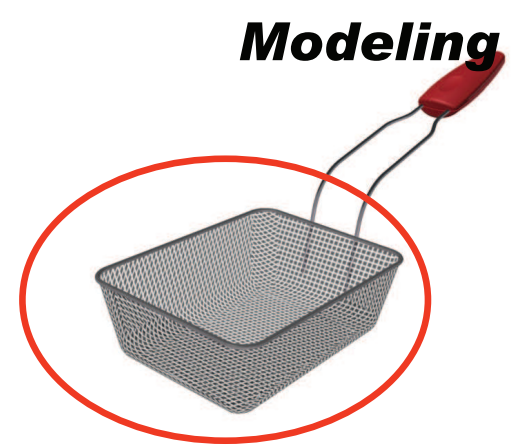


Tutorial 3B

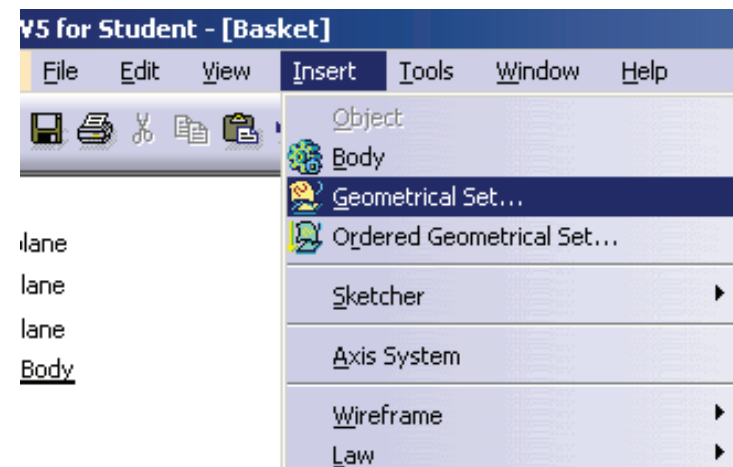
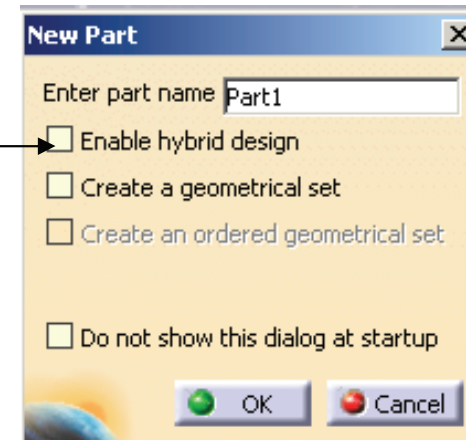
Building the basket:-

We are going to use **Surface-modeling** technique to build the basket...

- Select **File/New** on the menu bar
- Select “Part” in “List of Type
- Click ok to complete
- Enter “Basket” as Part Name
- Click ok to complete
- Select ‘**Start/Shape/Generative Shape Design**’ on the menu bar.
- Select “**Insert/Geometrical Set**” on the menu bar and click ok to complete (Now a new branch “Geometrical set” is created on the part tree, which is used to store all reference curves and surfaces)



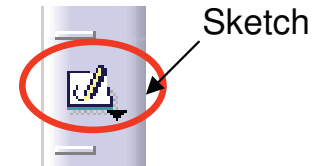
Don't check
“Hybrid
Design”



Tutorial 3B

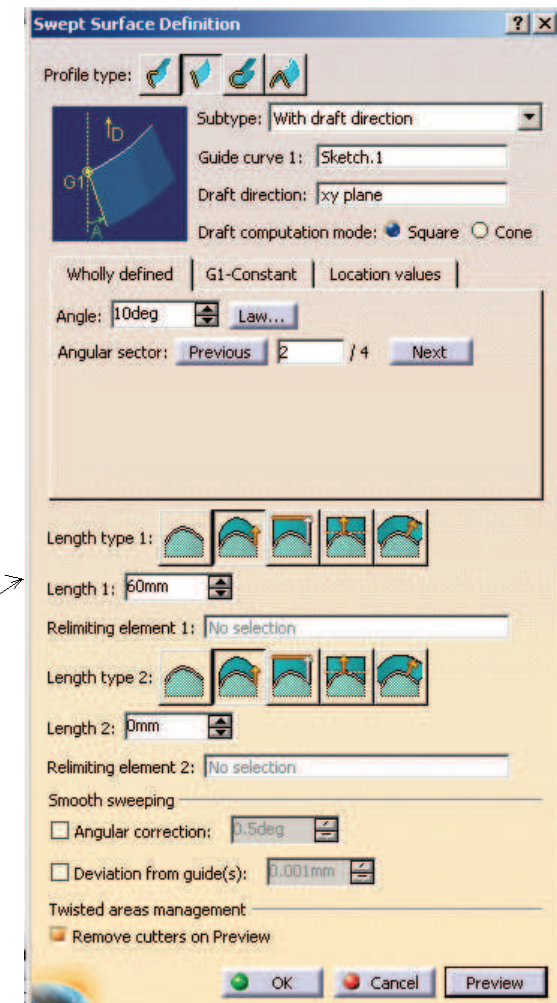
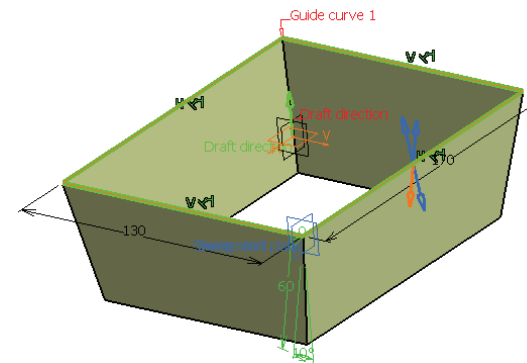
To build a sketch:-

- Click “**Sketch**” icon and select **xy plane**
- Draw** a **centered rectangle** (center at origin, 170mm x 130mm)
- Exit the workbench by clicking “**Exit**” icon



To create a swept surface:-

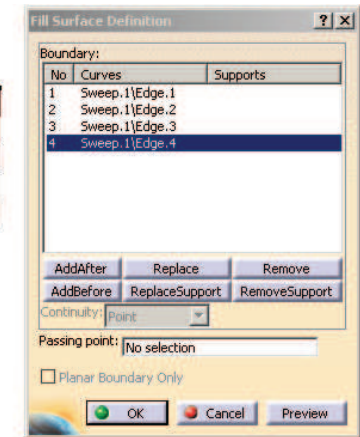
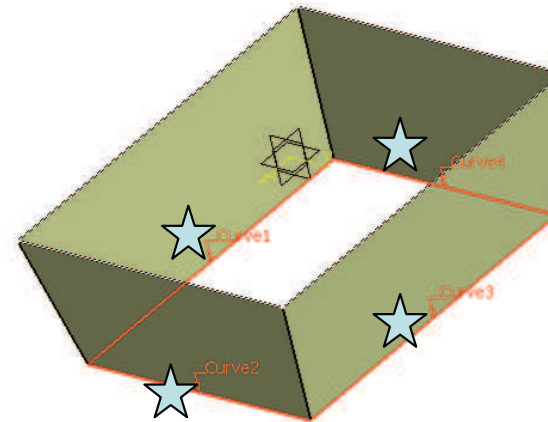
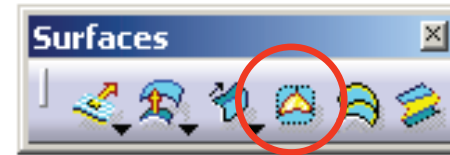
- Click “**Sweep**” icon
- Select “Line” as Profile Type
- Select “with draft direction” as subtype
- Select “Sketch.1” as Guide Curve.1
- Select xy plane as Draft Direction
- Enter 10 deg as Angle
- Enter 60mm as Length.1
- Click the arrow as shown (angular sector =2)
- Click ok to complete



Tutorial 3B

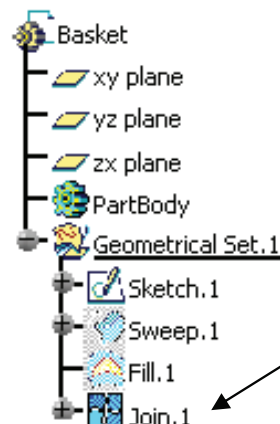
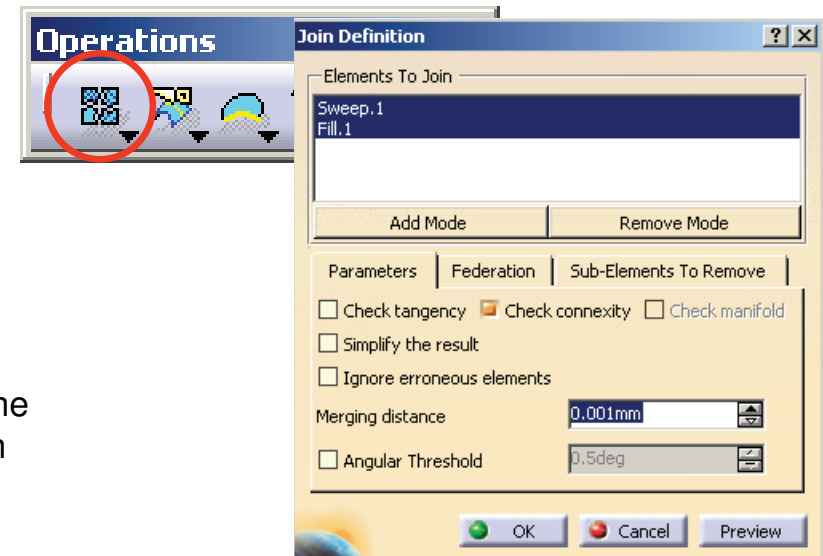
To create a surface from a closed boundary:-

- Click “**Fill**” icon
- Select all the four edges ★ of the smaller opening
- Click ok to complete (a surface will be created to fill the opening)



To Join surfaces into one:-

- Click “**Join**” icon
- Select surfaces “Sweep.1” & “Fill.1”
- Click ok to complete (a new surface is created to represent both surfaces; They are hidden now)



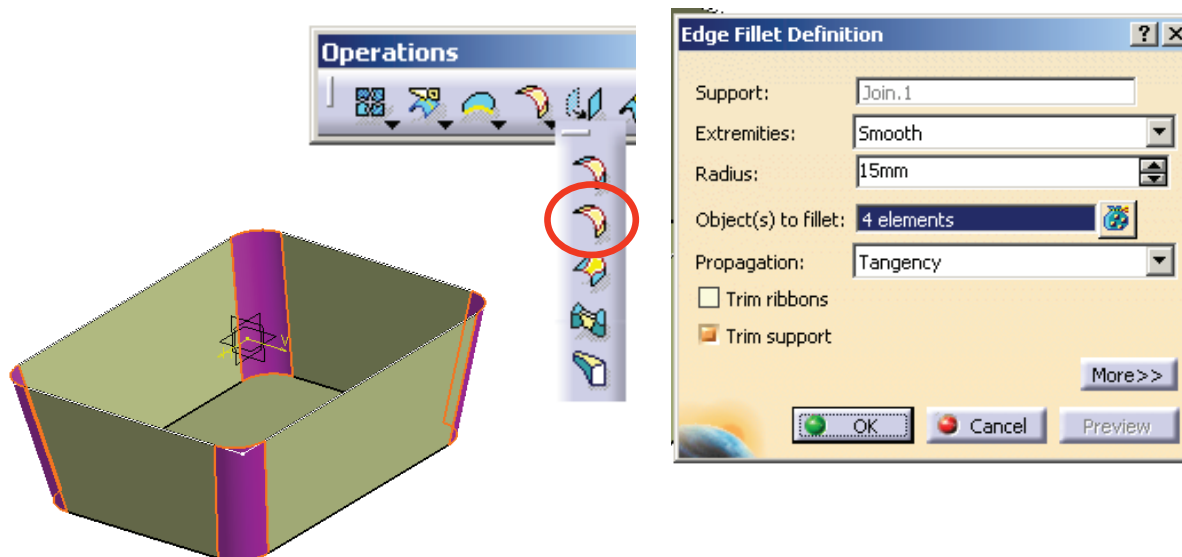
A new surface is created; the original surfaces are hidden after this creation

A- 54

Tutorial 3B

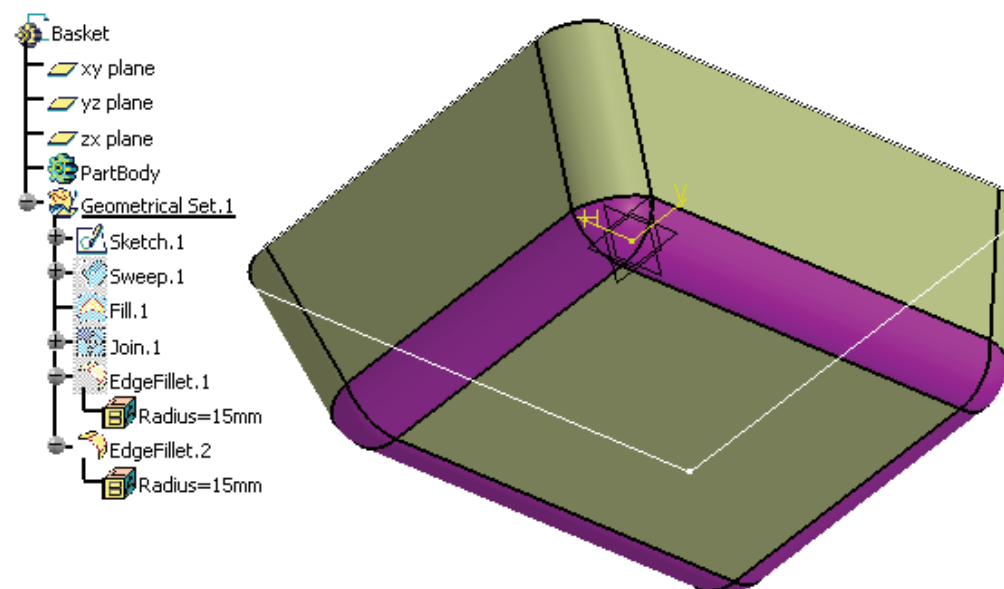
To add a Fillet on surface edges:-

- Click “Edge Fillet” icon
- Select all the four Vertical edges
- Enter 15mm as Radius
- Click ok to complete



To add another Fillet on surface edges:-

- Click “Edge Fillet” icon
- Select an edge of the bottom face
- Enter 15mm as Radius
- Click ok to complete



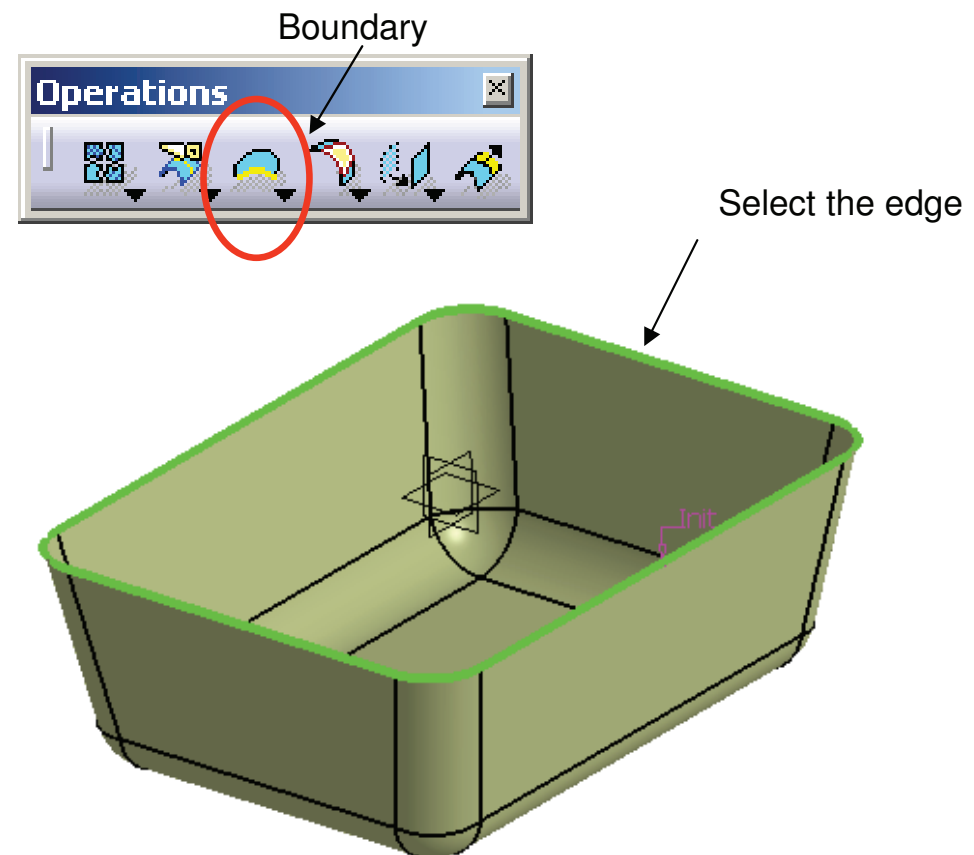
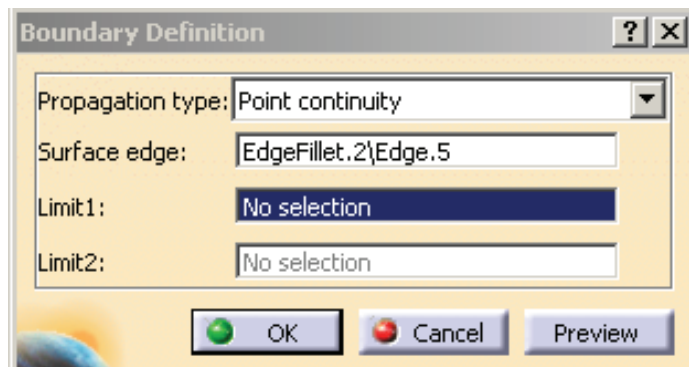
Tutorial 3B

To hide a sketch:-

- Right-Click on “Sketch.1”
- Select “**Hide/Show**”

To get a boundary from a surface:-

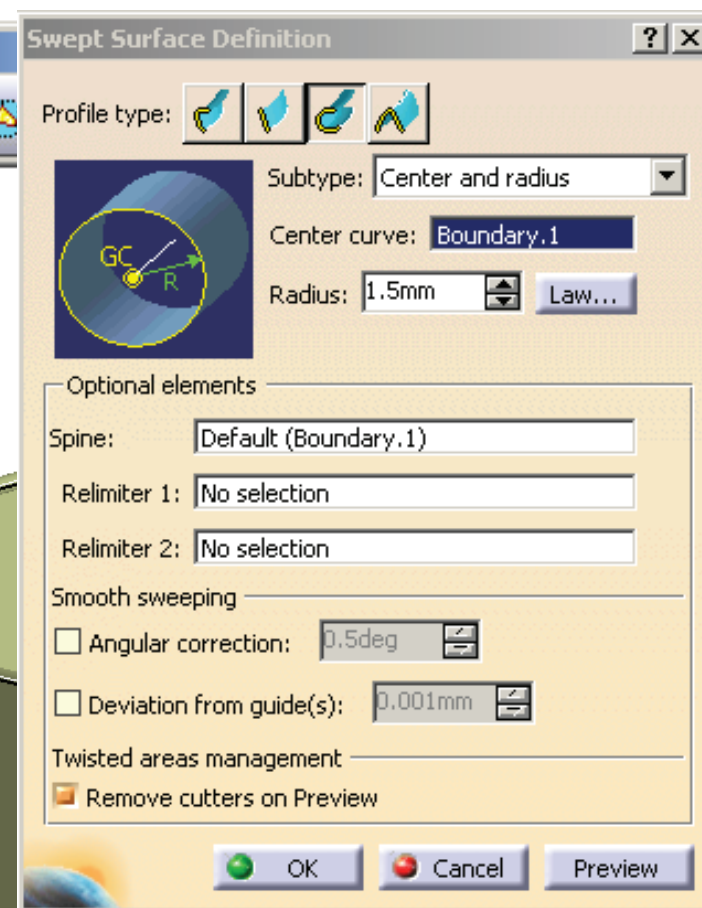
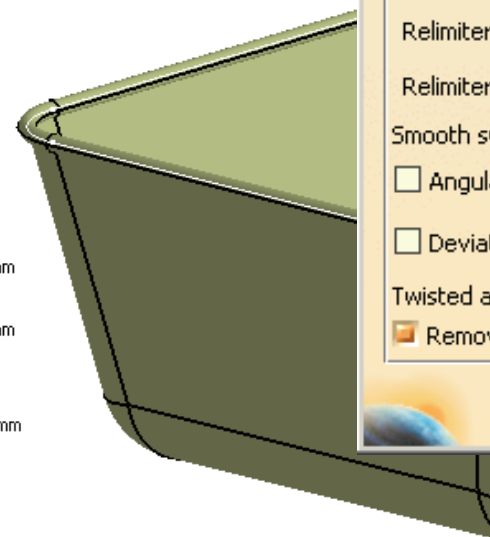
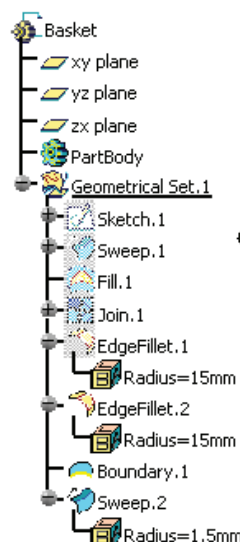
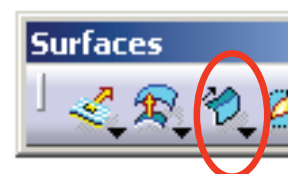
- Click “**Boundary**” icon
- Select “Point continuity” as propagation type
- Select an edge of the opening
- Click ok to complete



Tutorial 3B

To create a swept surface:-

- Click “**Sweep**” icon
- Select “Circle” as Profile Type
- Select “Center & Radius” as Subtype
- Select the curve “Boundary.1” as Center Curve
- Enter 1.5mm as Radius
- Click ok to complete



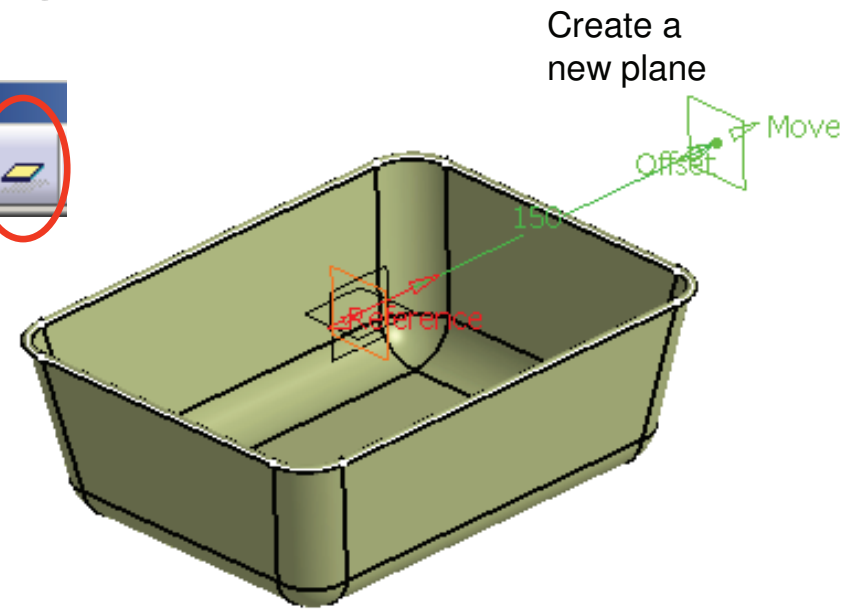
To hide a curve:-

- Right-click on “Boundary.1”
- Select “**Hide/Show**”

Tutorial 3B

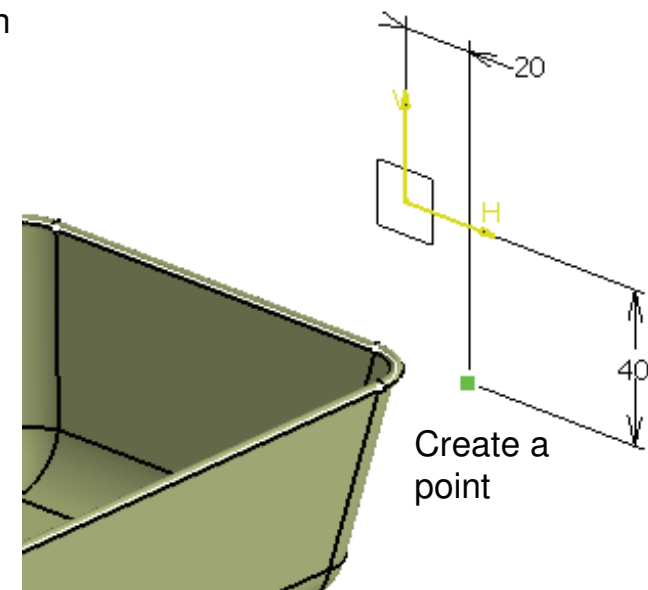
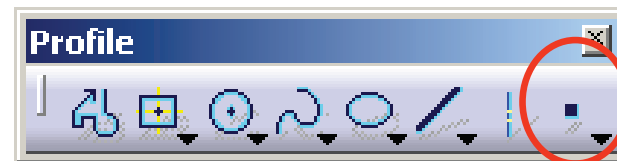
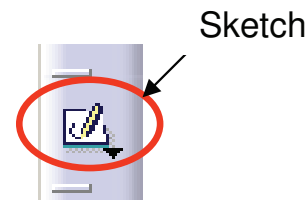
To create a reference plane:-

- Click “**plane**” icon
- Select “Offset from plane” as plane type
- Select “yz plane” as Reference
- Click “Reverse Direction” in the command window (The arrow should point to negative X)
- Enter 150mm as Offset value
- Click ok to complete



To make a point on a new sketch:-

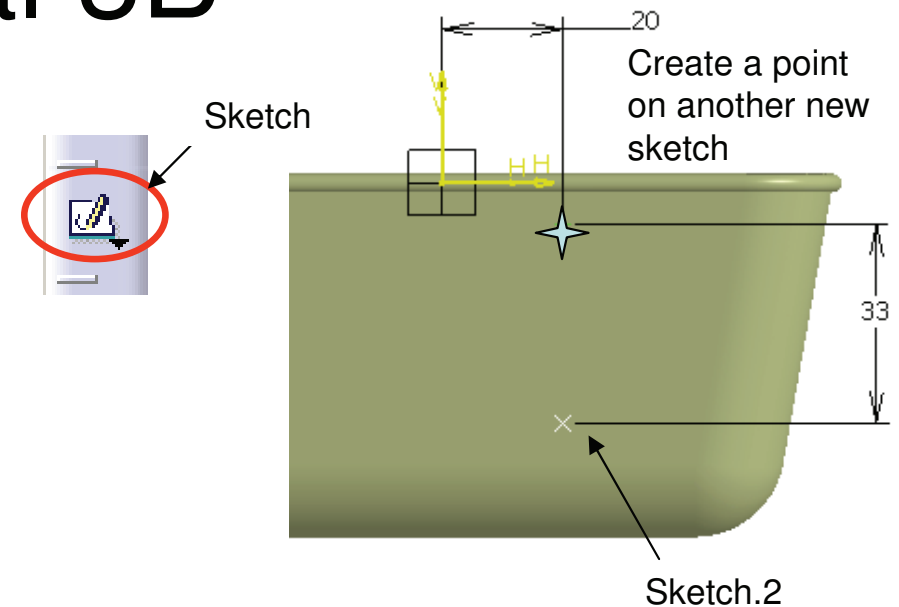
- Click “**Sketch**” icon and select **plane.1**
- **Draw a point** (x=20, y= - 40)
- Exit the workbench by clicking “**Exit**” icon



Tutorial 3B

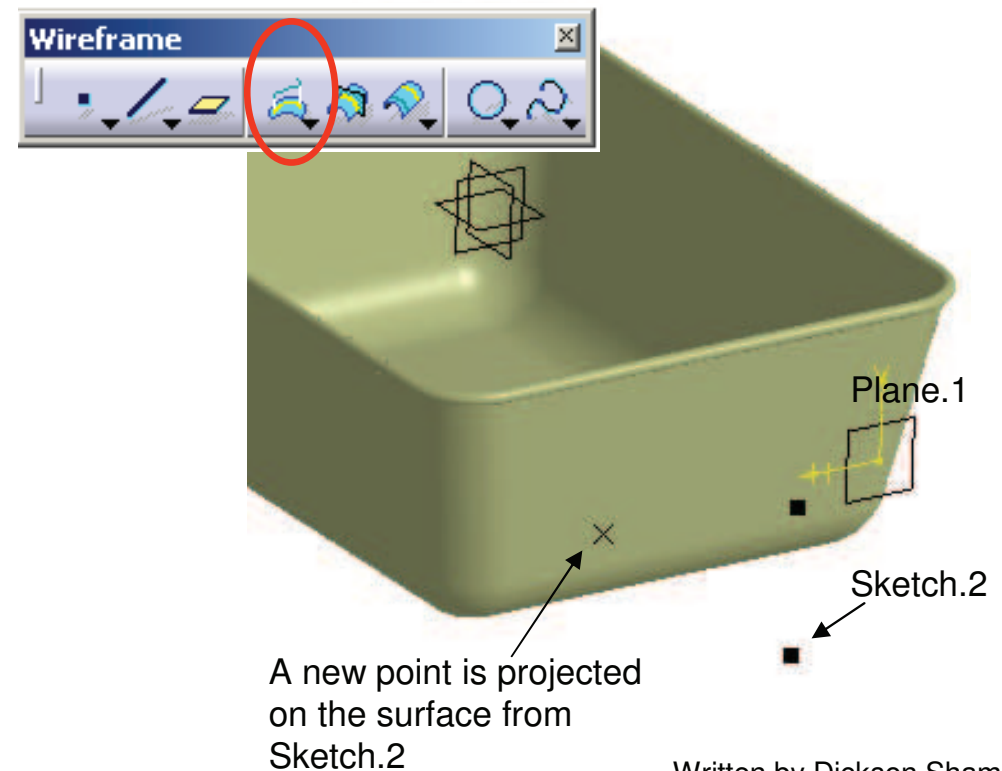
To make another point on a new sketch:-

- Click on empty space to deselect “Sketch.2”
- Click “**Sketch**” icon and select **plane.1**
- **Draw** a point ✨ (33mm above the previous point)
- Exit the workbench by clicking “**Exit**” icon



To project a point onto a surface:-

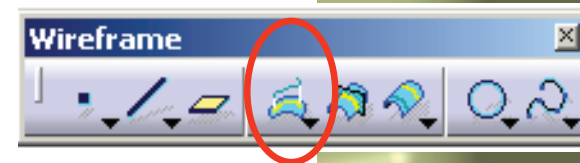
- Click “**Projection**” icon
- Select “Along a direction” as Projection Type
- Select the point “Sketch.2” as Projected
- Select the surface “Edgefillet.2” as Support
- Select Plane.1 as Direction
- Click ok to complete



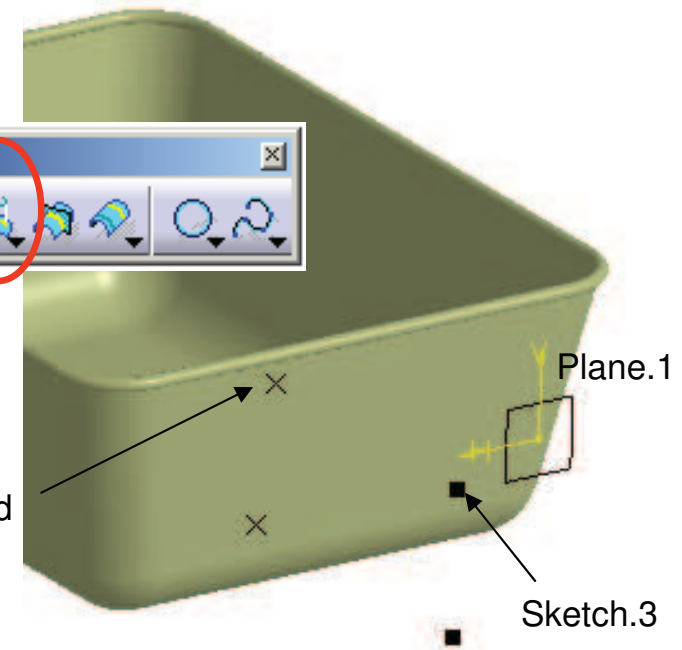
Tutorial 3B

To project another point onto a surface:-

- Click “**Projection**” icon again
- Select “Along a direction” as Projection Type
- Select the point “Sketch.3” as Projected
- Select the surface “Edgefillet.2” as Support
- Select Plane.1 as Direction
- Click ok to complete

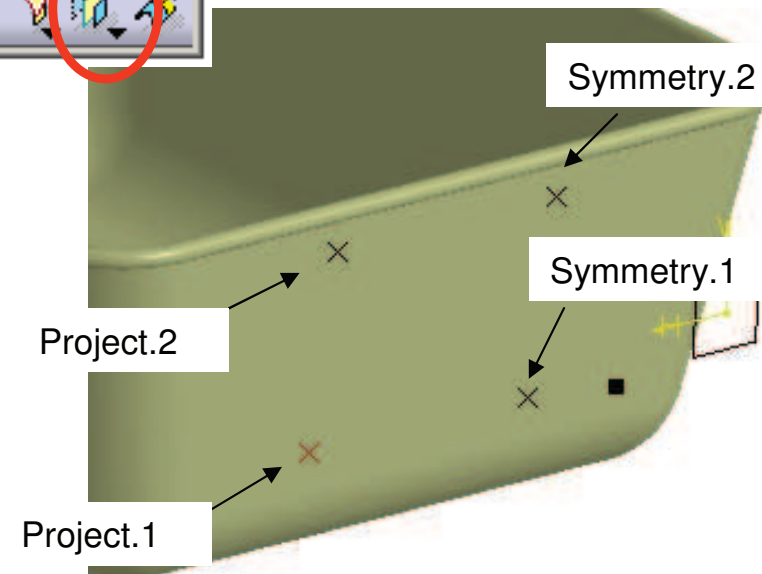


A new point is projected on the surface from Sketch.3



To make a mirror copy:-

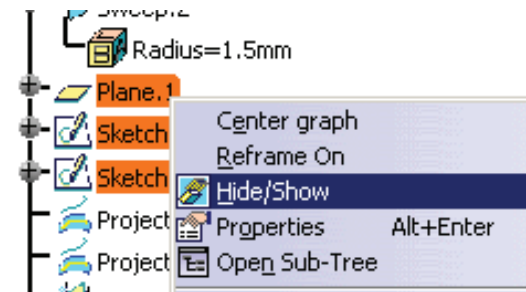
- Click “**Symmetry**” icon
- Select the point “Project.1” as Element
- Select zx plane as Reference
- Click ok to complete
- Click “**Symmetry**” icon again
- Select the point “Project.2” as Element
- Select zx plane as Reference
- Click ok to complete



Tutorial 3B

To hide all reference elements:-

- Multi-select Plane.1, Sketch.2 & Sketch.3
- Right-click on anyone
- Select **"Hide/Show"**



To add Material Texture:-

- Download a texture from a shared library at http://www.planit3d.com/source/texture_files/metal/metal.html
(The texture file has been stored in this folder:
(your DVD drive):\Model\mesh3.jpg)

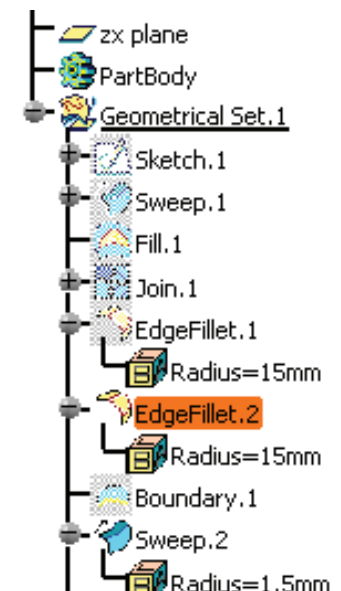
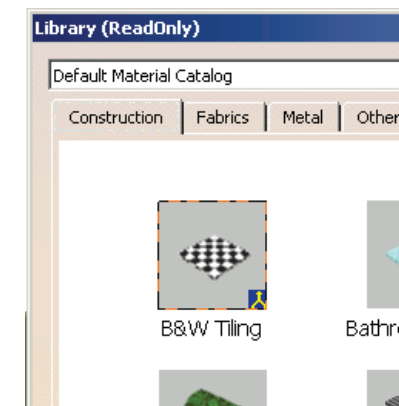


- Save the texture file into the project folder
- Click **"Apply Material"** icon
- Click B&W Tiling
- Select the surface "EdgeFillet.2" on the tree
- Click ok to complete



Apply Material

A- 61



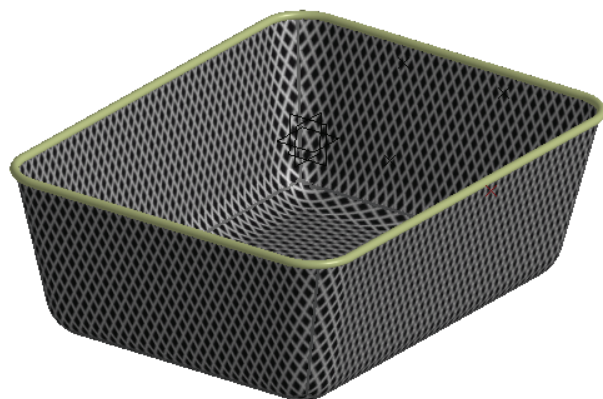
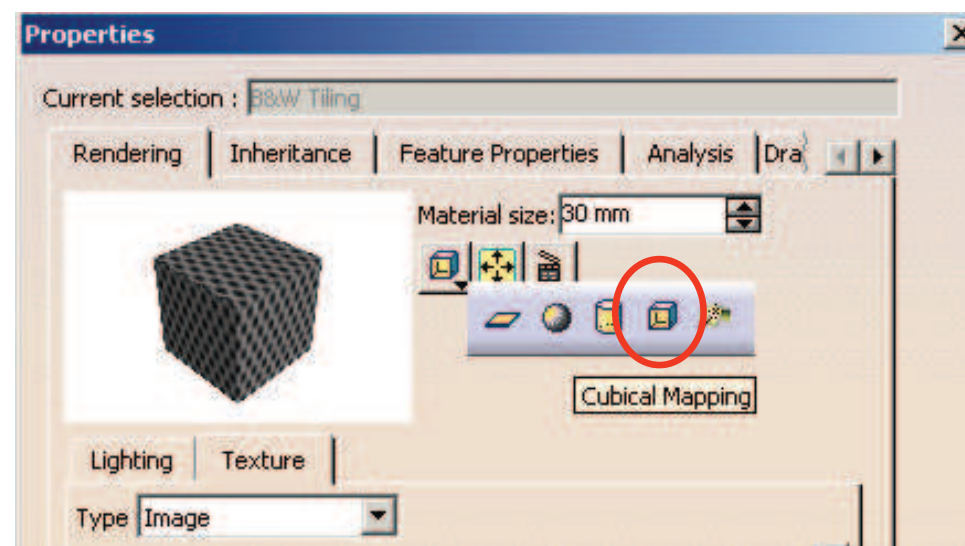
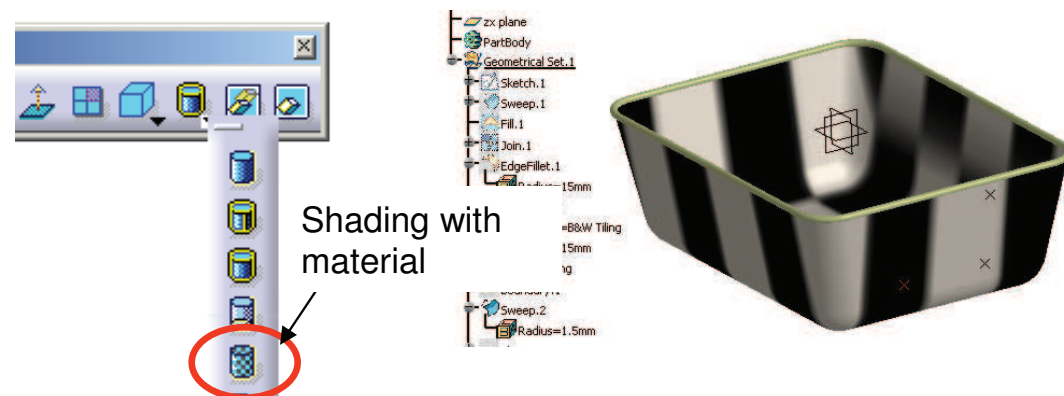
Tutorial 3B

To view the material texture:-

- Click “Shading with material” icon

To modify the texture:-

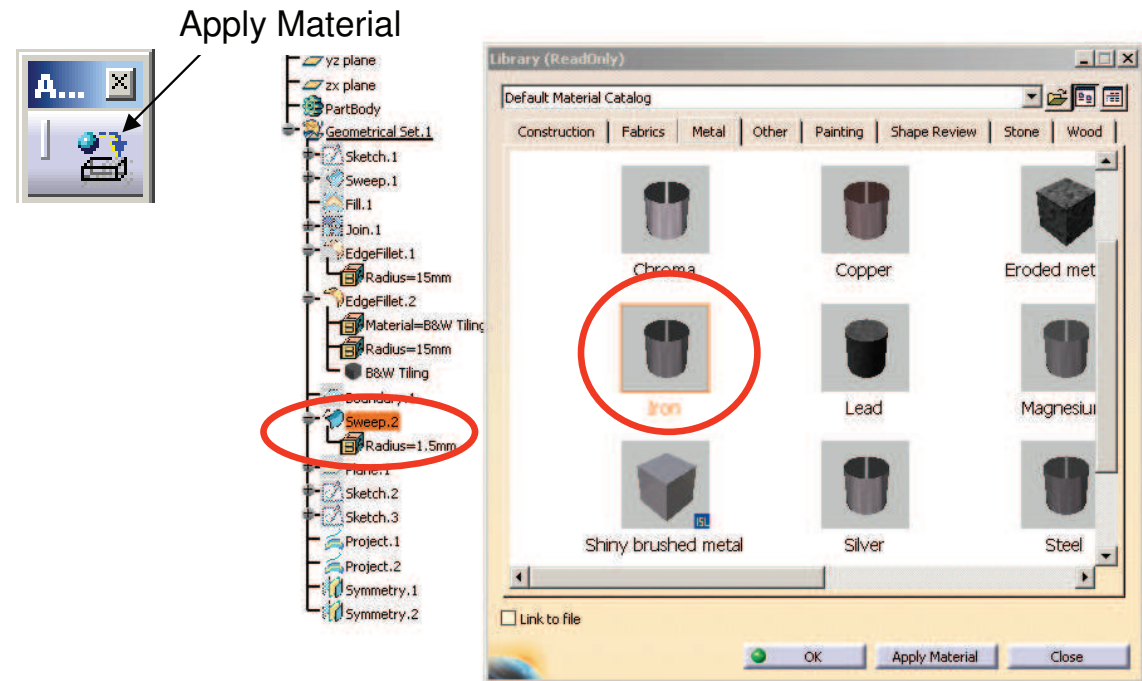
- Double-click “B&W” Tiling (under Edgefillet.2) on the tree
- Select the tab page “Texture”
- Select “Image” as Type
- Click “...” icon of Image Name
- Select the downloaded texture file
- Select “Cubical Mapping”
- Enter 30mm as Material Size
- Click ok to complete



Tutorial 3B

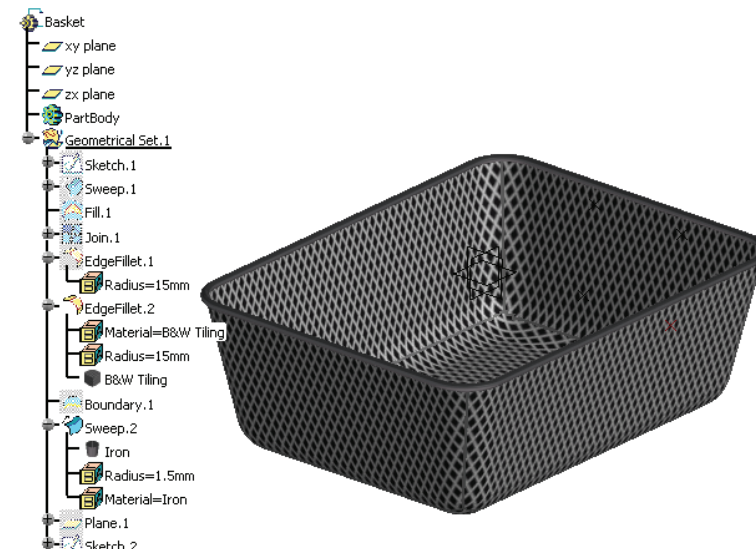
To add another material texture:-

- Click “**Apply Material**” icon
- Select “Iron” in the tab page “Metal”
- Select the surface “Sweep.2”
- Click ok to complete



To save the new part in Project Folder:-

- Save your current part as
“**Basket_a.CATPART**” into the folder.



A- 63

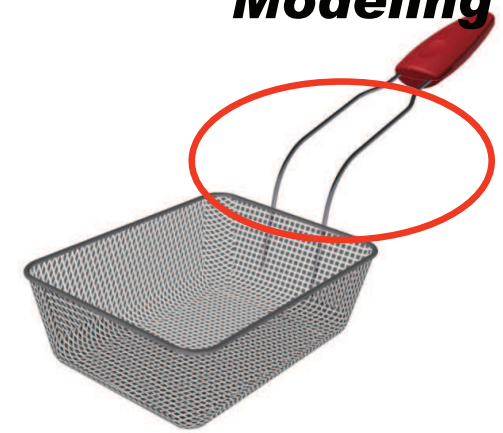
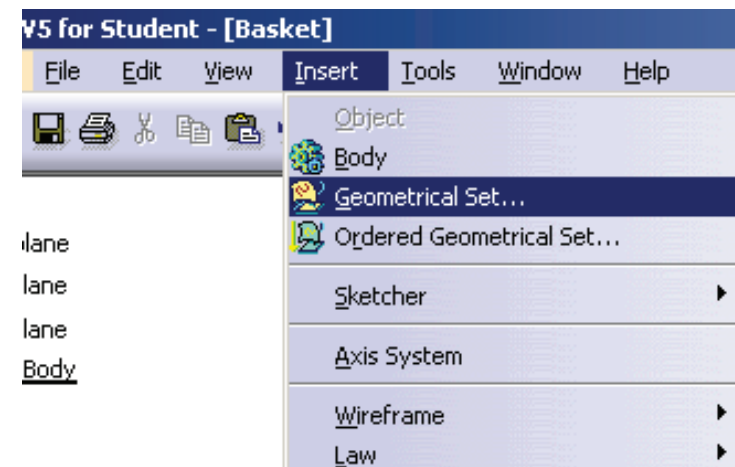
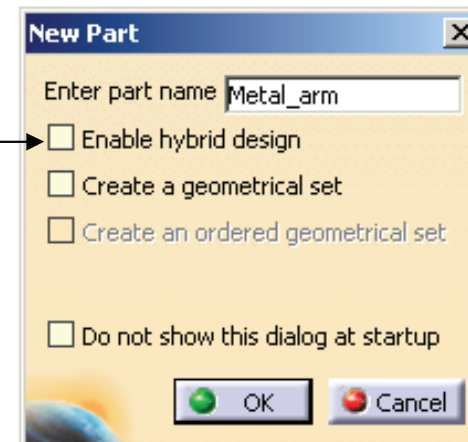
Tutorial 3B

Building the metal arm:-

We are going to learn how to build 3D curves to represent the metal arm...

- Select **"File/New"** on the menu bar
- Select "Part" in "List of Type"
- Click ok to complete
- Enter "Metal_arm" as Part Name
- Click ok to complete
- Select **'Start/Shape/Generative Shape Design'** on the menu bar.
- Select **"Insert/Geometrical Set"** on the menu bar and click ok to complete

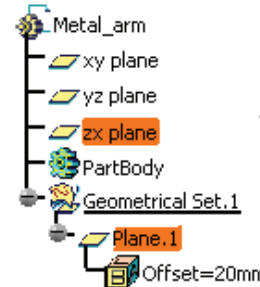
Don't check
"Hybrid
Design"



Tutorial 3B

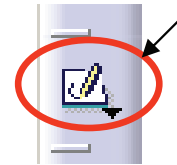
To create a reference plane:-

- Click “**plane**” icon
- Select “Offset from plane” as plane type
- Select “zx plane” as Reference
- Enter 20mm as Offset value
- Click ok to complete

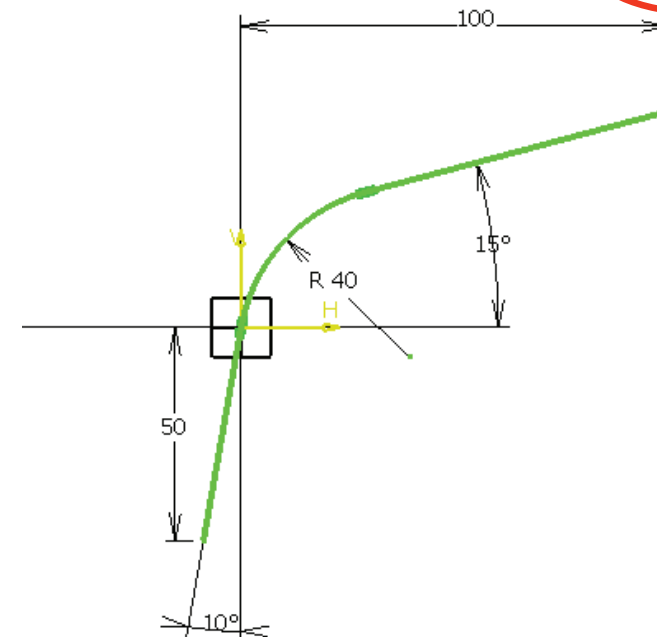
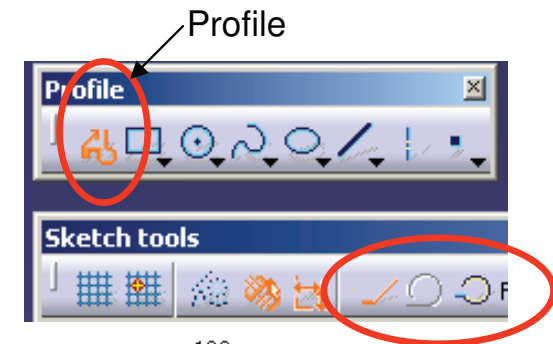


To make a Sketch:-

- Click “**Sketch**” icon
- Select Plane.1
- Draw the Profile as shown (You can use “Profile” command to draw the profile continuously)
- Add dimensional constraints
- Exit the workbench by clicking “**Exit**” icon



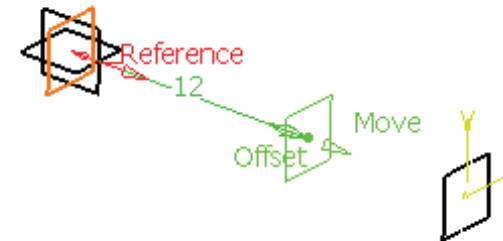
Sketch



Tutorial 3B

To create another reference plane:-

- Click “**plane**” icon
- Select “Offset from plane” as plane type
- Select “zx plane” as Reference
- Enter 12mm as Offset value
- Click ok to complete

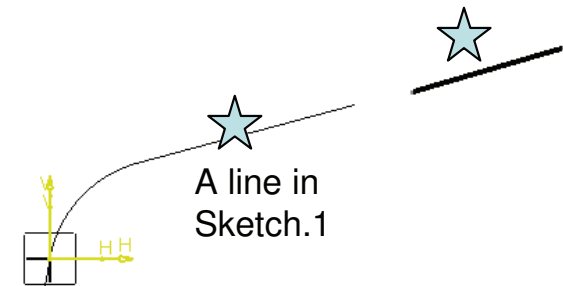


To make the 2nd Sketch:-

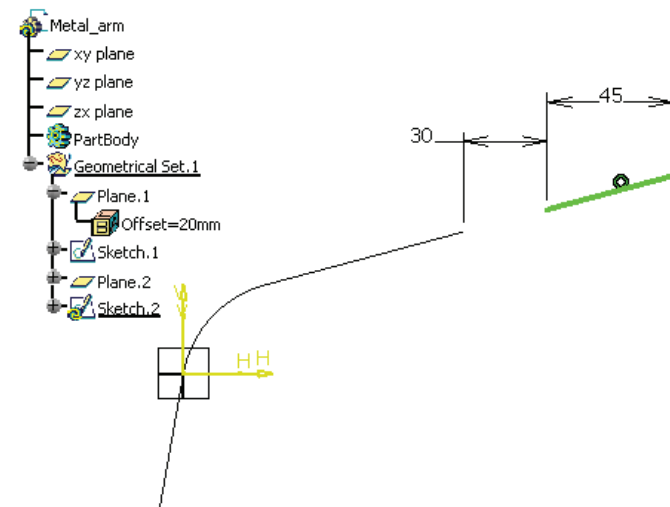
- Click “**Sketch**” icon
- Select Plane.2
- Draw a straight line as shown
- Multi-select the line ★ in the previous sketch and the current line; and add a Coincidence Constraint.
- Add two more dimensional constraints (remark: to have a horizontal dimension, right-click and select “Horizontal Measure Direction” while creating the constraint)
- Exit the workbench by clicking “**Exit**” icon



Sketch



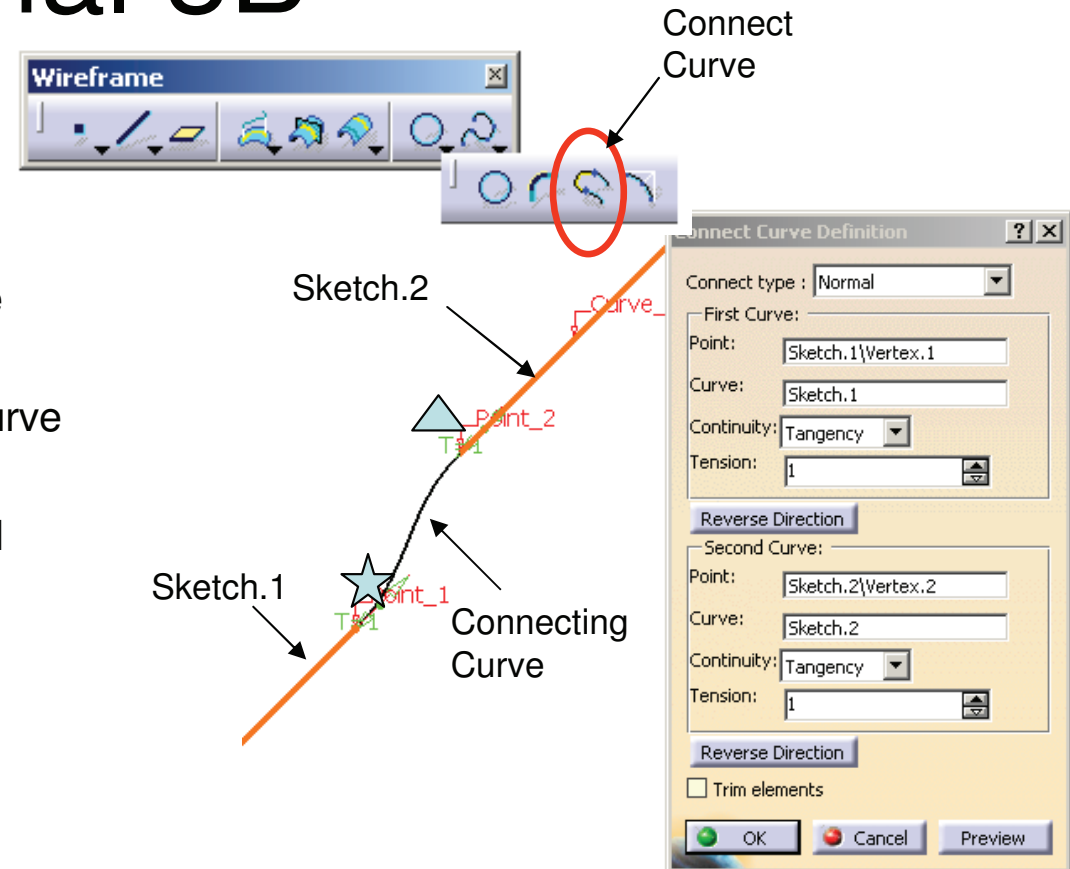
A line in Sketch.1



Tutorial 3B

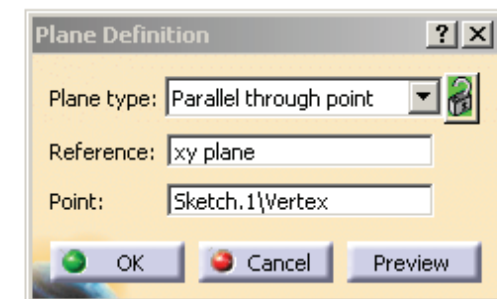
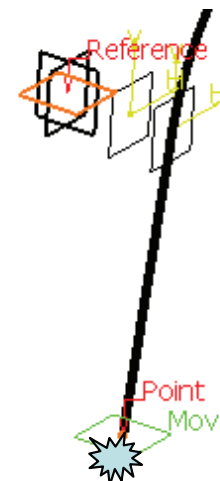
To create a connecting 3D curve:-

- Click “**Connect Curve**” icon
- Select the endpoint of Sketch.1 ★
- Select “Tangency” as Continuity of First Curve
- Select the endpoint of Sketch.2 ▲
- Select “Tangency” as Continuity of Second Curve
- Click “Preview” to have a preview
- Click “Reverse Direction” if the curve is flipped
- Click ok to complete



To create a reference plane:-

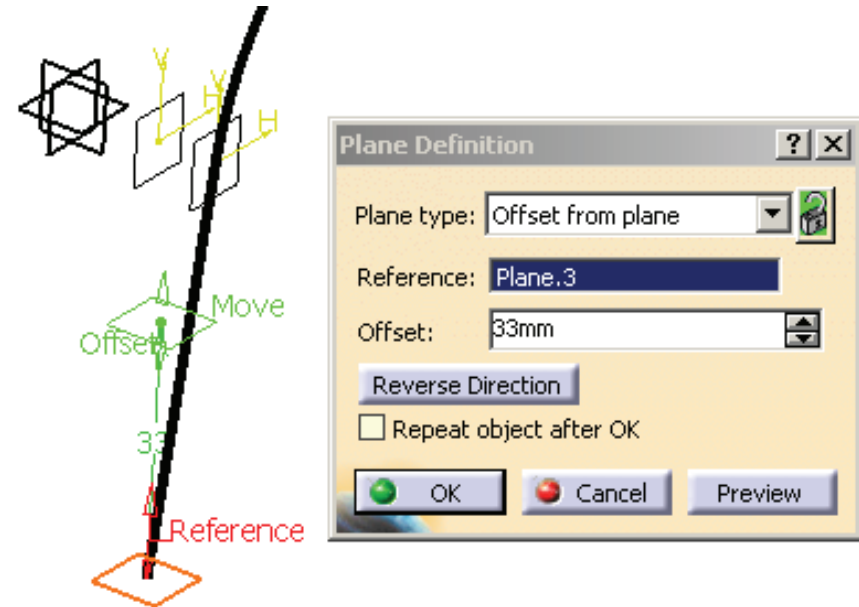
- Click “**plane**” icon
- Select “Parallel through point” as plane type
- Select “xy plane” as Reference
- Select the endpoint of Sketch.1 ★
- Click ok to complete



Tutorial 3B

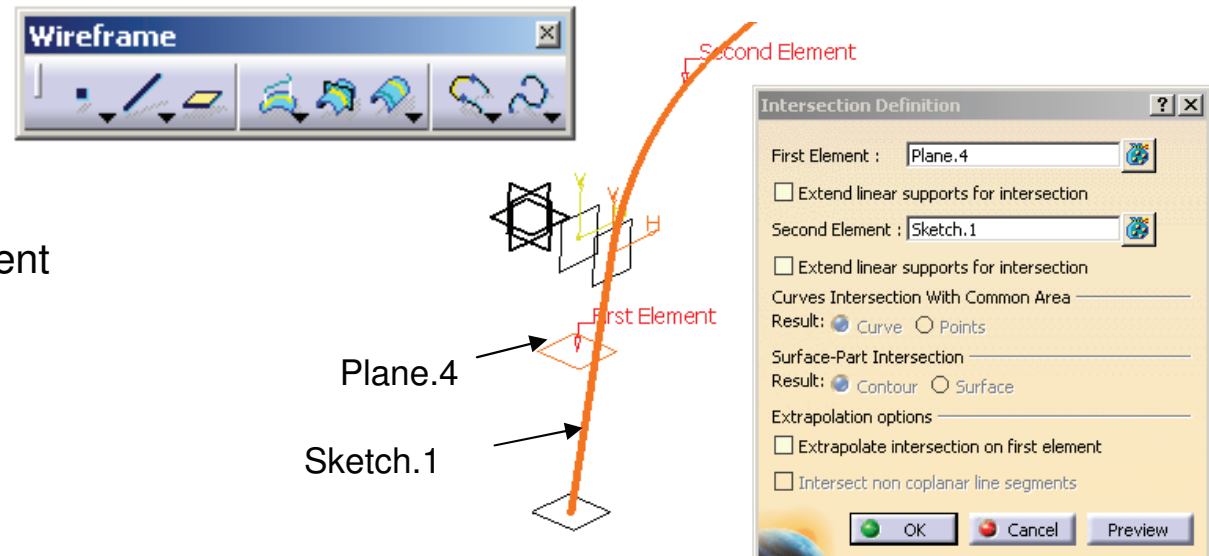
To create a reference plane:-

- Click “**plane**” icon
- Select “Offset from Plane” as plane type
- Select “plane.3” as Reference
- Enter 33mm as Offset
- Click ok to complete



To create an intersection point:-

- Click “**Intersection**” icon
- Select “Plane.4” as First Element
- Select “Sketch.1” as Second Element
- Click ok to complete

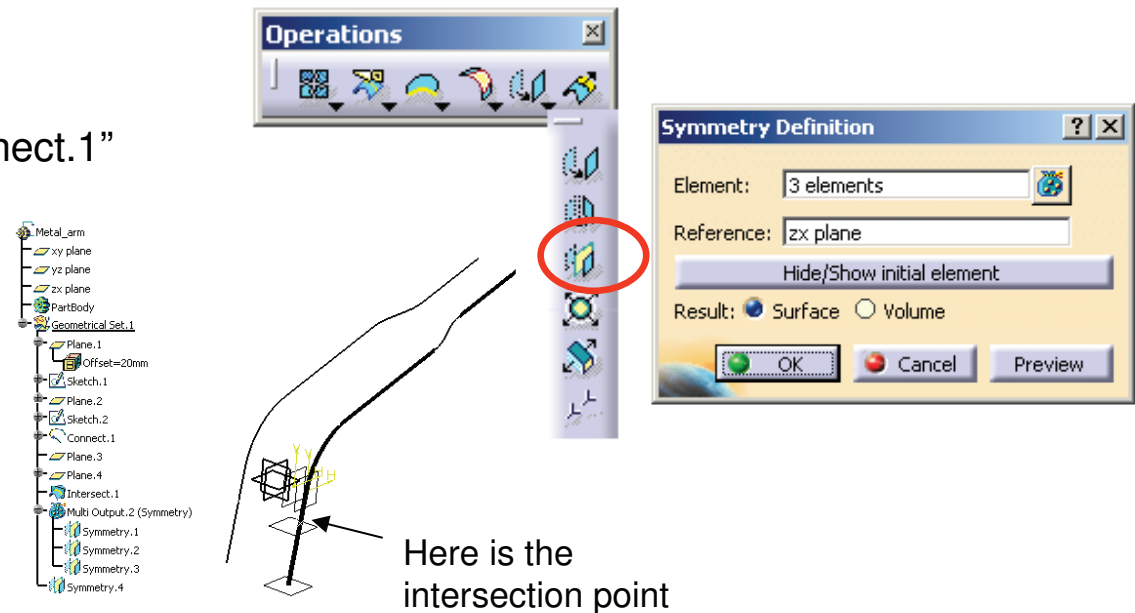


Tutorial 3B

To duplicate elements by mirroring:-

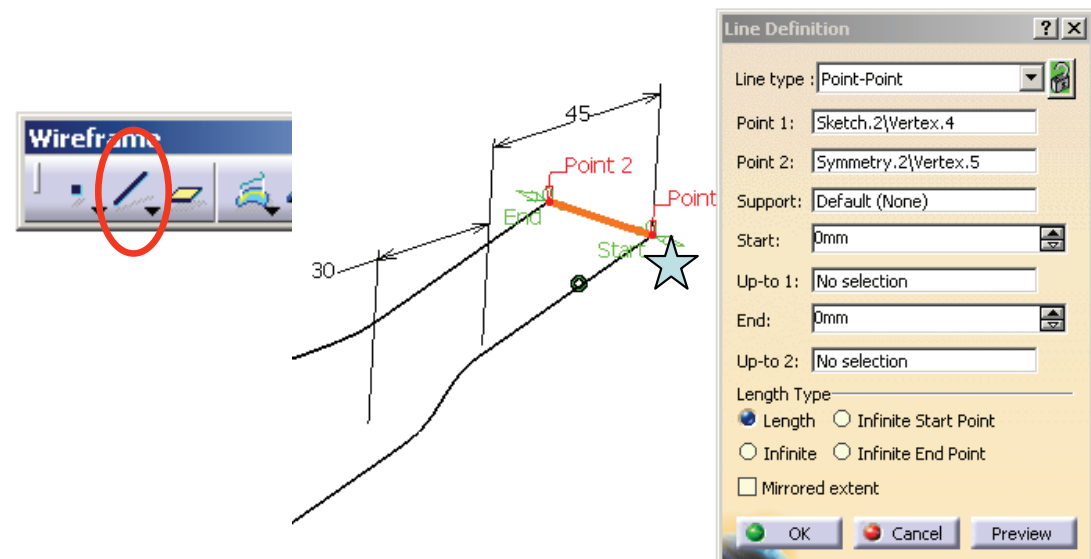
- Multi-select “Sketch.1” , “Sketch.2” & “Connect.1”
- Click “**Symmetry**” icon
- Select “zx plane” as Reference
- Click ok to complete

- Similarly, click “**Symmetry**” icon again
- Select the intersection point as Element
- Select “zx plane” as Reference
- Click ok to complete



To create a line in 3D space:-

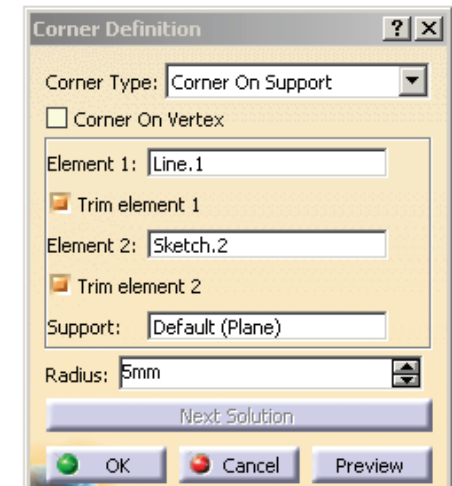
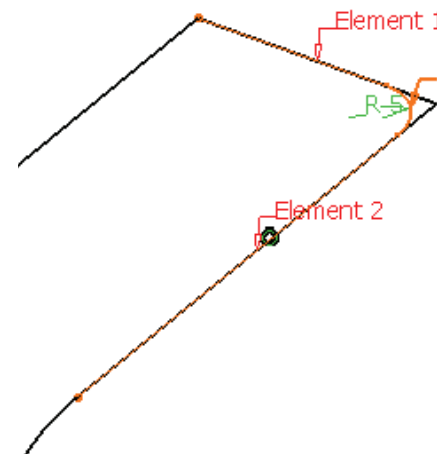
- Click “**Line**” icon
- Select the Endpoint of Sketch.2 ★
- Select the Endpoint of its mirrored image
- Click ok to complete



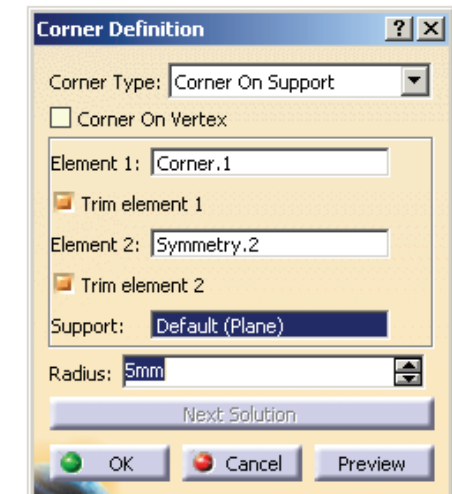
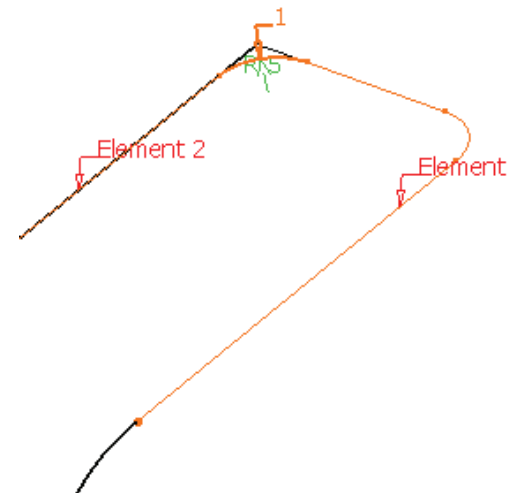
Tutorial 3B

To create a round corner between two lines:-

- Click “**Corner**” icon
- Select “Line.1” as Element 1
- Select “Trim element 1”
- Select “Sketch.2” as Element 2
- Select “Trim element 2”
- Enter 5mm as Radius
- Click ok to complete



- Similarly, click “**Corner**” icon again
- Select “Corner.1” as Element 1
- Select “Trim element 1”
- Select “Symmetry.2” as Element 2
- Select “Trim element 2”
- Enter 5mm as Radius
- Click ok to complete

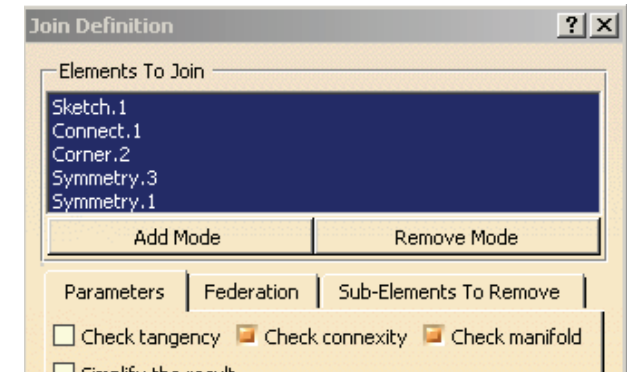
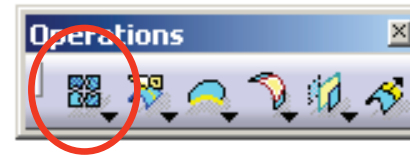


A- 70

Tutorial 3B

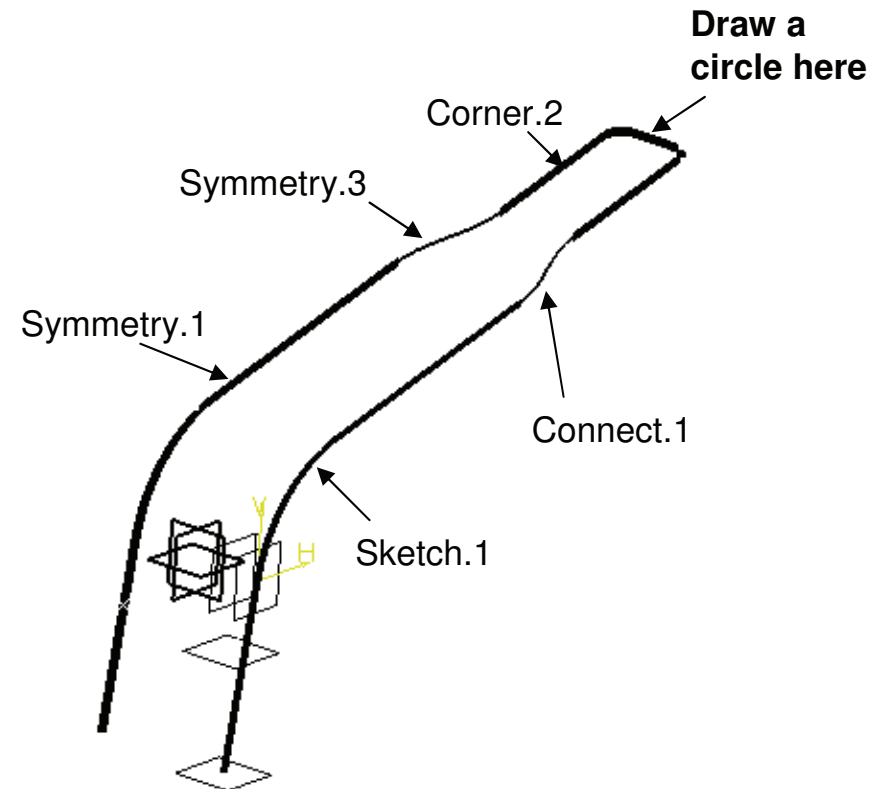
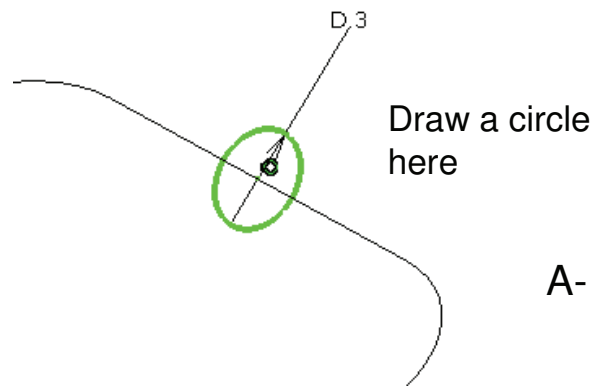
To group all lines & curves:-

- Click **“Join”** icon
- Select all lines & curves in the screen, which includes: “Sketch.1”, “Connect.1”, “Corner.2”, “Symmetry.3”& “Symmetry.1”
- Click ok to complete



To make a sketch:-

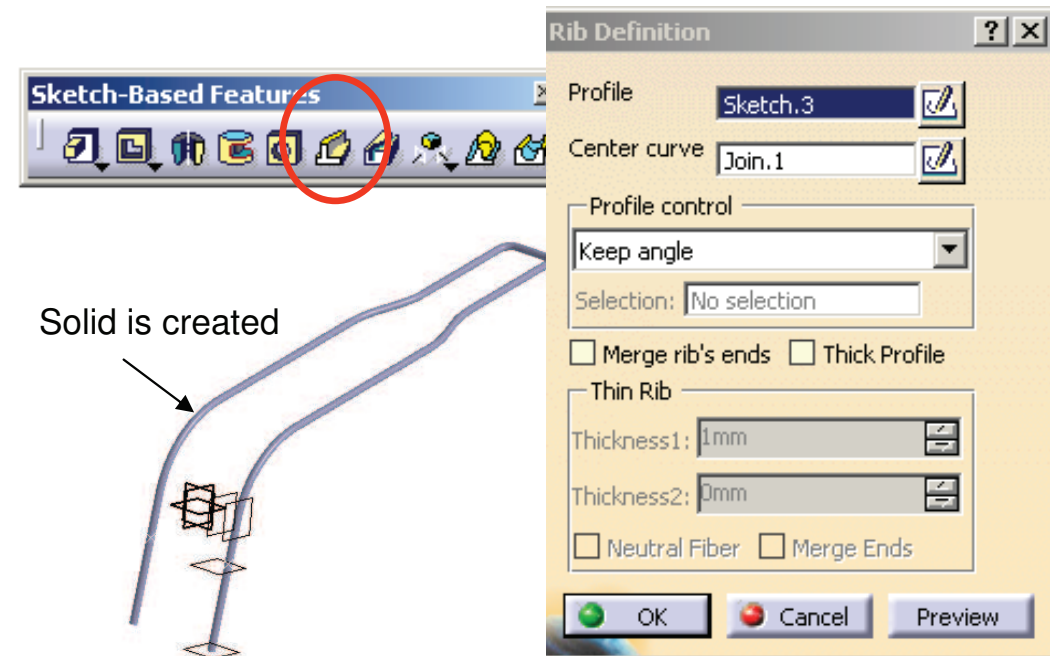
- Click **“Sketch”** icon and select zx plane
- Draw a circle (Dia 3.0)
- Add a coincidence constraints between the circle center and the line
- Exit the workbench by clicking **“Exit”** icon



Tutorial 3B

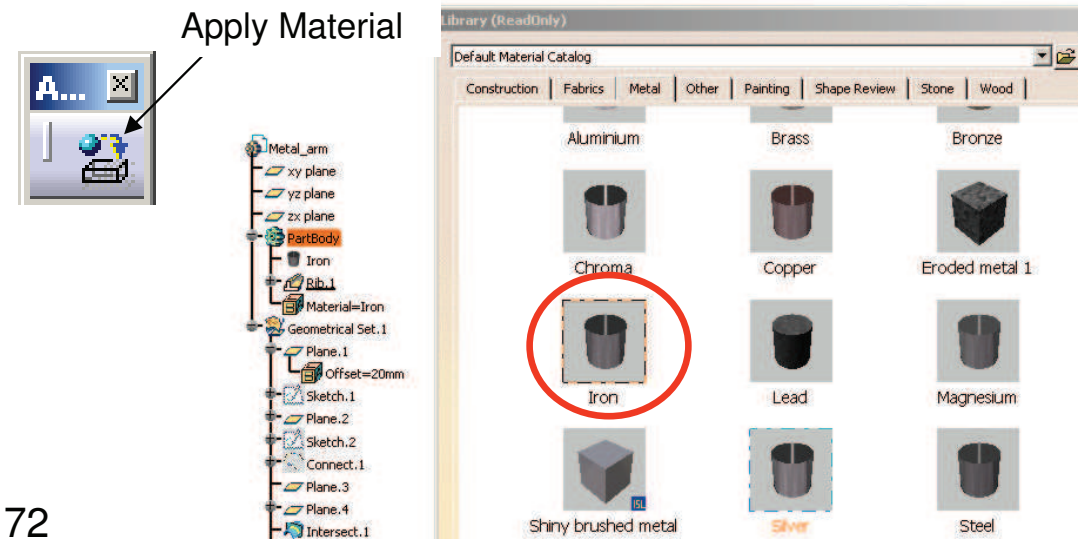
To make a solid:-

- Select '**Start/Mechanical Design/Part Design**' on the menu bar to go back to solid-modeling environment
- Click "**Rib**" icon
- Click ok on the warning window
- Select "Sketch.3" as Profile
- Select "Join.1" as Center Curve
- Click ok to complete



To add material texture:-

- Click "**Apply Material**" icon
- Select "Iron" in the tab page "Metal"
- Select "PartBody" on the tree
- Click ok to complete



To save the new part in Project Folder:-

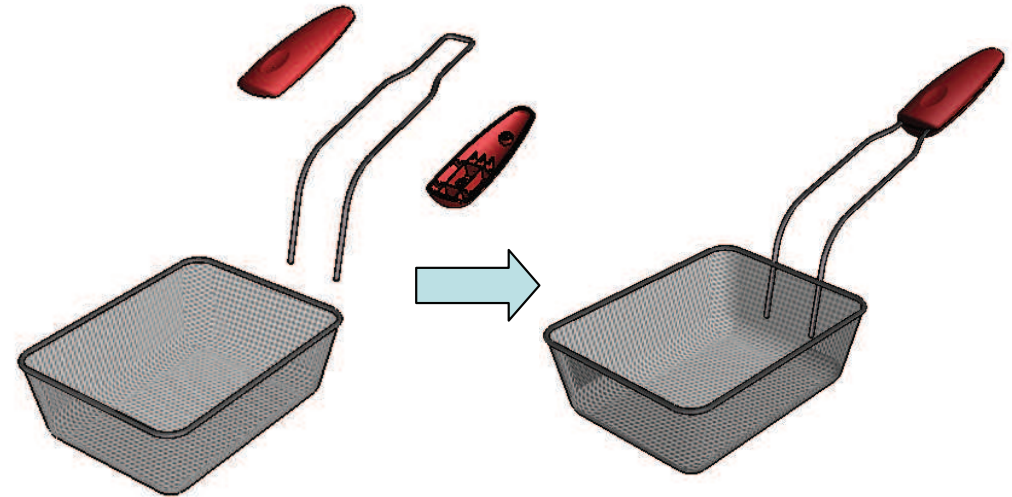
- Save your current part as "**Metal_arm_a.CATPART**" into the folder.

Tutorial 3B

Assemble components together...

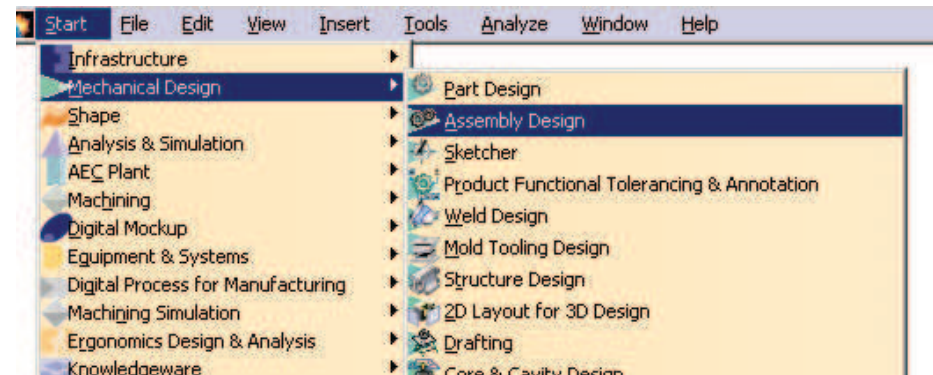
In the folder, you should have five part files;

- Master_handle_a.CATPART
- Upper_body_a.CATPART
- Lower_body_a.CATPART
- Basket_a.CATPART
- Metal_arm_a.CATPART



To go to a new Workbench:-

- Select '**Start/Mechanical Design/Assembly Design**' on the menu bar.
- You may need to reset the layout of the toolbars if the workbench isn't tidy.



To rename the tree:-

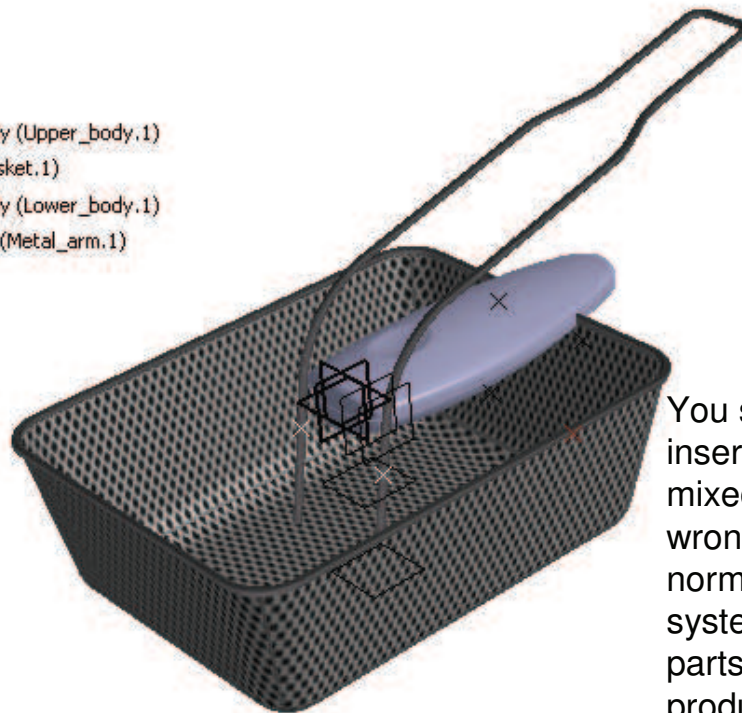
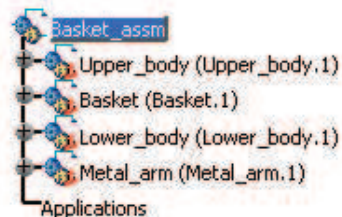
- Single-click "Product1" on the tree, right-click it, and then select "**Properties**".
- Modify Part Number as "Basket_assm" on the tab page "Product".
- Select "ok" to exit .



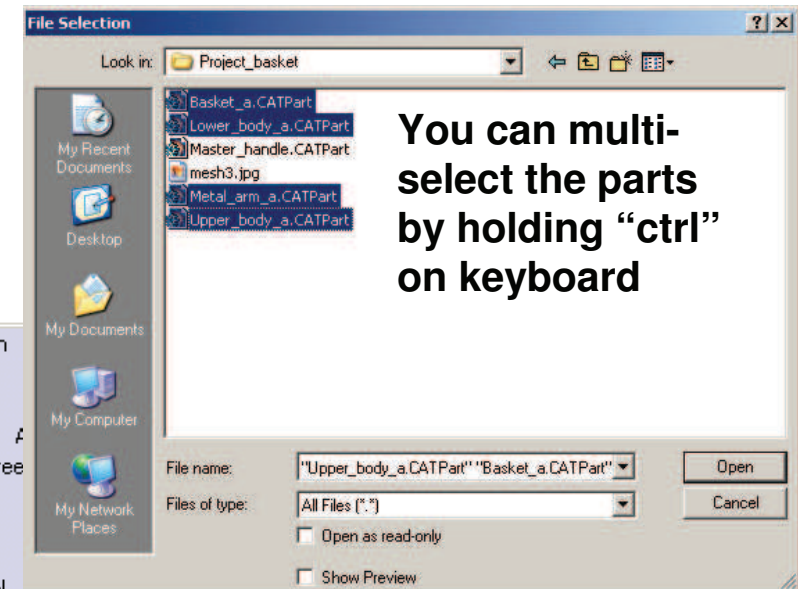
Tutorial 3B

To insert existing parts into a product:-

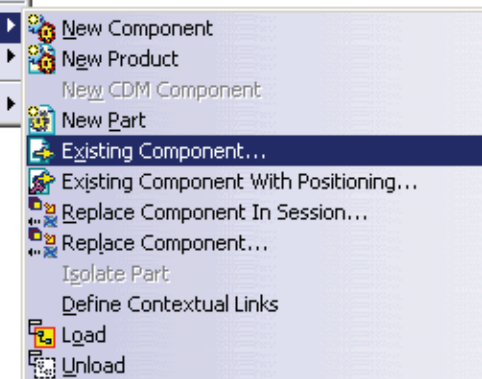
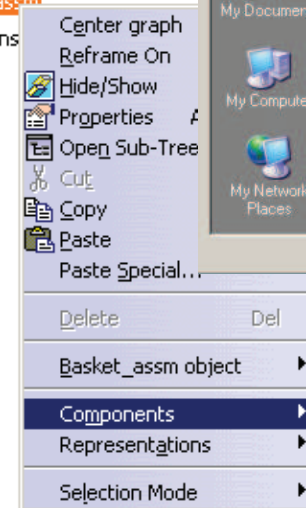
- Right-click the highest icon of the Product tree “Basket_assm”, then select **“Components/Existing component...”**
- Multi-Select all part files except “Master_handle_a.CATPART”
- Click “open”



You should see all inserted parts are mixed together and at wrong positions. It is normal because the system puts all the parts' origins onto the product's origin.



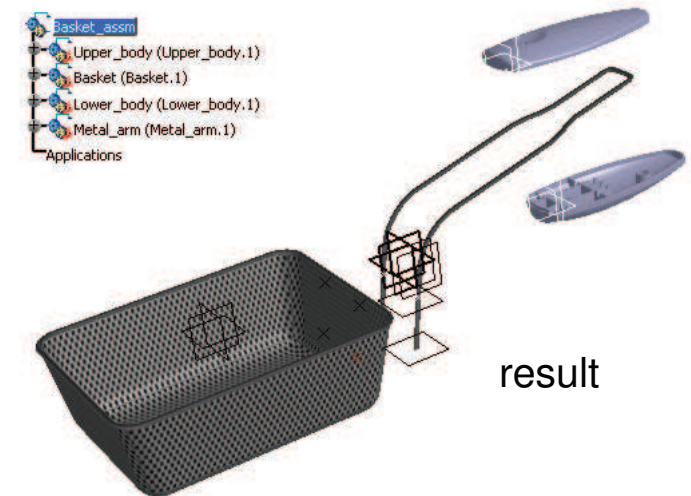
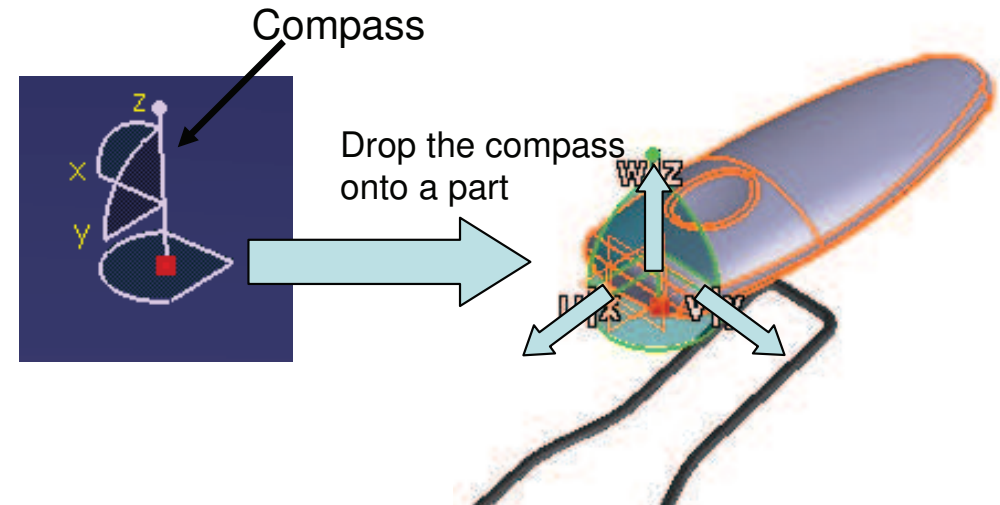
You can multi-select the parts by holding “ctrl” on keyboard



Tutorial 3B

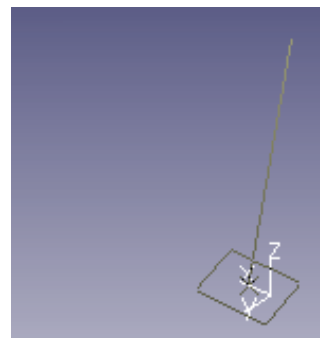
To move a part by “Compass”:-

- Click and hold the **RED** dot of the compass
- Drag it onto the part that you want to move
- The compass will then turn into green and its axis labels will be v-u-w
- Drag along the green lines/arcs of the compass to move the part to a desired position
- Repeat the steps so that all parts are **NEARLY** at desired positions
- Now the parts are separated. It is easier for us to select part features later



To reset “Compass” as original:-

- Click and hold the red dot of the compass.
- Drag it onto the coordinate system at lower right-hand corner of the window and then release.
- It will be auto-reset.



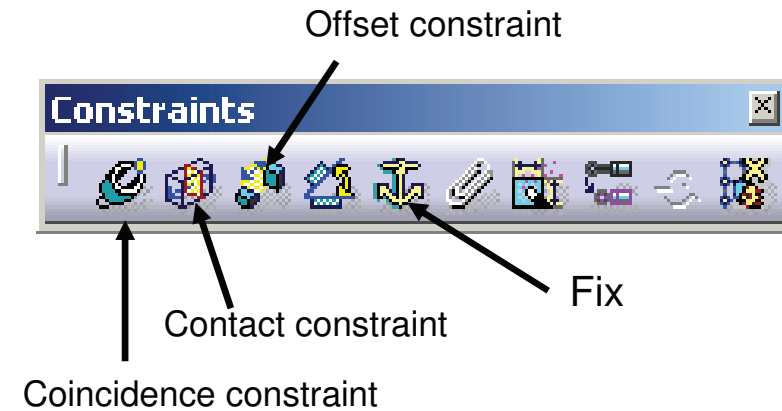
A- 75

Tutorial 3B

To assemble parts by adding constraints:-

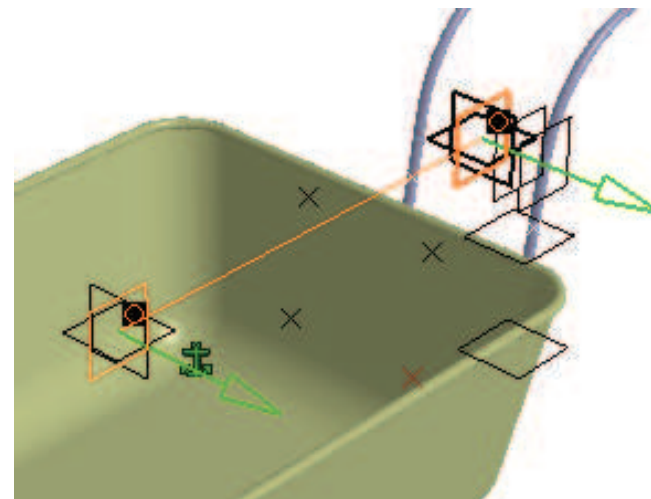
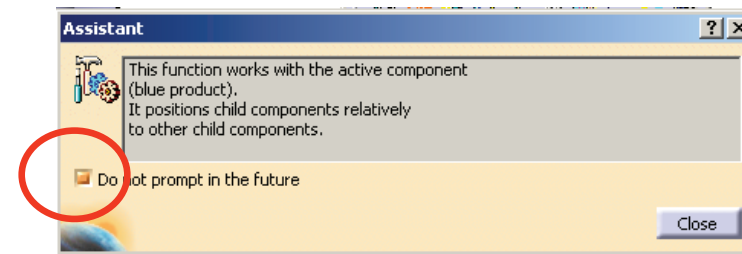
(1) Fix “Basket” in space

- Click “**Fix**” icon
- Select “Basket” on tree; Now the part “Basket” is fixed in position.



(2) Link “Metal Arm” to “Basket”

- Click “**Coincidence Constraint**” icon
- Check “Do not prompt in future” and click “close” to close the message box.
- Select zx plane of Metal Arm
- Select zx plane of Basket
- Click ok to complete

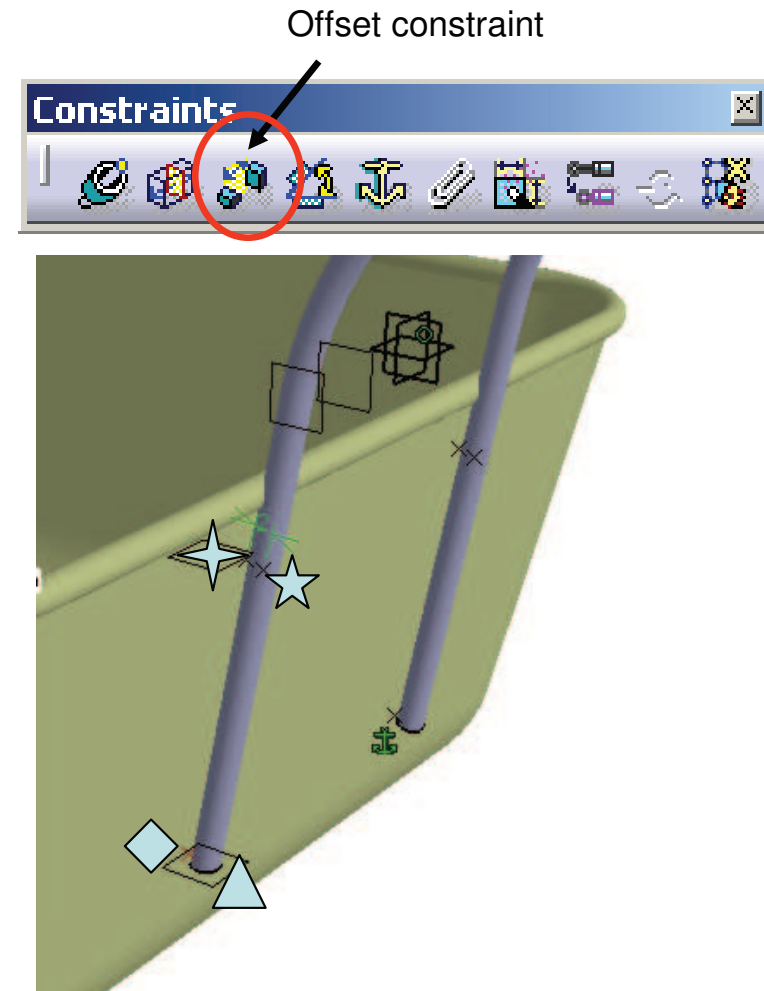


(If you want to delete a constraint, just click the constraint either on the model or on the tree, and then press “Delete” key on keyboard.)

Tutorial 3B

(Cont')

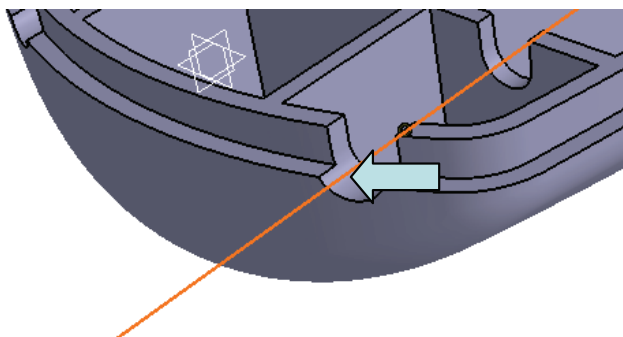
- Click “**Offset Constraint**” icon
- Select the point of Metal Arm ★
- Select the point of Basket ★
- Enter 2mm as Value
- Click ok to complete
- Click “**Update**” Icon  to update the position.
- **UNHIDE** the curve “Join.1” of Metal Arm
- Similarly, Click “**Offset Constraint**” icon again
- Select the point of Metal Arm ▲
- Select the point of Basket ◆
- Enter 2mm as Value
- Click ok to complete
- Click “**Update**” Icon  to update the position.
- **HIDE** the curve “Join.1” of Metal Arm again



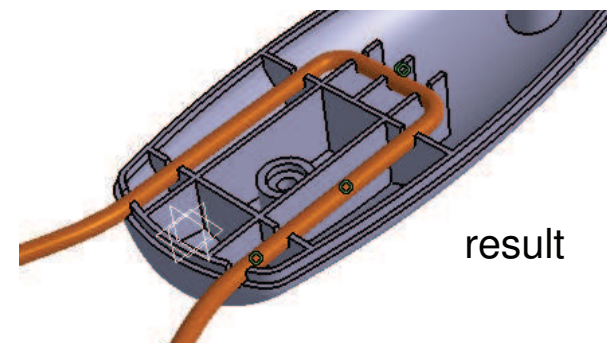
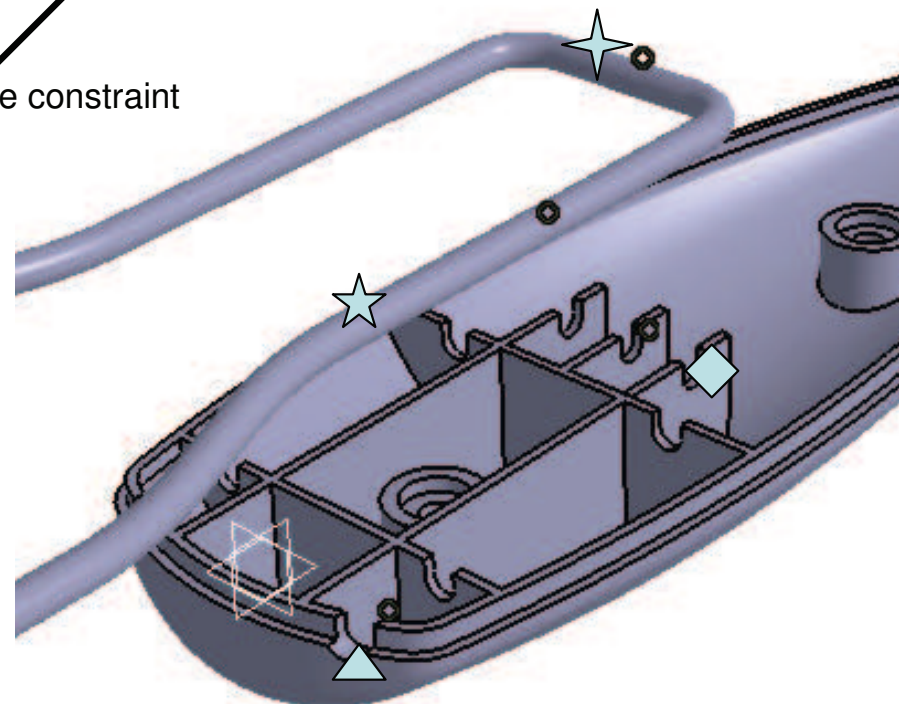
Tutorial 3B

(3) Link “Lower Body” to “Metal Arm”

- Click “**Coincidence Constraint**” icon
- Select the axis of Metal Arm ★
- Select the axis of Lower Body ▲
- Click ok to complete
- Similarly, Click “**Coincidence Constraint**” icon again
- Select the axis of Metal Arm ★
- Select the axis of Lower Body ◆
- Click ok to complete
- Click “**Update**” Icon  to update the position.
- (Remark: when the mouse cursor points on a circular face, the axis can be detected)



Coincidence constraint



result

A- 78

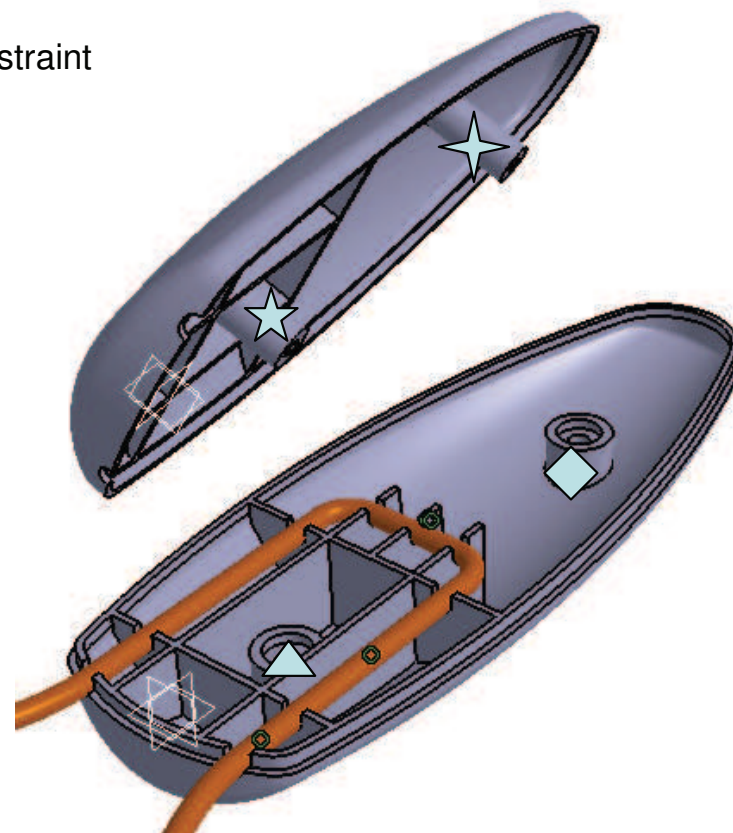
Tutorial 3B

(4) Link “Upper Body” to “Lower Body”

- Click “**Coincidence Constraint**” icon
- Select the axis of Upper Body ★
- Select the axis of Lower Body ▲
- Click ok to complete
- Similarly, Click “**Coincidence Constraint**” icon again
- Select the axis of Upper Body ★
- Select the axis of Lower Body ◆
- Click ok to complete
- Similarly, Click “**Coincidence Constraint**” icon again
- Select xy plane of Upper Body
- Select xy plane of Lower Body
- Click ok to complete
- Click “**Update**” Icon  to update the position.



Coincidence constraint



Tutorial 3B

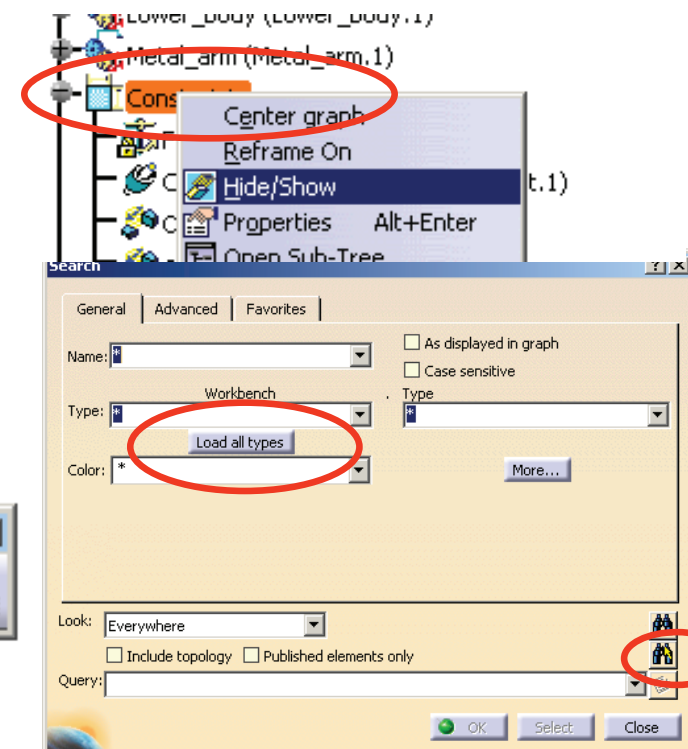
To hide all constraints:-

- Just single-click “Constraints” on the tree and right-click to show the contextual menu; then select “**Hide/Show**”

To hide all datum planes:-

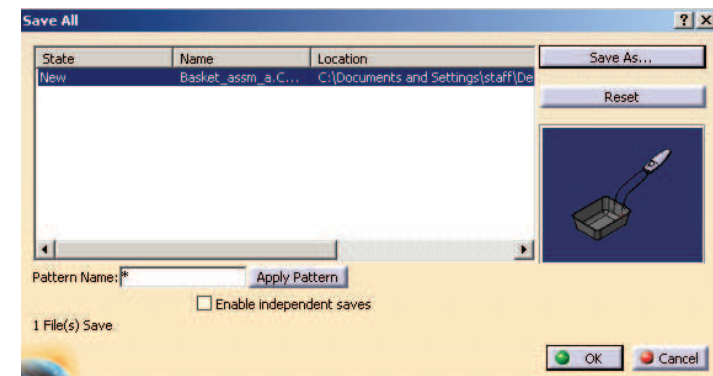
- Select “**Edit/Search..**” on the menu bar and then click “Load all type” icon
- Select “Plane” as Type
- Click “Search & Select” icon
- Click “**Hide/Show**” icon

Hide/Show



To Save all files:-

- Select “**File/Save all**”
- Click OK to close this message box (because you have to define the file location of the new Product file)
- Click “Save As...” icon
- Enter “Basket_assm_a.CATProduct” as filename and save it in your project folder.



****CLOSE ALL FILES****