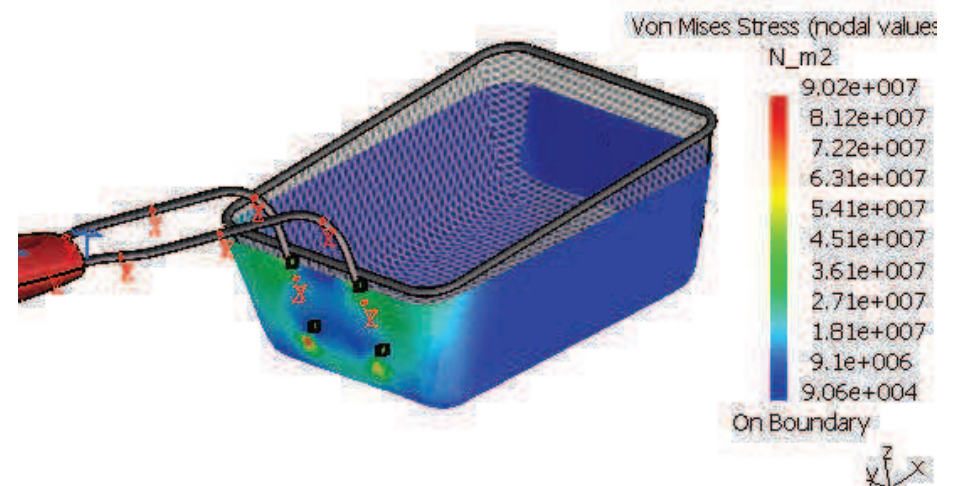
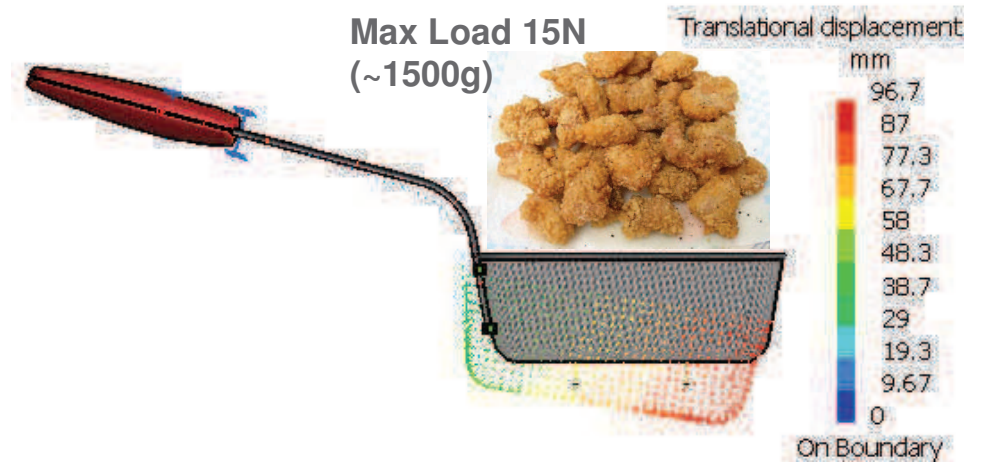


Tutorial 3C

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

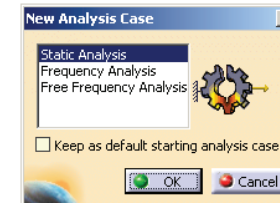
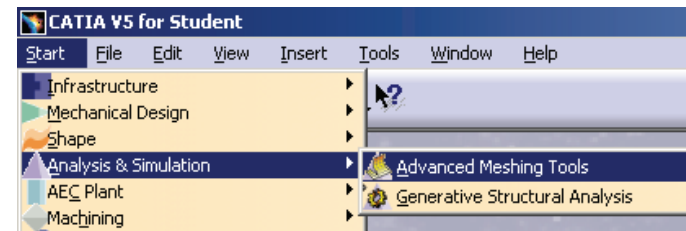


Tutorial 3C

File/Open/ Basket_assm_a.CATProduct

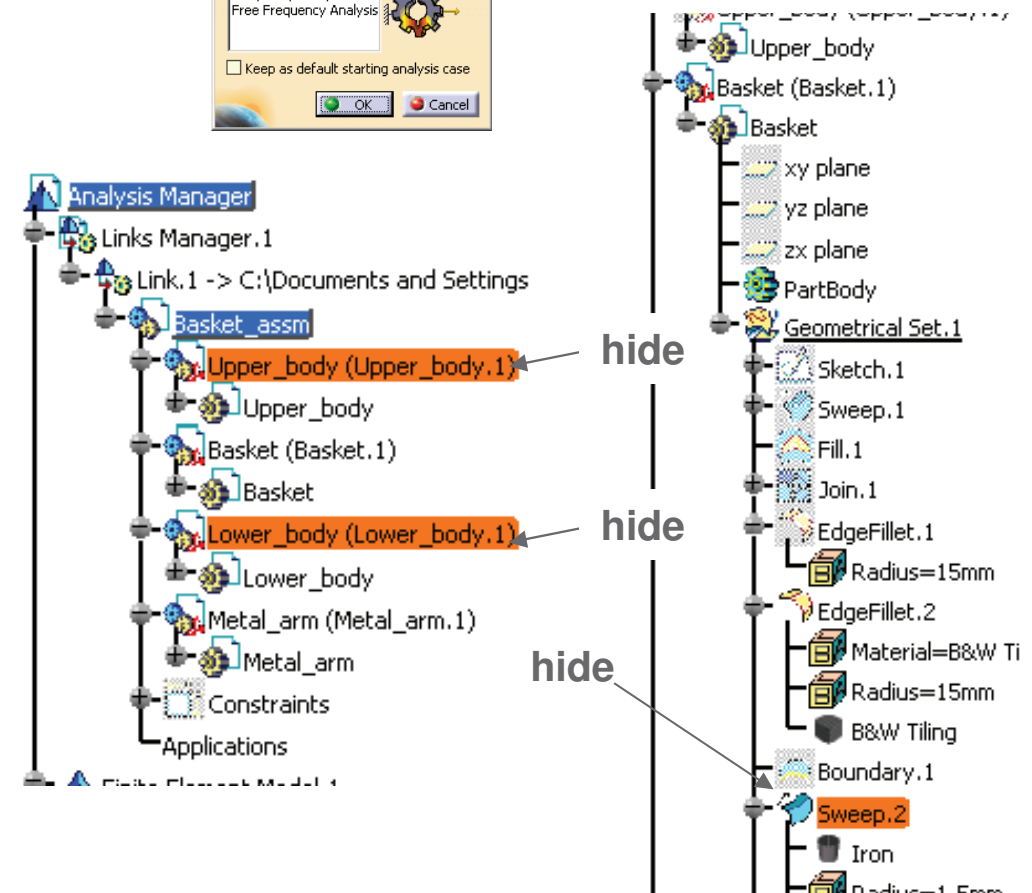
To go to a new Workbench:-

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



(1) To simplify the model for analysis:-

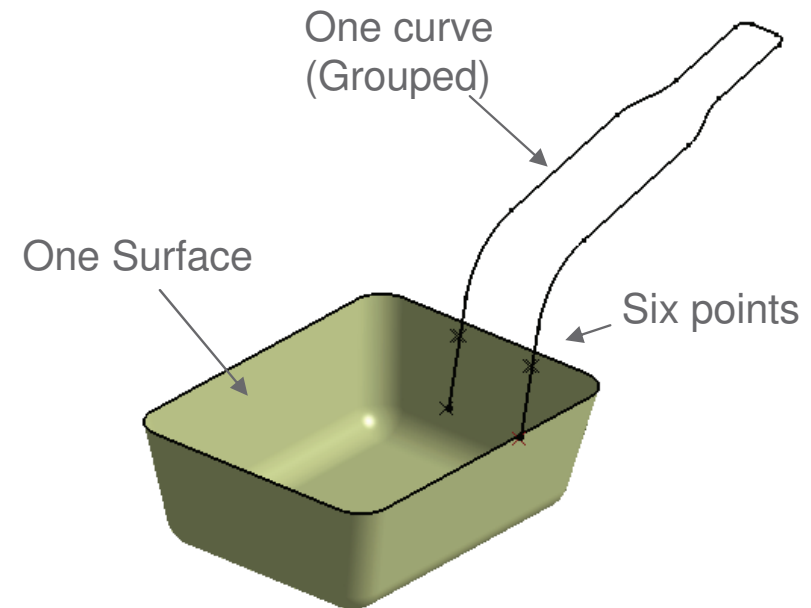
- Click “+” next to “Link Manager.1” on the tree
- Click “+” next to “Link.1” on the tree
- **Hide** *Upper_body.1* & *Lower_body.1*
- **Hide** the surface “Sweep.2” under *Basket.1*
- **Hide** “PartBody” under *Metal_arm.1*
- **Show** the curve “Join.1” under *Metal arm.1*



Tutorial 3C

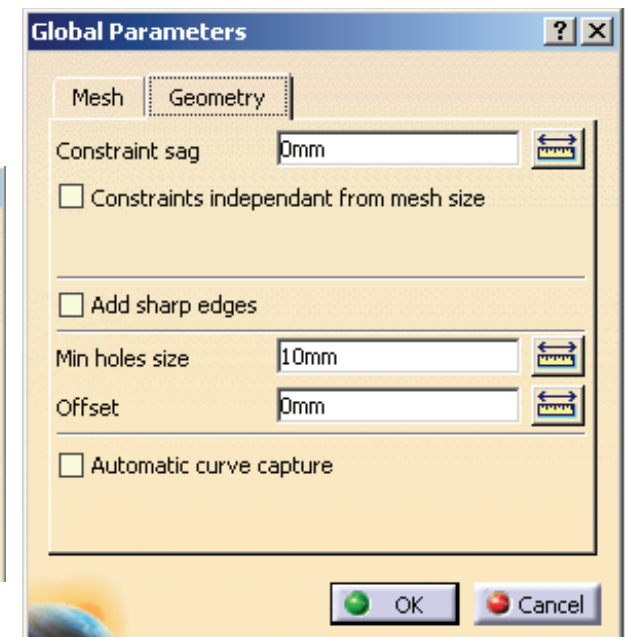
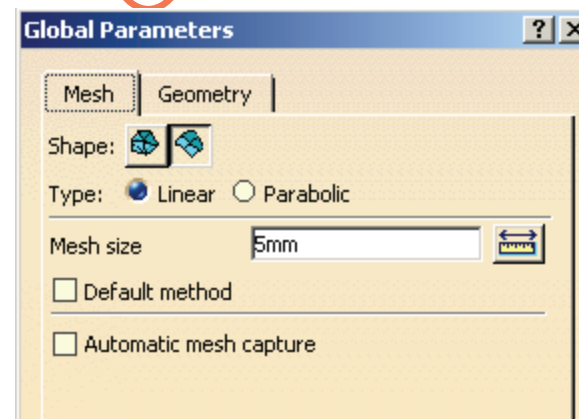
Now, we should see the below elements only:

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



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

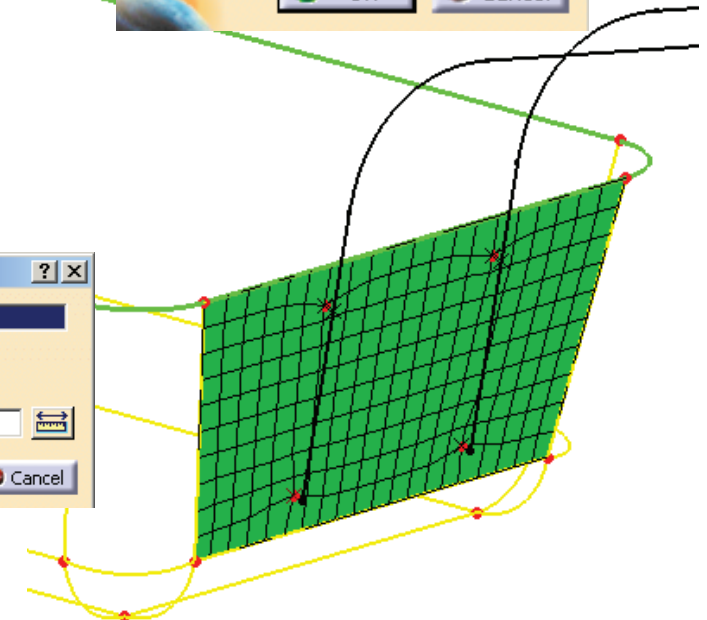
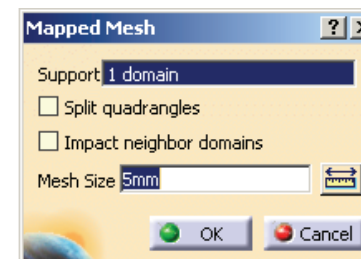
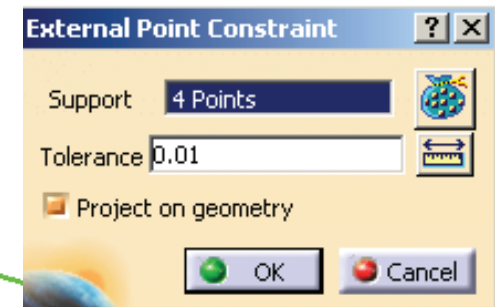
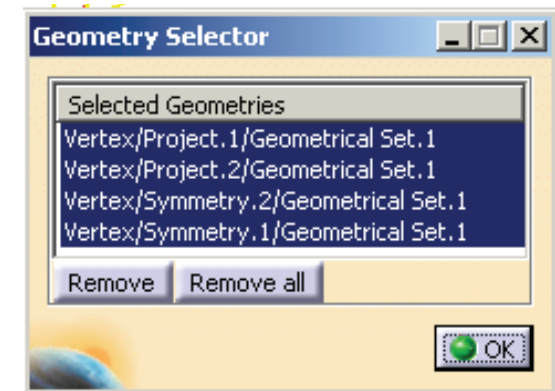
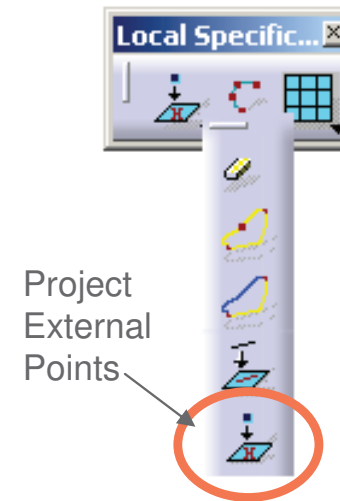
- Click “**Surface Mesher**” icon
- Select the surface
- Select “quadrangle” as Shape
- Select “Linear” as Type
- Enter 5mm as Mesh Size
- Leave all the rest default options unchecked
- Enter 0mm as Constraint sag
- Click ok to complete



Tutorial 3C

(Cont'):-

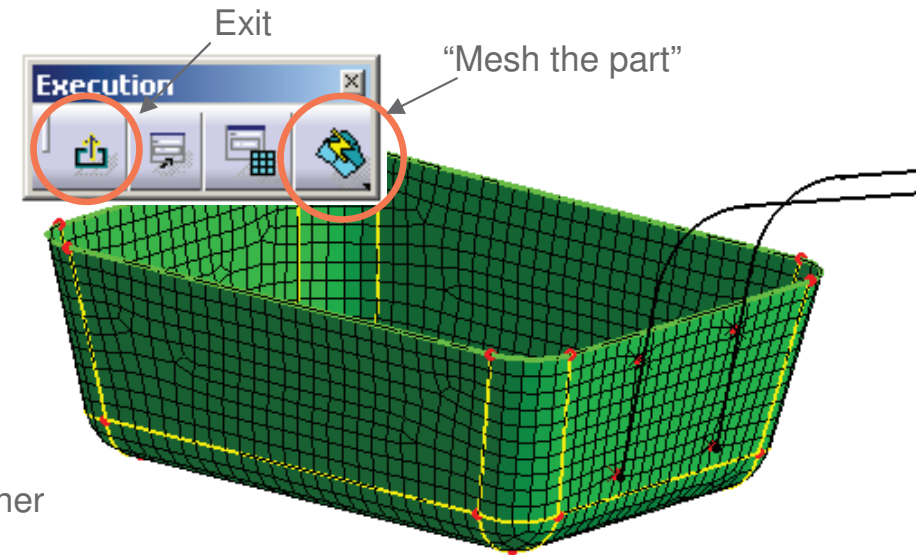
- Click “**Project External Points**” icon (so that nodes are created on the positions of spot welding)
 - Select the four points on the surface
 - If you select a wrong element accidentally, click  and then remove it from the list
 - Enter 0.01mm as Tolerance
 - Select “Project on geometry”
 - Click 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'):-

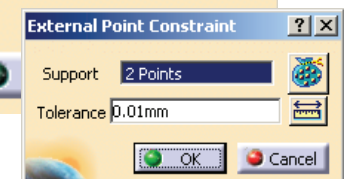
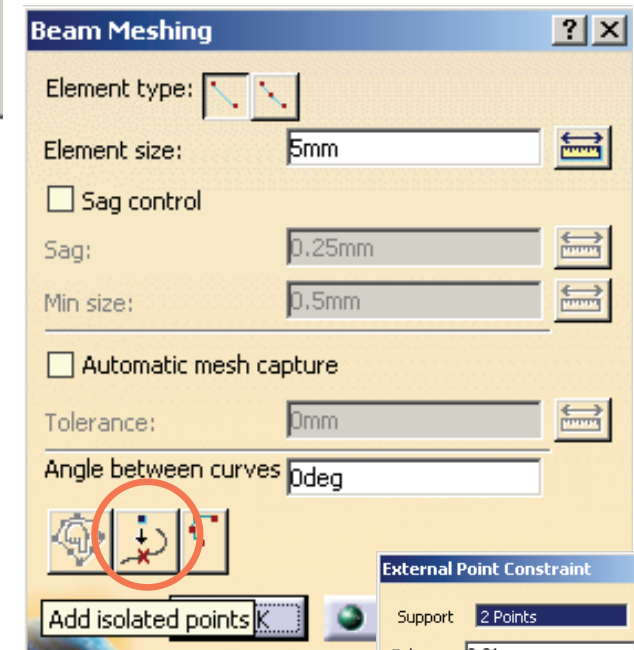
- Click “Mesh the part” icon (The system will mesh the remaining surfaces according to the specification you defined, i.e. linear quadrangle mesh with 5mm size)
- Click “Exit”



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

- 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

Beam Mesher



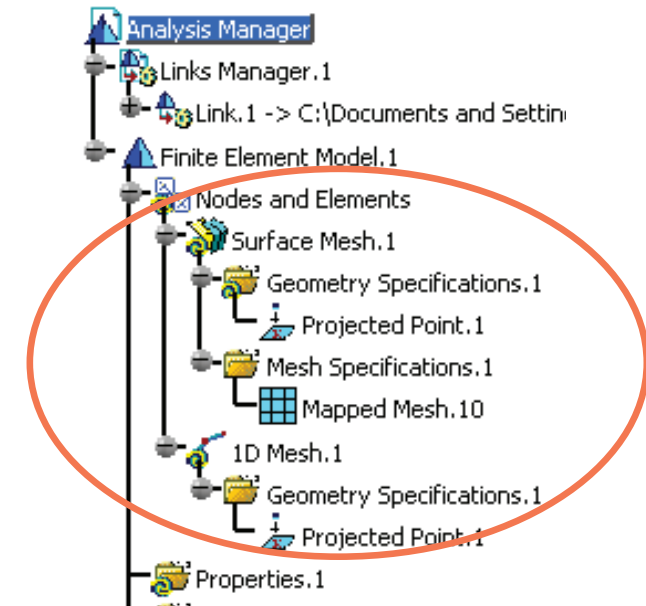
Tutorial 3C

Now, we have already created two meshes:

Surface Mesh.1 (for the basket)

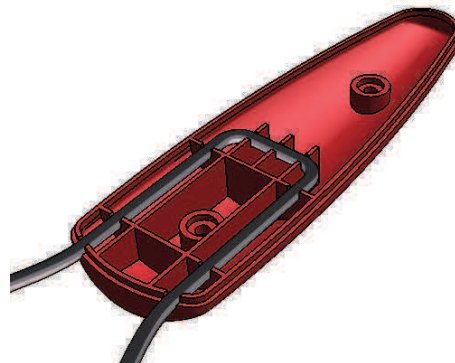
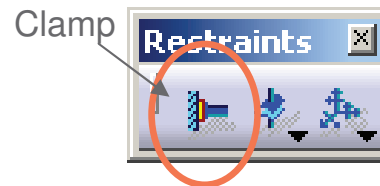
1D Mesh.1 (for the metal arm)

Select “Start /Analysis & Simulation/
Generative Structural Analysis”

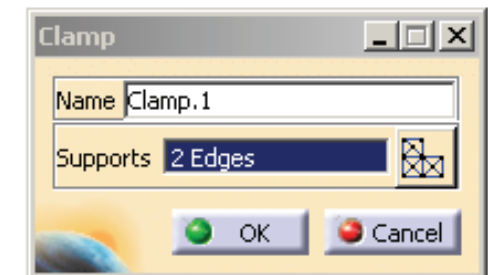
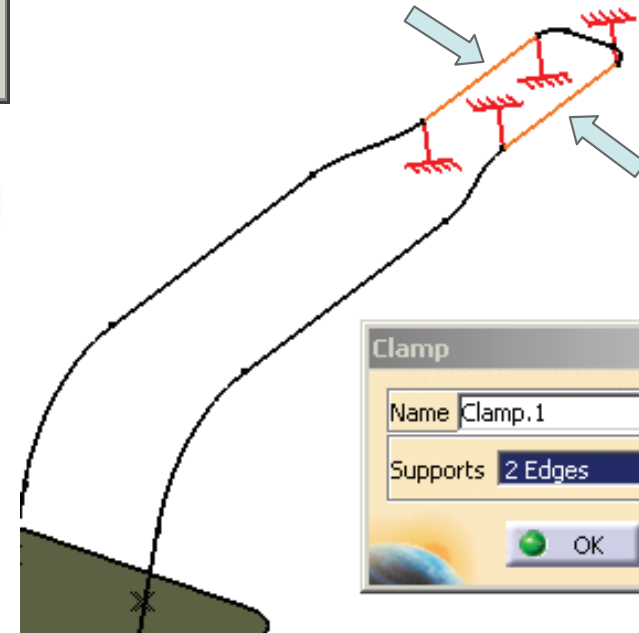


(3) To Create a constraint:-

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



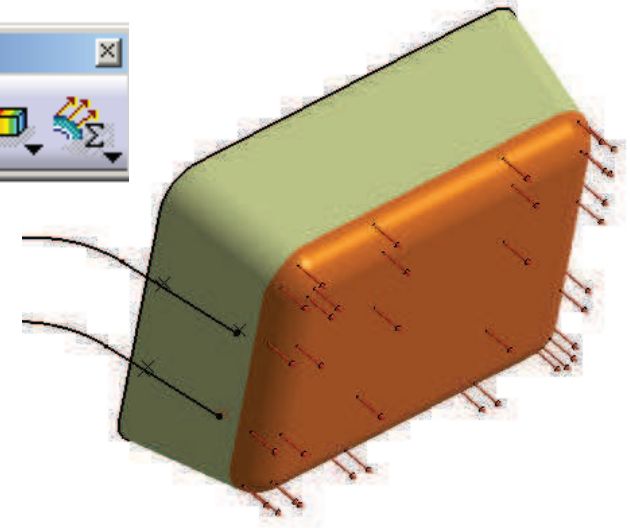
A- 86



Tutorial 3C

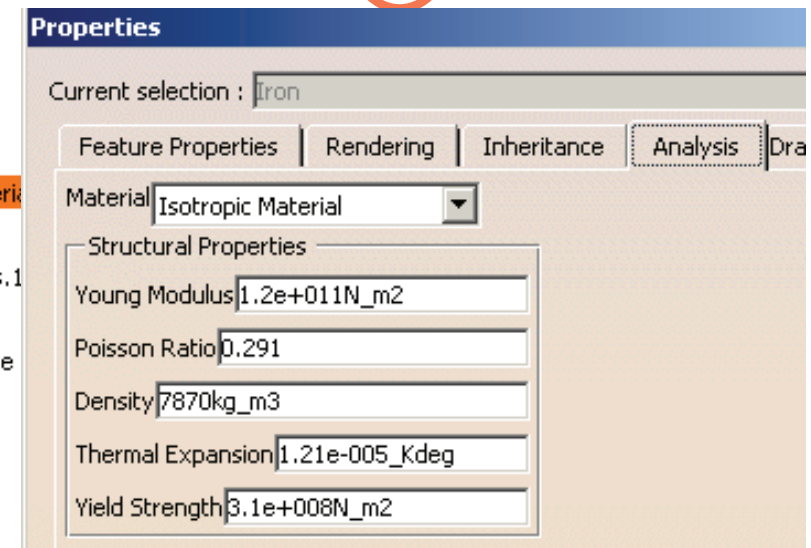
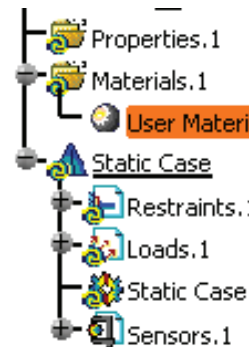
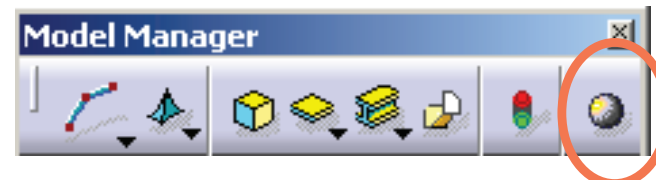
(4) To Create a Force:-

- 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:-

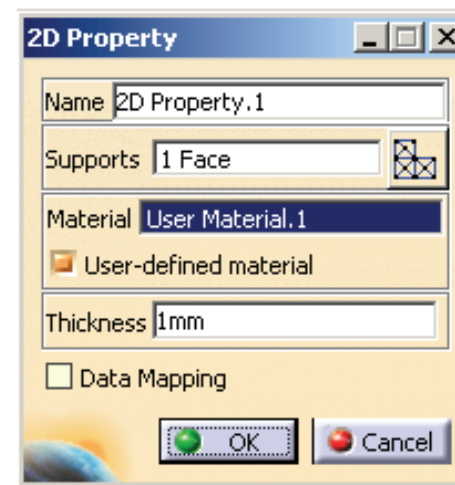
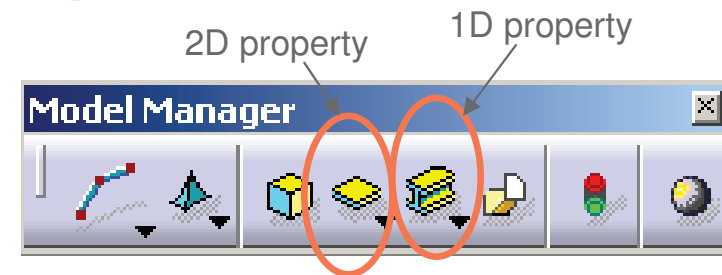
- Click “User Material” icon
- Select “Iron” from the catalog
- Click ok to complete (it will be created on the tree)
- Double-click it on the tree to view its properties
- Leave everything unchanged
- Click ok to quit



Tutorial 3C

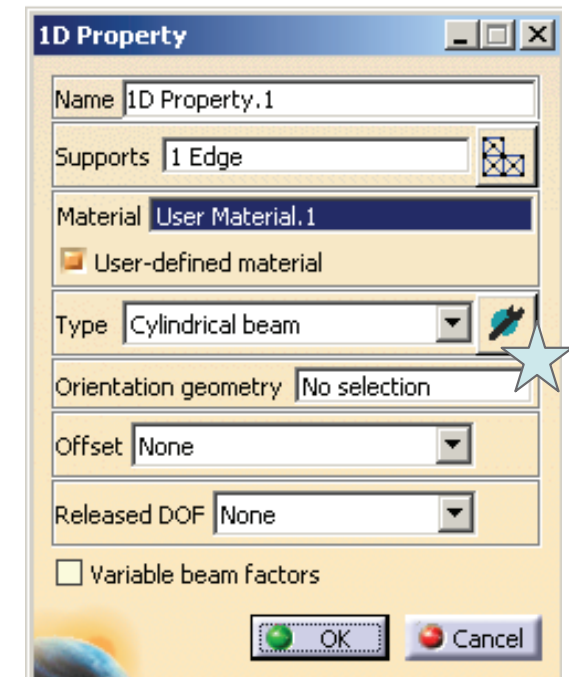
(6a) To define Properties of Surface Mesh.1:-

- 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:-

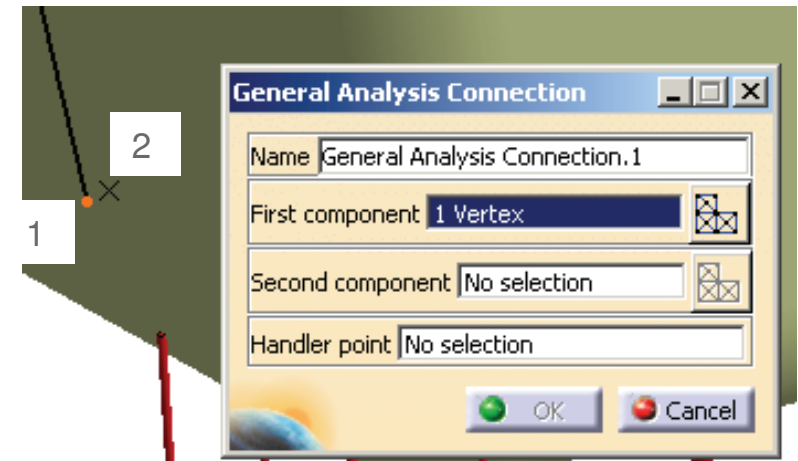
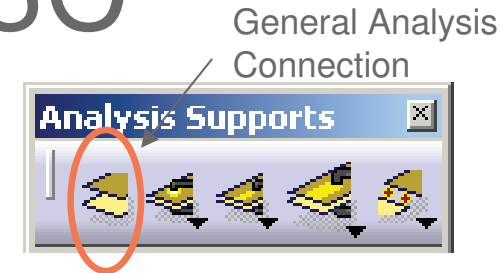
- Click “**1D Property**” icon
- Select the curve
- Select “User-defined material” option
- Click the entry box “No selection” once
- Select “User Material.1” on the tree
- Select “Cylindrical Beam” as Type
- Click the icon ★
- Enter 1.5mm as radius
- Click ok to complete



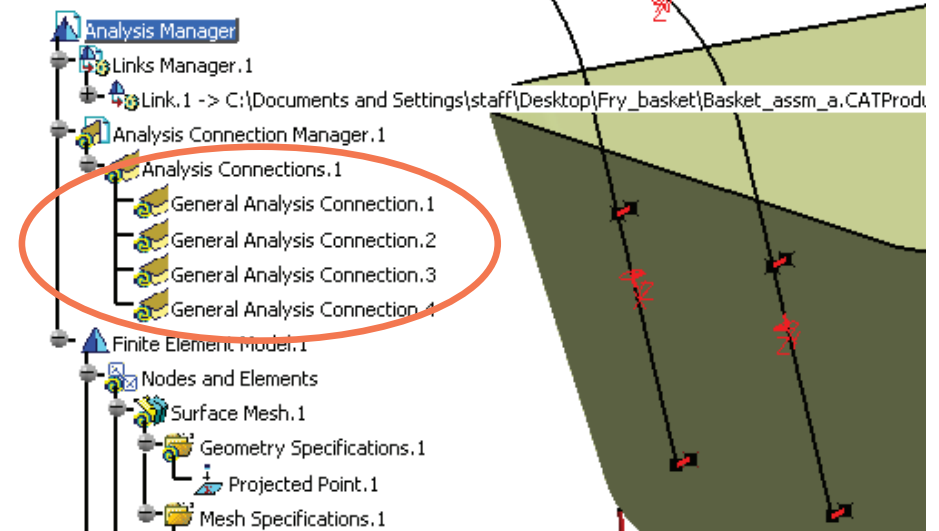
Tutorial 3C

(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



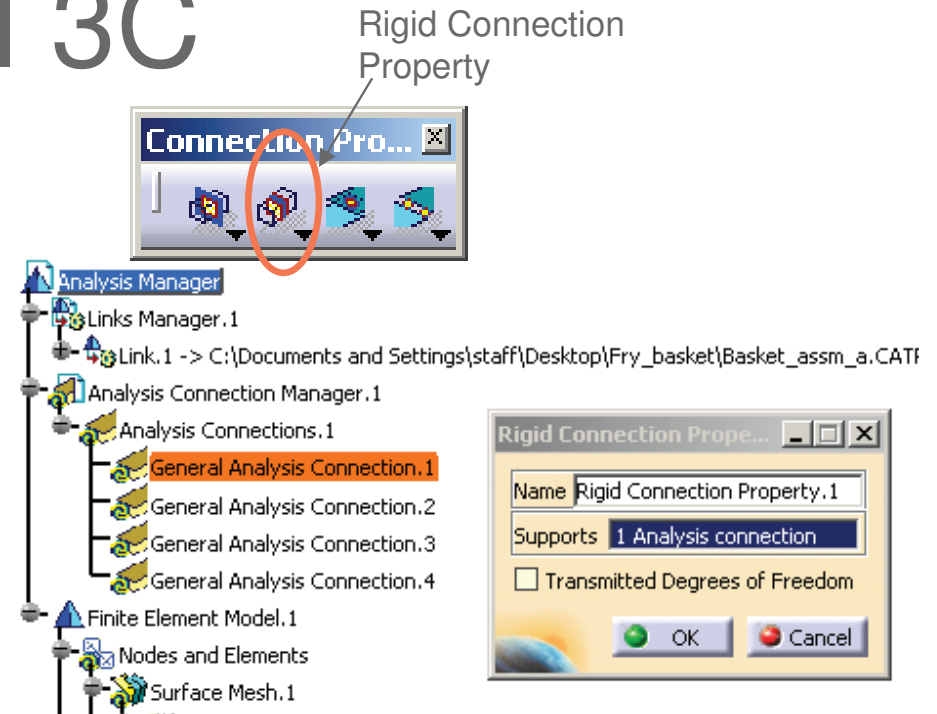
Four connections are made



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.CATAnalysis” as filename and save it in your project folder.

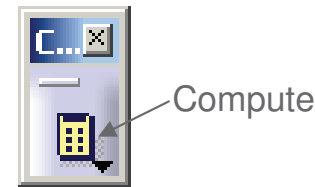
What we have already done...

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

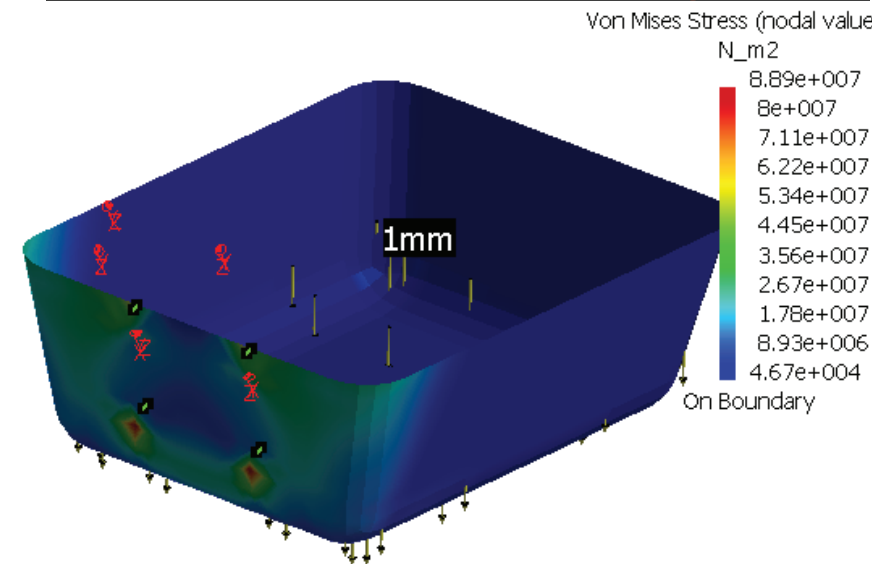
(9) To start Computation:-

- 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'):-

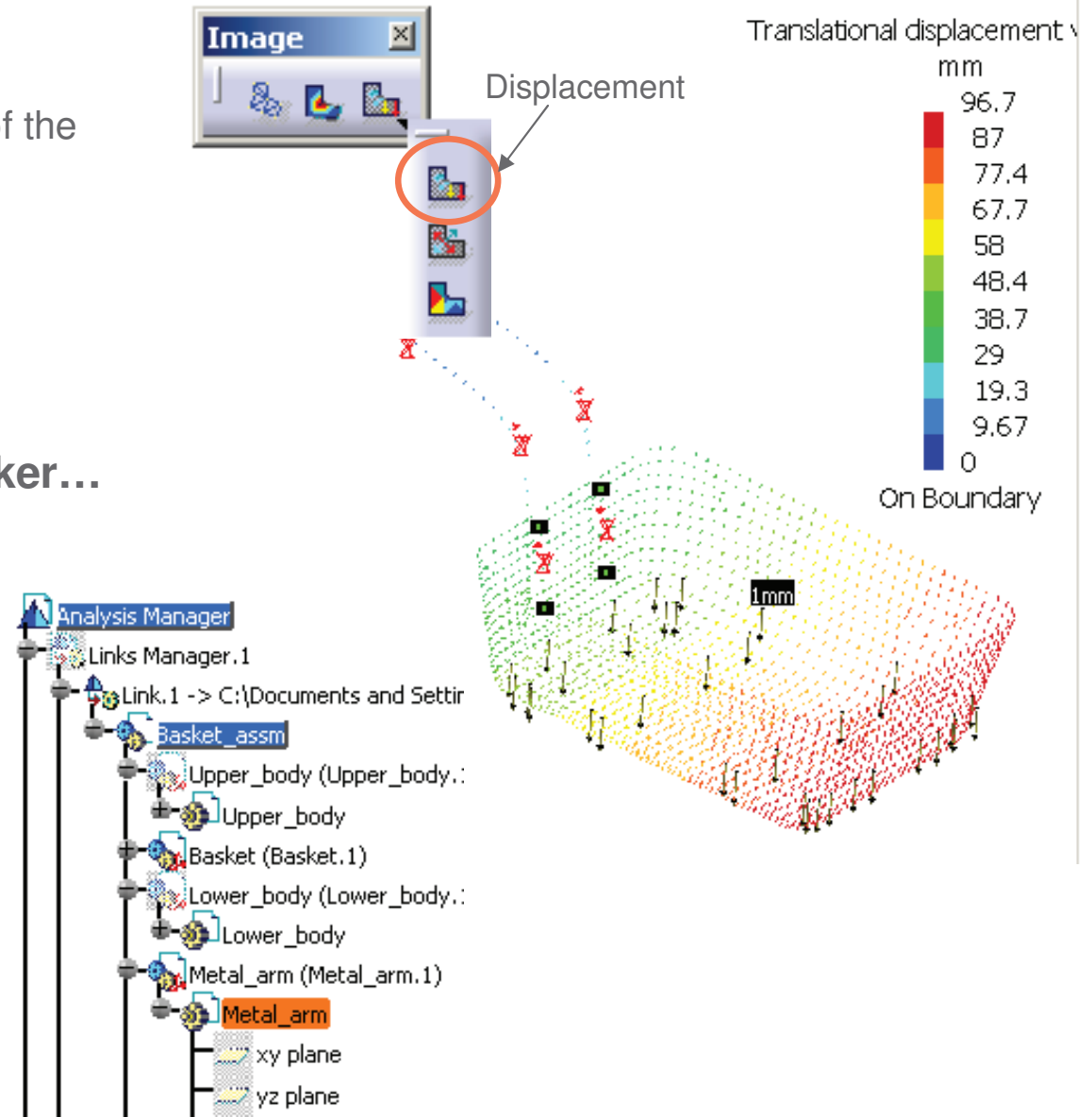
- Click “**Displacement**” icon
- (We can see the maximum displacement of the basket is about 96.7mm!)

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”:-

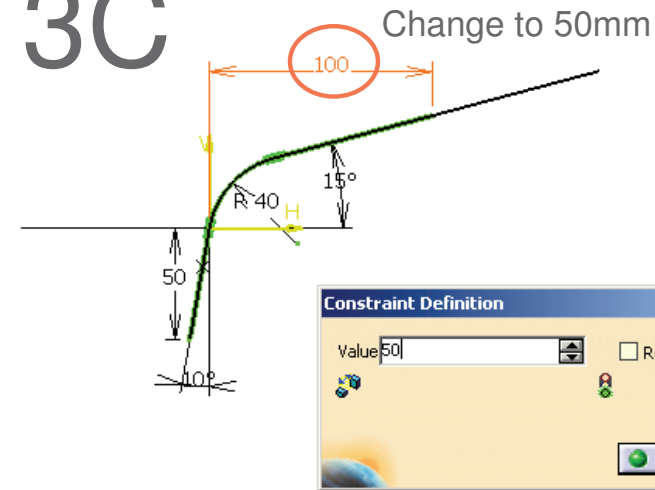
- Right-click “Metal arm” on tree
- Select Metal_arm.object/Open in new window



Tutorial 3C

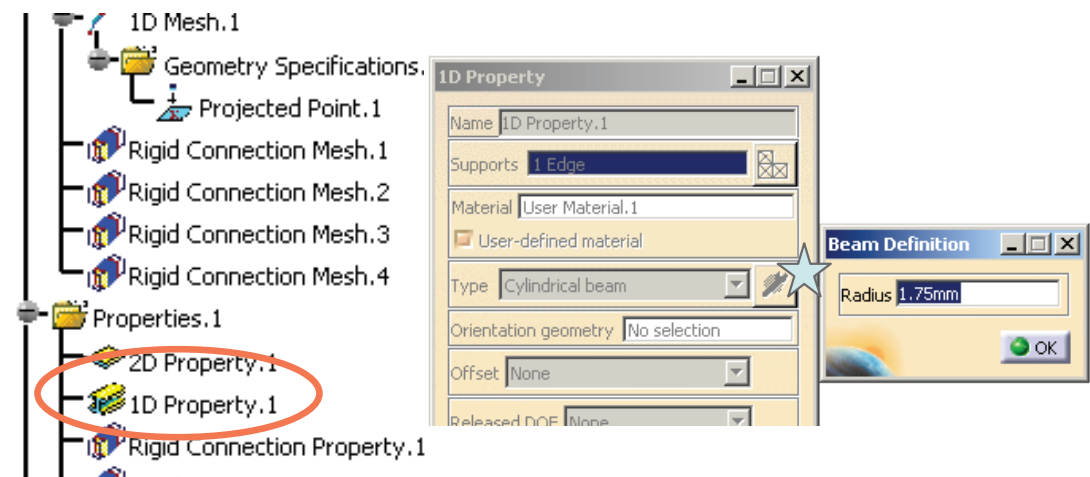
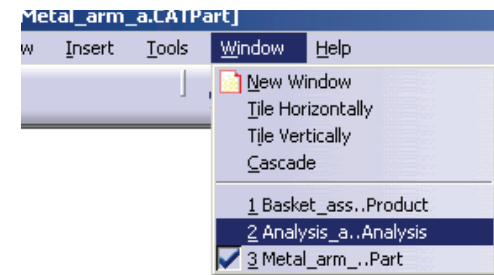
(Con't):-

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



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

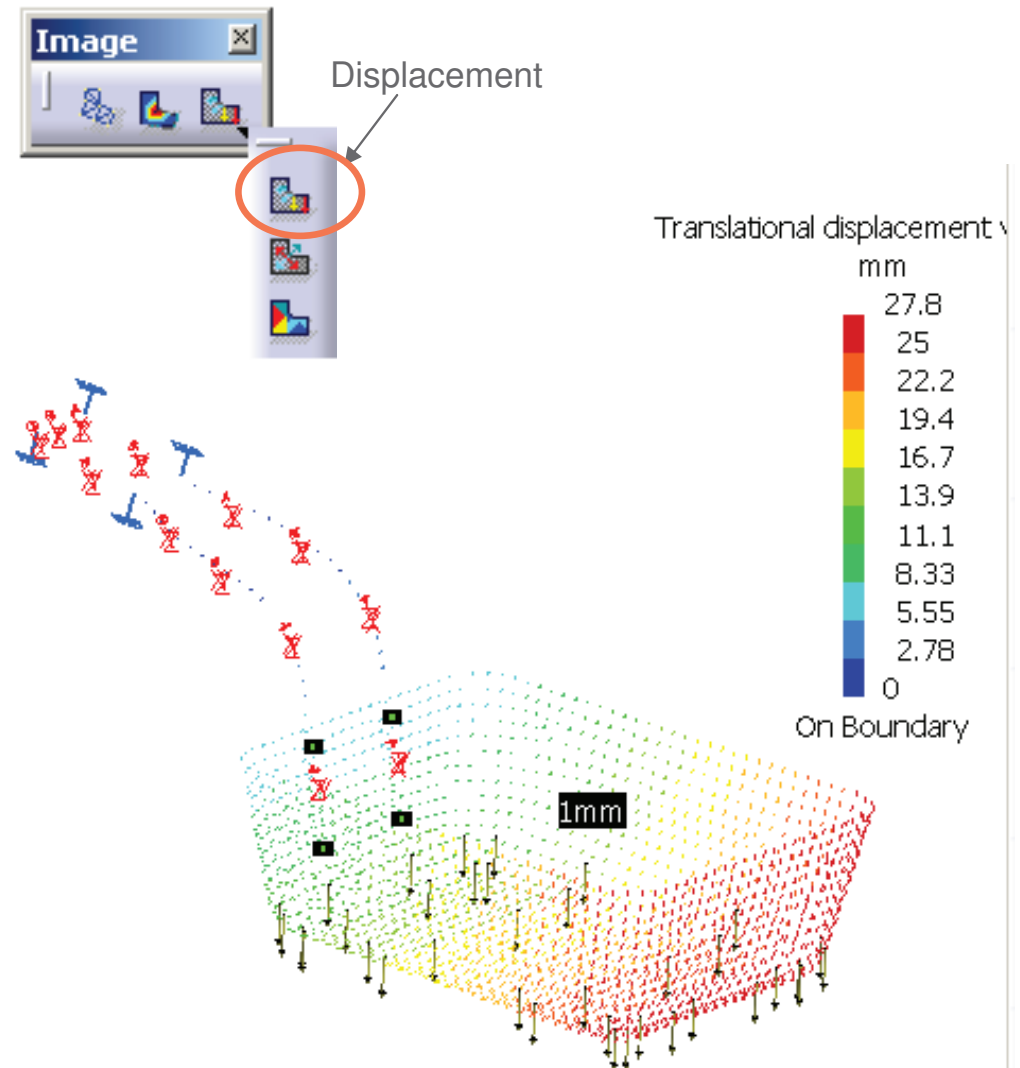
- Select “Window/Analysis1” on the menu bar to go back to the analysis workbench (The metal arm in the assembly is also updated)
- Double-Click “1D Property.1” icon on tree
- Click the icon ★
- Change Radius from 1.5 to 1.75
- Click ok to complete



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