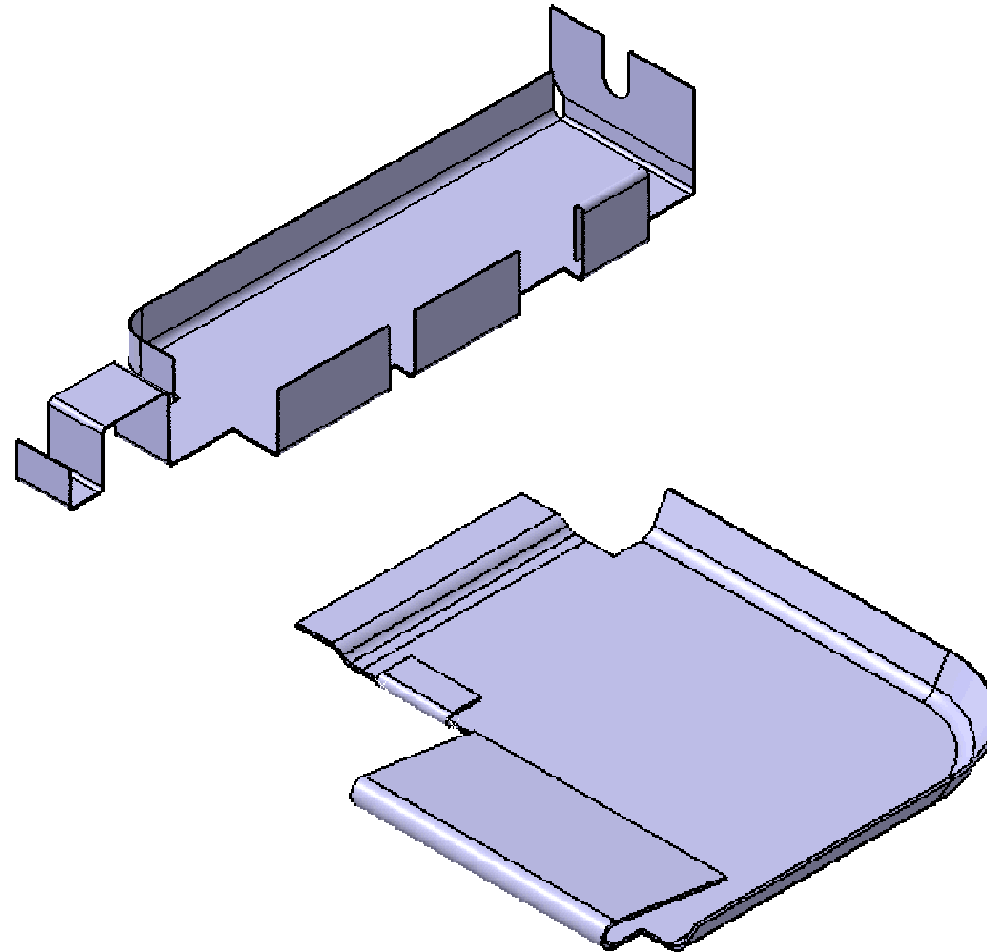
## Recap Exercises



30 min

In this Exercise you will create :

- A Simple Flange
- A Hem
- A Tear Drop
- A User Defined Flange



Student Notes:

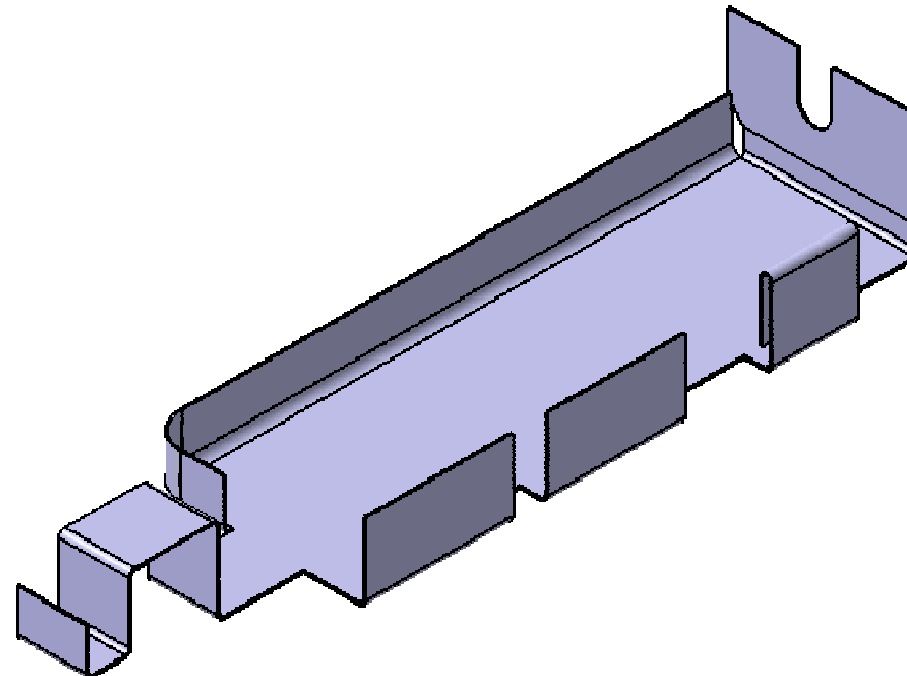
# Exercise 1

## Create Flanges for Bracket Model



In this Exercise you will create :

- A Simple Flange
- A Hem



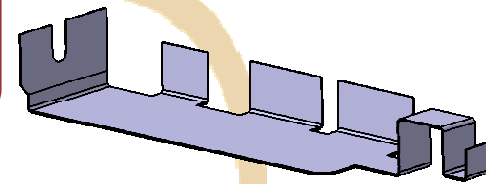
## Design Process: Flange Creation on Bracket



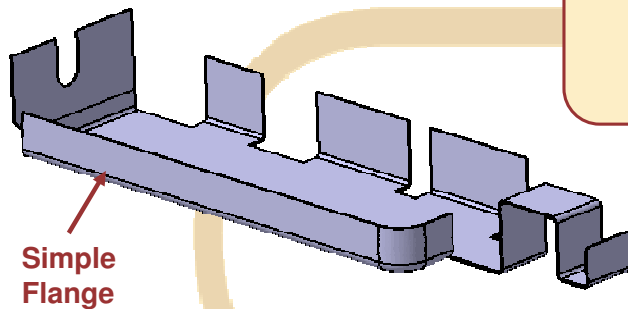
Part used: Bracket\_4.CATPart



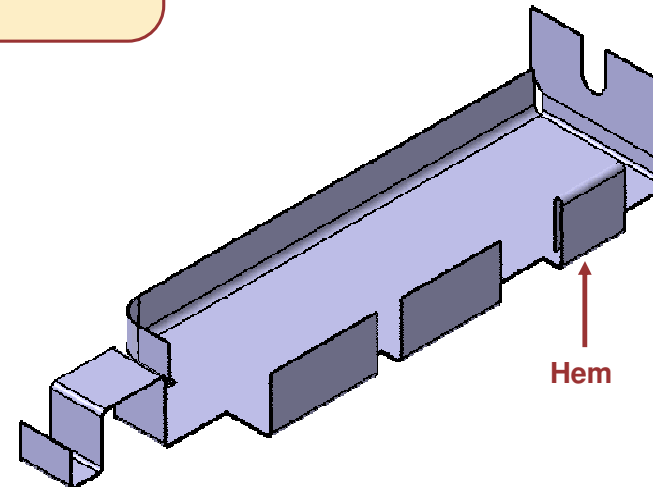
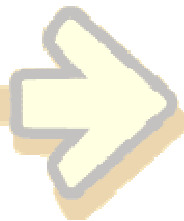
1. Open Bracket.CATPart. If you did not complete the exercise from lesson 3 you can use bracket\_4.CATPart instead.



2. Create a Simple flange as shown.  
Values are:  
Radius = 10.0mm  
Length = 50.0mm  
Angle = 90deg.



3. Create a hem as shown.  
Values are:  
Radius = 5.0mm  
Length = 70.0mm  
Save and close the model.



Student Notes:

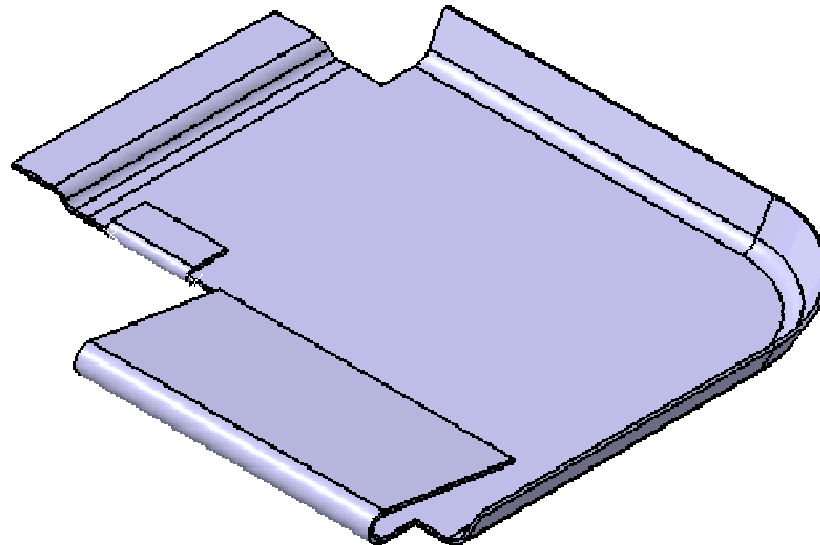
## Exercise 2

### Practice Creating Flanges



In this Exercise you will create :

- A Simple Flange with Propagation
- A Relimited Hem
- A Tear Drop Flange
- A User Defined Flange





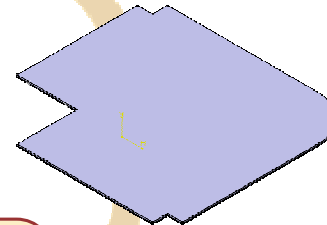
## Design Process: Practice Creating Flanges (1/2)



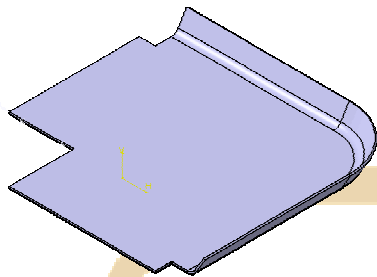
Part used: Sample\_Flanges.CATPart



1. Open  
Sample\_Flanges.CATPart.

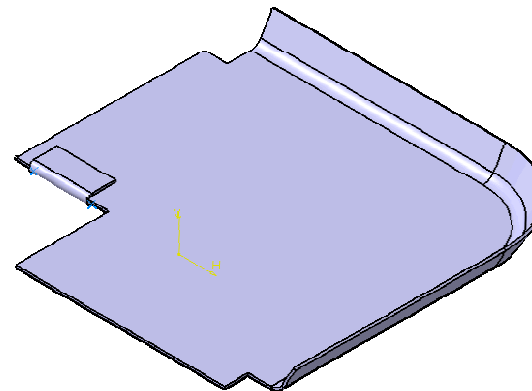
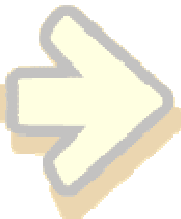


2. Create a Simple flange as shown.  
Values are:  
Radius = 5.0mm  
Length = 10.0mm  
Angle = 120deg.

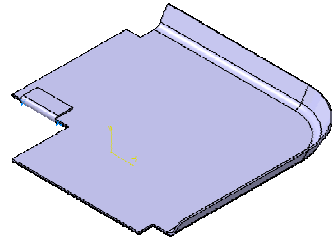


3. Create a hem as shown. Use the  
points on the edge as limits for the  
hem.

Values are:  
Radius = 1.0mm  
Length = 10.0mm

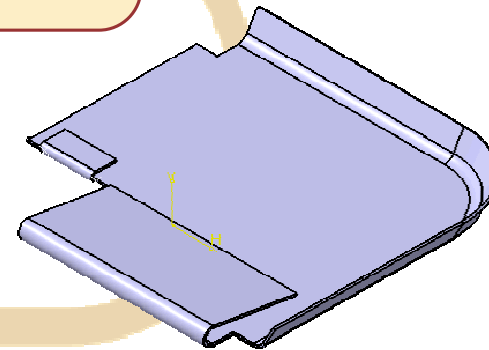


## Design Process: Practice Creating Flanges (2/2)



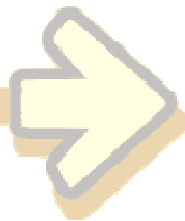
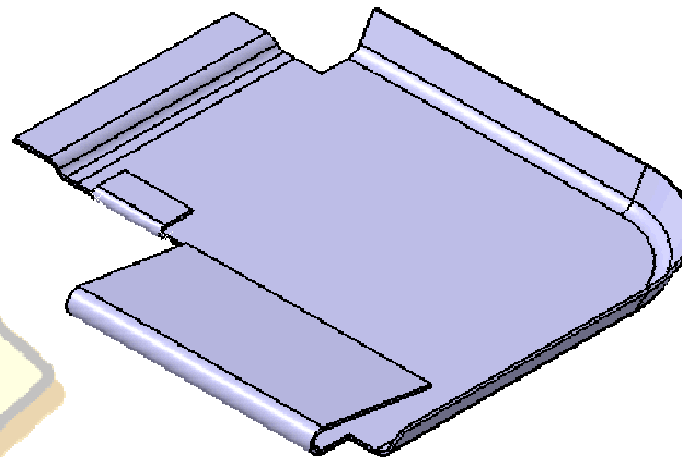
4. Create a Tear Drop flange as shown.

Values are:  
Radius = 3.0mm  
Length = 40.0mm



5. Create a User Defined Flange.  
Use "Sketch for User Flange" as the profile.

Save and close the file.



# Sheet Metal Features

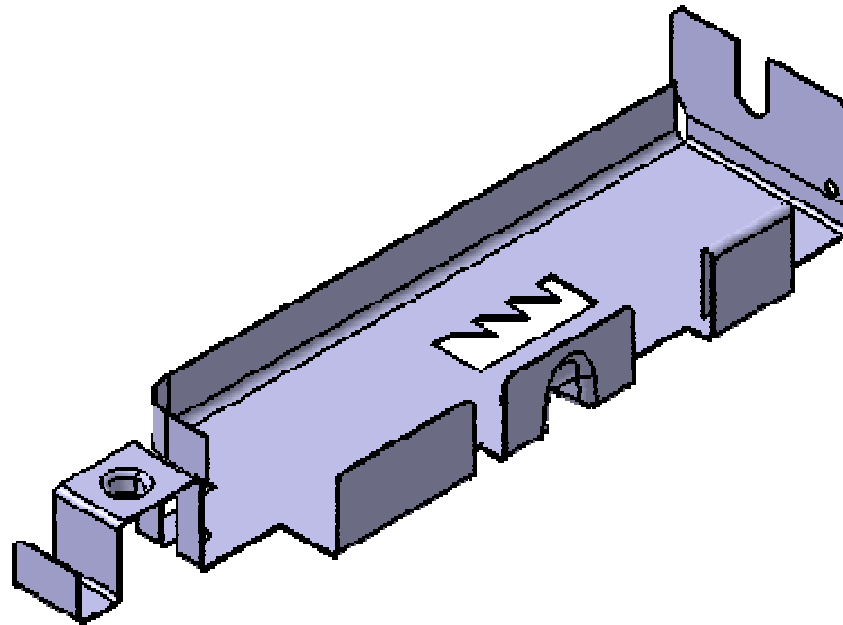
## Recap exercise



40 min

In this exercise you will create :

- A Cutout
- A Circular Cutout
- A Cutout in unfolded mode
- Corners
- Chamfers
- Flanged Hole
- Stiffening Rib



Student Notes:

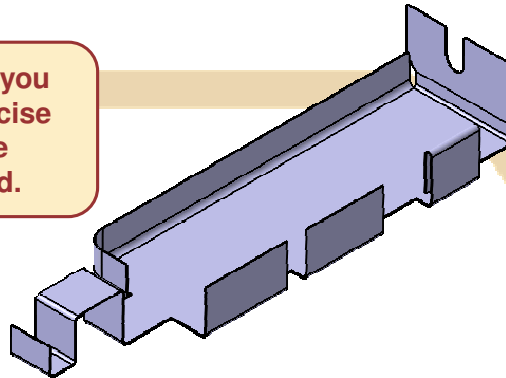
## Design Process: Sheet Metal Feature Creation (1/4)



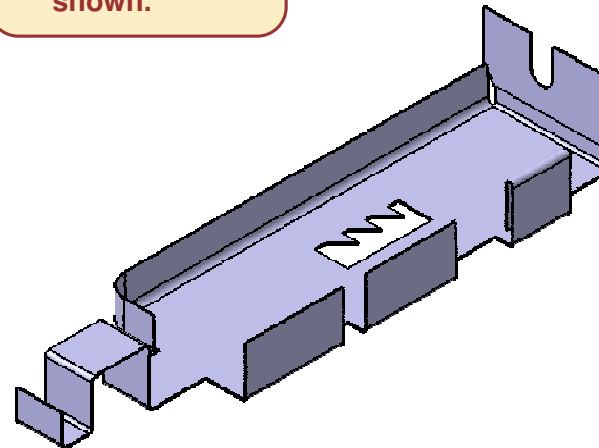
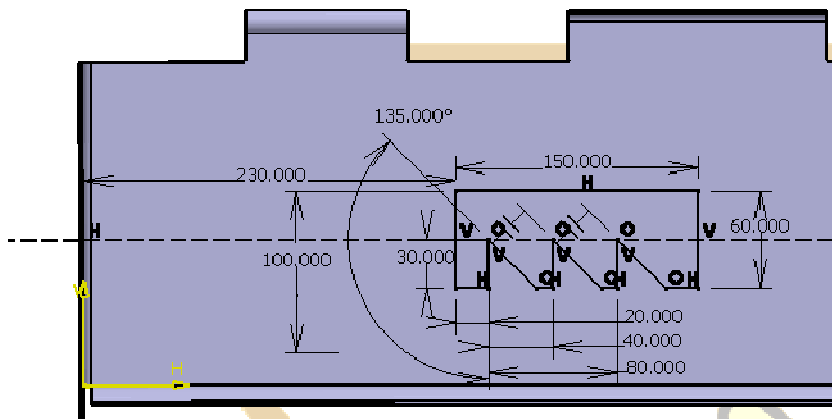
Part used: Bracket\_5.CATPart



1. Open Bracket.CATPart. If you did not complete the exercise from lesson 4 you can use bracket\_5.CATPart instead.

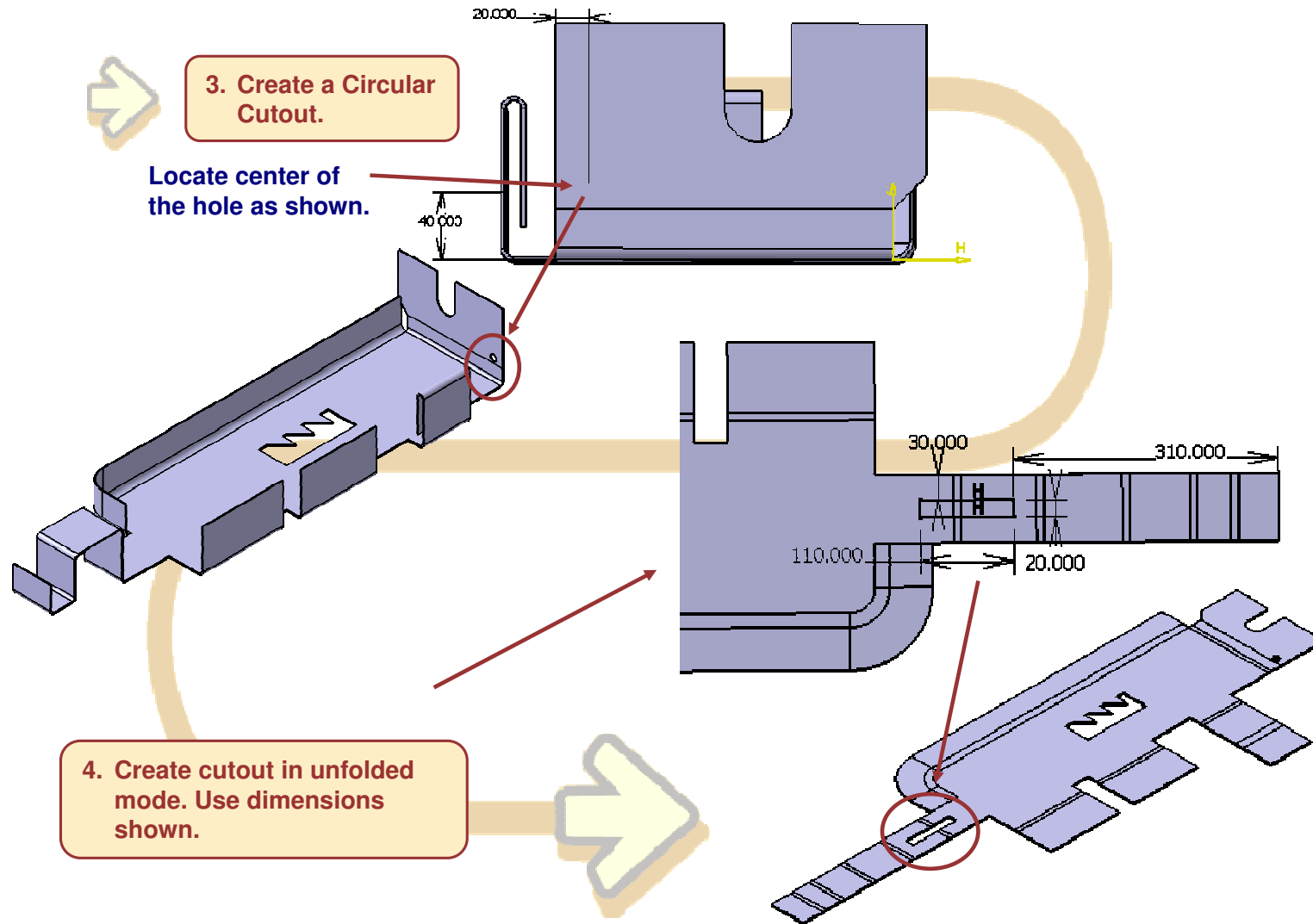


2. Create cutout using the dimensions shown.



Student Notes:

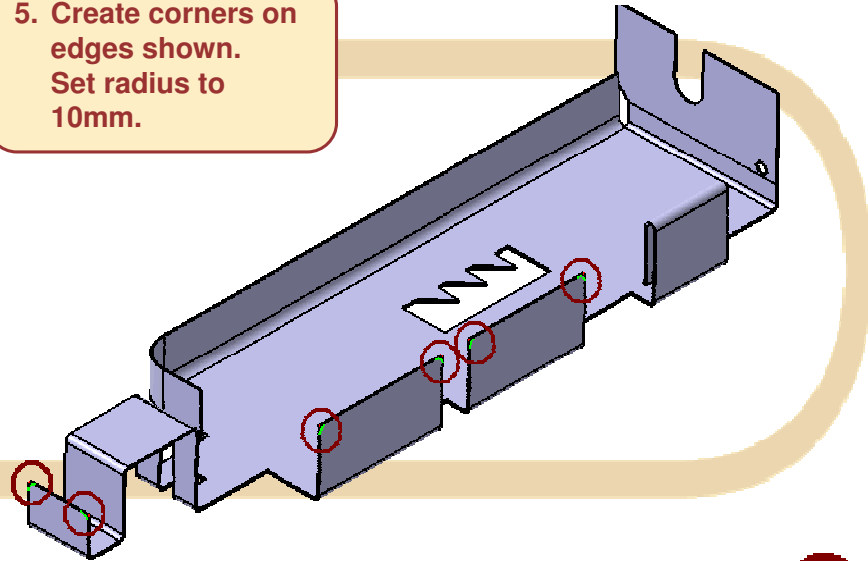
## Design Process: Sheet Metal Feature Creation (2/4)



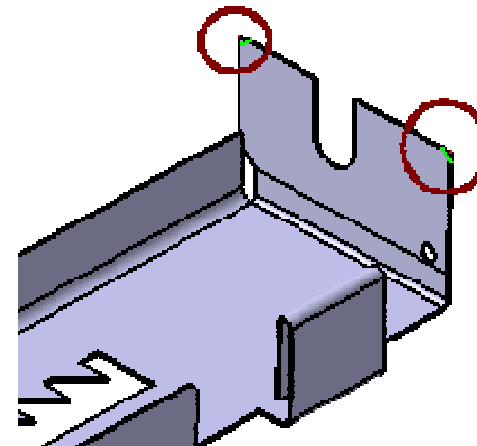
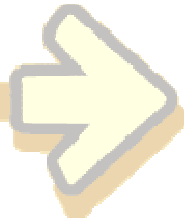
## Design Process: Sheet Metal Feature Creation (3/4)



5. Create corners on edges shown. Set radius to 10mm.



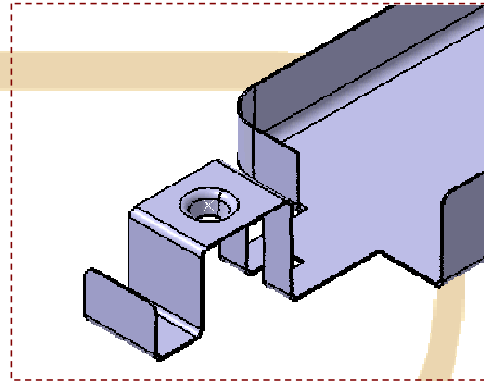
6. Create chamfers on edges shown. Create with 45 degree angle and L1 equal to 10mm.



## Design Process: Sheet Metal Feature Creation (4/4)

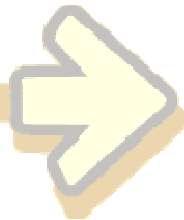
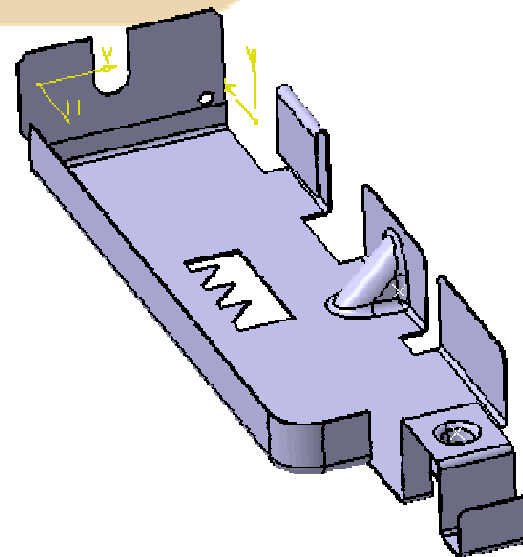


7. Create Flanged Hole.  
Use values:  
Height: 20mm  
R1: 5mm  
Angle: 80deg.  
Diameter: 35mm.



8. Create Stiffening Rib.  
Use values:  
Length: 100mm  
R1: 10mm  
R2: 20mm  
Angle: 80deg.

Save and close the file.



Student Notes:

# Transformations and Duplication

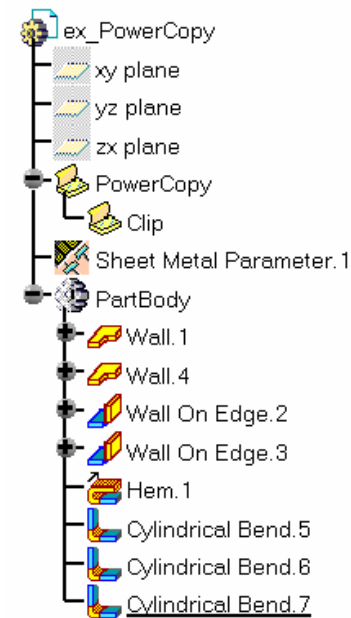
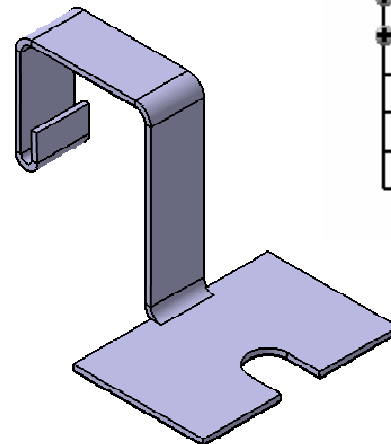
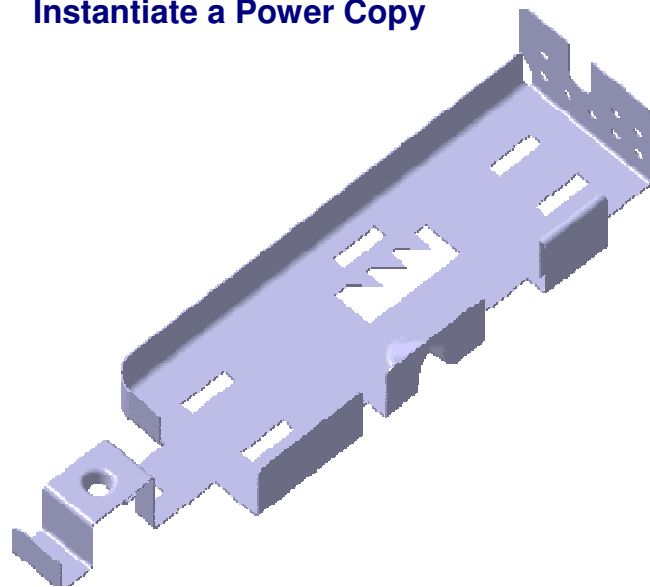
## Recap Exercises



55 min

In these exercises you will :

- Create a Rectangular Pattern
- Create a User Defined Pattern
- Create a Power Copy
- Instantiate a Power Copy





# Transformations and Duplication

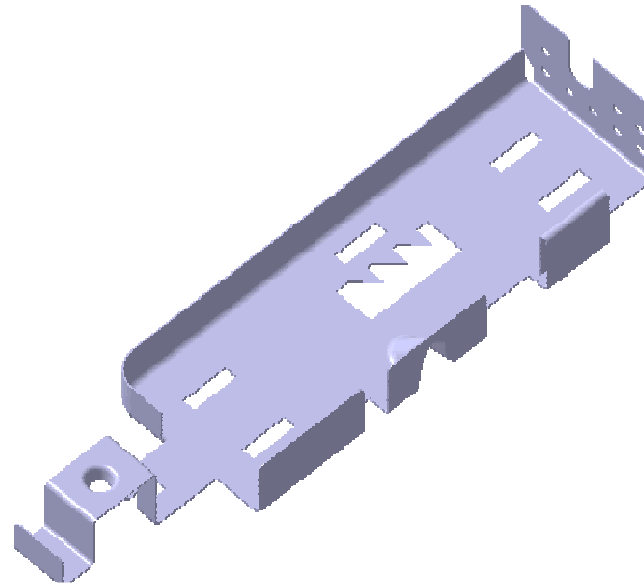
## Recap Exercise 1



20 min

In this exercise you will :

- Create a Rectangular Pattern
- Create a User Defined Pattern



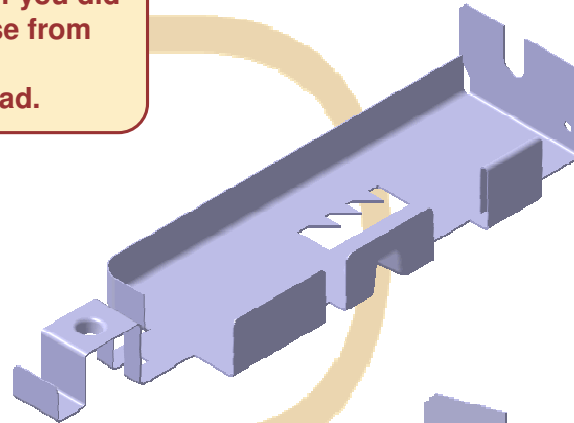
## Design Process: Duplication on Bracket (1/2)



Part used: Bracket\_6.CATPart

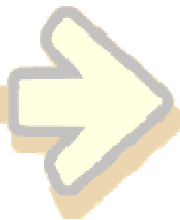
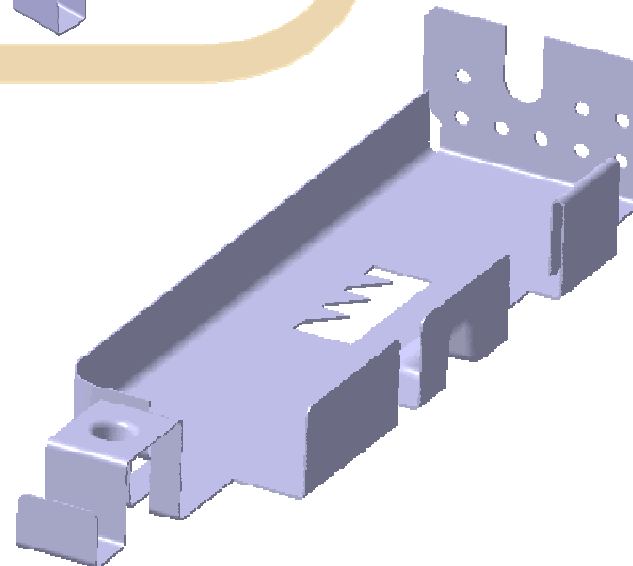


1. Open Bracket.CATPart. If you did not complete the exercise from lesson 5 you can use bracket\_6.CATPart instead.



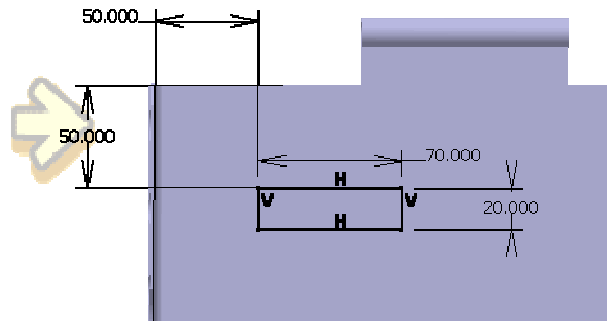
2. Create rectangular pattern. Instance spacing in both directions should be 40mm.

Remove the two instances that intersect the cutout.

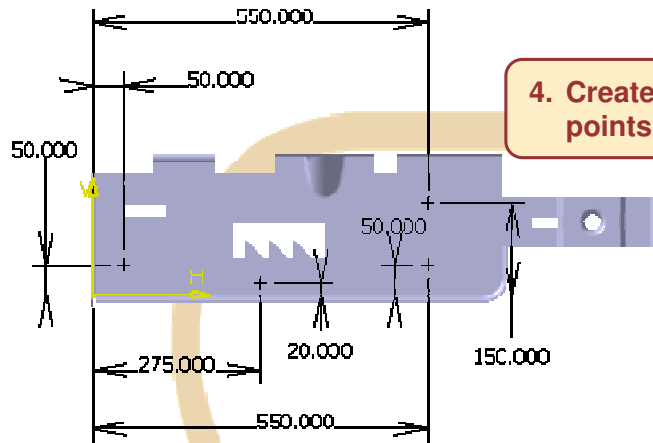
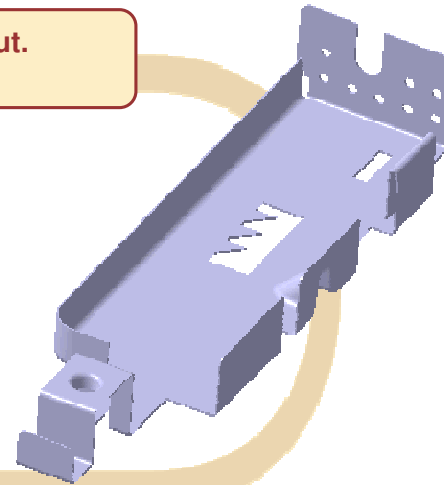


Student Notes:

## Design Process: Duplication on Bracket (2/2)

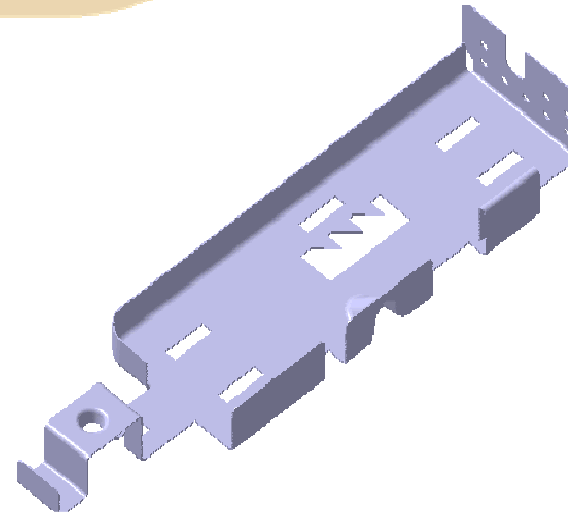
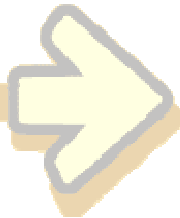


3. Create cutout.



4. Create sketch of 4 points.

5. Pattern cutout using points. Save and close the file.



# Transformations and Duplication

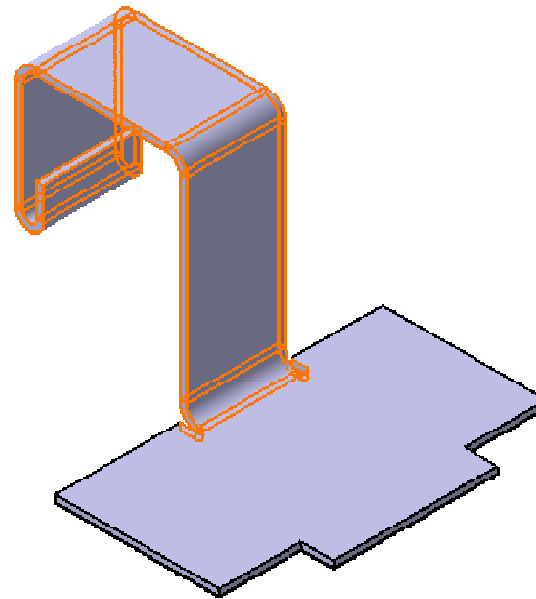
## Recap Exercise 2



20 min

In this exercise you will :

- Create a Power Copy



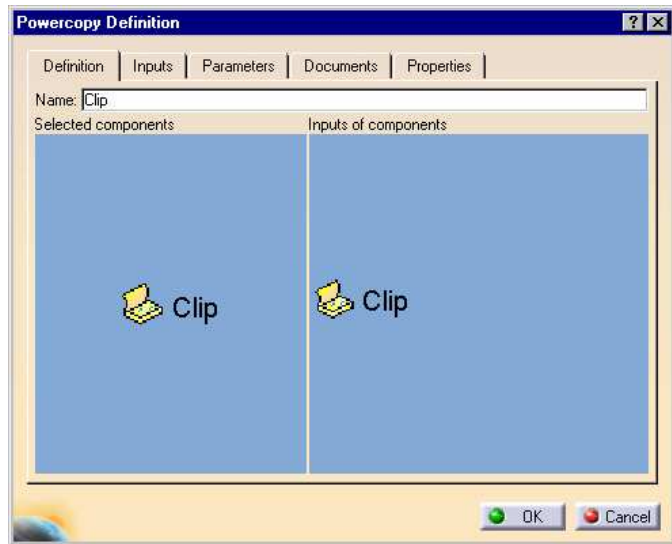
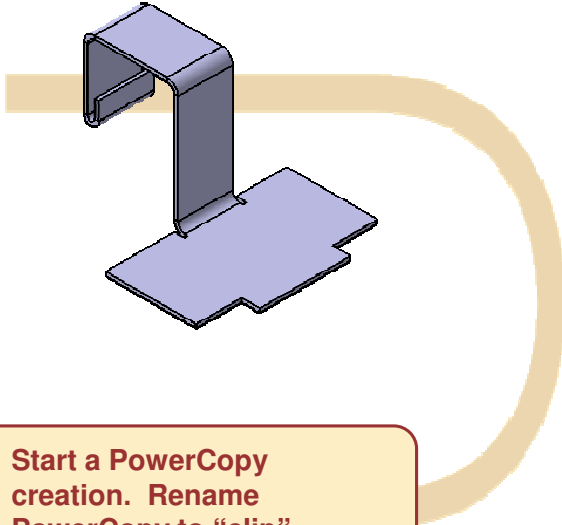
# Design Process: Create a PowerCopy (1/3)



Part used: ex\_PowerCopy.CATPart

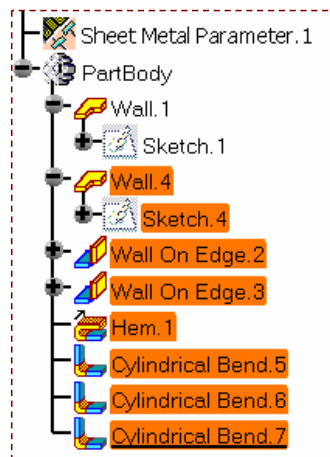
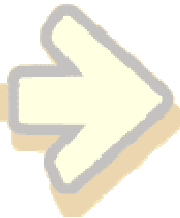


1. Open ex\_PowerCopy.CATPart.



2. Start a PowerCopy creation. Rename PowerCopy to "clip".

3. Add Features to PowerCopy.



## Design Process: Create a PowerCopy (2/3)

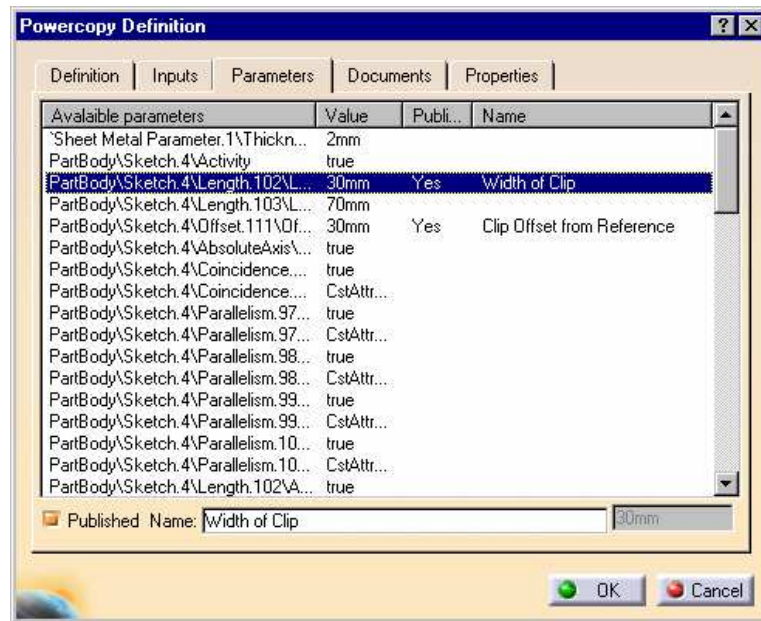
4. Rename inputs as shown.  
Note: Rename only the inputs shown, not all inputs.



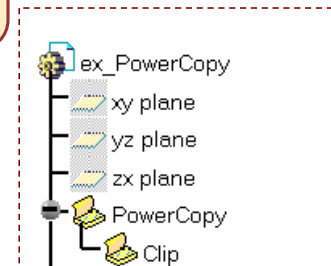
## Design Process: Create a PowerCopy (3/3)



**5. Publish parameters.**  
 Rename PartBody\Sketch.4\Length.102\Length to “Width of Clip”,  
 Rename PartBody\Sketch.4\Offset.111\Offset to “Clip Offset from  
 reference”.



**6. Close the  
 PowerCopy creation.  
 Save and close the  
 model.**



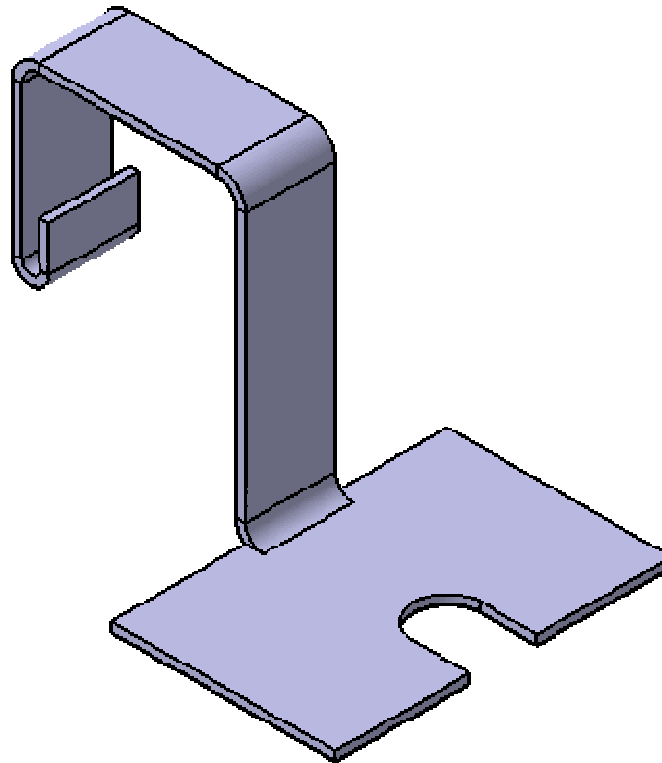
# Transformations and Duplication

## Recap Exercise 3



In this exercise you will :

- Instantiate a Power Copy





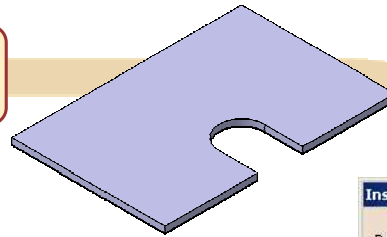
## Design Process: Instantiate a PowerCopy (1/2)



Parts used: Instantiation.CATPart, ex\_PowerCopy\_Final.CATPart

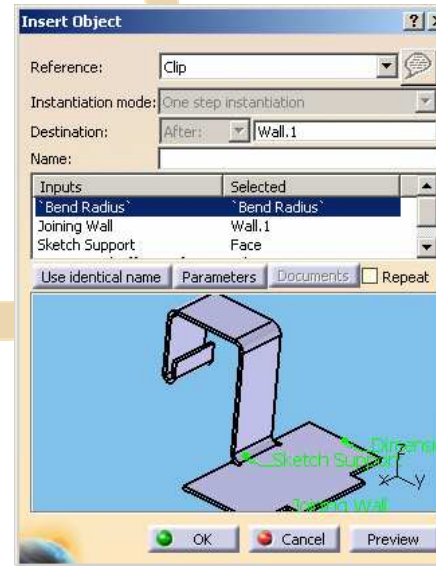


1. Open Instantiation.CATPart.

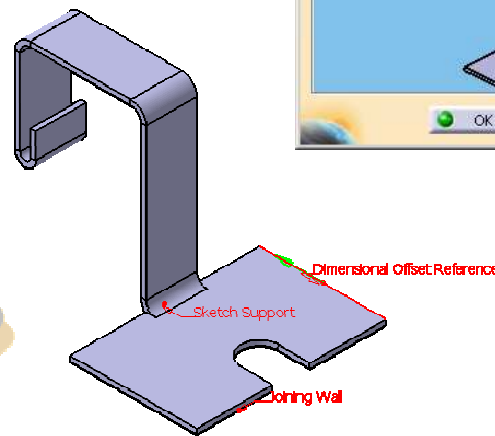
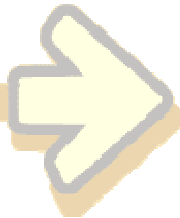


Note: You cannot instantiate a PowerCopy from an open part. If you are using ex\_PowerCopy\_Final; you must load the part, save it to your hard drive, close the part then use the saved copy to instantiate with.

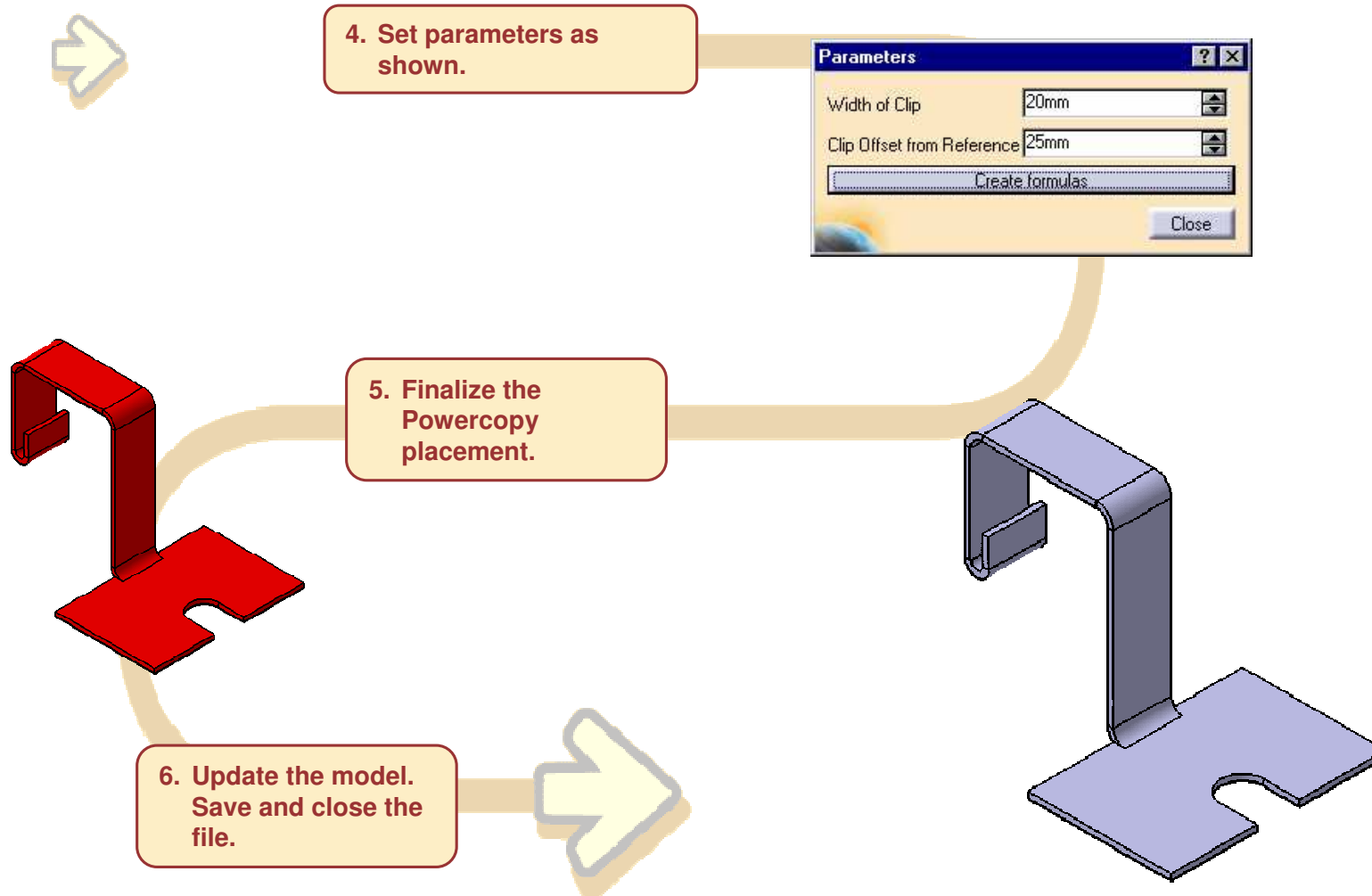
2. Instantiate from another document using the powerCopy created in the last exercise. If you did not finish the last exercise, use ex\_PowerCopy\_Final.CATPart instead.



3. Use the references as shown for placement.



## Design Process: Instantiate a PowerCopy (2/2)



Student Notes:

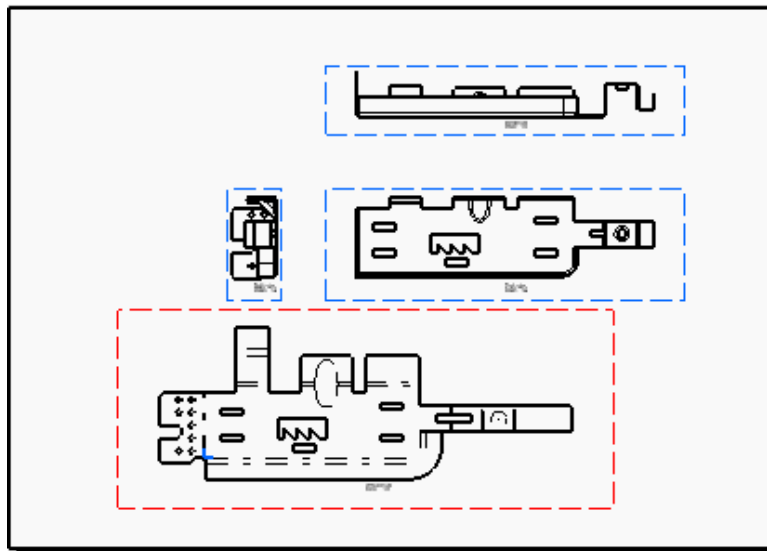
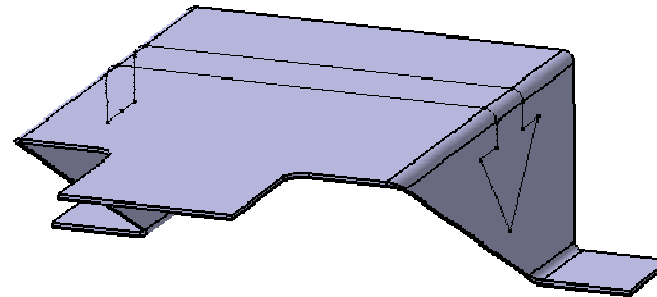
# Mapping and Output

## Recap Exercises



In these exercises you will :

- Map Curves from Folded to Unfolded mode
- Map sketch from Unfolded to Folded mode
- Create an unfolded view in the drawing



# Mapping and Output

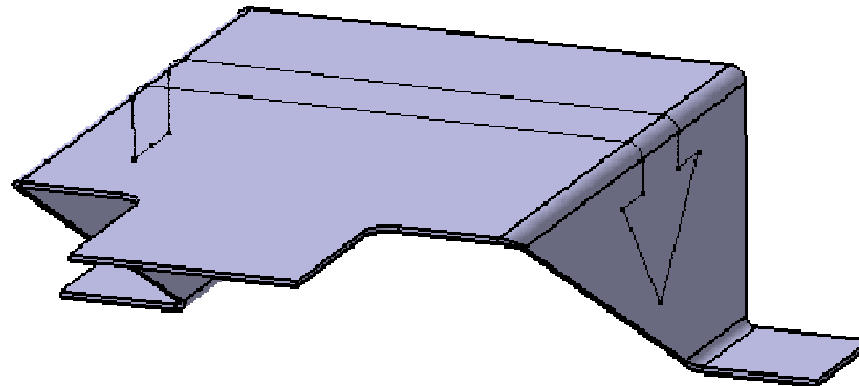
## Recap Exercise 1



20 min

In this exercise you will :

- Map Curves from Folded to Unfolded mode
- Create Cutouts from mapped curves



This exercise can be replayed in 'P2 configuration only'

## Design Process: Mapping Curves (1/2)



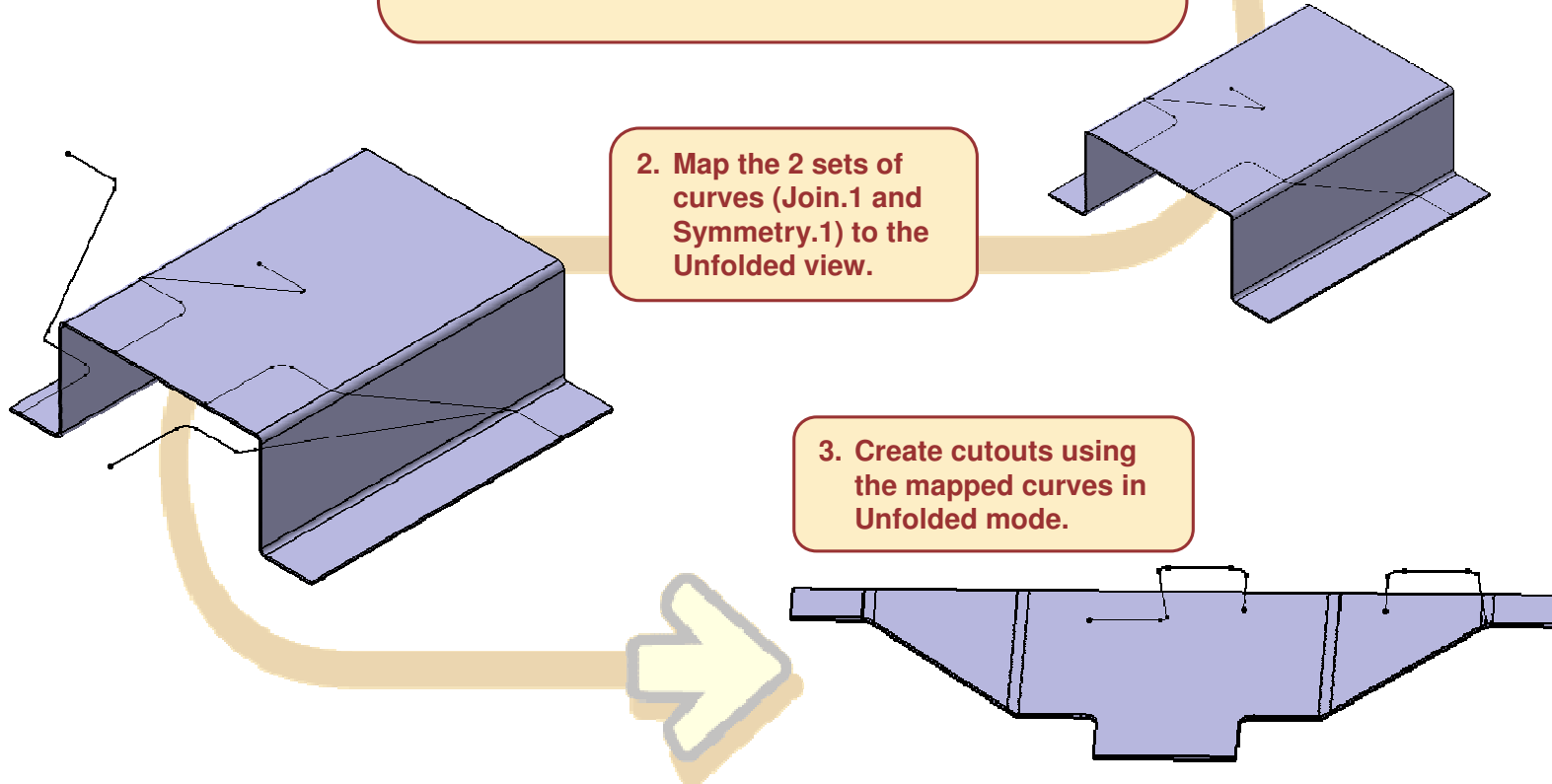
Part used: CATSMD\_Contour\_Mapping.CATPart



1. Open CATSMD\_Contour\_Mapping.CATPart. Note curves are created on the folded model. These curves were created in the Wireframe and Surfacing Workbench and now need to be used to create cutouts on the sheet metal model.

2. Map the 2 sets of curves (Join.1 and Symmetry.1) to the Unfolded view.

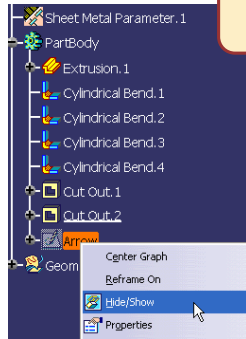
3. Create cutouts using the mapped curves in Unfolded mode.



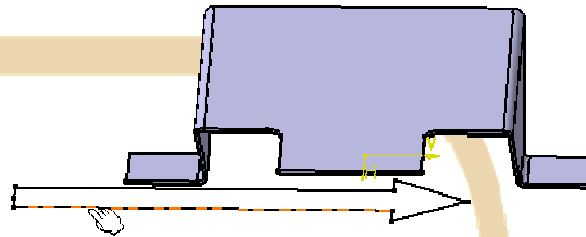
## Design Process: Mapping Curves (2/2)



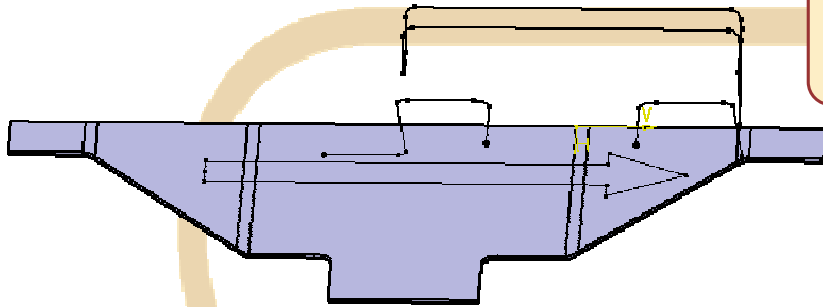
You can compare your result with "CATSMD\_Contour\_Mapping\_end.CATPart".



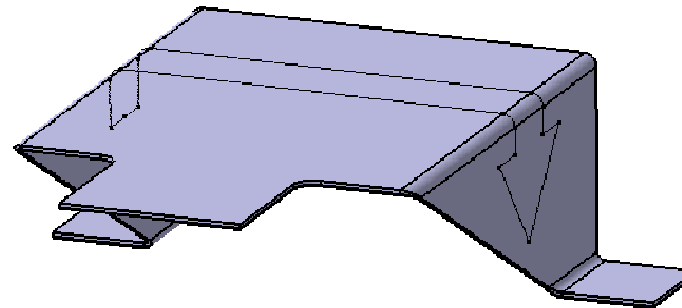
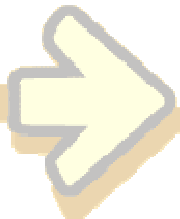
4. Unhide the Sketch Arrow.



5. Map the Arrow sketch onto the Folded view.



6. Return to folded view. Re-hide the Arrow Sketch. Save and close the model.



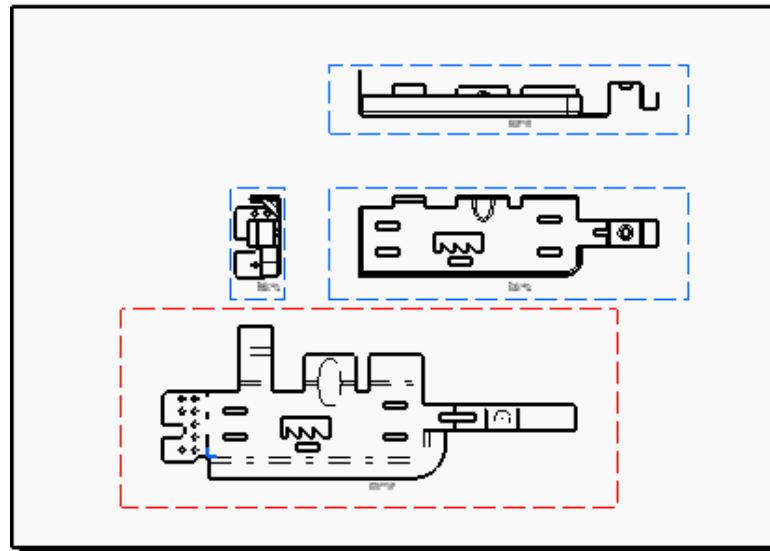
# Mapping and Output

## Recap Exercise 2



In this exercise you will :

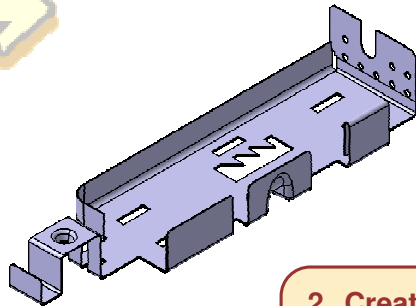
- Create an Unfolded Drawing view



## Design Process: Create an Unfolded View (1/2)

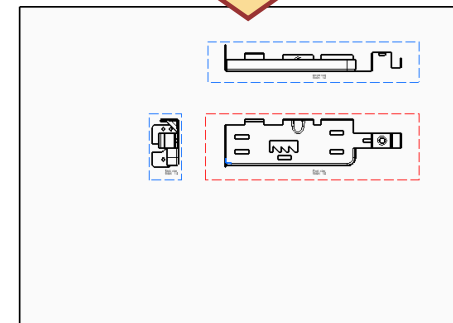
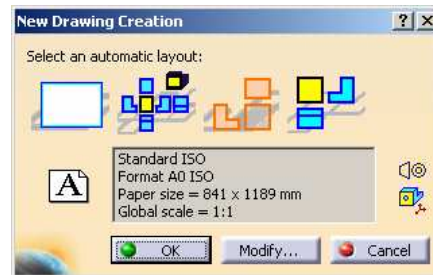
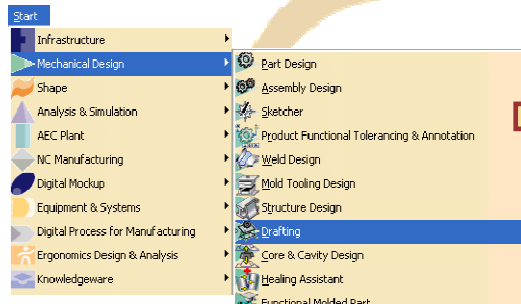


Part used: Bracket\_8.CATPart



1. Open Bracket.CATPart. If you did not complete the exercise from lesson 6 you can use bracket\_8.CATPart instead.

2. Create a drawing. Create the 3 main views (Front, bottom and right) using the view wizard. Use ISO standard and a page size of A0 ISO.





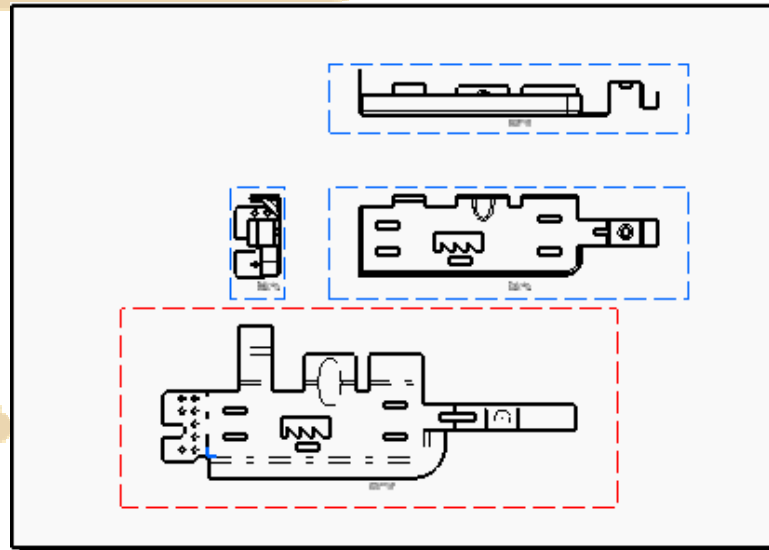
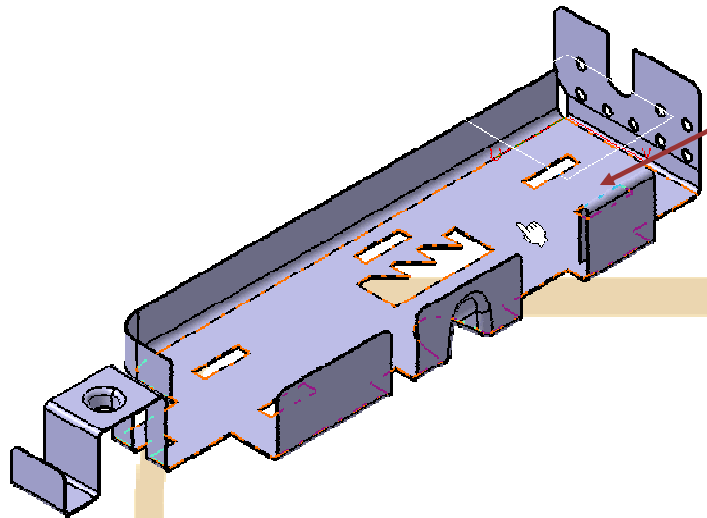
Student Notes:

## Design Process: Create an Unfolded View (2/2)



3. Create Unfolded view. Save and close the drawing. Name the drawing Bracket.CATDrawing.

Use this surface as orientation reference for the view.



# Multi-Body Cover

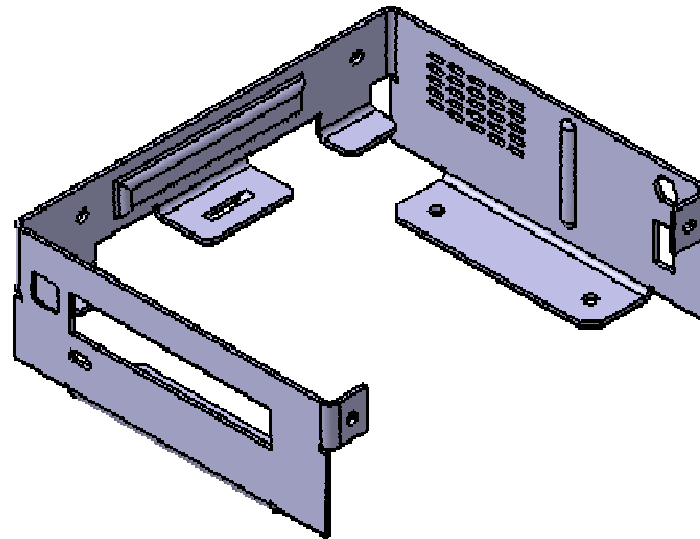
## Recap Exercise



In this exercise you will use multi-body methodology to design a sheet metal cover.

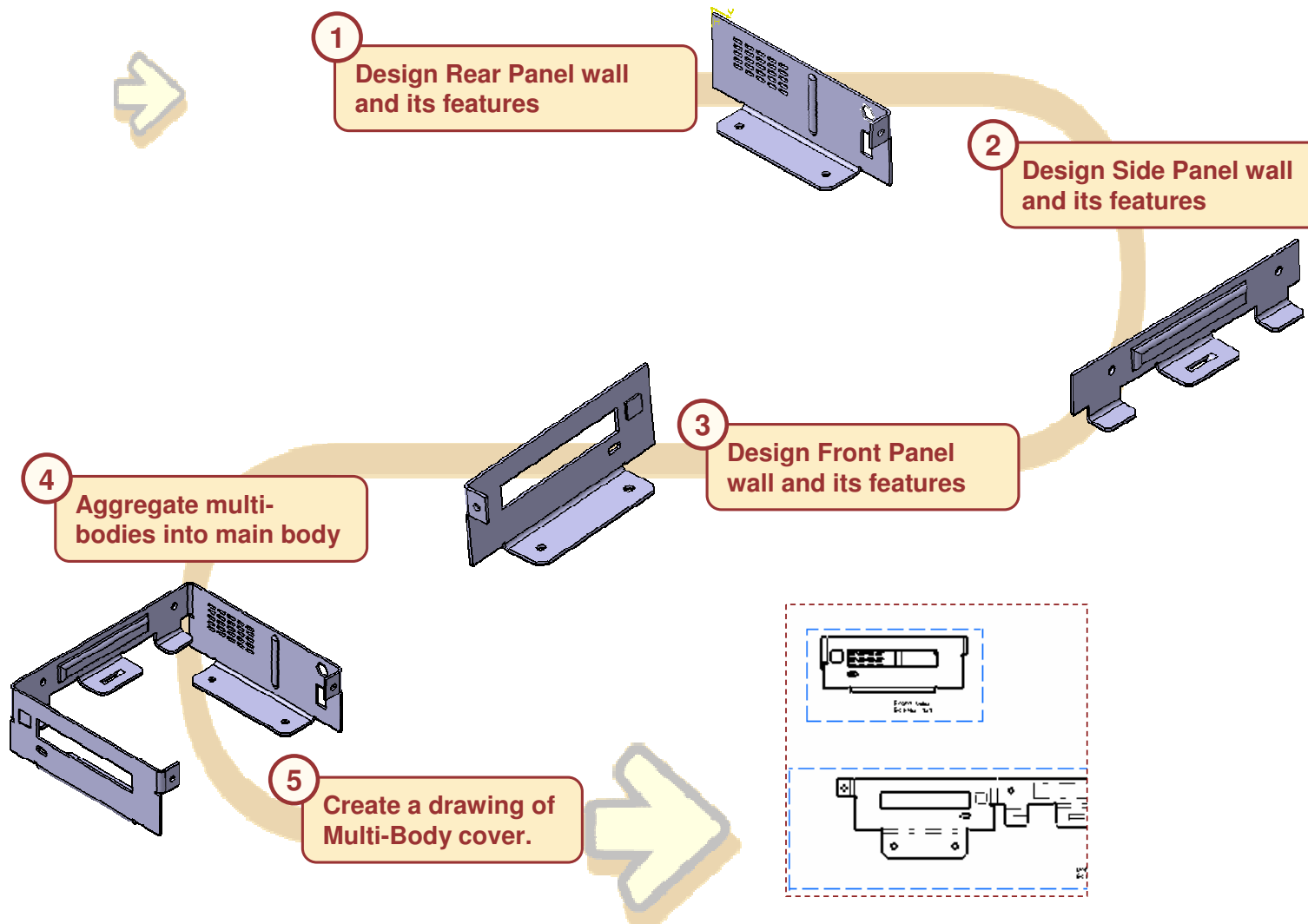
You will design various sheetmetal walls and its features in various bodies as given below :

- Front Panel
- Side Panel
- Rear Panel



Student Notes:

# Design Process: Multi-Body Cover



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Student Notes:

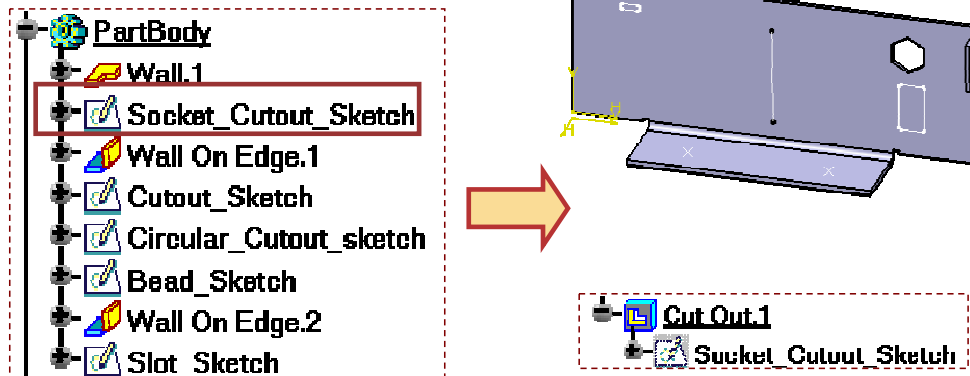
## Step 1 : Design Rear Panel (1/4)



Part used: CATSMD\_MultiBodyCoverStep1.CATPart

In this step, you will complete the design of 'Rear Panel'.

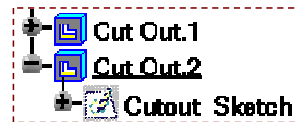
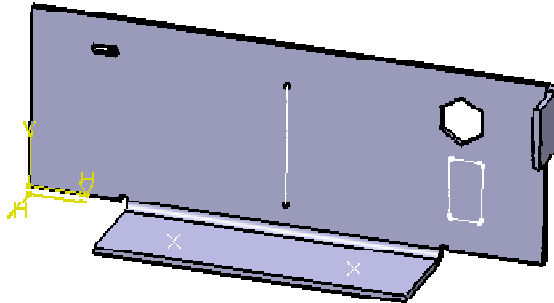
- Activate 'PartBody' and hide the 'SidePanel' and 'FrontPanel' bodies.
- Create Socket cutout in 'Rear Panel' using the existing sketch 'Socket\_Cutout\_Sketch'.



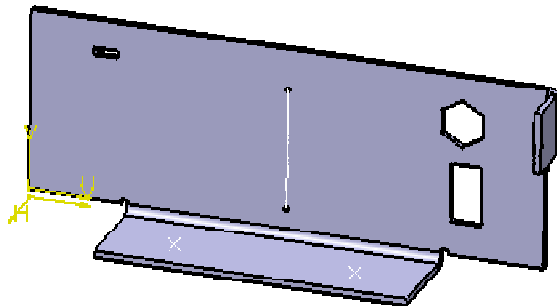
Student Notes:

## Step 1 : Design Rear Panel (2/4)

- Create another cutout using the existing sketch 'Cutout\_Sketch'.

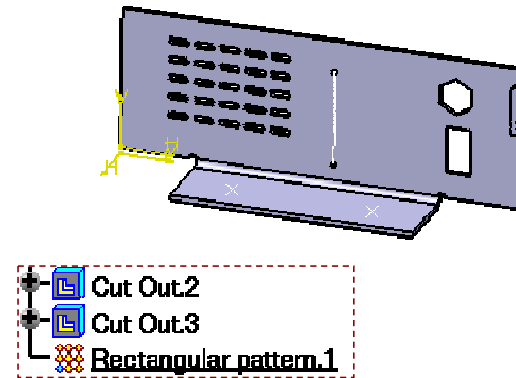
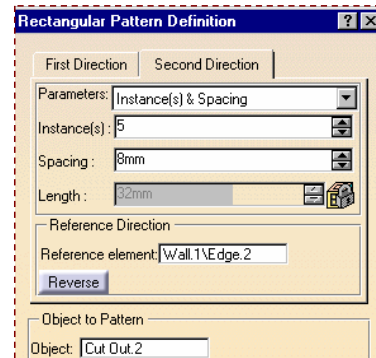
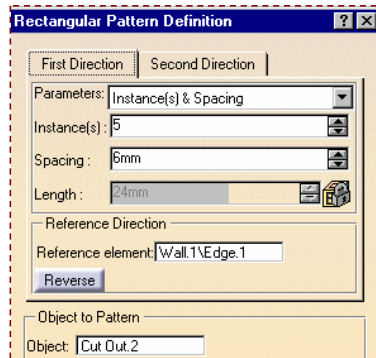


- Create another cutout using the existing sketch 'Slot\_Sketch'.

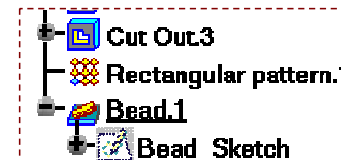
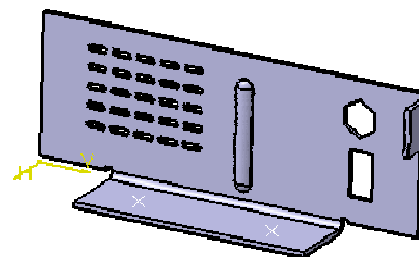
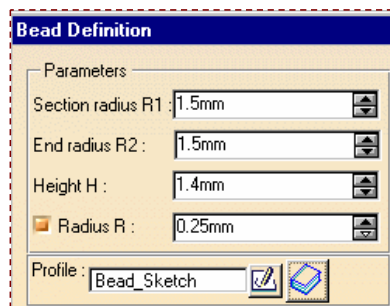


## Step 1 : Design Rear Panel (3/4)

- Create a rectangular pattern of 'Cut Out.2' with parameters as shown below.

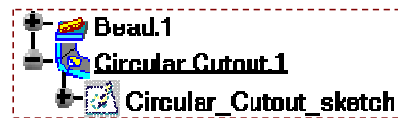
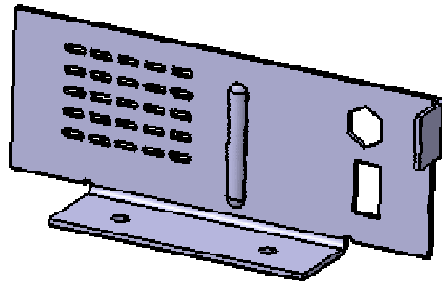


- Create a 'Bead' using 'Bead\_sketch' and parameters as shown below.

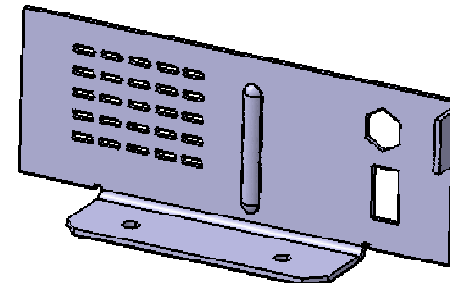


## Step 1 : Design Rear Panel (4/4)

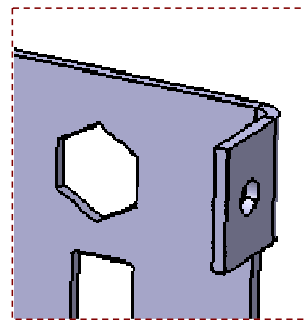
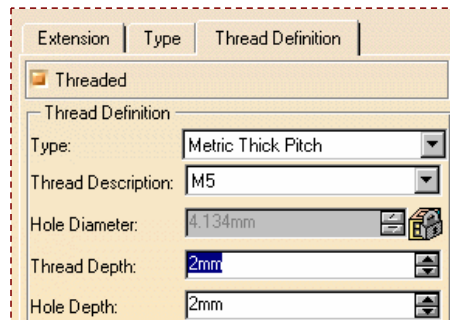
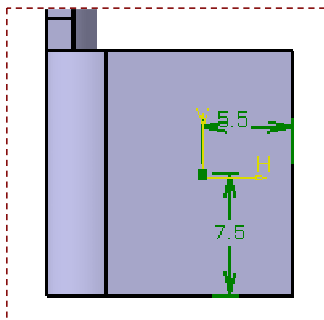
- Create a 'Circular Cutout' of diameter 4.5mm, using 'Circular\_Cutout\_sketch'



- Create a 5mm x 45° chamfer on edges of 'Wall on Edge'



- Create a M5 x 2mm threaded hole on 'Wall on Edge.2' and position the sketch as shown.



## Step 2 : Design Side Panel (1/3)

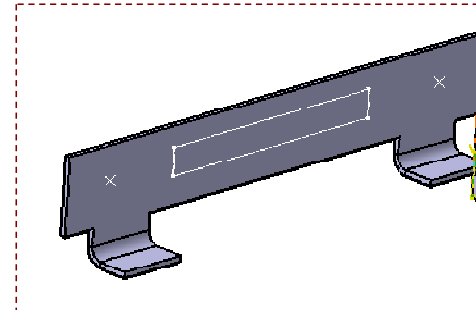
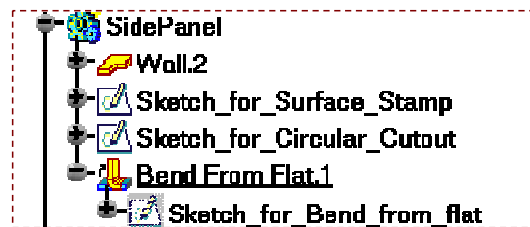
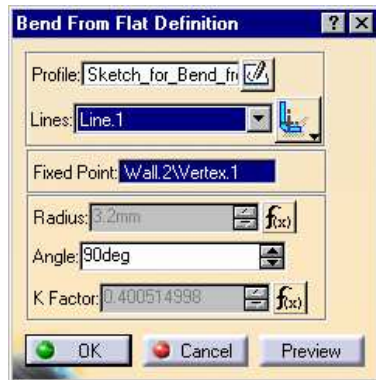


Part used: CATSMD\_MultiBodyCoverStep2.CATPart

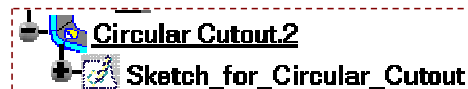
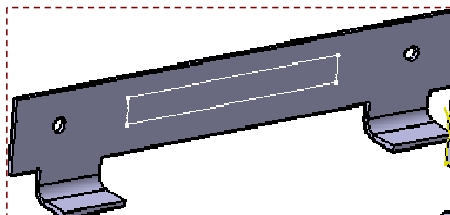
In this step, you will complete the design of 'Side Panel'.

• Unhide and activate 'SidePanel' and hide the 'PartBody'.

• Create a 'Bend From Flat' using the profile 'Sketch\_for\_Bend\_from\_flat'



• Create a 'Circular Cutout' of diameter 4.5mm, using 'Sketch\_for\_Circular\_Cutout'

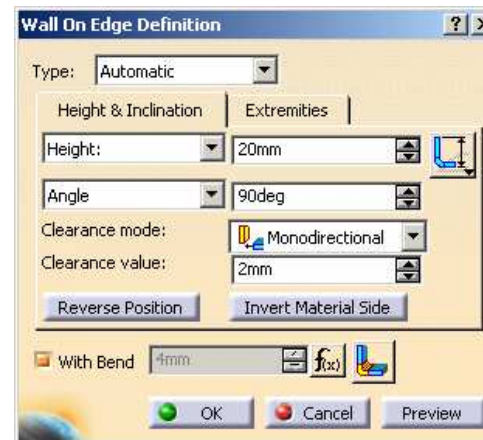
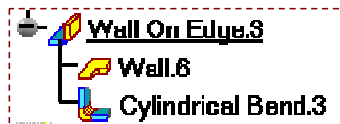
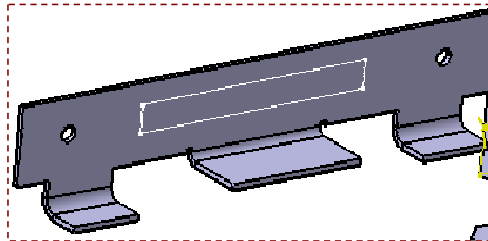




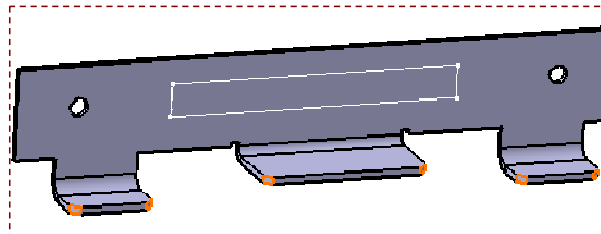
## Step 2 : Design Side Panel (2/3)

• Create a 'Wall On Edge' as shown with following parameters

- ◆ Height = 20mm
- ◆ Left limit offset = 25mm
- ◆ Right limit offset = 25mm
- ◆ With Clearance
- ◆ With Bend

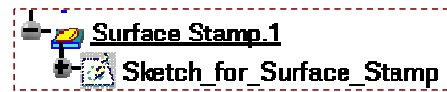
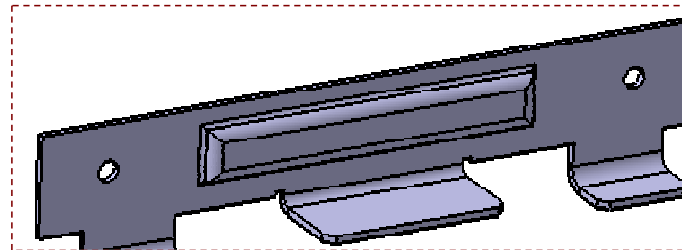
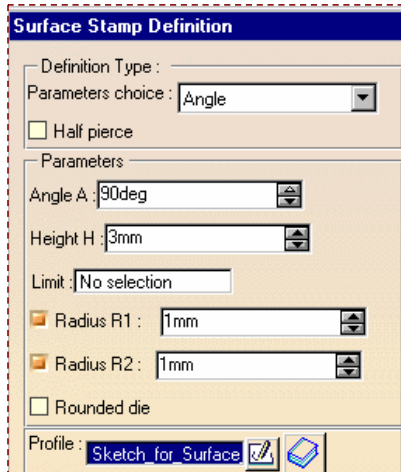


• Create a 5mm x 45° chamfer on edges as shown

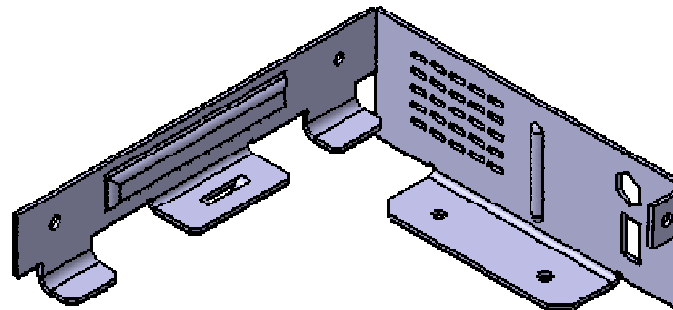
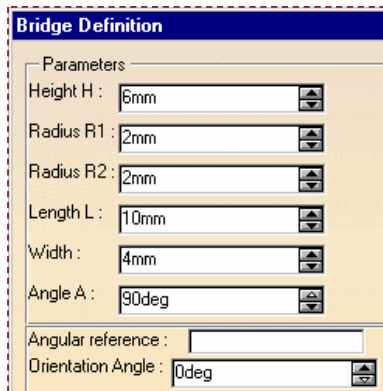


## Step 2 : Design Side Panel (3/3)

- Create a 'Surface Stamp' with parameters as shown in the panel. 



- Create a 'Bridge' and position it as shown on the wall on edge. 



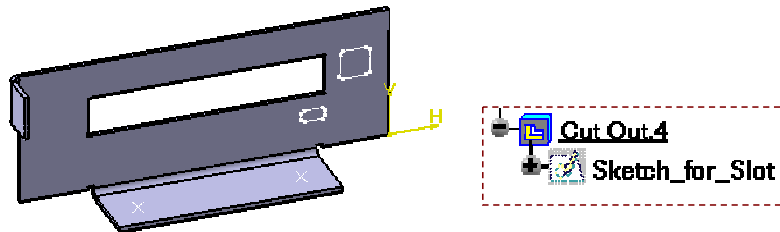
### Step 3 : Design Front Panel (1/3)

 Part used: CATSMD\_MultiBodyCoverStep3.CATPart

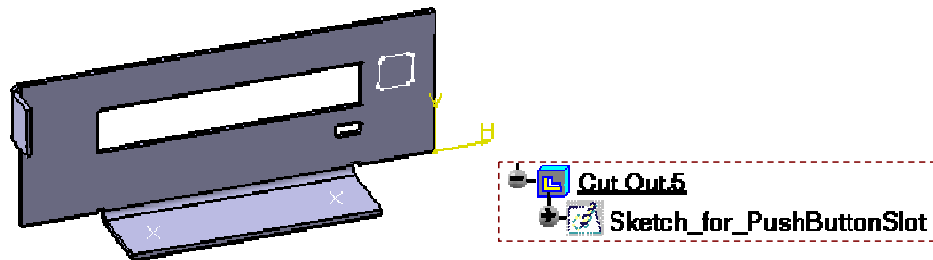
In this step, you will complete the design of 'Front Panel'.

• Unhide and activate 'Front Panel' and hide the 'SidePanel' body.

• Create a Cutout using 'Sketch\_for\_Slot'.



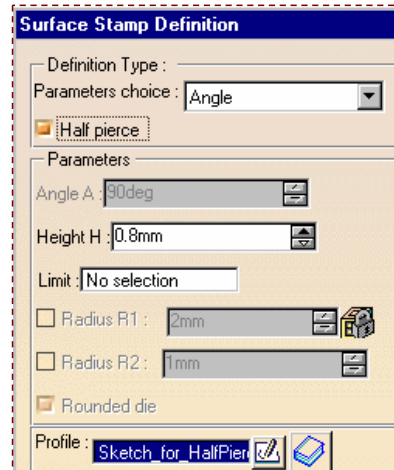
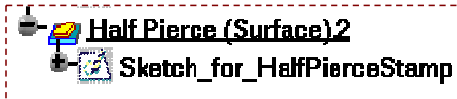
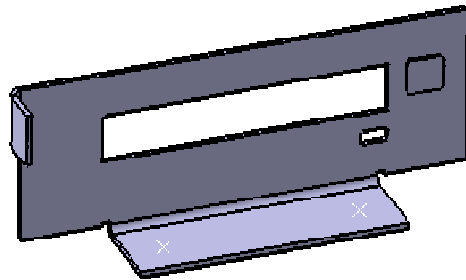
• Create another Cutout using 'Sketch\_for\_PushButtonSlot'.



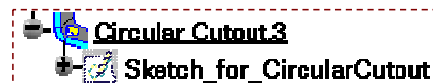
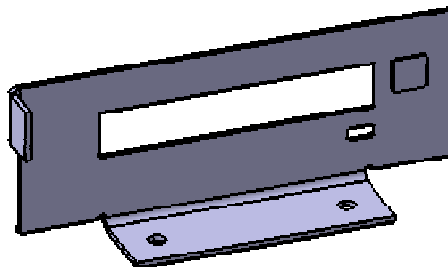
Student Notes:

### Step 3 : Design Front Panel (2/3)

- Create a 'Half Pierce Surface Stamp' using 'Sketch\_for\_HalfPierceStamp' and parameters as shown.



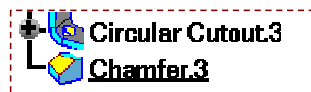
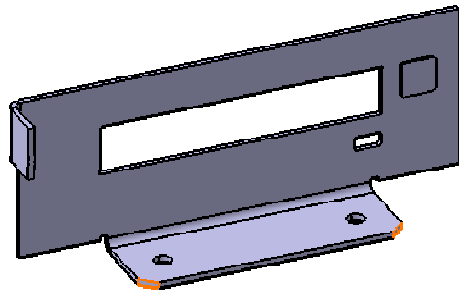
- Create a 'Circular Cutout' of diameter 5.0 mm, using 'Sketch\_for\_Circular\_Cutout'



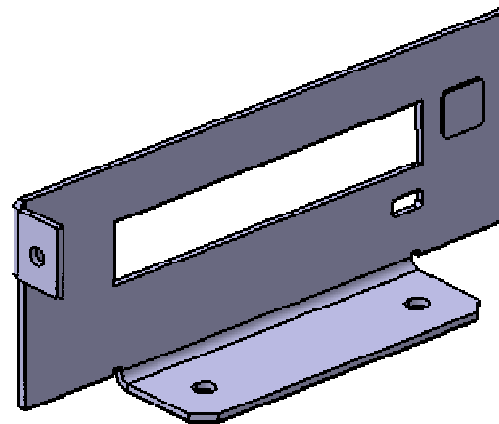
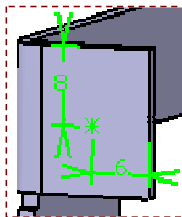
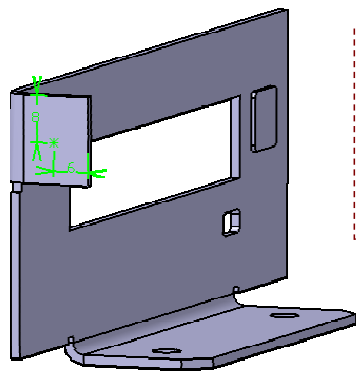
Student Notes:

### Step 3 : Design Front Panel (3/3)

- Create a 4mm x 45° chamfer on edges of as shown



- Create a M5 x 2mm threaded hole on the wall and position the sketch as shown.



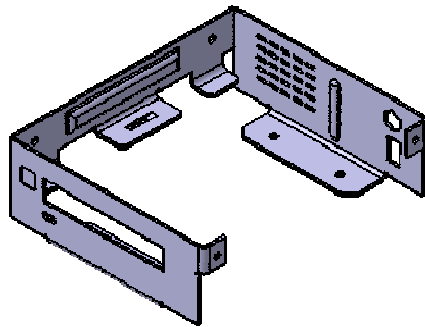
## Step 4 : Aggregate Multi-Bodies using Bend (1/3)



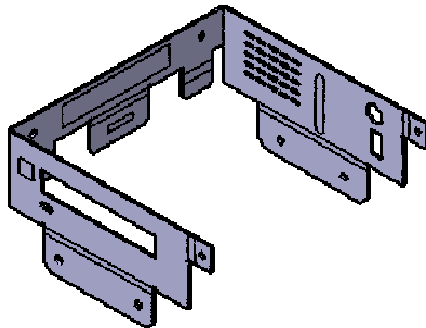
Part used: CATSMD\_MultiBodyCoverStep4.CATPart

In this step, you will aggregate the multi-bodies into a single body.

- Unhide and activate 'PartBody' and unhide the 'SidePanel' body.



- Unfold the part using 'Fold /Unfold' function.

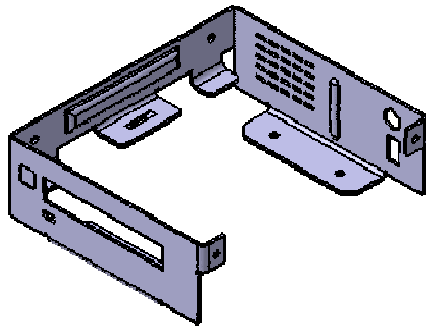


Only bends in individual walls are unfolded.

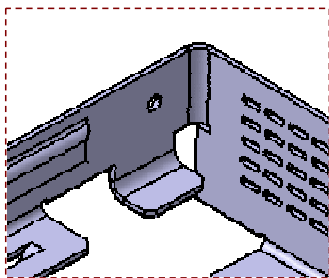
These walls belong to separate bodies still.

## Step 4 : Aggregate Multi-Bodies using Bend (2/3)

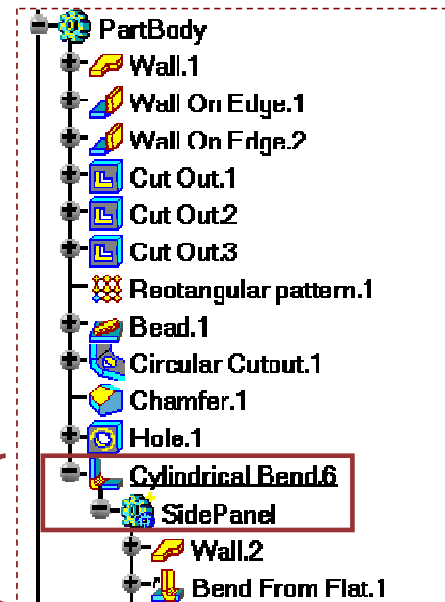
- ◆ Fold the part using 'Fold /Unfold' function.



- ◆ Create a 'Cylindrical Bend' by selecting following supports
  - ◆ 'Wall.1' in PartBody
  - ◆ 'Wall.2' in SidePanel body

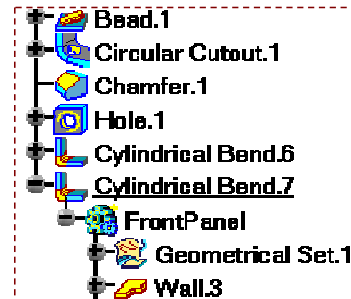
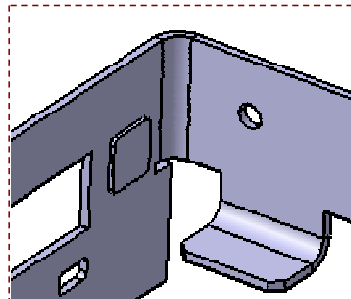
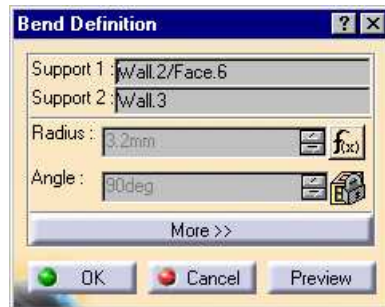


'SidePanel' body is aggregated under 'Cylindrical Bend' feature inside 'PartBody'.

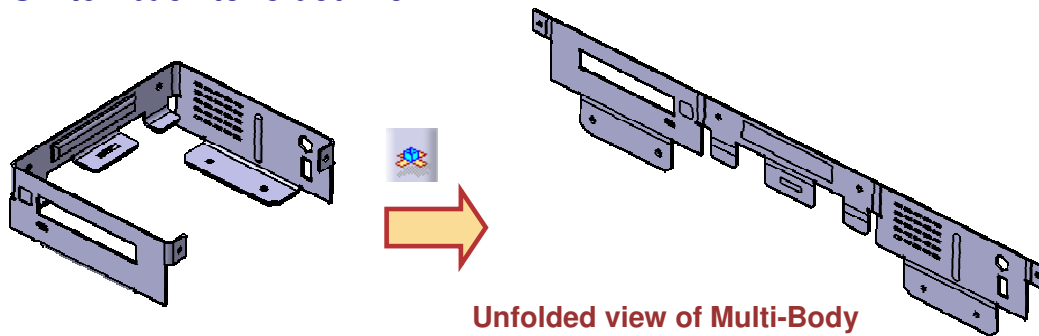


## Step 4 : Aggregate Multi-Bodies using Bend (3/3)

- Activate 'PartBody' again.
- Create another 'Cylindrical Bend' by selecting following supports
  - ◆ 'Wall.2' in SidePanel body
  - ◆ 'Wall.3' in FrontPanel body



- **Unfold the part using 'Fold /Unfold' function to see the unfolded view. Switch back to folded view.**



Unfolded view of Multi-Body Cover after aggregation



Student Notes:

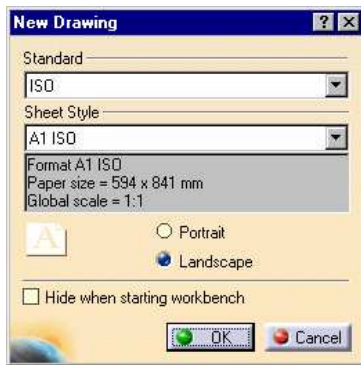
## Step 5 : Multi-Body Cover Drawing



Part used: CATSMD\_MultiBodyCoverStep5.CATPart

In this step, you will create a drawing of the 'Multi-Body Cover'.

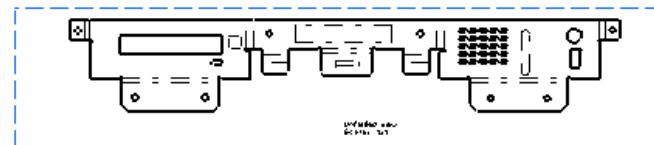
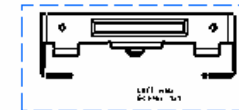
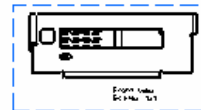
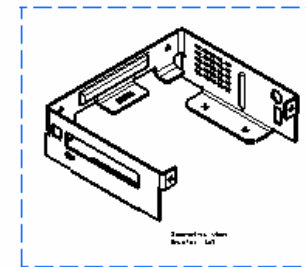
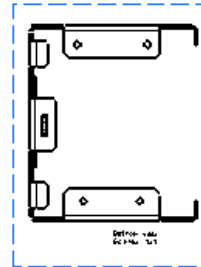
- Create a new drawing with parameters as shown.



- Create the following views in the drawing:

- ◆ Front View
- ◆ Bottom View
- ◆ Left View
- ◆ Isometric View
- ◆ Unfolded View

- Save the drawing as 'CATSMD\_MultiBodyCover\_Drawing.CATDrawing.'



# Master Exercise

*In this exercise you will put the skills you have learned in this course together to create an assembly consisting of 2 sheet metal parts.*

- ▣ Casing Assembly
- ▣ Casing Assembly (1): Wall Creation
- ▣ Casing Assembly (2): Bend and Flange Creation
- ▣ Casing Assembly (3): Cutout, Circular Cutout and Stamp Creation
- ▣ Casing Assembly (4): Pattern Creation
- ▣ Casing Assembly (5): Unfolded View Creation
- ▣ Casing Assembly Recap

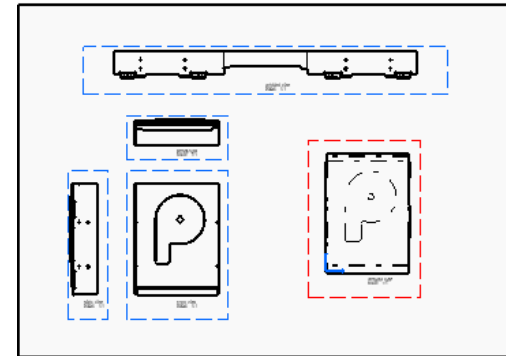
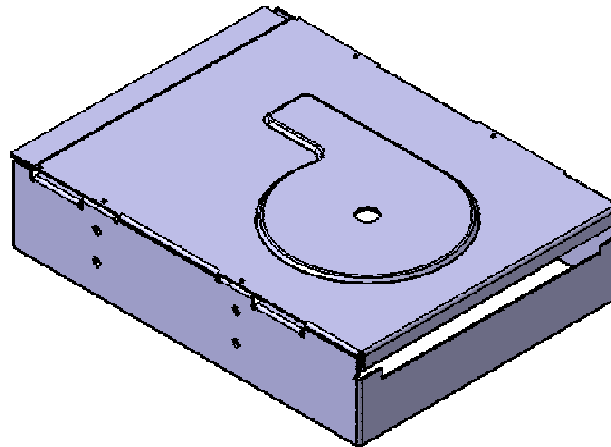
# Casing Assembly

## Master Exercise



In this exercise you will create :

- Walls
- Bends
- Flanges
- Cutouts
- Circular Cutouts
- Stamps
- Patterns
- Unfolded View



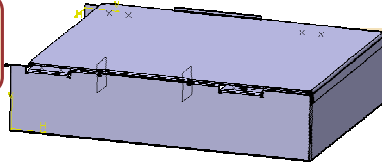
Student Notes:

Student Notes:

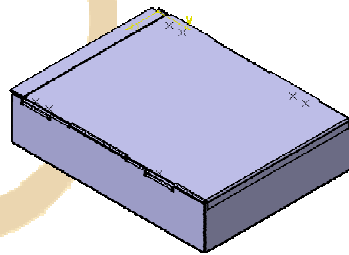
# Design Process: Casing Assembly



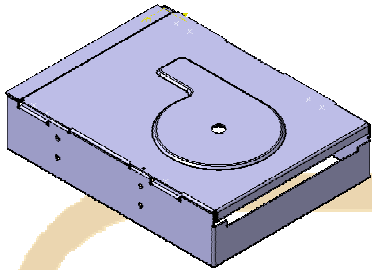
1. Create Walls.



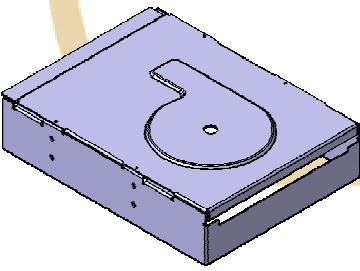
2. Create Bends, add Flanges.



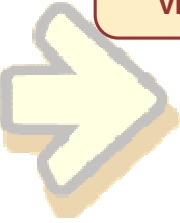
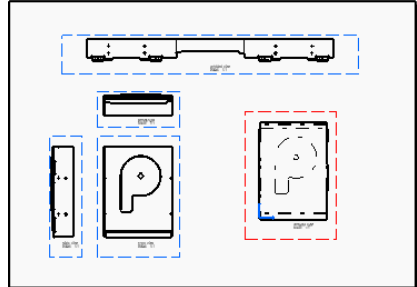
3. Create Cutouts, Circular Cutouts, and Stamp.



4. Create Pattern Cutout and Circular Cutout.



5. Create drawing with Unfolded views.



Student Notes:

# Casing Assembly

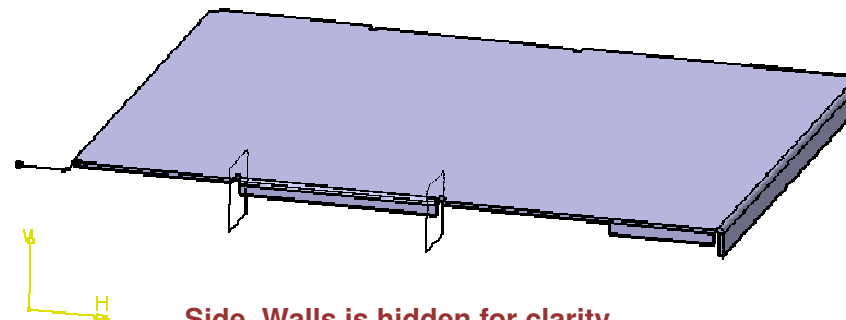
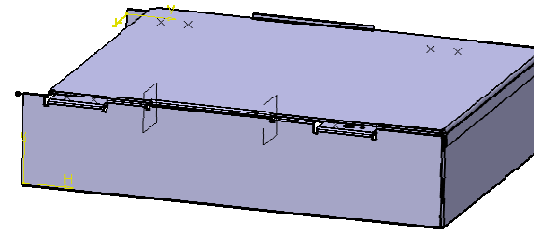
## Master Exercise Step 1



20 min

In this exercise you will create :

- Profile Wall
- Edge Walls



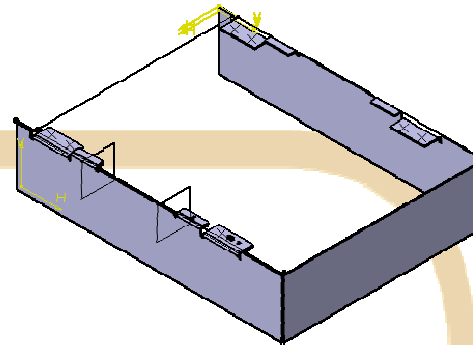
# Design Process: Wall Creation (1/4)



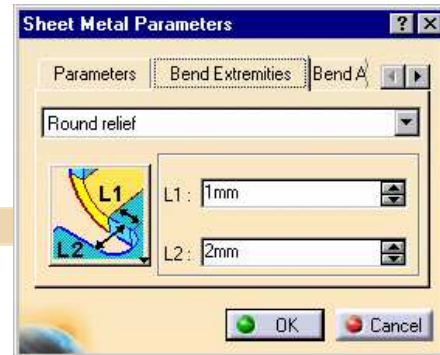
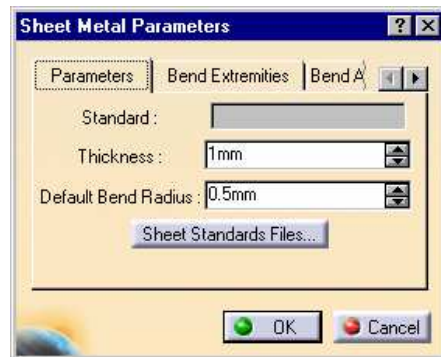
Part used: Casing.CATProduct



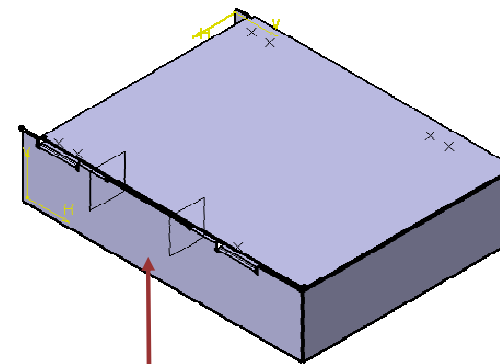
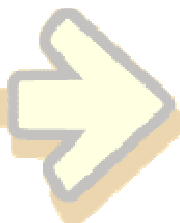
1. Open Casing.CATProduct.



2. Activate the Top\_Cover part. Set Sheet Metal Parameters as shown.



3. Create Profile wall using Sketch.1 as the profile. Ensure material is added above the profile.

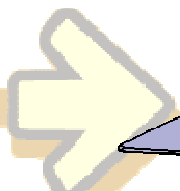
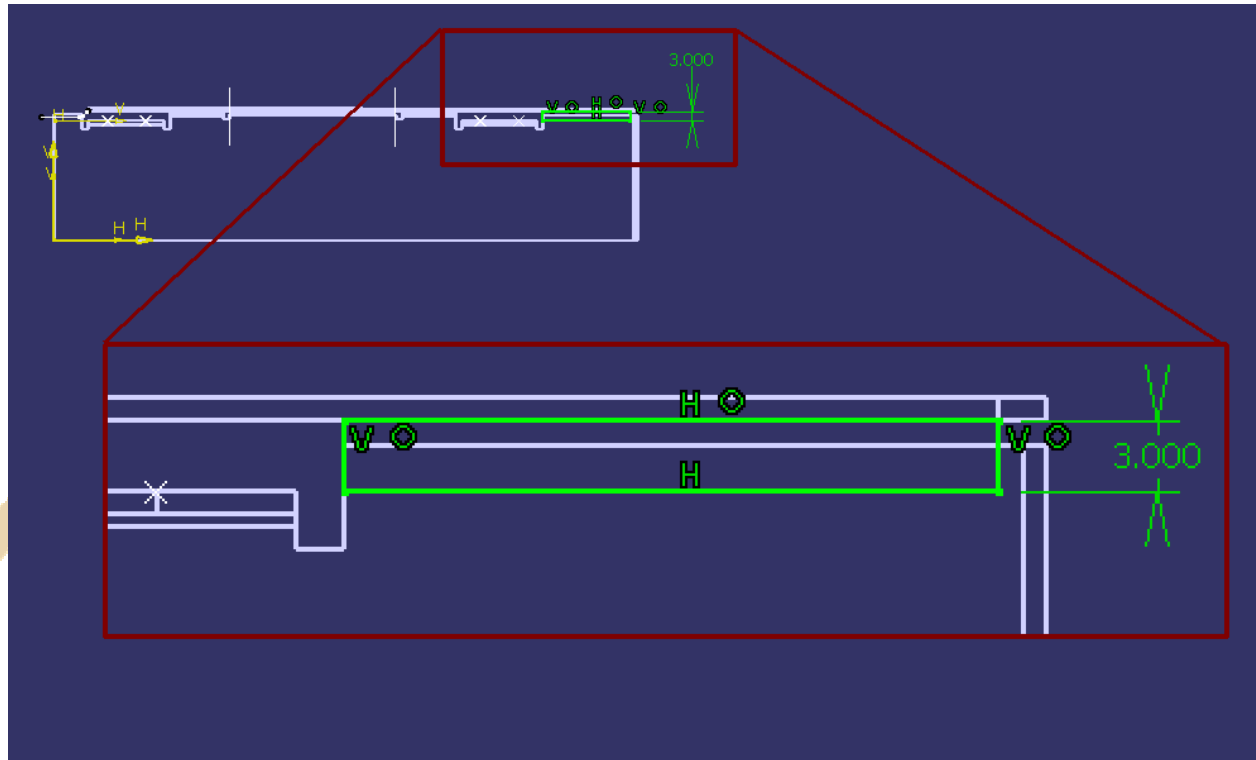


Add material in this direction

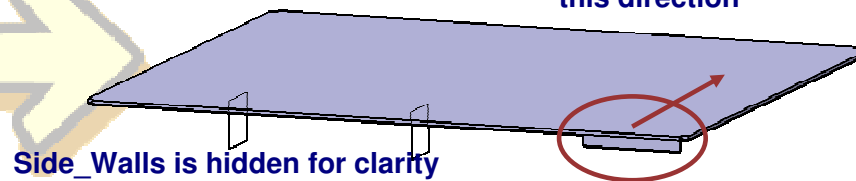
Student Notes:

## Design Process: Wall Creation (2/4)

4. Create Profile wall as shown. Add material toward the inside of the part.



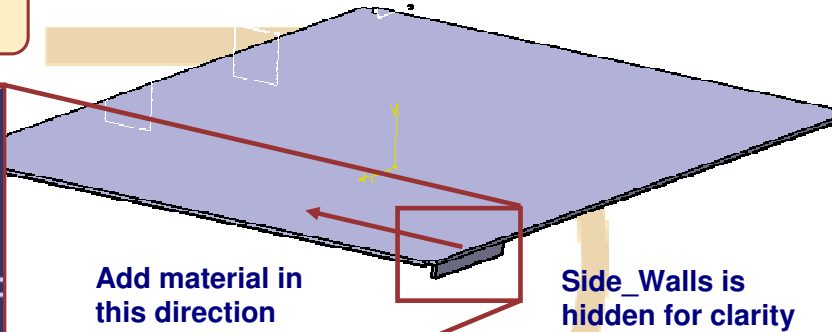
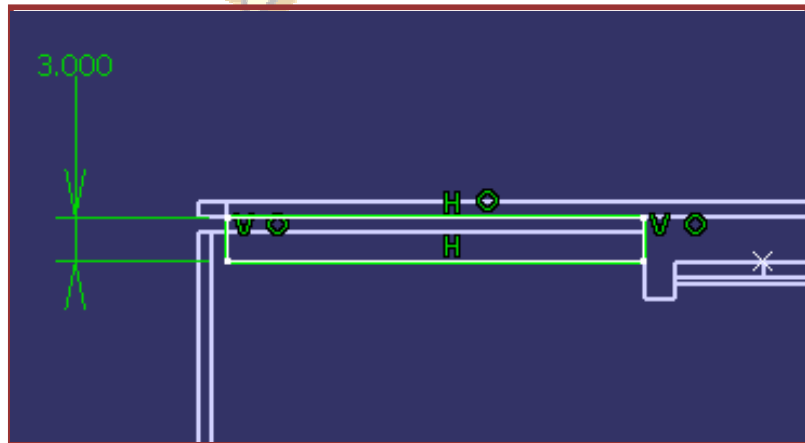
Add material in this direction



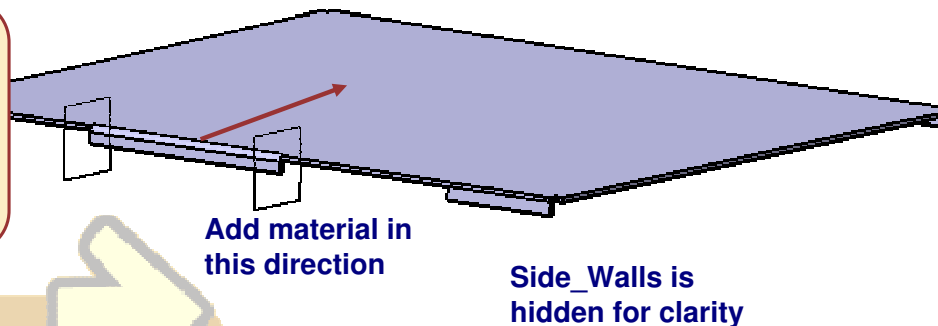
Student Notes:

## Design Process: Wall Creation (3/4)

5. Create another profile wall on the opposite side of part.



6. Create Edge wall. Use the two reference planes to limit the wall. Create the wall 3mm in height with a bend. Define the height from the top of the reference wall.



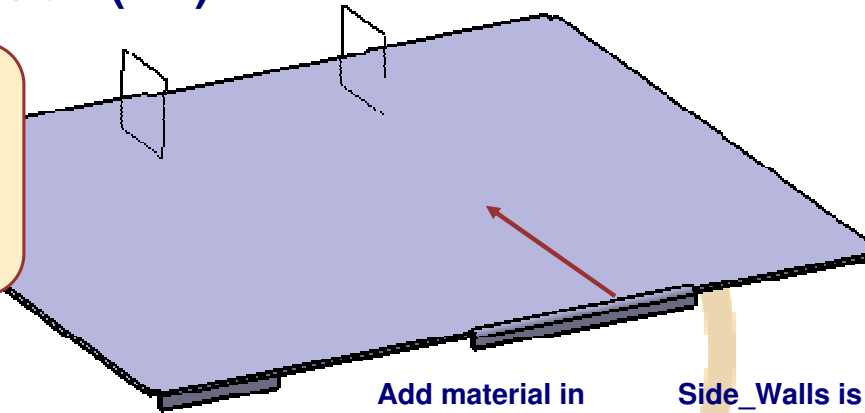


Student Notes:

## Design Process: Wall Creation (4/4)



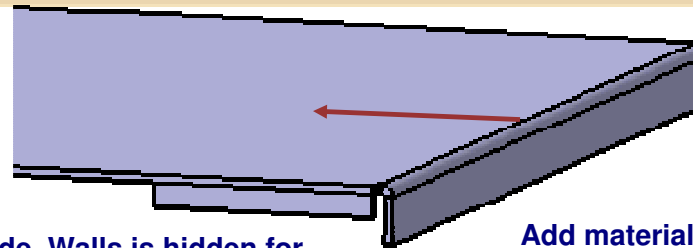
7. Create another Edge wall using the same values and limiting planes on the opposite side.



Add material in this direction

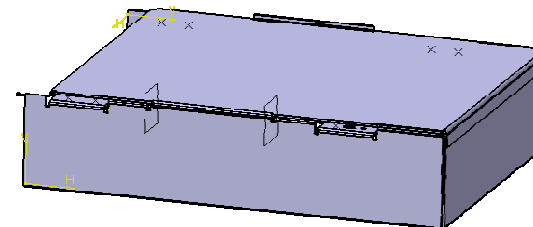
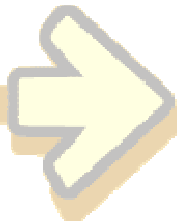
Side\_Walls is hidden for clarity

8. Create Edge wall on the back of the Top\_cover. Create wall 7.5mm in height with a bend. Define the height from the bottom of the reference wall.



Side\_Walls is hidden for clarity

Add material in this direction



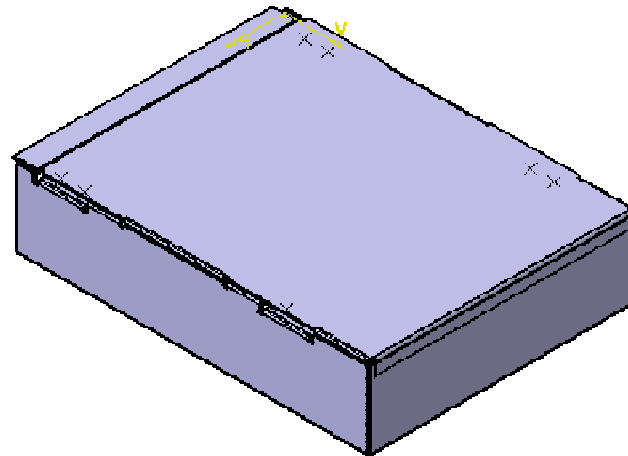
# Casing Assembly

## Master Exercise Step 2



In this exercise you will create :

- Bends
- Hem
- User Defined Flange



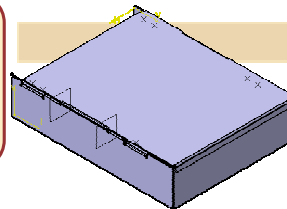
Student Notes:

## Design Process: Bend and Flange Creation

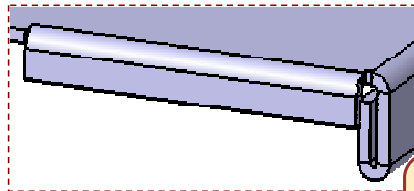
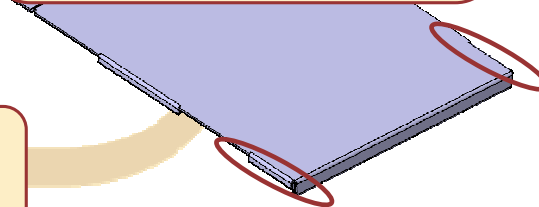


Part used: Casing\_2.CATProduct

1. Open Casing.CATProduct. If you did not complete Casing Assembly(1) open Casing\_2.CATProduct instead.

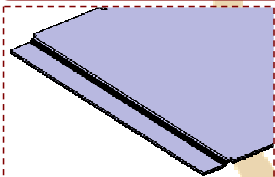


2. Activate the Top\_Cover part. Hide the Side\_Walls part. Create bends between Wall.1 and the the two side profile walls created in the last exercise.

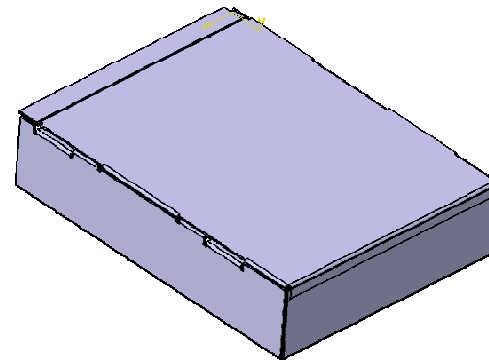
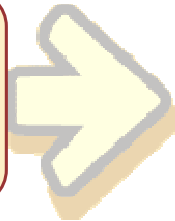


3. Create a hem on the back edge wall. R = 0.3mm, Length = 5mm

4. Create a user-defined flange using Sketch.2 as the profile.



5. Hide the Geometrical Set.1 body in the Top\_Cover part. Show the Side\_Walls part.



Student Notes:

# Casing Assembly

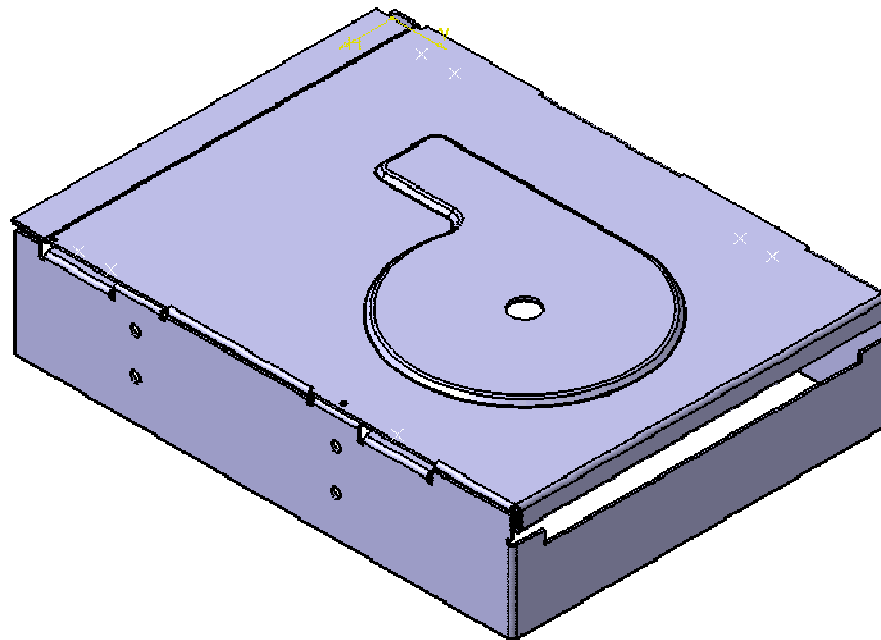
## Master Exercise Step 3



20 min

In this exercise you will create :

- A Cutout
- Circular Cutouts
- A Stamp



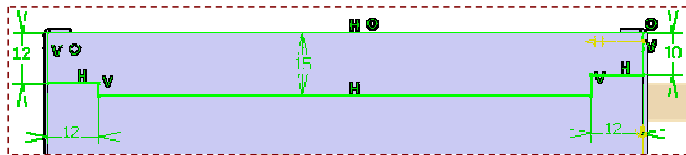
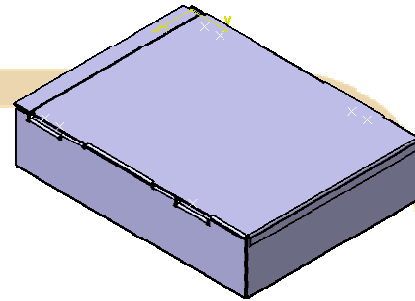
# Cutout, Circular Cutout and Stamp Creation (1/4)



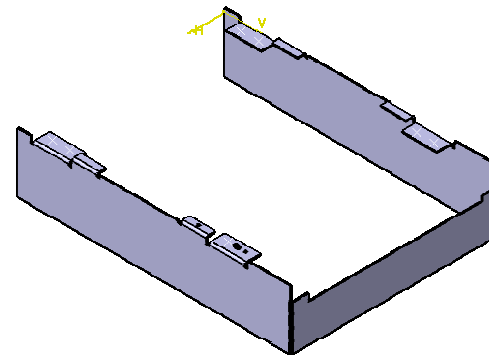
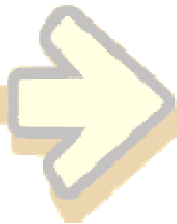
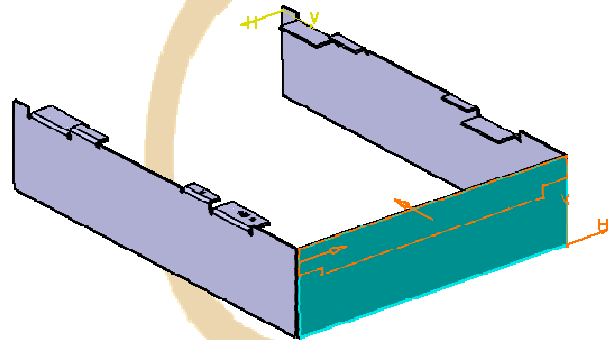
Part used: Casing\_3.CATProduct



1. Open Casing.CATProduct. If you did not complete Casing Assembly(2) open Casing\_3.CATProduct instead.



2. Activate the Side\_Walls part. Hide the Top\_Cover part. Show the Cutout\_Profile sketch. Create a cutout using the Cutout\_Profile sketch as its profile.

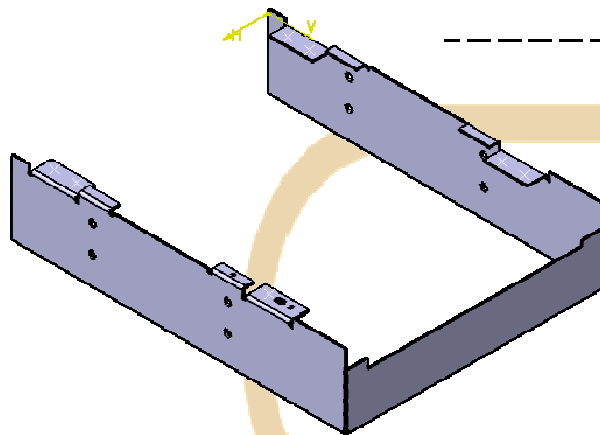
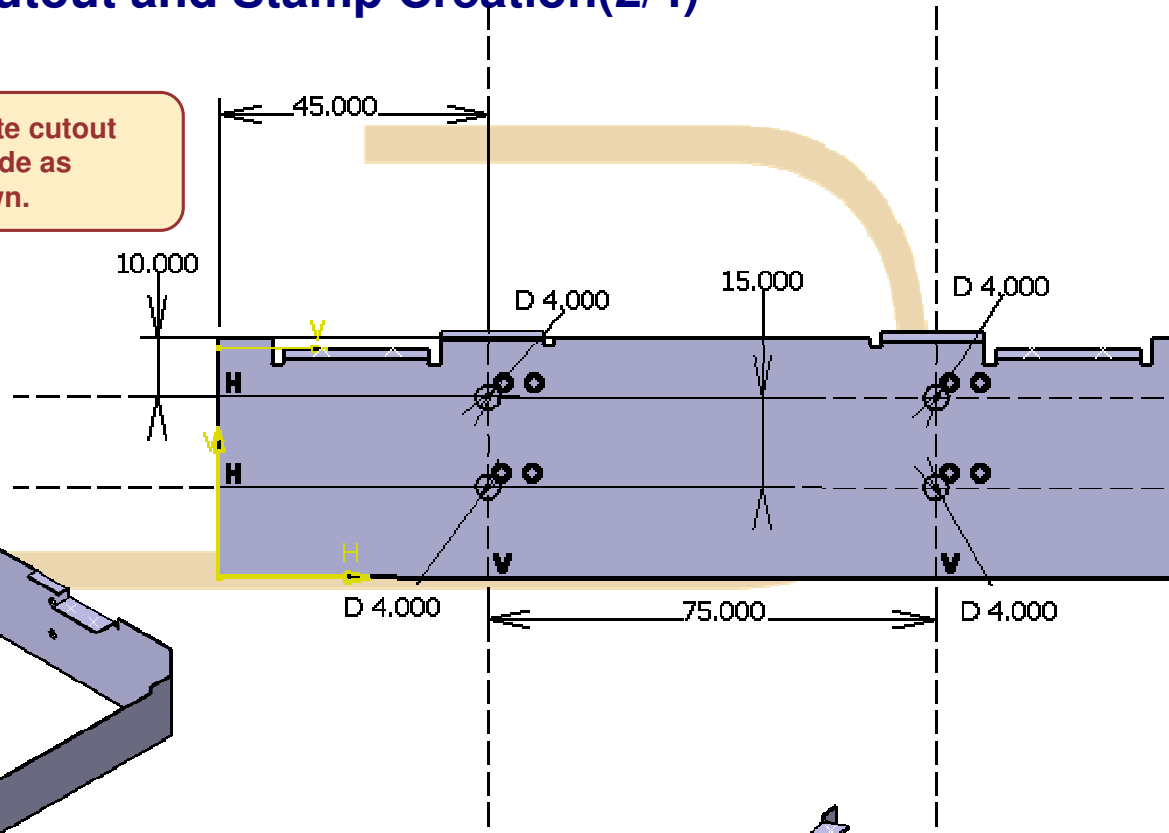


Student Notes:

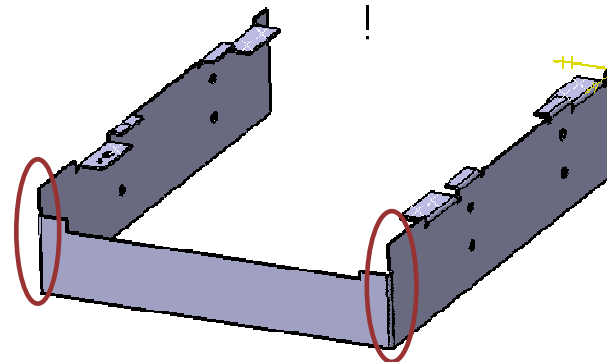
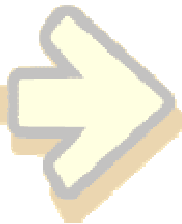
### Cutout, Circular Cutout and Stamp Creation(2/4)



3. Create cutout on side as shown.



4. Create bends as shown.

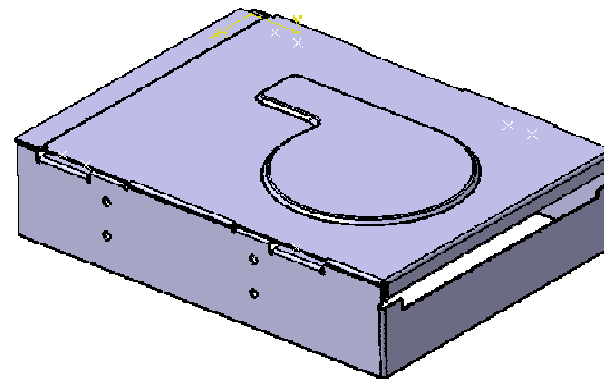
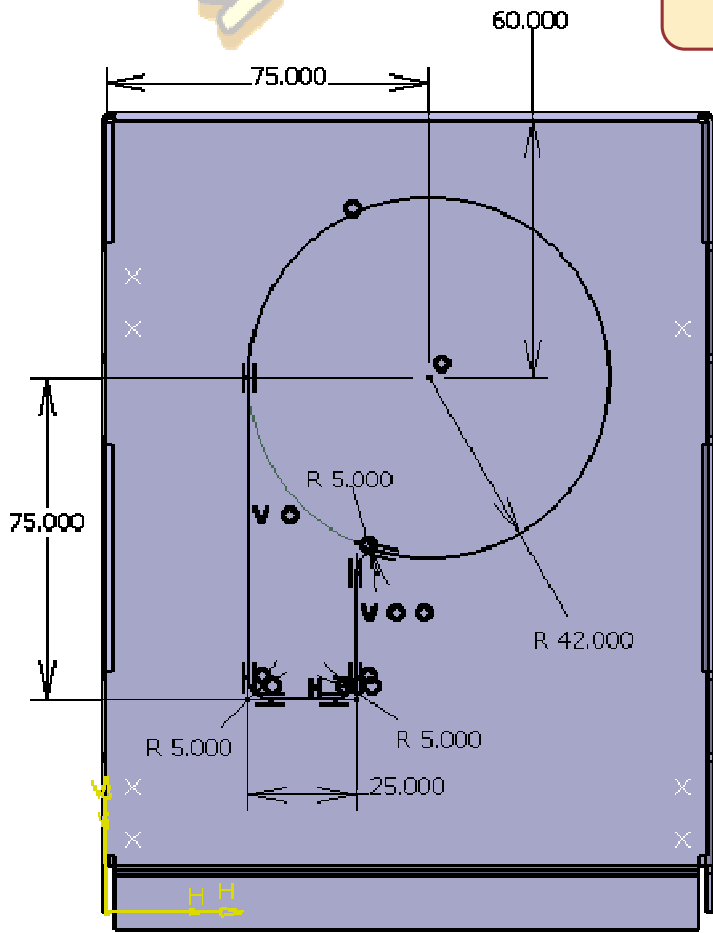


Student Notes:

### Cutout, Circular Cutout and Stamp Creation (3/4)

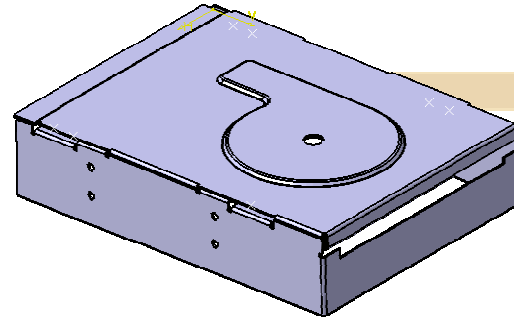
5. Activate the Top\_Cover part. Create sketch as shown.

6. Create surface stamp.  
Height = 2mm, R1 = 0.5mm, R2 = 0.5mm, Angle = 45deg.

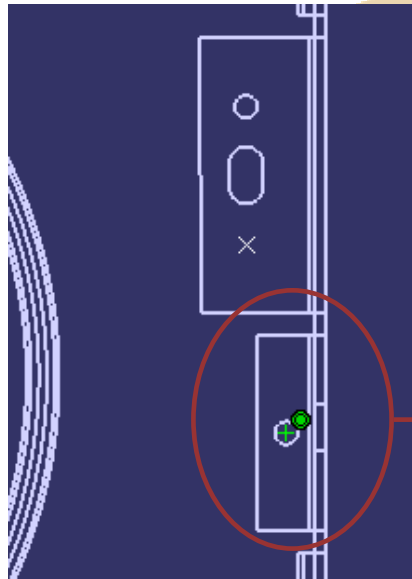


Student Notes:

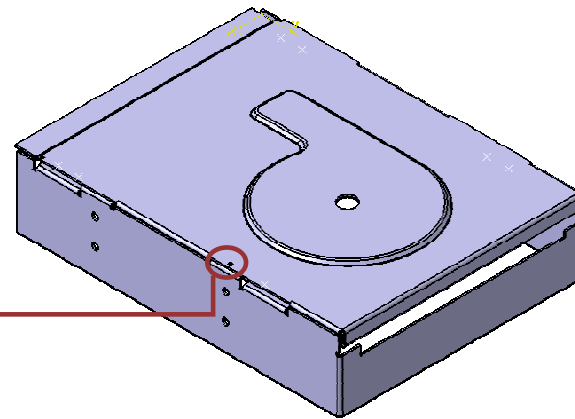
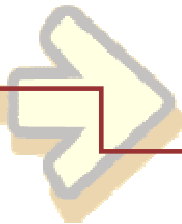
## Cutout, Circular Cutout and Stamp Creation (4/4)



7. Create a circular cutout concentric with the arc from the surface stamp profile. Diameter of the hole should be 10mm.



8. Create a circular cutout concentric with the hole in the side\_wall part. Diameter of the hole should be 2mm.





Student Notes:

# Casing Assembly

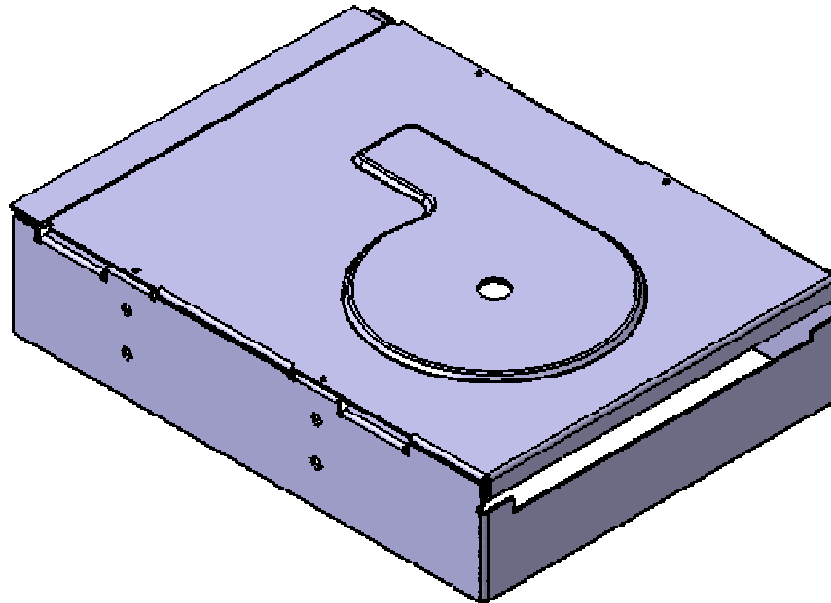
## Master Exercise Step 4



15 min

In this exercise you will create :

- A Rectangular Pattern
- A User Defined Pattern



Student Notes:

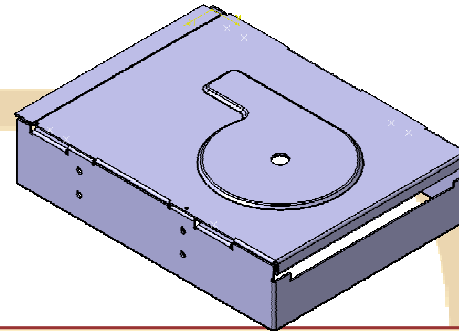
## Design Process: Pattern Creation (1/2)



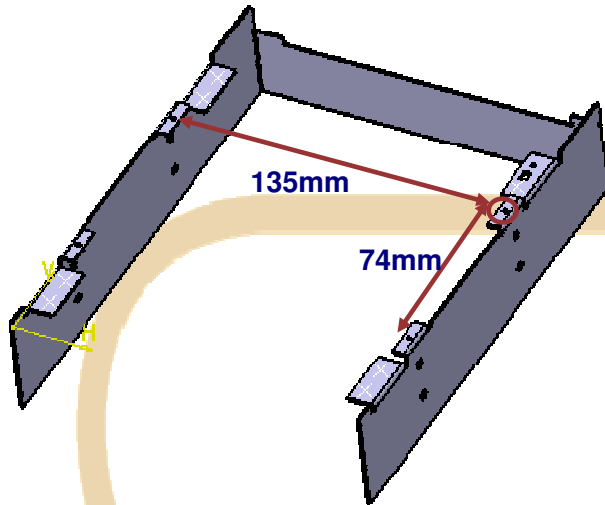
Part used: Casing\_4.CATPart



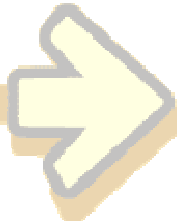
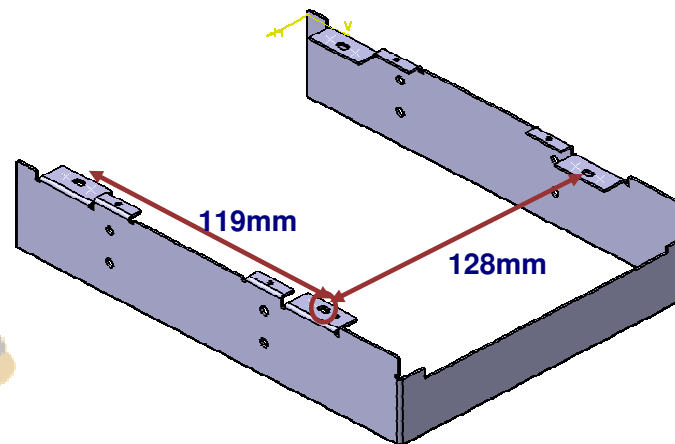
1. Open Casing.CATProduct. If you did not complete Casing Assembly(3) open Casing\_4.CATProduct instead.



2. Activate the Side\_Walls part. Hide the Top\_Cover part. Create rectangular pattern of hole. Pattern 2 instances 74mm apart in one direction and 2 instances 135mm apart in the other direction.



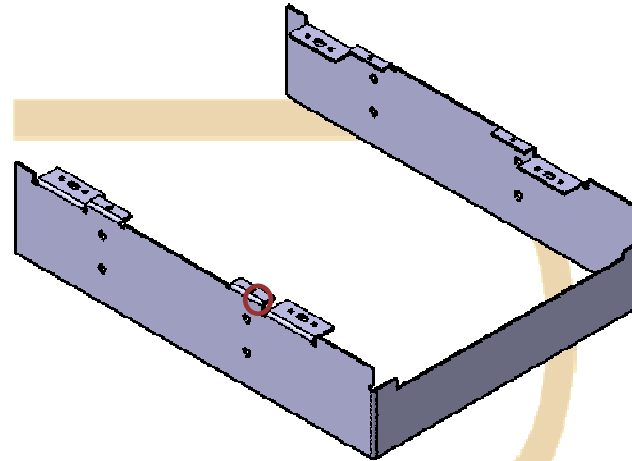
3. Create rectangular pattern of cutout. Pattern 2 instances 119mm apart in one direction and 2 instances 128mm apart in the other.



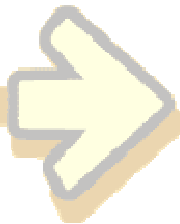
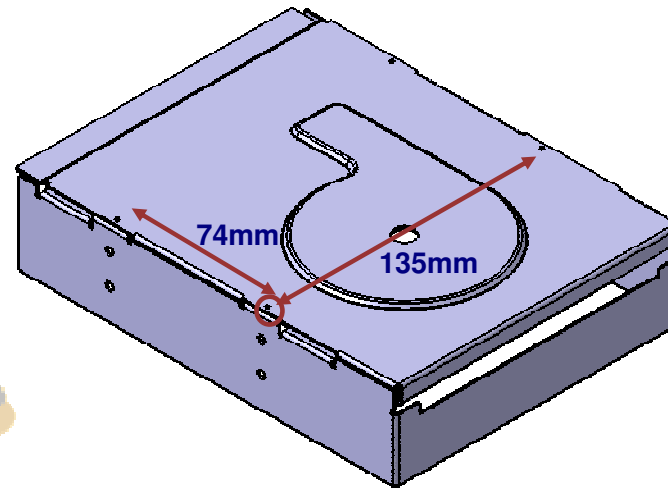
## Design Process: Pattern Creation (2/2)



4. Create user-defined pattern of hole. Use sketch "User\_pattern\_locations" for the positions.



5. Activate the Top\_Cover part. Create rectangular pattern of hole. Pattern 2 instances 74mm apart in one direction and 2 instances 135mm apart in the other direction.



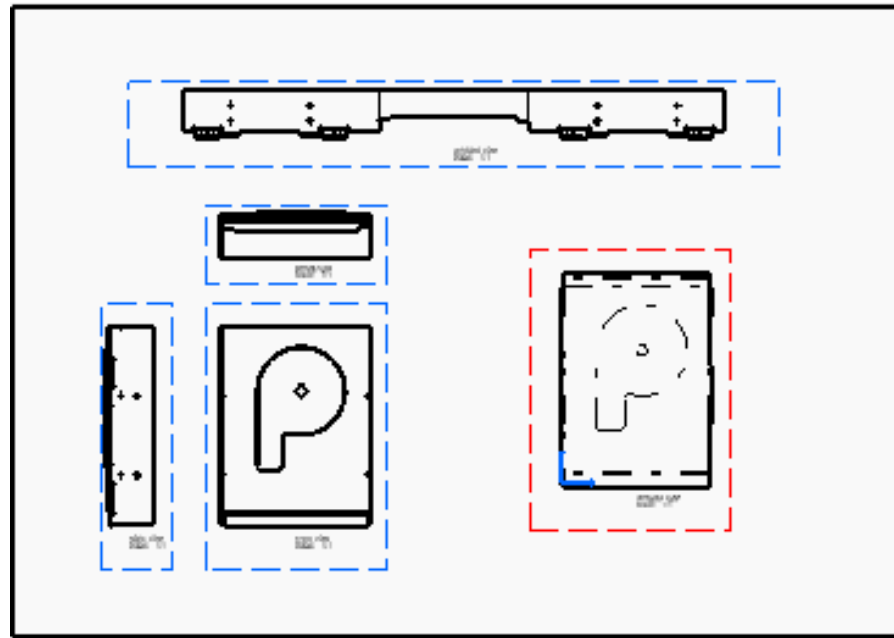
# Casing Assembly

## Master Exercise Step 5



In this exercise you will create :

- A drawing with three main views
- Unfolded views



Student Notes:

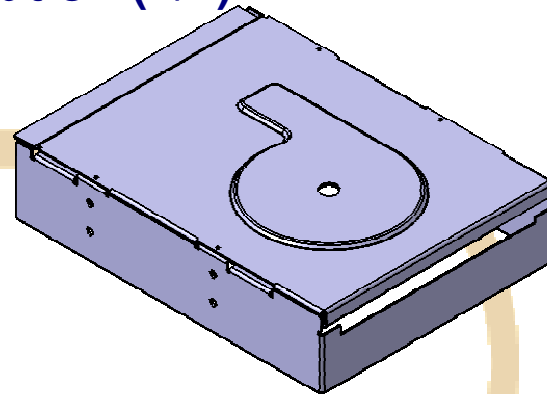
# Design Process: Unfolded View Creation (1/2)



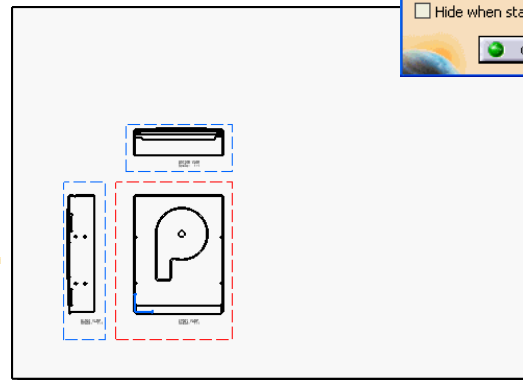
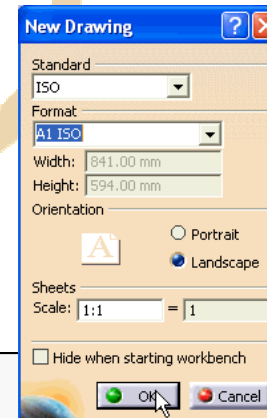
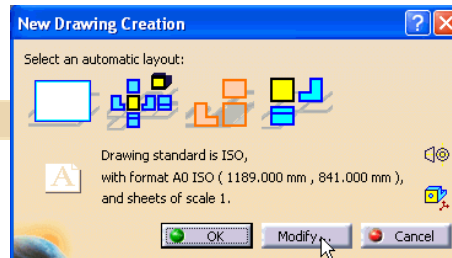
Product used : Casing\_5.CATProduct



1. Open Casing.CATProduct. If you did not complete Casing Assembly(4) open Casing\_5.CATProduct instead.



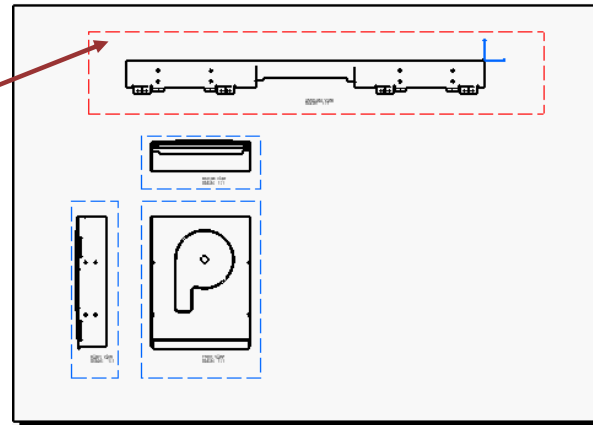
2. Create drawing with 3 main views, Front, Bottom, Right. Use ISO format and A1 ISO page size.



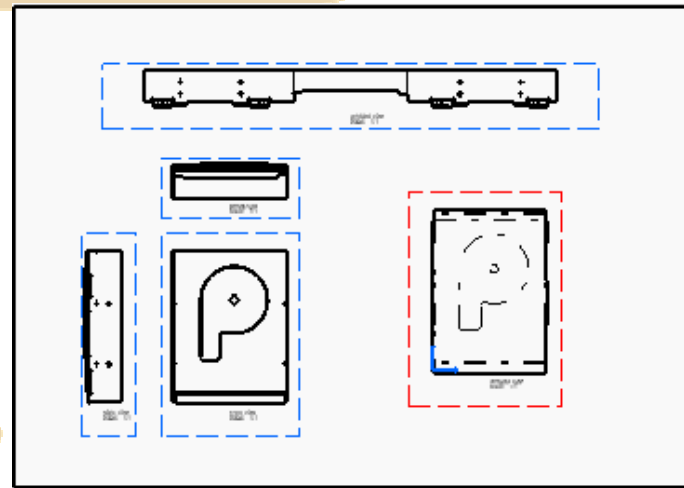
## Design Process: Unfolded View Creation (2/2)



3. Create unfolded view of the Side\_Walls part.



4. Create unfolded view of the Top\_Cover part. Save the drawing with the name Casing.CATDrawing.



Student Notes:

## Casing Assembly Recap

- ✓ Design intent to create a casing assembly
- ✓ Create walls for the Top Cover:
  - Profile wall
  - Edge Wall
- ✓ Create bends
- ✓ Create hem and user defined flange
- ✓ Create Cutouts
- ✓ Create holes
- ✓ Create Surface stamp
- ✓ Pattern cutout and holes using rectangular and user-defined patterns
- ✓ Create drawing of assembly including 2 unfolded views.

