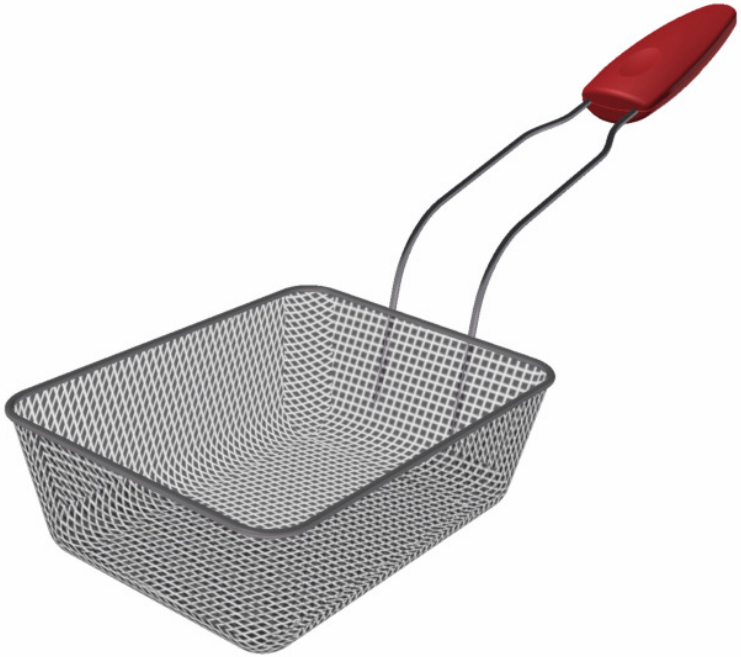
CATIA V5 Design with Analysis

(Tutorial 3 – Deep Fry Basket)



Infrastructure

Sketcher

Part Design (Solid-modeling)

GSD (Surface-modeling)

Assembly Design

Generative Structural Analysis

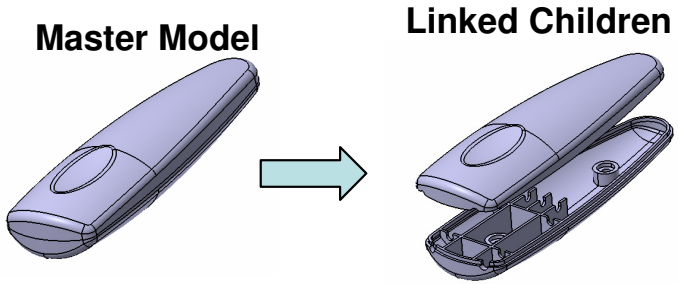
Product Engineering Optimizer

By Dickson Sham (ME Dept, HKPU)

Overview of Tasks

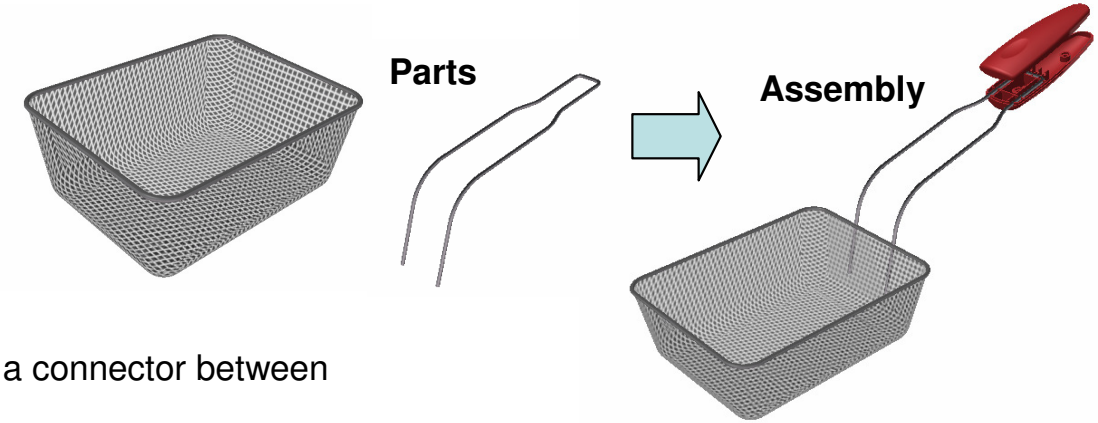
Tutorial 3A - Modeling

- Build a Master Model of the basket handle
- Create the upper & the lower parts from the Master Model
- Build the mechanical features on the both parts
- Get the both parts auto-updated after modifying the outlook of the master model



Tutorial 3B - Modeling

- Build the metal arm
- Build the basket
- Add material texture onto all components
- Assemble components



Tutorial 3C – Structural analysis

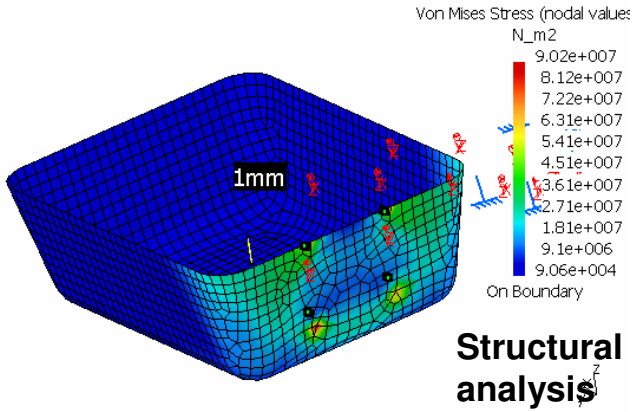
- Simplify the model for analysis
- Create Meshes onto two components and create a connector between them
- Create boundary conditions & define properties
- Analyze displacements & stresses

Tutorial 3D – Structural analysis (By Nastran)

- Repeat Tutorial 3C with the use of Nastran

Tutorial 3E – Design optimization

- Create a user parameter “volume”
- Run optimization to get the minimum volume of the metal arm with the smallest part deformation



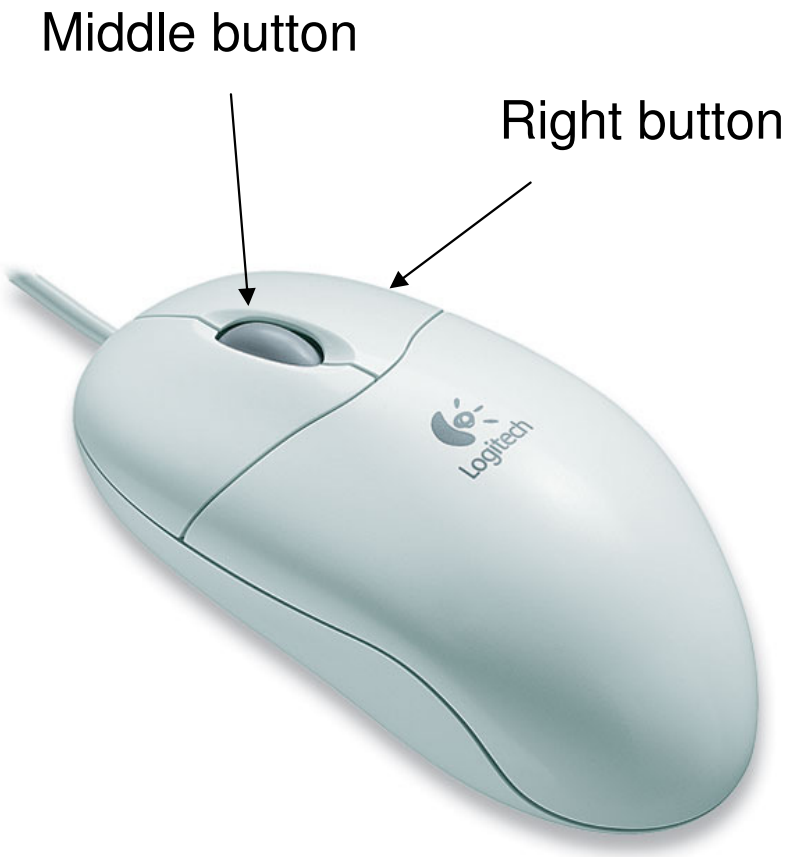
Structural analysis

Change the view with the mouse

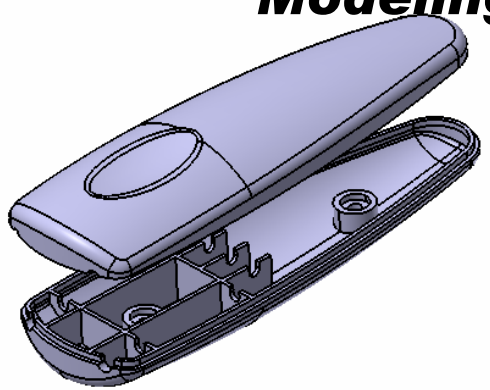
A. ***Panning*** enables you to move the model on a plane parallel to the screen. Click and hold the middle mouse button, then drag the mouse.

B. ***Rotating*** enables you to rotate the model around a point. Click and hold the middle mouse button and the right button, then drag the mouse.

C. ***Zooming*** enables you to increase or decrease the size of the model. Click and hold the middle button, then click ONCE and release the right button, then drag the mouse up or down.

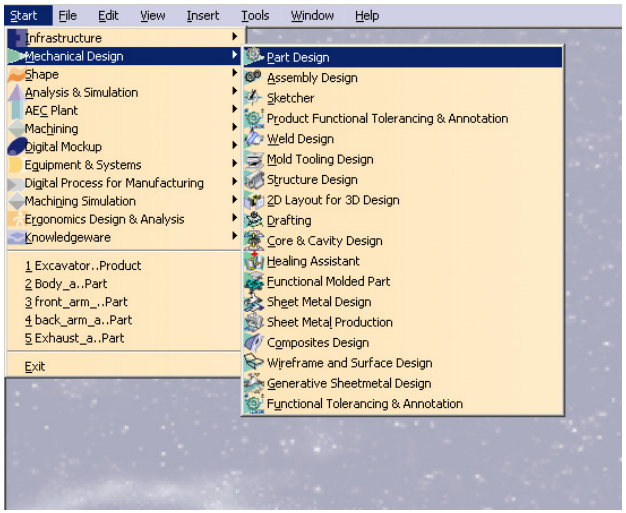


Tutorial 3A

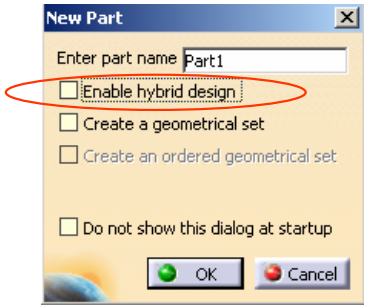


- Enter CATIA by double-clicking its icon on the desktop
- (If a license menu pops up), select **ED2** and close CATIA. Then reopen again
- By default, a empty “Product” file is created. But now, you don’t need this, just select “**File/Close**” on the menu

- Select ‘**Start/Mechanical Design/Part Design**”on the menu bar



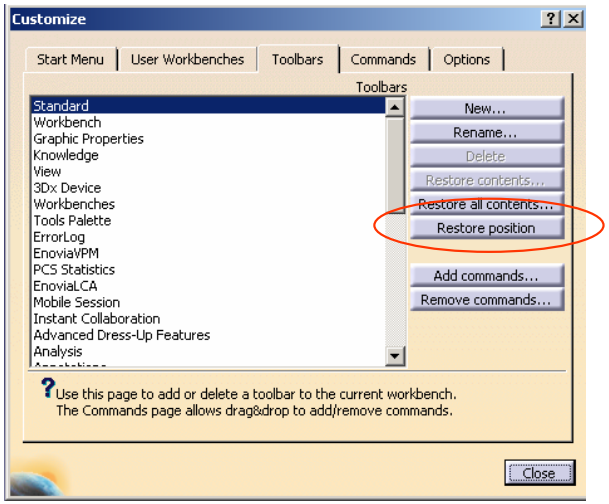
- Uncheck “Enable Hybrid Design” and then click “ok”
- An empty part is now created on “Part Design” workbench. You can see a specification tree at the upper left-hand corner and xyz datum planes in the middle of the screen



Tutorial 3A

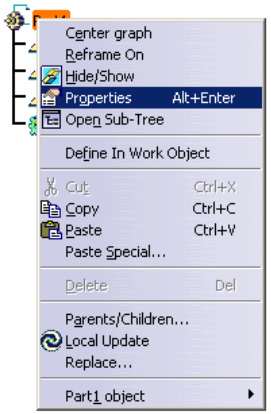
To reset the layout of workbench (optional):-

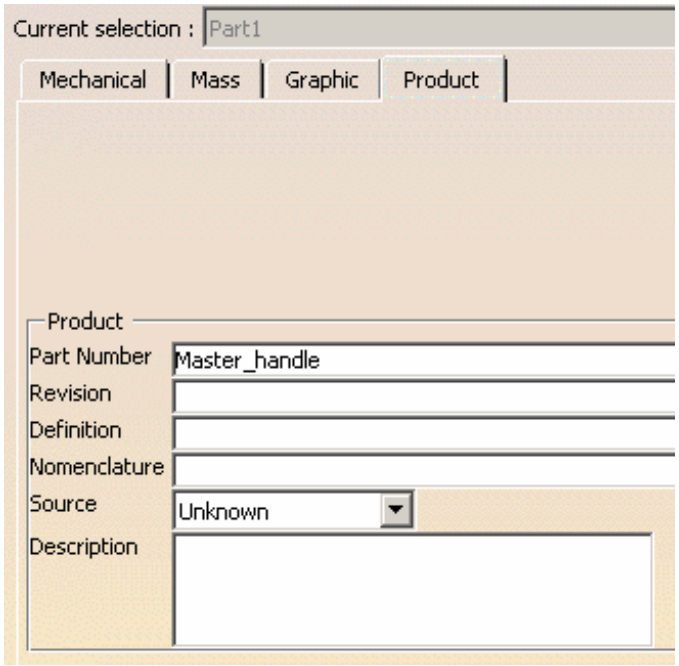
- Sometimes the workbench may not be tidy before you use; some toolbars are missing and some are at wrong positions. To reset the layout, select "**View/Toolbars/Customize**" and select "**Toolbar/restore position**" on the pop-up window; Close and exit



To rename the tree:-

- Single-click "Part1" on the tree, right-click it, and then select "**Properties**"
- Modify Part Number as "Master_handle" on the tab page "Product"
- Select "ok" to exit



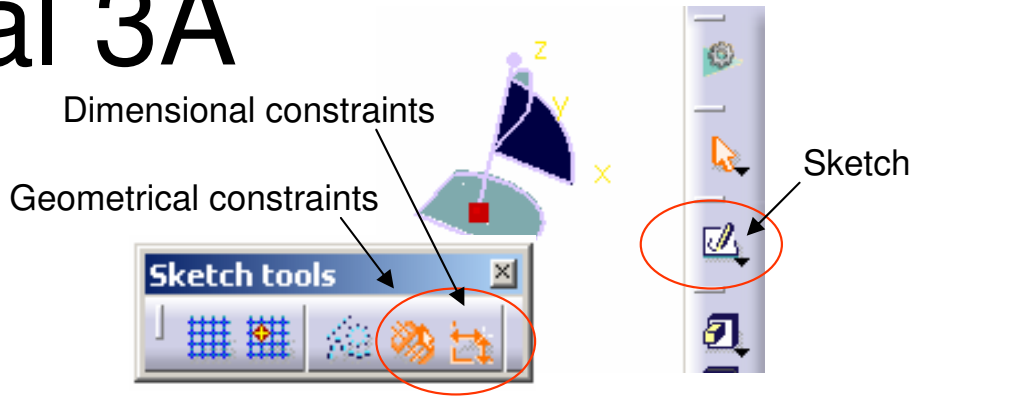


Tutorial 3A

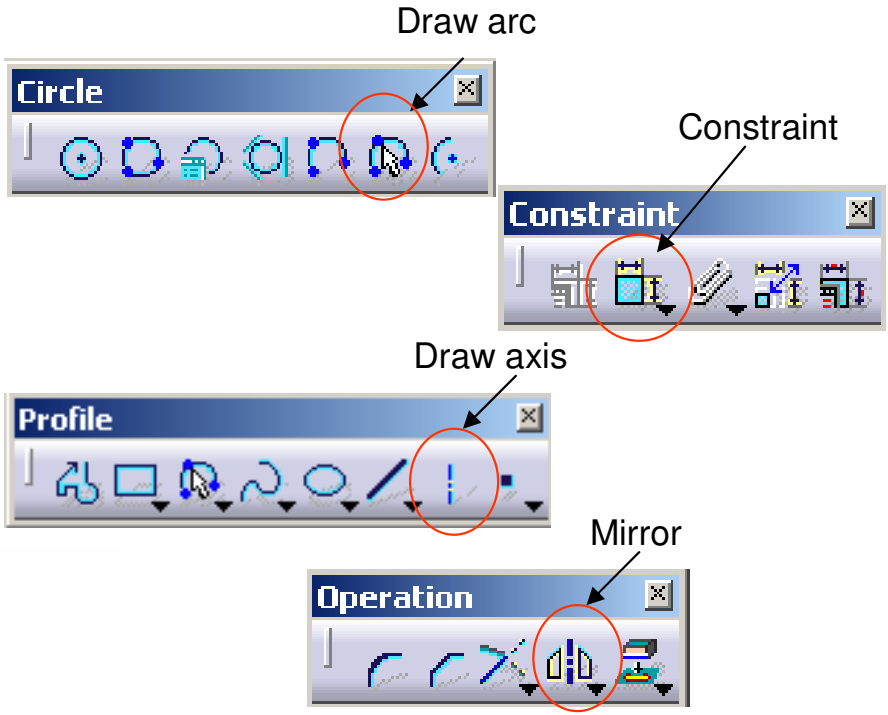
To build 1st sketch:-

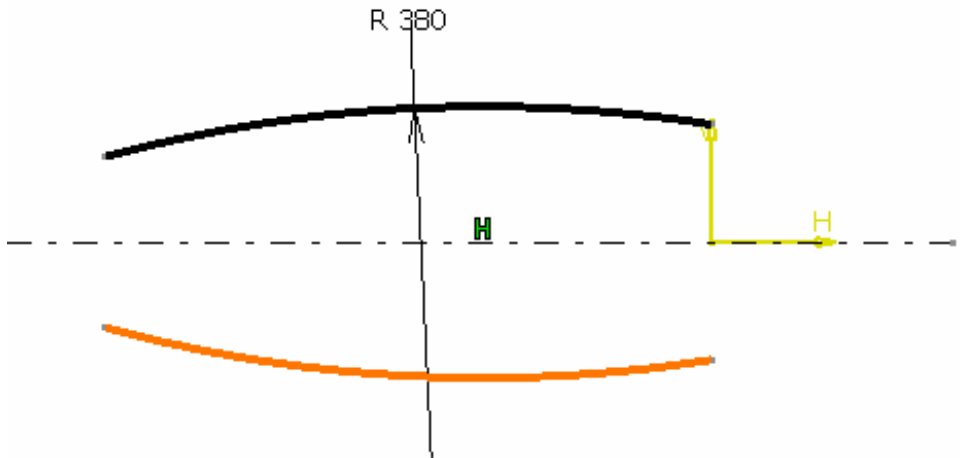
- click "**Sketch**" icon and select **xy plane**

- Ensure "Dimensional Constraints" & "Geometrical Constraints" are activated in the Toolbar "**Sketch tools**"



- **Draw** an arc with one end touching y-axis
- Add a dimension **constraint** onto the circle by clicking "constraint" icon and then selecting the circle.
- Double-click on the dimension and **modify** the radius as 380mm; the arc will be resized automatically
- **Draw** an **axis** along the x-axis
- Click **"Mirror"** icon and then select the arc and the axis to duplicate the arc on the opposite side





Tutorial 3A

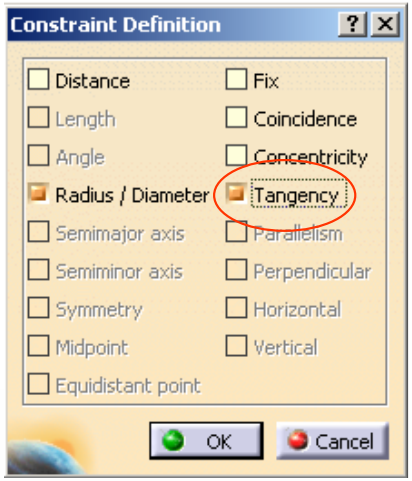
To build 1st sketch (Cont'):-

- **Draw** an **arc** (R35) connecting the bigger arcs , on the positive-x side, which is NOT tangent to them
- Similarly, **Draw** another smaller **arc** (R10) connecting the bigger arcs on the negative-x side

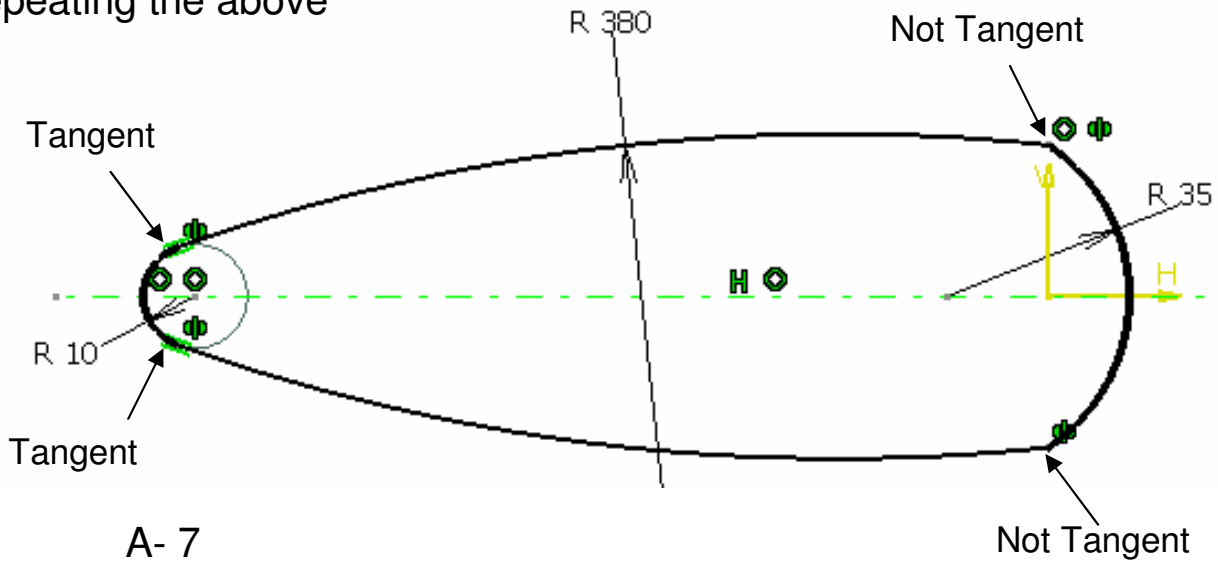
Draw arc

- **Multi-select** the arc (R10) and the upper arc(R380) by pressing and holding "**ctrl**" key on the keyboard
- Then select "**Constraints defined in dialog box**" icon
- Select "**Tangency**" and "ok"

Constraints defined in dialog box



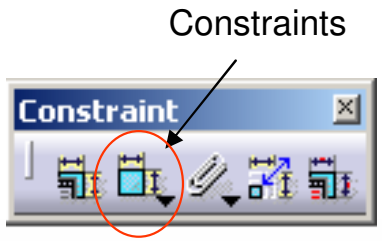
- Add another Tangency constraint between the arc R10 and the lower arc(R380) by repeating the above steps



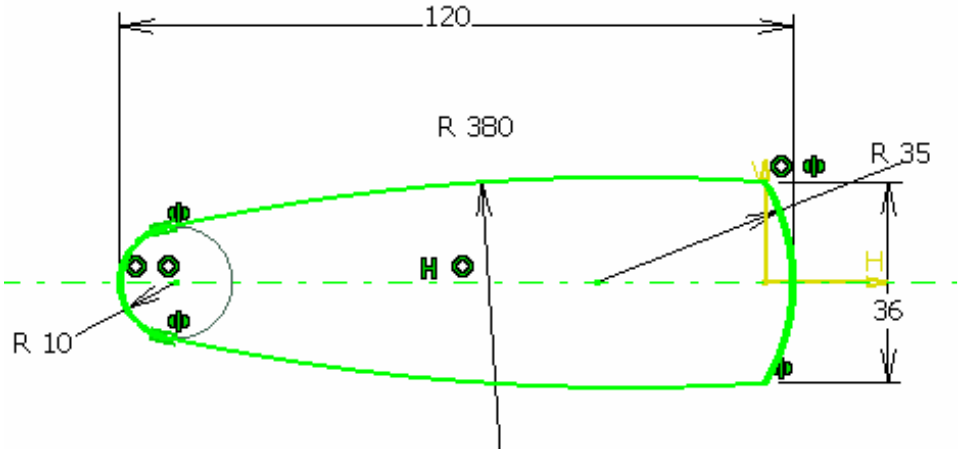
Tutorial 3A

To build 1st sketch (Cont'):-

- Click "**Constraint**" icon and then select the arcs R10 & R35
- Modify the value to 120mm

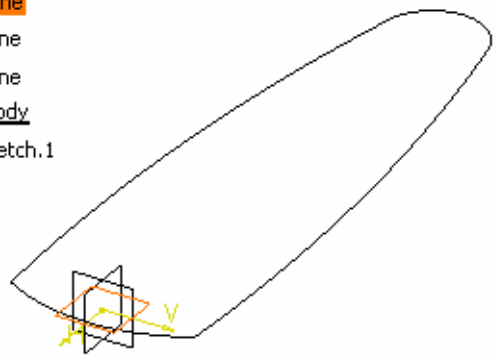


- Similarly, Click "**Constraint**" icon and then select the end points of arc R35
- Modify the value to 36mm
- (After that, the sketch color should become green since it has been fully constrained.)



- Exit the workbench by clicking "**Exit**" icon

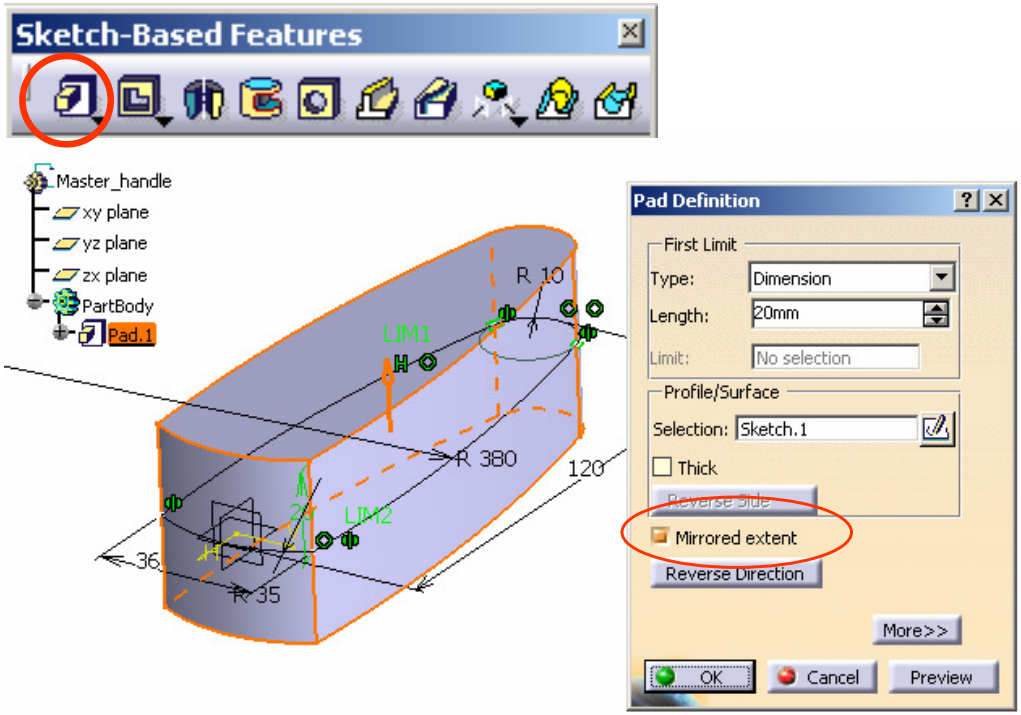
- Now, you are back to Part Design Workbench (3D environment) and "Sketch.1" is created on the tree.



Tutorial 3A

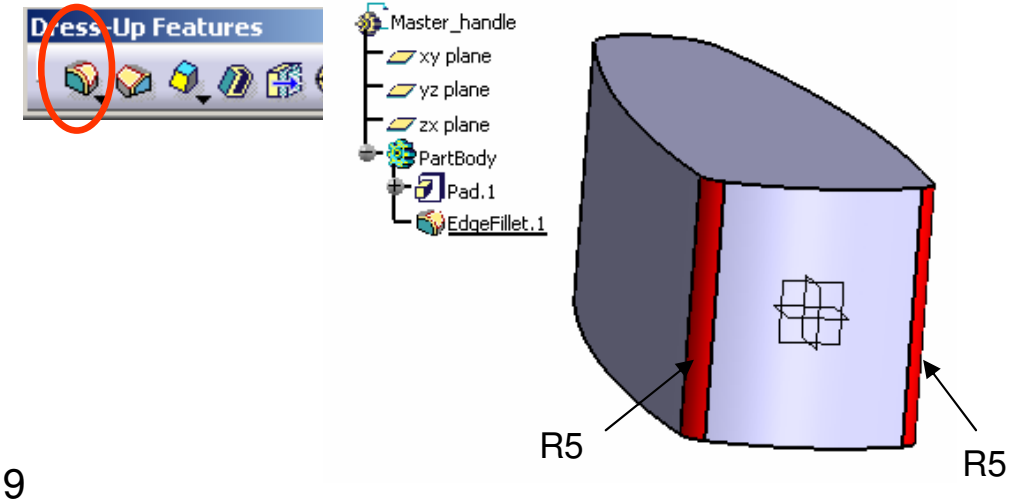
To build a solid:-

- Select “Sketch.1” on the tree / directly click on the geometry
- Click “**Pad**” icon
- Enter 20mm as the length of First Limit
- Select “Mirror extent”
- Click “ok”
- A solid is created



To round the sharp edges:-

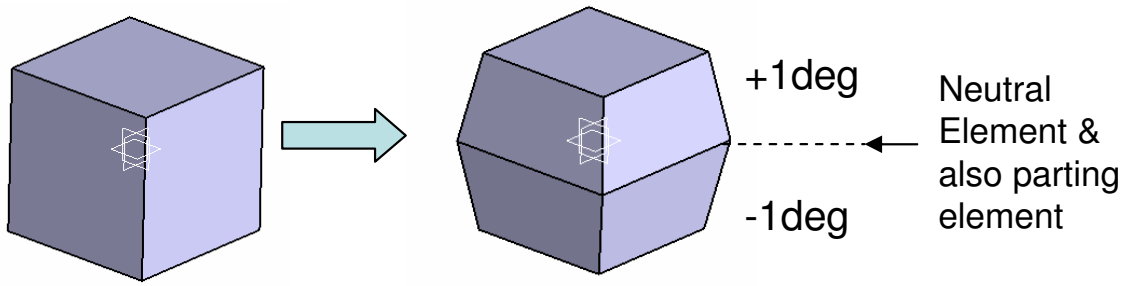
- Add a “**Edge Fillet**” R5mm onto the vertical sharp edges

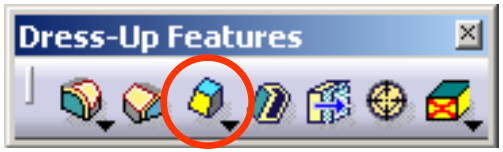


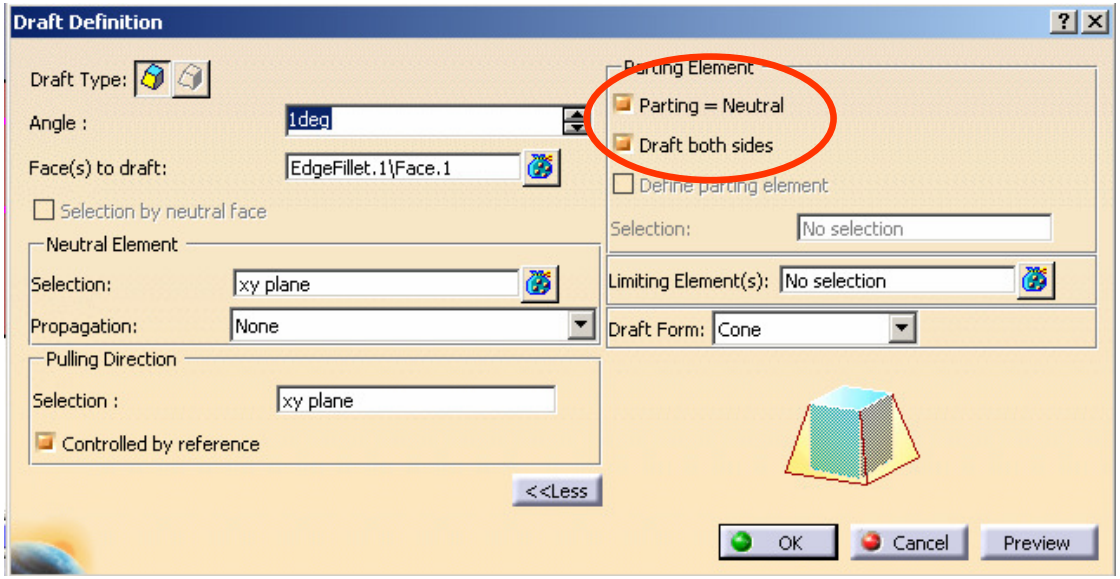
Tutorial 3A

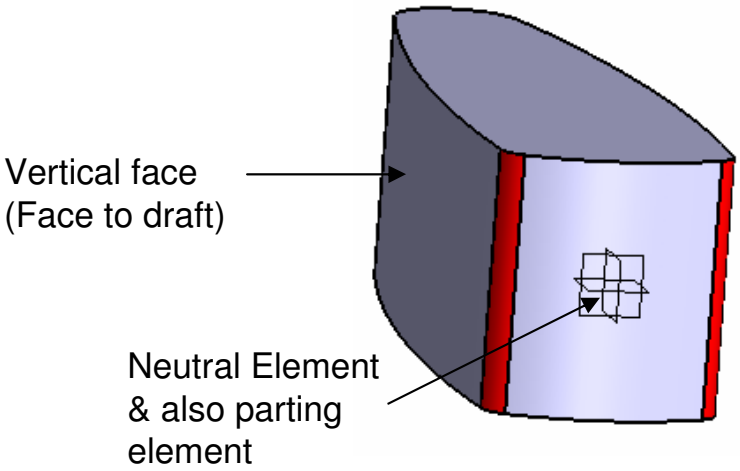
To draft both sides of the solid:-

- Click "**Draft angle**" icon
- Enter 1deg as Angle
- Select a vertical face as "Face to draft" (after that, all tangent faces will be automatically selected and turn red)
- Click the selection box of "Neutral element" and then select xy plane
- Click "More"
- Check "Parting=Neutral"
- Check "Draft both sides"
- Click ok to complete







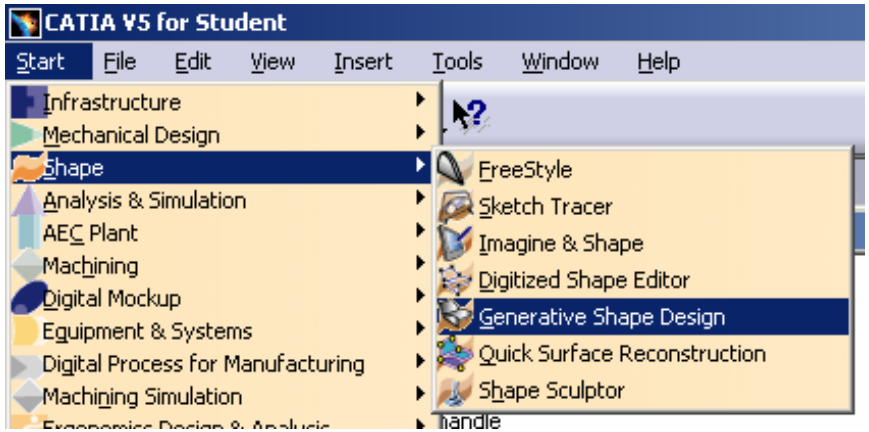


Tutorial 3A

To change the workbench:-

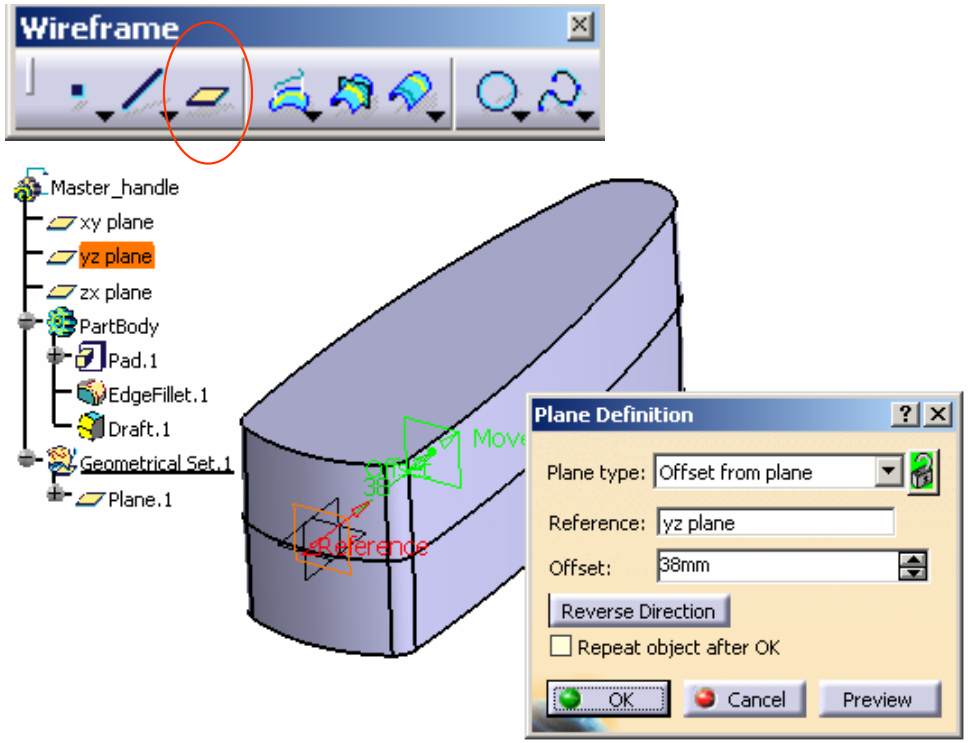
- Select '**Start/Mechanical Design/Generative Shape Design**"on the menu bar

- 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)



To create a reference plane:-

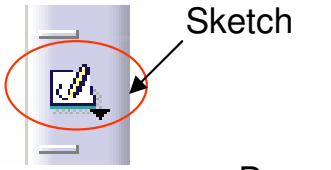
- Click "**plane**" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window
- Enter 38mm as Offset value
- Click ok to complete

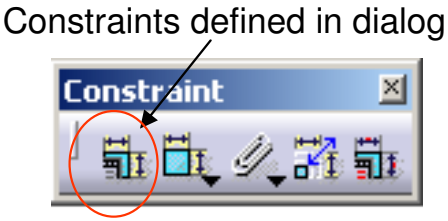


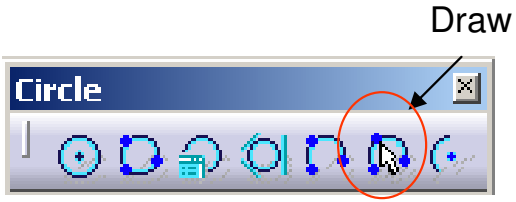
Tutorial 3A

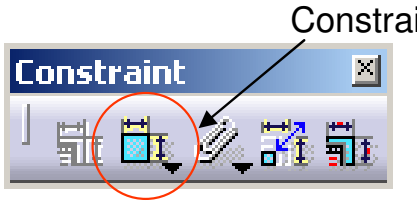
To build 2nd sketch:-

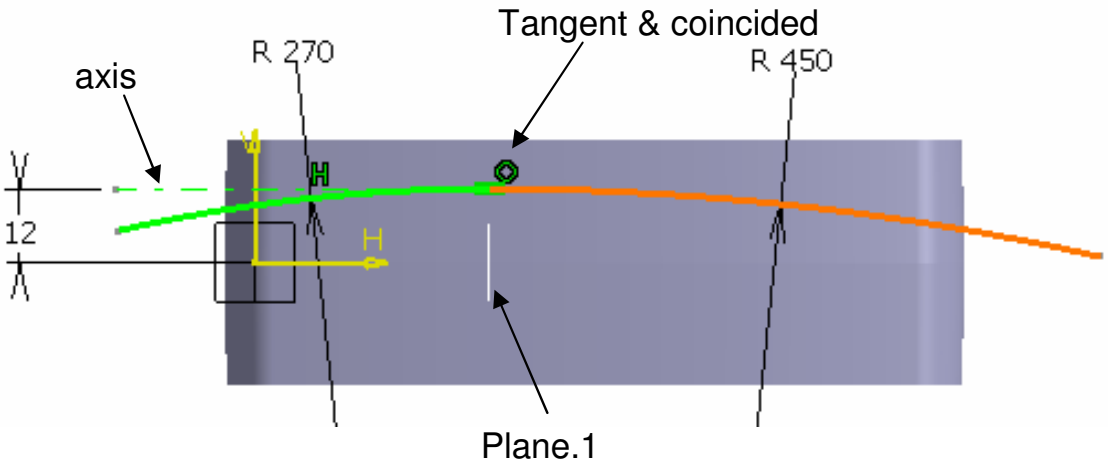
- click "**Sketch**" icon and select **zx plane**
- **Draw** a horizontal **axis** as shown
- **Multi-select** the axis and plane.1 by pressing and holding "**ctrl**" key on the keyboard
- Then select "**Constraints defined in dialog box**" icon
- Select "**Coincidence**" and "ok"
- **Draw** another two **arcs** (R450 & R270) and add the corresponding constraints as shown
- Exit the workbench by clicking "**Exit**" icon
- *Click on an empty space to deselect the sketch*







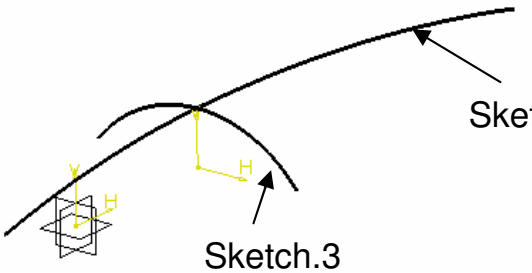


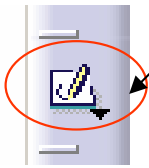


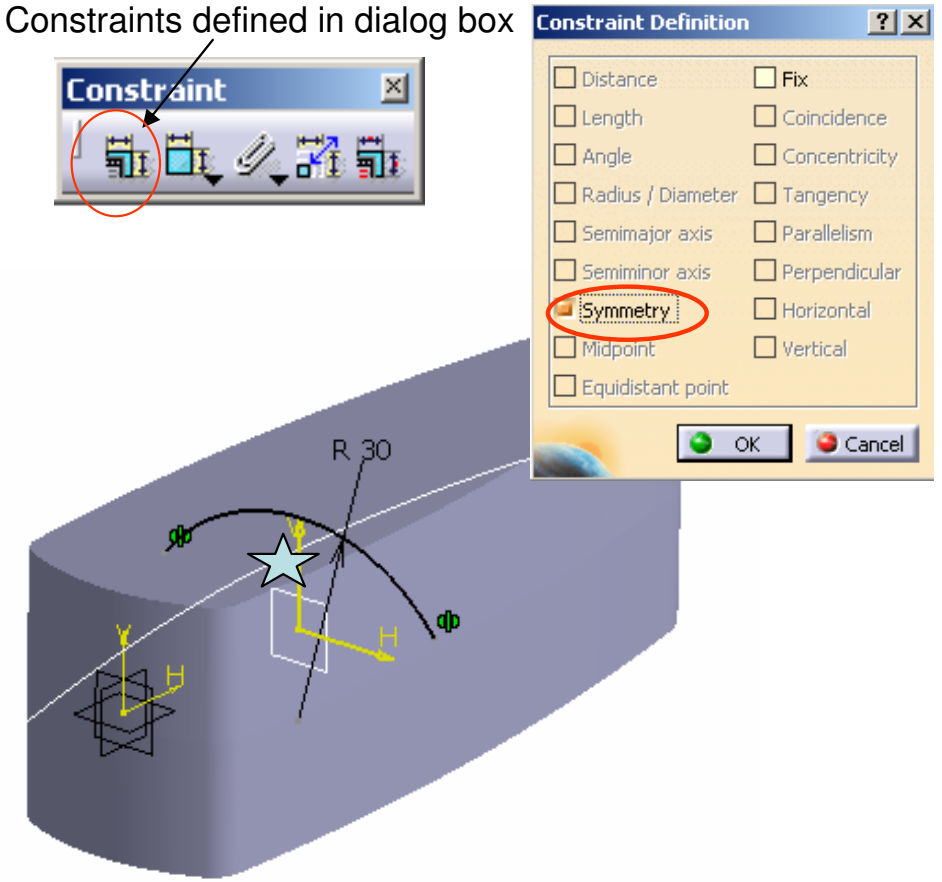
Tutorial 3A

To build 3rd sketch:-

- Click "**Sketch**" icon and select **plane.1**
- **Draw** an **arc** as shown
- **Multi-select** the endpoints then the y-axis by pressing and holding "**ctrl**" key on the keyboard
- Then select "**Constraints defined in dialog box**" icon
- Select "**Symmetry**" and "ok"
- Add Constraint R30 onto the arc
- Rotate the model by mouse to have an isometric view
- **Multi-select** the arc and the point ☆ by pressing and holding "**ctrl**" key on the keyboard.
- Then select "**Constraints defined in dialog box**" icon
- Select "**Coincidence**" and "ok"
- (Now the arc should be coincided with Sketch.2)
- Exit the workbench by clicking "**Exit**" icon



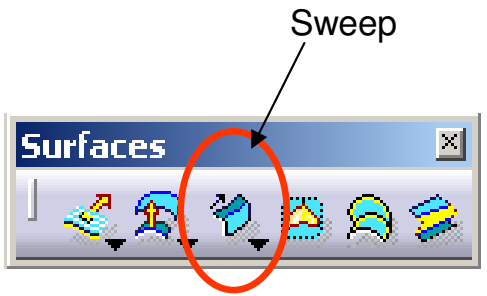


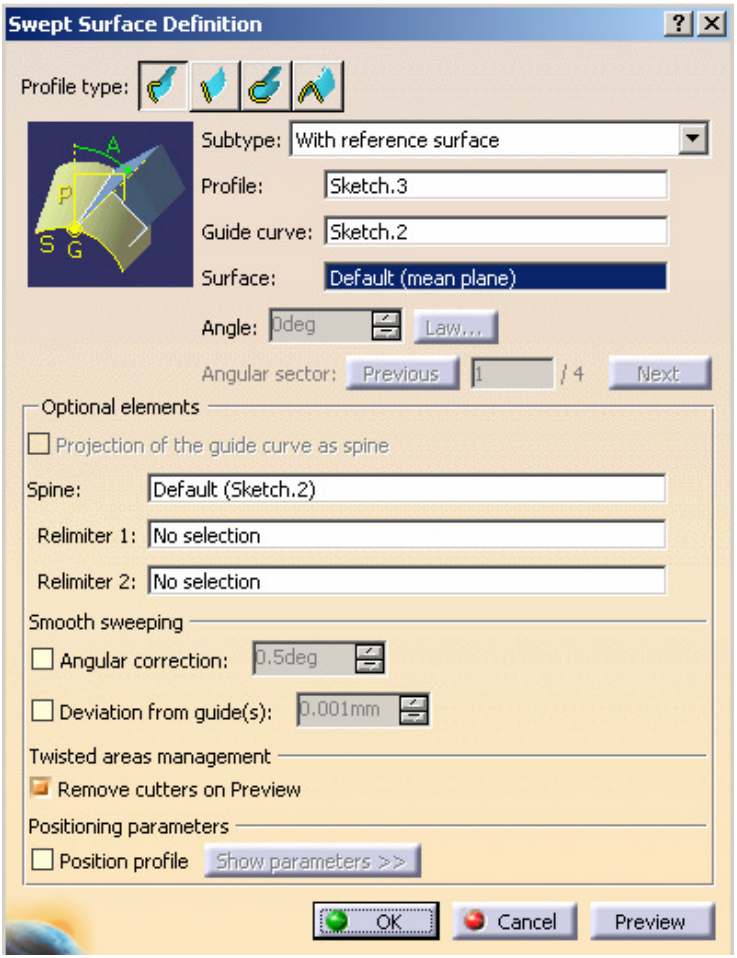


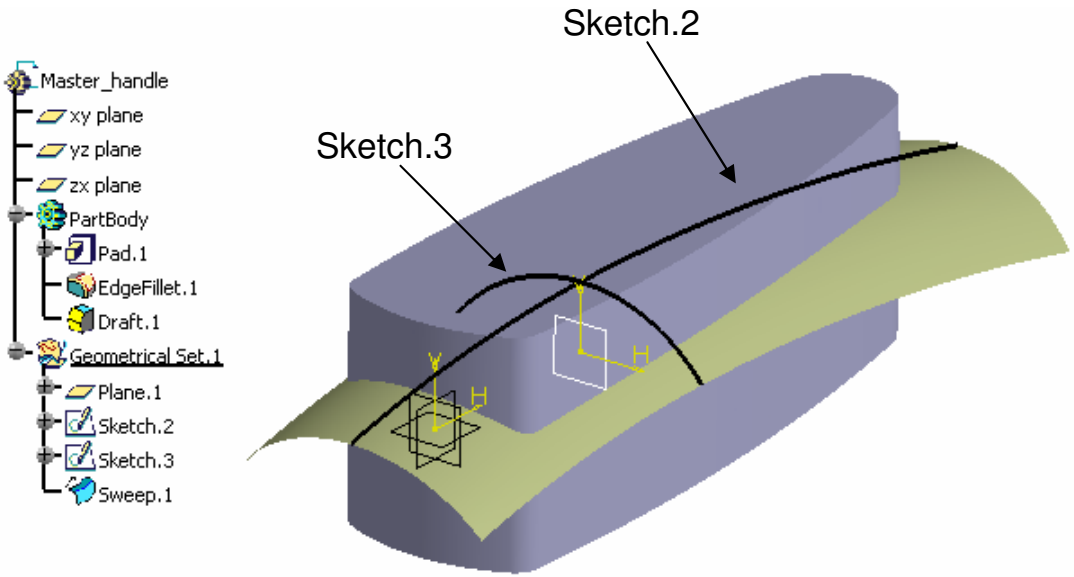
Tutorial 3A

To build a Surface:-

- Click "**Sweep**" icon
- Select "Explicit" as Profile Type
- Select "Sketch.3" as Profile
- Select "Sketch.2" as Guided Curve
- Click ok to complete
- (On the tree, this surface is stored in "Geometrical Set.1", so it will not be mixed with solids.)







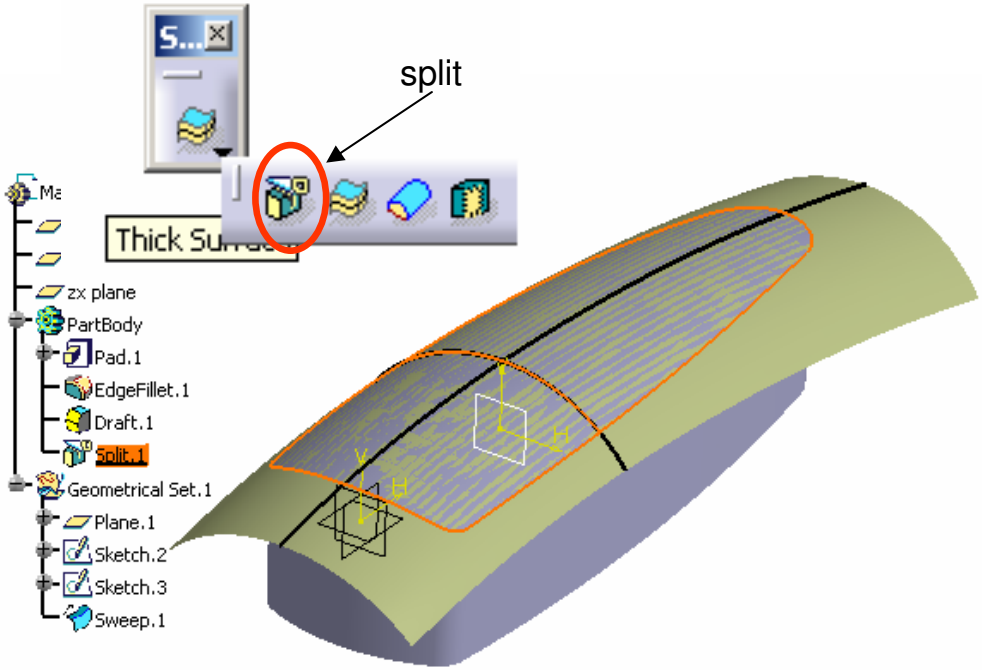
Tutorial 3A

To change the workbench:-

- Select '**Start/Mechanical Design/ Part Design**" on the menu bar to go back to solid-modeling environment

To cut the solid with this SURFACE:-

- Click "**Split**" icon
- Click OK on the warning window
- Select the Yellow Surface "Sweep.1"
- Click on the arrow so that it is pointing downwards
- Click ok to complete

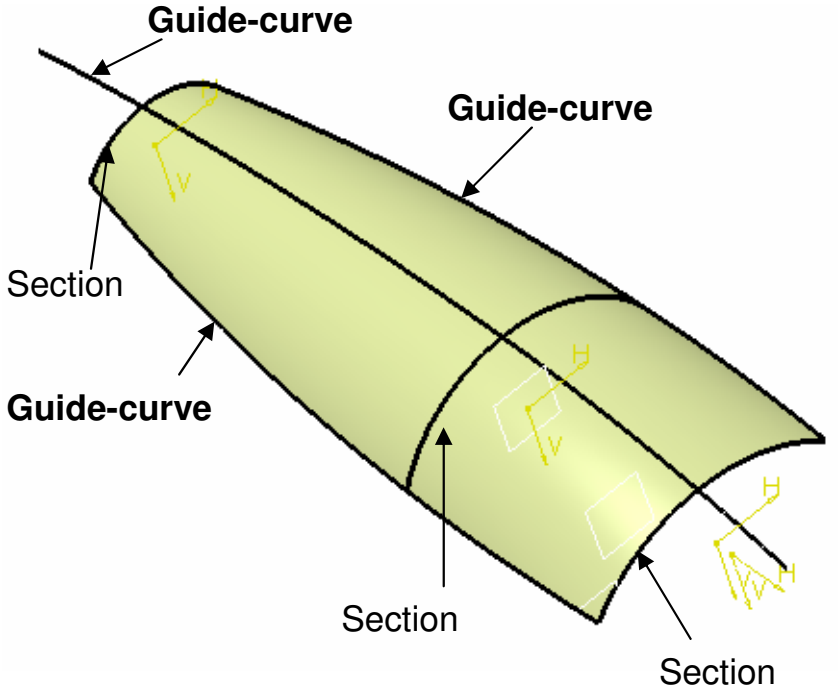


To hide the surface & its curves:-

- Select the surface "Sweep.1" and click "**hide/show**" icon
- **Hide** Sketch.2 & Sketch.3 too

Tutorial 3A

Now, we are going to create a Multi-section surface for the bottom face. Before that, we need to construct three guide curves and three different sections

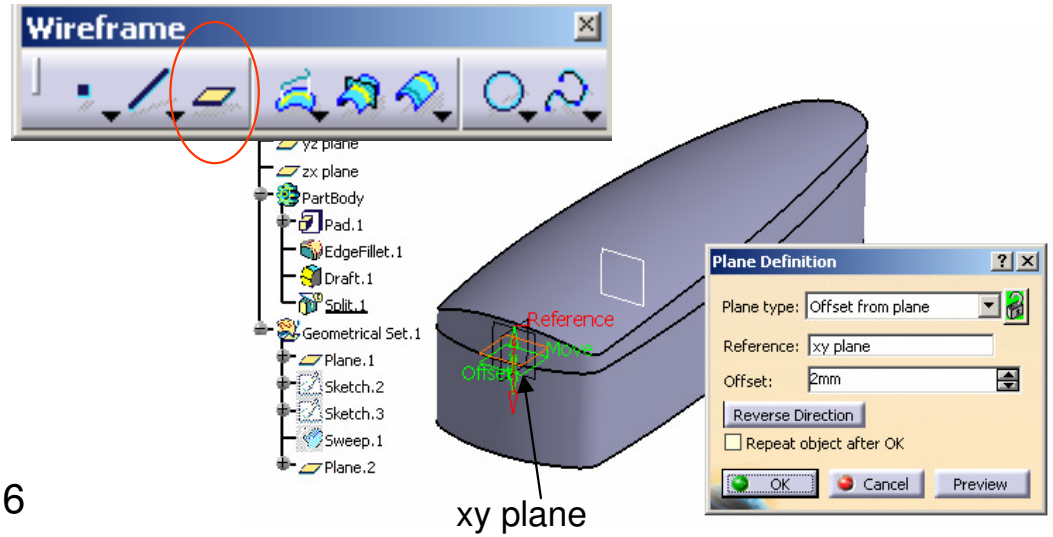


To change the workbench:-

- Select '**Start/Mechanical Design/ Generative Shape Design**" on the menu bar to go back to surface-modeling environment

To create a reference plane:-

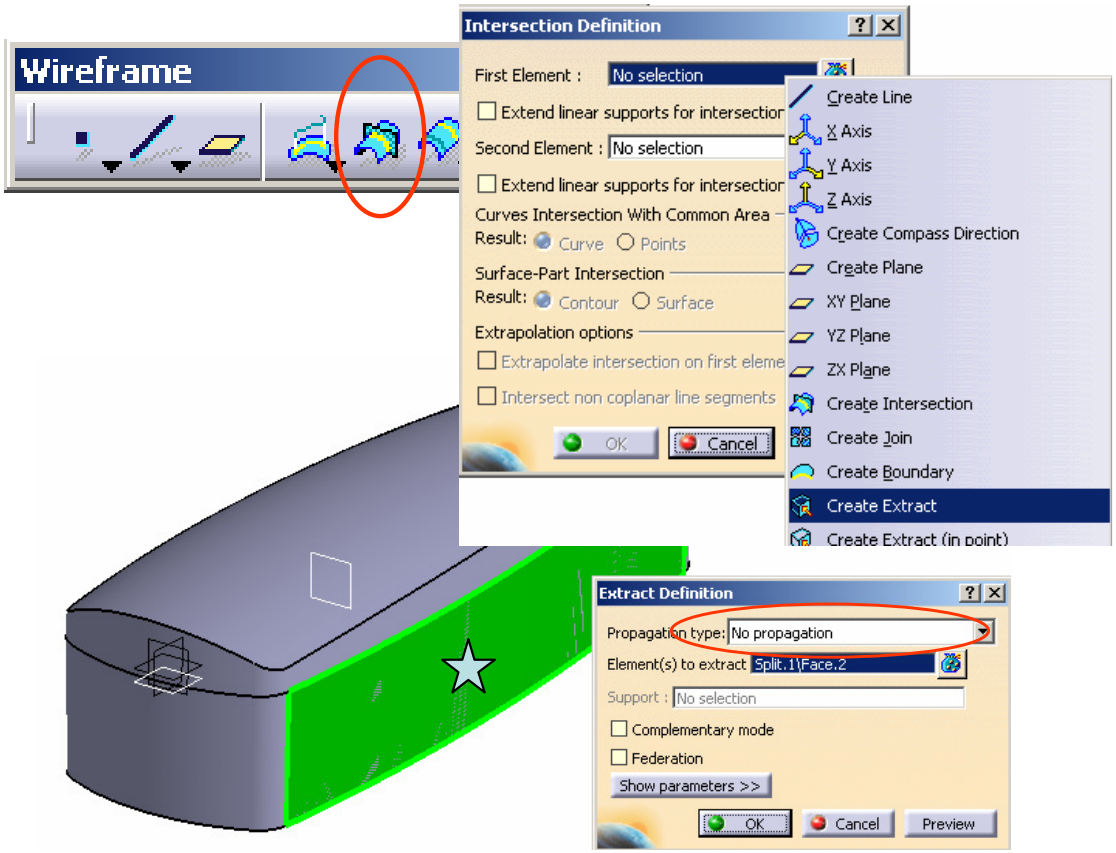
- Click "**plane**" icon
- Select "Offset from plane" as plane type
- Select "xy plane" as Reference
- Click "Reverse Direction" in the command window (The arrow points to negative Z)
- Enter 2mm as Offset value
- Click ok to complete



Tutorial 3A

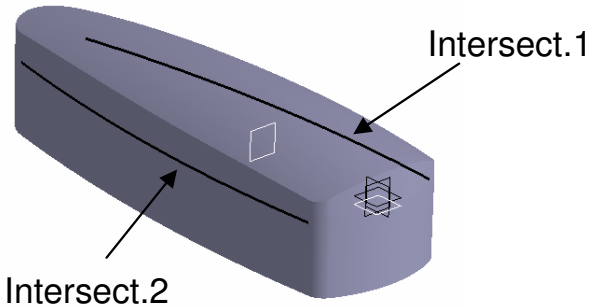
To create a intersection curve:-

- Click “**intersection**” icon
- Right-Click on the entry box of First Element
- Select “Create Extract”
- Select “No propagation “ for Extract Definition
- Select the Face ☆
- Select Plane.2 as Second Element
- Click ok to complete



To create another intersection curve on the opposite side:-

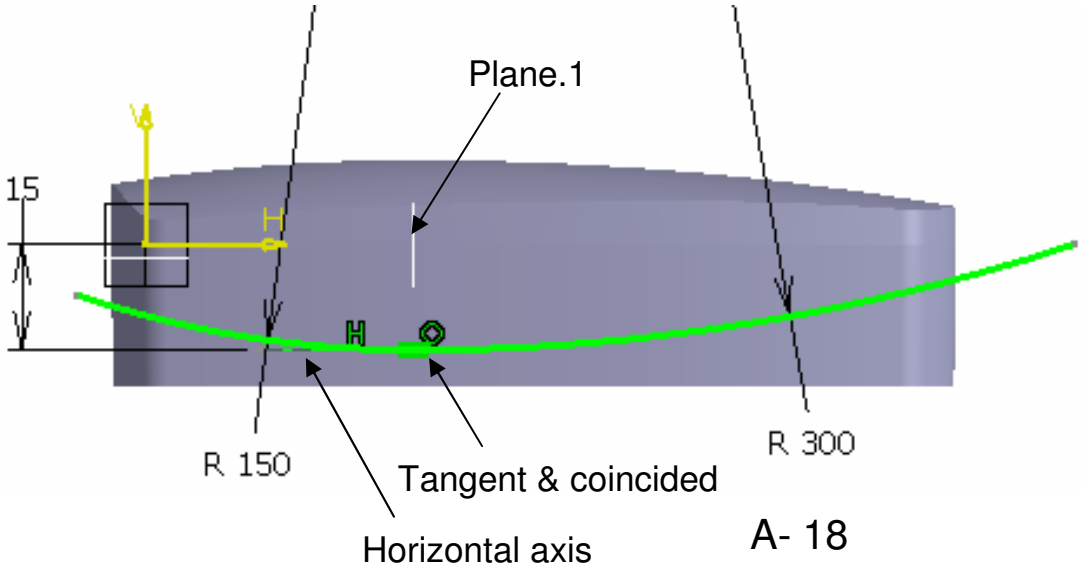
- Repeat the above steps but select the face opposite to Face ☆ as First Element

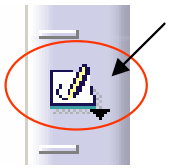


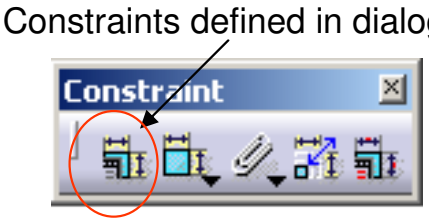
Tutorial 3A

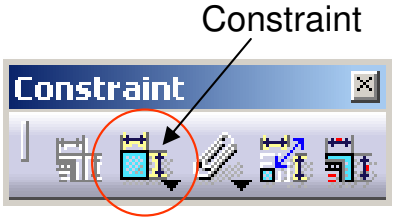
To build 4th sketch:-

- Click "**Sketch**" icon and select **zx plane**
- **Draw** a horizontal **axis** as shown
- **Multi-select** the axis and plane.1 by pressing and holding "**ctrl**" key on the keyboard.
- Then select "**Constraints defined in dialog box**" icon
- Select "**Coincidence**" and "ok"
- **Draw** another two **arcs** (R300 & R150) and add the corresponding constraints as shown
- Exit the workbench by clicking "**Exit**" icon
- *Click on an empty space to deselect the sketch*





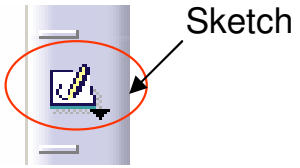




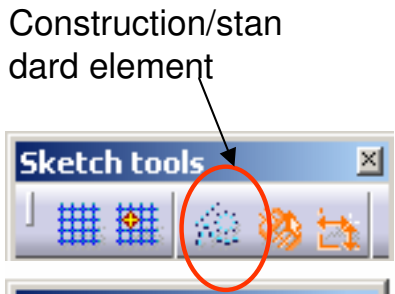
Tutorial 3A

To build 5th sketch:-

- click "**Sketch**" icon and select **plane.1**

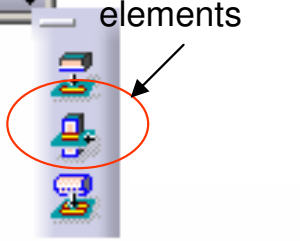


- Click "**Construction/Standard element**" icon so that the coming elements will be considered as construction (reference) elements

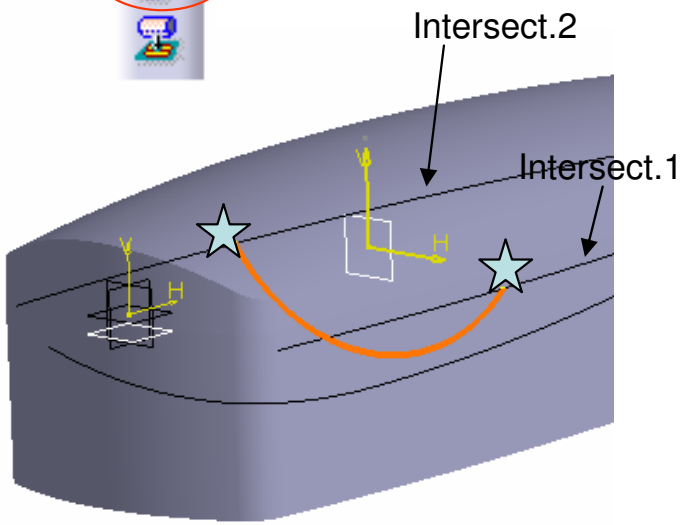


- Rotate the model by mouse to have an isometric view

- Click "**Intersect 3D elements**" icon
- Select the curve "Intersect.1" (A point is created)



- Similarly, click "**Intersect 3D elements**" icon
- Select the curve "Intersect.2" (A point is created)



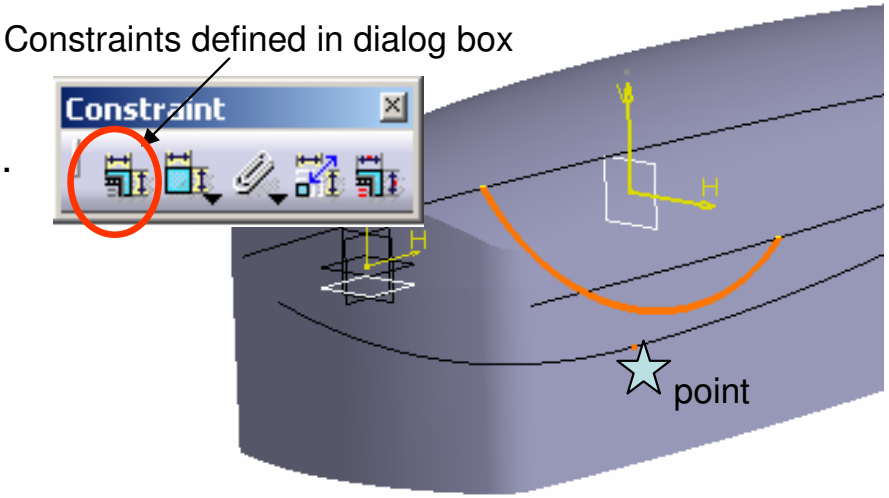
- Click "**Construction/Standard element**" icon again to deactivate this mode.

- **Draw** an **arc** by selecting the two intersection points as the endpoints ☆

Tutorial 3A

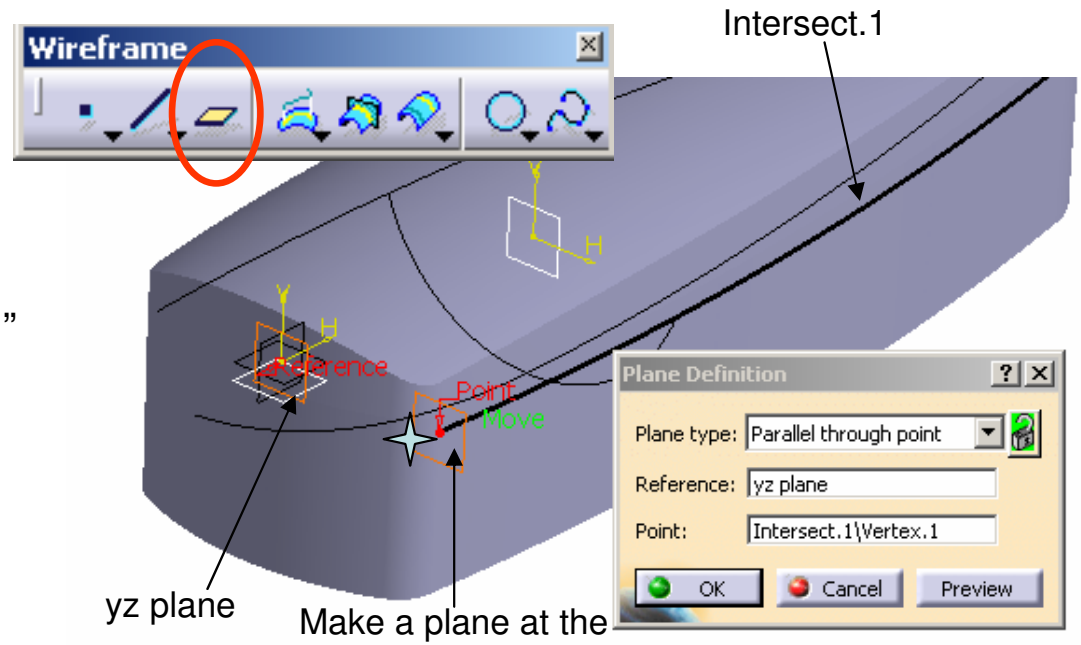
To build 5th sketch (Cont'):-

- **Multi-select** the arc and the point ☆ by pressing and holding "**ctrl**" key on the keyboard.
- Then select "**Constraints defined in dialog box**" icon.
- Select "**Coincidence**" and "ok".
- Exit the workbench by clicking "**Exit**" icon.

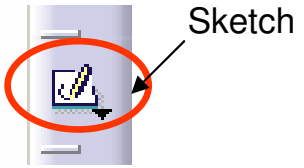


To create a reference plane:-

- Click "**plane**" icon
- Select "Parallel through point" as plane type
- Select "yz plane" as Reference
- Select a Endpoint ✧ of the curve "Intersect.1"
- Click ok to complete

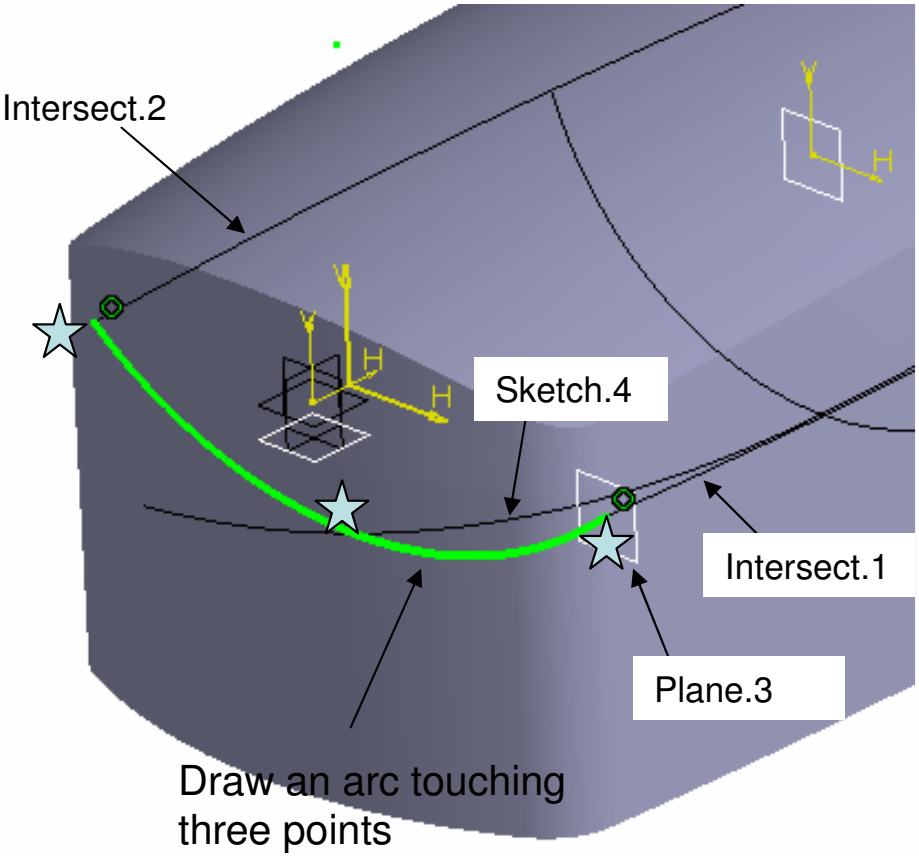


Tutorial 3A



To build 6th sketch:-

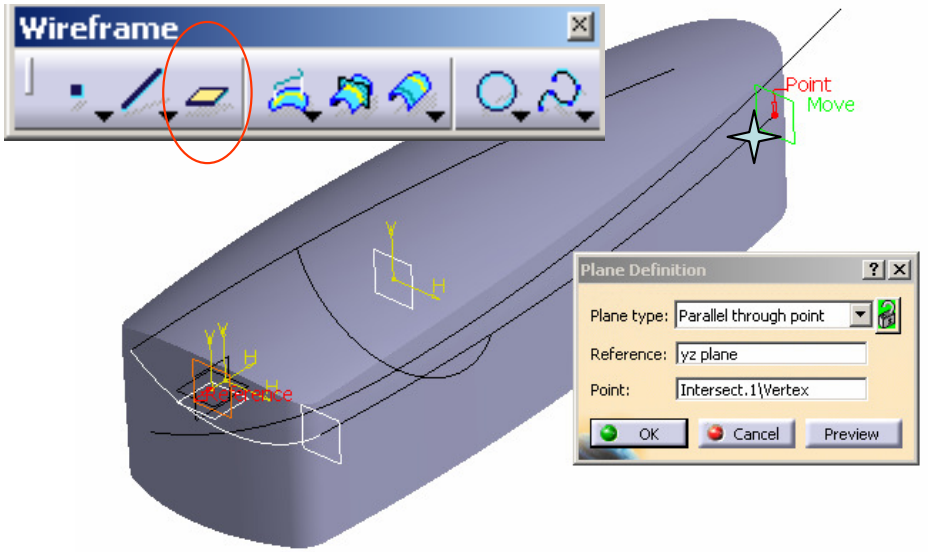
- click "**Sketch**" icon and select **plane.3**
- Click "**Construction/Standard element**" icon so that the coming elements will be considered as construction (reference) elements
- Rotate the model by mouse to have an isometric view
- click "**Intersect 3D elements**" icon
- Select the curve "Sketch.4" (A point is created)
- Click "**Construction/Standard element**" icon again to deactivate this mode.
- **Draw** an **arc** with the endpoints near the extreme points of Intersect.1 & Intersect.2
- Add three Coincidence constraints to align the arc onto the points ☆
- Exit the workbench by clicking "**Exit**" icon.



Tutorial 3A

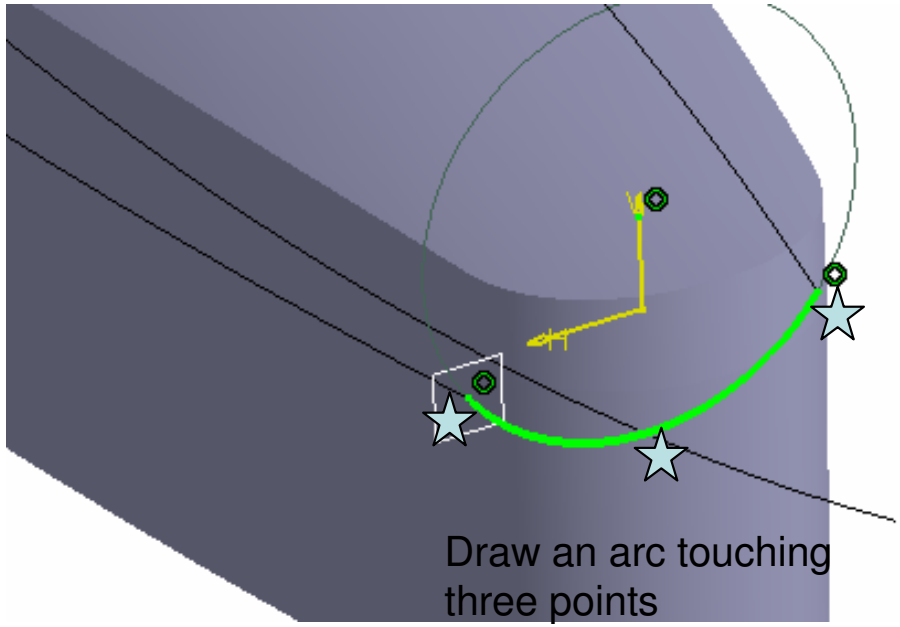
To create a reference plane:-

- Click “**plane**” icon
- Select “Parallel through point” as plane type
- Select “yz plane” as Reference
- Select another Endpoint ✦ of the curve “Intersect.1”
- Click ok to complete



To build 7th sketch:-

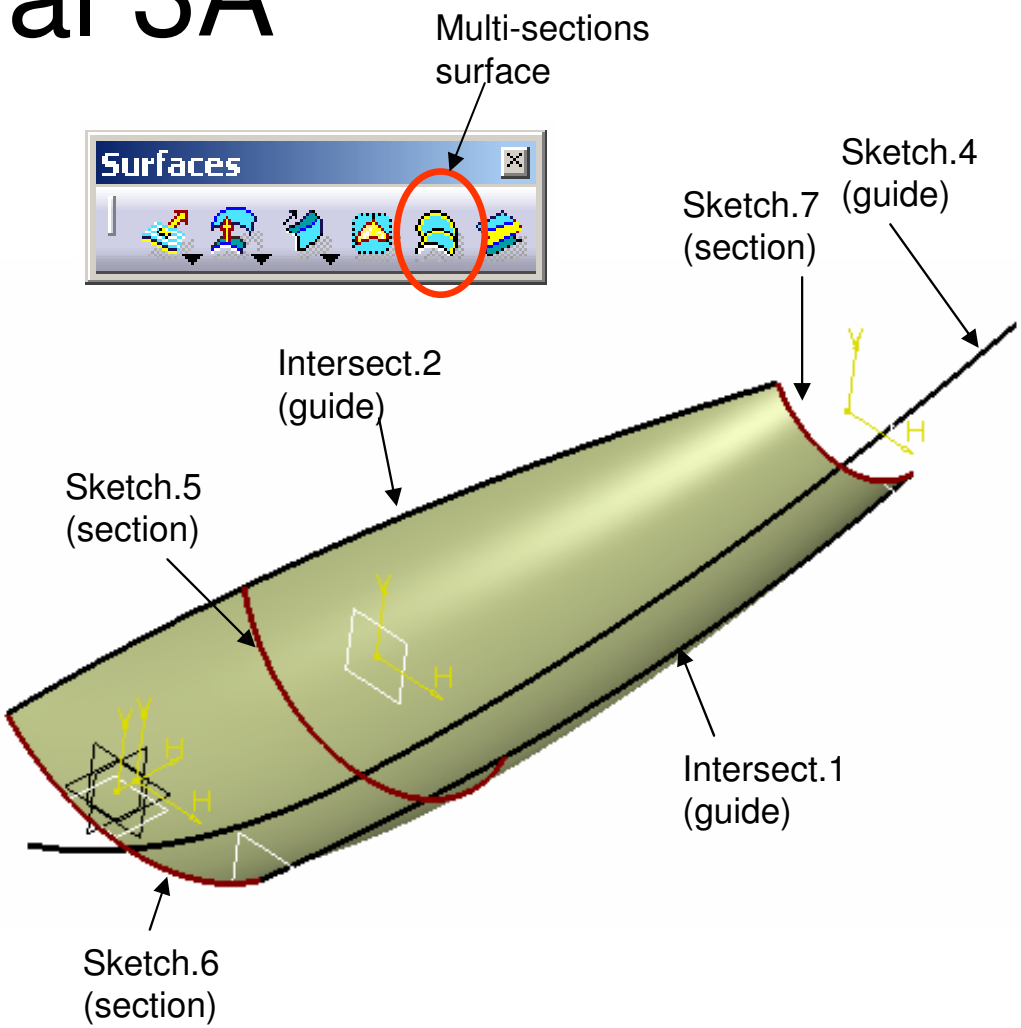
- **Draw** an **arc** with the endpoints near the endpoints of Intersect.1 & Intersect.2
- Add three Coincidence Constraints to align the arc onto the points ★
- (Refer to the steps of building 6th sketch)

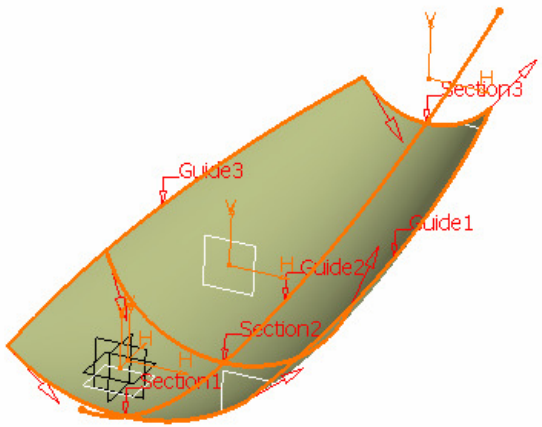


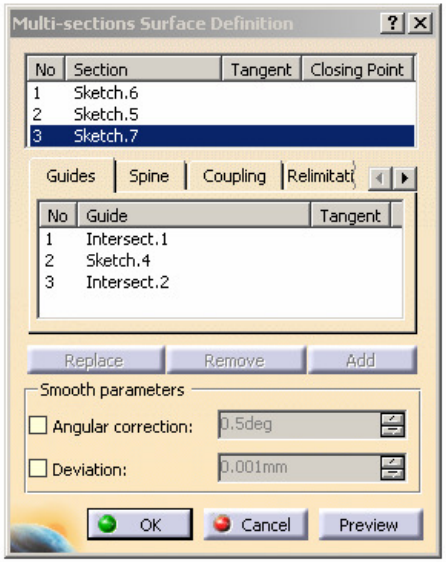
Tutorial 3A

To create a Multi-sections surface:-

- Click “**Multi-sections surface**” icon
- Select “Sketch.6”, “Sketch.5”, & “Sketch.7” in order (They will then be inserted into the entry box of **Section**)
- If any red arrows are not pointing to the same direction, click it once to reverse.
- Then click the entry box of **Guides** once
- Select “Intersect.1”, “Sketch.4” & “Intersect.2”
- Click ok to complete





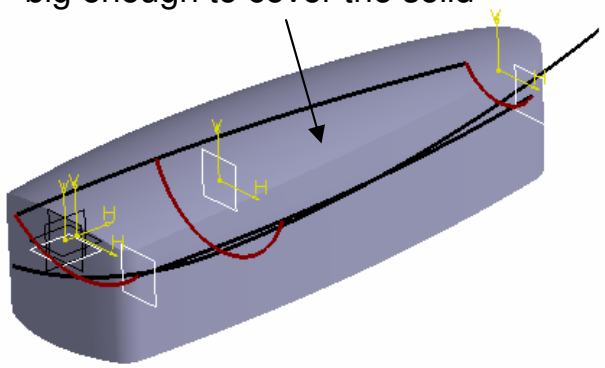


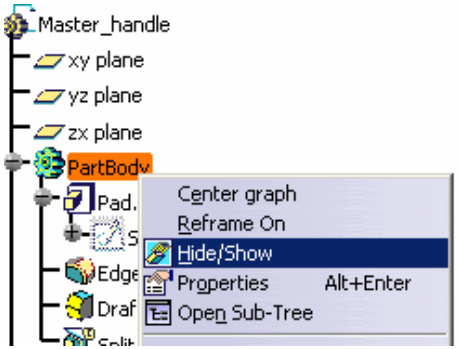
All red arrows should point to the same direction, otherwise the created surface will be twisted

Tutorial 3A

As seen, the multi-sections surface is not big enough to cover the whole solid…

The multi-sections face is not big enough to cover the solid



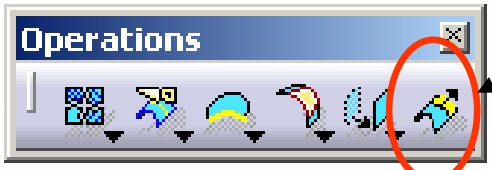


To hide the solid:-

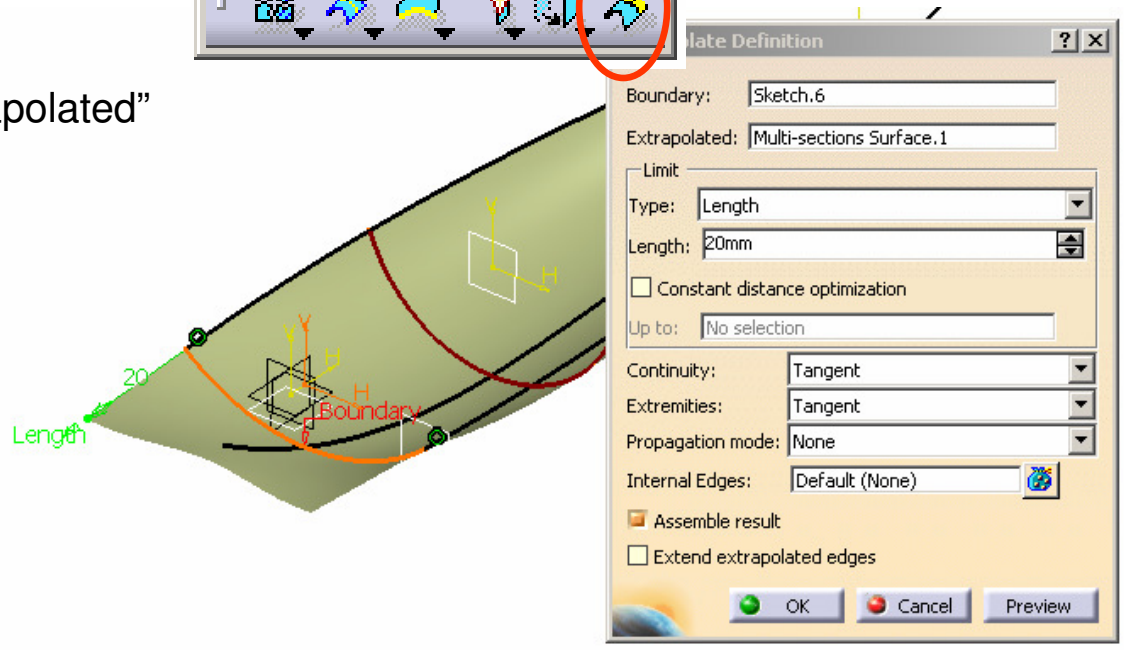
- Right-click on “PartBody” on the tree
- Select **Hide/Show**

To extend the surface:-

- Click “**Extrapolate**” icon
- Select “Sketch.6” as Boundary
- Select “Multi-sections surface.1” as “Extrapolated”
- Enter 20mm as Length
- Enter Tangency as Continuity
- Select “Assembly Result”
- Click Ok to complete



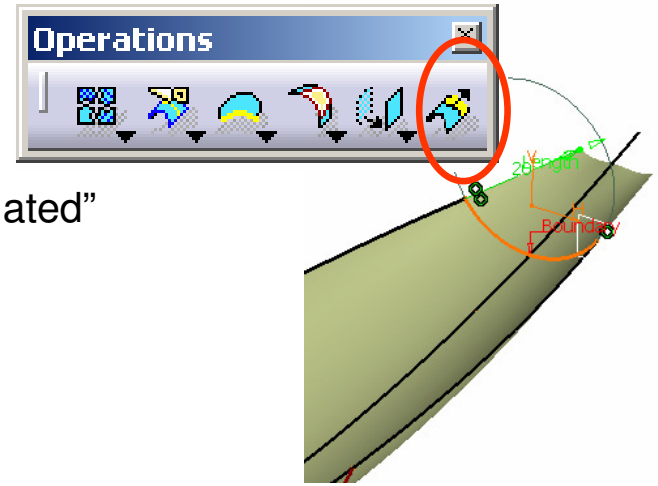
extrapolate

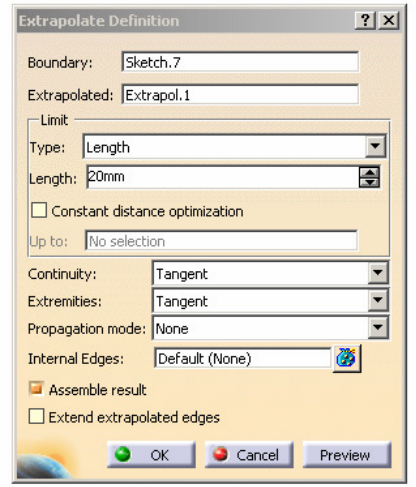


Tutorial 3A

To extend the surface on the other end:-

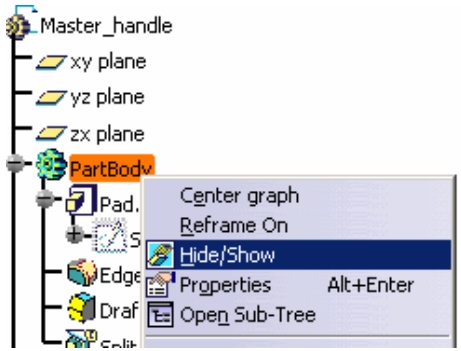
- Click “**Extrapolate**” icon
- Select “Sketch.7” as Boundary
- Select “Multi-sections surface.1” as “Extrapolated”
- Enter 20mm as Length
- Enter Tangency as Continuity
- Select “Assembly Result”
- Click ok to complete





To change the workbench:-

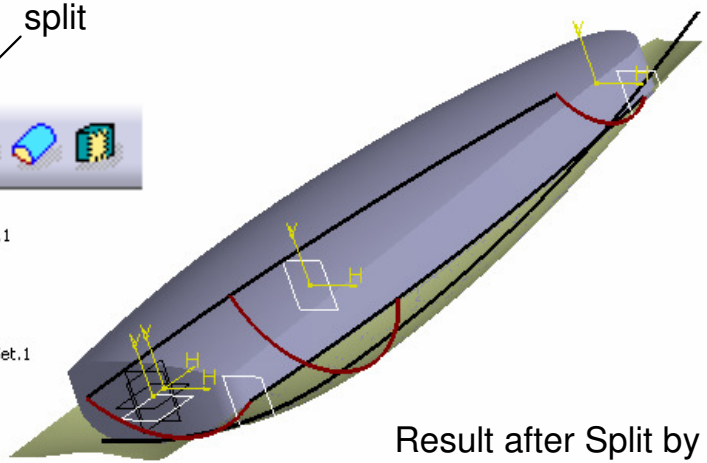
- Select ‘**Start/Mechanical Design/ Part Design**” on the menu bar to go back to solid-modeling environment
- **Unhide** “PartBody”



Tutorial 3A

To cut the solid with this extended surface:-

- Click "**Split**" icon.
- Click OK on the warning window.
- Select the Yellow Surface "Extrapol.2"
- Click on the arrow so that it is pointing upwards.
- Click ok to complete

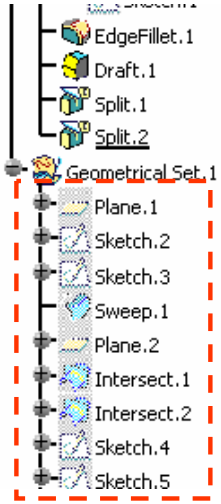


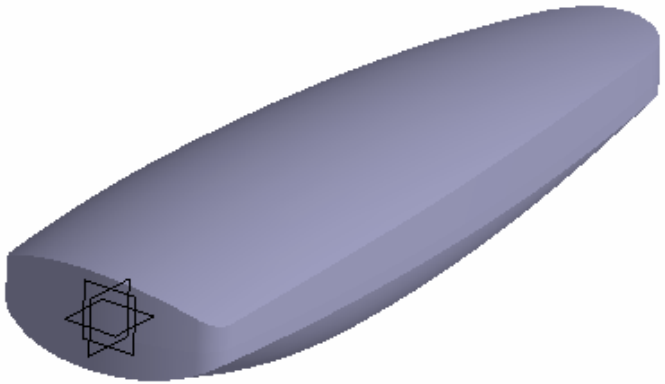
Result after Split by the extended surface

To hide the surface & its curves:-

- **Hide** Everything except "Partbody", xy plane, yx plane and zx plane.

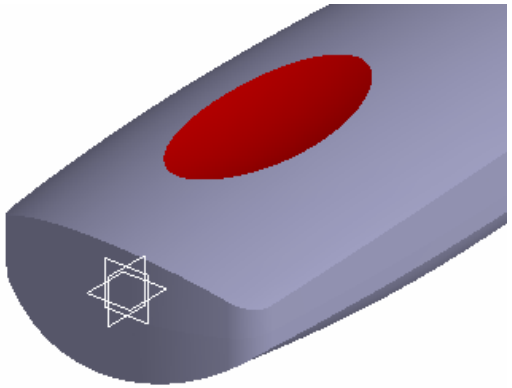
All reference curves & surfaces are now hidden →





Tutorial 3A

Next, we are going to create a curve-based pocket on the top face:-

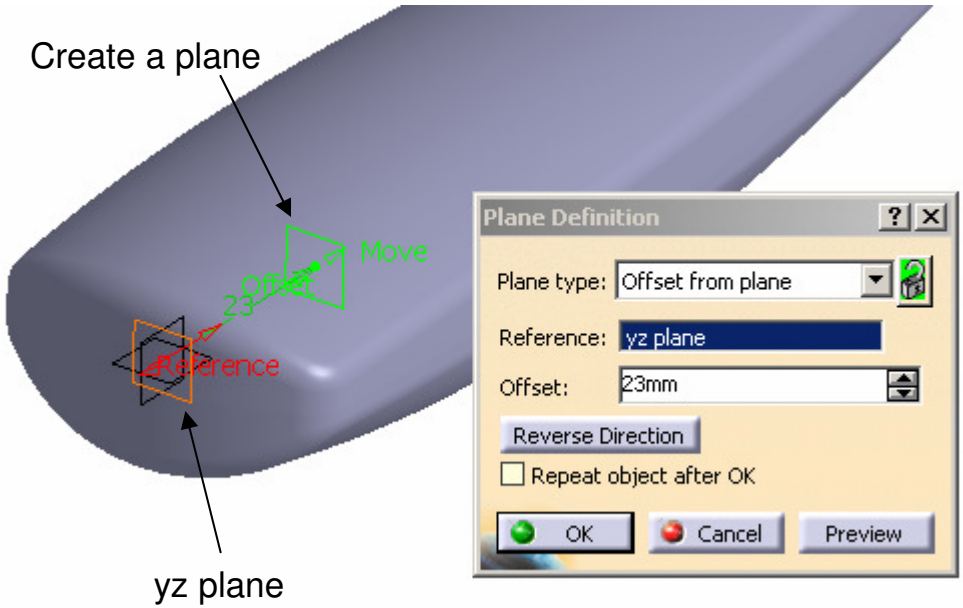


To change the workbench:-

- Select '**Start/Mechanical Design/ Generative Shape Design**" on the menu bar to go back to the surface-modeling environment

To create a reference plane (plane.5):-

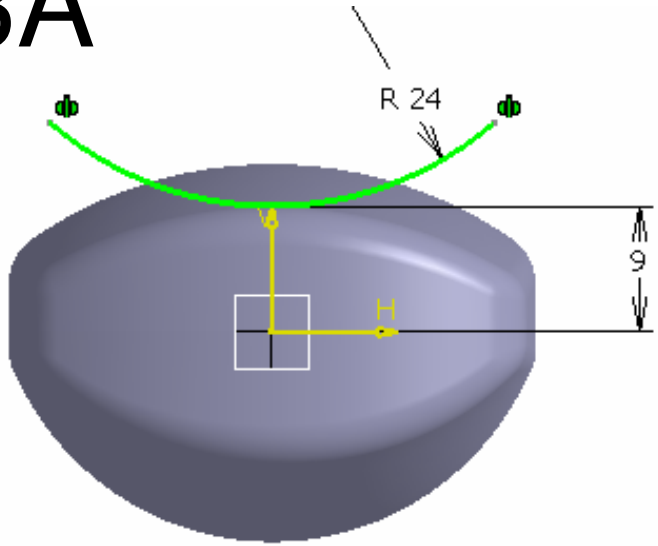
- Click "**plane**" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window (The arrow should point to negative X)
- Enter 23mm as Offset value
- Click ok to complete



Tutorial 3A

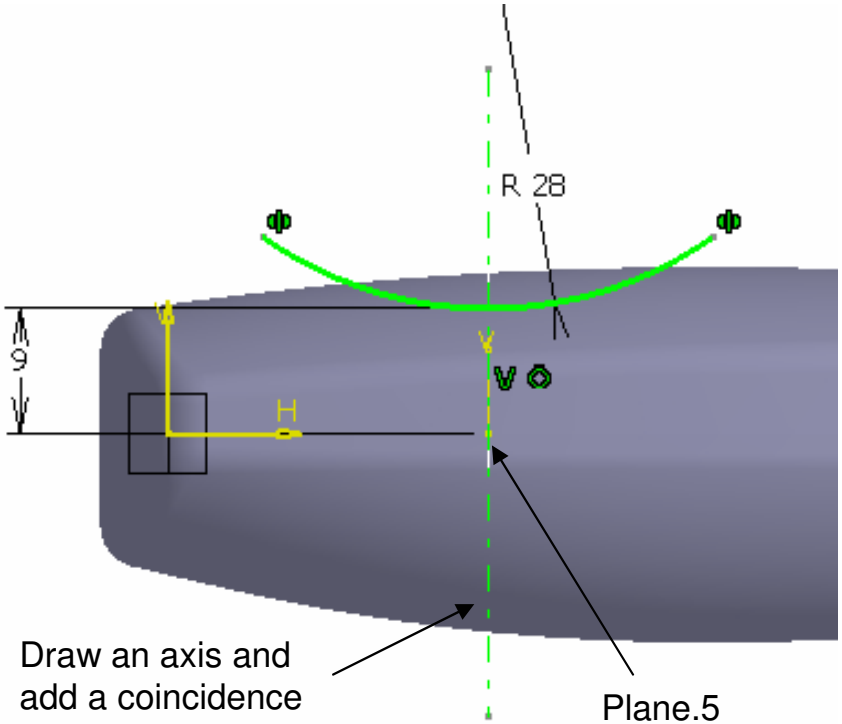
To build 8th sketch:-

- Click "**Sketch**" icon and select **plane.5**
- **Draw** an **arc** (R24, endpoints symmetric about y-axis)
- Add a Constraint (9mm) to define the distance between the arc and x-axis
- Exit the workbench by clicking "**Exit**" icon
- *Click on an empty space to deselect the sketch*



To build 9th sketch:-

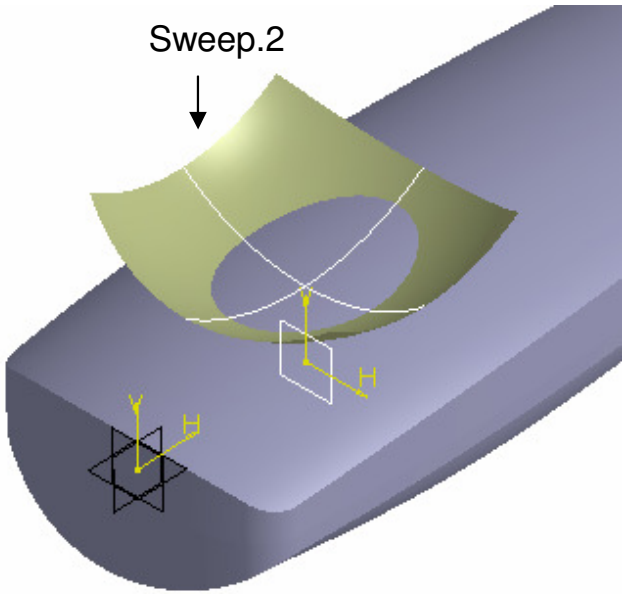
- Click "**Sketch**" icon and select **zx plane**
- **Draw** an **axis**
- Add a **coincidence** constraint between the axis and plane.5
- **Draw** an **arc** (R28, endpoints symmetric about the axis)
- Add a Constraint (*9mm*) to define the distance between the arc and x-axis
- Exit the workbench by clicking "**Exit**" icon
- *Click on an empty space to deselect the sketch*



Tutorial 3A

To build a Surface:-

- Click "**Sweep**" icon
- Select "Explicit" as Profile Type
- Select "Sketch.8" as Profile
- Select "Sketch.9" as Guided Curve
- Click ok to complete

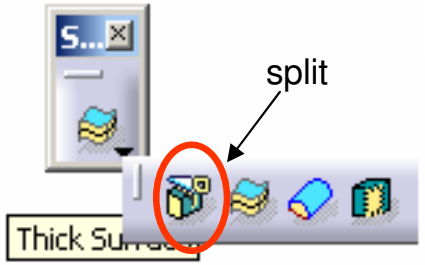


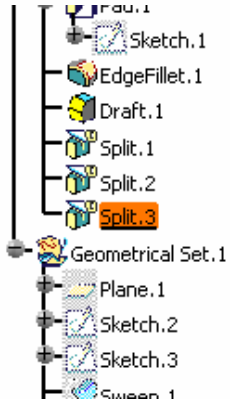
To change the workbench:-

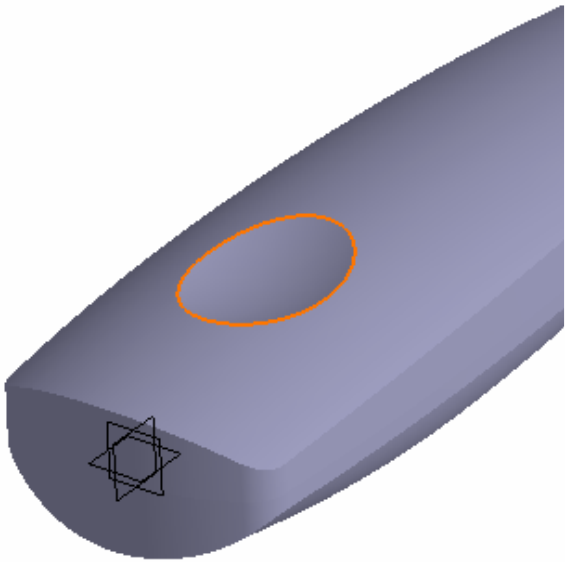
- Select '**Start/Mechanical Design/ Part Design**" on the menu bar to go back to solid-modeling environment

To cut the solid with this SURFACE:-

- Click "**Split**" icon.
- Click OK on the warning window.
- Select the Yellow Surface "Sweep.2"
- Click on the arrow so that it is pointing downwards.
- Click ok to complete







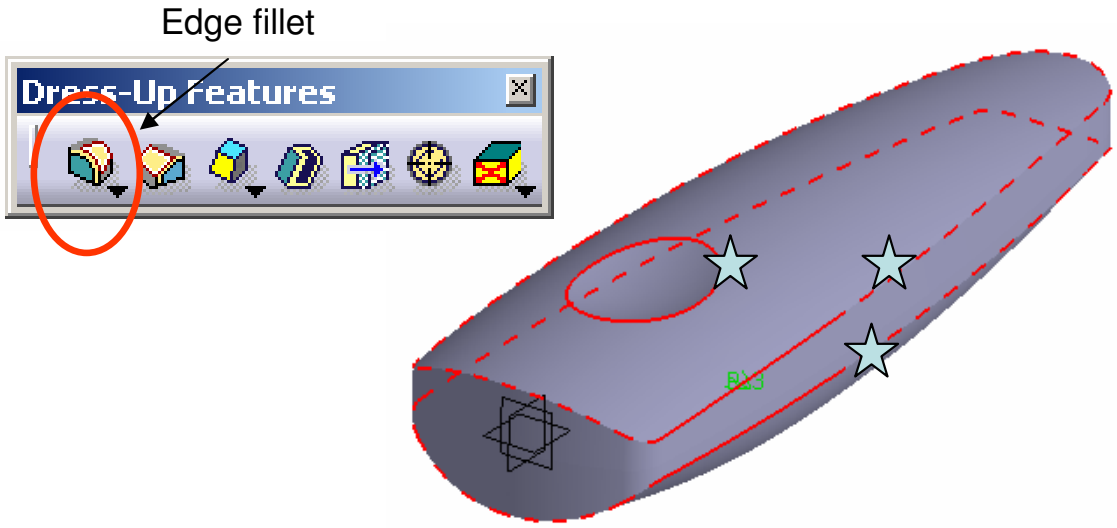
To hide the surface & its curves:-

- Select "Sweep.2", "Sketch.8", "Sketch.9" & "Plane.5" and click "hide/show" icon.

Tutorial 3A

To add Edge Fillets:-

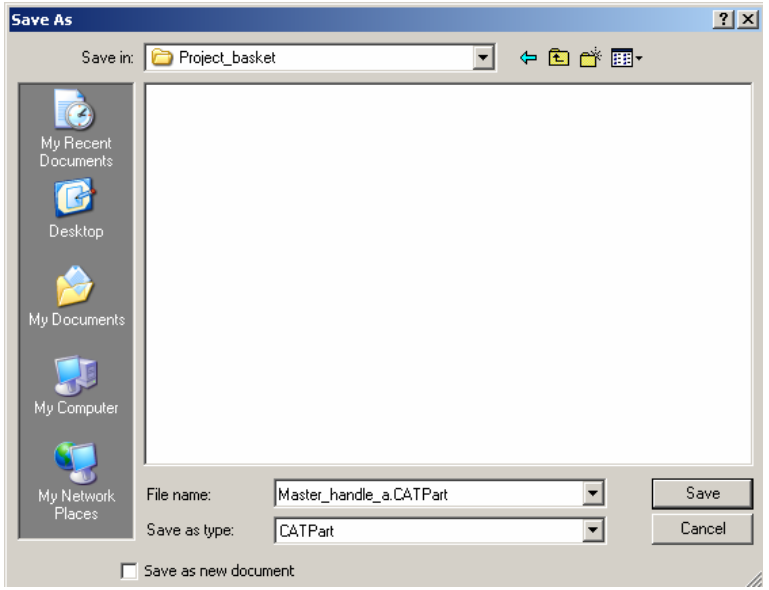
- Click "**Edge Fillet**" icon
- Enter 3mm as Radius
- Select Tangency as Propagation
- Select the three sharp edges ☆
- Click ok to complete



To save the new part in a Project Folder:-

It is a good practice to store all part files of a product in one specific folder.

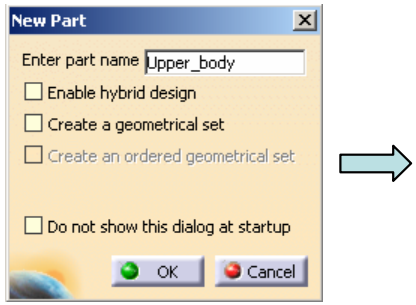
- Create a folder wherever you can save (by MS window technique).
- Save your current part as "**master_handle_a.CATPART**" into the folder.
- Add "a" after its name to remind us its version.



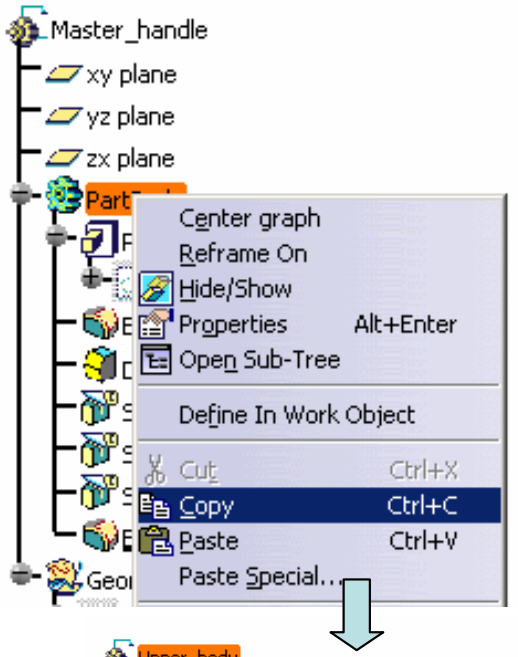
Tutorial 3A

To create the upper body:-

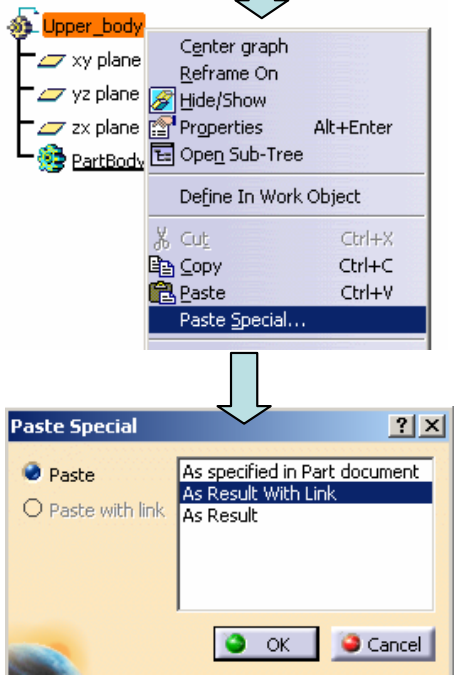
- Select File/New on the menu bar
- Select Part as type
- Enter Upper_body as part name
- Click ok to complete

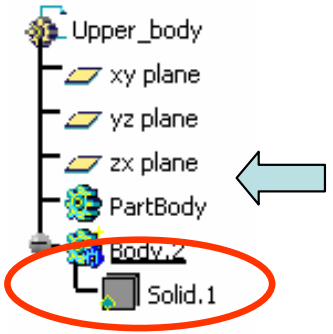


- Select Window/Tile Vertically (we can see Master_handle & Upper Body at the same time)
- Right-click "PartBody" of master_handle_a.CatPart;
- and then select **"Copy";**



- Right-click "Upper_body" of the tree of Upper_body and then select **"Paste Special…"**
- Select "**As Result with link**";
- Click ok to complete.

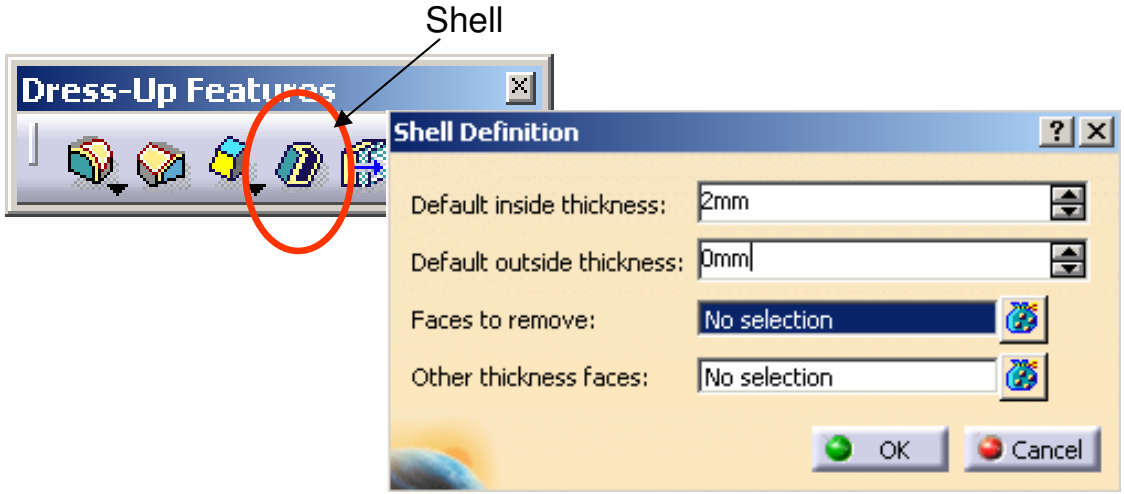




Tutorial 3A

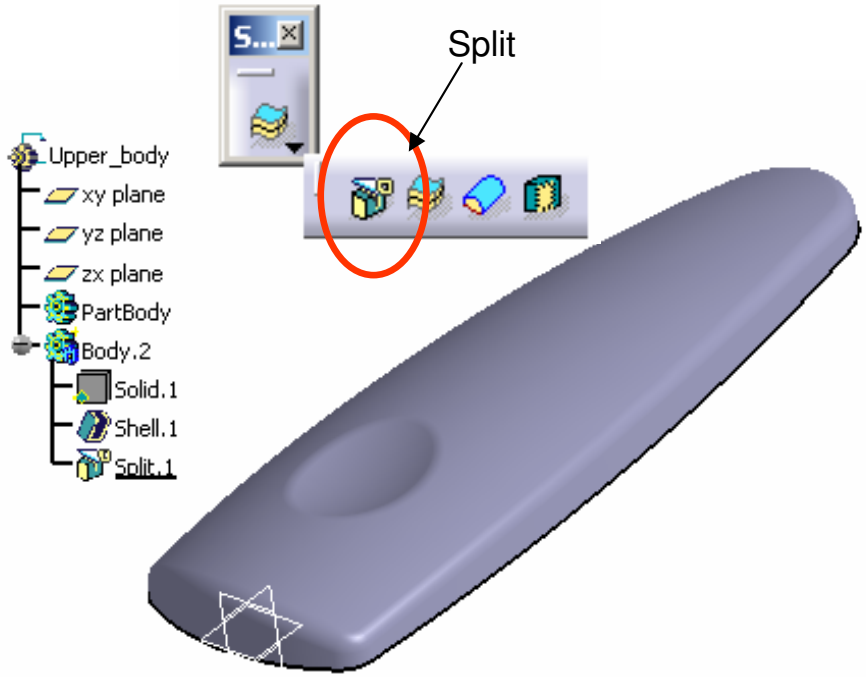
To make the solid hollow:-

- Click "**Shell**" icon
- Enter 2mm as Default inside thickness
- Click ok to complete.
- (the material inside the solid is removed)



To cut the solid with a plane:-

- Click "**Split**" icon.
- Select xy plane
- Click on the arrow so that it is pointing upwards.
- Click ok to complete



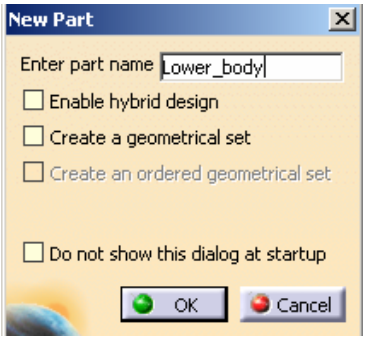
To Save the new part in a Project Folder:-

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

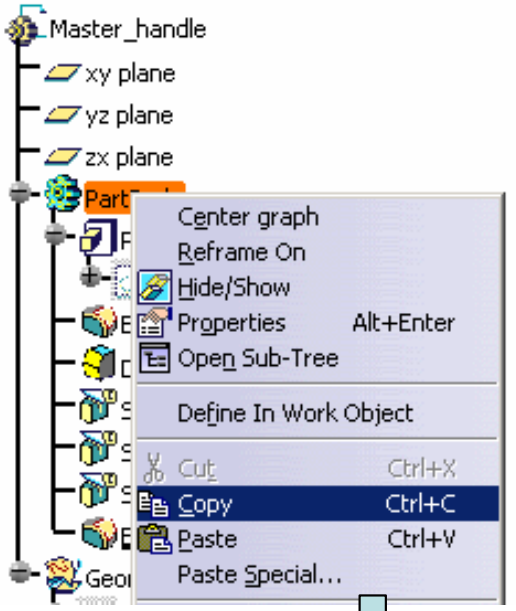
Tutorial 3A

To create the lower body:-

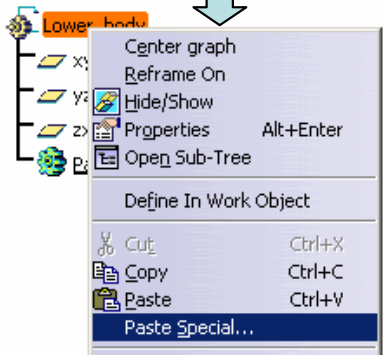
- Select File/New
- Select Part as type
- Enter Lower_body as part name
- Click ok to complete

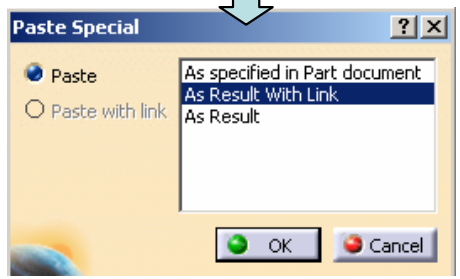


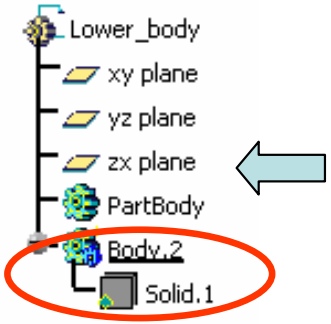
- Select Window/Tile Vertically (we can see Master_handle & Lower Body at the same time)
- Right-click "PartBody" of master_handle_a.CatPart
- and then select **"Copy"**



- Right-click "Lower_body" of the tree of Lower_body and then select "**Paste Special…**"
- Select **"As Result with link"**
- Click ok to complete



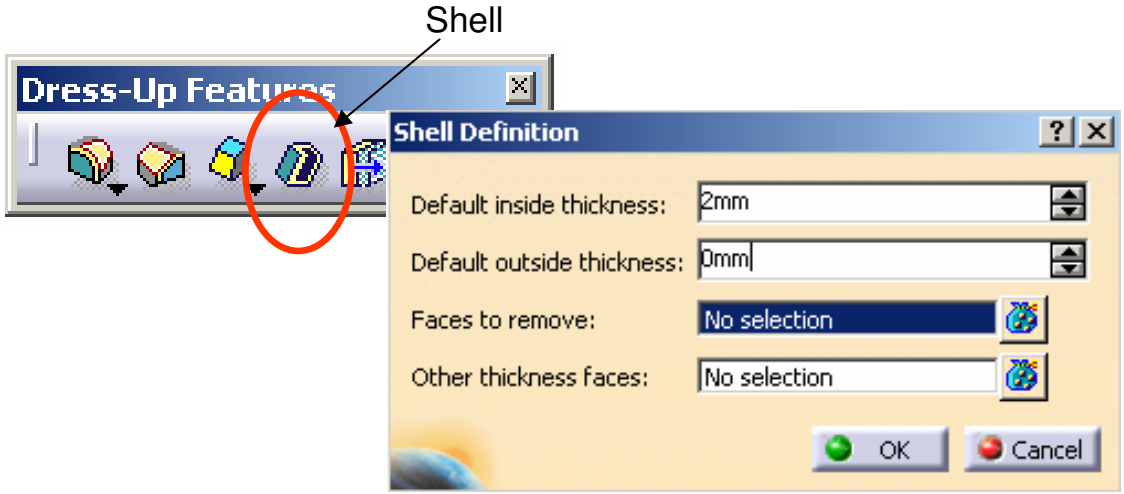




Tutorial 3A

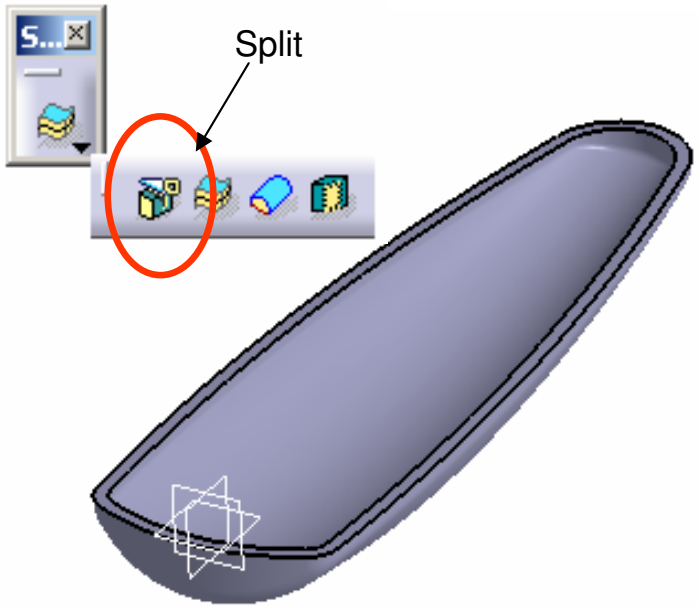
To make the solid hollow:-

- Click "**Shell**" icon
- Enter 2mm as Default inside thickness
- Click ok to complete.
- (the material inside the solid is removed)



To cut the solid with a plane:-

- Click "**Split**" icon.
- Select xy plane
- Click on the arrow so that it is pointing downwards.
- Click ok to complete

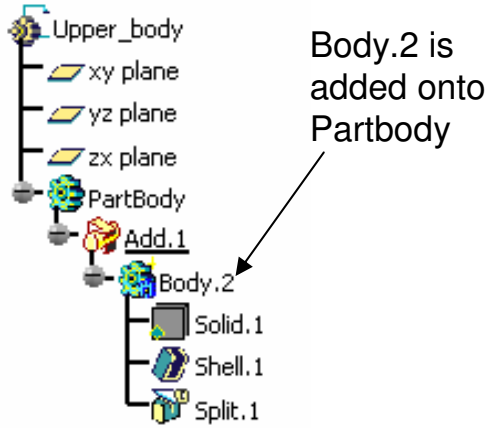


To save the new part in Project Folder:-

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

Tutorial 3A

Building mechanical features on Upper Body:-

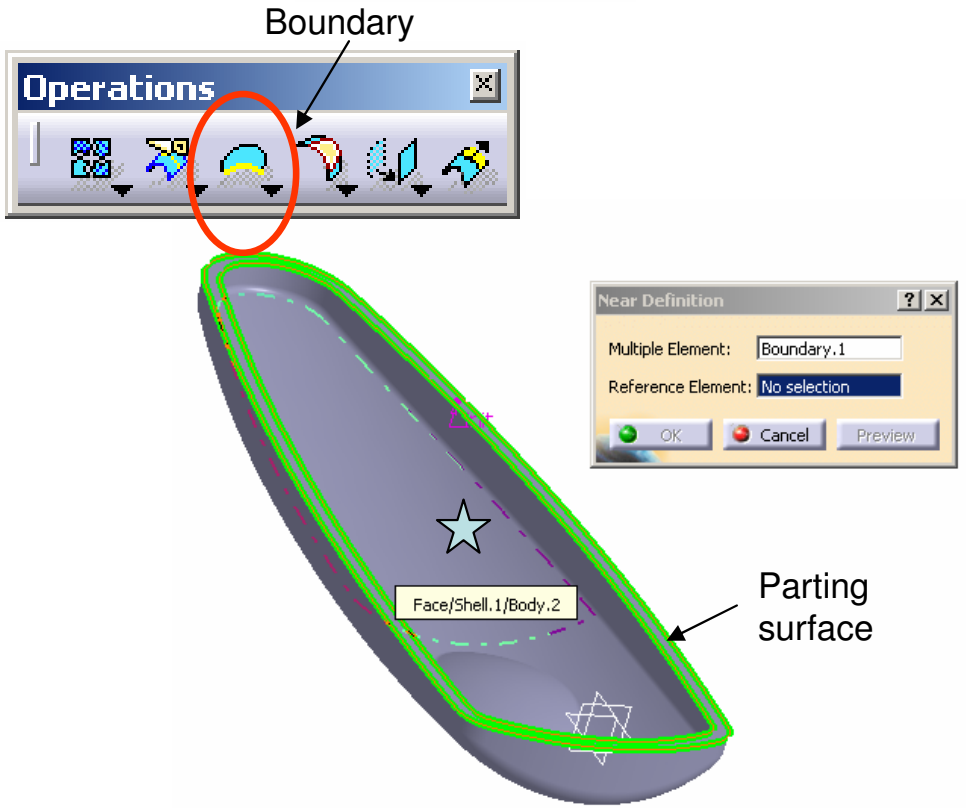


To add two Bodies together:-

- Right-click “Body.2” on the tree
- Select Body.2 **object/add…** (Body.2 will become a branch of PartBody)

To get a boundary curve from the solid:-

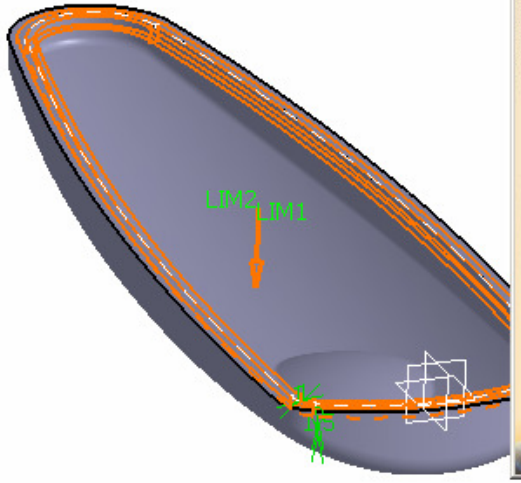
- Select ‘**Start/Mechanical Design/ Generative Shape Design**” on the menu bar to go back to surface-modeling environment
- Click “**Boundary**” icon
- Select “Point continuity” as propagation type
- Select the parting surface (both inner & outer edges will be highlighted)
- Click ok to complete
- Select “Keep only one sub-element by a Near” in the message window “Multi-result management”
- Click ok
- Select an inner face ☆ as Reference Element
- Click ok to complete

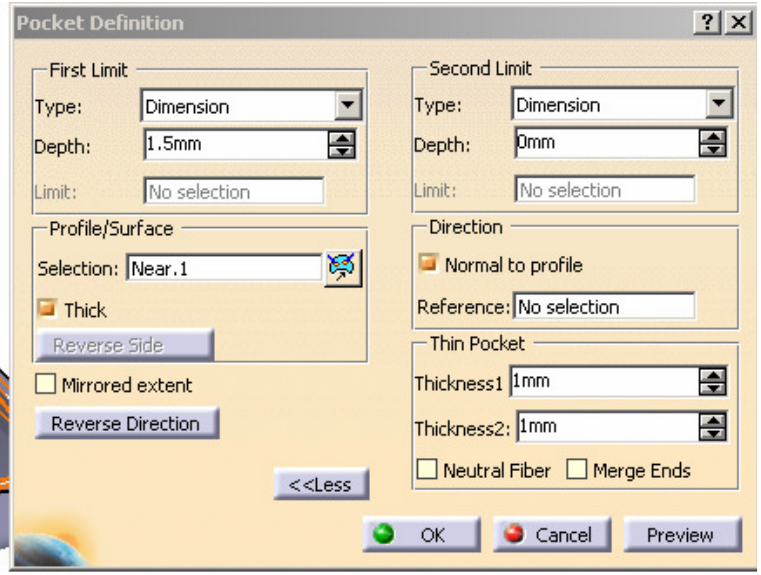


Tutorial 3A

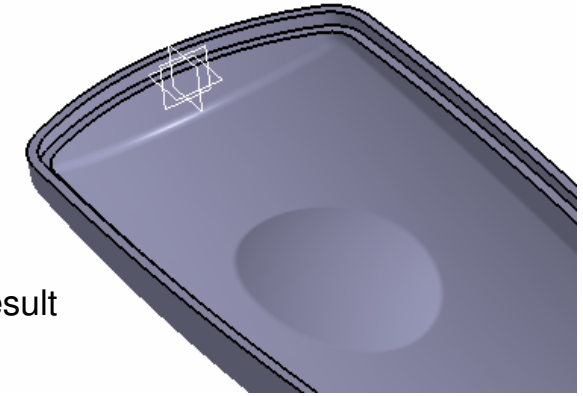
To remove material from the boundary:-

- Select '**Start/Mechanical Design/ Part Design**" on the menu bar to go back to solid-modeling environment
- Click "**Pocket**" icon
- Click OK on the warning window.
- Select the curve "Near.1"
- Select "Reverse Direction"
- Enter 1.5mm as First Limit
- Select "Thick" option
- Enter 1mm as thickness.1
- Enter 1mm as thickness.2
- Click ok to complete





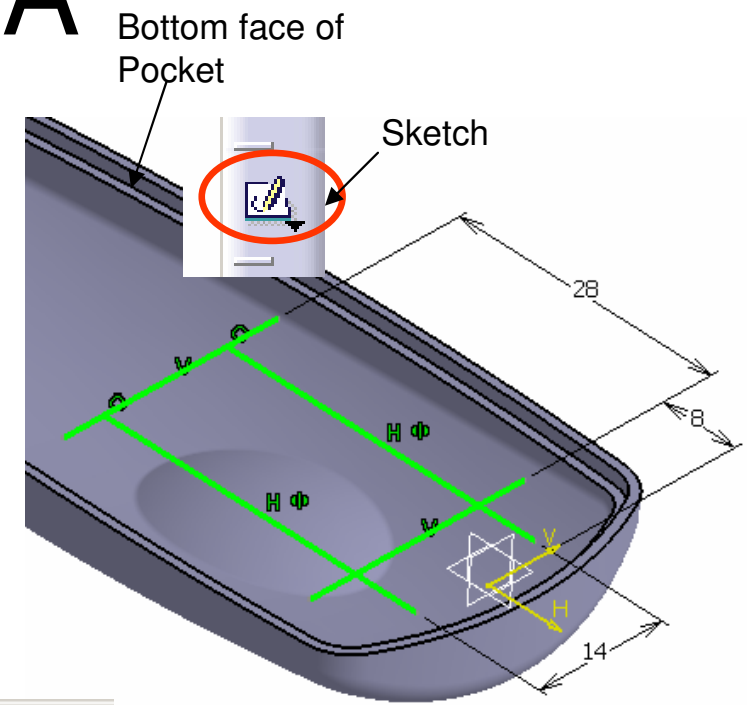
Result



Tutorial 3A

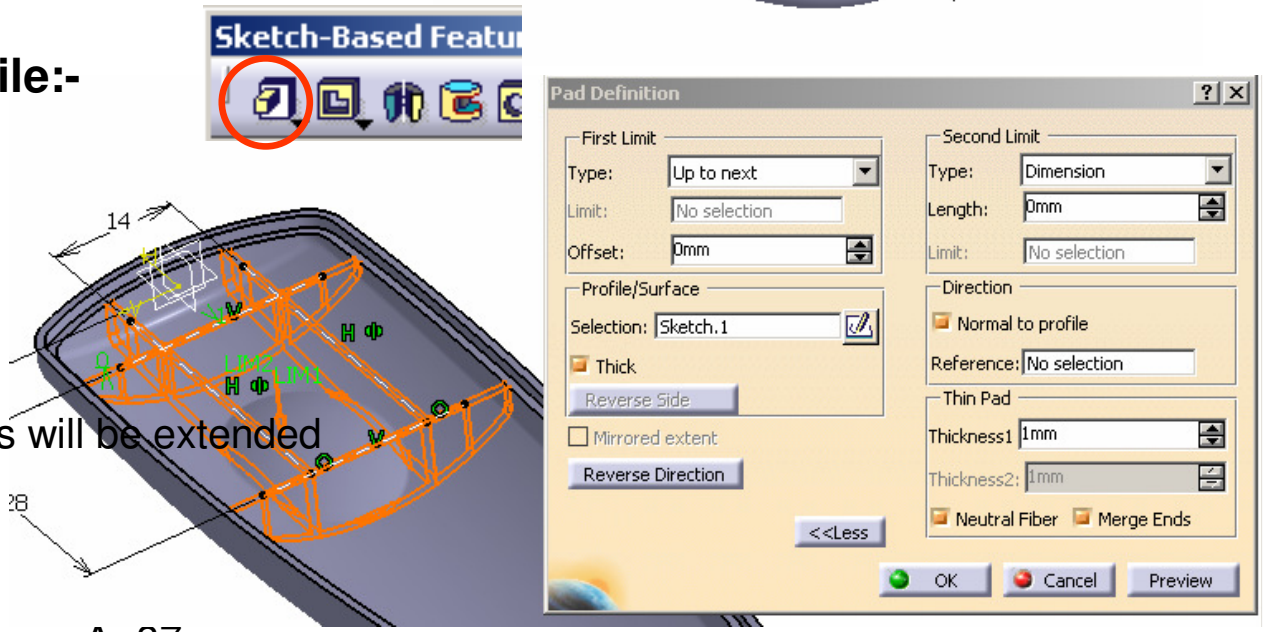
To build a sketch:-

- Click "**Sketch**" icon and select **the bottom face of Pocket.1**
- **Draw** 4 **lines** (two horizontal & two vertical)
- Add a Symmetric Constraint between two horizontal lines
- Add 3 more dimensional constraints (28mm, 8mm & 14mm)
- Exit the workbench by clicking "**Exit**" icon



To build a solid from the open profile:-

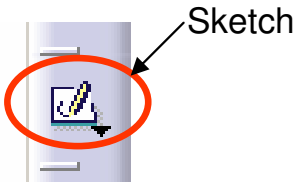
- click "**Pad**" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter 1mm as thickness.1
- Select "Merge Ends" so that the lines will be extended until they touch the solid face
- Select "Up to Next" as First Limit
- Click ok to complete

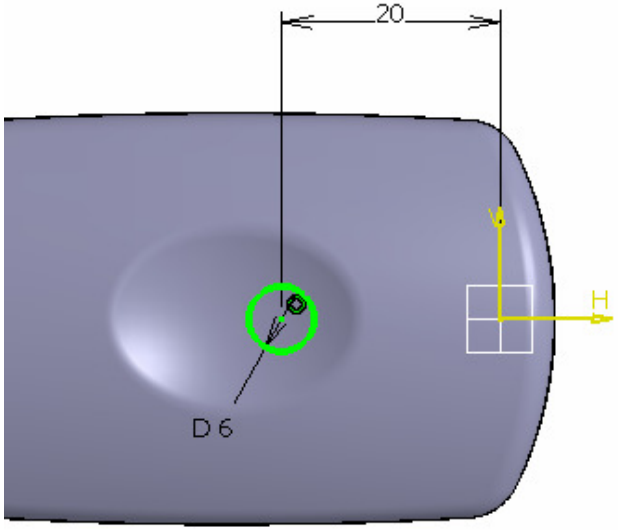


Tutorial 3A

To build another sketch:-

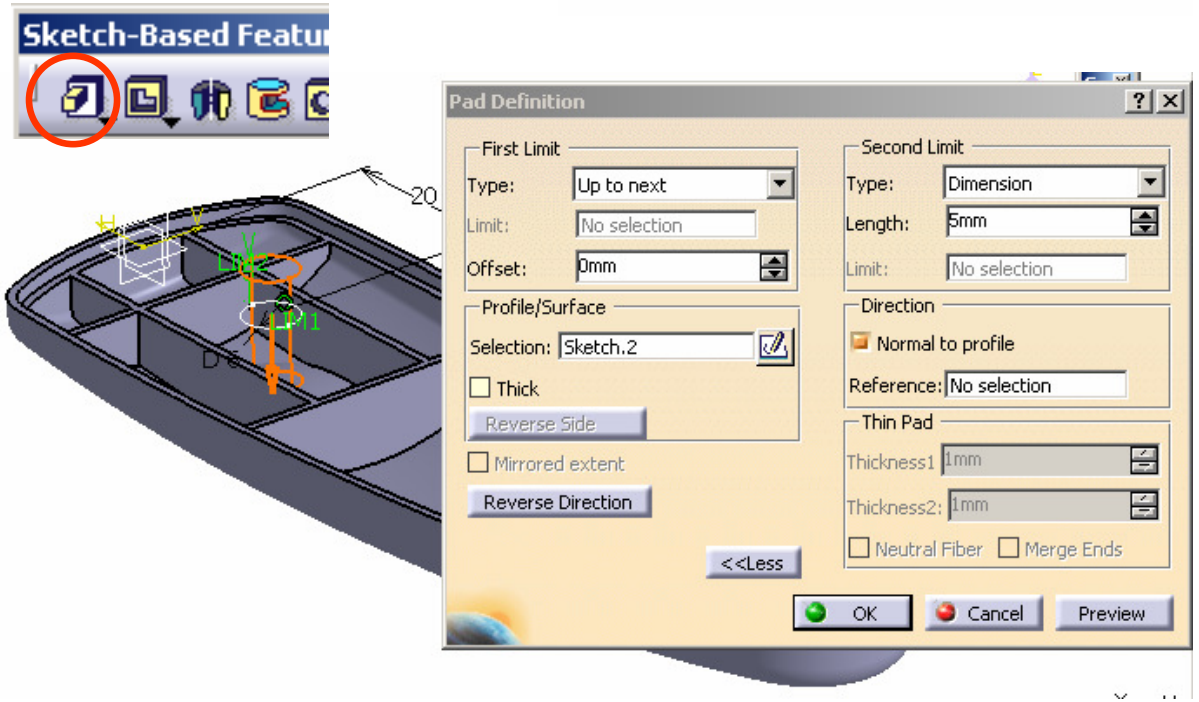
- Click "**Sketch**" icon and select **xy plane**
- **Draw** a **circle** (Dia6.0) on x-axis
- Add a dimensional Constraint (20mm)
- Exit the workbench by clicking "**Exit**" icon





To build a solid:-

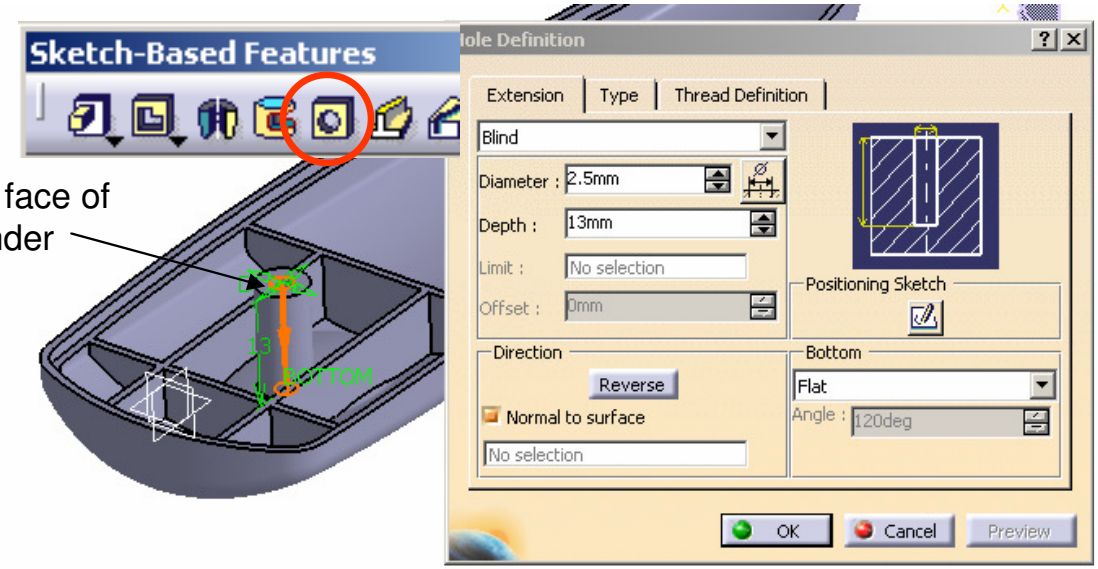
- click "**Pad**" icon
- Select "Up to Next" as First Limit
- Select "More"
- Enter 5mm as Second Limit
- Click ok to complete



Tutorial 3A

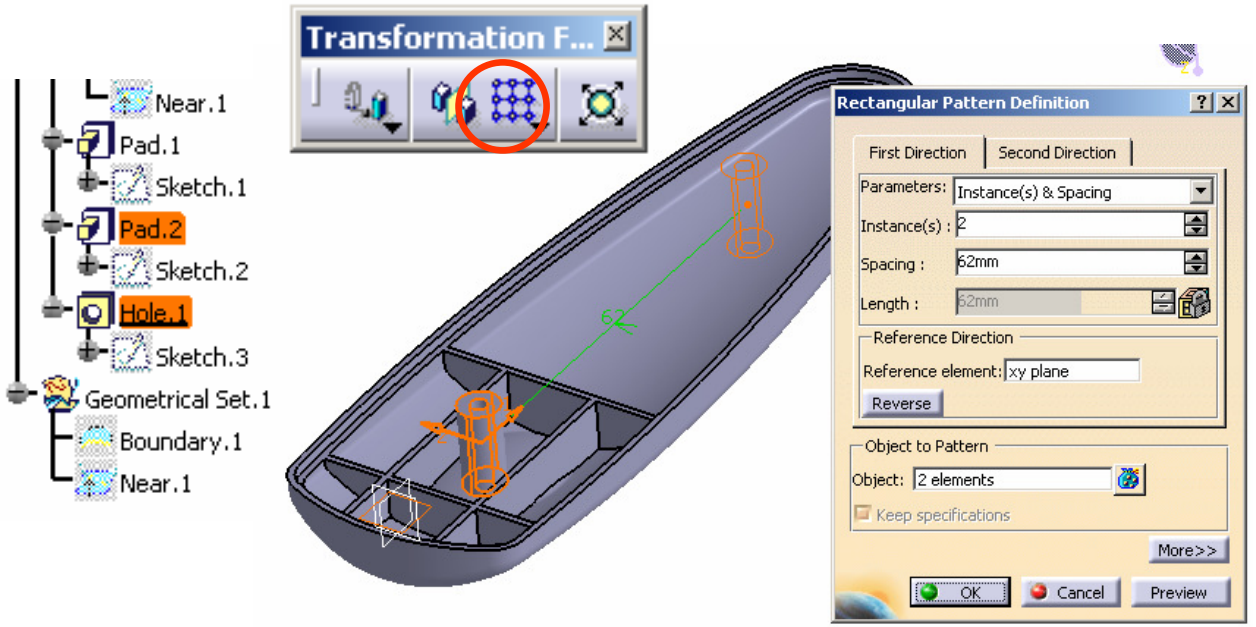
To make a hole:-

- Click "**Hole**" icon and select **the top face of the cylinder**
- Select "Blind" as Extension type
- Enter 2.5mm as Diameter
- Enter 13mm as Depth
- Click ok to complete



To make a pattern:-

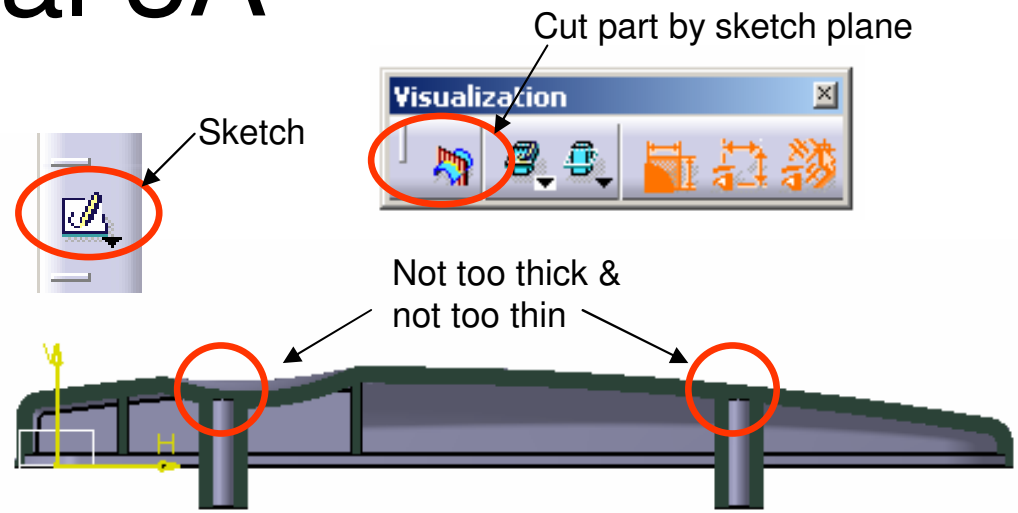
- Multi-select Pad.2 & Hole.1
- Click "**Rectangular Pattern**" icon
- Click the box "Reference Element"
- Select xy plane
- Click "Reverse"
- Enter 2 as Instance
- Enter 62mm as Spacing
- Click ok to complete



Tutorial 3A

To check the depth of holes:-

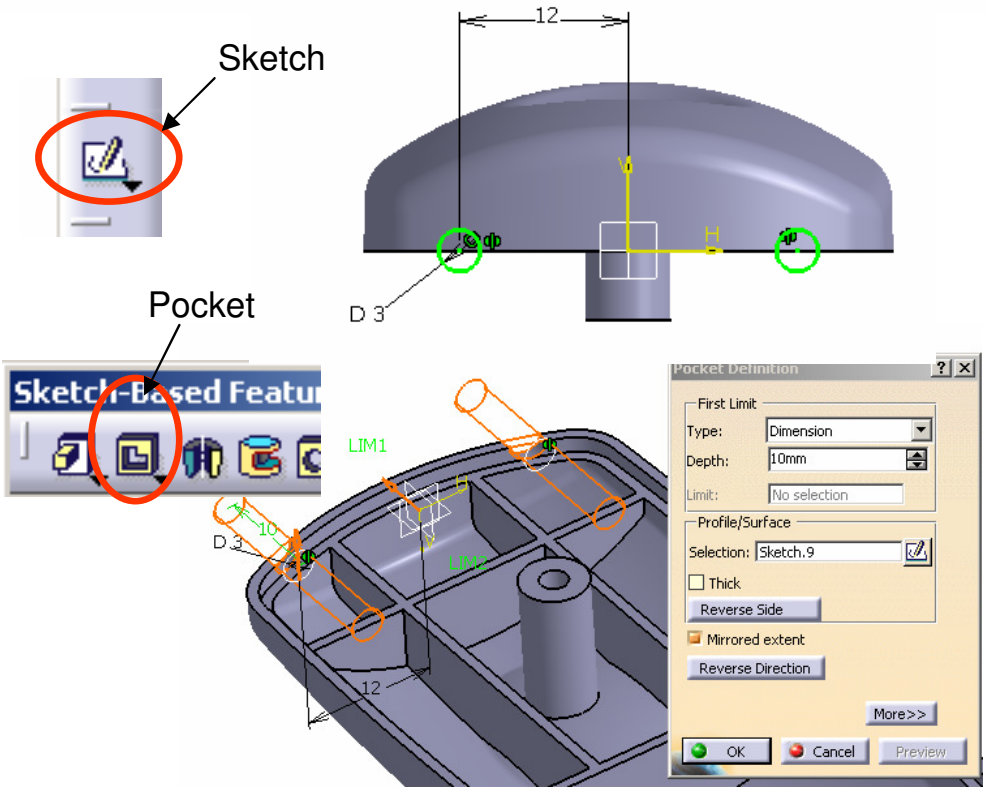
- Click "**Sketch**" icon and select **zx plane**
- Click "Cut Part by Sketch Plane" icon
- Visual Check whether the holes are too deep or not deep enough (for this case, they are accepted)
- Exit the workbench by clicking "**Exit**" icon



To make a pocket:-

- Click "**Sketch**" icon and select **yz plane**
- Draw a **circle** (Dia3.0) at (-12.0mm, 0)
- Select the circle, click "**Mirror**" icon, and then click y-axis
- Exit the workbench by clicking "**Exit**" icon

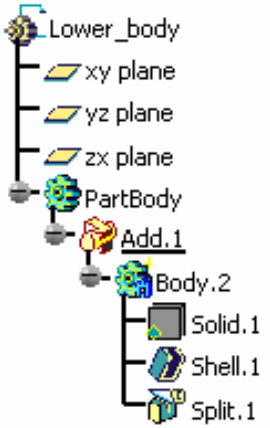
- Click "**Pocket**" icon
- Enter 10mm as First Limit
- Check " Mirrored extent" option
- Click ok to complete



***** SAVE THE FILE AGAIN *****

Tutorial 3A

Building mechanical features on Lower Body:-

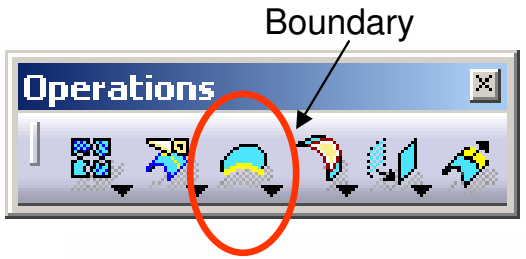


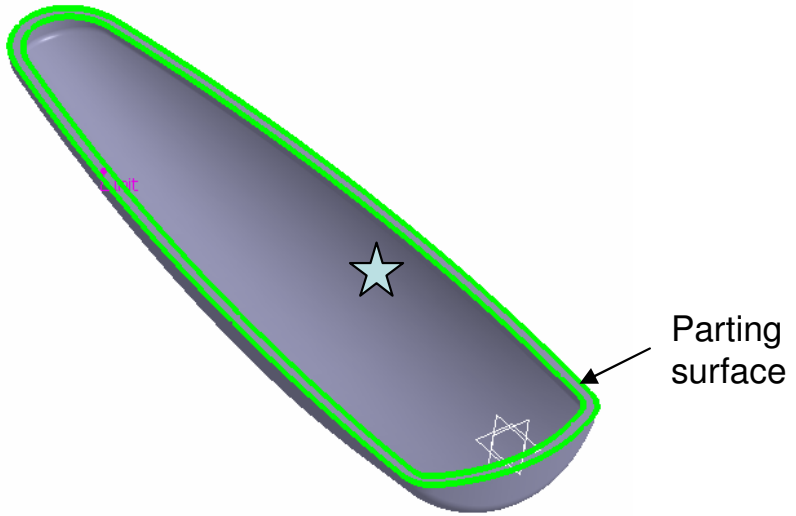
To add Bodies together:-

- Right-click “Body.2” on the tree
- Select Body.2 **object/add…** (Body.2 will become a branch of PartBody)

To get a boundary curve from the solid:-

- Select ‘**Start/Mechanical Design/ Generative Shape Design**” on the menu bar to go back to surface-modeling environment
- Click “**Boundary**” icon
- Select “Point continuity” as propagation type
- Select the parting surface (both inner & outer edges will be highlighted)
- Click ok to complete
- Select “Keep only one sub-element by a Near” in the message window “Multi-result management”
- Click ok
- Select an inner face ☆ as Reference Element
- Click ok to complete

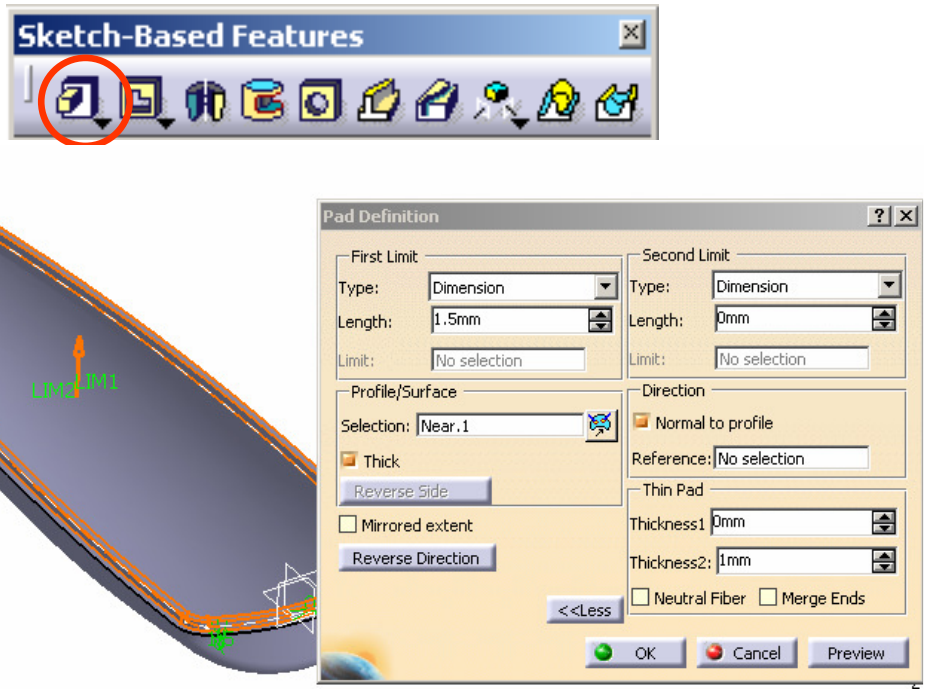




Tutorial 3A

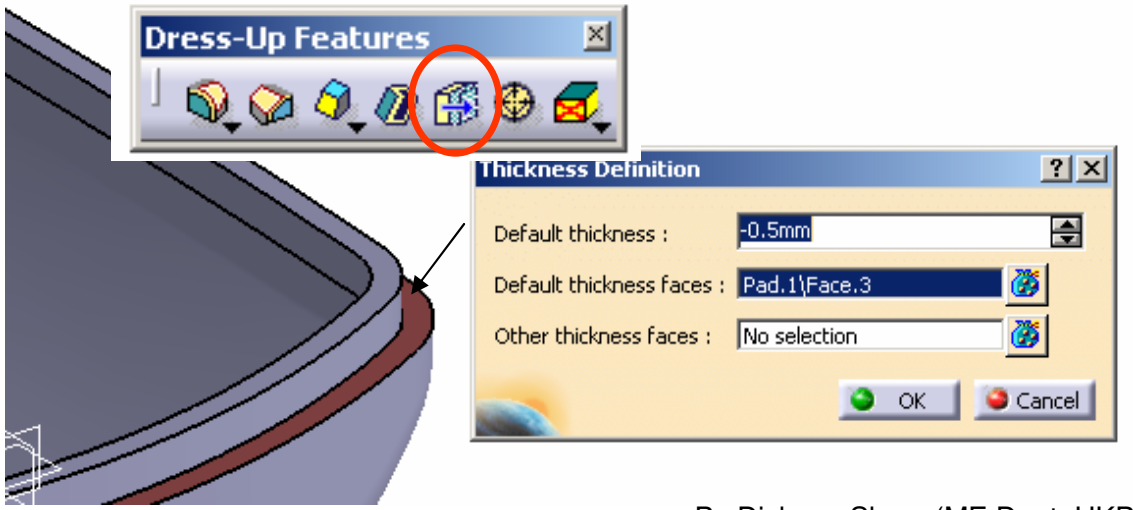
To add material from the boundary:-

- Select '**Start/Mechanical Design/ Part Design**" on the menu bar to go back to solid-modeling environment
- Click "**Pad**" icon
- Click OK on the warning window.
- Select the curve "Near.1"
- Enter 1.5mm as First Limit
- Select "Thick" option
- Enter 0mm as thickness.1
- Enter 1mm as thickness.2
- Click ok to complete



To Offset a solid face:-

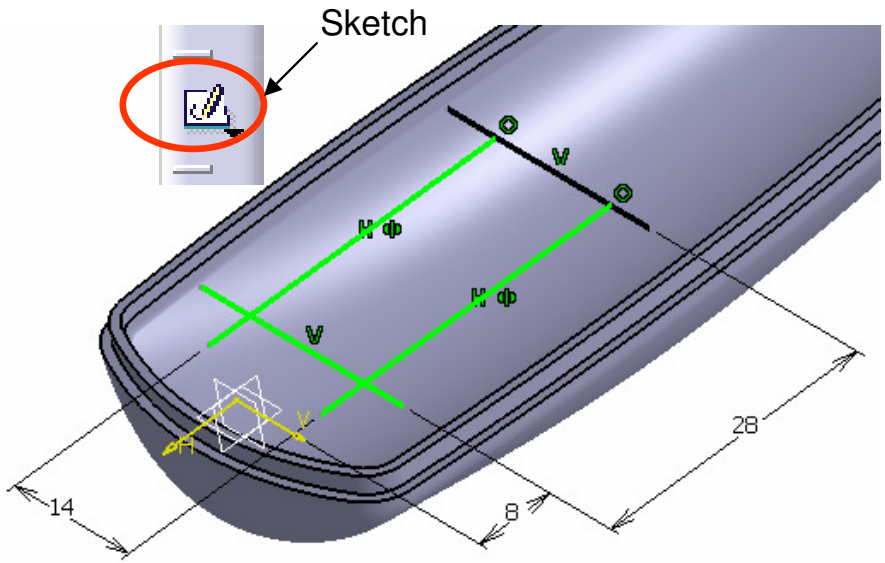
- Click "**Thickness**" icon
- Select the parting surface
- Enter -0.5mm as Default thickness
- Click ok to complete



Tutorial 3A

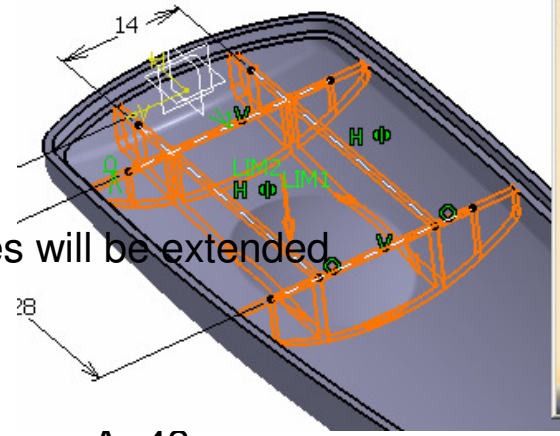
To build a sketch:-

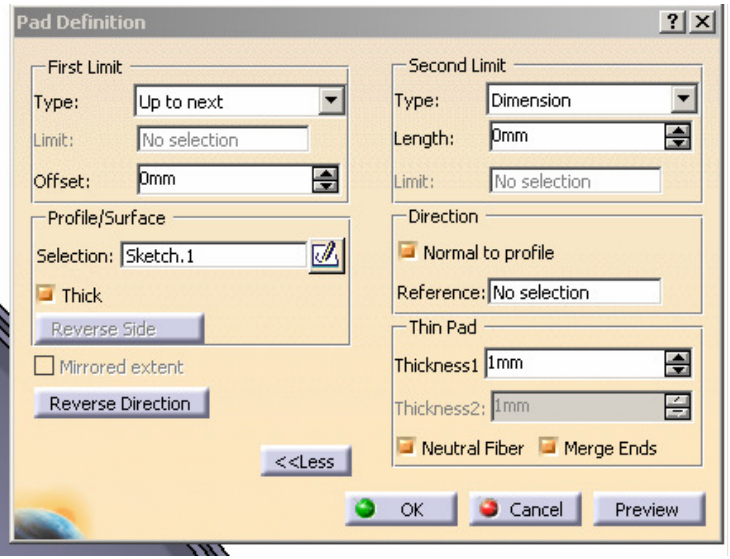
- Click "**Sketch**" icon and select **the top face of Pad.1**
- **Draw** 4 **lines** (two horizontal & two vertical)
- Add a symmetric Constraint between two horizontal lines
- Add 3 more dimensional constraints (28mm, 8mm & 14mm)
- Exit the workbench by clicking "**Exit**" icon



To build a solid from the open profile:-

- Click "**Pad**" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter 1mm as thickness.1
- Click "Reverse Direction"
- Select "Merge Ends" so that the lines will be extended until they touch the solid face
- Select "Up to Next" as First Limit
- Click ok to complete

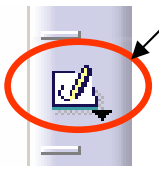


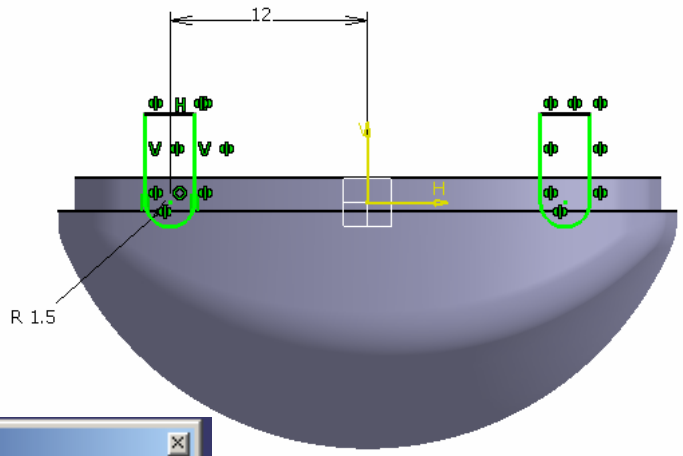


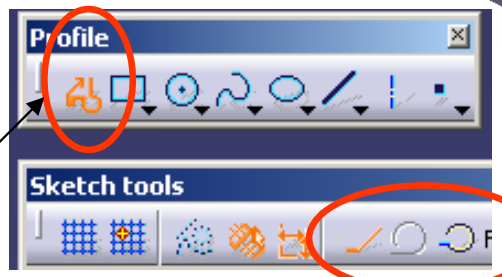
Tutorial 3A

To make a pocket:-

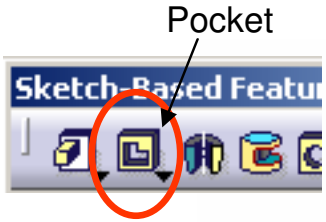
- Click "**Sketch**" icon and select **yz plane**
- Draw a **Profile as shown** (you can use "Profile" command to draw the straight lines continuously, and then click "Tangent Arc" option in the toolbar "Sketch tools" to draw the arc.
- Select the profile, click "**Mirror**" icon, and then click y-axis
- Exit the workbench by clicking "**Exit**" icon

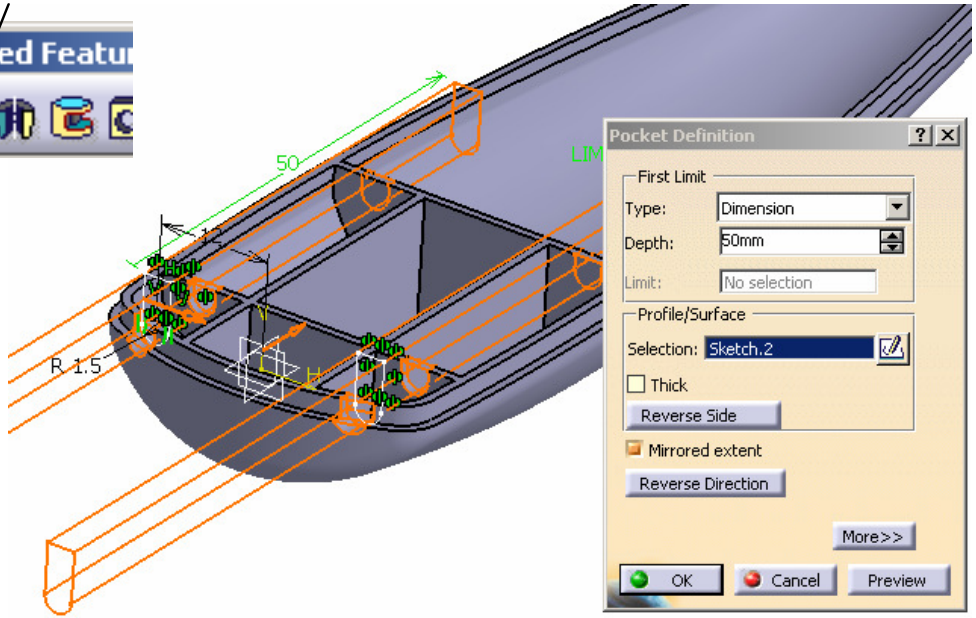


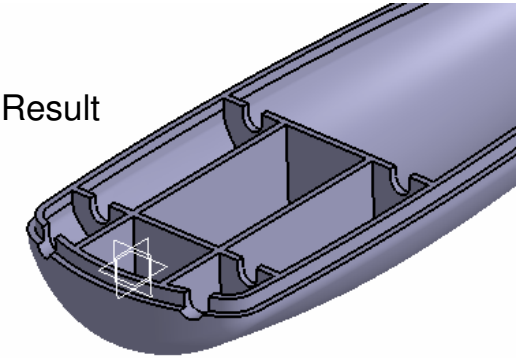




- Click "**Pocket**" icon
- Enter <u>50mm</u> as First Limit
- Check " Mirrored extent" option
- Click ok to complete



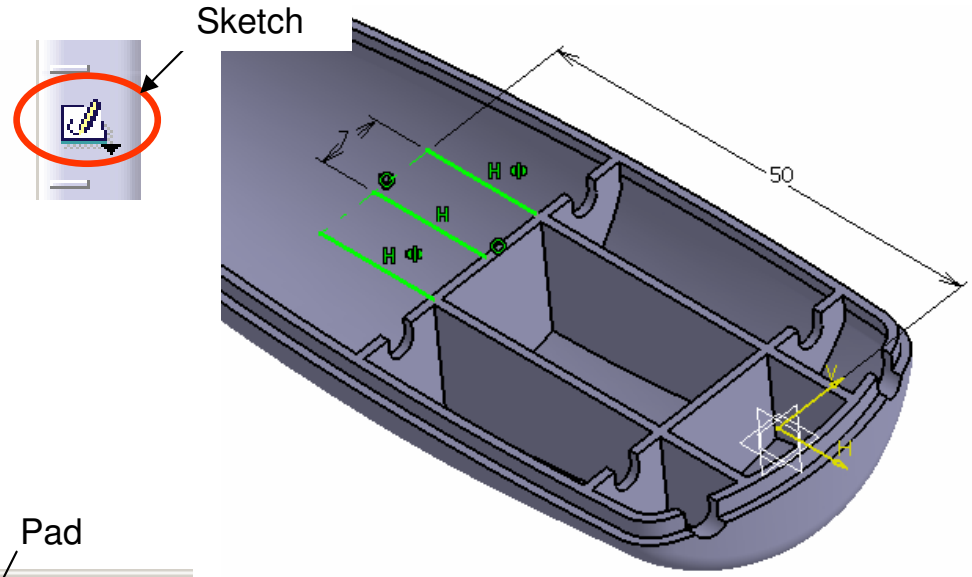




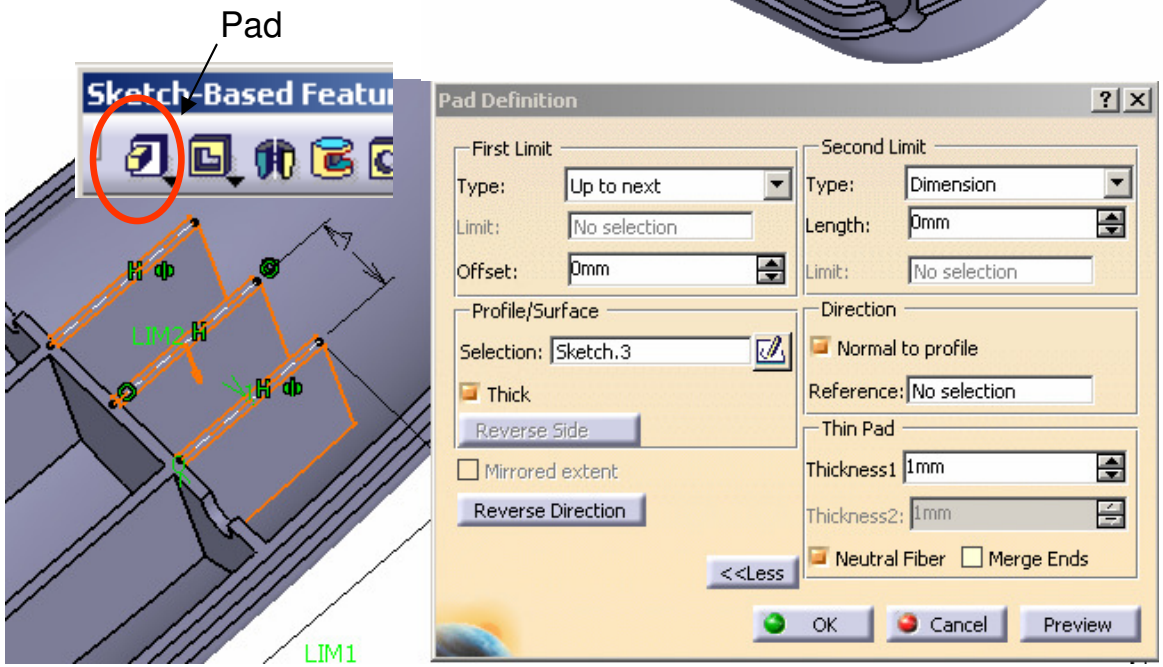
Tutorial 3A

To make a pad:-

- Click "**Sketch**" icon and select **the top face of Pad.1**
- **Draw** 3 **lines** (three horizontal lines)
- Add a symmetric Constraint between two horizontal lines
- Add 2 more dimensional constraints (50mm, & 7mm)
- Exit the workbench by clicking "**Exit**" icon



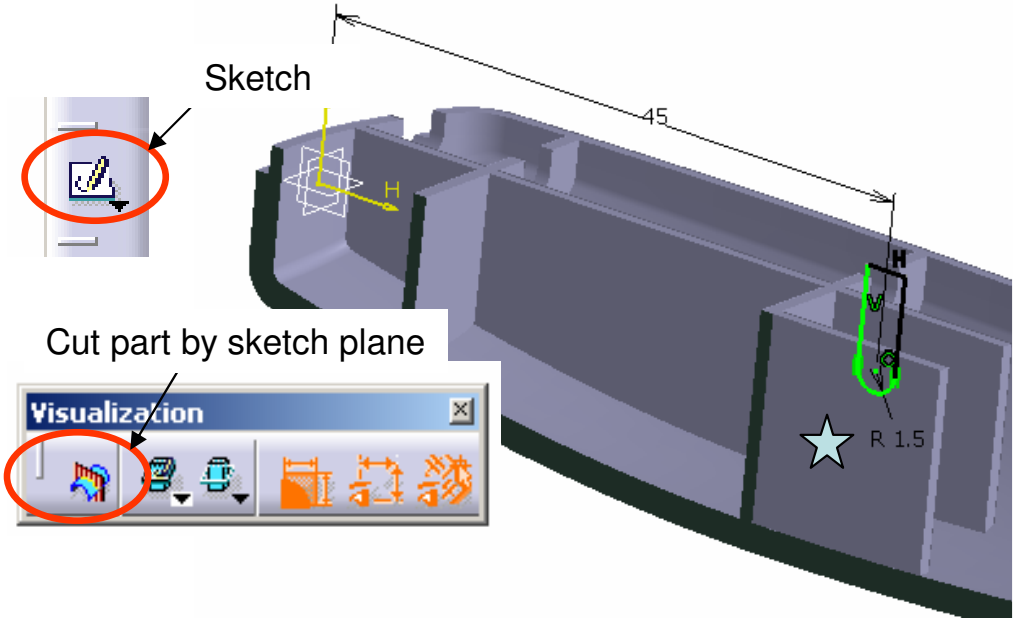
- Click "**Pad**" icon
- Click ok on the warning window
- Select "Thick" option
- Select "Neutral Fiber"
- Enter 1mm as thickness.1
- Click "Reverse Direction"
- Select "Up to Next" as First Limit
- Click ok to complete



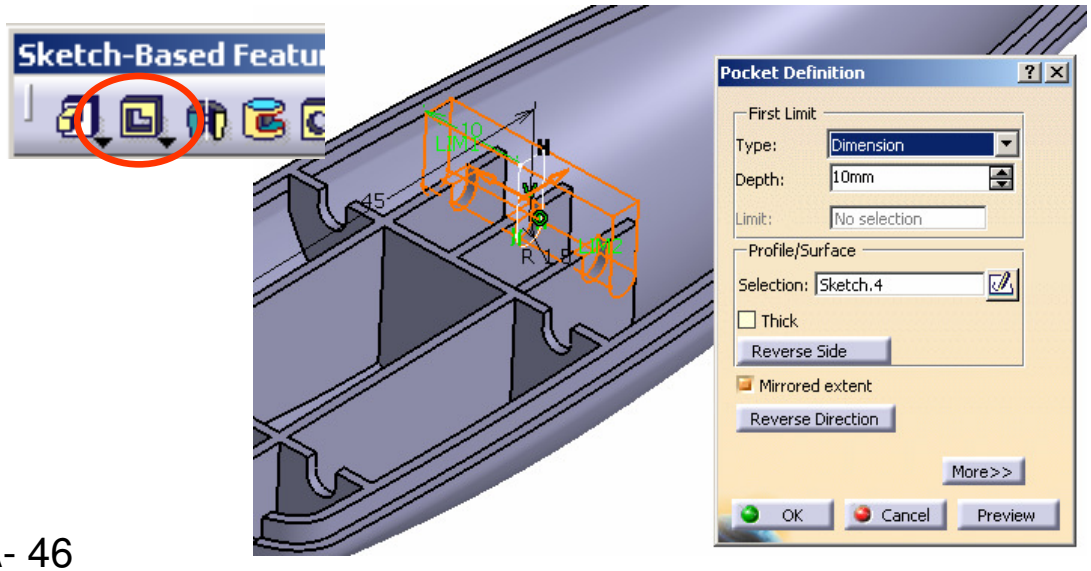
Tutorial 3A

To make a pocket:-

- Click "**Sketch**" icon and select the **face of Pad.3** ☆
- Click **"Cut Part by sketch plane"** icon to view the sketch plane
- **Draw** a **Profile** (two vertical lines, one horizontal line & an arc)
- Add a dimensional constraint R1.5 on the arc
- Add another dimensional constraint (45mm) between the circle centre and the y-axis
- Exit the workbench by clicking "**Exit**" icon



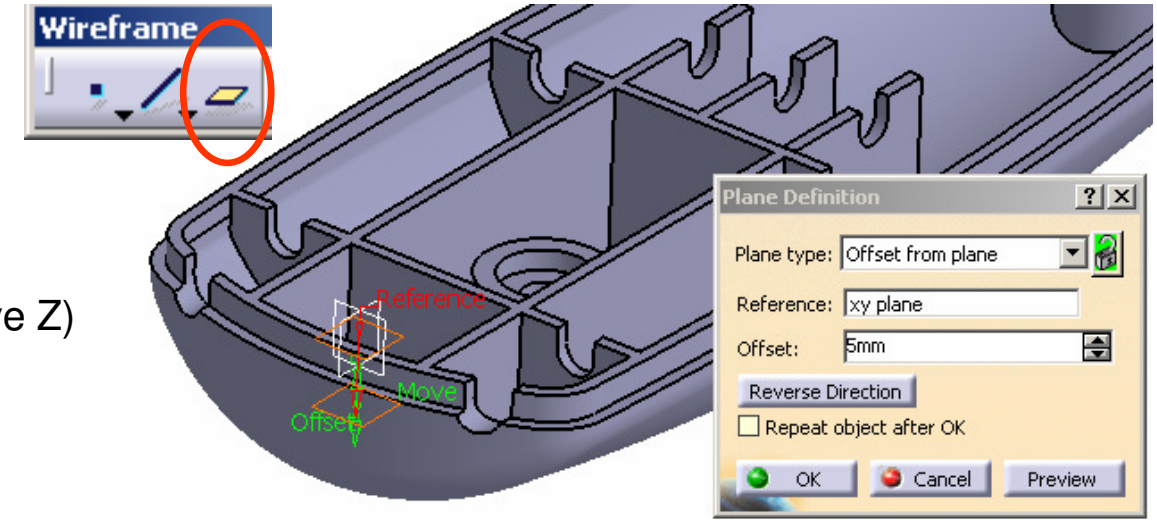
- Click "**Pocket**" icon
- Enter <u>10mm</u> as First Limit
- Check "Mirrored Extent" option
- Click ok to complete



Tutorial 3A

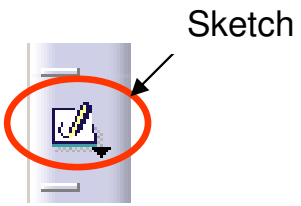
To create a reference plane:-

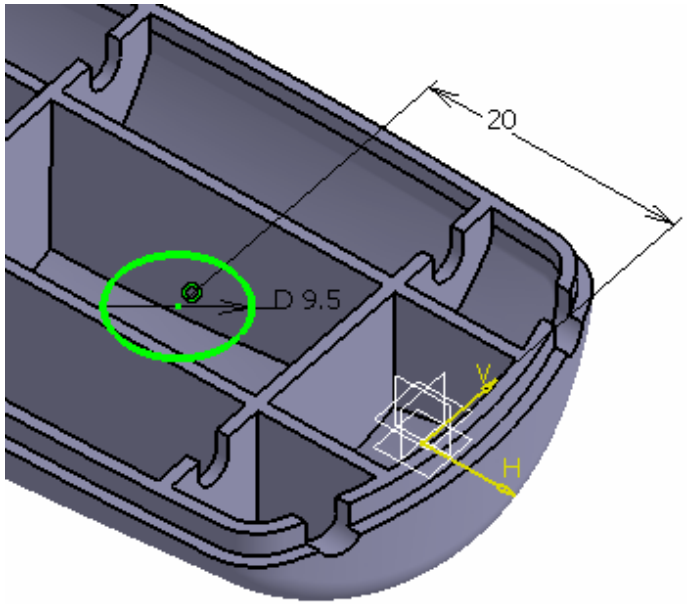
- Click “**plane**” icon
- Select “Offset from plane” as plane type
- Select “xy plane” as Reference
- Click “Reverse Direction” in the command window (The arrow should point to negative Z)
- Enter 2.5mm as Offset value
- Click ok to complete



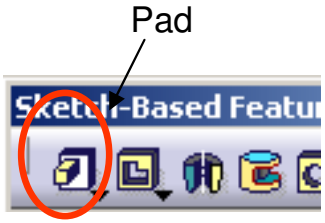
To make a pad:-

- Click “**Sketch**” icon and select **xy plane**
- **Draw** a **Circle** (Dia9.5mm)
- Add a dimensional constraint (20mm) between the circle center and the y-axis
- Exit the workbench by clicking “**Exit**” icon





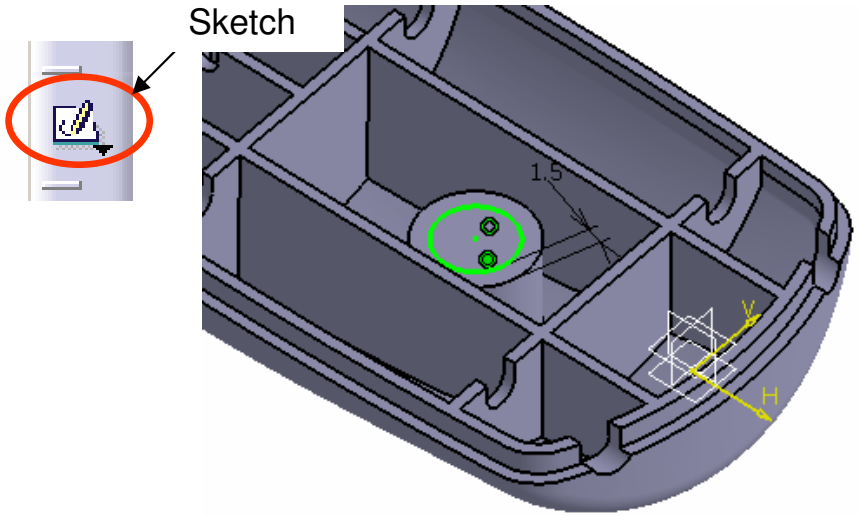
- Click “**Pad**” icon
- Click “Reverse Direction”
- Select “Up to Next” as First Limit Type
- Click ok to complete



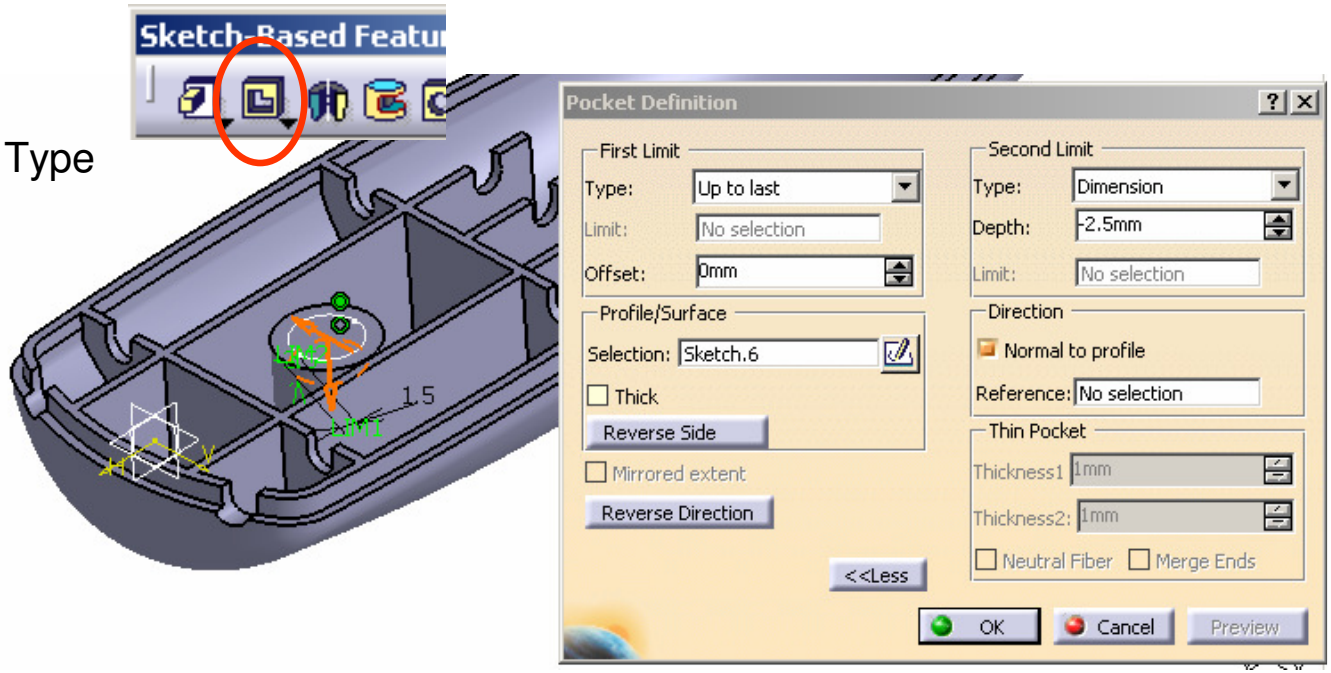
Tutorial 3A

To make a pad:-

- Click "**Sketch**" icon and select the top face of the cylinder
- **Draw** a **circle**
- Add a Concentric Constraint
- Add a dimensional constraint (1.5mm)
- Exit the workbench by clicking "**Exit**" icon



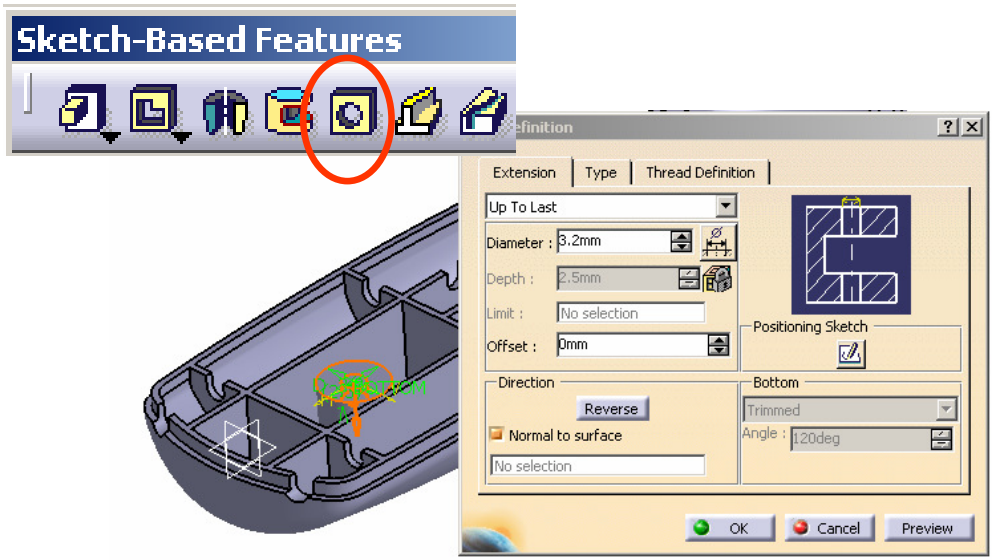
- Click "**Pocket**" icon
- Select "Up to Last" as First Limit Type
- Click "More" option
- Enter -2.5mm as Second Limit
- Click ok to complete



Tutorial 3A

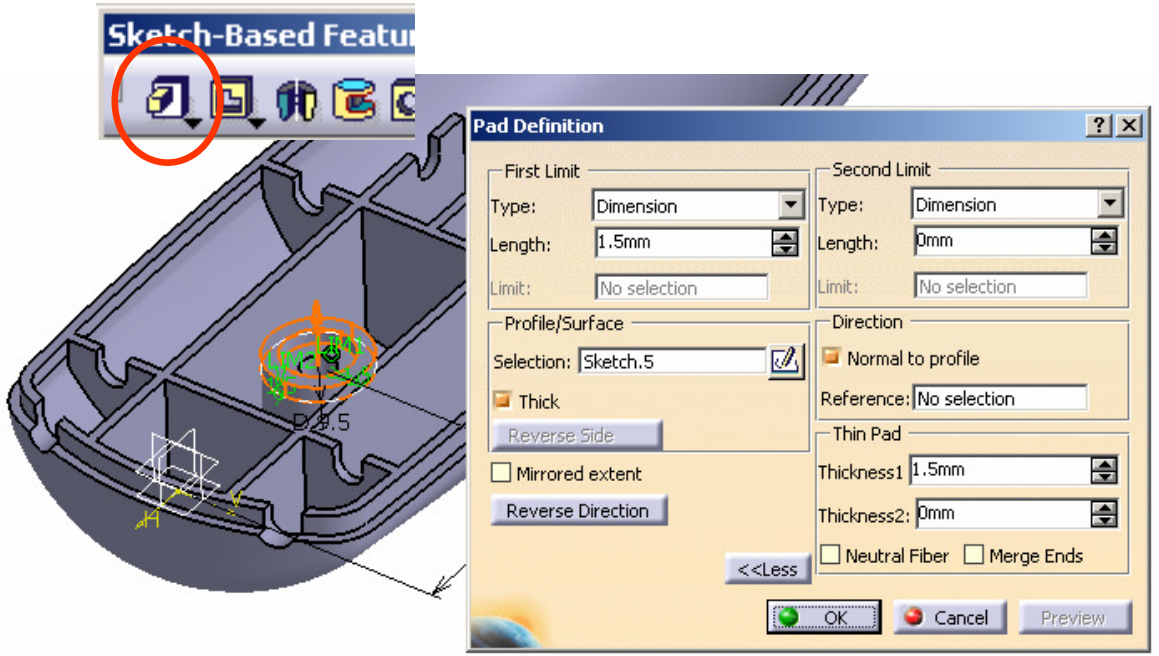
To make a hole:-

- Click "**Hole**" icon and select the top face of the cylinder
- Select "Up to Last" as Extension Type
- Enter 3.2mm as Diameter
- Click ok to complete



To make a solid by an used sketch:-

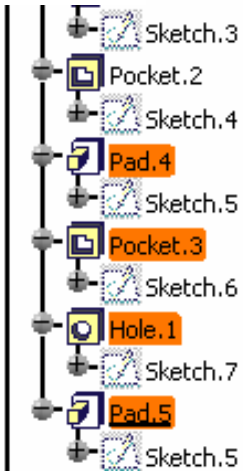
- Click "**Pad**" icon
- Select "Sketch.5" as Profile (Sketch.5 has been used before and it is now hidden)
- Enter 1.5mm as First Limit
- Check "Thick" option
- Enter 1.5mm as Thickness.1
- Enter 0mm as Thickness.2
- Click ok to complete

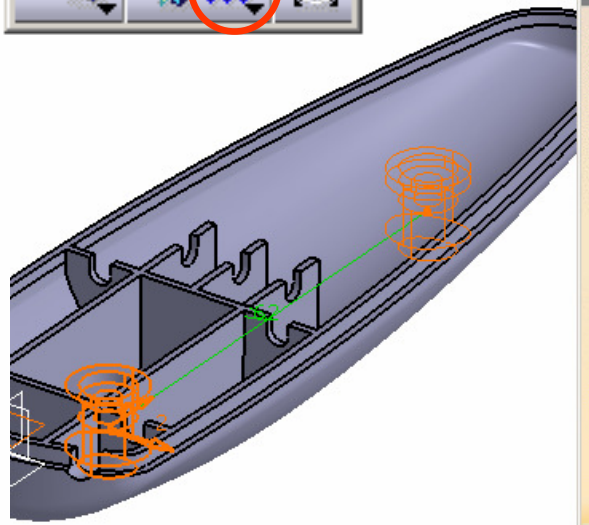


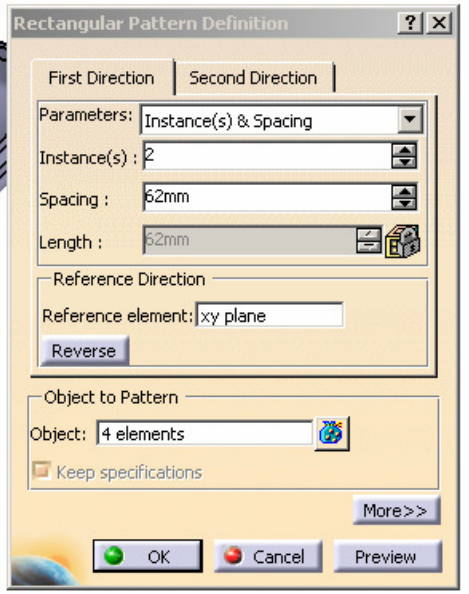
Tutorial 3A

To make a pattern:-

- Multi-select Pad.4, Pocket.3, Hole.1 & Pad.5
- Click "**Rectangular Pattern**" icon
- Click the box "Reference Element"
- Select xy plane
- Click "Reverse"
- Enter 2 as Instance
- Enter 62mm as Spacing
- Click ok to complete







To hide a plane:-

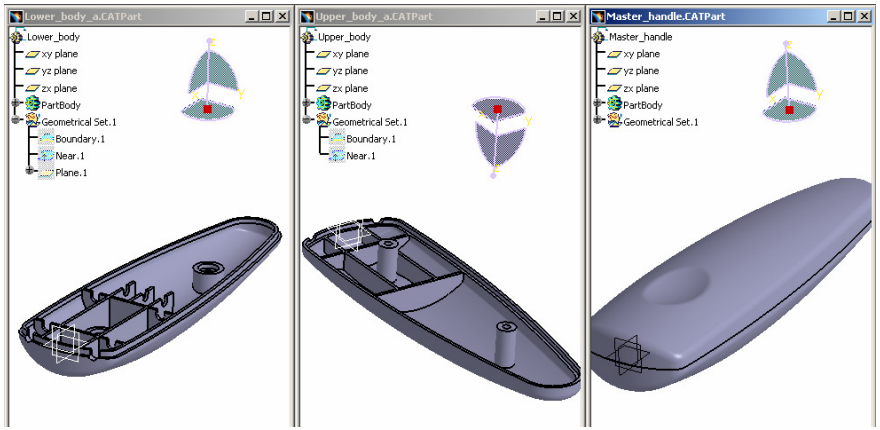
- Right-click on Plane.1
- Select "**Hide/Show**"

*****SAVE THE FILE AGAIN*****

Tutorial 3A

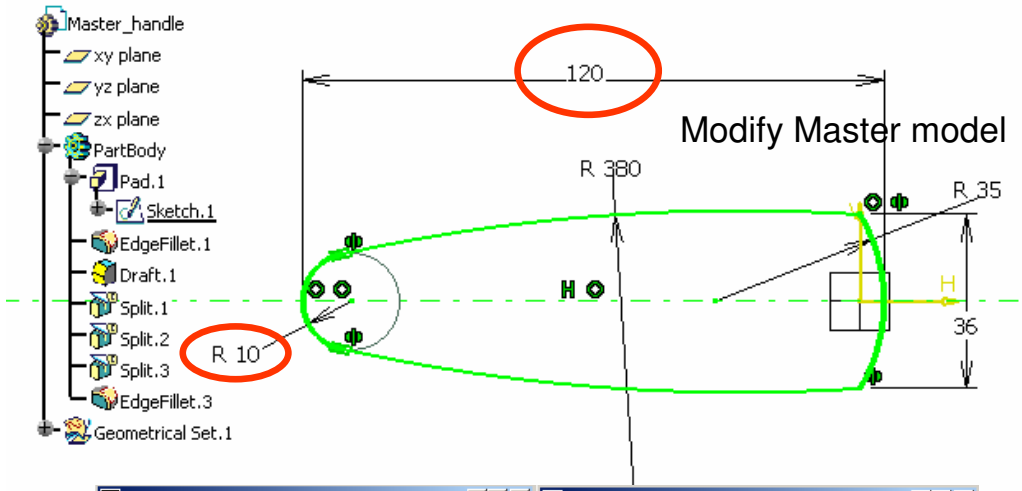
Now we have three part files:

- *Master_handle* (which controls the handle outlook)
- *Upper_body* (which is a child of Master_handle and has its own mechanical features)
- *Lower_body* (which is a child of Master_handle and has its own mechanical features)



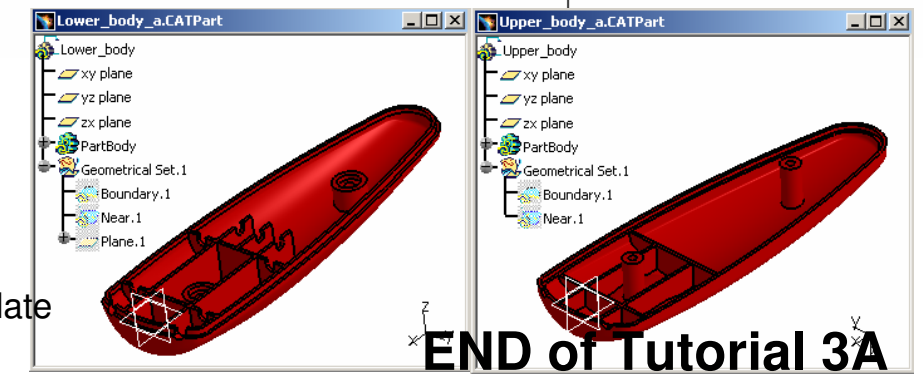
To modify *Master Handle*:-

- Single Click “Sketch.1” in Master_handle
- Change R380 to R250
- Change R10 to R12
- Exit the workbench by clicking “**Exit**” icon



To get Upper & Lower bodies updated:-

- Activate the window of *Upper_body*
- Click “**Update**” icon
- After a few second, the model turns from red to blue; by then the model is updated in shape

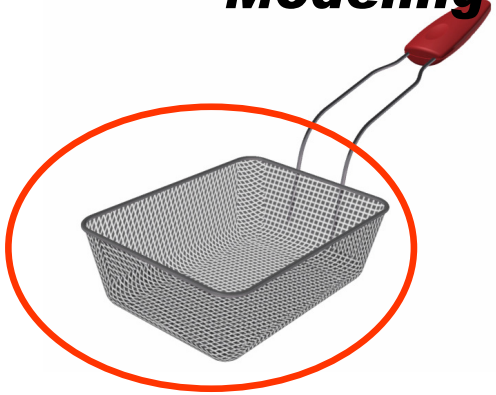


- For *Lower body*, the steps are the same.

update

END of Tutorial 3A

Tutorial 3B

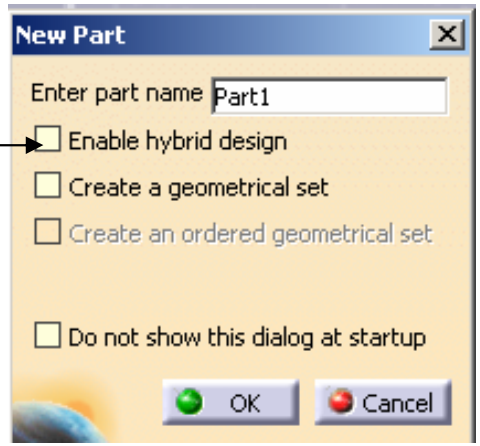


Building the basket:-

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

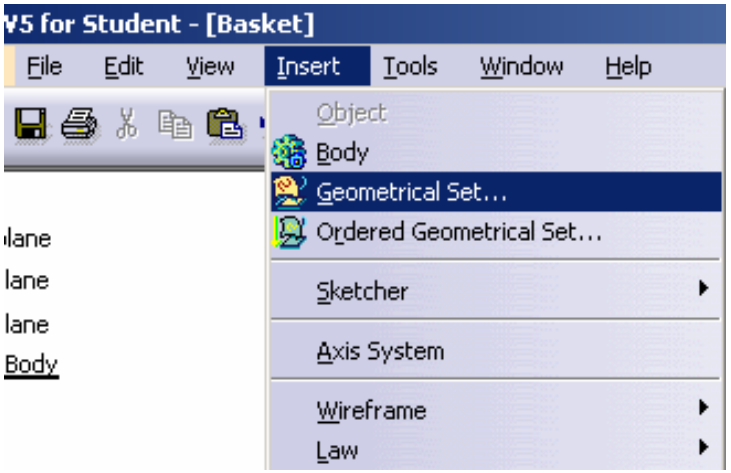
- Select **File/New** on the menu bar
- Select "Part" in "List of Type
- Click ok to complete
- Enter "Basket" as Part Name
- Click ok to complete

Don't check "Hybrid Design"



- Select '**Start/Mechanical Design/Generative Shape Design**"on the menu bar.

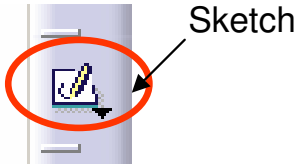
- Select "**Insert/Geometrical Set**" on the menu bar and click ok to complete (Now a new branch "Geometrical set" is created on the part tree, which is used to store all reference curves and surfaces)

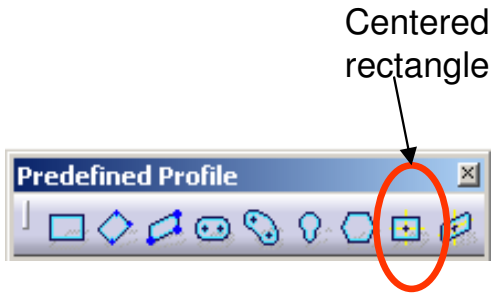


Tutorial 3B

To build a sketch:-

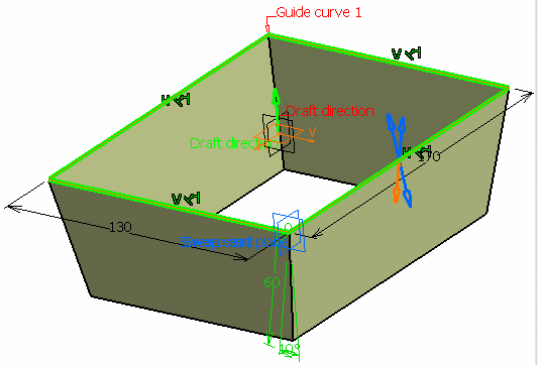
- Click "**Sketch**" icon and select **xy plane**
- **Draw** a **centered rectangle** (center at origin, 170mm x 130mm)
- Exit the workbench by clicking "**Exit**" icon

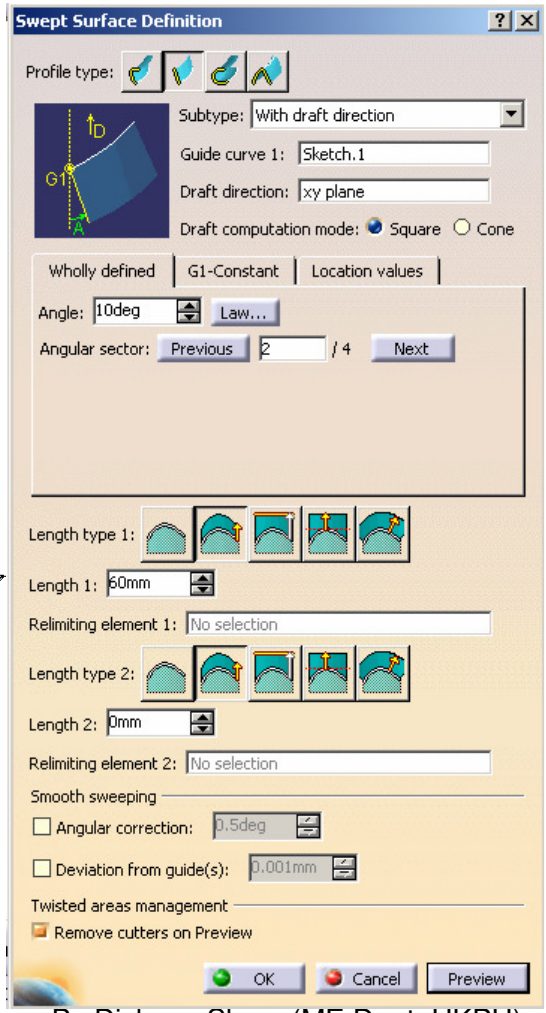




To create a swept surface:-

- 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

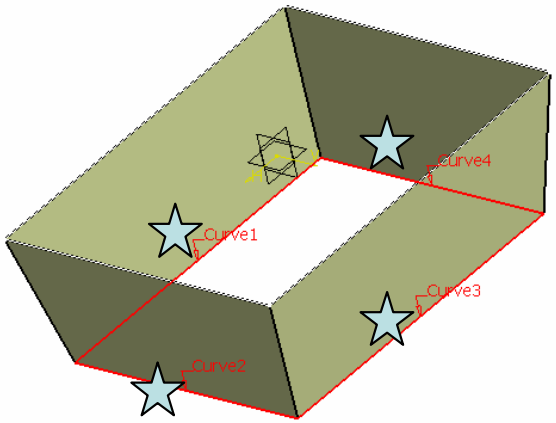


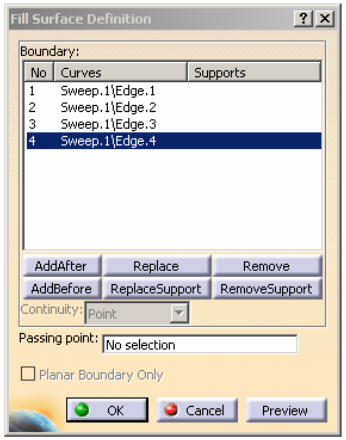


Tutorial 3B

To create a surface from a closed boundary:-

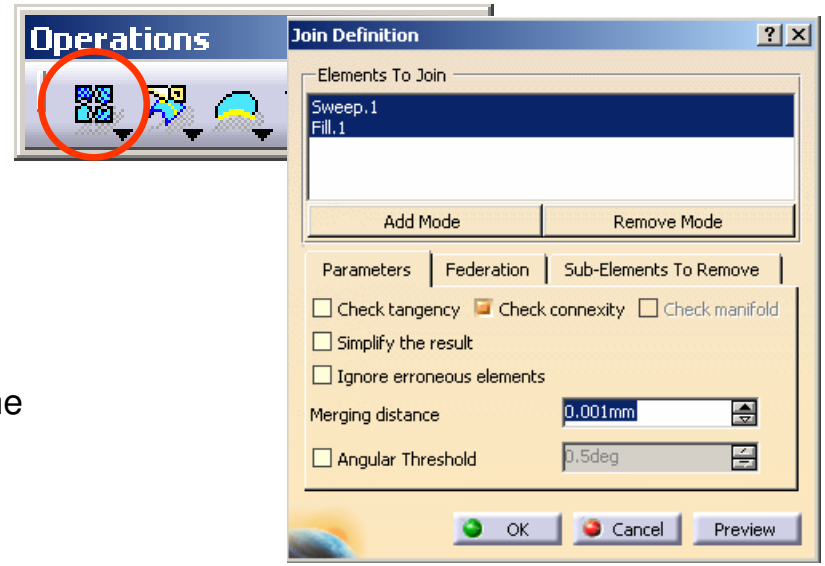
- Click "**Fill**" icon
- Select all the four edges ☆ of the smaller opening
- Click ok to complete (a surface will be created to fill the opening)

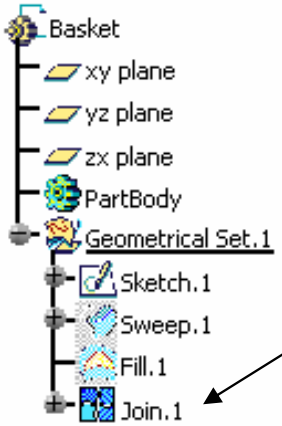




To Join surfaces into one:-

- 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)



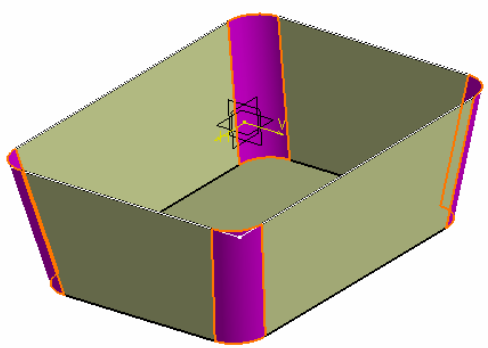


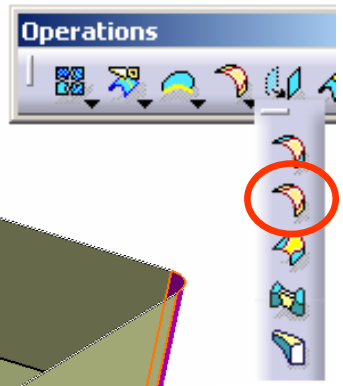
A new surface is created; the original surfaces are hidden after this creation

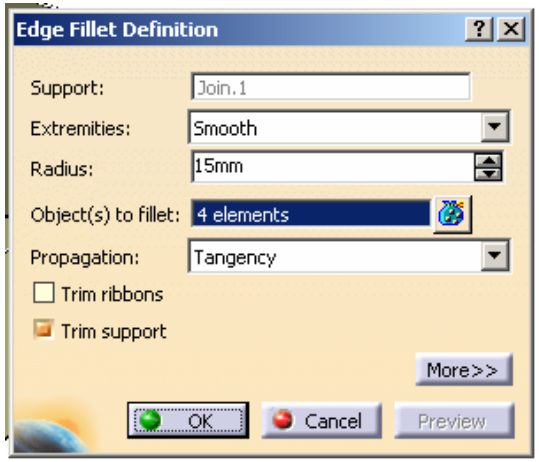
Tutorial 3B

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

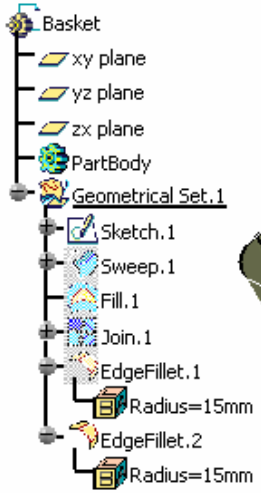


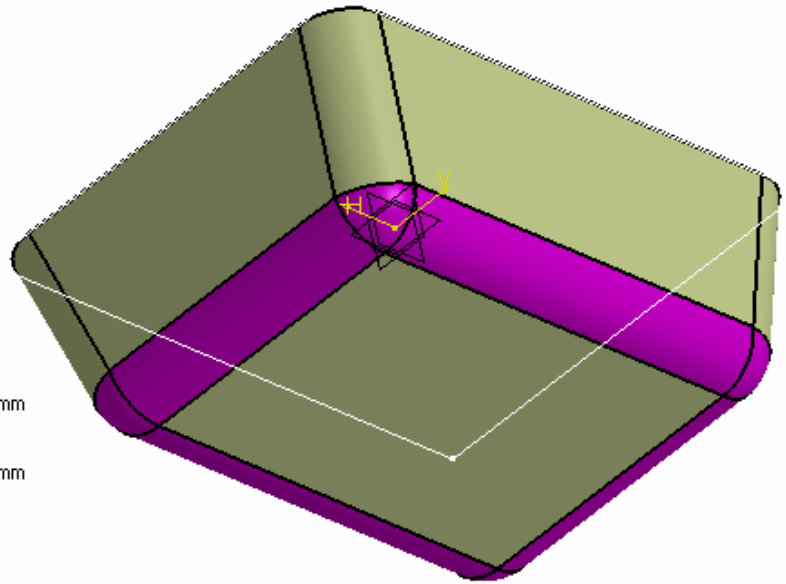




To add another Fillet on surface edges:-

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





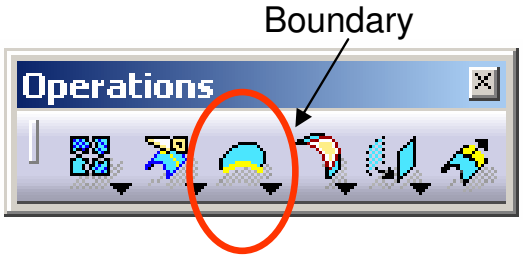
Tutorial 3B

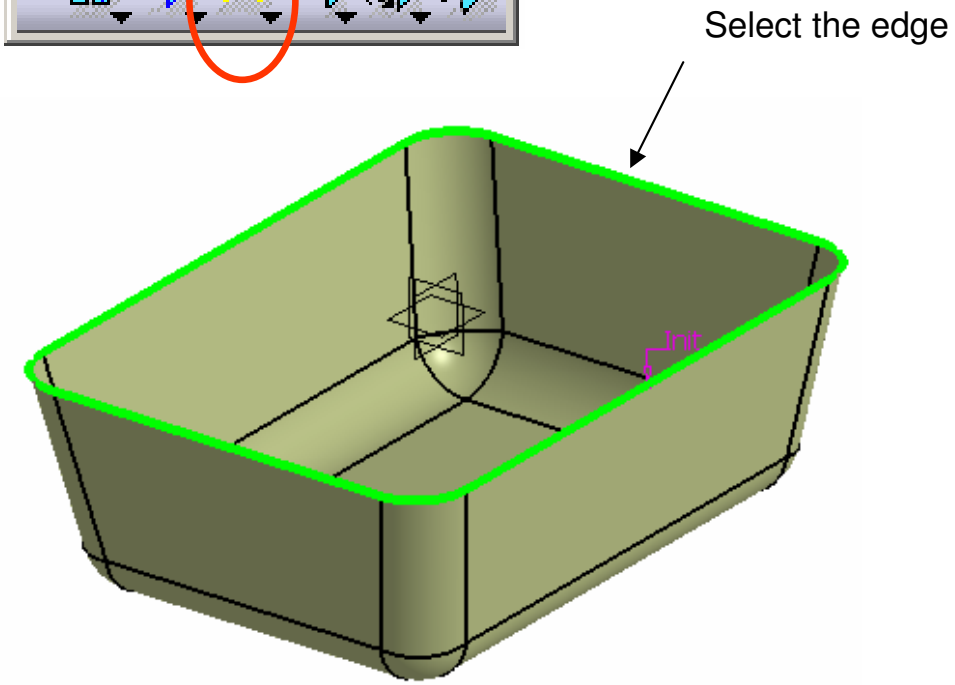
To hide a sketch:-

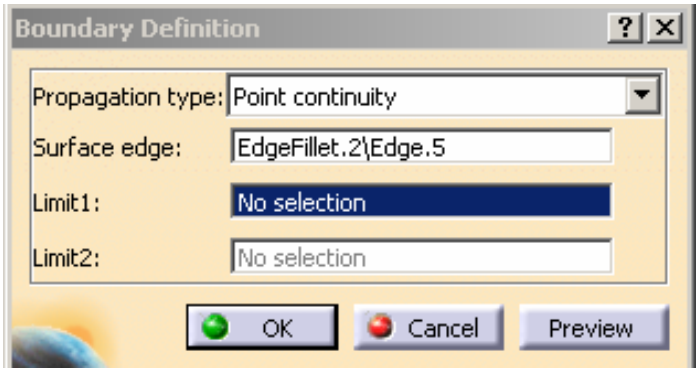
- Right-Click on “Sketch.1”
- Select “**Hide/Show**”

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



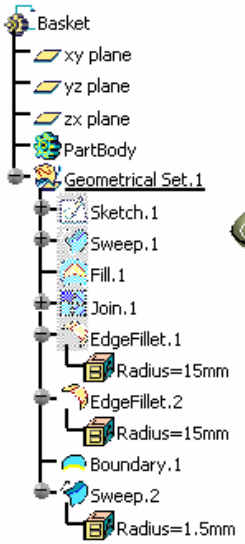


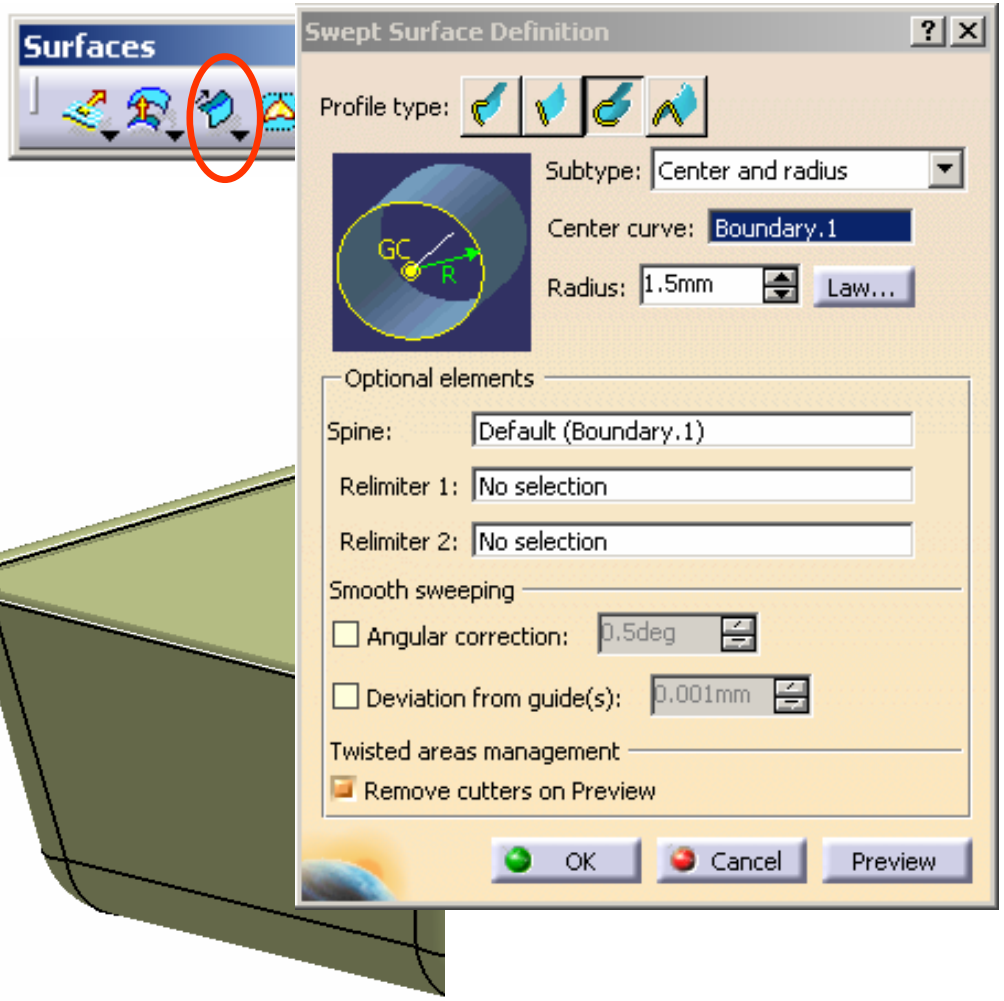


Tutorial 3B

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





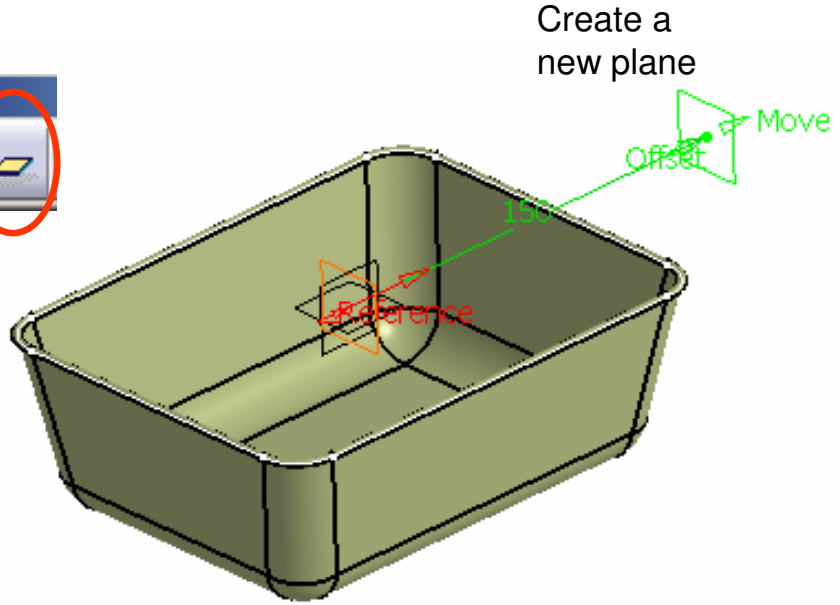
To hide a curve:-

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

Tutorial 3B

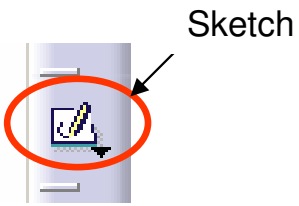
To create a reference plane:-

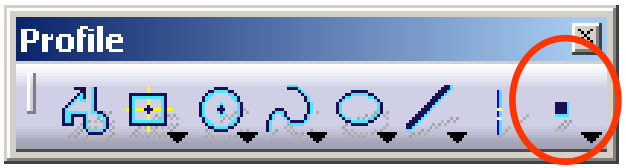
- Click "**plane**" icon
- Select "Offset from plane" as plane type
- Select "yz plane" as Reference
- Click "Reverse Direction" in the command window (The arrow should point to negative X)
- Enter 150mm as Offset value
- Click ok to complete

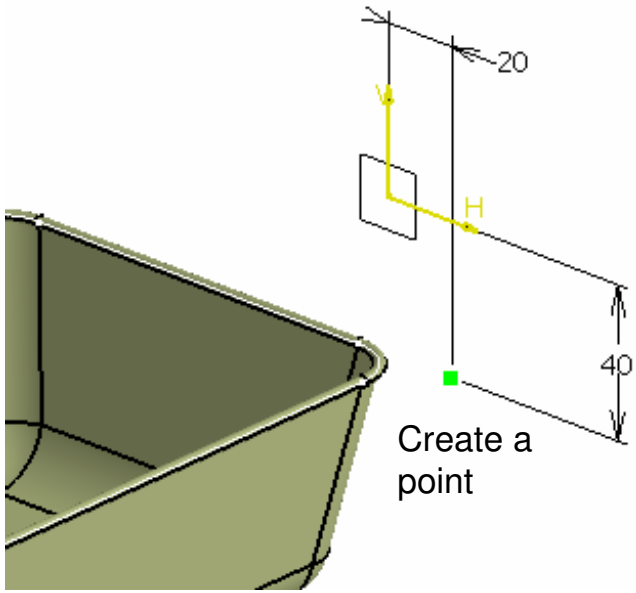


To make a point on a new sketch:-

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



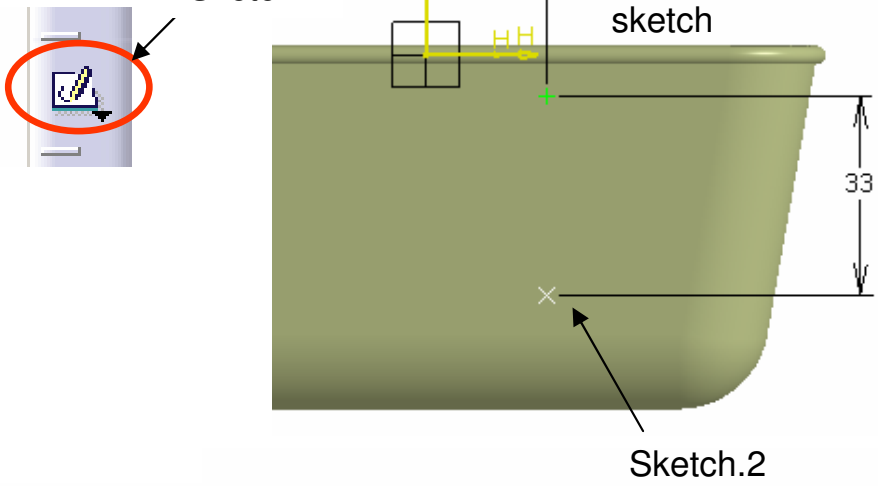




Tutorial 3B

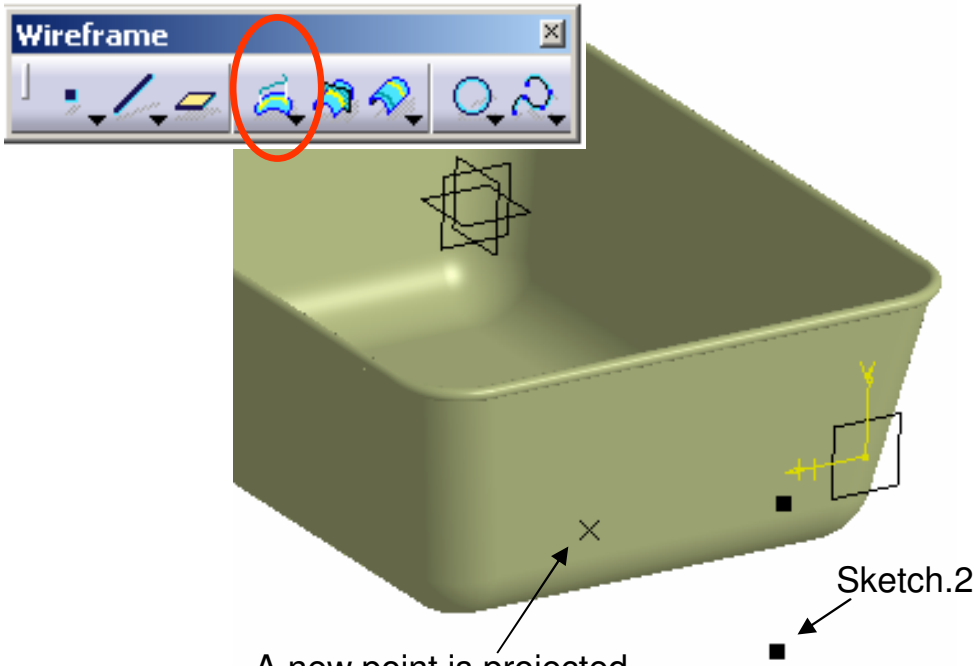
To make another point on a new sketch:-

- *Click on empty space to deselect "Sketch.2"*
- Click "**Sketch**" icon and select **plane.1**
- **Draw** a **point** (x=20, y= 33 above the previous point)
- Exit the workbench by clicking "**Exit**" icon



To project a point onto a surface:-

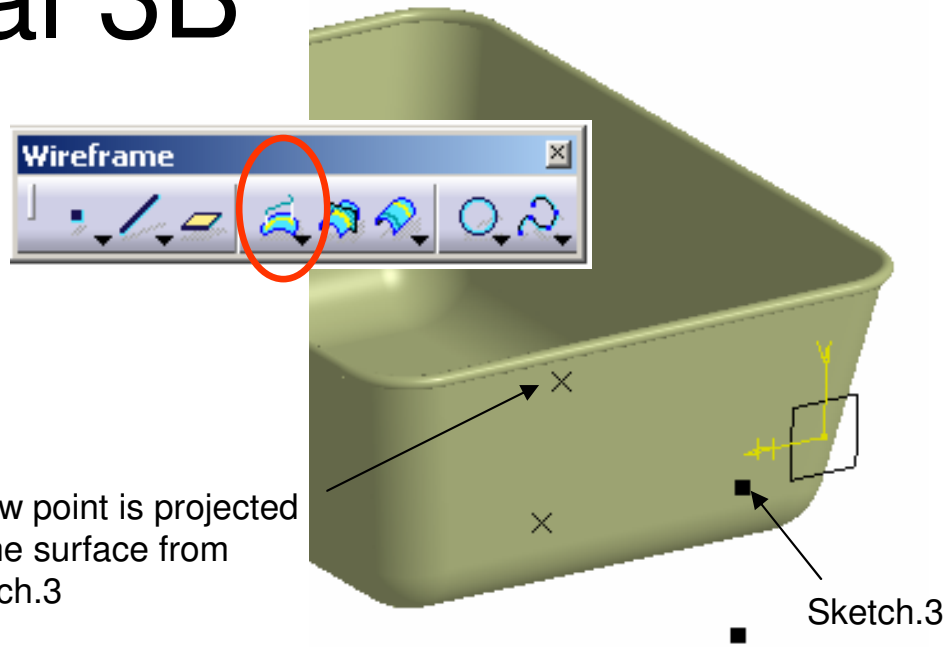
- Click "**Projection**" icon
- Select "Along a direction" as Projection Type
- Select the point "Sketch.2" as Projected
- Select the surface "Edgefillet.2" as Support
- Select Plane.1 as Direction
- Click ok to complete



Tutorial 3B

To project another point onto a surface:-

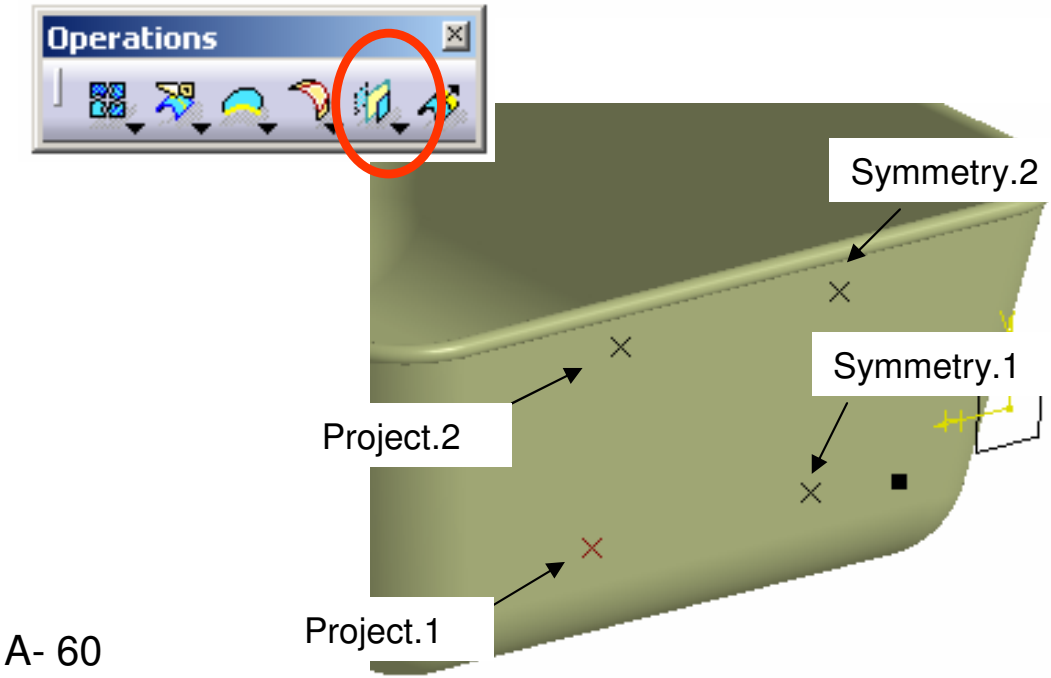
- Click "**Projection**" icon again
- Select "Along a direction" as Projection Type
- Select the point "Sketch.3" as Projected
- Select the surface "Edgefillet.2" as Support
- Select Plane.1 as Direction
- Click ok to complete



To make a mirror copy:-

- Click "**Symmetry**" icon
- Select the point "Project.1" as Element
- Select zx plane as Reference
- Click ok to complete

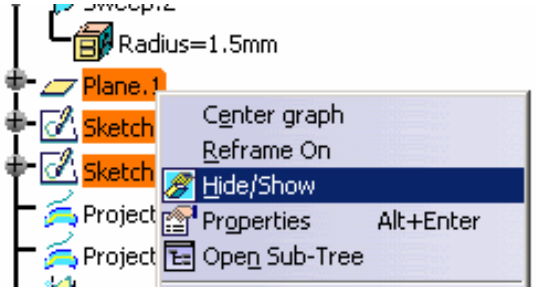
- Click "**Symmetry**" icon again
- Select the point "Project.2" as Element
- Select zx plane as Reference
- Click ok to complete



Tutorial 3B

To hide all reference elements:-

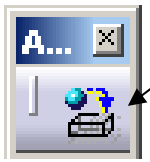
- Multi-select Plane.1, Sketch.2 & Sketch.3
- Right-click on anyone
- Select "**Hide/Show**"



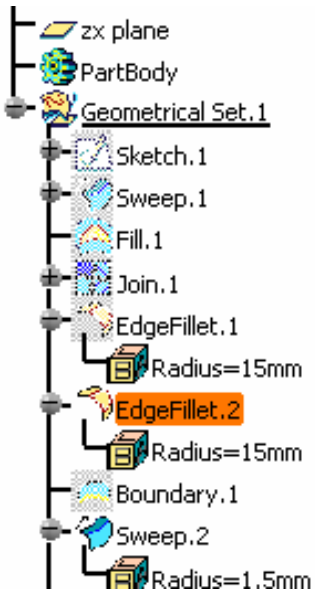
To add Material Texture:-

- Download a texture from a shared library at
  http://www.planit3d.com/source/texture_files/metal/metal.html

- 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



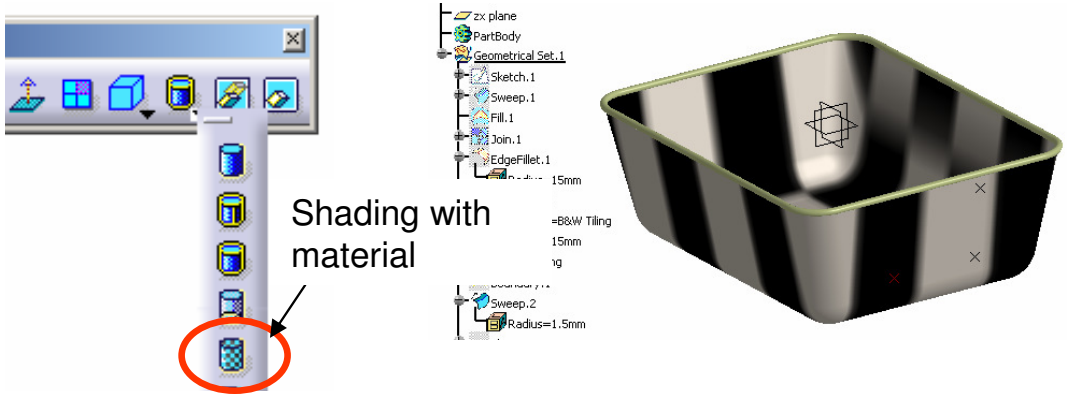
Apply Material



Tutorial 3B

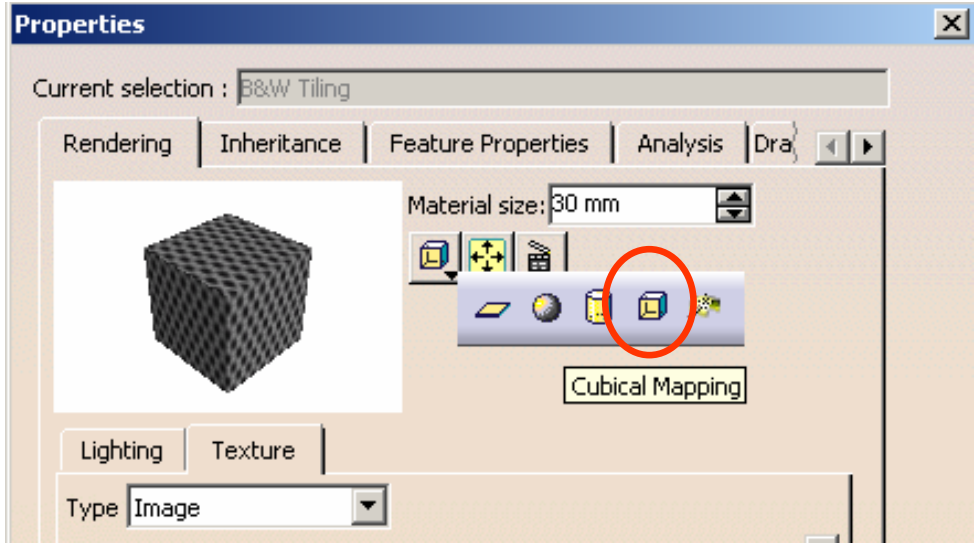
To view the material texture:-

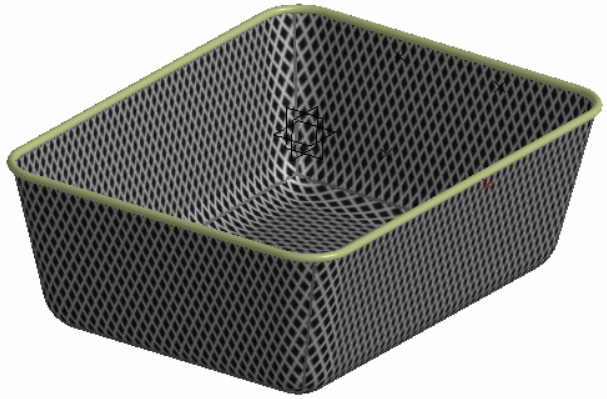
- Click "**Shading with material**" icon



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

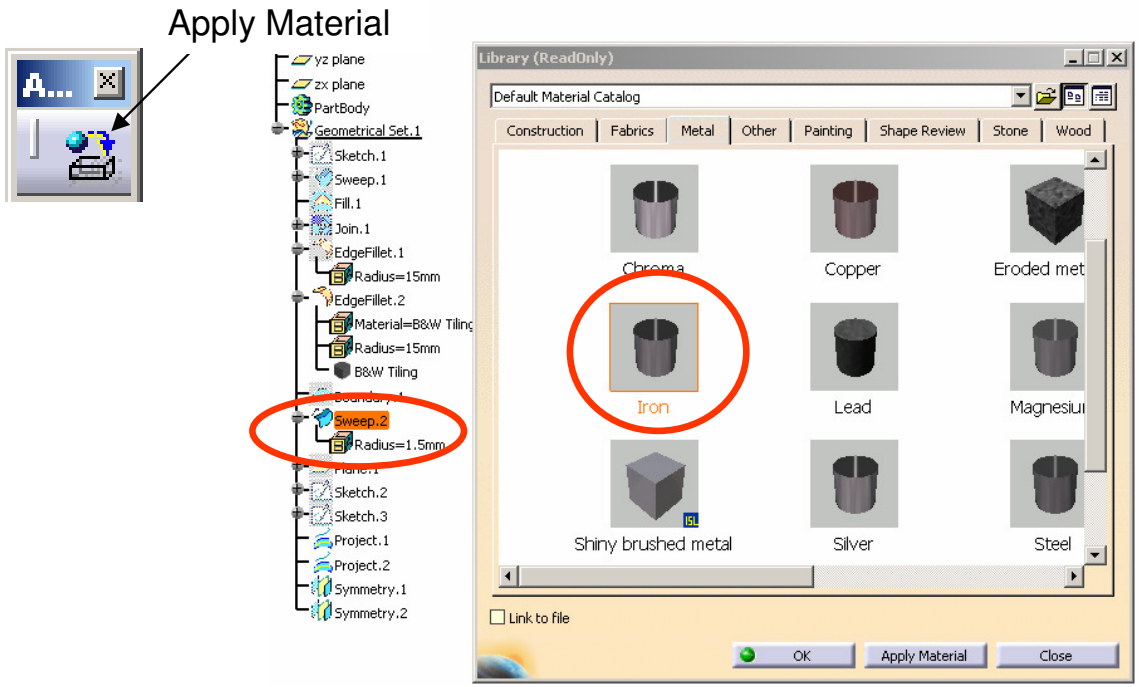




Tutorial 3B

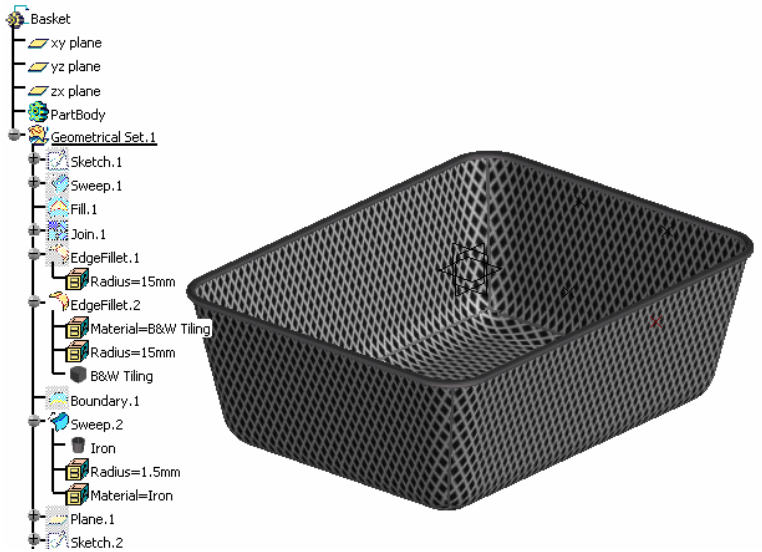
To add another material texture:-

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

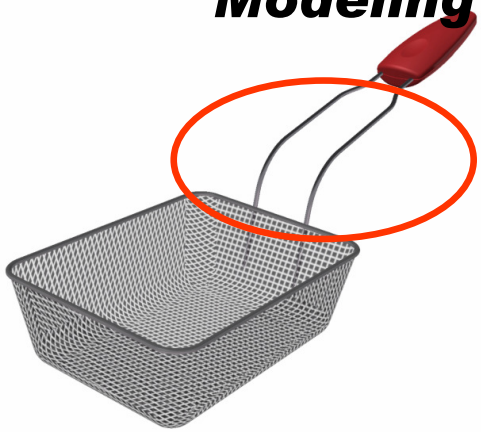


To save the new part in Project Folder:-

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



Tutorial 3B

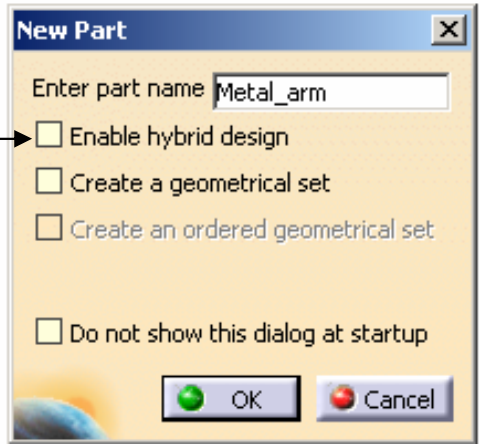


Building the metal arm:-

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

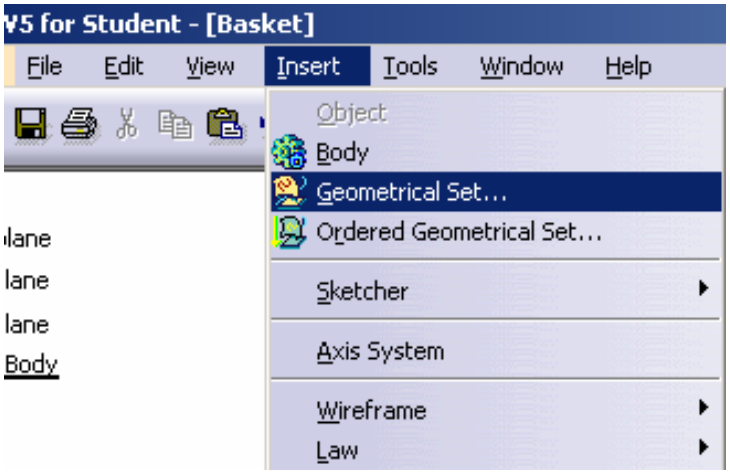
- Select "**File/New**" on the menu bar
- Select "Part" in "List of Type
- Click ok to complete
- Enter "Metal_arm" as Part Name
- Click ok to complete

Don't check "Hybrid Design"

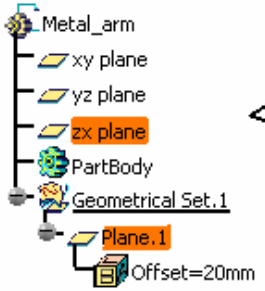


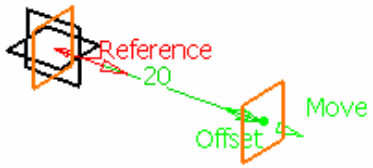
- Select '**Start/Mechanical Design/Generative Shape Design**"on the menu bar.

- Select "**Insert/Geometrical Set**" on the menu bar and click ok to complete



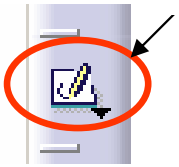
Tutorial 3B





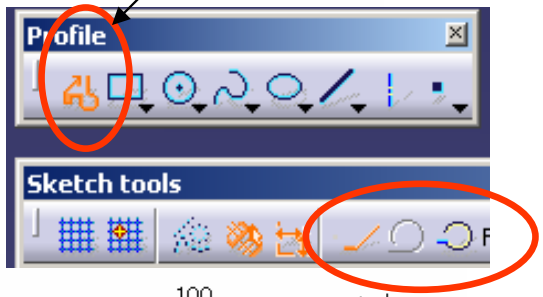
To create a reference plane:-

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



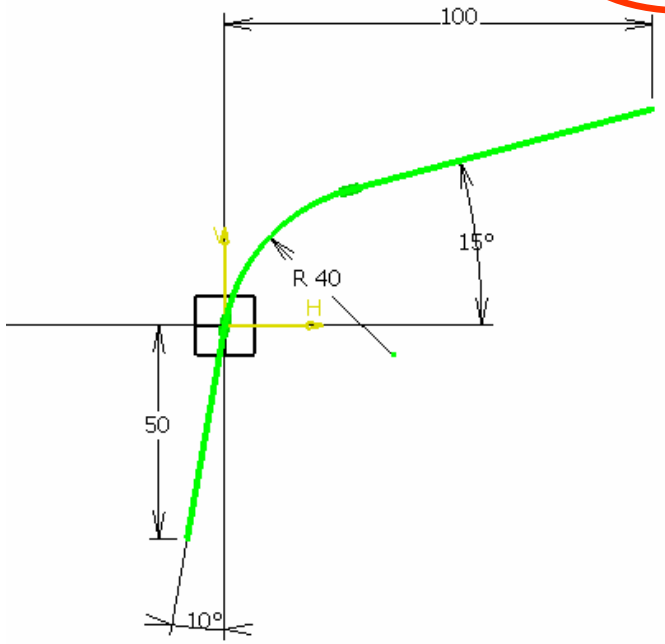
Sketch

Profile



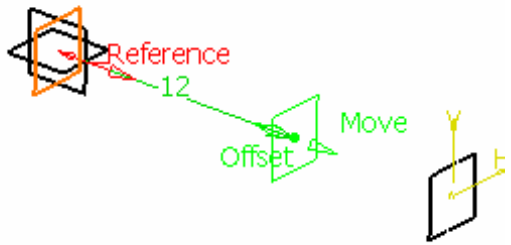
To make a Sketch:-

- 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



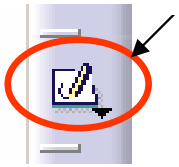
Tutorial 3B

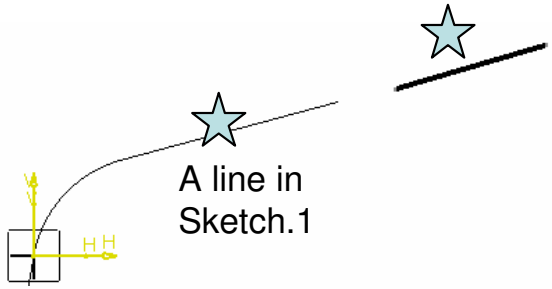
To create another reference plane:-



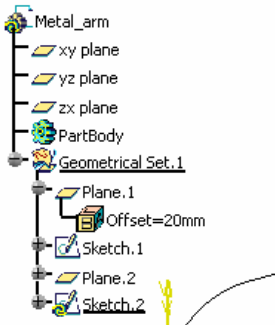
- Click “**plane**” icon
- Select “Offset from plane” as plane type
- Select “zx plane” as Reference
- Enter 12mm as Offset value
- Click ok to complete

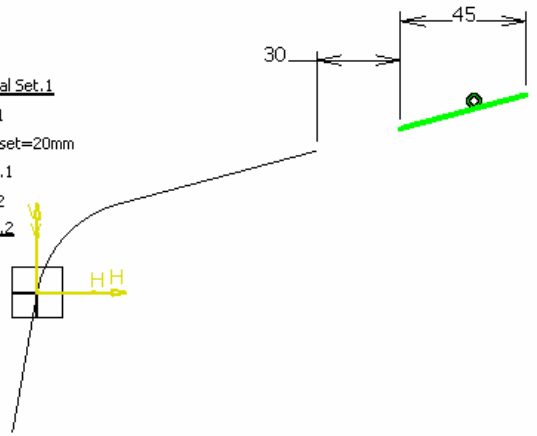
To make the 2nd Sketch:-





- 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

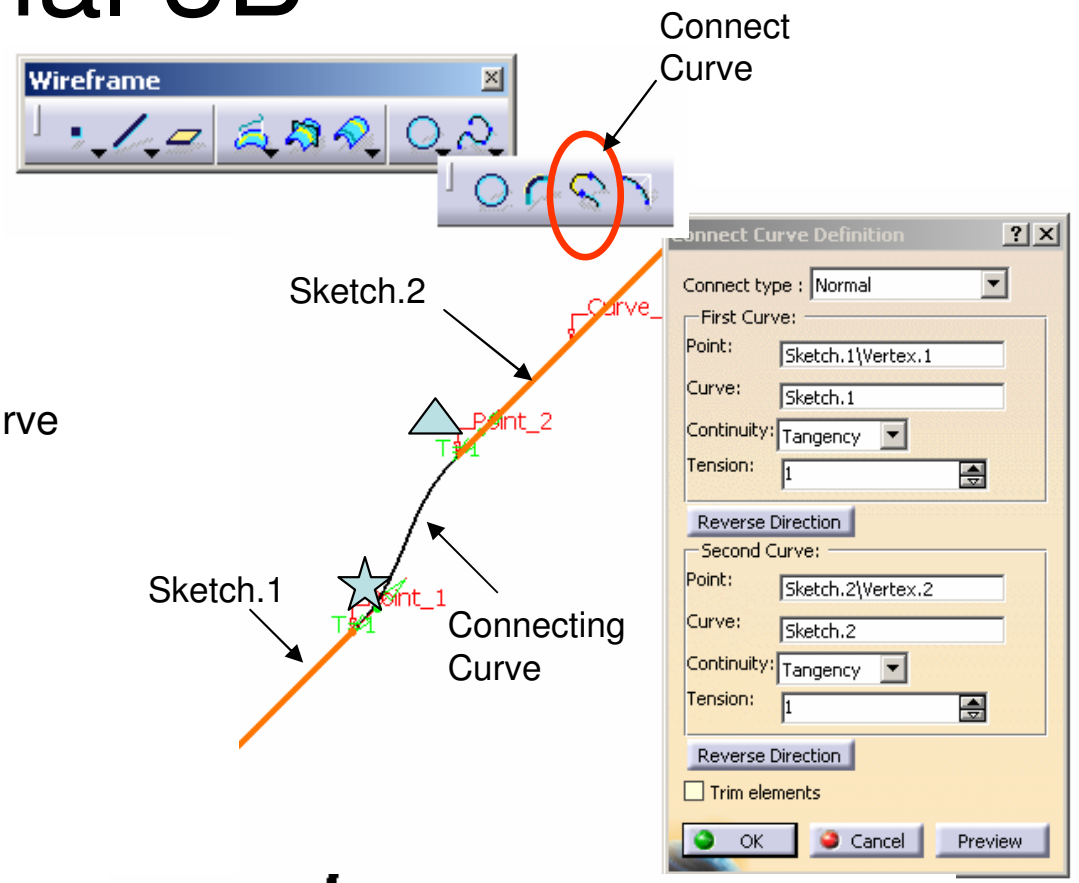




Tutorial 3B

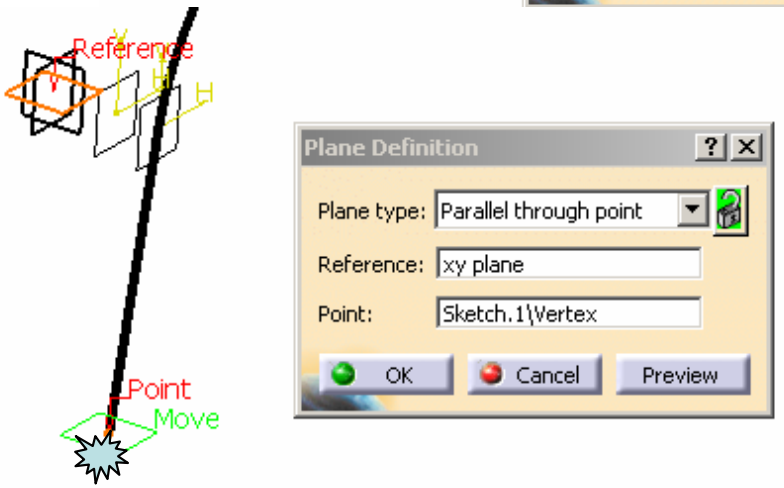
To create a connecting 3D curve:-

- 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:-

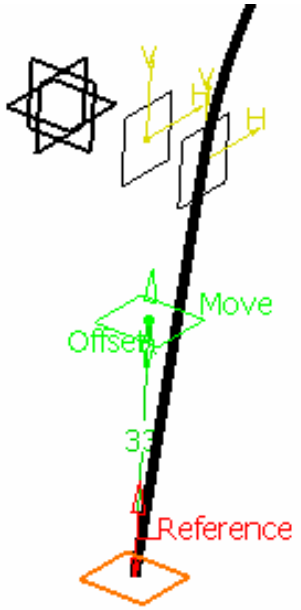
- Click “**plane**” icon
- Select “Parallel through point” as plane type
- Select “xy plane” as Reference
- Select the endpoint of Sketch.1
- Click ok to complete

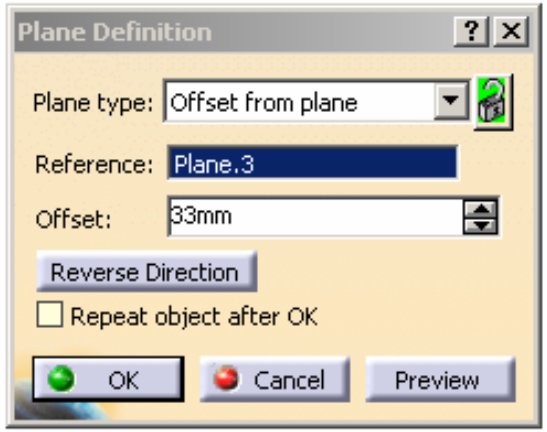


Tutorial 3B

To create a reference plane:-

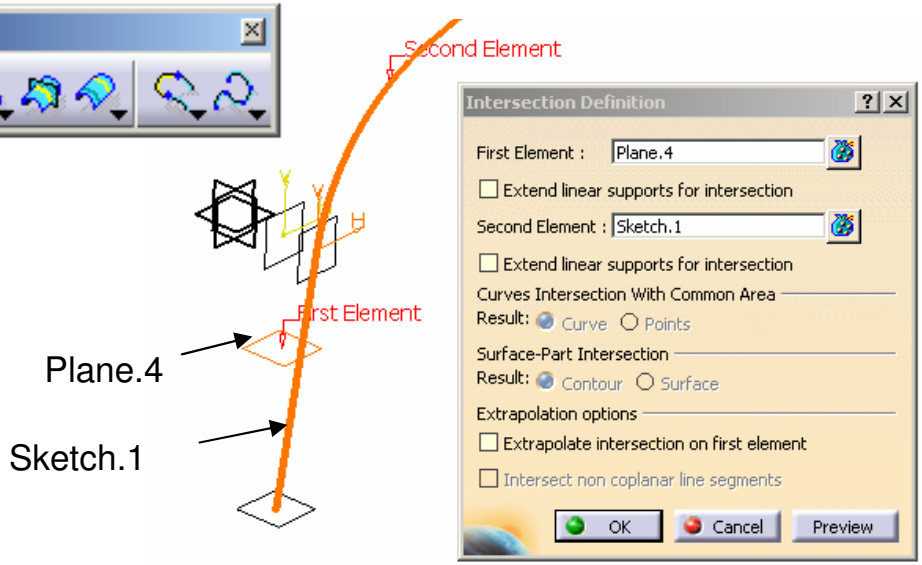
- Click “**plane**” icon
- Select “Offset from Plane” as plane type
- Select “plane.3” as Reference
- Enter 33mm as Offset
- Click ok to complete





To create an intersection point:-

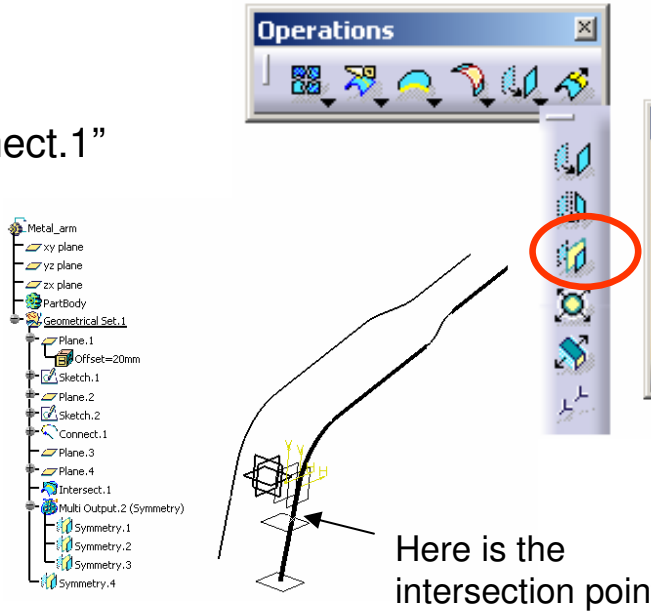
- Click “**Intersection**” icon
- Select “Plane.4” as First Element
- Select “Sketch.1” as Second Element
- Click ok to complete



Tutorial 3B

To duplicate elements by mirroring:-

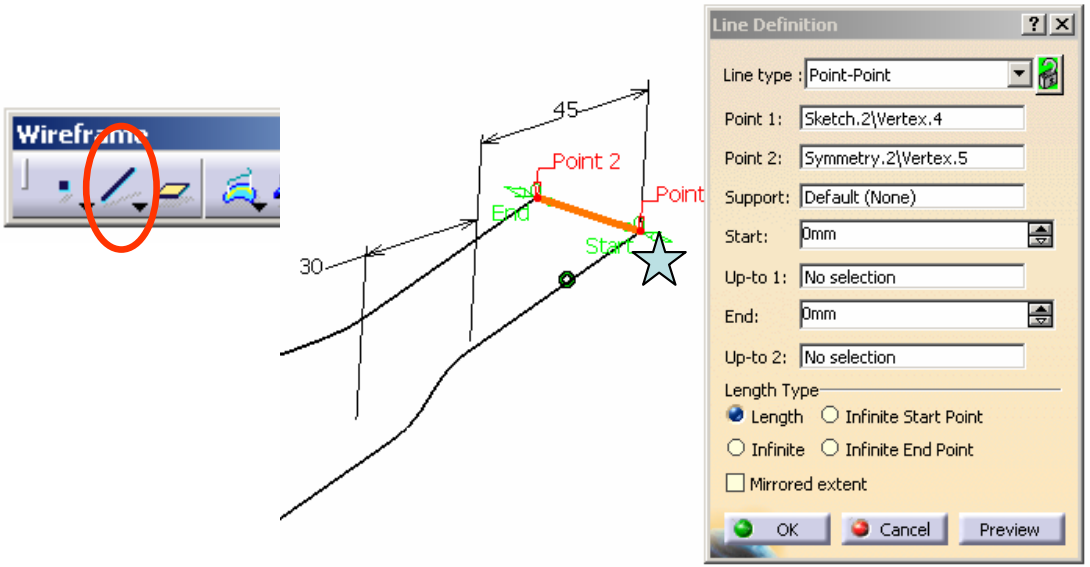
- Multi-select “Sketch.1” , “Sketch.2” & “Connect.1”
- Click “**Symmetry**” icon
- Select “zx plane” as Reference
- Click ok to complete



- 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:-

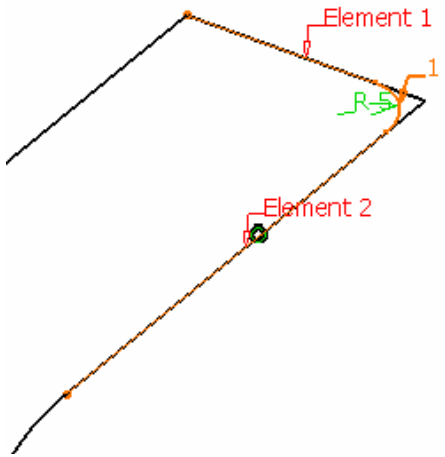
- Click “**Line**” icon
- Select the Endpoint of Sketch.2 ☆
- Select the Endpoint of its mirrored image
- Click ok to complete

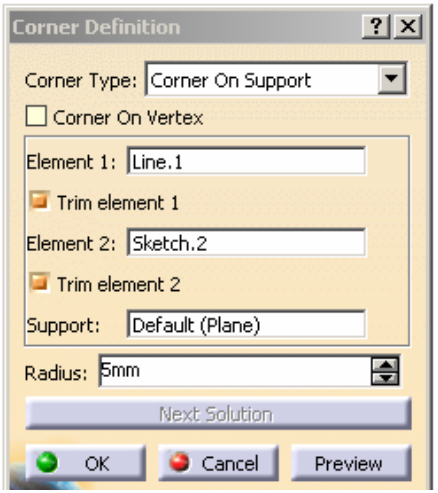


Tutorial 3B

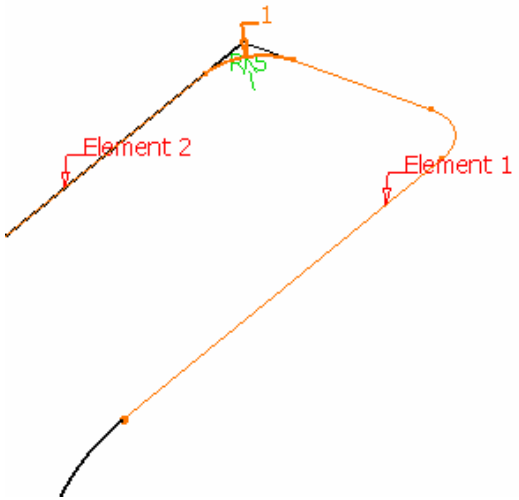
To create a round corner between two lines:-

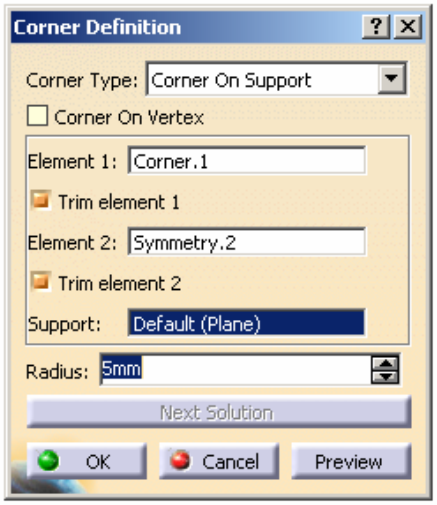
- Click “**Corner**” icon
- Select “Line.1” as Element 1
- Select “Trim element 1”
- Select “Sketch.2” as Element 2
- Select “Trim element 2”
- Enter 5mm as Radius
- Click ok to complete





- 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

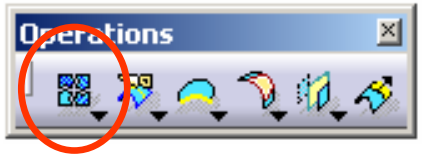


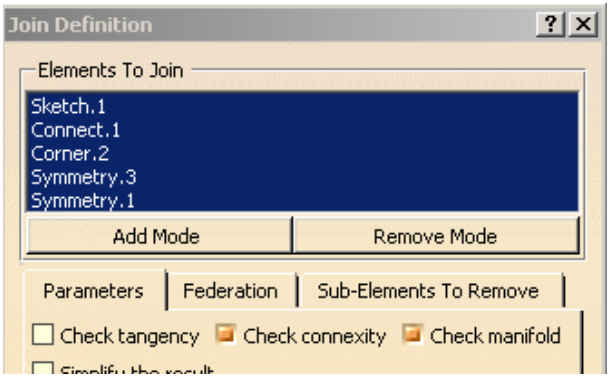


Tutorial 3B

To group all lines & curves:-

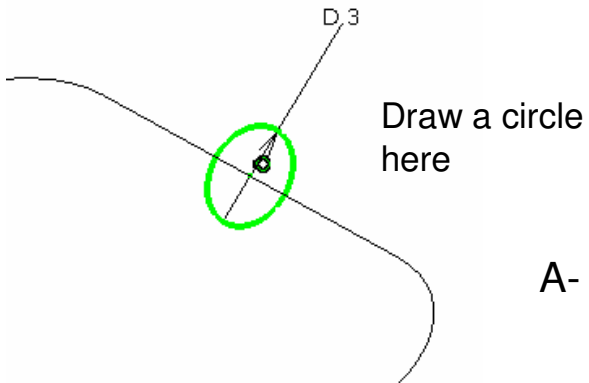
- Click "**Join**" icon
- Select all lines & curves in the screen, which includes: "Sketch.1", "Connect.1", "Corner.2", "Symmetry.3"& "Symmetry.1"
- Click ok to complete

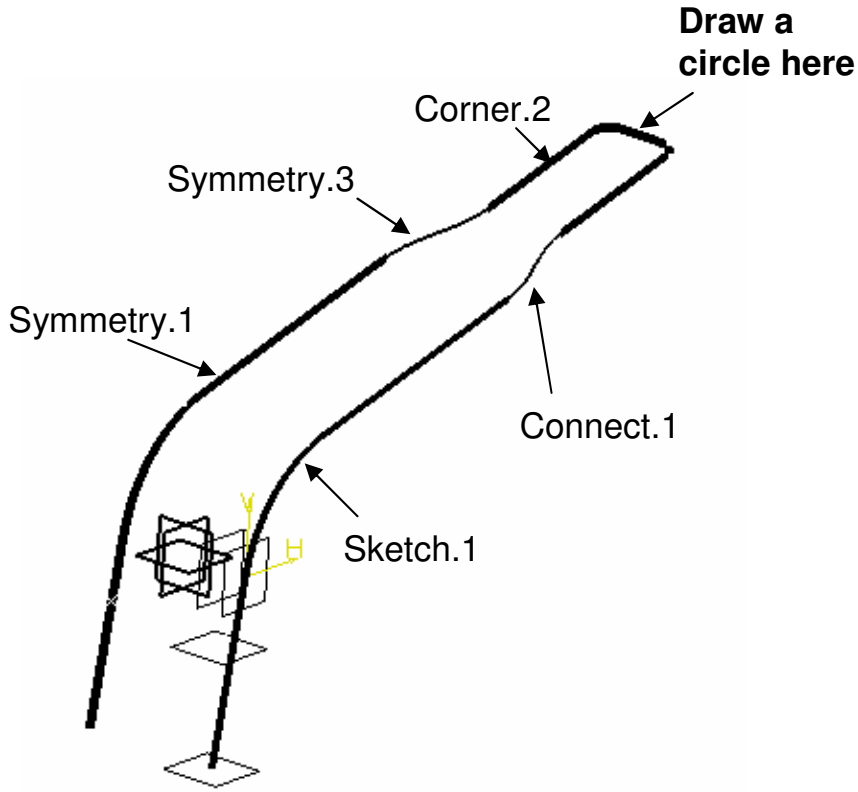




To make a sketch:-

- 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

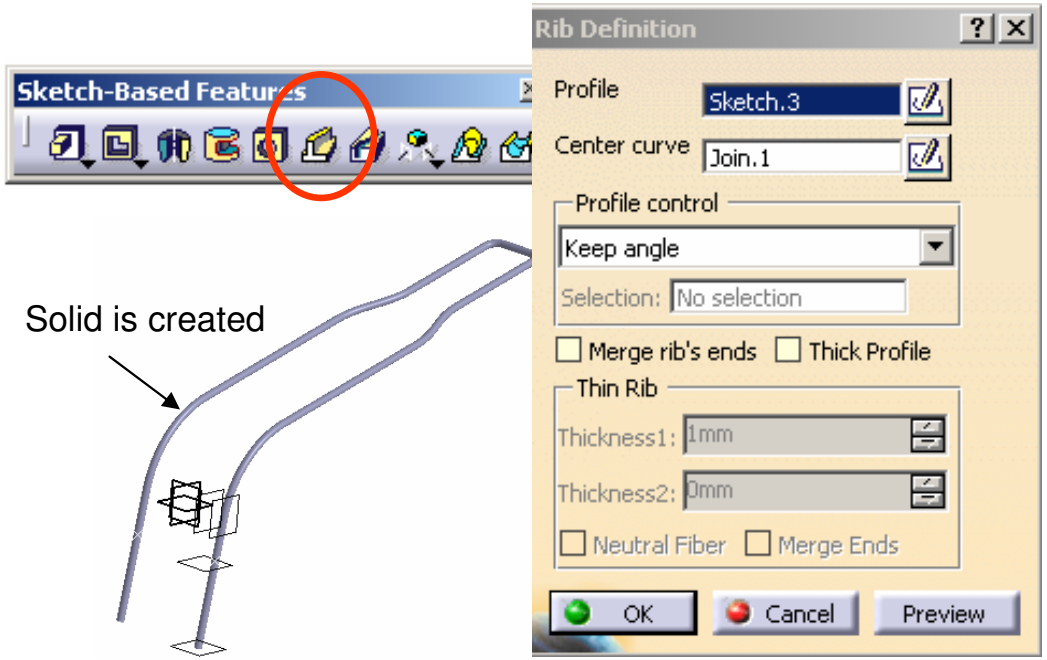




Tutorial 3B

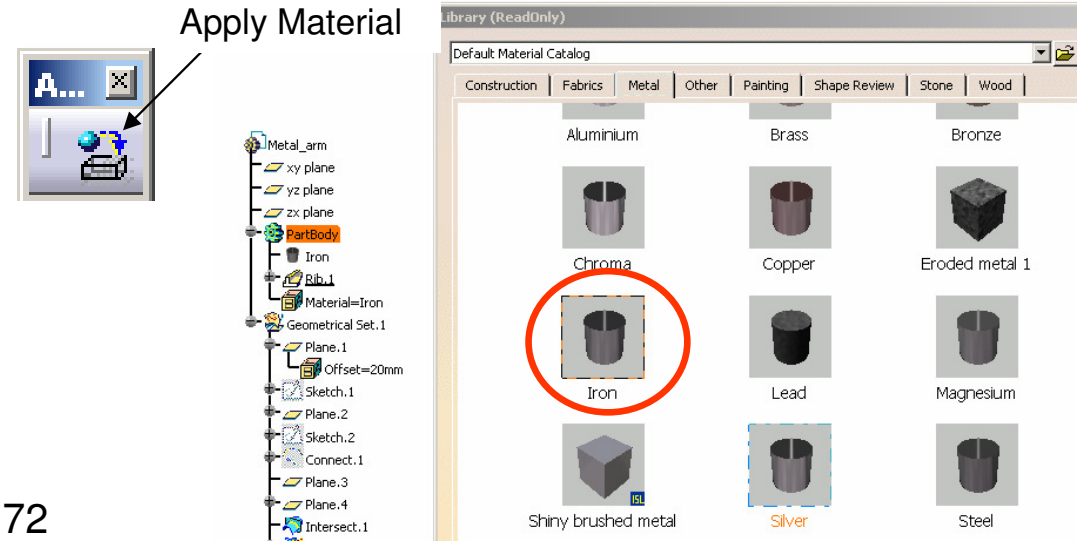
To make a solid:-

- 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:-

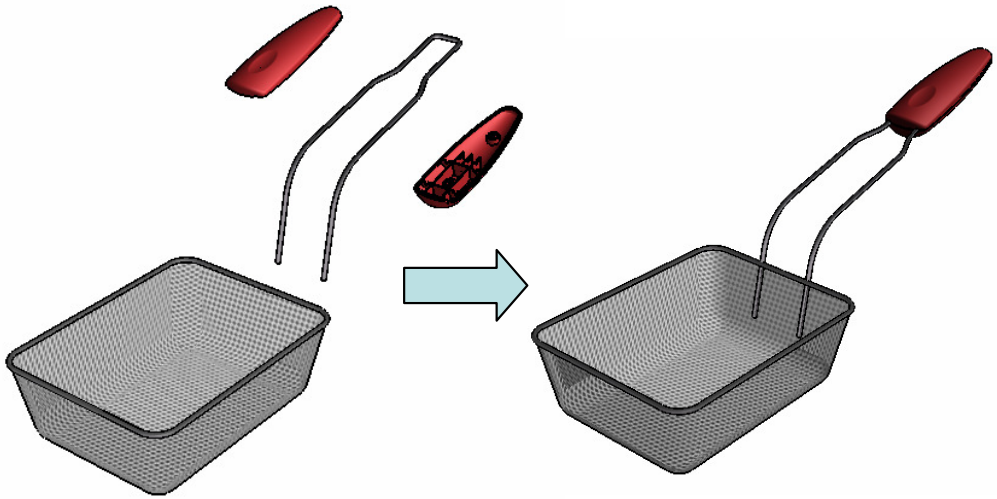
- Save your current part as "**Metal_arm_a.CATPART**" into the folder.

Tutorial 3B

Assemble components together…

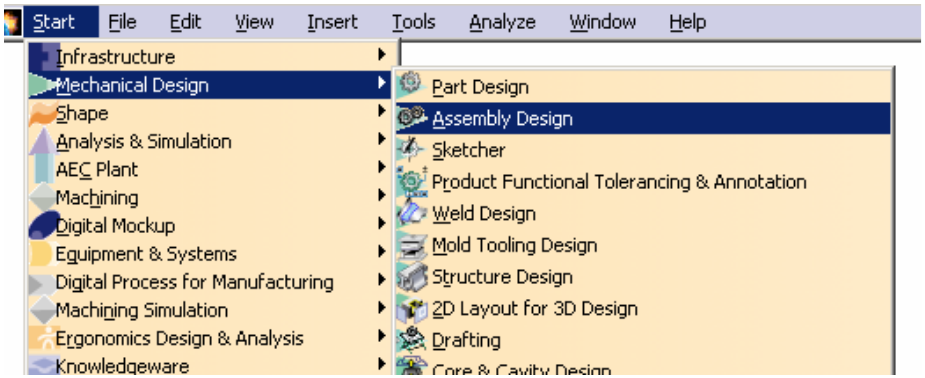
In the folder, you should have five part files;

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



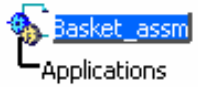
To go to a new Workbench:-

- 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:-

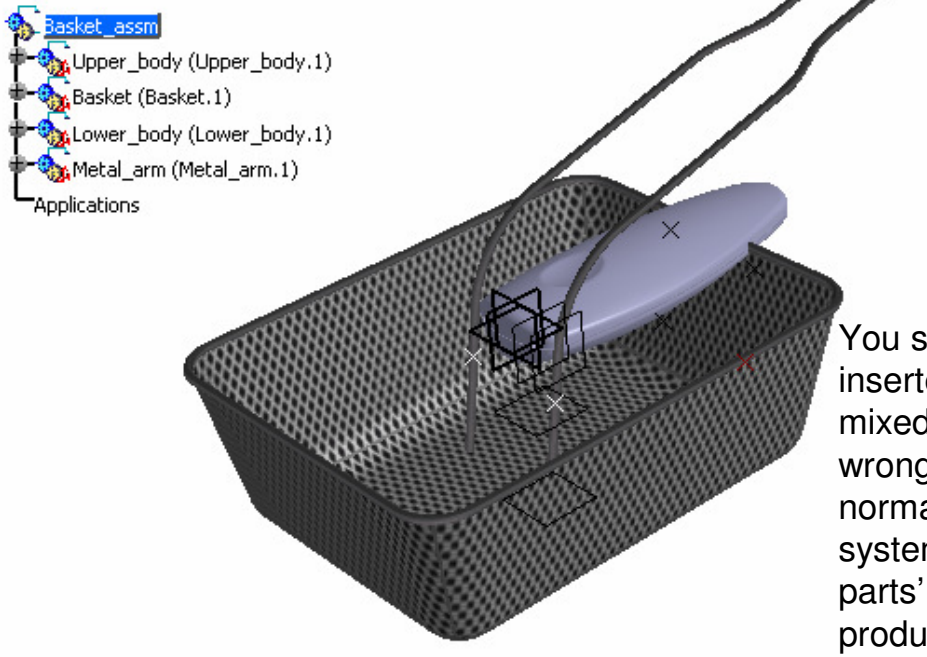
- Single-click "Product1" on the tree, right-click it, and then select "**Properties**".
- Modify Part Number as "Basket_assm" on the tab page "Product".
- Select "ok" to exit .

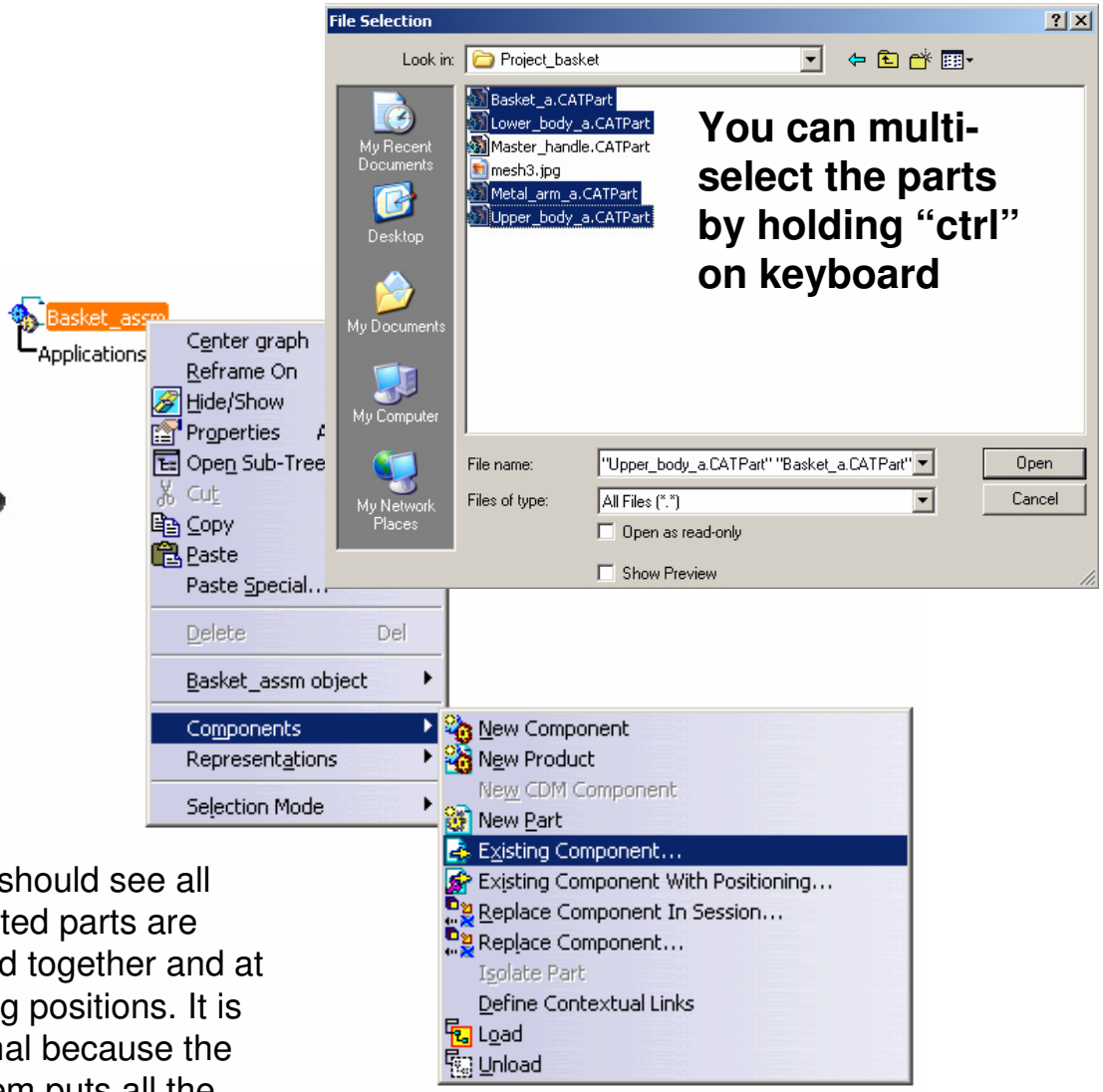


Tutorial 3B

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”



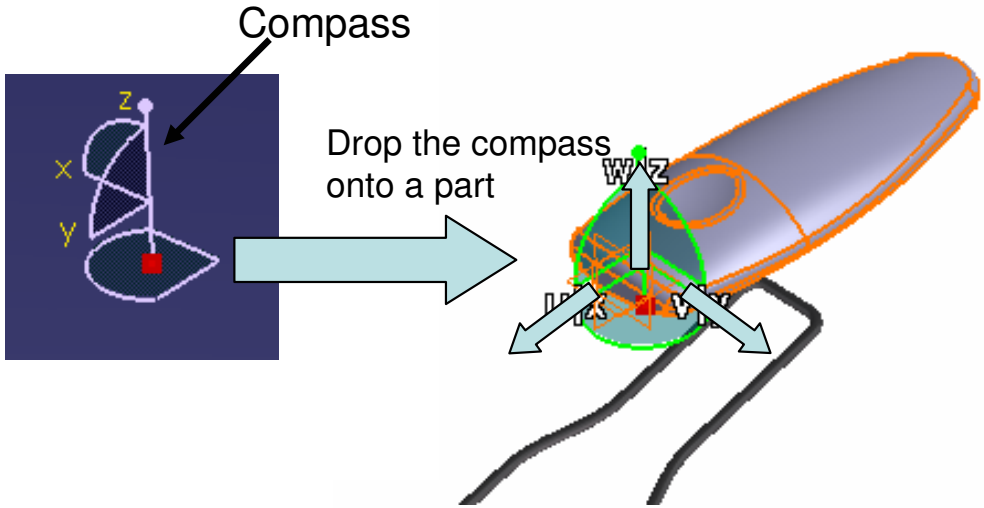


You should see all inserted parts are mixed together and at wrong positions. It is normal because the system puts all the parts’ origins onto the product’s origin.

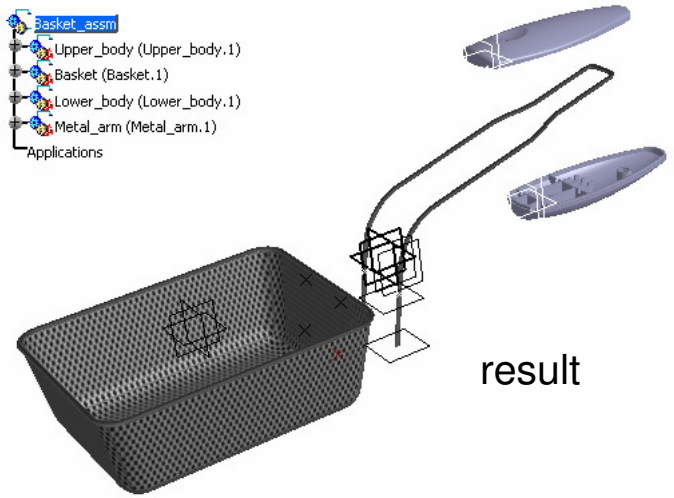
Tutorial 3B

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

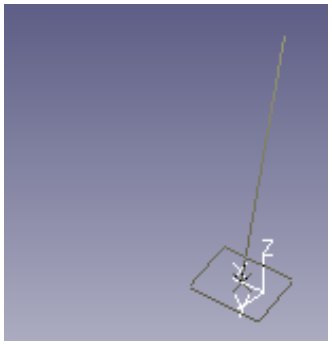


- Repeat the steps so that all parts are NEARLY at desired positions
- Now the parts are separated. It is easier for us to select part features later



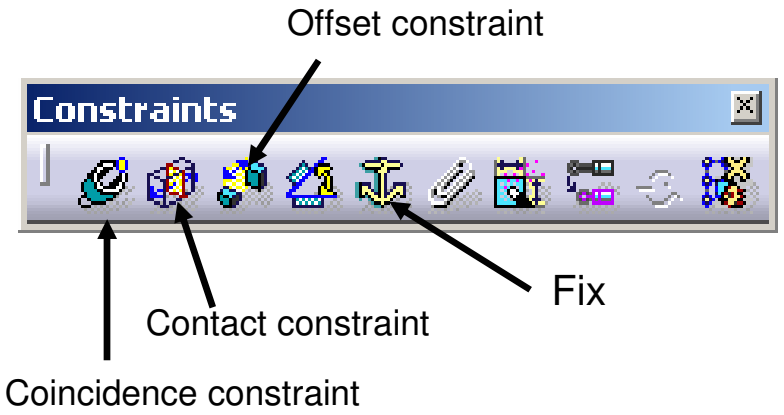
To reset "Compass" as original:-

- 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.



Tutorial 3B

To assemble parts by adding constraints:-

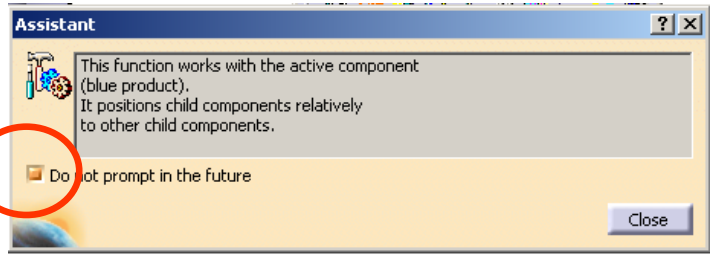


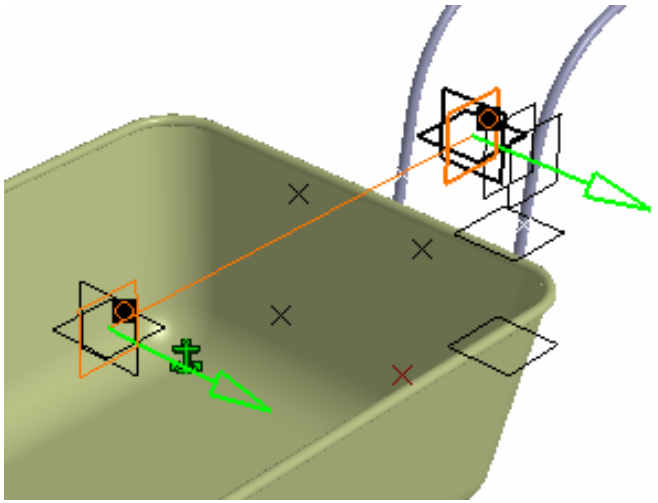
(1) Fix "Basket" in space

- 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

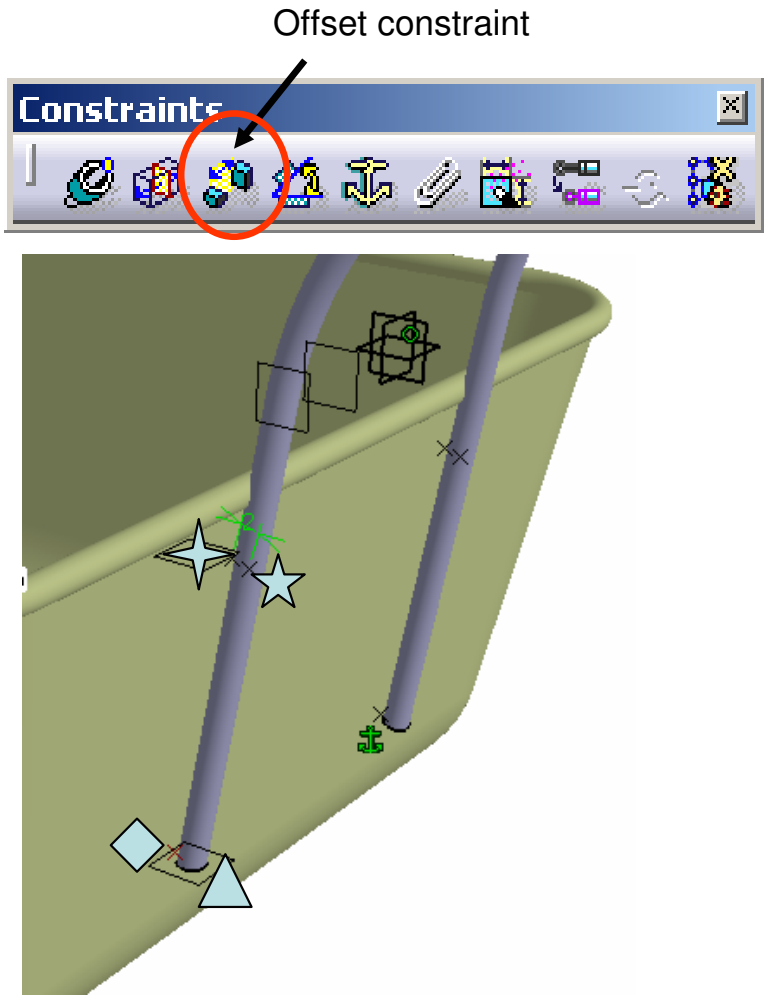
(Cont')

- 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 Basket

- 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 Basket again

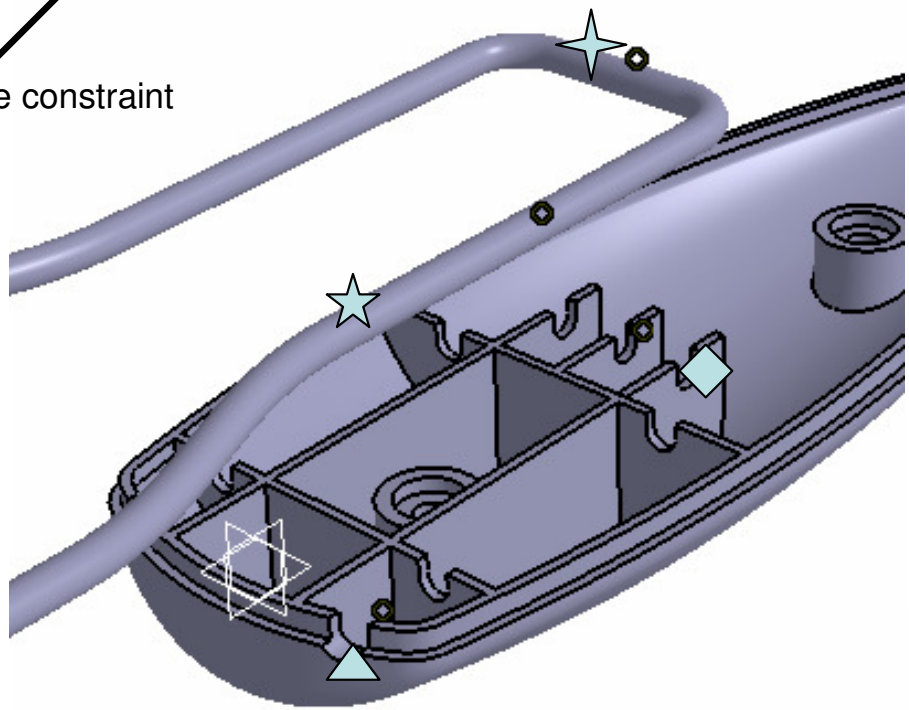


Tutorial 3B

(3) Link "Lower Body" to "Metal Arm"

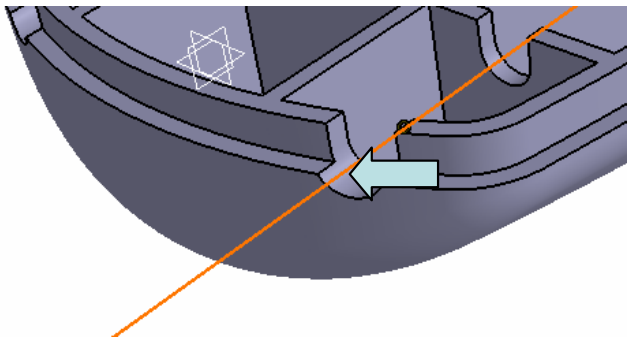
- Click "**Coincidence Constraint**" icon
- Select the axis of Metal Arm
- Select the axis of Lower Body
- Click ok to complete

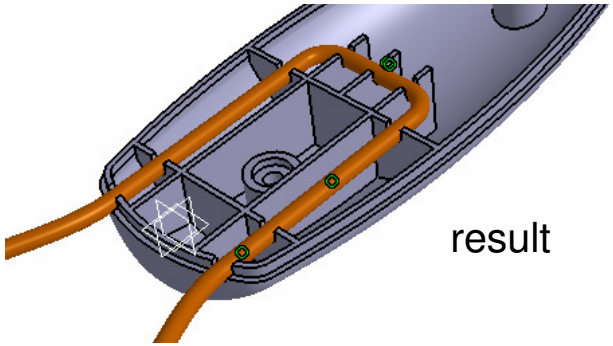
Coincidence constraint



- 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)**





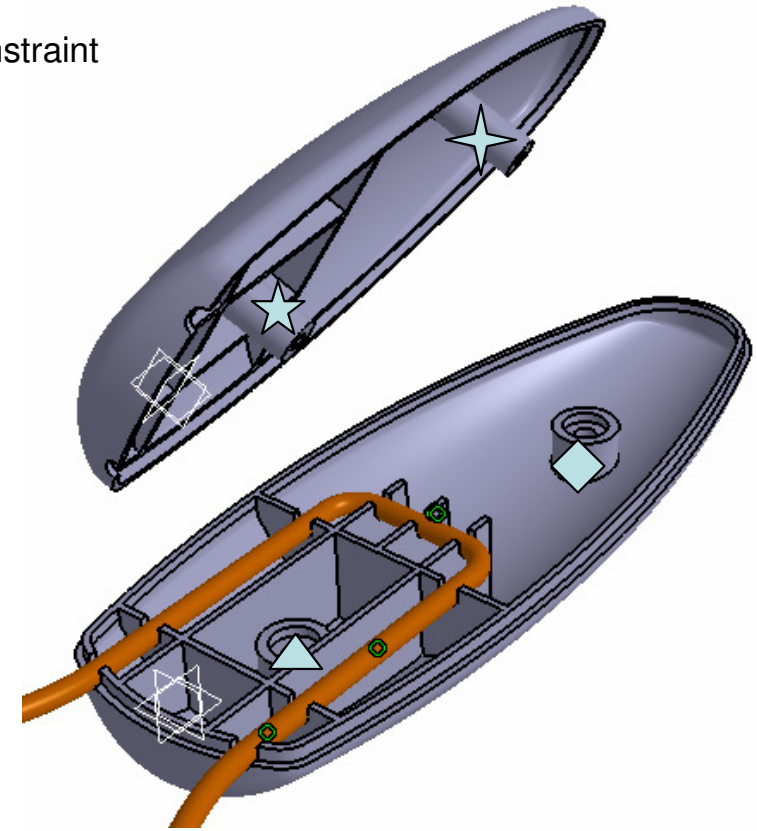
result

Tutorial 3B

(4) Link "Upper Body" to "Lower Body"

Coincidence constraint

- 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

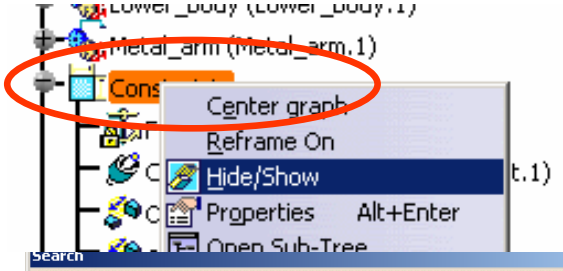
- Similarly, Click "**Coincidence Constraint**" icon again
- Select xy plane of Upper Body
- Select xy plane of Lower Body
- Click ok to complete

- Click "**Update**" Icon to update the position.

Tutorial 3B

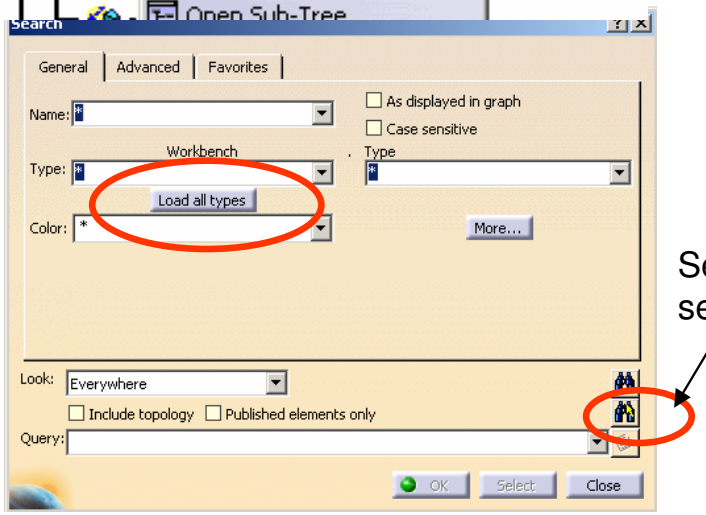
To hide all constraints:-

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



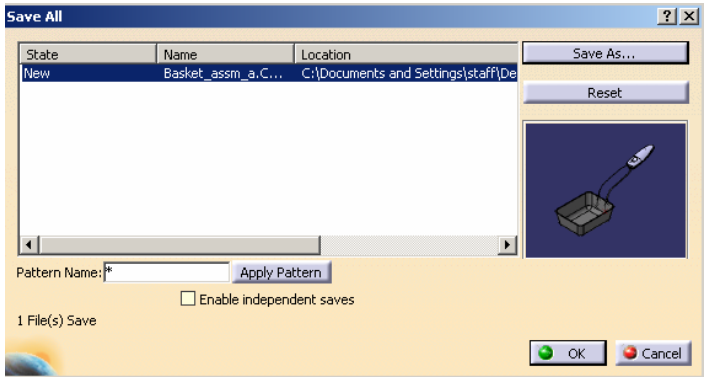
To hide all datum planes:-

- 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



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*****

END of Tutorial 3B

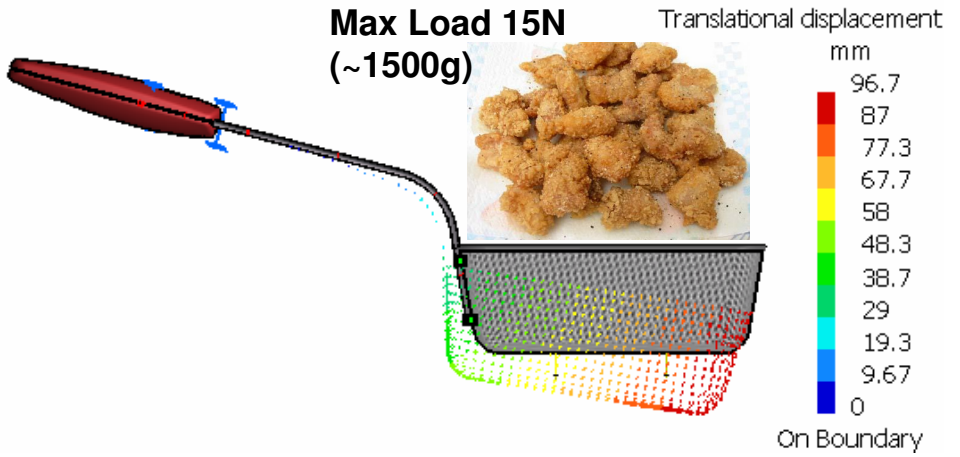
By Dickson Sham (ME Dept, HKPU)

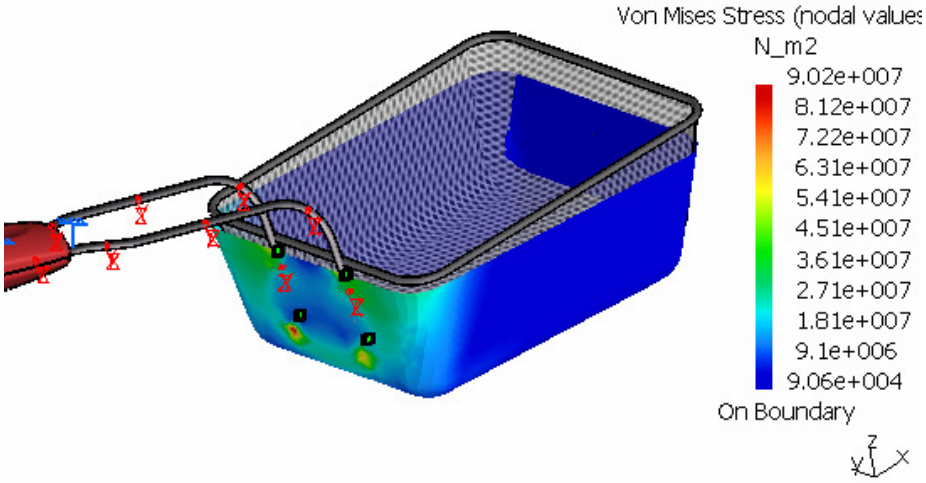
Tutorial 3C

We are going to estimate the deflection of the basket under the maximum load by Finite Element Analysis…

Assumptions:

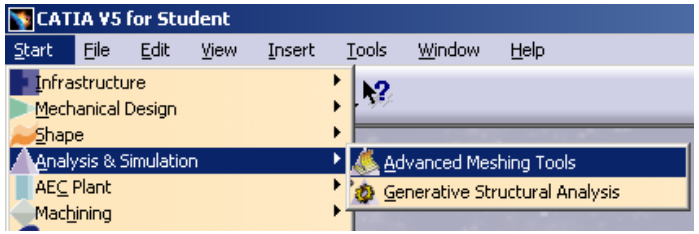
- Linear Behavior of the material
- Displacements will be small such that a linear solution is valid
- The spot weld joint between the basket and the metal arm will not break under the load
- Loading rate should be sufficiently low
- Load is uniformly distributed on the bottom faces of the basket
- The deformation of the basket is much lower than that of the metal arm

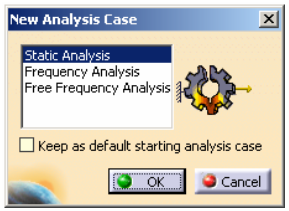




Tutorial 3C

File/Open/ Basket_assm_a.CATProduct



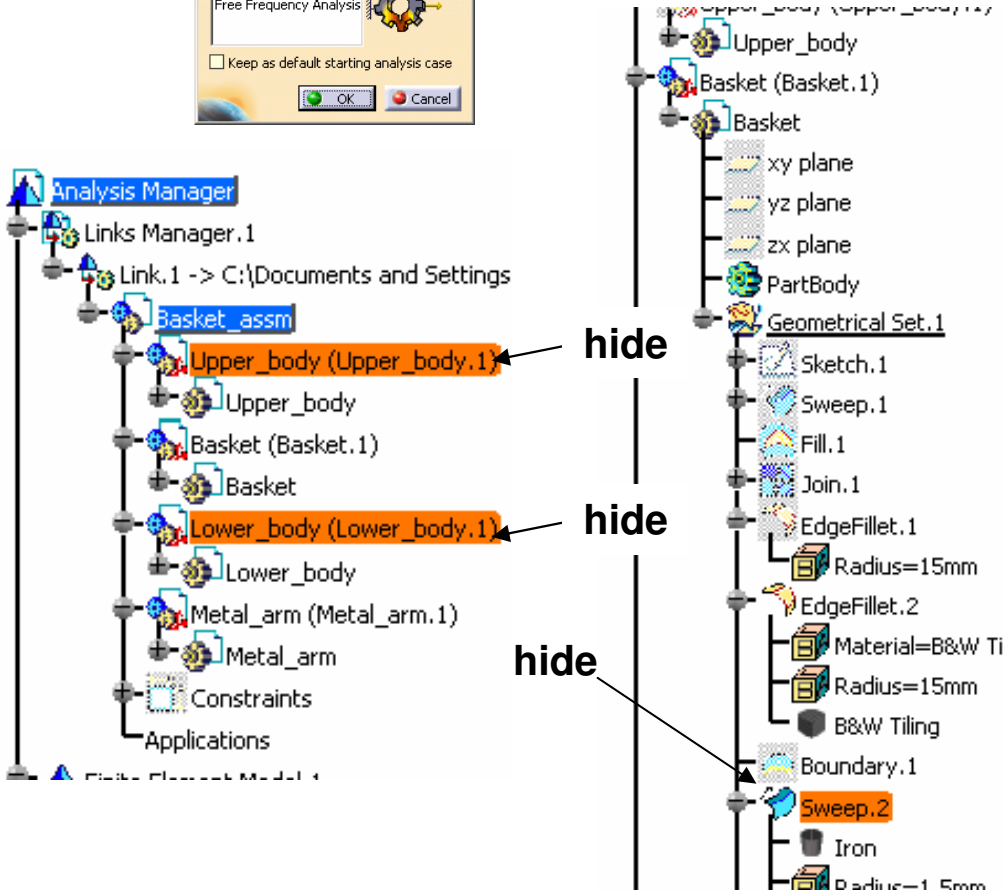


To go to a new Workbench:-

- Select '**Start/Analysis/Advanced Meshing Tools"** on the menu bar.
- Select **Static Analysis** and then click ok

(1) To simplify the model for analysis:-

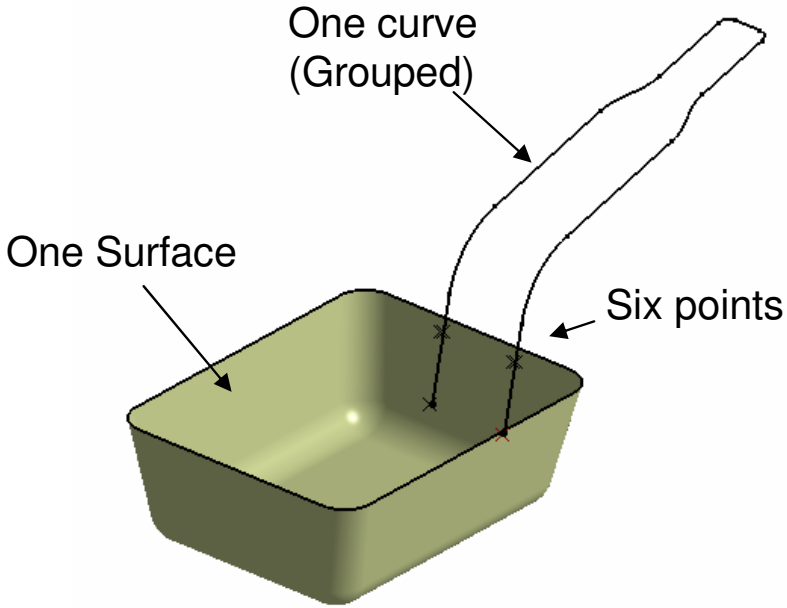
- Click "+" next to "Link Manager.1" on the tree
- Click "+" next to "Link.1" on the tree
- **Hide** *Upper_body.1* & *Lower_body.1*
- **Hide** the surface "Sweep.2" under *Basket.1*
- **Hide** "PartBody" under *Metal_arm.*1
- **Show** the curve "Join.1" under *Metal arm.1*



Tutorial 3C

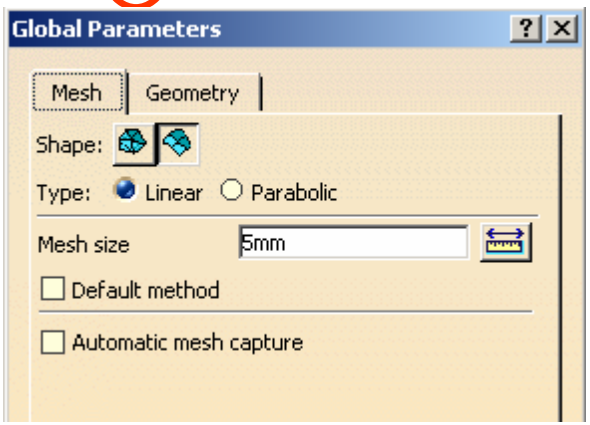
Now, we should see the below elements only:

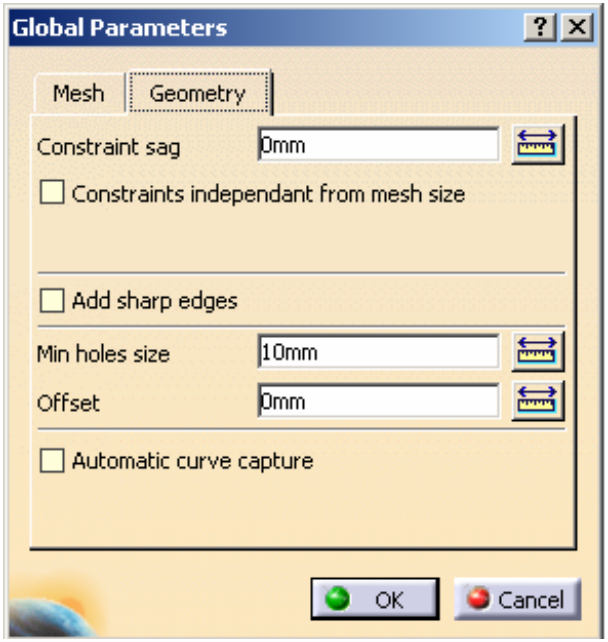
- One surface
- Six points (4 points on surface, 2 points on curve)
- One Curve (grouped)



(2a) To create a 2D mesh:-

- Click "**Surface Mesher**" icon
- Select the surface
- Select "quadrangle" as Shape
- Select "Linear" as Type
- Enter 5mm as Mesh Size
- Leave all the rest default options unchecked
- Click ok to complete

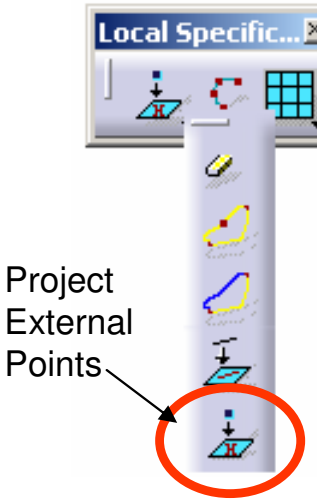


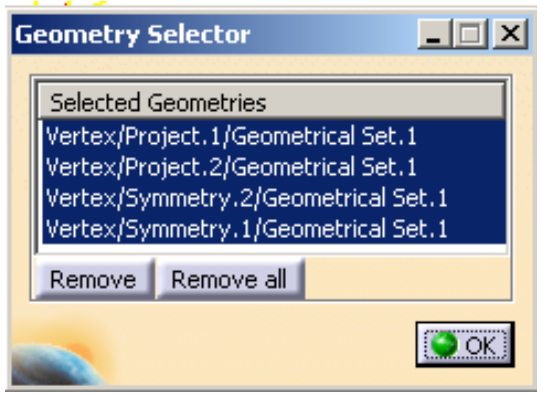


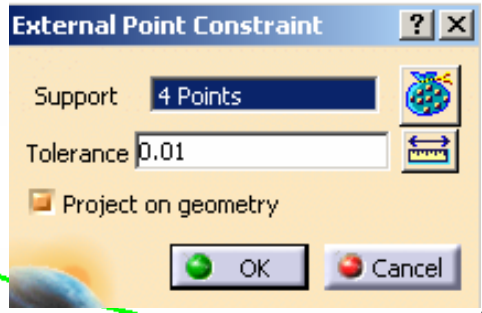
Tutorial 3C

(Cont'):-

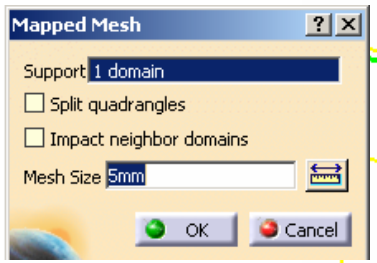
- Click "**Project External Points**" icon (so that nodes are created on the positions of spot welding)
- Select the four points on the surface
- If you select a wrong element accidentally, click and then remove it from the list
- Enter 0.01mm as Tolerance
- Select "Project on geometry"
- Click ok to complete (Now 4 red dots appears on the surface)

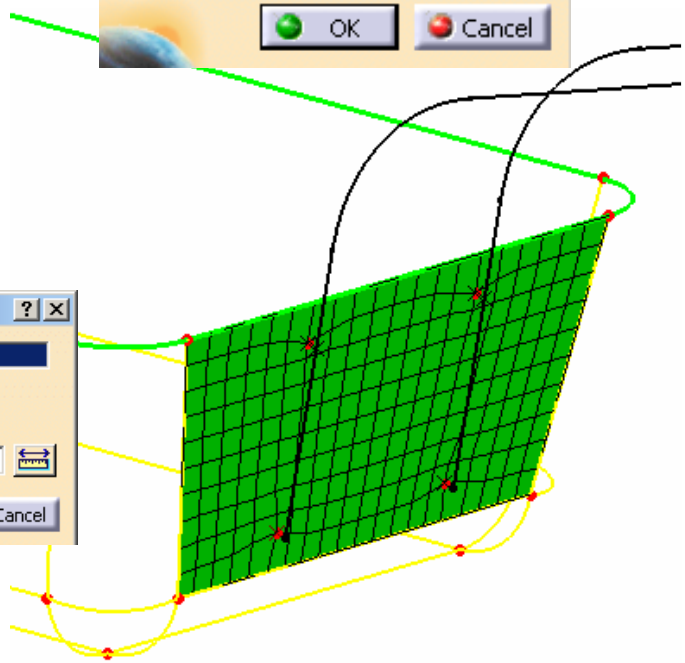






- Click "**Mapped Method**" icon
- Select the surface on which 4 red dots have been just created
- Enter 5mm as Mesh Size
- Click ok to complete (Meshing this surface before other surfaces will result in an uniform mesh distribution on this surface)

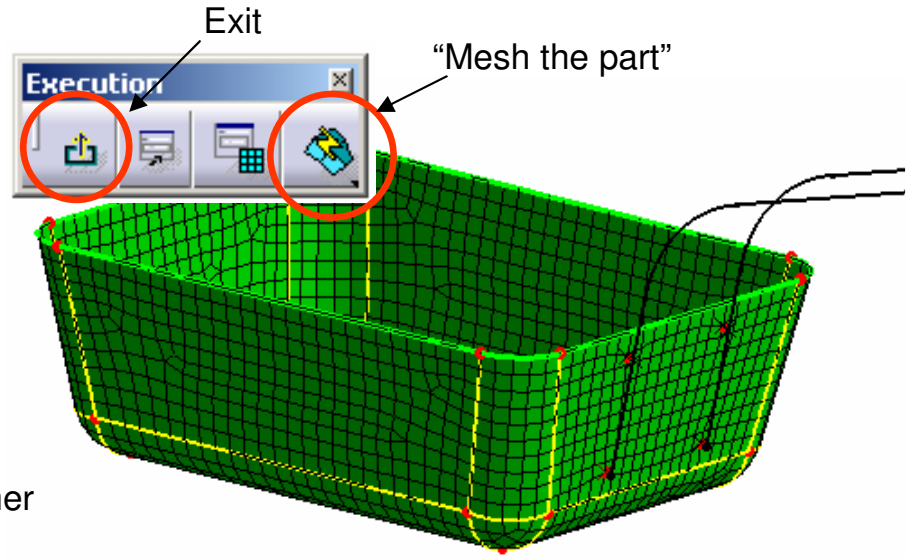




Tutorial 3C

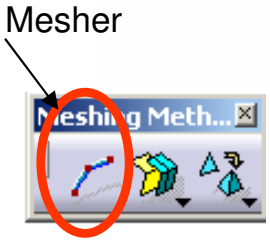
(Cont'):-

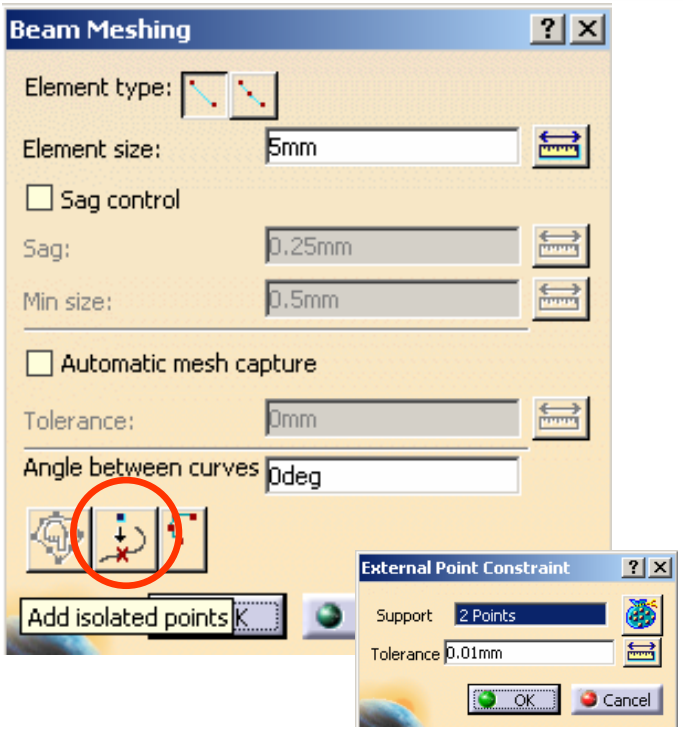
- Click "Mesh the part" icon (The system will mesh the remaining surfaces according to the specification you defined, i.e. linear quadrangle mesh with 5mm size)
- Click "Exit"



(2b) To create a 1D mesh:-

- Select '**Start/Analysis/Advanced Meshing Tools**" on the menu bar again
- Click "**Beam Mesher**" icon
- Select the curve
- Click "Add Isolated points" on the command window
- Select the two points on the curve (so that nodes are created on the positions of spot welding)
- Enter 0.01mm as Tolerance
- Click ok to complete



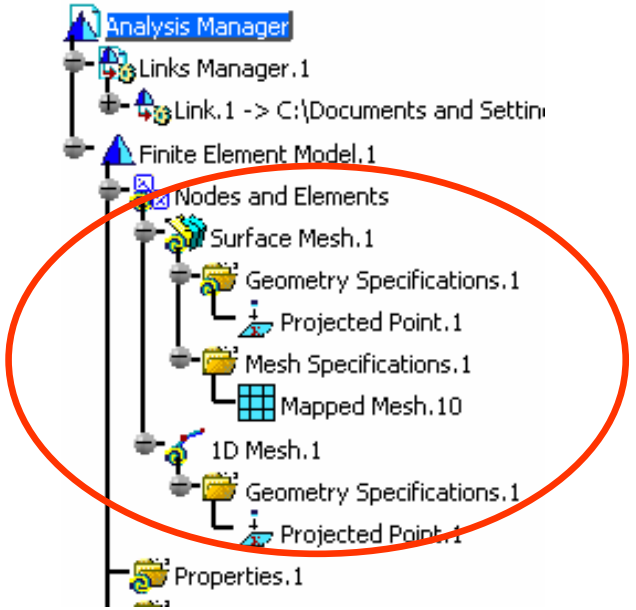


Tutorial 3C

Now, we have already created two meshes:

Surface Mesh.1 (for the basket)

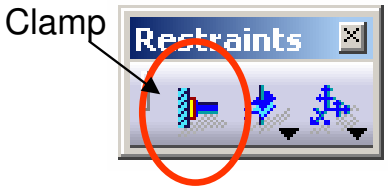
1D Mesh.1 (for the metal arm)

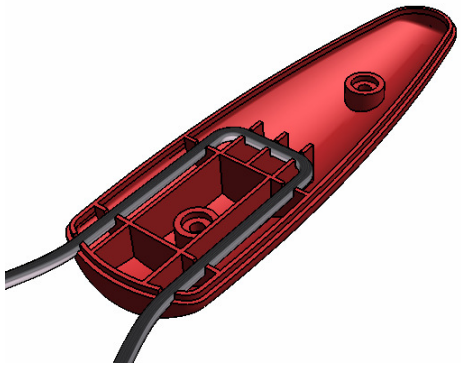


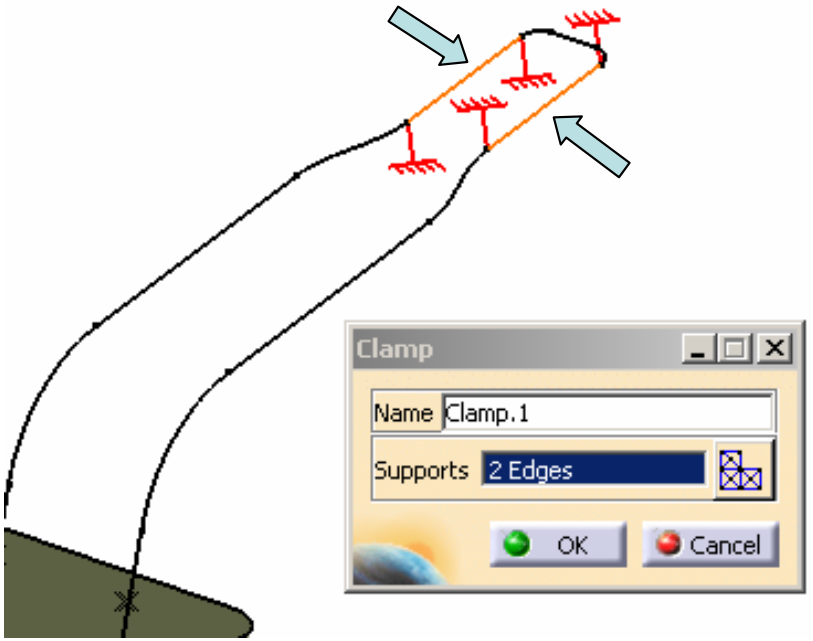
The workbench has been automatically switched to "Generative Structural Analysis"

(3) To Create a constraint:-

- Click "**Clamp**" icon
- Select the two straight lines as shown
- Click ok to complete



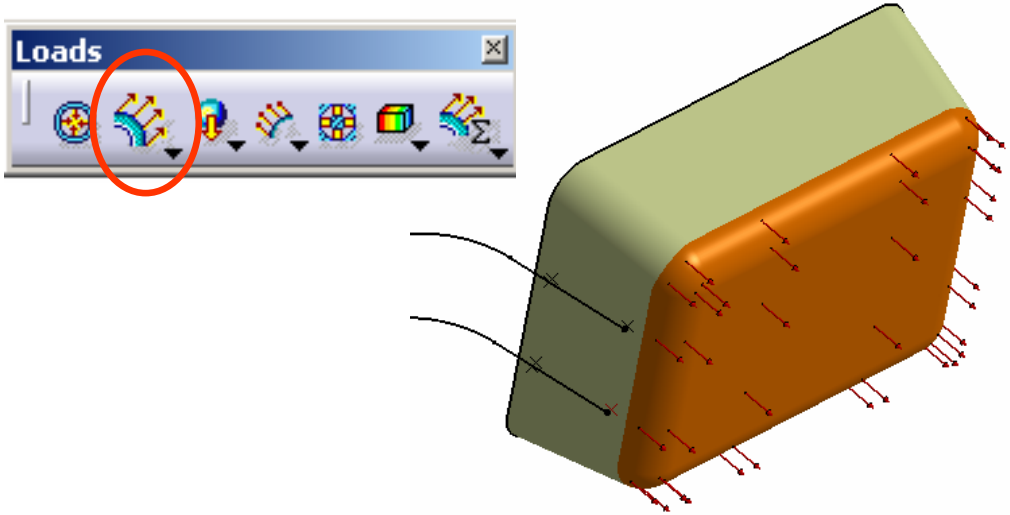




Tutorial 3C

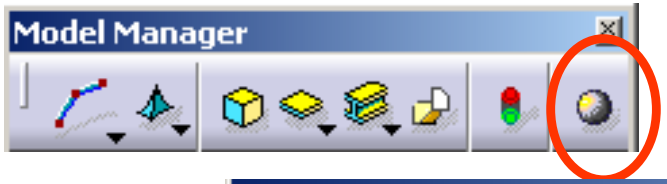
(4) To Create a Force:-

- Click “Distributed Force” icon
- Select the bottom faces (9 faces) of the surface as shown
- Enter -15N as Z Force
- Click ok to complete

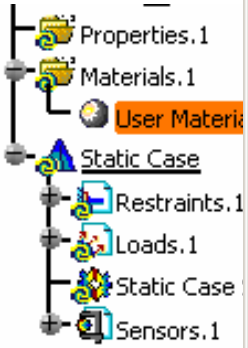


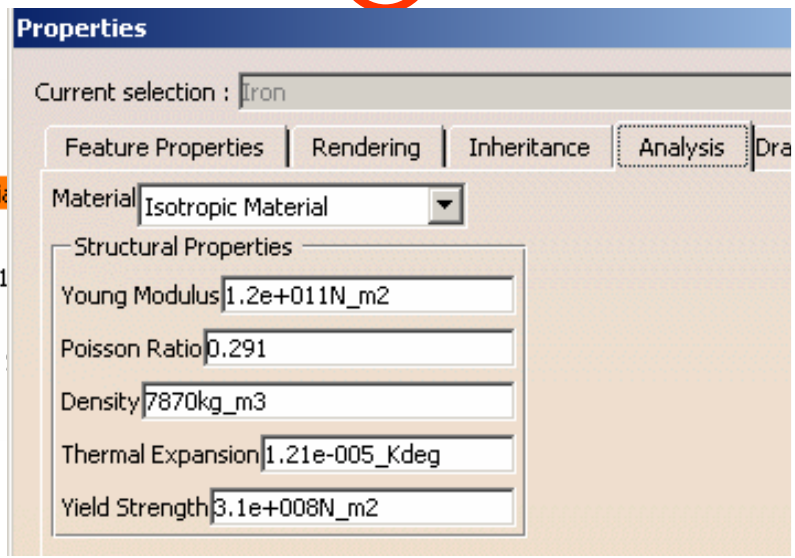
(5) To Create a User Material:-

- Click “User Material” icon
- Select “Iron” from the catalog
- Click ok to complete (it will be created on the tree)



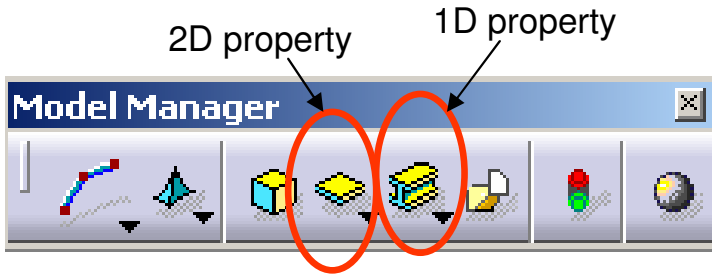
- Double-click it on the tree to view its properties
- Leave everything unchanged
- Click ok to quit



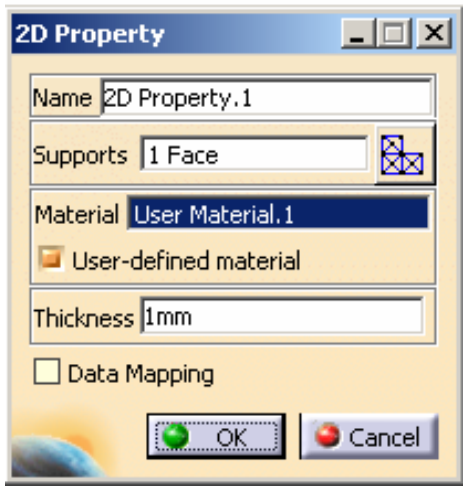


Tutorial 3C

(6a) To define Properties of Surface Mesh.1:-

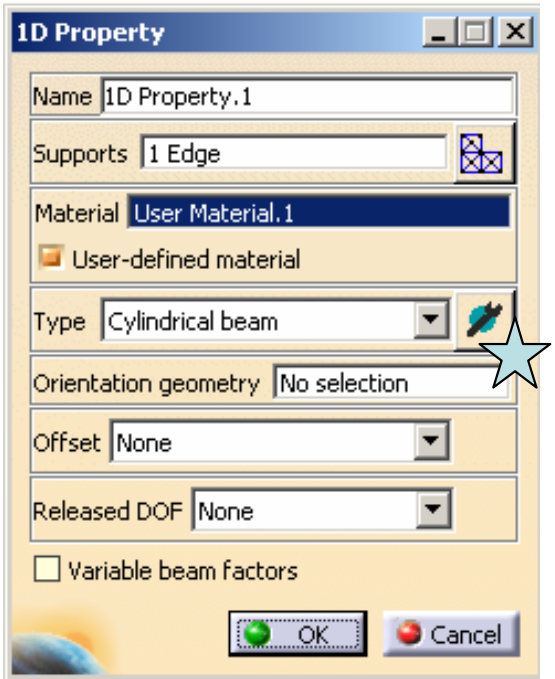


- Click "**2D Property**" icon
- Select the surface
- Select "User-defined material" option (because the original material is used for rendering only, not the actual material of the basket)
- Click the entry box "No selection" once
- Select "User Material.1" on the tree
- Enter 1mm as Thickness
- Click ok to complete



(6b) To define Properties of 1D Mesh.1:-

- Click "**1D Property**" icon
- Select the curve
- Select "User-defined material" option
- Click the entry box "No selection" once
- Select "User Material.1" on the tree
- Select "Cylindrical Beam" as Type
- Click the icon ☆
- Enter 1.5mm as radius
- Click ok to complete



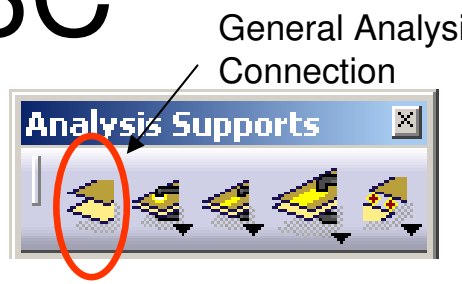
Tutorial 3C

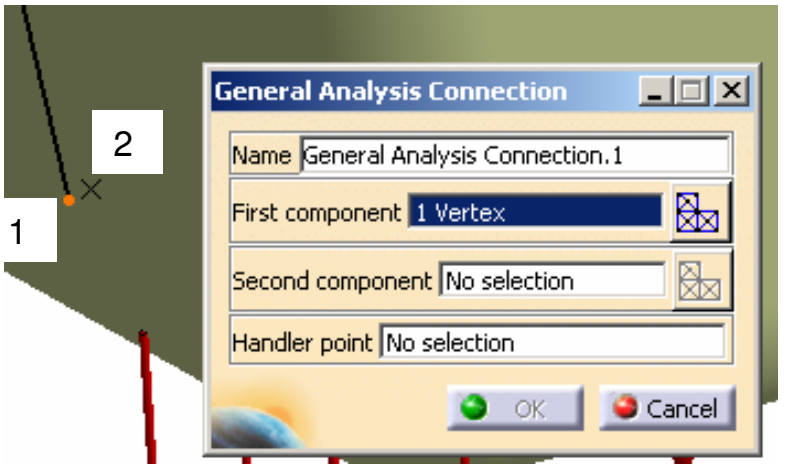
(7) To build Connections between meshes:-

- (We have four pairs of points and we are going to build a connection for EVERY PAIR independently)

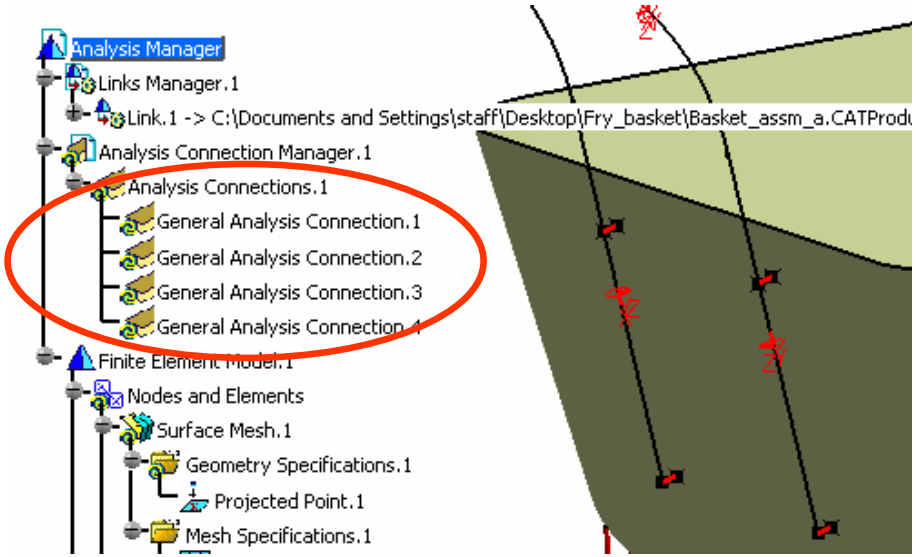
- Click "**General Analysis Connection**" icon
- Select *point 1* as First Element
- Click the box "No selection" of Second Element
- Select *point 2* as Second Element
- Click ok to complete

- Repeat the above steps for the remaining three pairs





Four connections are made

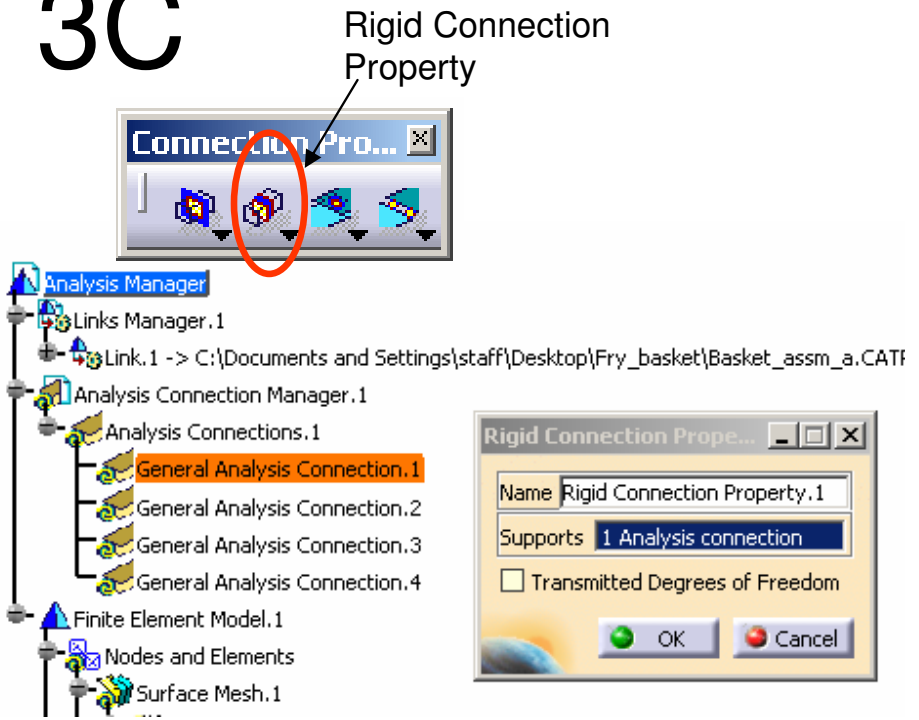


Tutorial 3C

(8) To define Properties of connections:-

- Click "**Rigid Connection Property**" icon
- Select 'General Analysis Connection.1" on tree
- Click ok to complete

- Repeat the above steps for
  'General Analysis Connection.2"
  'General Analysis Connection.3"
  'General Analysis Connection.4"



***** 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 "Analysis_a.CATProduct" as filename and save it in your project folder.

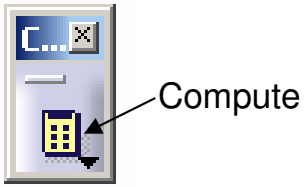
What we have already done…

1. Simplify the model (Hide unnecessary parts & features)
2. Create Meshes (2D & 1D)
3. Create a constraint (Clamp)
4. Create a force (distributed force)
5. Create a User material
6. Define Properties of Meshes
7. Create connectors
8. Define Properties of Connectors

Tutorial 3C

(9) To start Computation:-

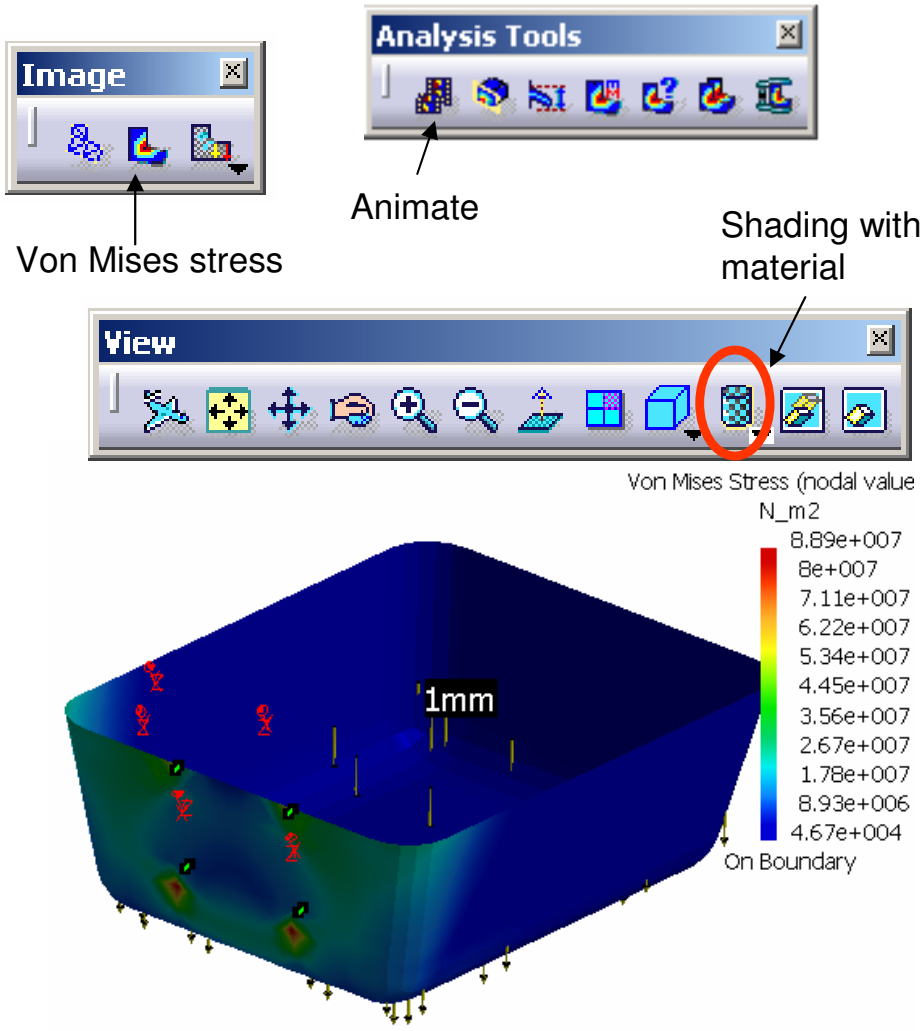
- Click "**Compute**" icon
- Select "All"
- Click ok



(10) To View the result:-

- Click "**Von Mises stress**" icon
- Click "**Shading with material**"
- (We can see the stress distribution on the basket. The stress value is not the same as the real case because we simplify the metal net as a metal sheet)

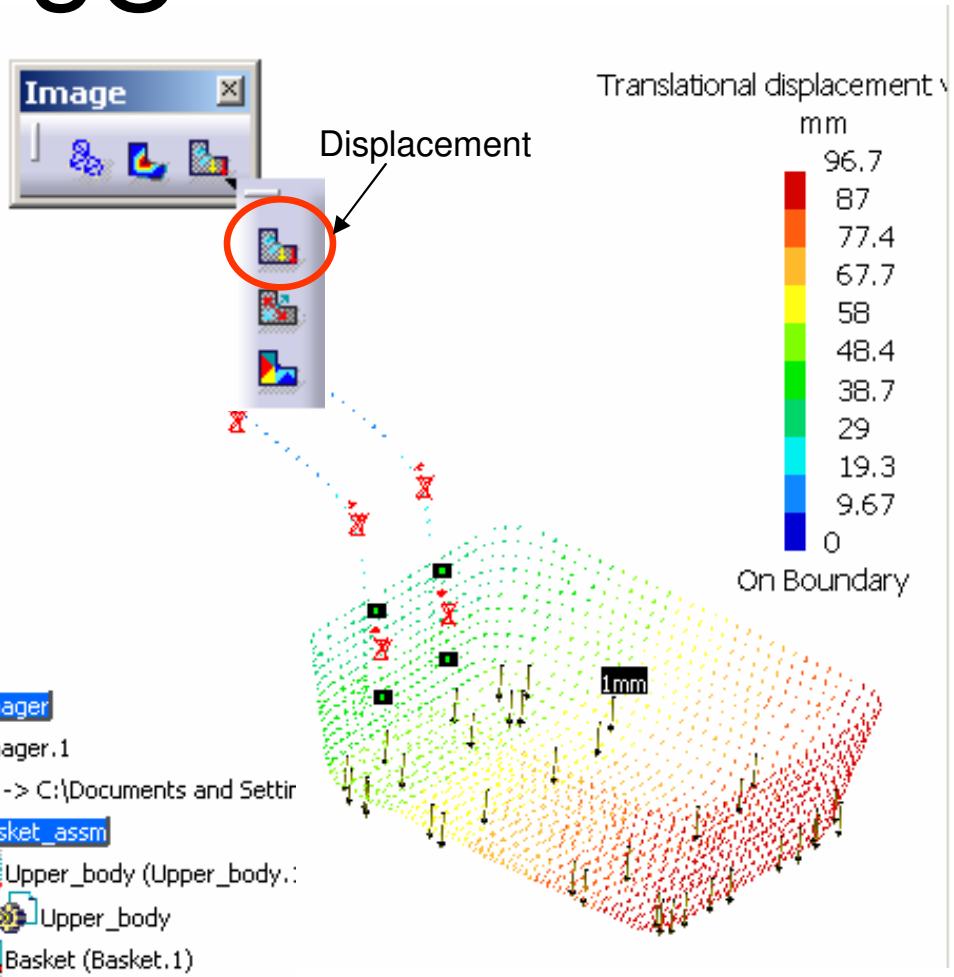
- Click "**Animate**" icon to see the variations in stress with different degrees of displacement



Tutorial 3C

(Cont'):-

- Click "**Displacement**" icon
- (We can see the maximum displacement of the basket is about 96.7mm!)

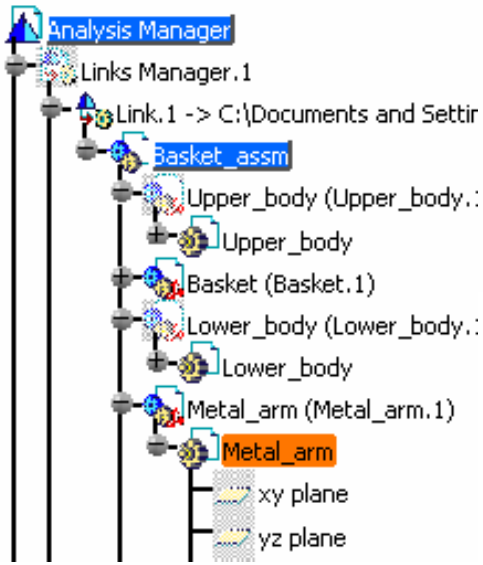


The Displacement is too big to accept.

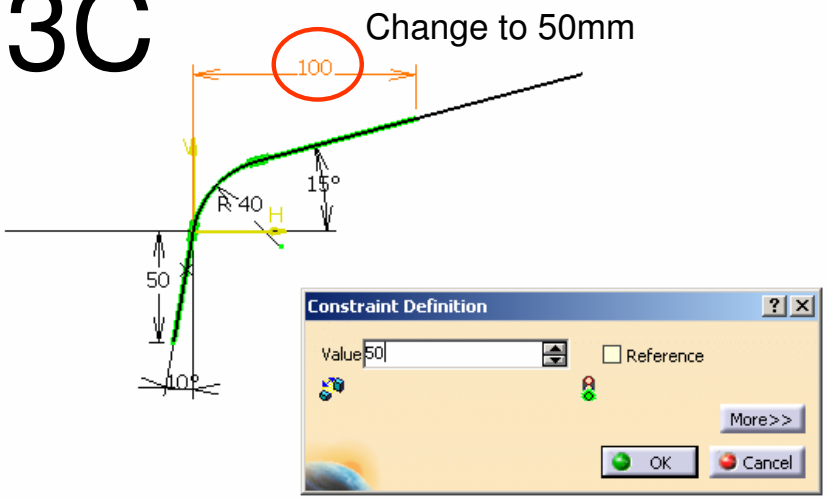
Now we are going to shorten the length of the metal arm and make it thicker…

To edit the part "Metal Arm":-

- Right-click "Metal arm" on tree
- Select Metal_arm.object/Open in new window



Tutorial 3C

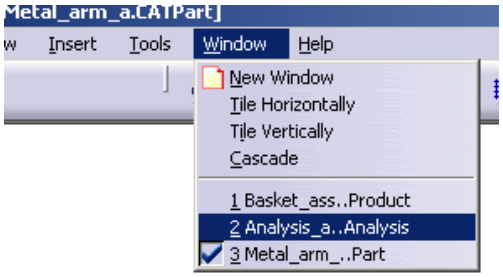


(Con't):-

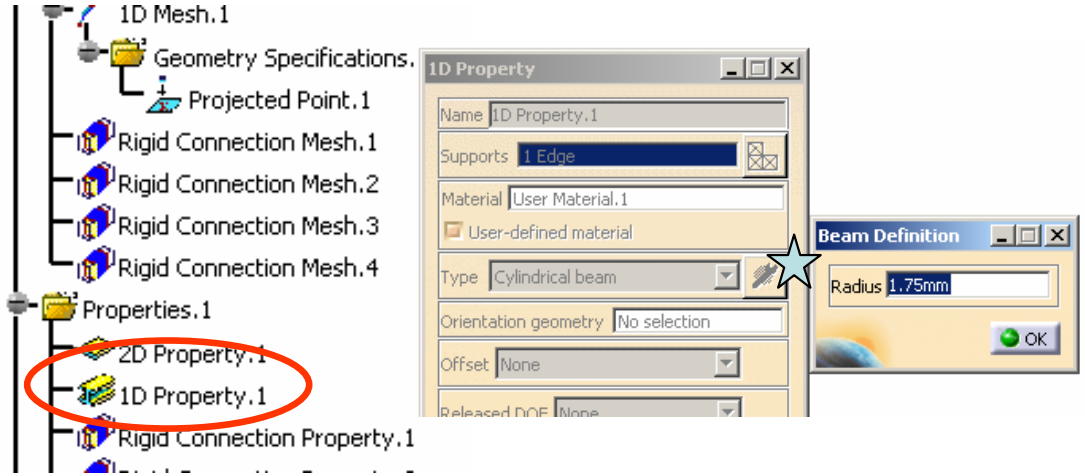
- Double-click "Sketch.1" on tree
- Modify the dimension 100mm to <u>50mm</u>
- Exit the workbench by clicking "**Exit**" icon
- (The metal arm is automatically updated)

To increase the diameter of Metal Arm (1D- Mesh):-

- Select "Window/Analysis1" on the menu bar to go back to the analysis workbench (The metal arm in the assembly is also updated)



- Double-Click "1D Property.1" icon on tree
- Click the icon ☆
- Change Radius from 1.5 to <u>1.75</u>
- Click ok to complete



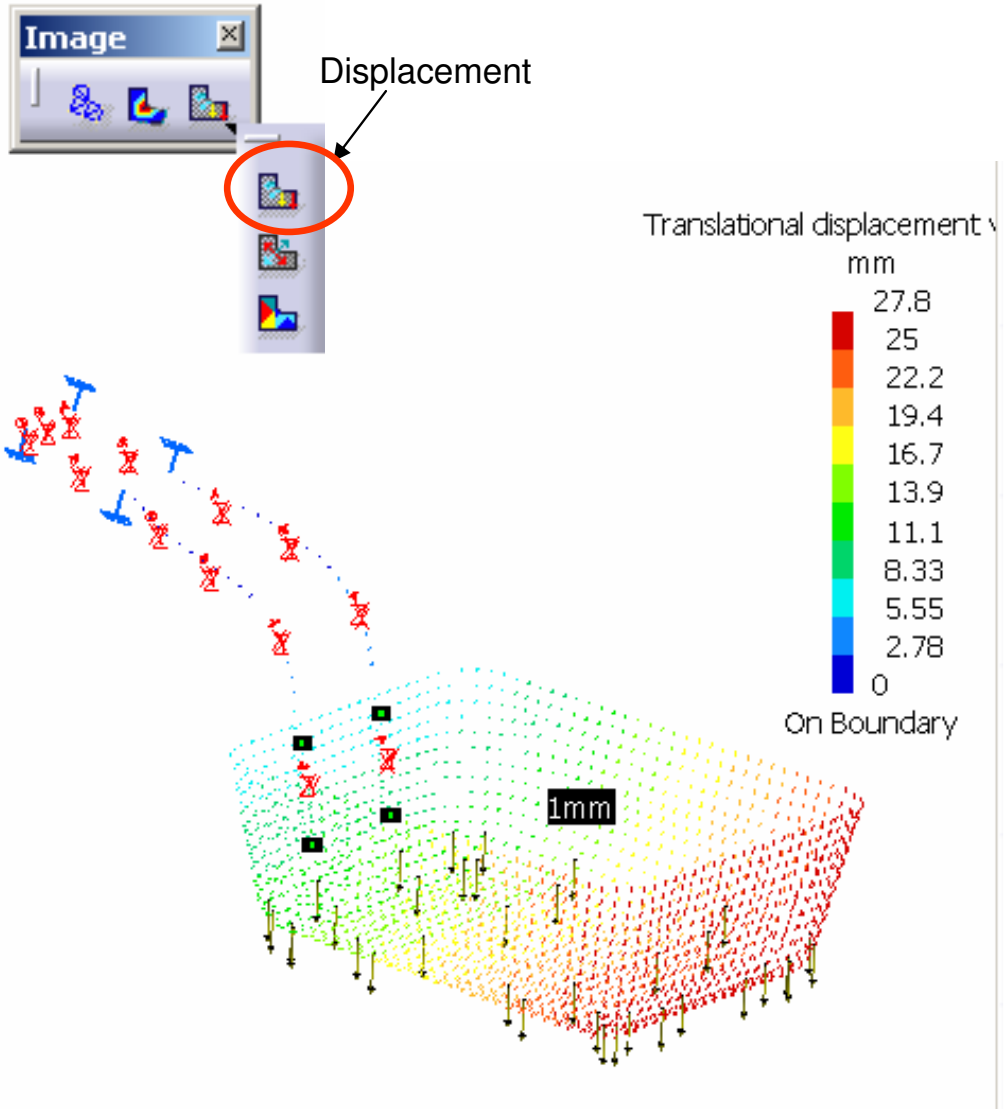
Tutorial 3C

To Compute Analysis again:-

- Click "**Compute**" icon
- Click ok

- Click "**Displacement**" icon to view the update displacement

- (The maximum displacement is now decreased to 27.8mm after the modification of the metal arm)



***** CLOSE ALL FILES WITHOUT SAVING*****

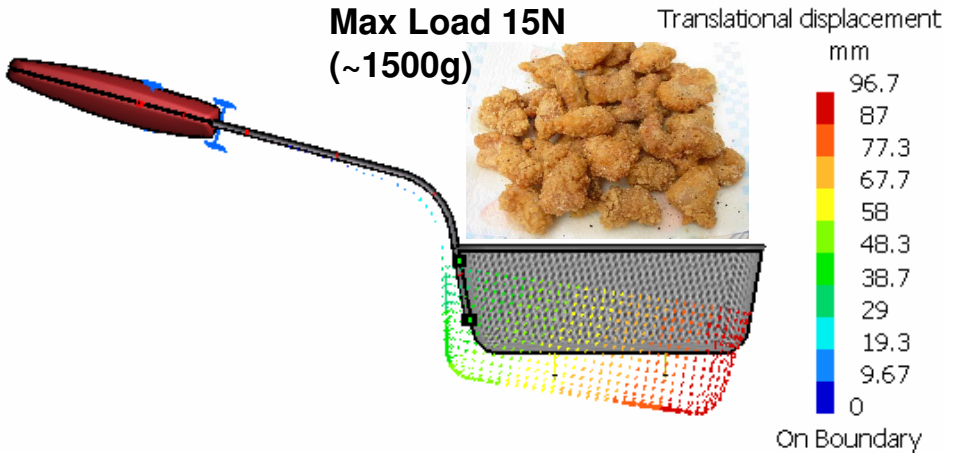
END of Tutorial 3C

Tutorial 3D

We are going to estimate the deflection of the basket again by another FEA tools, MSC PATRAN…

Assumptions: (Same as Tutorial 3C)

- Linear Behavior of the material
- Displacements will be small such that a linear solution Is valid
- The spot weld joint between the basket and the metal arm will not break under the load
- Loading rate should be sufficiently low
- Load is uniformly distributed on the bottom faces of the basket
- The deformation of the basket is much lower than that of the metal arm



Tutorial 3D

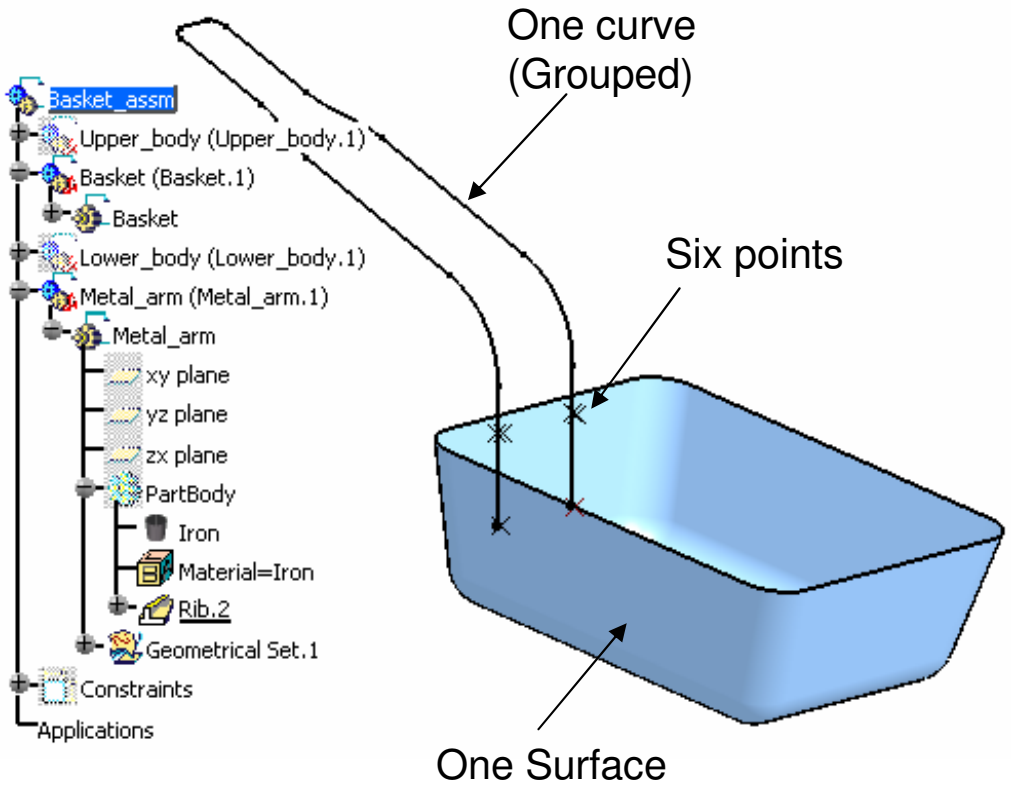
(1) To prepare the 3D model for analysis:-

- Enter CATIA
- File/Open/ Basket_assm_a.CATProduct

Now, we should see the below elements only:

- One surface
- Six points (4 points on surface, 2 points on curve)
- One Curve (grouped)

- Hidden elements are:-
  - Upper_body.1 & Lower_body.1
  - “Sweep.2” under Basket.1
  - “PartBody” under Metal_arm.1



Tutorial 3D

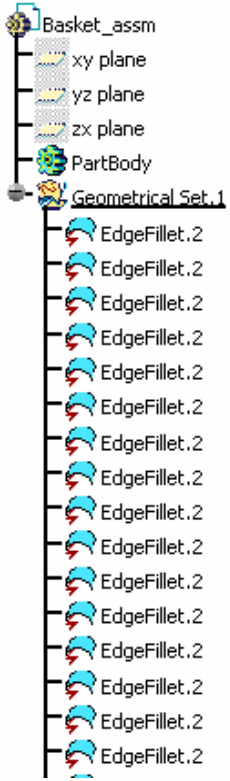
Export the file in an IGES file:-

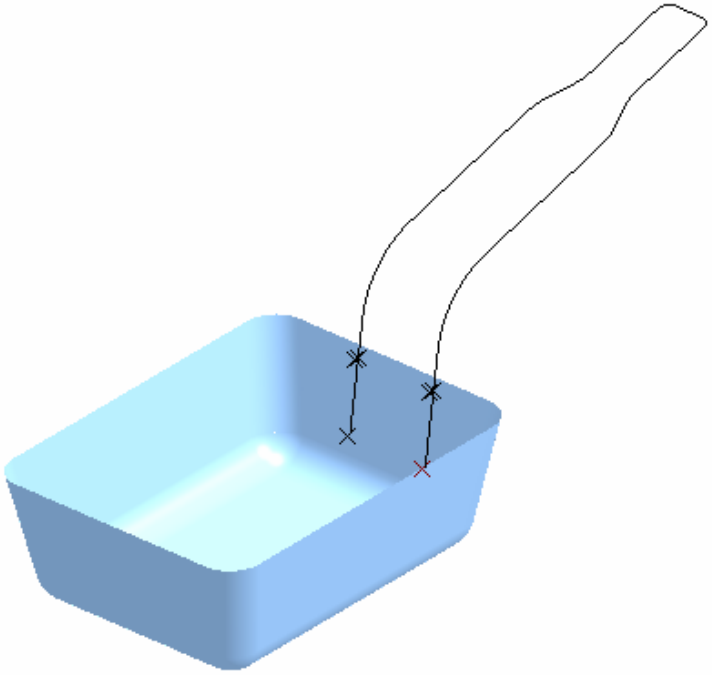
- File/Save as
- Select “igs” as File Type
- Enter “Basket_assm_a.igs” as File Name
- (Keep the file folder unchanged)
- Click “Save” to complete

File/Close/Basket_assm_a.Catproduct

Check and Re-save the IGES again:-

- File/Open/Basket_assm_a.igs
- (From the file, we can see that all elements are stored in the same level of the tree, and the product structure has been eliminated)
- Select “File/Save as” on the menu bar
- Select “igs” as File Type
- Select the file “Basket_assm_a.igs”
- Click “Save” and then “yes” to overwrite the file





Tutorial 3D

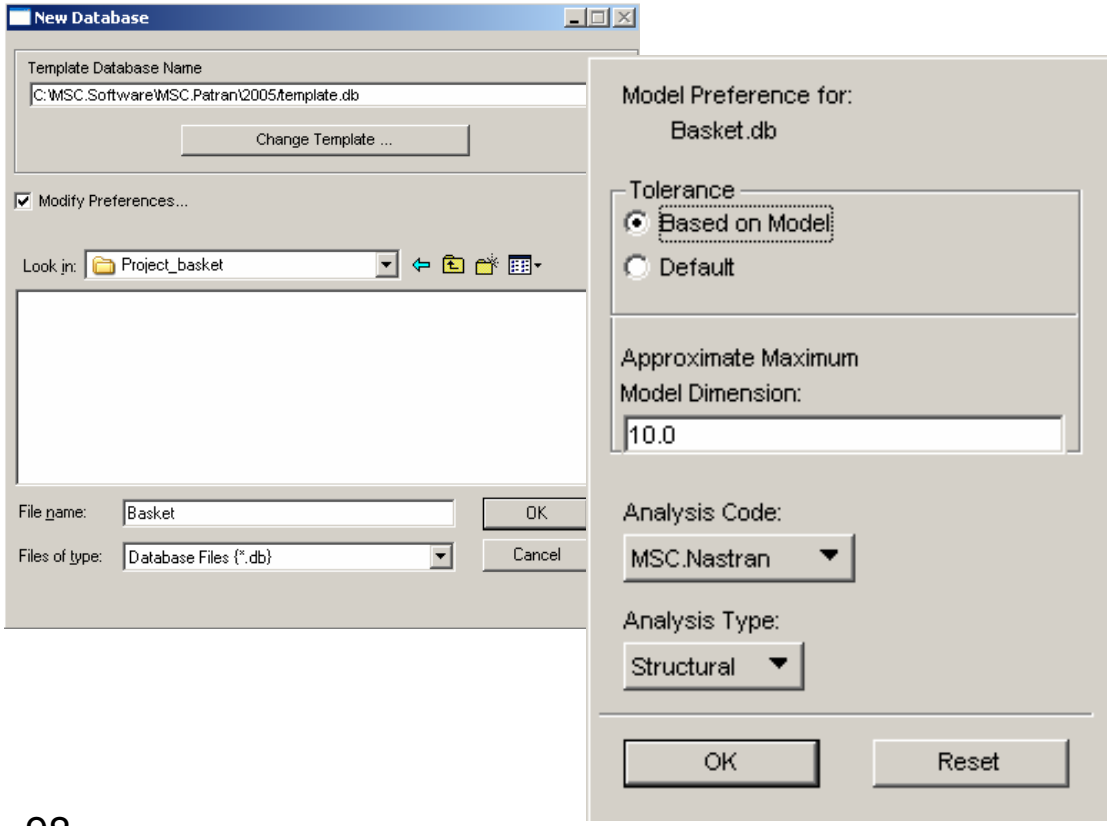
Close CATIA*

Enter MSC Patran*

- Select “MSC Software/MSC Patran2005/MSC Patran 2005” on the START menu of WindowsXP

(2) To Create a NEW Database:

- File/New
- Select Your project folder
- Enter “basket.db” as File Name
- Click OK
- Select “Based on Model” as Tolerance
- Select “MSC Nastran” as Analysis Code
- Select “Structural” as Analysis Type
- Click OK



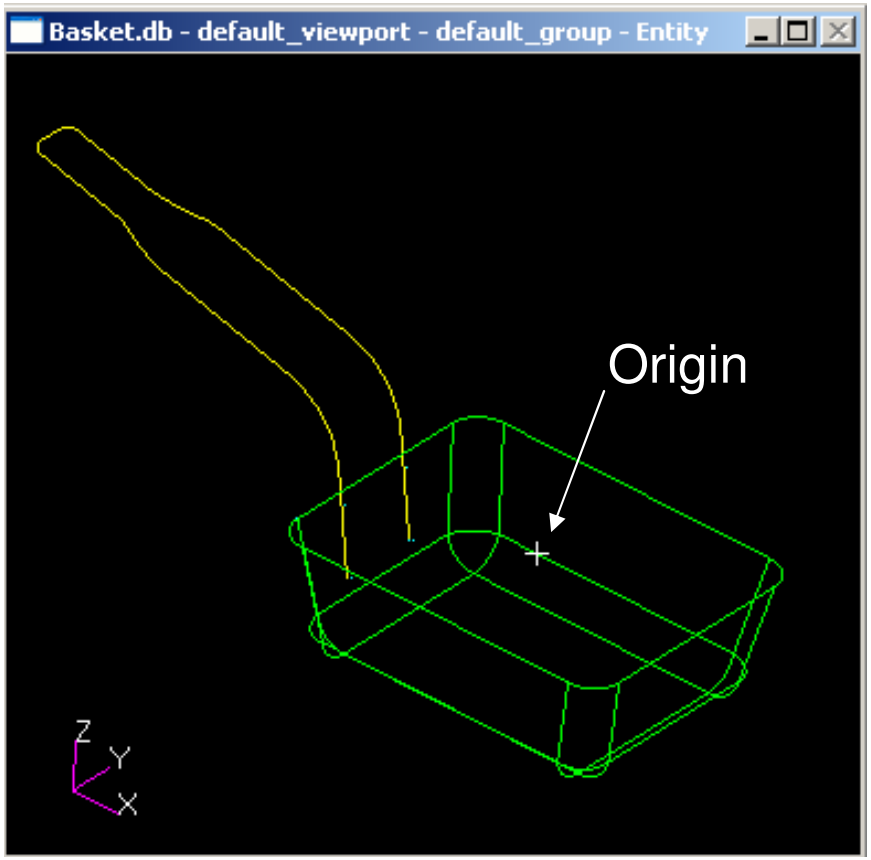
Tutorial 3D

(3) Import the model geometry:-

- File/Import
- Select “IGES” as Source
- Select your project folder
- Select the file “Basket_assm_a.igs”
- Click APPLY
- (all elements in IGES have been imported; the white “+” is the model origin)

PATRAN Entities Created

Model Space Scale	PATRAN Entity Type	Quantity
1.	Point	6
1.	Curve	1
1.	Surface	13
1.	Trimmed Surface	4



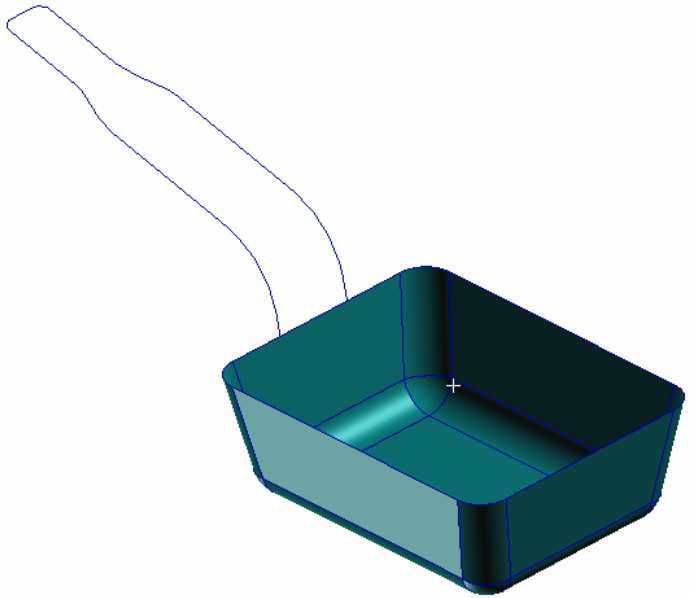
- The unit of CATIA models is mm. To have the unit consistency in Patran, remember to use the SI(mm) units as shown

Quantity	SI	SI(mm)
Length	m	mm
Force	N	N
Mass	kg	$tonne\ (10^3 kg)$
Time	s	s
Stress	$Pa\ (\mathrm{N}/m^2)$	$MPa\ (\mathrm{N}/mm^2)$
Energy	J	$mJ\ (10^{-3}\mathrm{J})$
Density	kg/m^3	$tonne/mm^3$

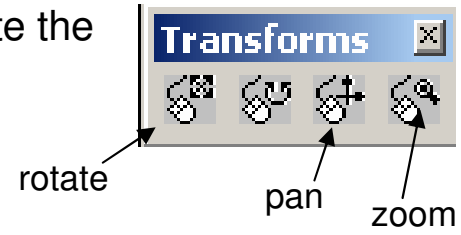
Tutorial 3D

View the model geometry:-

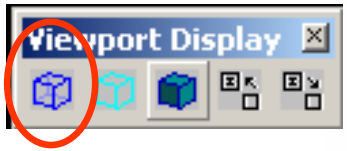
- Click "**Smooth Shaded**" icon to switch to the shading mode



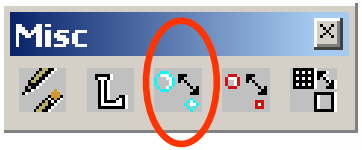
- Use the middle mouse button to rotate the model

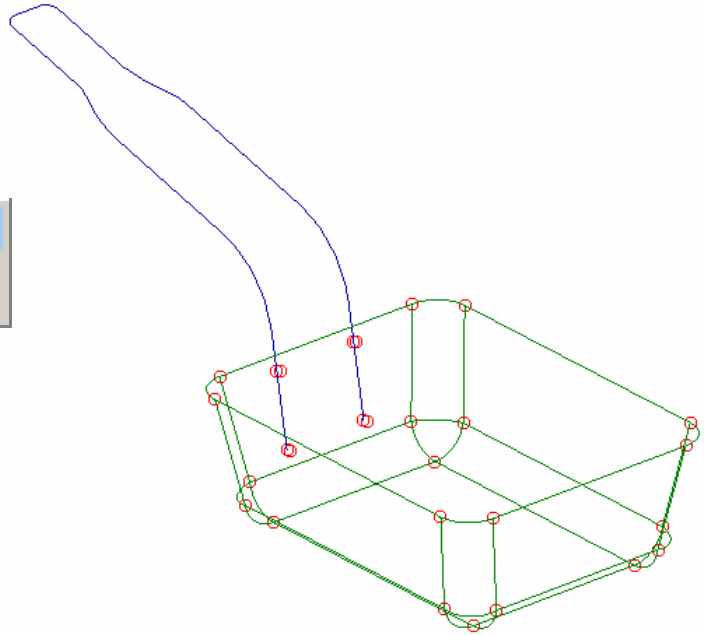


- Click "**Wireframe**" icon to switch to the wireframe mode



- Click "**Point Size**" icon to increase the point size so that we can see all locations of points



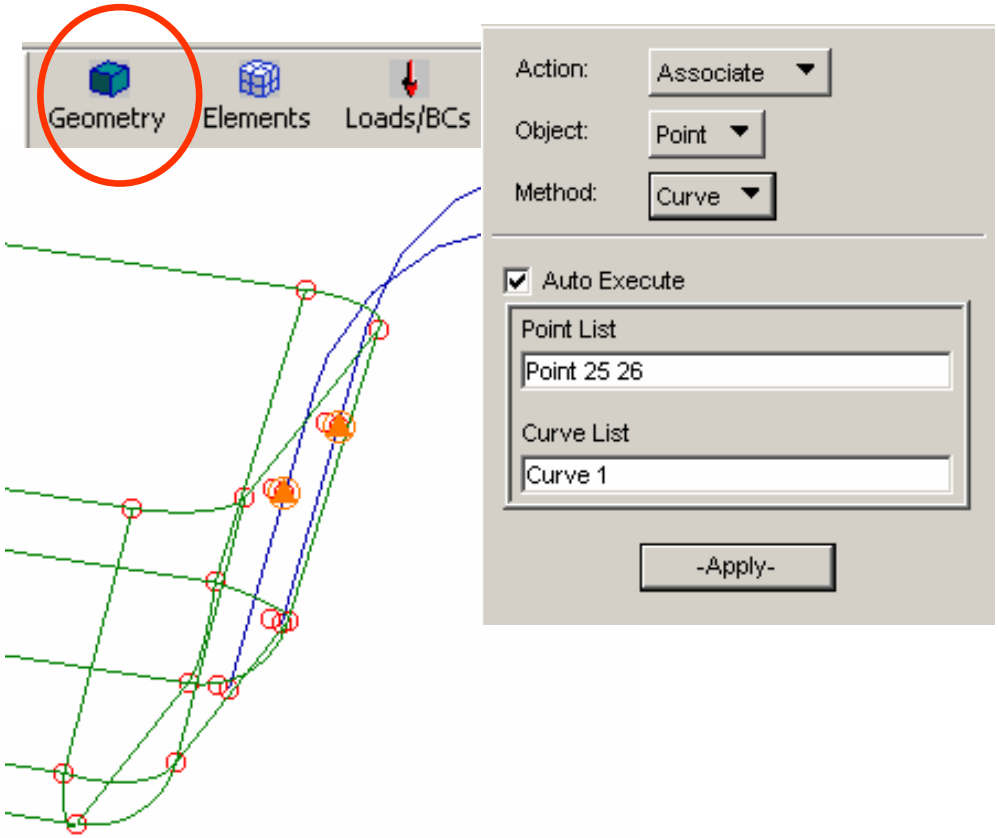


Tutorial 3D

(4a) To associate points to a curve:-

- Click "**Geometry**" icon on the top menu
- Select *Action/Associate*
- Select *Object/Point*
- Select *Method/Curve*
- Click the Entry Box of *Point List* Once, then select a point on the curve
- Click the box again
- Press and hold "SHIFT" key
- Select another point on the curve

- Click the Entry Box of *Curve List*, then select the curve
- (If Auto-execute is checked, it is not necessary to click Apply)

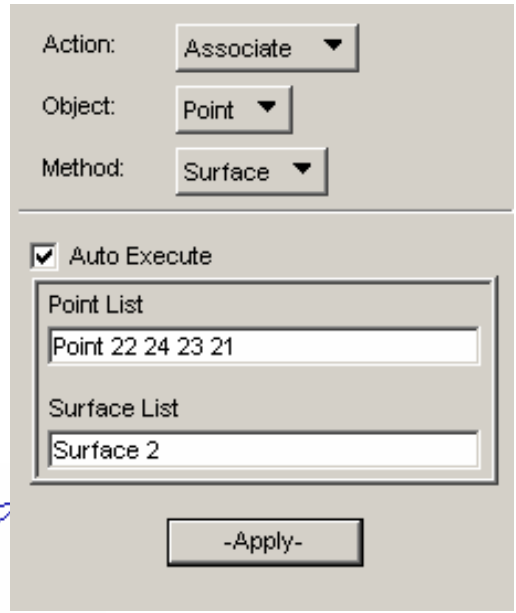


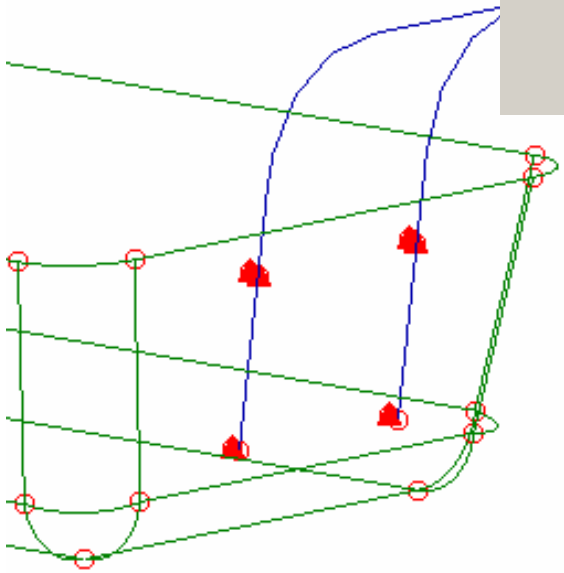
Tutorial 3D

(4b) To associate points to a surface:-

- (Keep *Action/Associate*)
- Select *Object/Point*
- Select *Method/Surface*
- Click the Entry Box of *Point List* Once, then select a point on the surface
- Click the box again
- Press and hold “SHIFT” key
- Select another point on the surface
- (Repeat the steps until all 4 points are selected)

- Click the Entry Box of *Surface List*, then select the Surface (at which the 4 points are located)
- (If Auto-execute is checked, it is not necessary to click Apply)





Tutorial 3D

(5a) To create a 1D Mesh:-

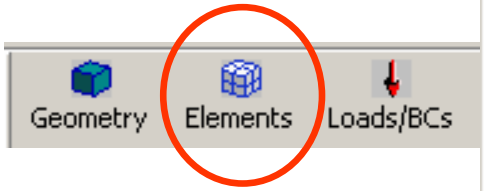
- Click "**Elements**" icon on the top menu
- Select *Action/Create*
- Select *Object/Mesh*
- Select *Type/Curve*
- Select *Bar2* as Topology

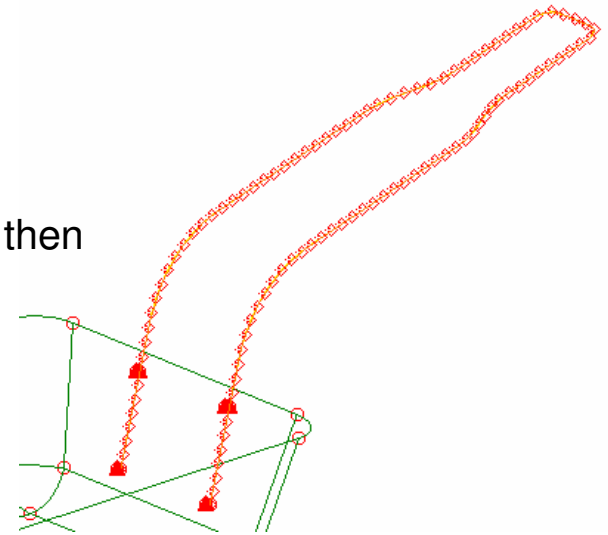
- Click the Entry Box of *Curve List* Once, then select the curve on screen

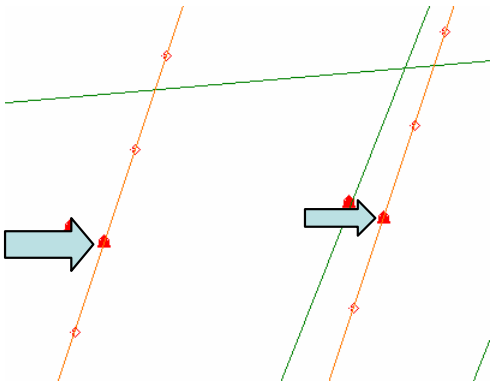
- Deselect Automatic Calculation
- Enter 5 for the *Global Edge Length*

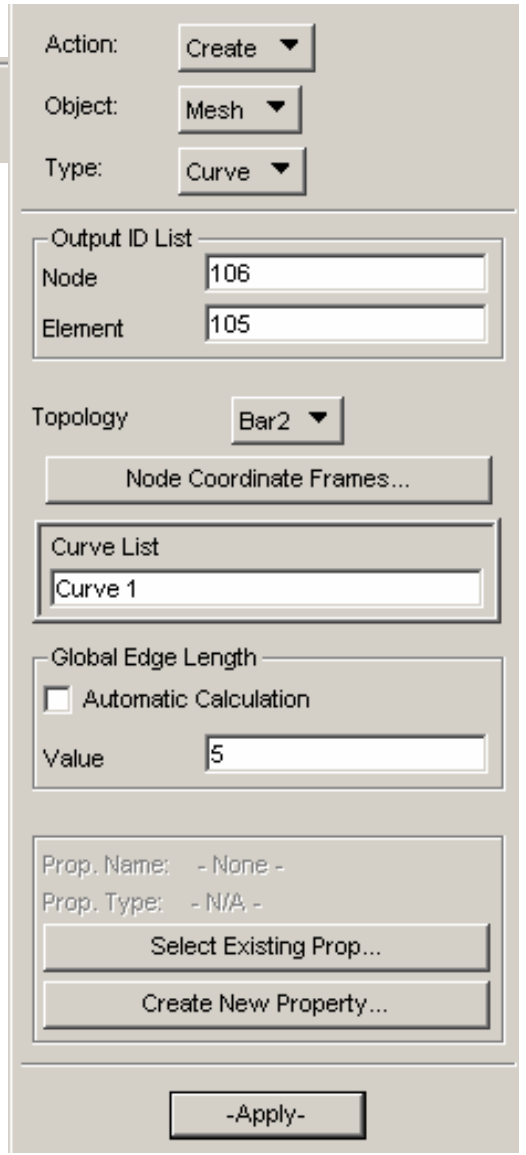
- Click **Apply**

- (If we look closer to the nodes, we can see that a node is created on the location of the associated point)





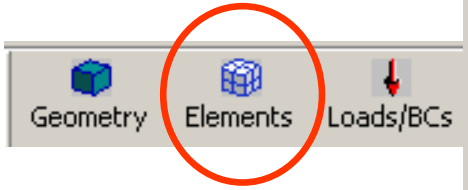


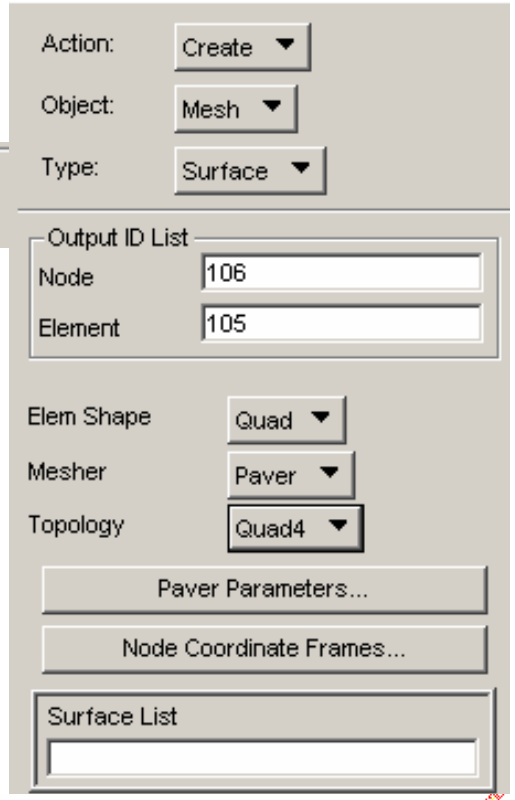


Tutorial 3D

(5b) To create a 2D Mesh:-

- Click "**Elements**" icon on the top menu
- Select *Action/Create*
- Select *Object/Mesh*
- Select *Type/Surface*

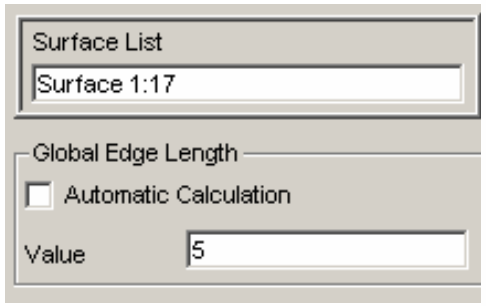




- Select *Quad* as Element Shape
- Select *Paver* as Mesher
- Select *Quad4* as Topology

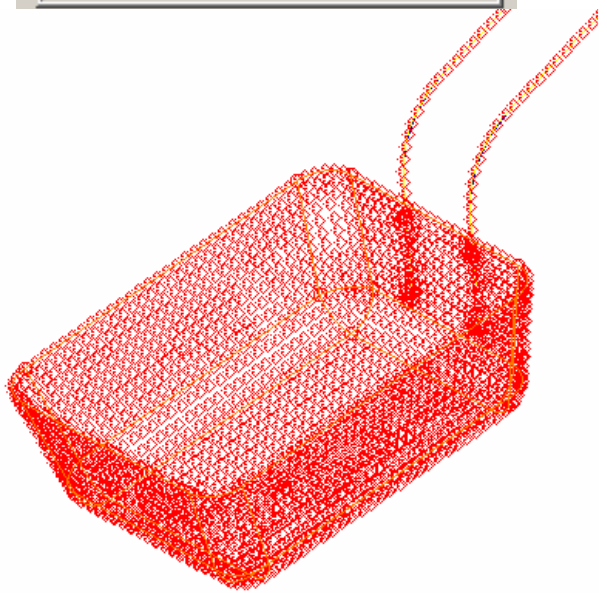
- Click the Entry Box of *Surface List* Once, then Click the Surface icon under Picking Filters ☆

- Select all Surfaces on screen



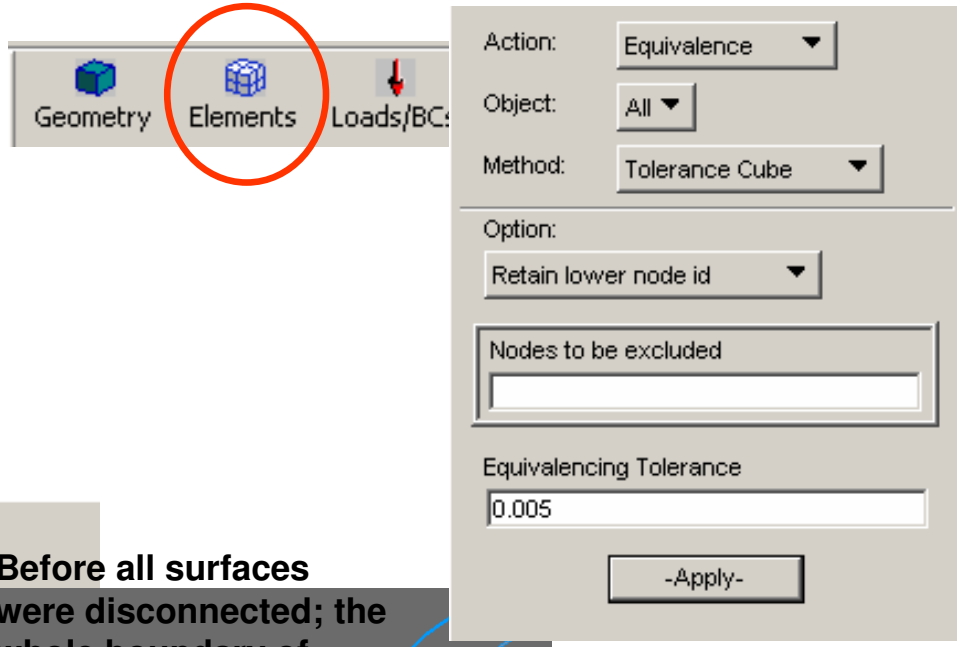
- Deselect Automatic Calculation
- Enter 5 for the *Global Edge Length*

- Click **Apply**
- (If we look closer to the nodes, we can see that a node is created on the location of the associated point)



Tutorial 3D

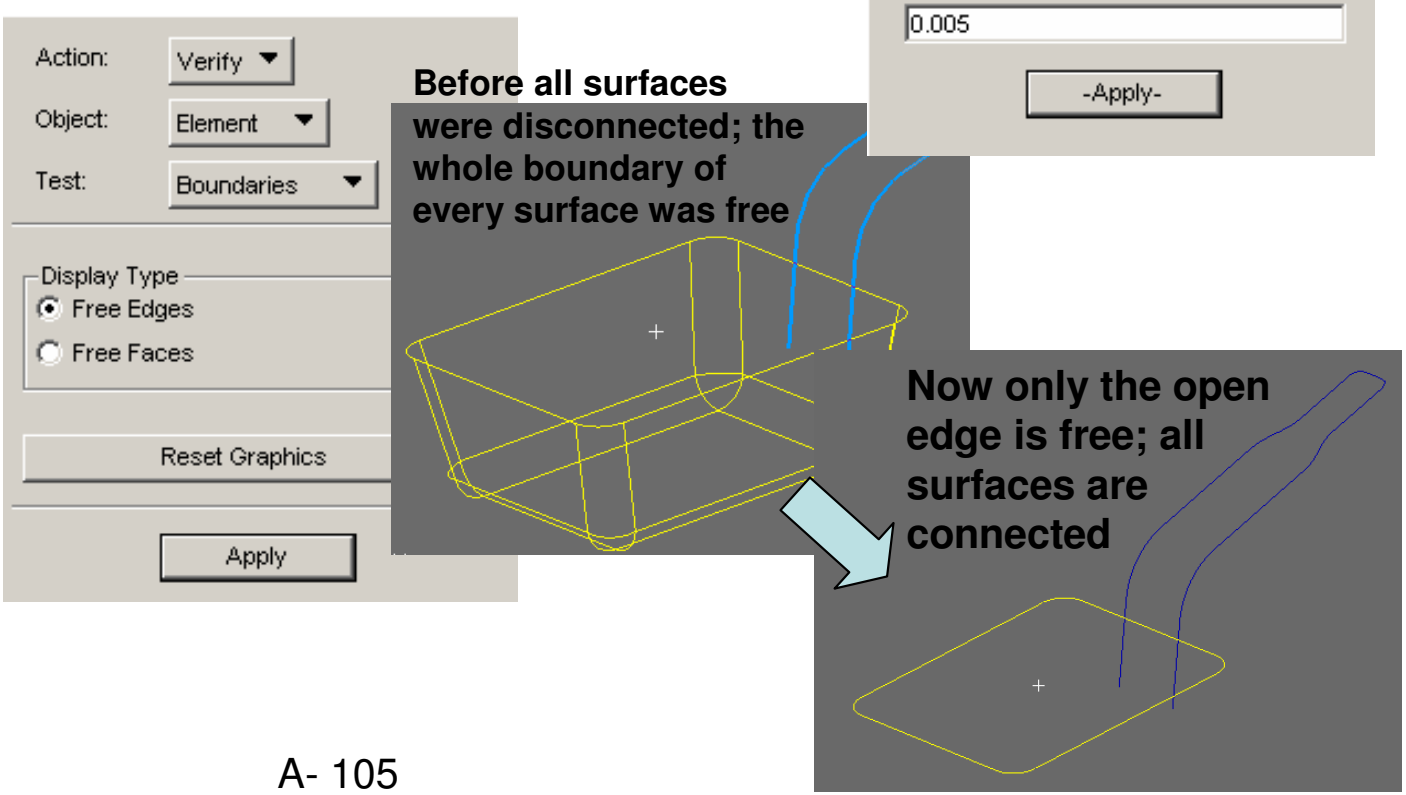
(6) To equivalence Nodes to connect 2D Quad Elements:-



- Select *Action/Equivalence*
- Select *Object /All*
- Select *Method/Tolerance Cube*
- Click **Apply**

- Select *Action/Verify*
- Select *Object/Element*
- Select *Test/Boundaries*
- Click **Apply**

- **Reset Graphics**



Tutorial 3D

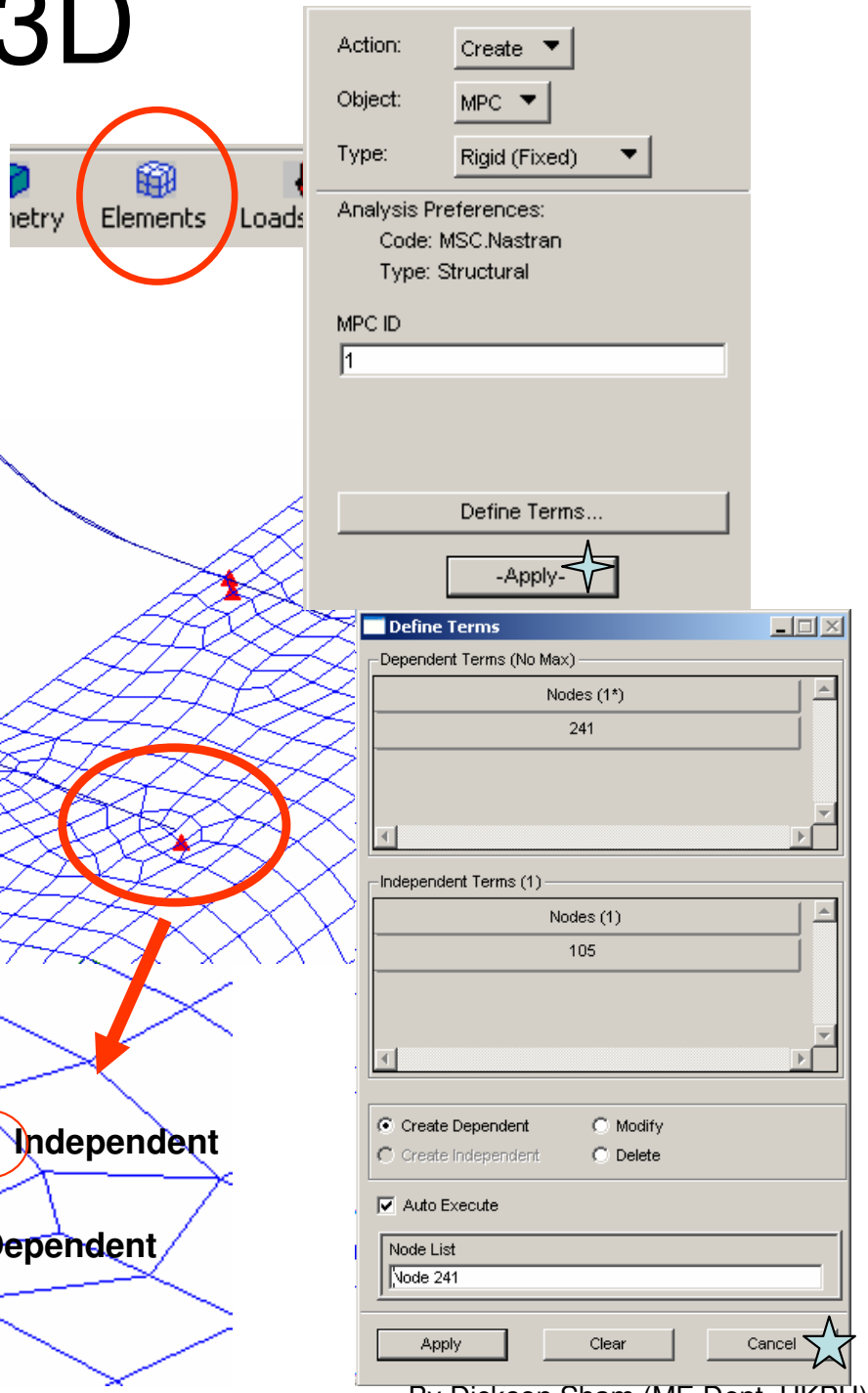
(7) To create Rigid Joints between two meshes:-

- Click "**Elements**" icon on the top menu
- Select *Action/Create*
- Select *Object/MPC*
- Select *Type/Rigid(Fixed)*

- Select "**Define Terms..**"
- Select "Create Independent" and then the end node of the curve
- Select the node with ▲ on the surface as Dependent Node

- (If Auto-execute is checked, it is not necessary to click Apply)
- Click "Cancel" ☆ on the submenu
- Click "Apply" ✦ on the top menu

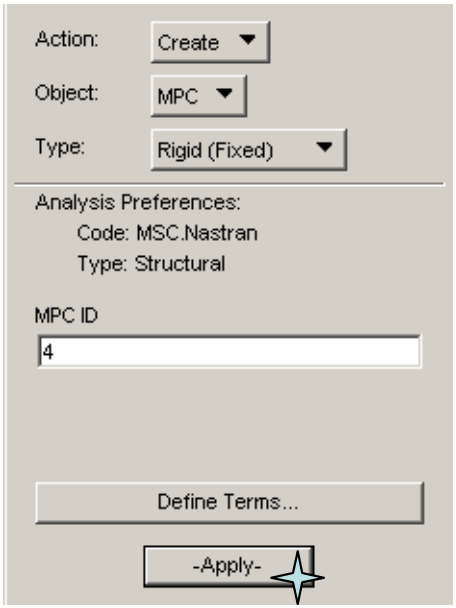
- (NOW MPC.1 is created)

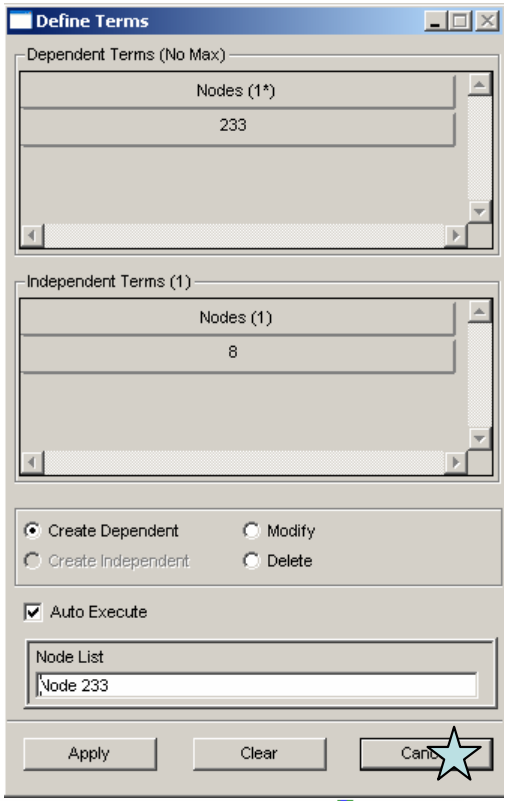


Tutorial 3D

(Cont'):-

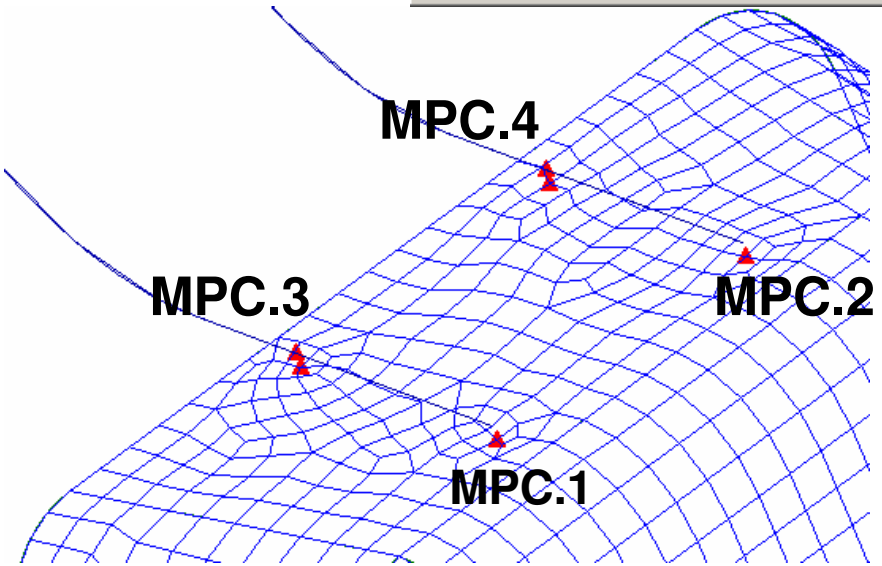
- Select "**Define Terms..**"
- Select "Create Independent" and then another node of the curve
- Select the corresponding node on the surface as Dependent Node





- (If Auto-execute is checked, it is not necessary to click Apply)
- Click "Cancel" ☆ on the submenu
- Click "Apply" ✦ on the top menu

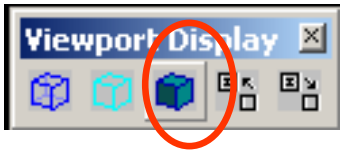
- Repeat the steps until all four MPCs are created
  - MPC.1
  - MPC.2
  - MPC.3
  - MPC.4



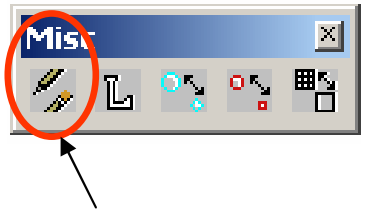
Tutorial 3D

To show the meshes only:-

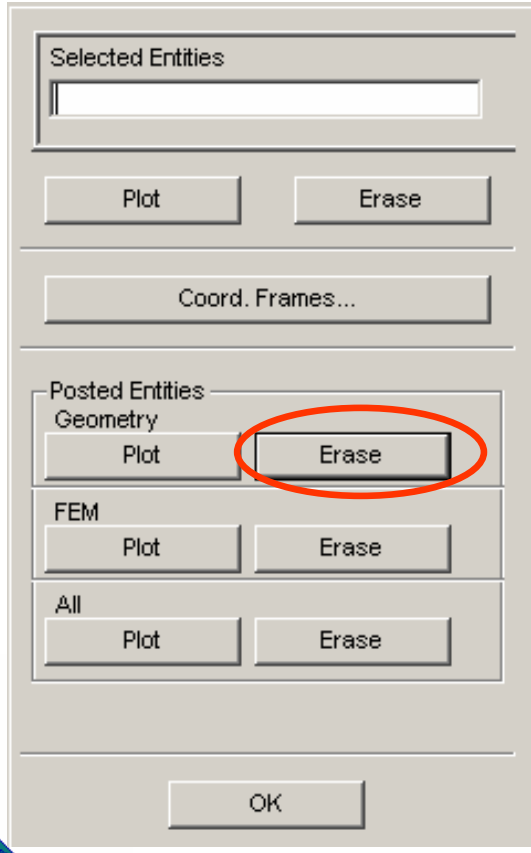
- Click "**Smooth Shaded**" icon to switch to the shading mode

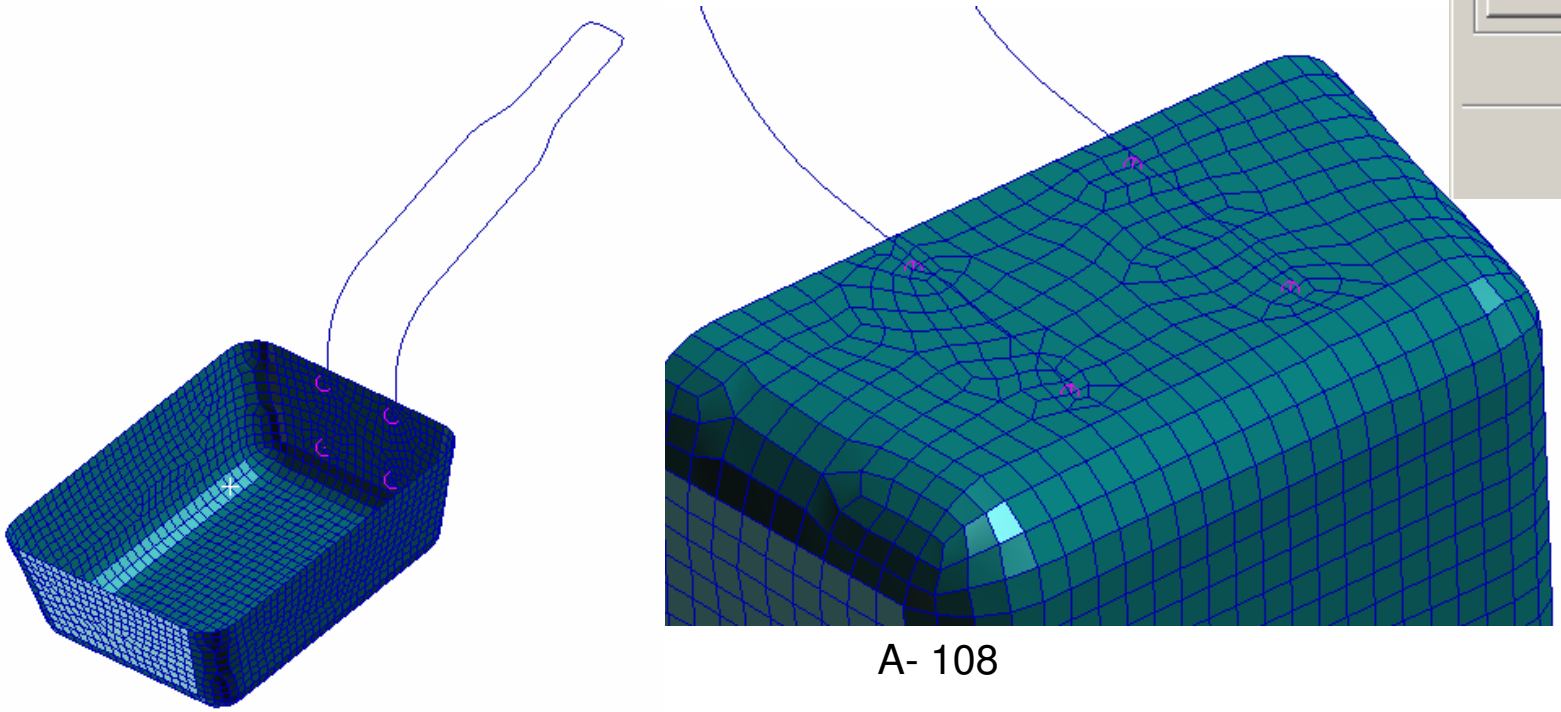


- Click "**Plot/Erase**" icon
- Select "**Erase**" under Geometry
- Click ok
- (Previously, the geometry and the meshes were overlapped together, but now only meshes are shown on the screen)



Plot/Erase





Tutorial 3D

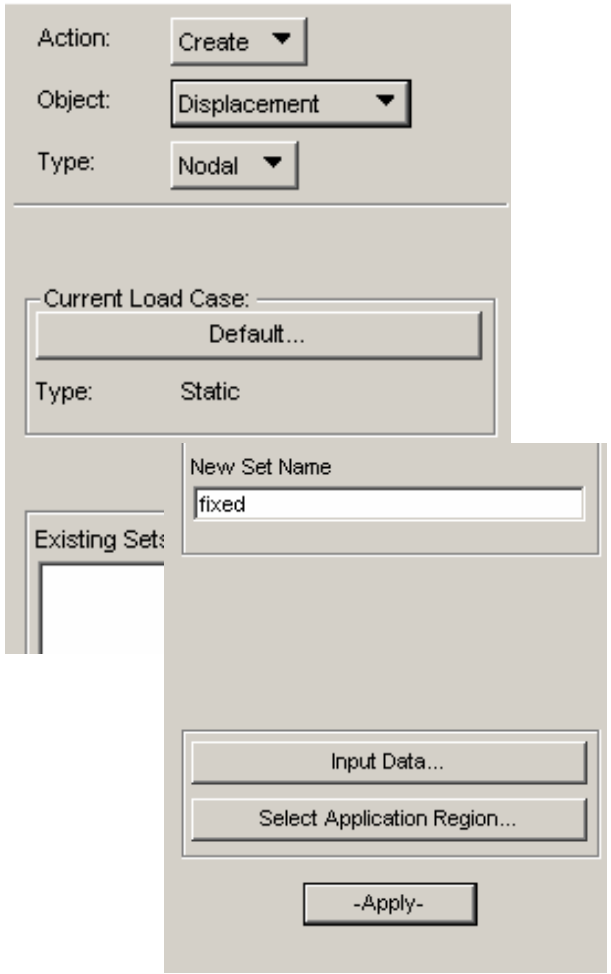
(8) To create a Constraint:-

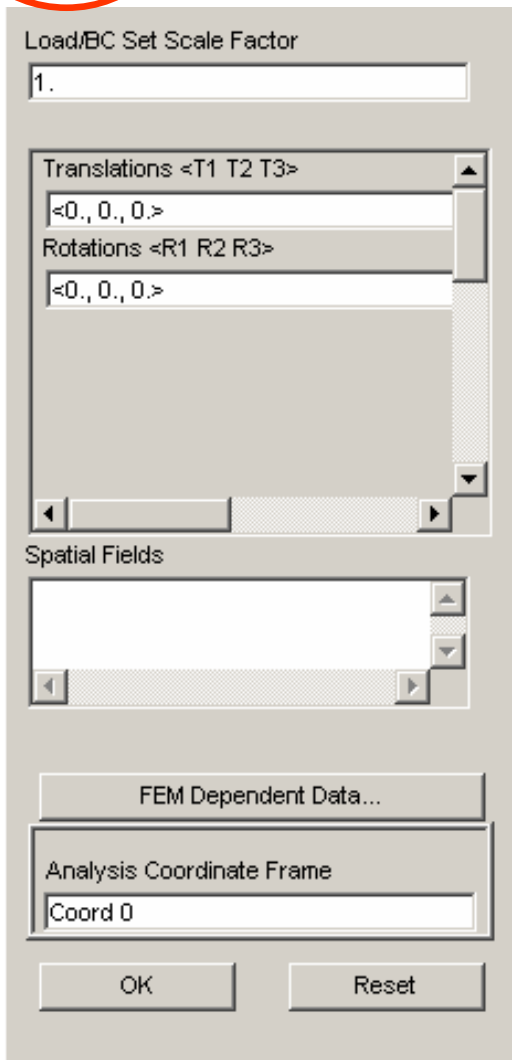
- Click "**Loads/BCs**" icon on the top menu

- Select *Action/Create*
- Select *Object/Displacement*
- Select *Type/Nodal*

- Click the entry box of New Set Name
- Enter "fixed" in the box

- Click "**Input Data…**" icon
- Enter <0 0 0> under Translation
- Enter <0 0 0> under Rotation
- Click ok

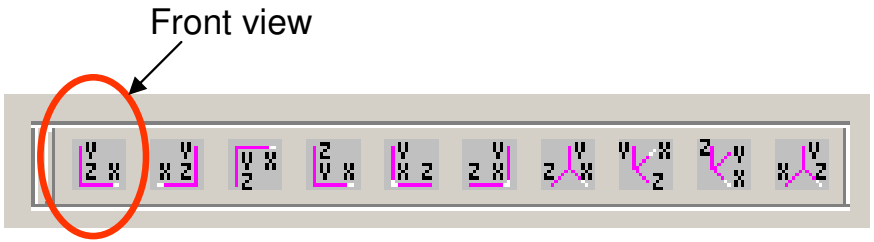


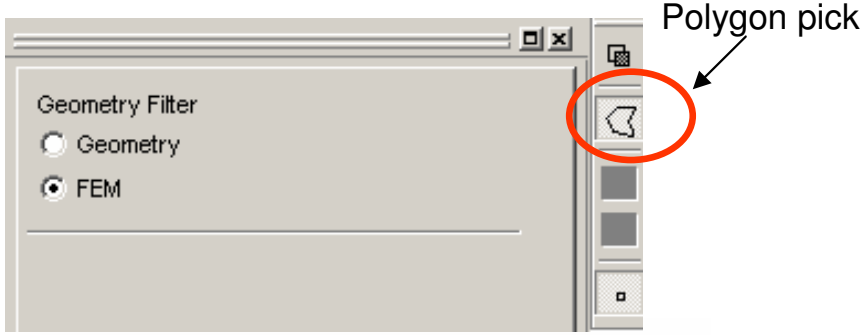


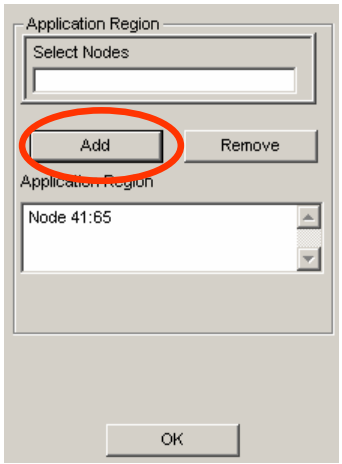
Tutorial 3D

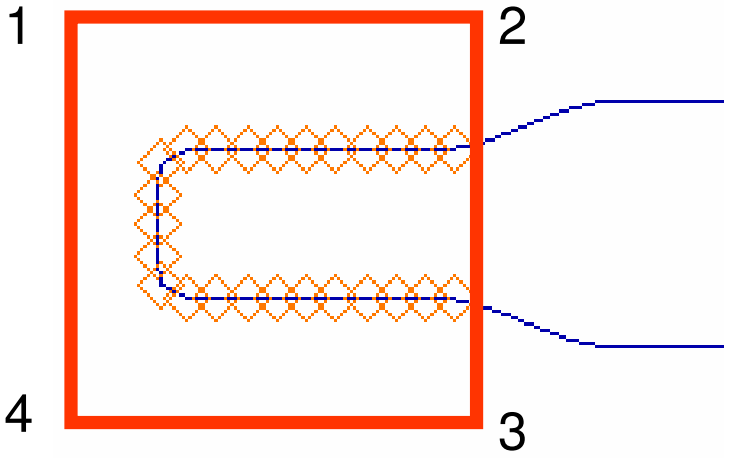
(Cont') :-

- Click "**Select Application Region**" icon
- Select FEM under Geometry Filter
- Click "Front View" icon
- Click "Polygon pick" icon
- Click at ***1 , 2, 3***
- then Double-Click at ***4*** to select all nodes within the region
- Click "**Add**" icon
- Click ok
- Finally Click Apply









Tutorial 3D

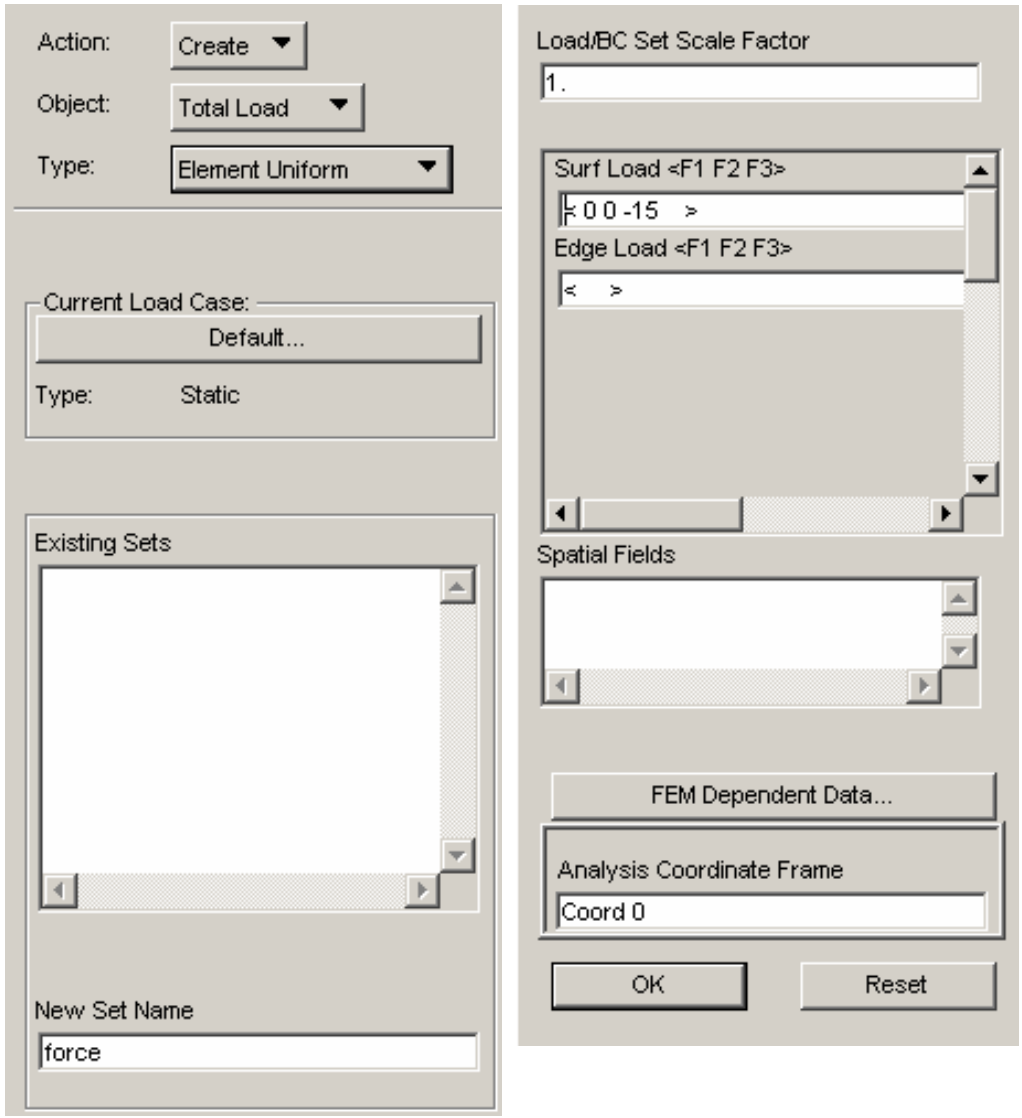
(9) To create a Distributed Load :-

- Select *Action/Create*
- Select *Object/Total Force*
- Select *Type/Nodal*

- Click the entry box of New Set Name
- Enter “Load” in the box

- Select *2D* as *Target Element Type*

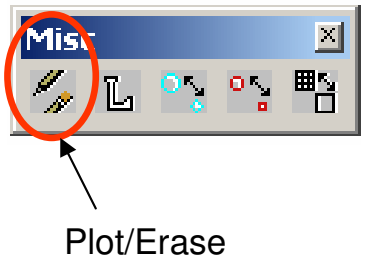
- Click “**Input Data…**”
- Enter <0 0 -15> under Surf. Load (Patran will distributed the 15N load evenly over the area of the Application Region)
- Click ok



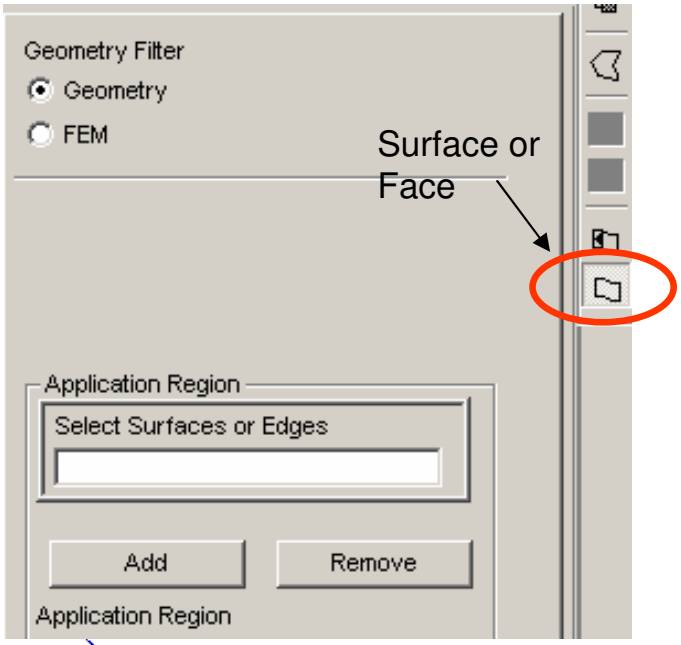
Tutorial 3D

(Cont') :-

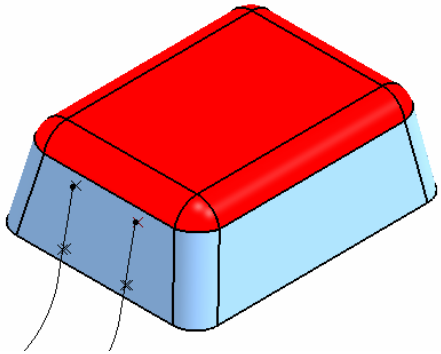
- Click "**Plot/Erase**" icon
- Select "Plot" under Geometry
- Click ok

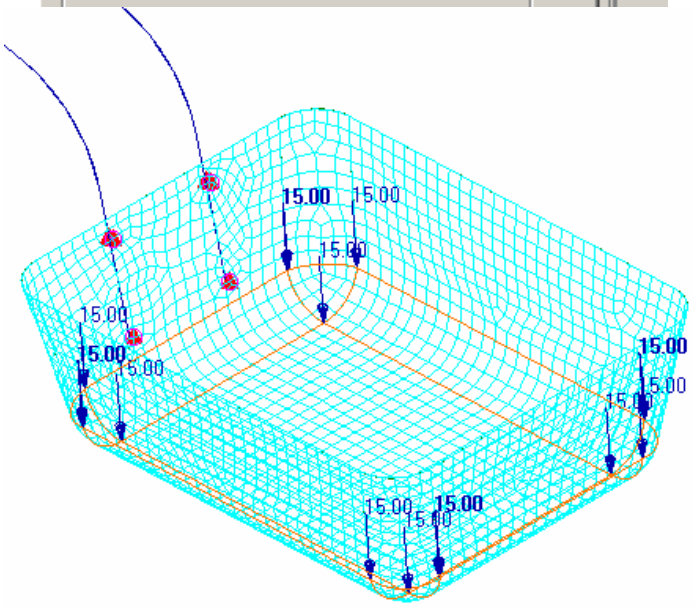


- Click "**Select Application Region**" icon
- Select "Geometry" under Geometry Filter
- Click "Surface or Face" icon for Picking Filter



- Press and Hold "Shift" key on the keyboard
- Multi-select all bottom faces (9 faces)
- Click "**Add**" icon
- Click OK
- Click Apply

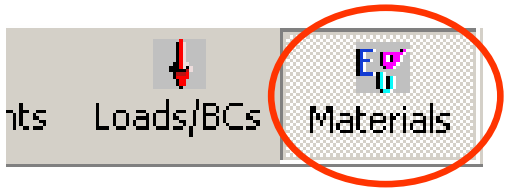




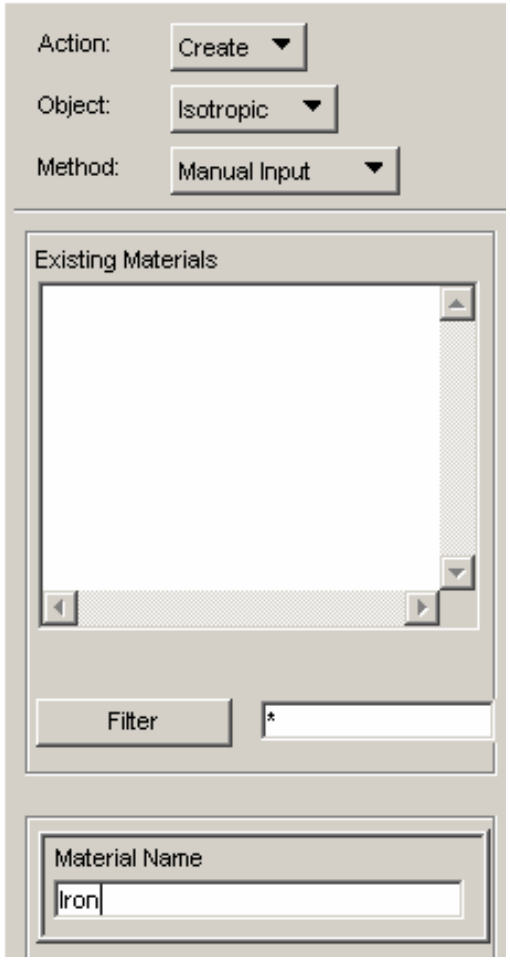
Tutorial 3D

(10) To create Material Properties :-

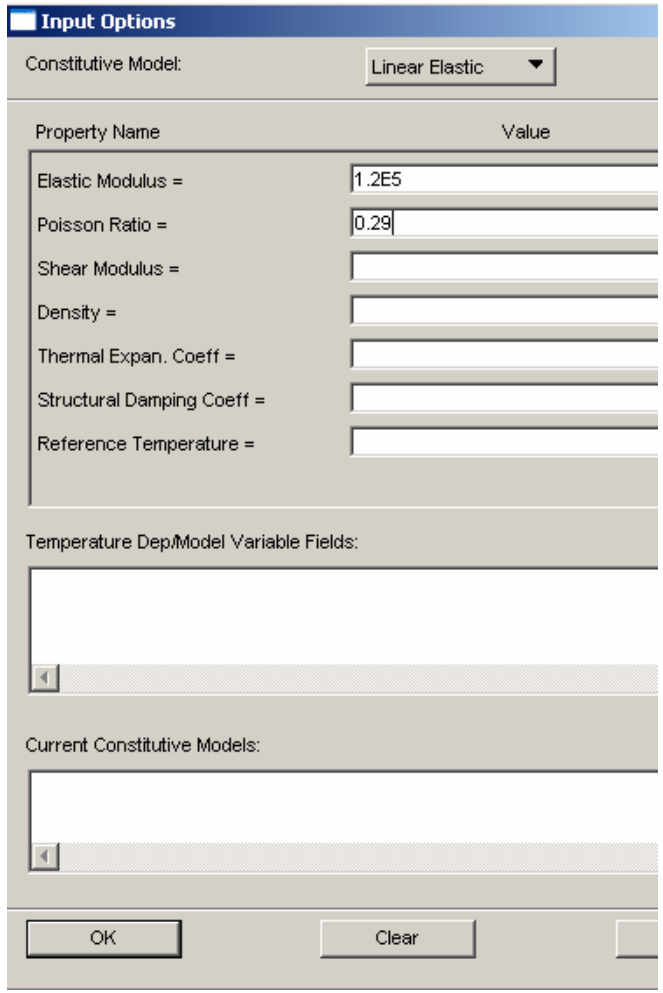
- Click "**Materials**" icon on the top menu



- Select *Action/Create*
- Select *Object/Isotropic*
- Select *Method/Manual Input*



- Enter "Iron" for Material Name
- Click on "Input Properties…"
- Enter 1.2E5 and 0.29 for Elastic Modulus(N_mm2) and Poisson ratio respectively



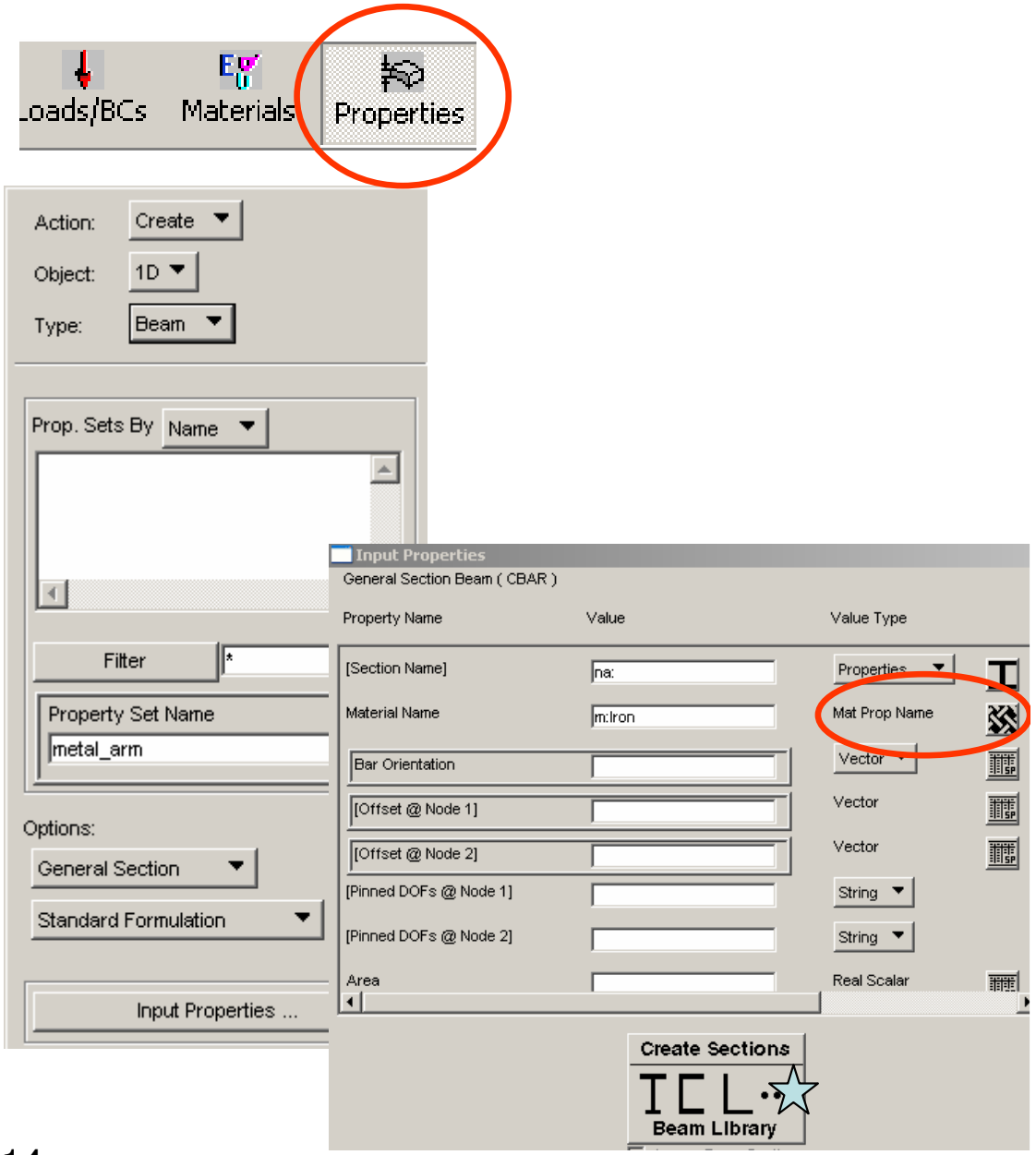
- Click ok
- Click Apply

Tutorial 3D

(11) To create Element Properties :-

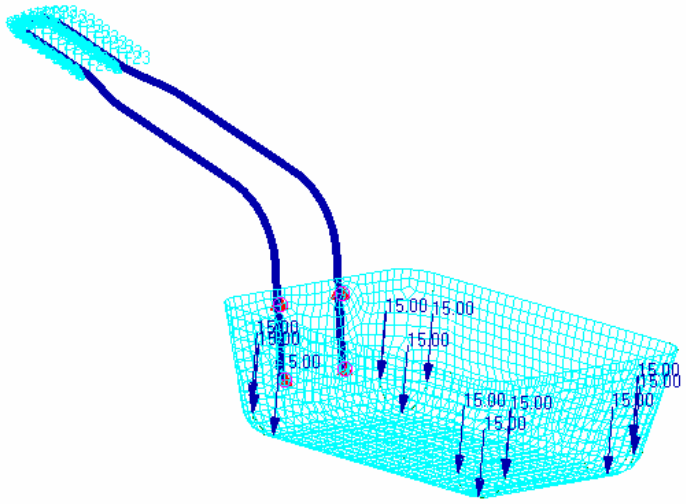
(FOR 1D MESH)

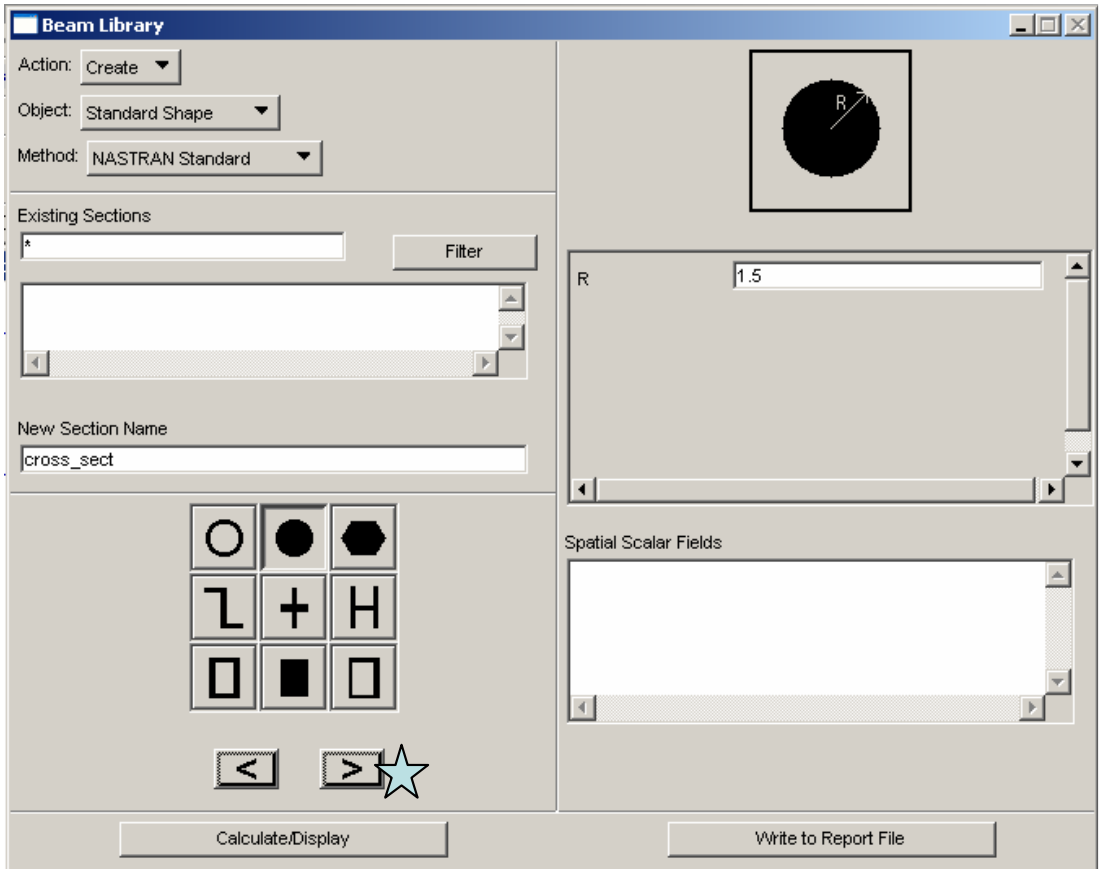
- Click "**Properties**" icon on the top menu
- Select *Action/Create*
- Select *Object/1D*
- Select *Type/Beam*
- Enter "metal_arm" as Property Set Name
- Click "Input Properties" icon
- Click the icon next to "Mat Prop Name" and then select Iron
- Click "Create Sections Beam Library" icon

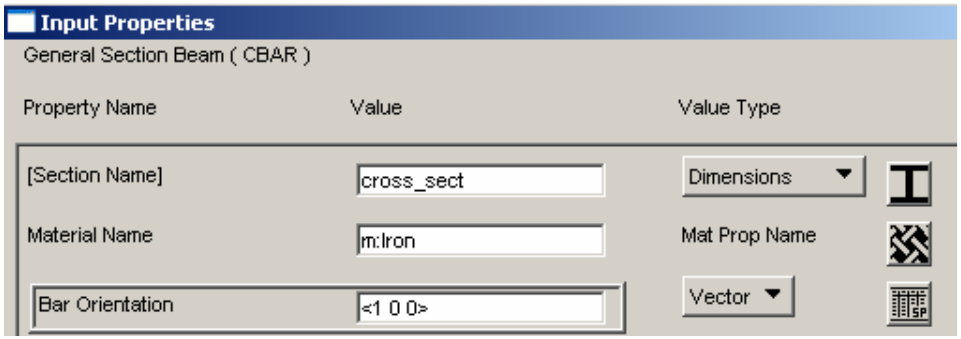


Tutorial 3D

- Enter “Cross_sect” as New Section Name
- Click the arrow icon
- Select the cross-section with a solid circle
- Enter 1.5 as R
- Click ok
- Enter <1 0 0> as *Bar Orientation*
- Click the entry box of “Select Members” and then select the curve on the screen
- Click Add
- Click Apply
- (Optional) Display the cross-section by selecting…
  - Display/Load/BC/Elem. Props…using Beam Display/3D: Full-Span +Offsets.







Tutorial 3D

(Cont') :-

(FOR 2D MESH)

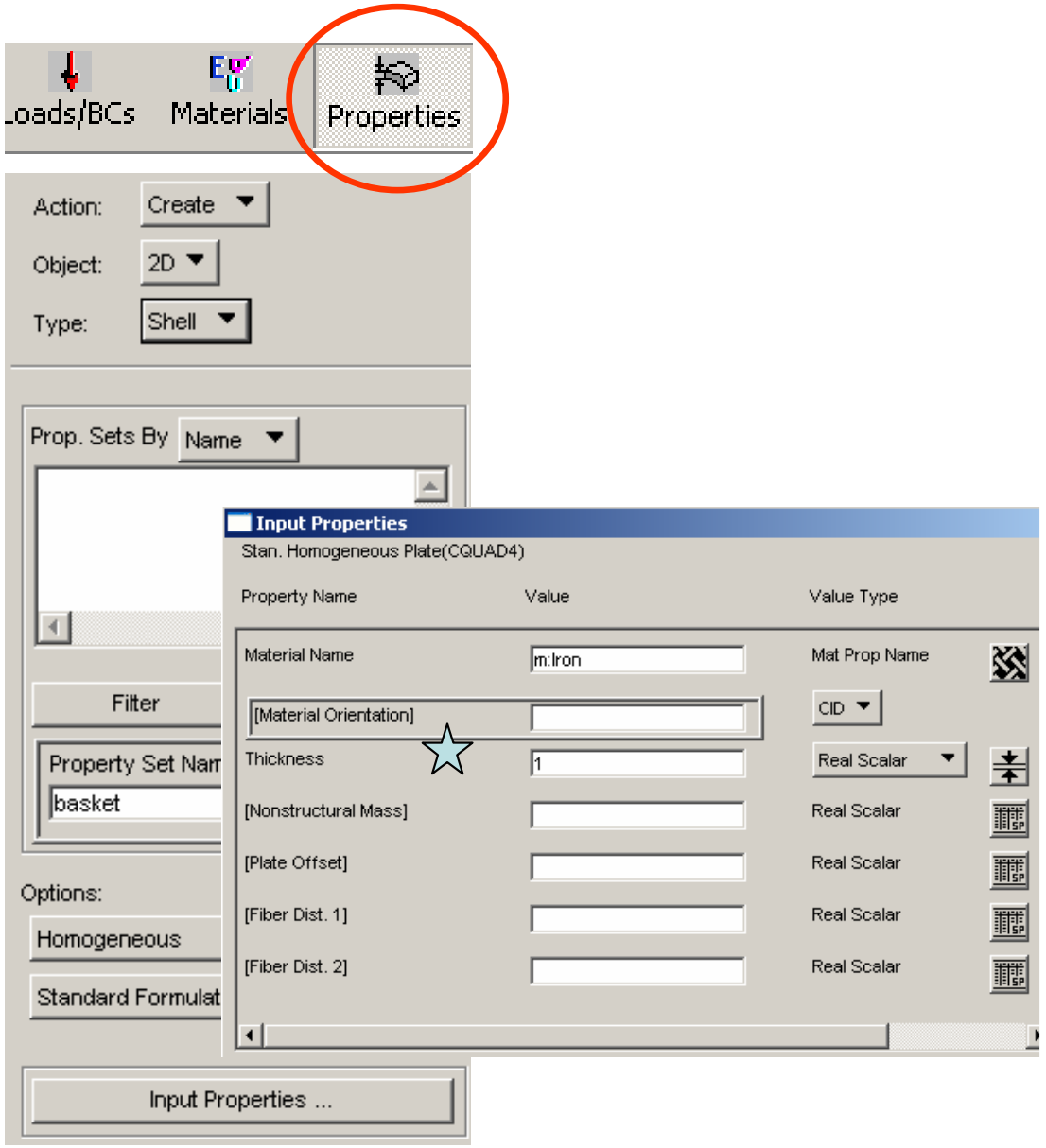
- Select *Action/Create*
- Select *Object/2D*
- Select *Type/Shell*
- Enter "basket" as Property Set Name

- Click "Input Properties" icon

- Click the icon next to "Mat Prop Name" and then select Iron (it has been selected by default)

- Enter 1 as Thickness and Click OK

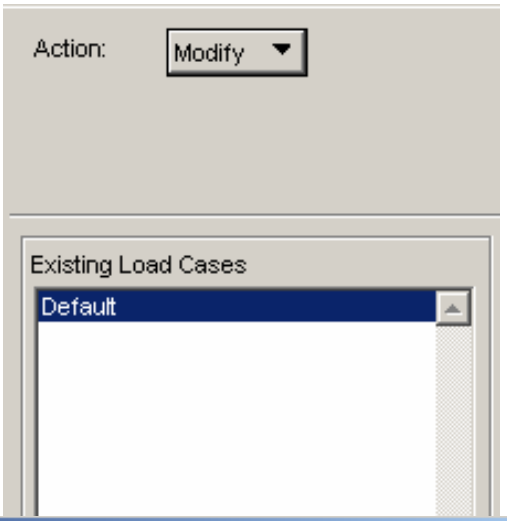
- Click the entry box of "Select Members" and then select all surfaces on the screen
- Click Add
- Click Apply

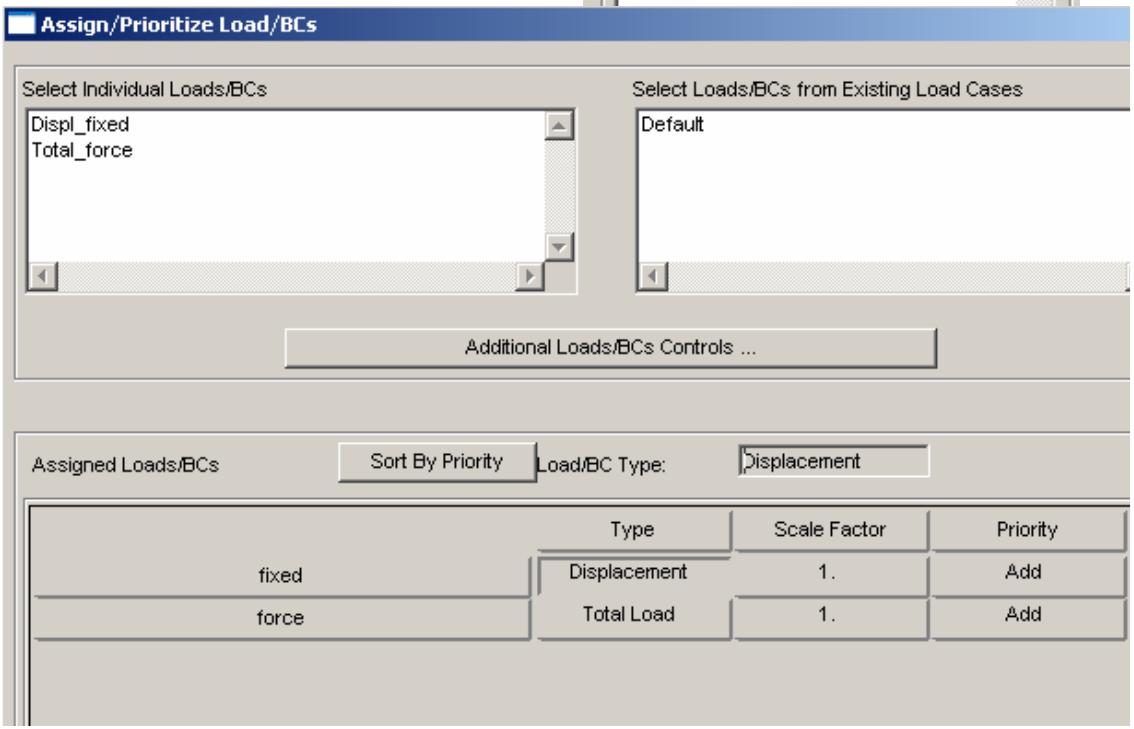


Tutorial 3D

(12) To check assignment of Loads and Boundary Conditions :-

- Click "**Load Case**" icon on the top menu
- Select *Action/Modify*
- Select Load case **Default** from "Select Load Case to Modify"
- Check to see that the Total Load and the fixed constraint are assigned to the default load case
- Click **Cancel**





Tutorial 3D

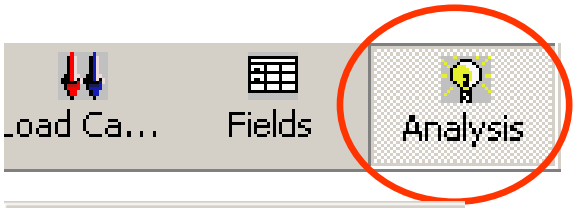
(13) To Run the Analysis:-

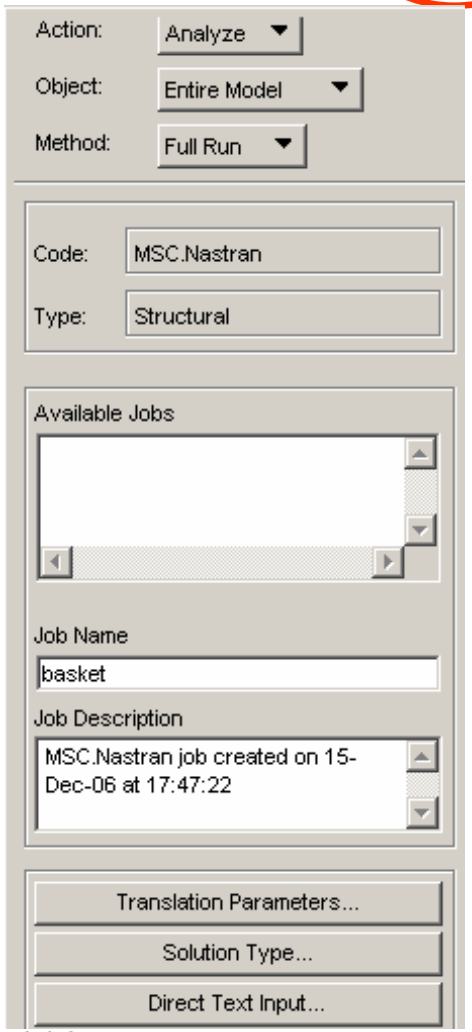
- Click "**Analysis**" icon on the top menu
- Select *Action/Analyze*
- Select *Object/Entire Model*
- Select *Method/Full Run*

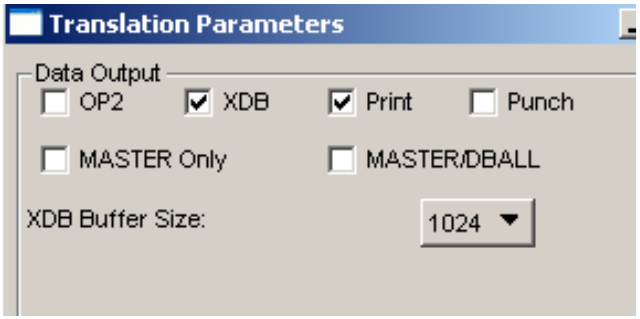
- Click "**Translation Parameters…**"
- Select **XDB** and **Print**
- Click OK

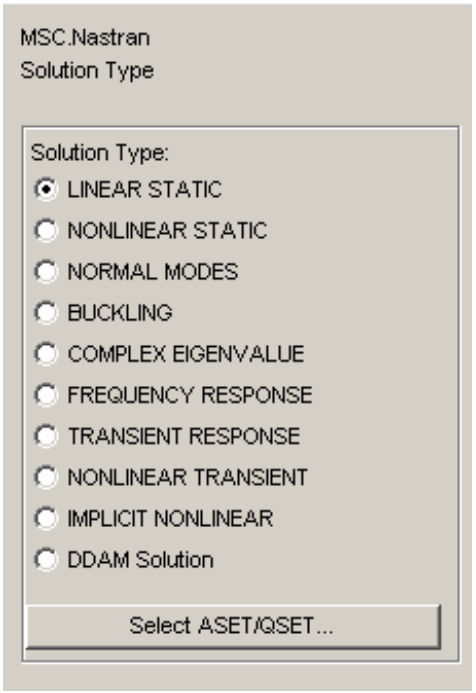
- Click "**Solution Type**"
- Select "**Linear Static**" as Solution Type
- Click Ok

- Click Apply





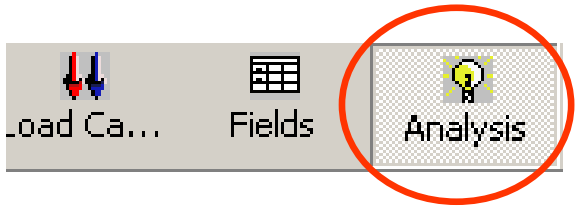


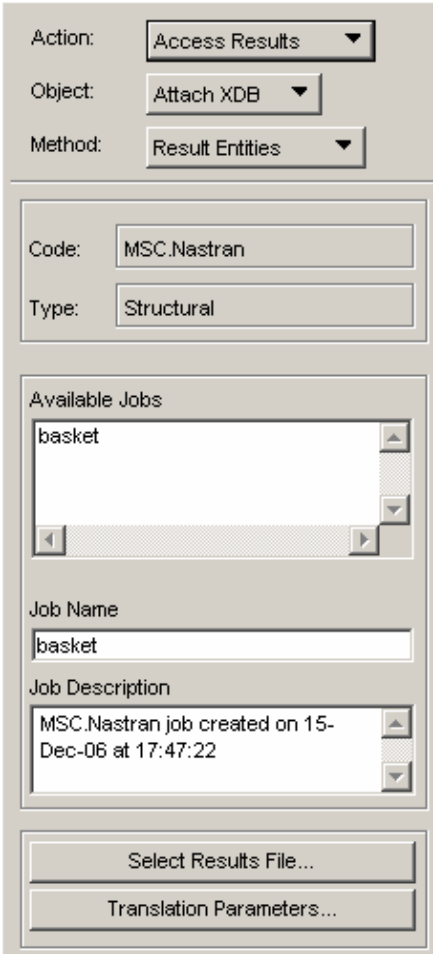


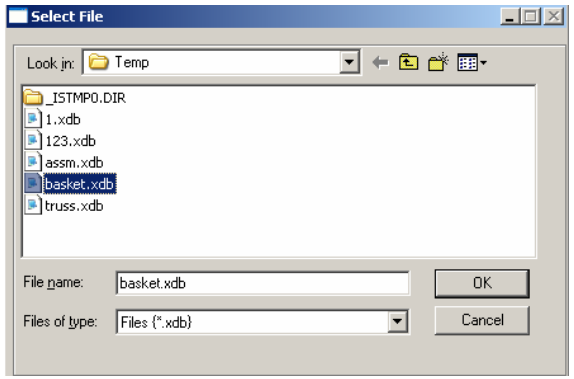
Tutorial 3D

(14) To Read the results:-

- Select *Action/Access Results*
- Select *Object/Attach XDB*
- Select *Method/Result Entities*

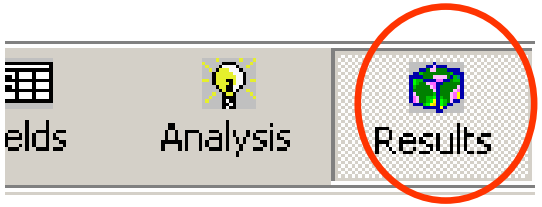


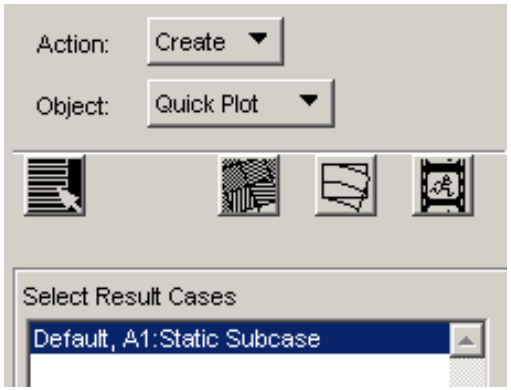




- Click "**Select Results File…**"
- Select the file "basket.xdb" and Click OK
- Click Apply

- Click "**Results**" icon on the top menu
- Select *Action/Create*
- Select *Object/Quick Plot*



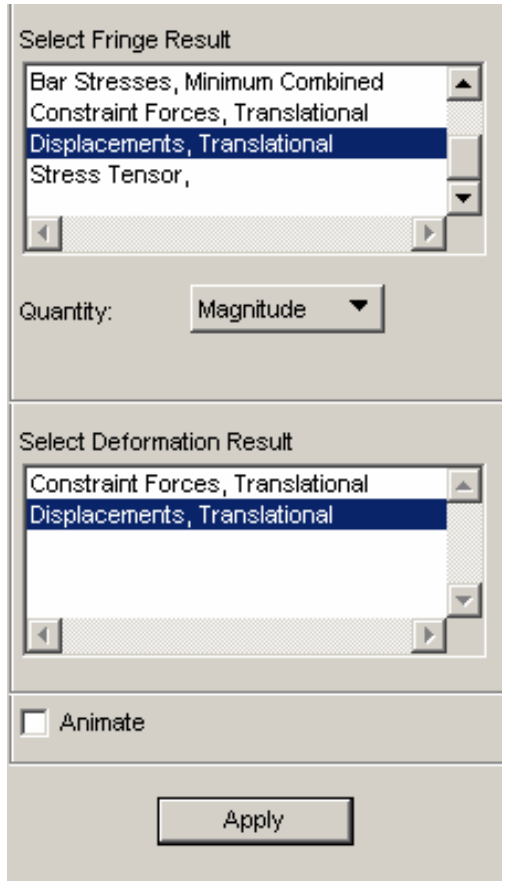


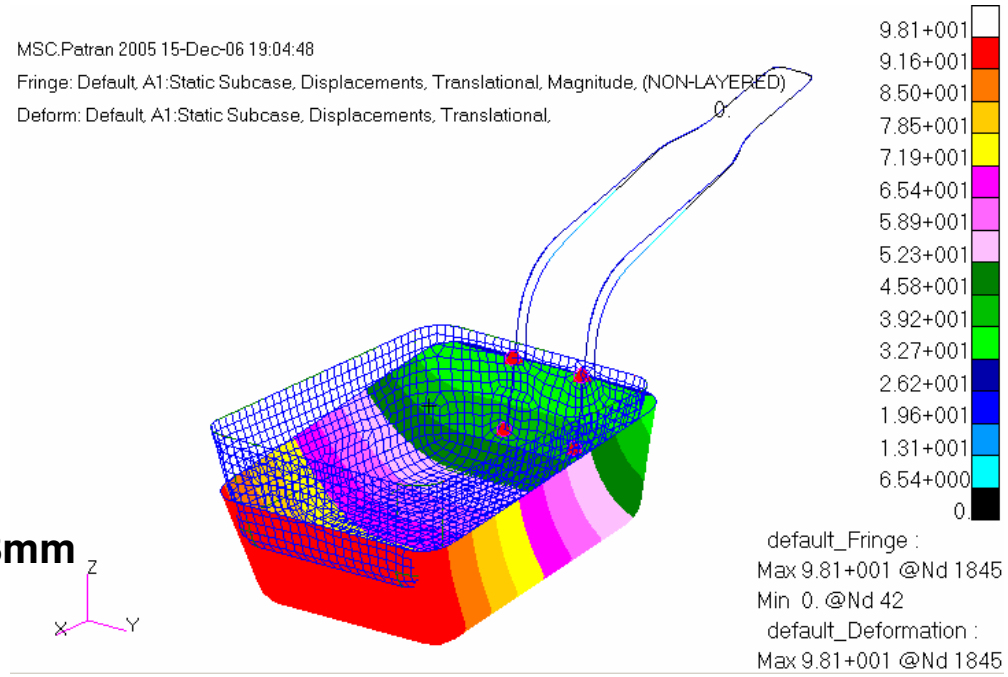
Tutorial 3D

(Con't):-

- Select "Displacement, Translational" for Fringe Result
- Select "Displacement, Translational" for Deformation Result

- Click Apply





The maximum Displacement = 98mm

Result by CATIA = 97mm

END of Tutorial 3D

By Dickson Sham (ME Dept, HKPU)

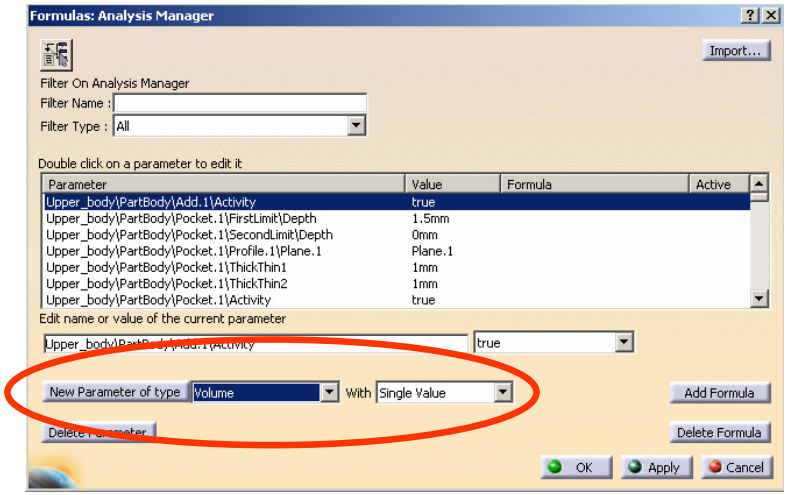
Tutorial 3E

We know we can get a stronger metal arm by shortening its length and/or increasing its diameter. But what are their best values so that we can use the minimal material to support the load? (Less Volume Less Cost)

***** File/Open… Analysis_a.CATProduct*****

To Define a User Parameter "Volume":-

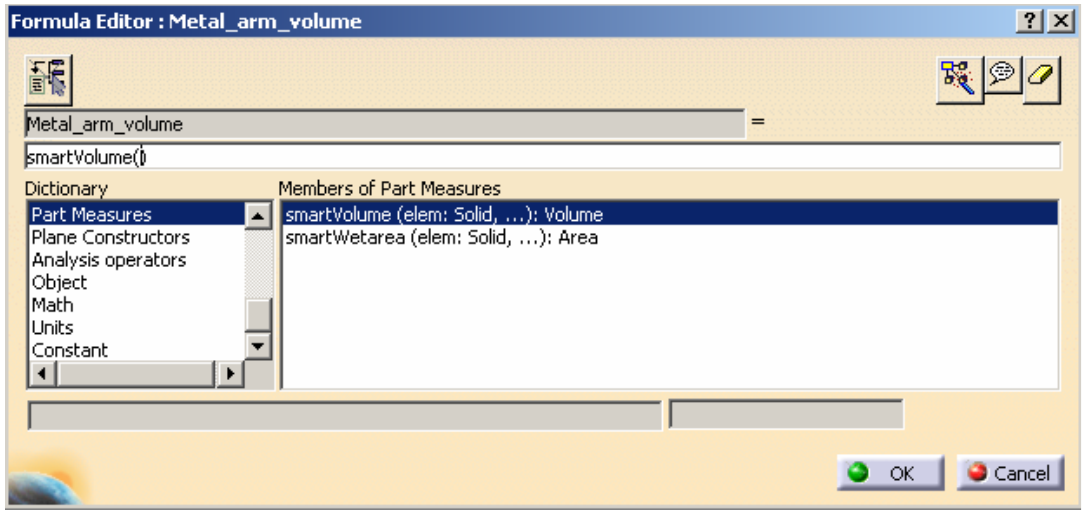
- Click "**Formula**" icon
- Select "Volume" as Type
- Select "Single Value"
- Click "New Parameter of Type"
- Rename "Volume.1" to "Metal_arm_volume"
- Click "Add Formula"



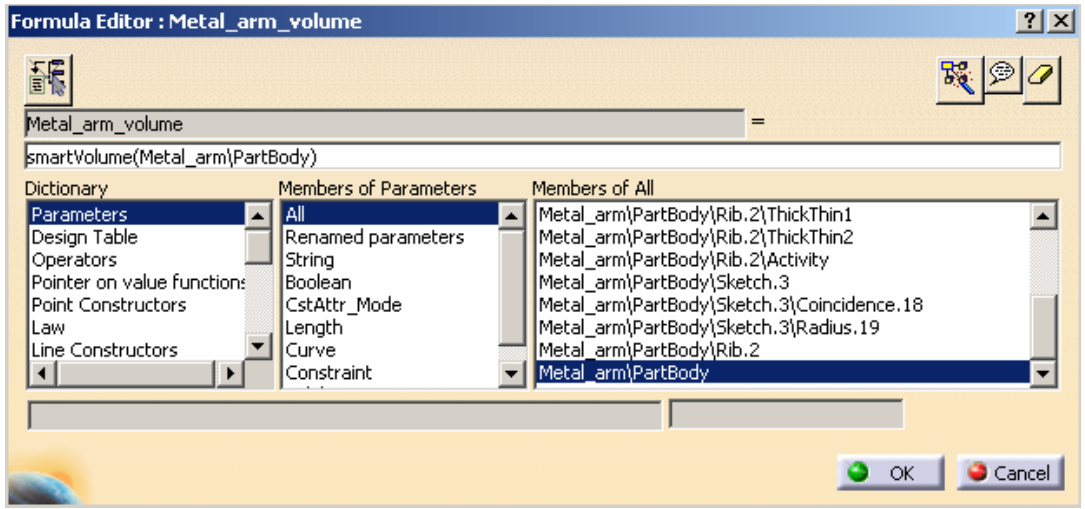
Tutorial 3E

(Con't):-

- Select "Part Measures" on the list of "Dictionary"
- **Double Click** "smartvolume(elem:solid,…)" under the list of "Members of Part Measures"
- (now Metal_arm_volume=smartvolume())
- Then click the space between two blankets



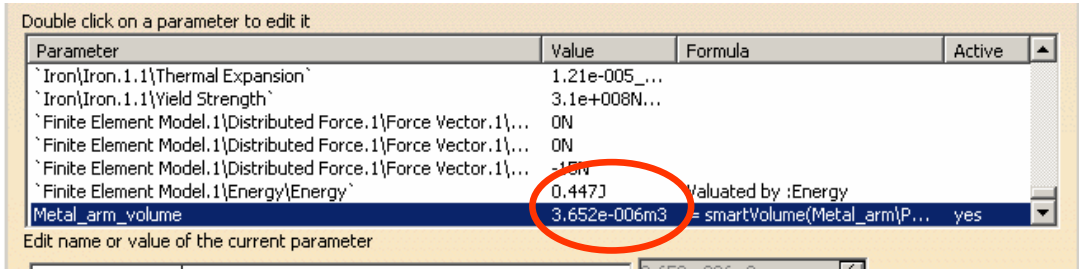
- Maximize the product tree and maximize the part tree of Metal Arm
- Click "Partbody" under *Metal_arm* once
- **Double-click** "Metal_arm/Partbody" on the list of "Member of all"
- Click ok to complete



Tutorial 3E

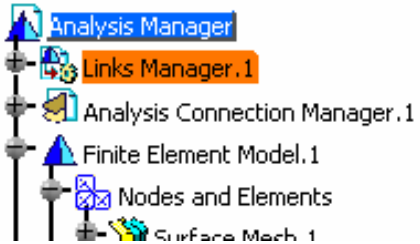
(Con't):-

- (The system measured the volume of "Metal_arm" and return the value as 3.652e-006m^3)
- Click ok to complete



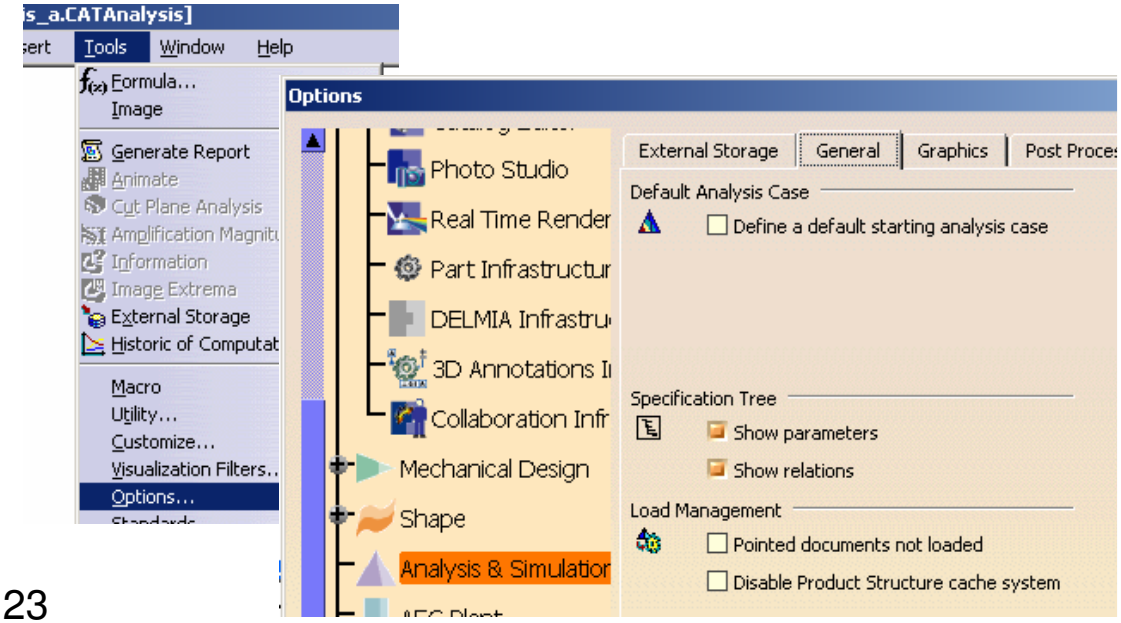
To minimize the tree:-

- Click "+" next to "Link Manger.1"



To Display the User-defined Parameter "Volume" on the tree:-

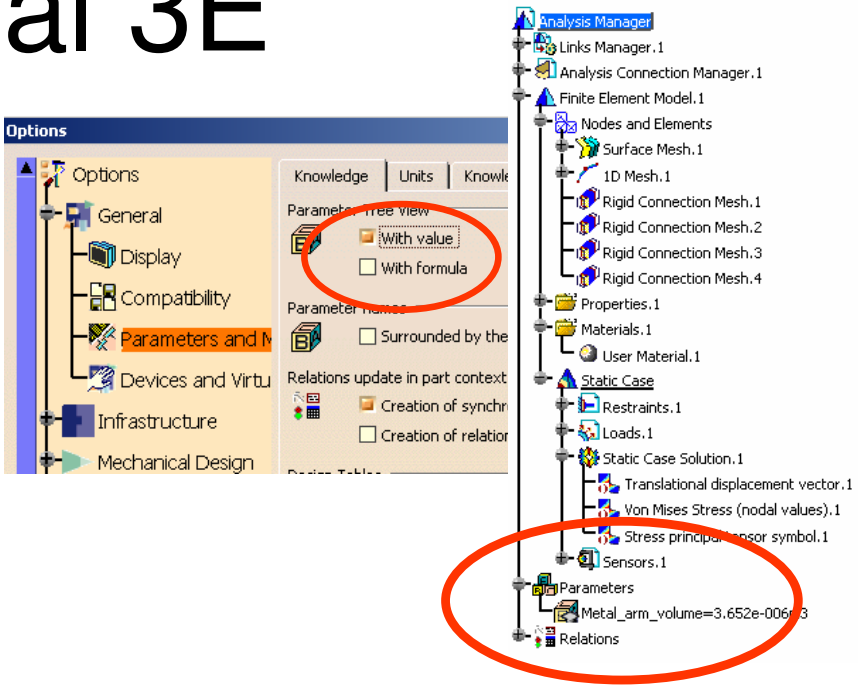
- Select Tools/Options… on the menu bar
- Select "Analysis & Simulation" on the left list
- Select the tab page "General"
- Select "Show parameters"
- Select "Show relations"



Tutorial 3E

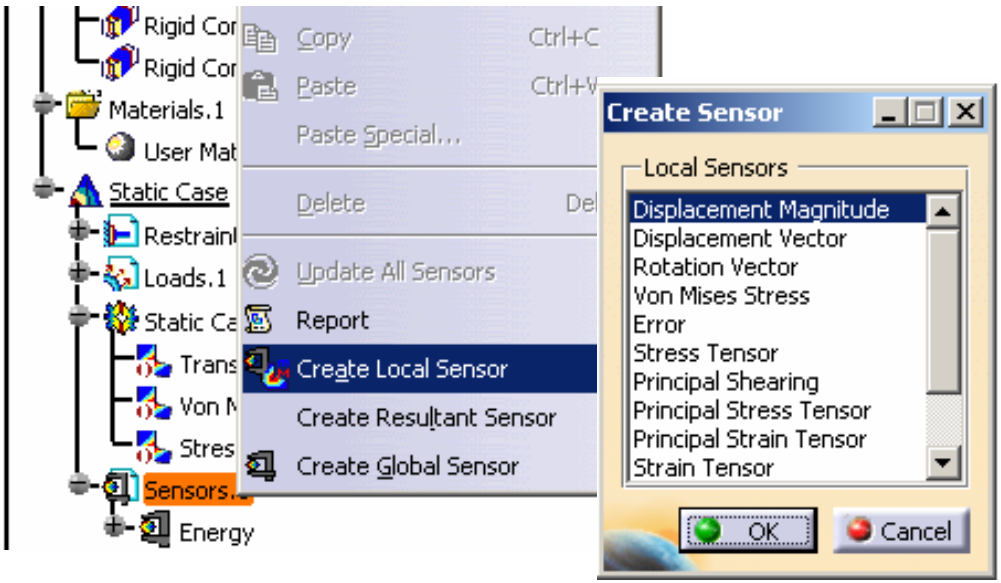
(Con't):-

- Select "Parameters and Measures" on the left list
- Select the tab page "Knowledge"
- Select "With Value"
- Click ok to complete (Now we can see the user-defined parameter "volume" with its value on the tree)



To Create a measurement senor:-

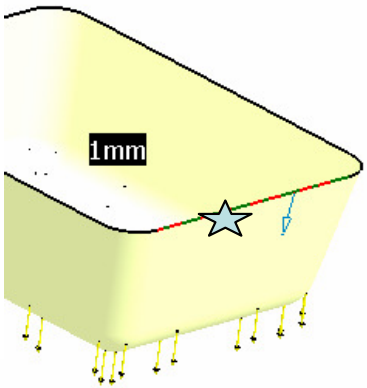
- Right-click "**Sensors**" on the tree
- Select "Create Local Sensor"
- Select "**Displacement Magnitude**" on the list
- Click ok to complete

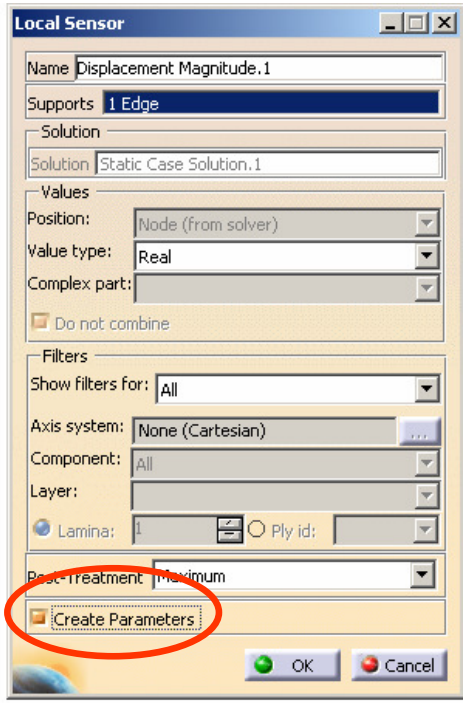


Tutorial 3E

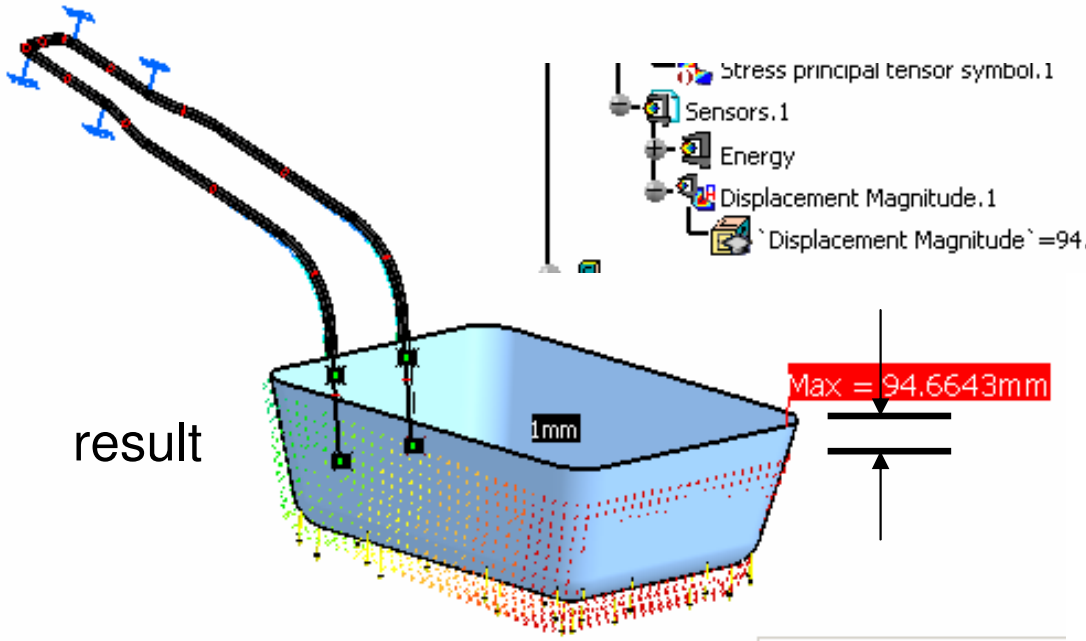
(Cont'):-

- Double-Click "Displacement Magnitude.1" on the tree
- Select the edge ☆ as Supports (Test point)
- Select "Maximum" as Post-Treatment
- Select "Create Parameters"
- Click ok to complete





result



Now we have a sensor to measure the maximum displacement of the basket

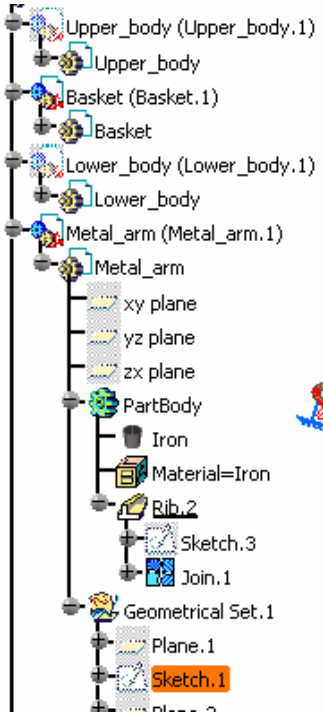
Tutorial 3E

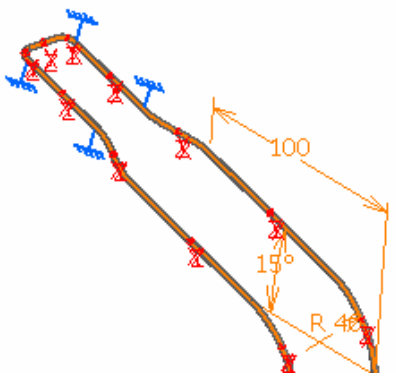
To Create a case of Optimization:-

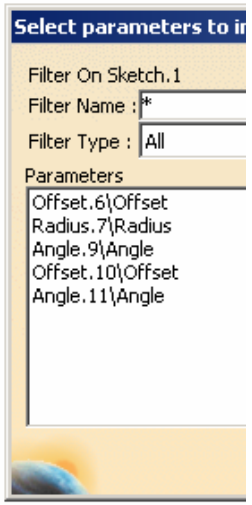
- Select “Start/Knowledgeware/Product Engineering Optimizer” on the menu bar

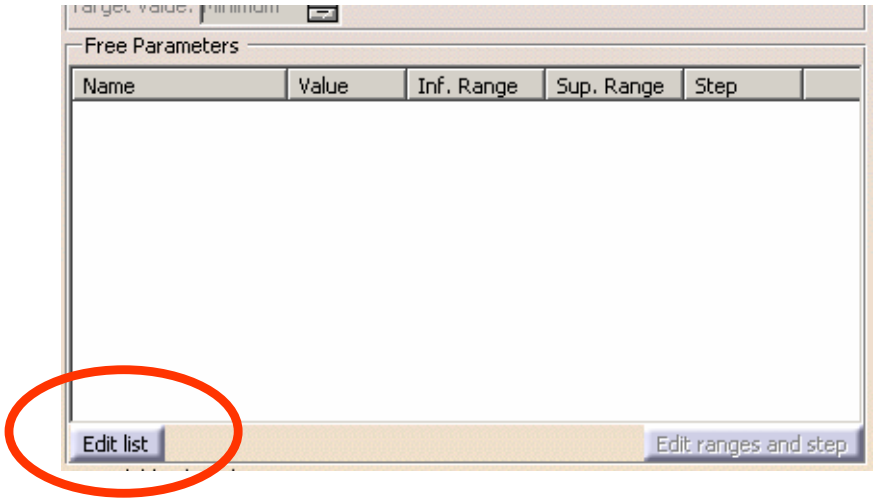
- Click “**Optimization**” icon

- (1) To define Free Parameters:-
  - (For this case, we choose
    - Diameter of the metal wire
    - Length of a portion of the metal arm)

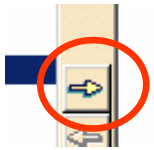






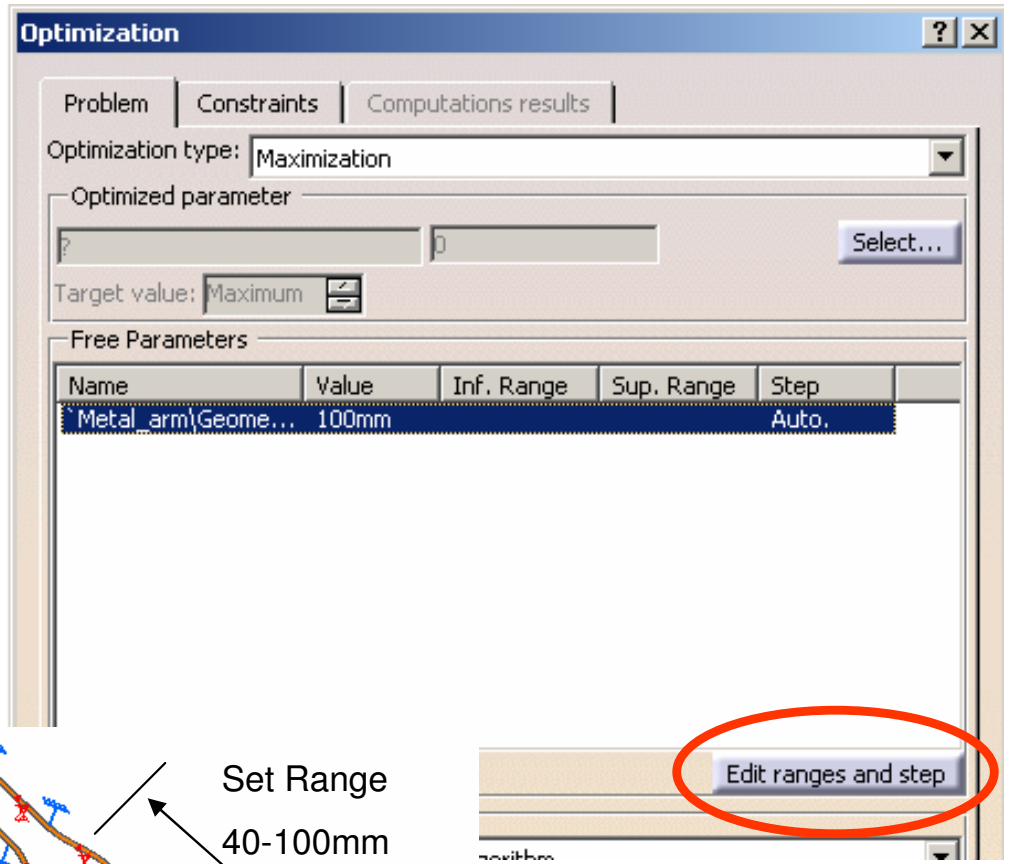


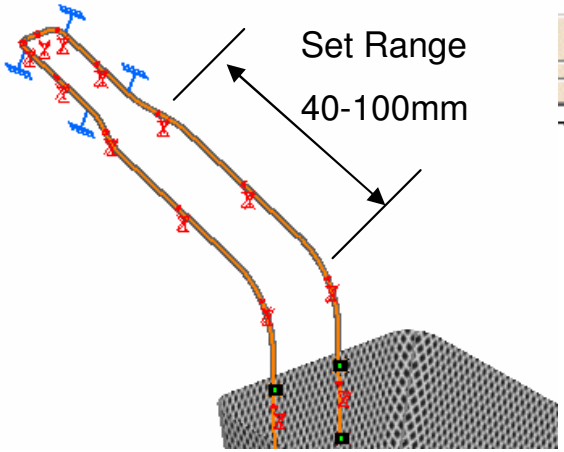
  - Click “**Edit List**” icon
  - Maximize the tree so that we can see the Geometrical set.1 of Metal Arm
  - Click “**Sketch.1**” once (only related parameters are shown on the list)
  - **Click on the dimension”100”** or click “Offset.10” on the list (They are the same)
  - Click the arrow icon
  - Click ok to complete

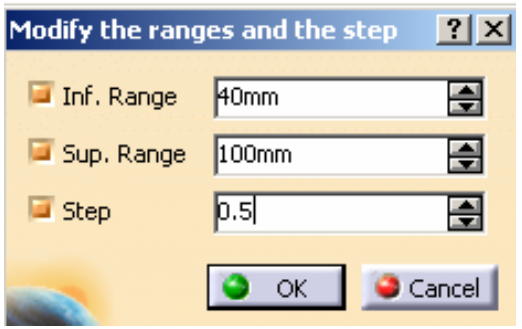


Tutorial 3E

- (Con't) :-
  - (Now a parameter is added on the list of Free Parameters)
  - Single Click on this parameter
  - Click "Edit ranges and step"
  - Select the box next to Inf. Range
  - Enter 40mm as Inf. Range (lower limit)
  - Select the box next to Sup. Range
  - Enter 100mm as Sup. Range (upper limit)
  - Select the box next to Step
  - Enter 0.5mm as Step
  - Click ok to complete

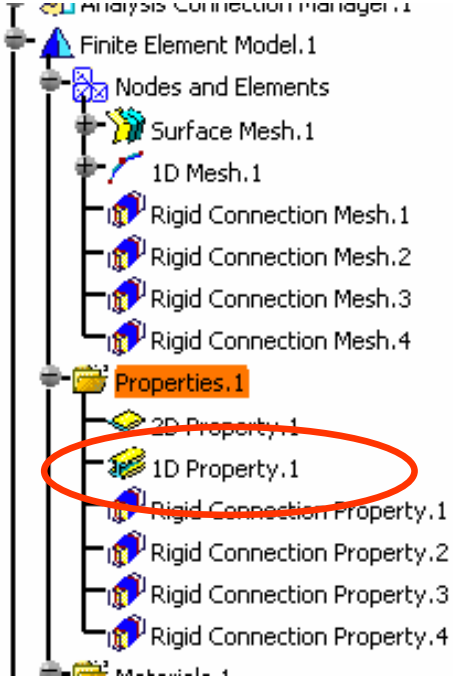


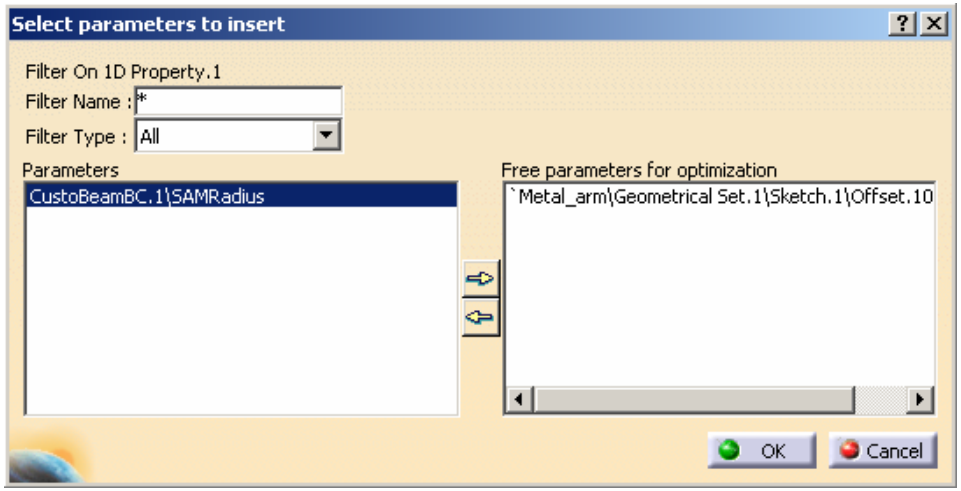




Tutorial 3E

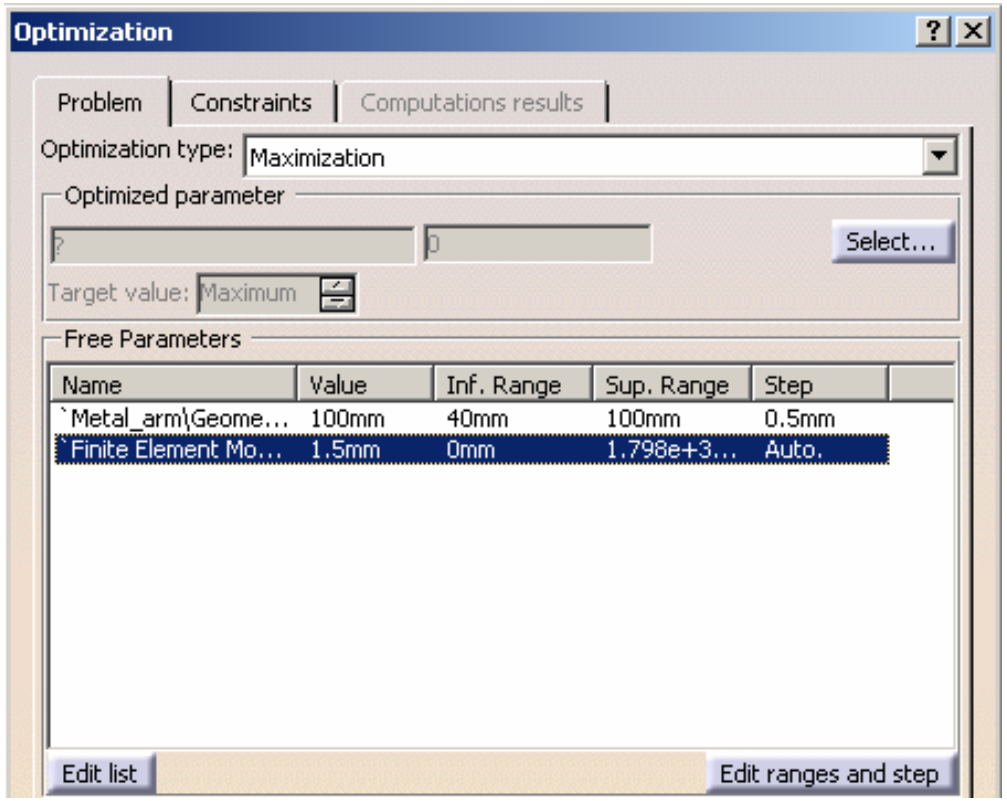
- (Con't):-
  - Click "+" next to "Link Manger.1" to minimize the tree
  - Click "**Edit List**" icon again
  - Maximize the tree so that we can see the Properties.1 of Finite Element Model.1
  - Click "1D Property.1" once (only related parameters are shown on the list)
  - Click the one and the only parameter on the left list, which is the radius of the 1D mesh
  - Click the arrow icon
  - Click ok to complete





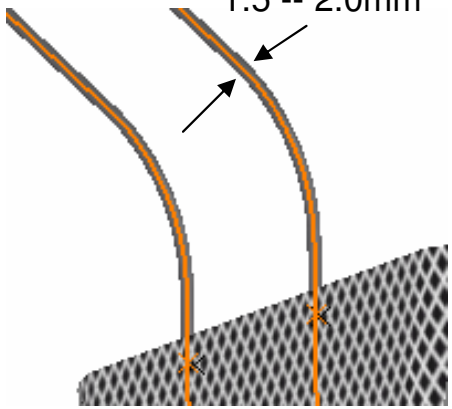
Tutorial 3E

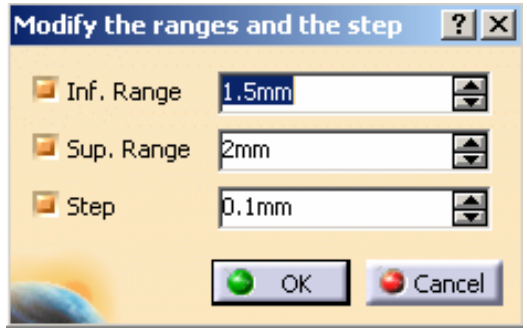
- (Con't) :-
  - (Now the 2nd parameter is added on the list of Free Parameters)
  - Single Click on this parameter
  - Click "Edit ranges and step"
  - Select the box next to Inf. Range
  - Enter 1.5mm as Inf. Range (lower limit)
  - Select the box next to Sup. Range
  - Enter 2mm as Sup. Range (upper limit)
  - Select the box next to Step
  - Enter 0.1mm as Step
  - Click ok to complete



Set Range

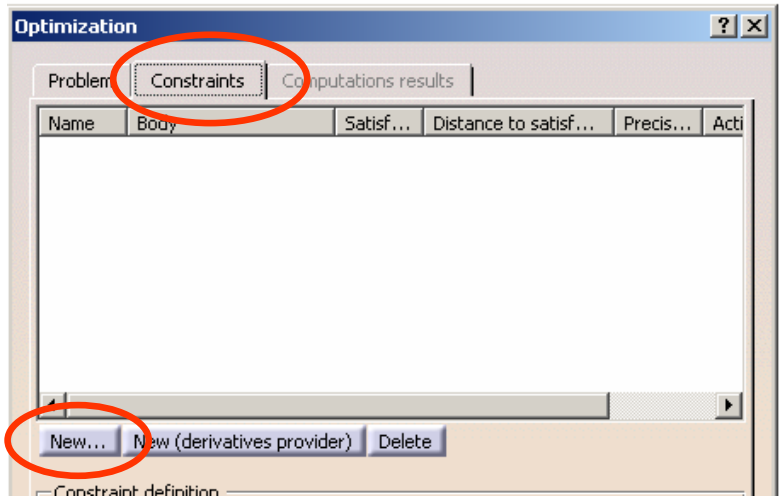
1.5 -- 2.0mm

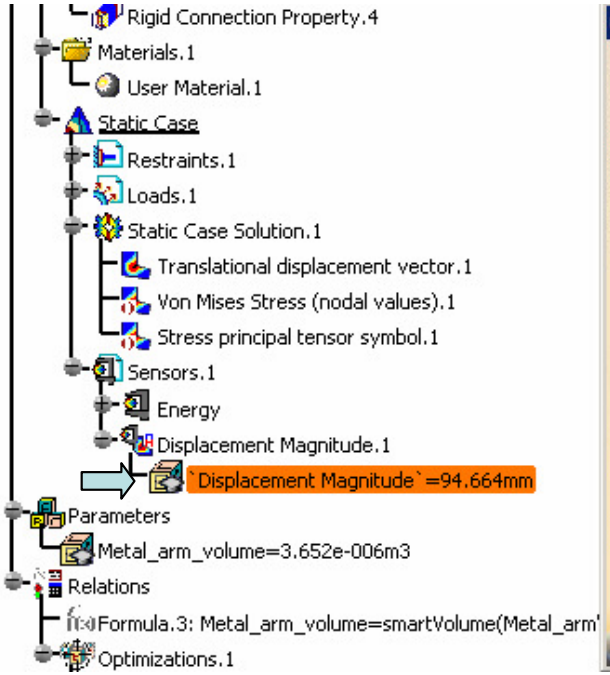


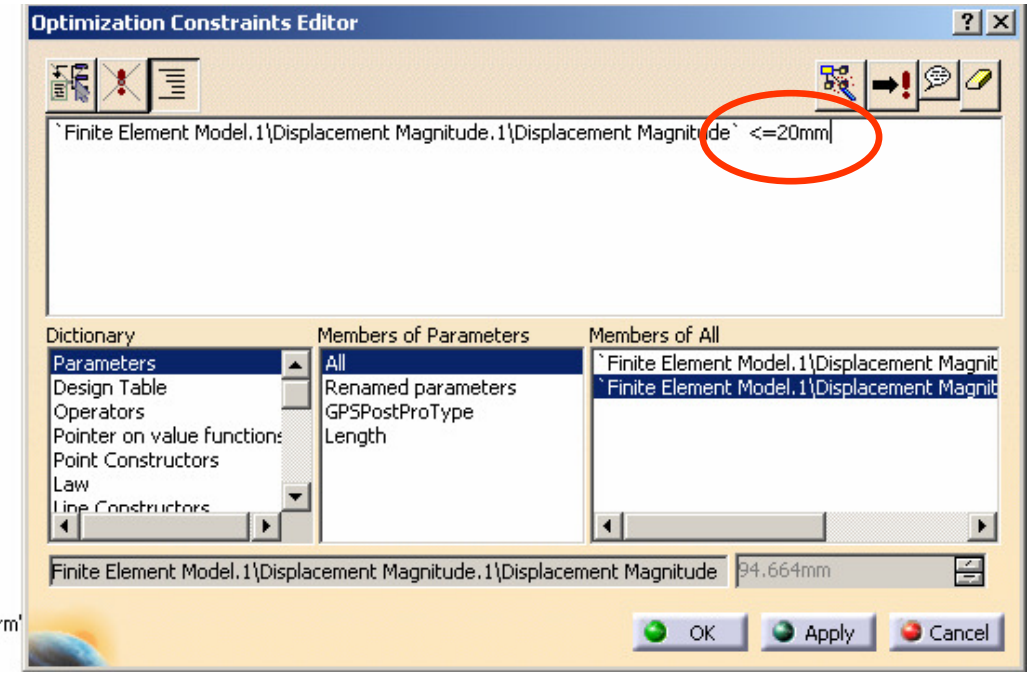


Tutorial 3E

- (2) To define Constraints:-
  - Select the tab page "**Constraint**"
  - Click "New…" icon
  - Select "Displacement Magnitude" under Sensor.1 on the tree
  - Then add "<=20mm" after the words
  - Click ok to complete







Tutorial 3E

- (3) Define computation method:-
  - Select the tab page "**Problem**"
  - Select " **Only Constraints**" as Optimization Type
  - Select "Simulated Annealing Algorithm" as Algorithm type"
  - Set Termination Criteria:

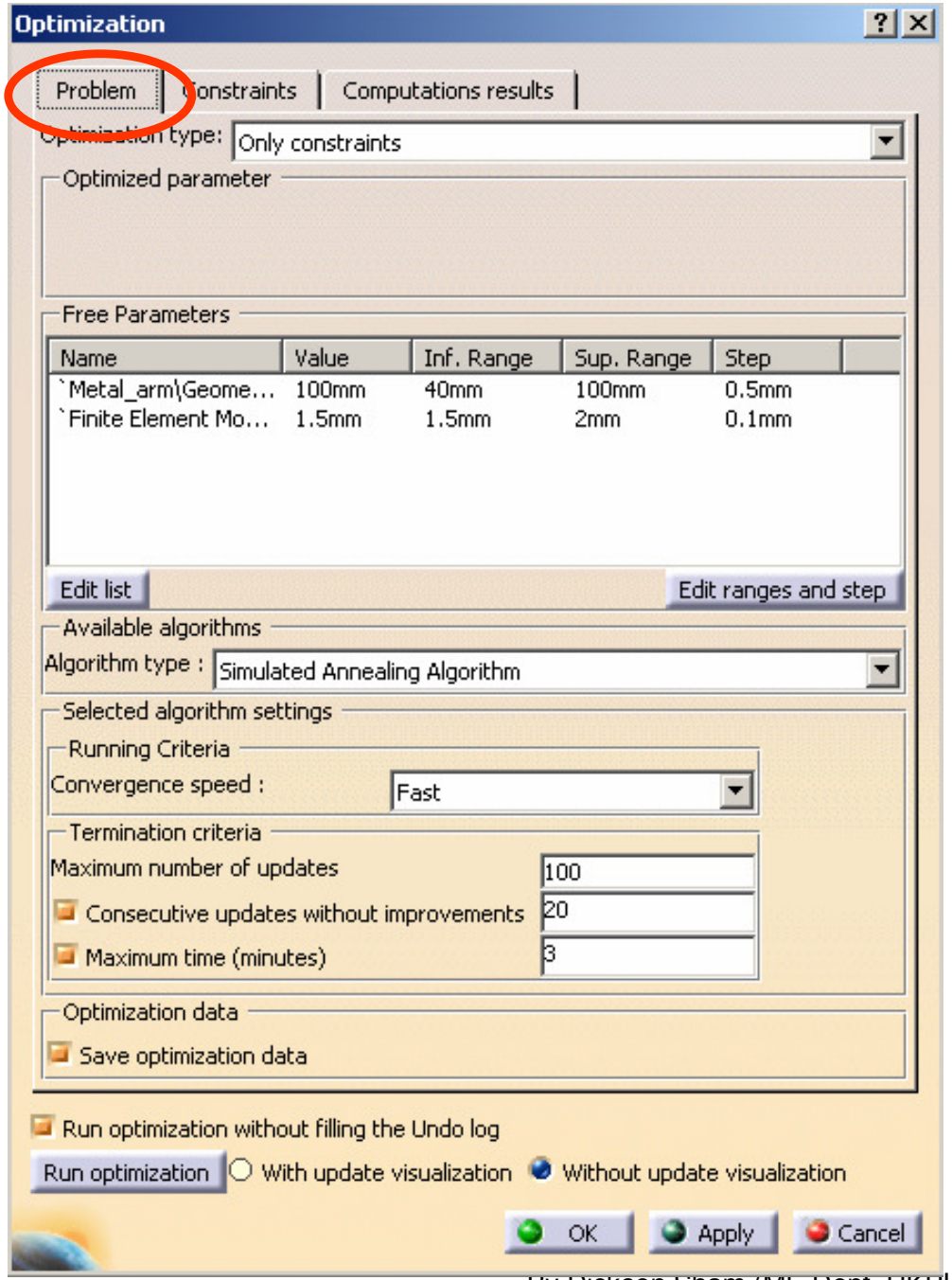
a. Enter 100 as Maximum number of updates

b. Enter 20 as Consecutive updates without improvements

c. Enter 3 as Maximum Time (minutes)

(If any of these is fulfilled, the computation will stop)

  - Select "Save Optimization data" (so that we can see all raw data after computation)



Tutorial 3E

To RUN Optimization:-

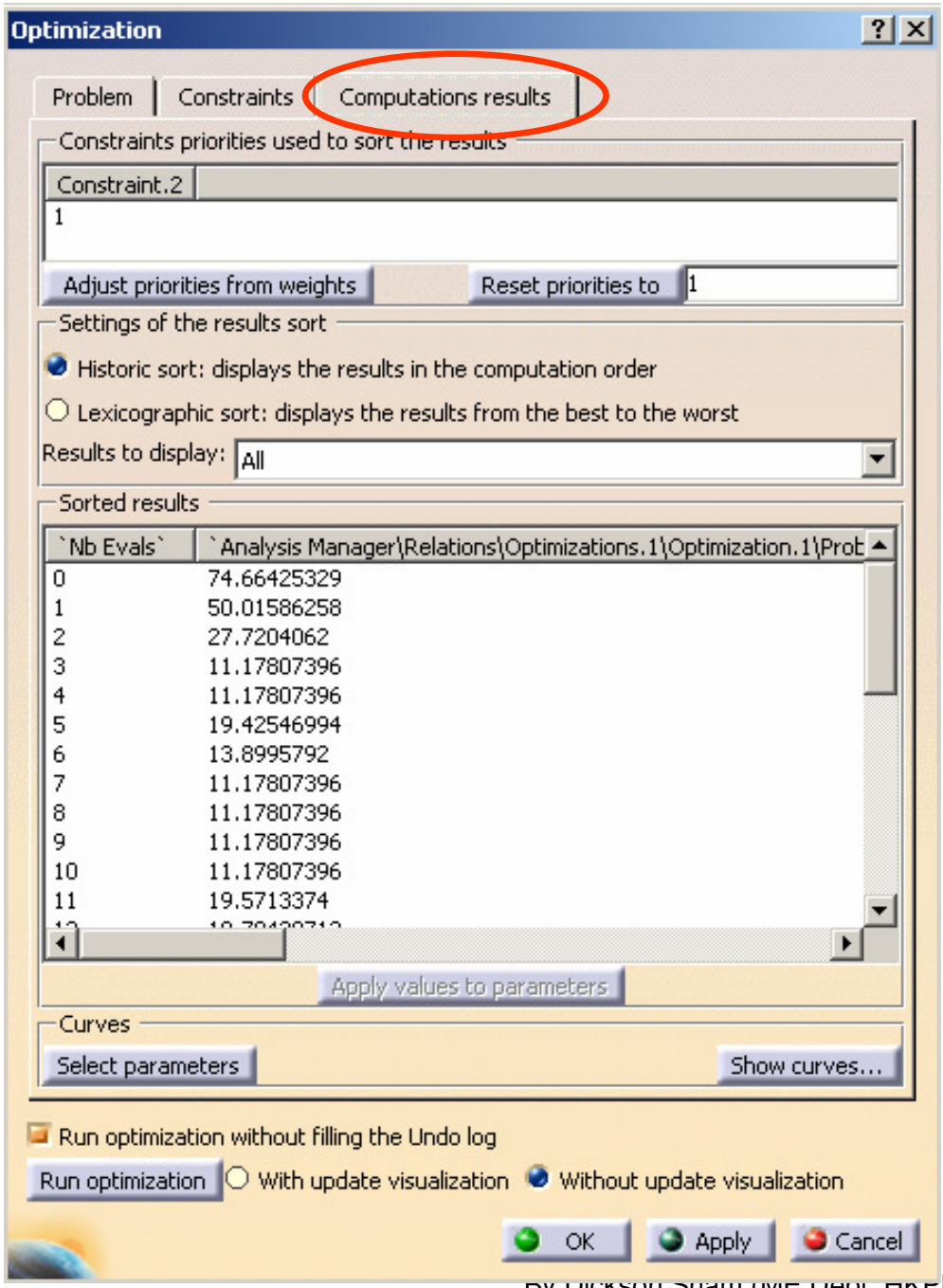
- Click "**Run Optimization**" icon
- Enter "Data" as file name (all the raw data will be stored in this excel file)
- Click "Save" to start computation

(The computer starts to search all possible values of the two free parameters so that the constraint can be met)

(After 3 minutes)

To review the results:-

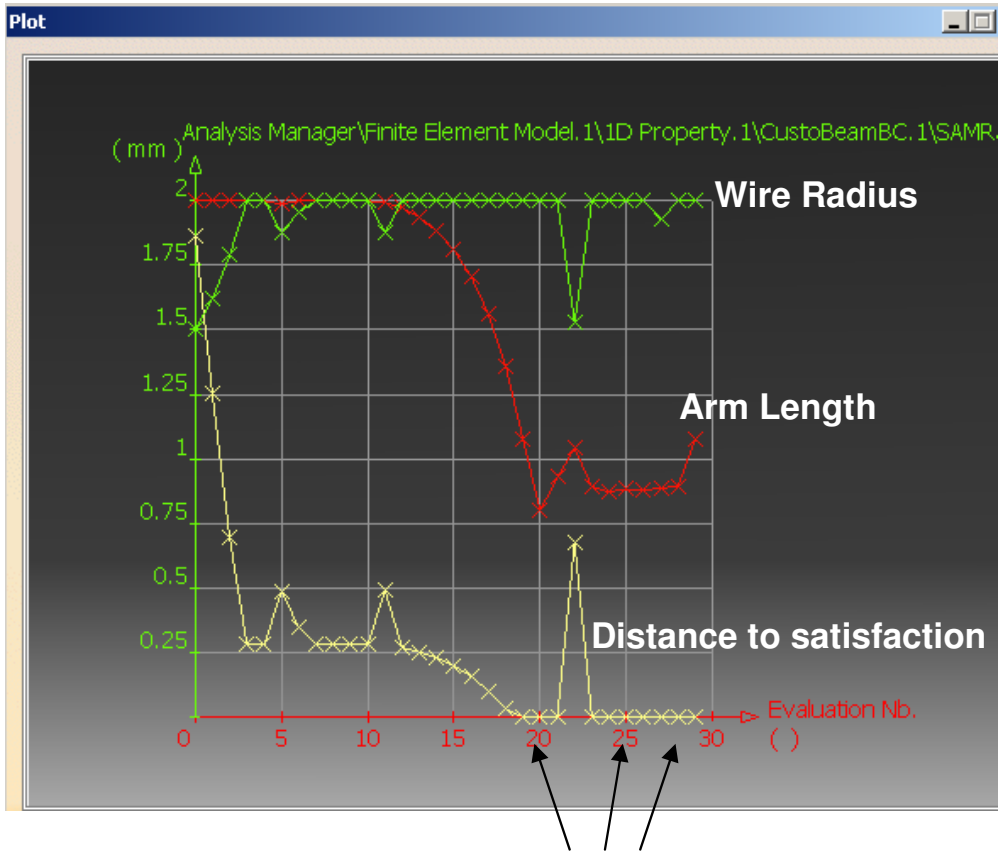
- Select the tab page "**Computation Results**"
- On the list, all attempts failed until #19 (i.e. Result – Target = 0)



Tutorial 3E

(Cont'):-

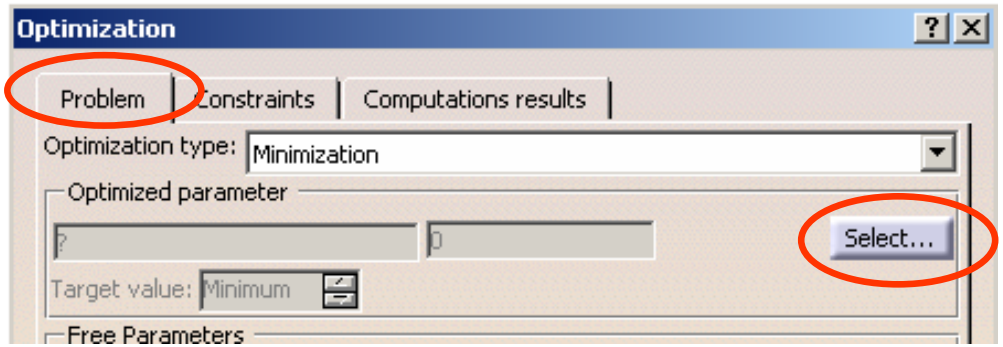
- Select "Show Curve" icon
- On the list, the four attempts failed but the rest can meet the requirement
  (i.e. Result – Target = 0)



Only a few can meet the requirement. But which one can give the minimum volume?

To Further optimize the parameters (for the minimum volume of metal arm):-

- Select the tab page "**Problem**" again
- Select "**Minimization**" as Optimization type
- Click "Select" icon



Tutorial 3E

(Cont') :-

- Select "Metal_arm_volume" on the list
- Click ok to complete

- Select "**Gradient Algorithm with Constraints**" as Algorithm type"

- Set Termination Criteria:

a. Enter 100 as Maximum number of updates

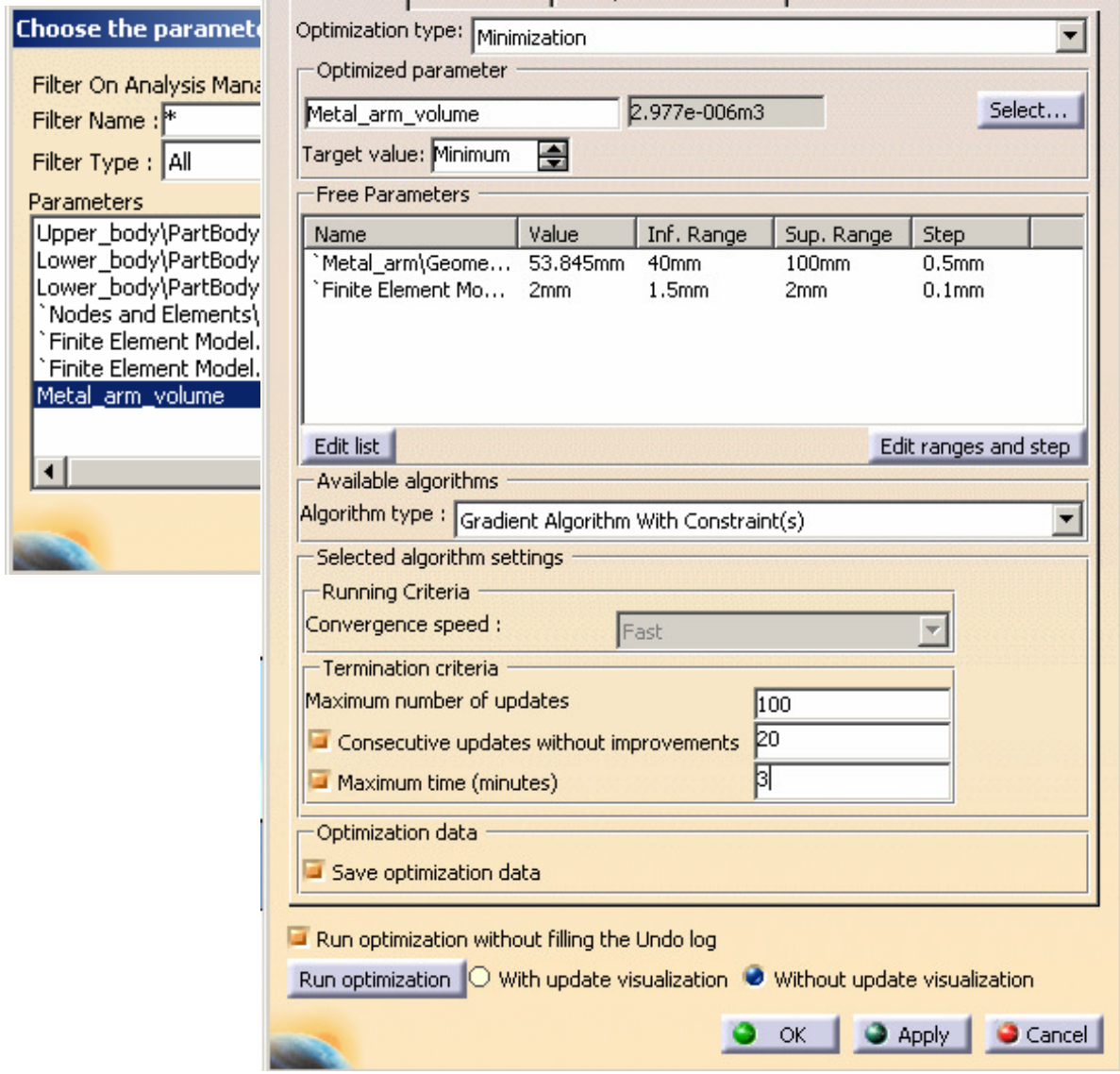
b. Enter 20 as Consecutive updates without improvements

c. Enter 3 as Maximum Time (minutes)

(If any of these is fulfilled, the computation will stop)

To RUN Optimization again:-

- Click "Run Optimization" icon
- Select Yes to overwrite the data file



Tutorial 3E

To review the result:-

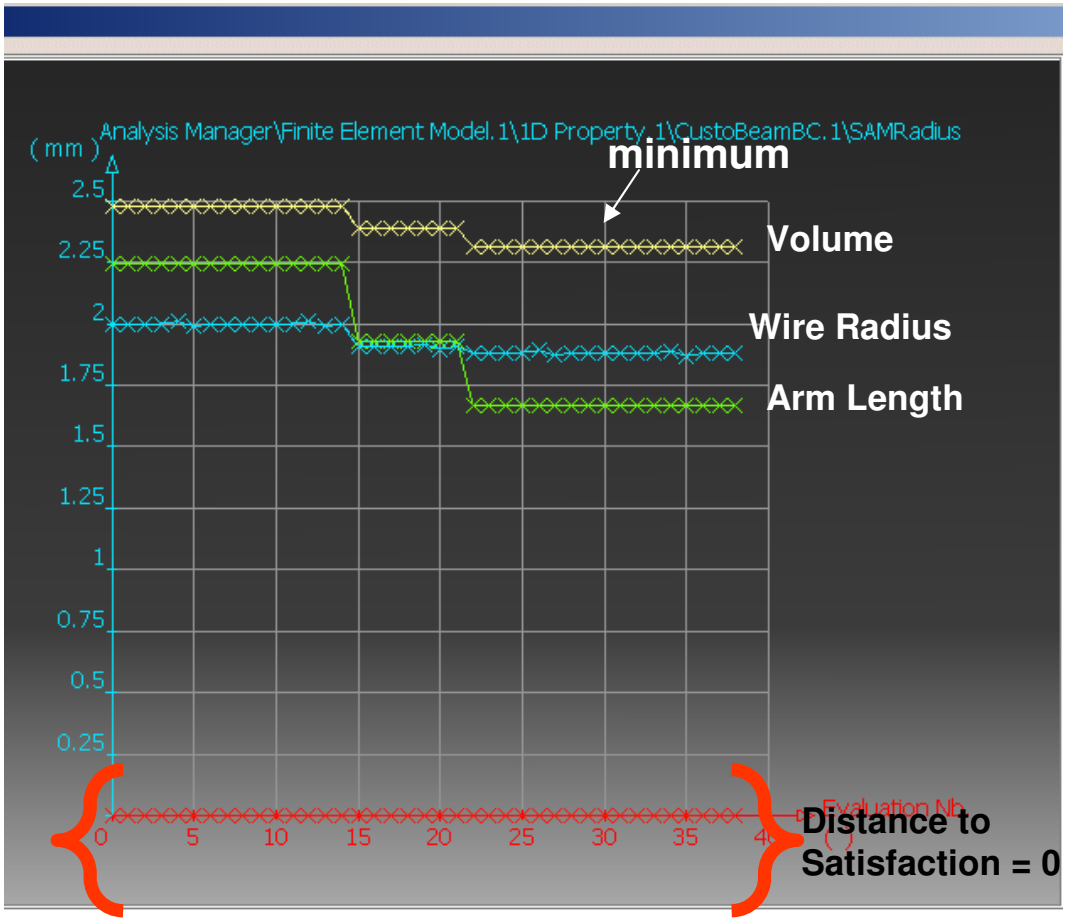
- Select the tab page "**Computation Results**"
- Select "Show Curve" icon

From the curve, the best values are:

Wire Radius = 1.9mm

Arm Length = 40mm

Its volume is the smallest but it is still so strong that the deflection is lower than the required limit.



In this revised optimization, all cases can meet the requirement (i.e. deflection of basket is smaller than 20mm)

To Save all files:-

- Select "**File/Save all**"

END of Tutorial 3E