

Objectives of the WorkShop

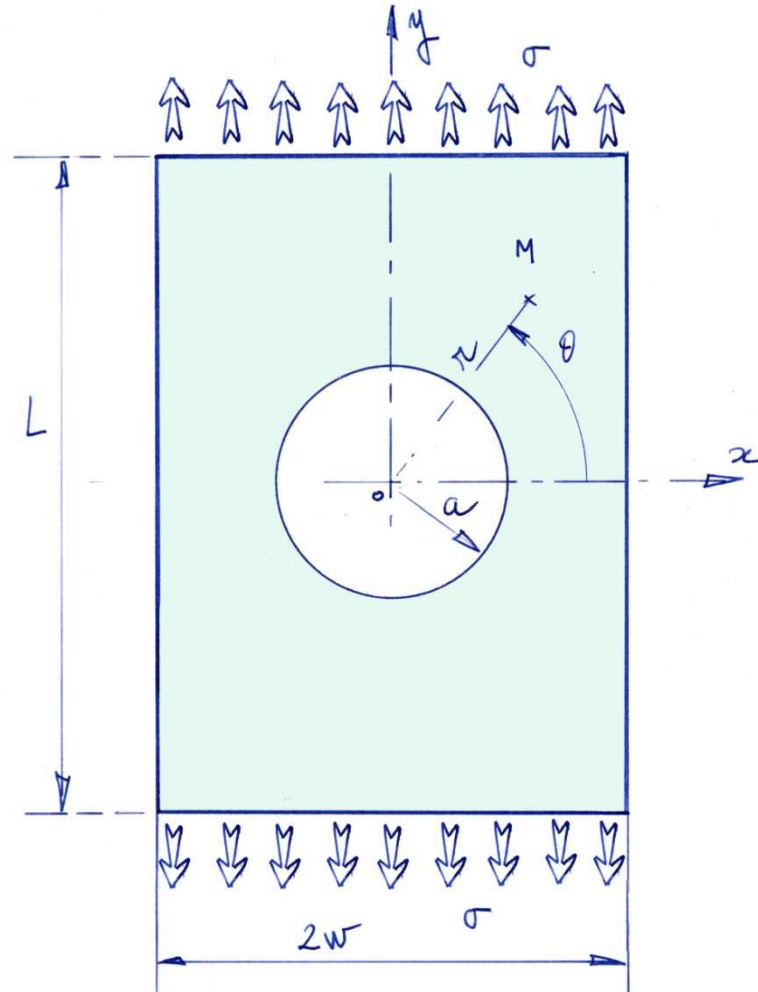
- Learn to use the elements of membrane,
- Learn to create a MeshSeed,
- Learn to create an accurate meshing,
- Learn to read the BDF File,
- Learn to use the F06 File,
- Learn to create images with the results,
- Learn to verify the accuracy of the modelling,
- Learn to create images and curves with the results.

Structure studied

A plate with a hole:

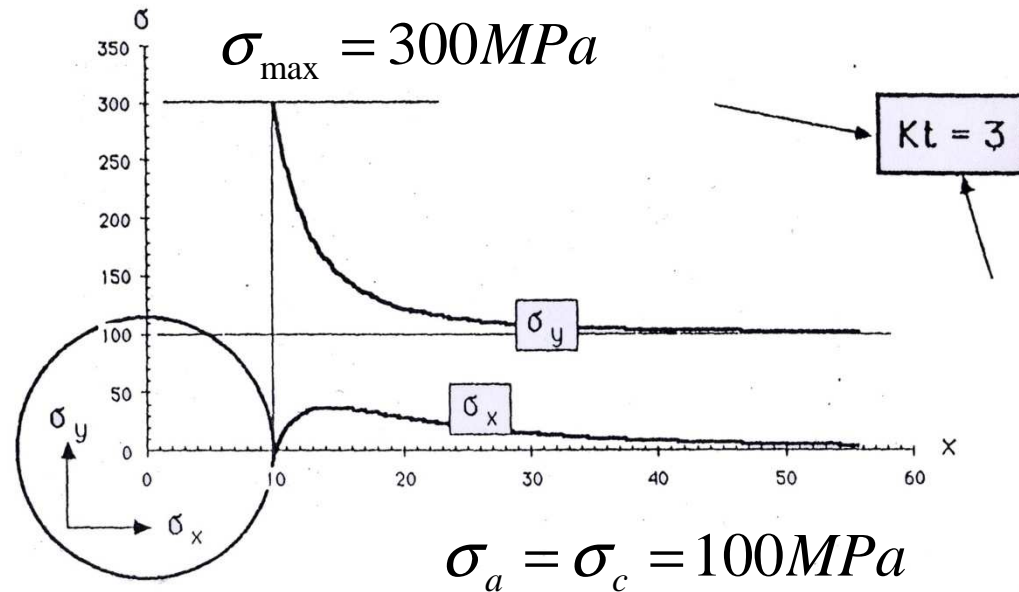
- Thickness $e = 1$ mm
- Length $L = 600$ mm
- Width $2W = 200$ mm
- Drilling $a = 50$ mm

The load is a uniform tensile stress of 100 Mpa on two opposite edges



Particular Case for an infinite plate ($r \ll W$)

$$\sigma_y = \frac{\sigma_a}{2} \left[2 + \left(\frac{a}{x} \right)^2 + 3 \left(\frac{a}{x} \right)^4 \right]$$
$$\sigma_x = \frac{3\sigma_a}{2} \left[\left(\frac{a}{x} \right)^2 - \left(\frac{a}{x} \right)^4 \right]$$



Infinite Plate : Solution in cylindrical axis

If the point M of the plate is described in a cylindrical axis system with the two coordinates r and θ , then the solution is :

$$\sigma_{rr} = \frac{\sigma}{2} \left(1 - \frac{a^2}{r^2} \right) + \frac{\sigma}{2} \left(1 - 4 \frac{a^2}{r^2} + 3 \frac{a^4}{r^4} \right) \cos 2\theta$$

$$\sigma_{\theta\theta} = \frac{\sigma}{2} \left(1 + \frac{a^2}{r^2} \right) - \frac{\sigma}{2} \left(1 + 3 \frac{a^4}{r^4} \right) \cos 2\theta$$

$$\sigma_{r\theta} = -\frac{\sigma}{2} \left(1 + 2 \frac{a^2}{r^2} - 3 \frac{a^4}{r^4} \right) \sin 2\theta$$

Case of a finite Plate

If the plate is a real plate with finite dimensions, as the one studied in this workshop, the results described below are accurate.

$$\sigma_{net} = \sigma_{\infty} \frac{w}{w-r} = \sigma_{\infty} \frac{100}{100-50} = 100\sigma_{\infty} = 200 \quad MPa$$

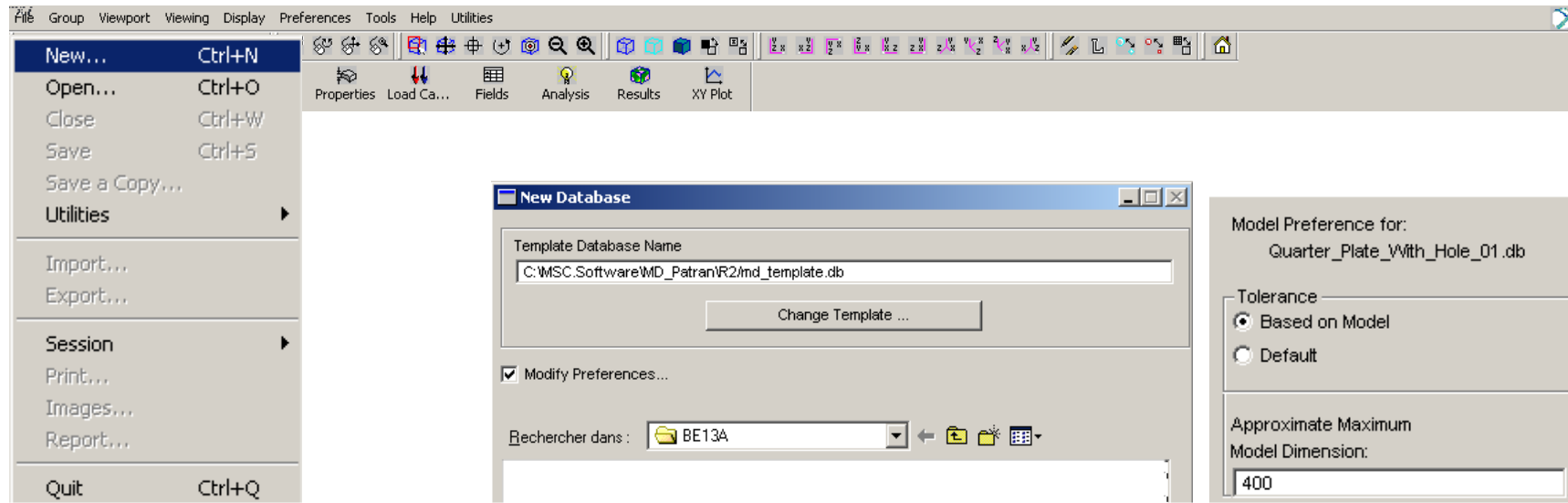
$$\sigma^{Max} = K_t \sigma_{net}$$

with

$$K_t = 2 + \left(1 - \frac{r}{w}\right)^3 = 2 + \left(1 - \frac{50}{100}\right)^3 = 2.125$$

$$\sigma^{Max} = 425 \quad MPa$$

Step 1 : Creation of a New Database



Create a new database

a File / New

b Enter **Plate_With_Hole** as the File name

c Click **OK**

d Select **Based on Model** for Tolerance

e Enter **400** For **Model Dimension**

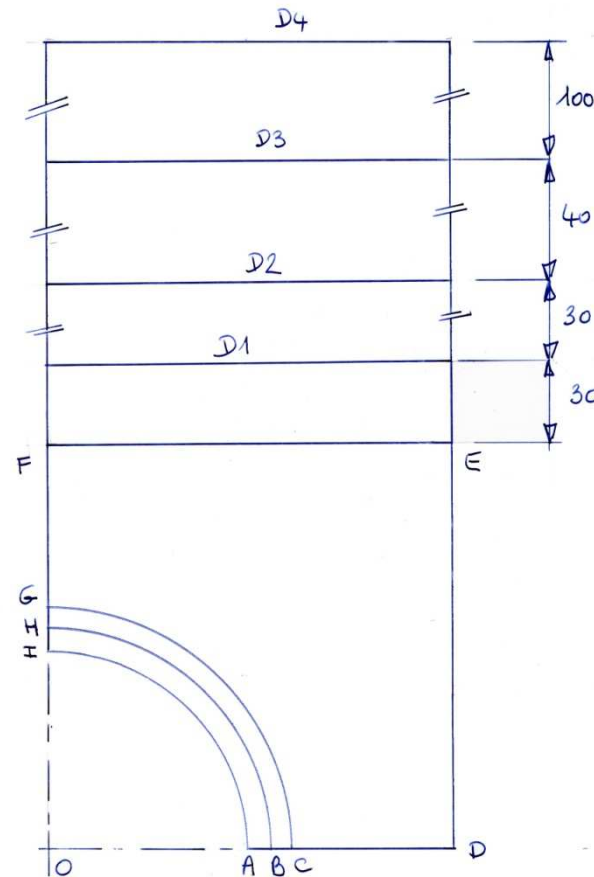
f Select **MD Nastran** as the **Analysis Code**

g Select **Structural** as the Analysis Type

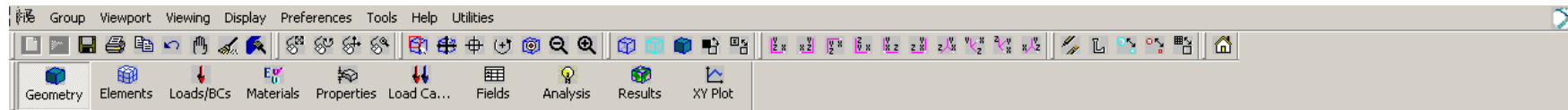
h Click **OK**

Points O, A to I and Lines D1 to D4

Description of the geometry of a quarter of the plate.



Step 1 : Creation of Geometry - Points



Create the first point :

- a Geometry : **Create / Point / XYZ**
- b Unselect **Auto Execute**
- c Enter [0,0,0] for the **Point Coordinate List**.
- d Click **Apply**

Create the other points (A....I) with the same method.

Point	x	y	z	Nastran Id
A	50	0	0	1
B	53	0	0	3
C	56	0	0	4
D	100	0	0	5
E	100	100	0	6
F	0	100	0	7
G	0	56	0	8
H	0	53	0	9
I	0	50	0	10

Action: **Create** ▼

Object: **Point** ▼

Method: **XYZ** ▼

Point ID List

1

Refer. Coordinate Frame

Coord 0

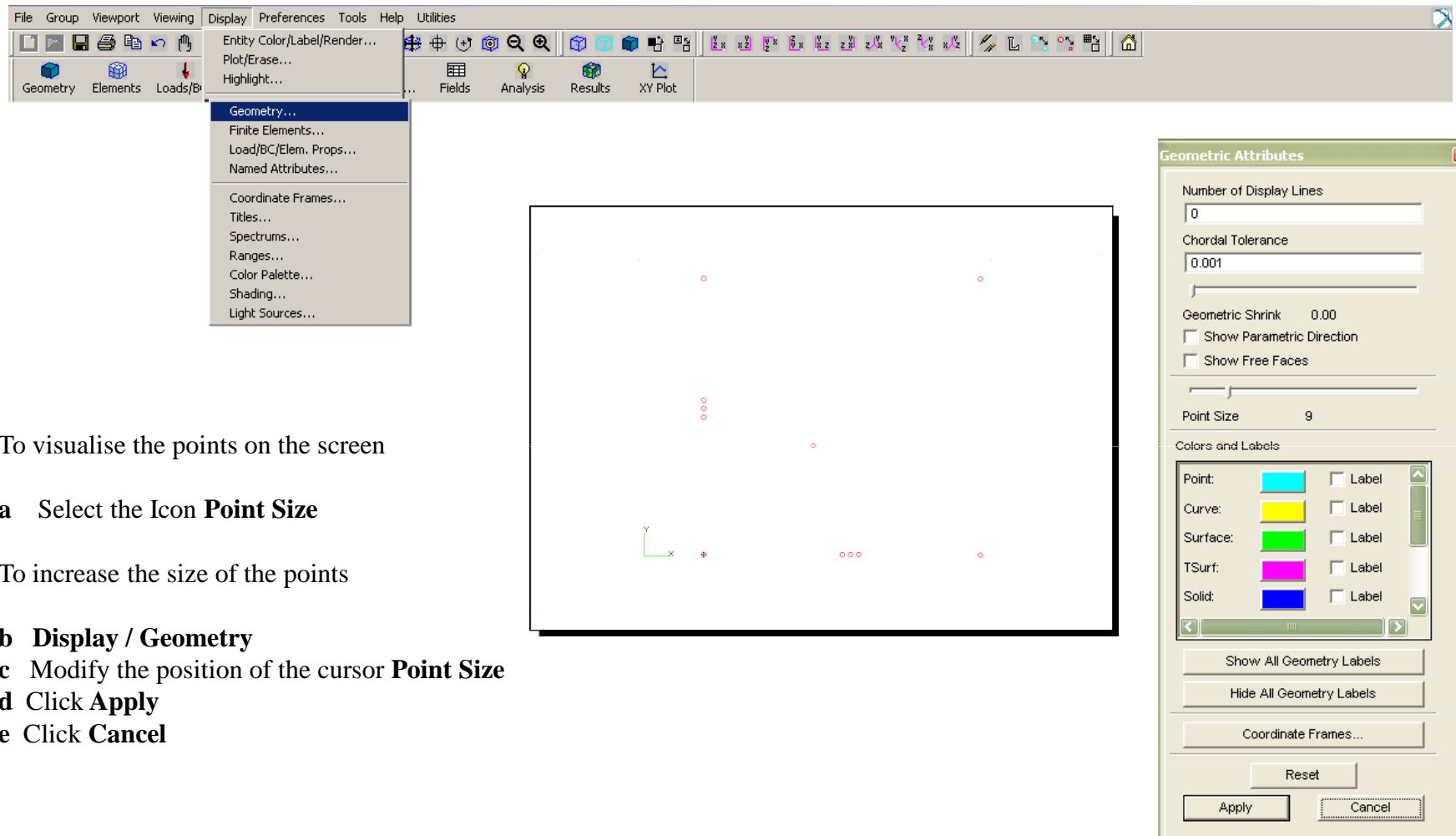
☐ Auto Execute

Point Coordinates List

[0 0 0]

-Apply-

Viewing of the Points



The screenshot shows a software interface with a menu bar (File, Group, Viewport, Viewing, Display, Preferences, Tools, Help, Utilities) and a toolbar. The 'Display' menu is open, showing options like 'Entity Color/Label/Render...', 'Plot/Erase...', 'Highlight...', 'Geometry...', 'Finite Elements...', 'Load/BC/Elem. Props...', 'Named Attributes...', 'Coordinate Frames...', 'Titles...', 'Spectrums...', 'Ranges...', 'Color Palette...', 'Shading...', and 'Light Sources...'. The 'Geometry...' option is selected. The 3D viewport shows a coordinate system with x, y, and z axes and several red points. The 'Geometric Attributes' dialog box is open, showing settings for 'Number of Display Lines' (0), 'Chordal Tolerance' (0.001), 'Geometric Shrink' (0.00), 'Show Parametric Direction' (unchecked), 'Show Free Faces' (unchecked), 'Point Size' (9), and 'Colors and Labels' (Point: cyan, Curve: yellow, Surface: green, TSurf: magenta, Solid: blue). The 'Show All Geometry Labels' button is highlighted.

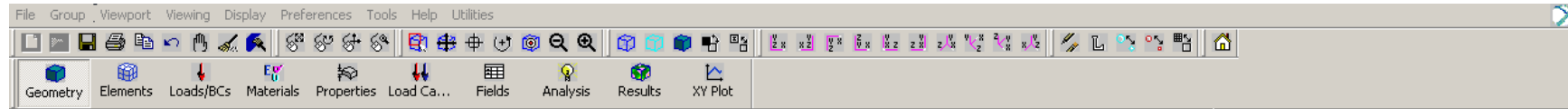
To visualise the points on the screen

- Select the Icon **Point Size**

To increase the size of the points

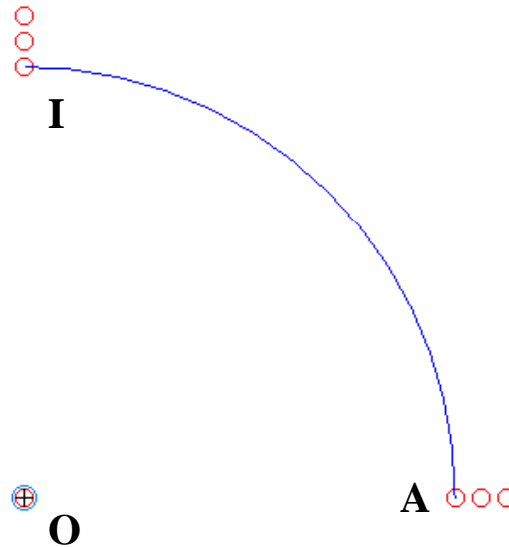
- Display / Geometry**
- Modify the position of the cursor **Point Size**
- Click **Apply**
- Click **Cancel**

Step 1 : Creation of Geometry – Curves



Create the first curve (a quarter of circle) :

- a Geometry : **Create / Curve / 2D Arc2Point**
- b Select option : **Center**
- c Unselect **Auto Execute**
- d Activate **Center Point List** with the left click
- e Screen Pick the point O (Point 1) for the **Center Point List**.
- f Activate **Starting Point List** with the left click
- g Screen Pick the point A (Point 2) for the **Starting Point List**.
- h Activate **Ending Point List** with the left click
- i Pick the point I (Point 10) for the **Ending Point List**.
- j Click **Apply**



Action: **Create** ▼

Object: **Curve** ▼

Method: **2D Arc2Point** ▼

Curve ID List
1

Option: **Center** ▼

Arc2Point Parameters...

☐ Project to Plane

Construction Plane List
Coord 0.3

☐ Auto Execute

Center Point List
Point 1

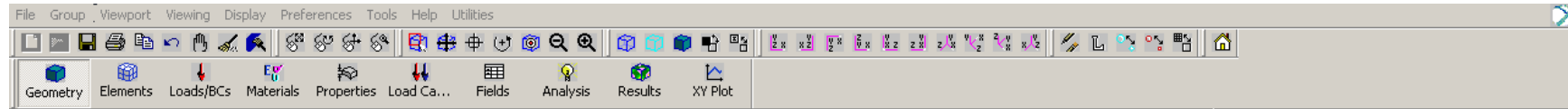
Starting Point List
Point 2

Ending Point List
Point 10

-Apply-

Create the two other Arc circles (BH and CG) with the same method.

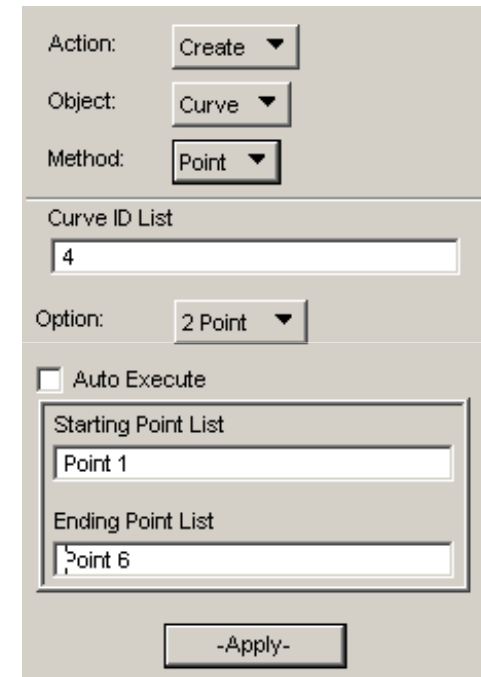
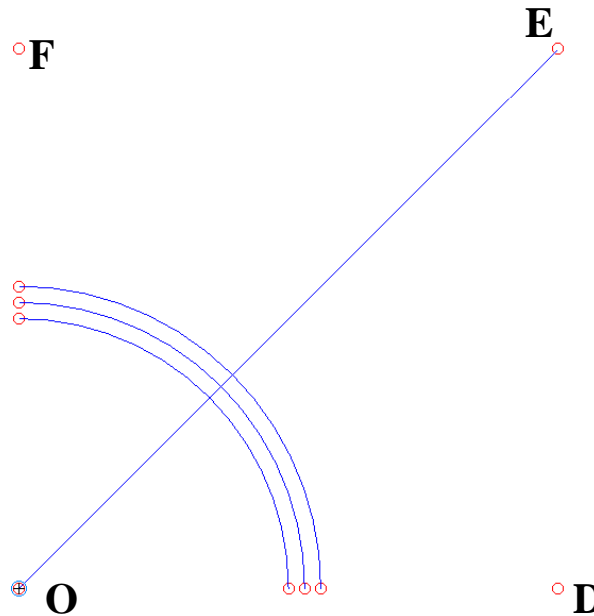
Step 1 : Creation of Geometry – Curves



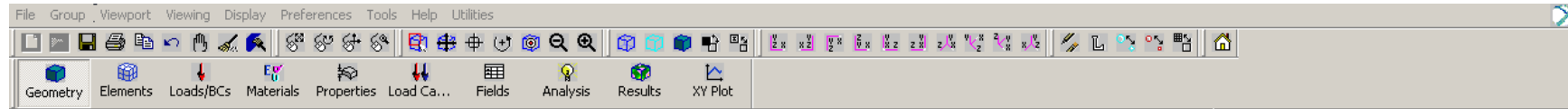
Create the line OE (this line will be used later) :

- Geometry : **Create / Curve / Point - Option : 2Point**
- Unselect **Auto Execute**
- Screen Pick the point O (Point 1) for the **Starting Point List**.
- Screen Pick the point E (Point 6) for the **Ending Point List**.
- Click **Apply**

Create the other lines **DE** and **FE**



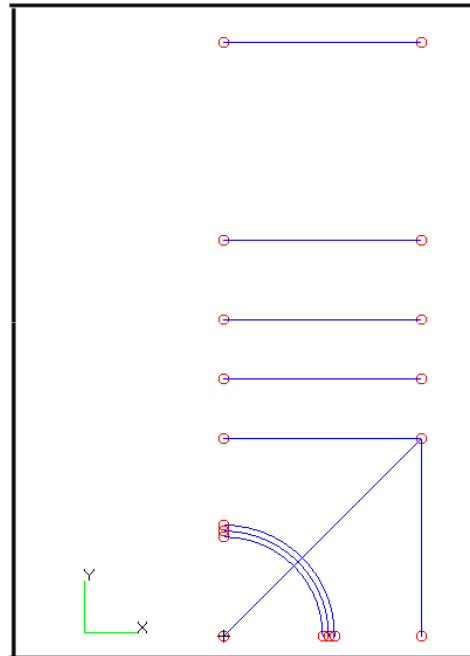
Step 1 : Creation of Geometry – Curves



Create the line D1 :

- a Geometry : **Create / Curve / XYZ**
- b Unselect **Auto Execute**
- c Enter **[0,130,0]**, the coordinates of the beginning point of the line in **Origin Coordinates List**
- d Enter **[100,0,0]**, the projected lengths of the line in **Vector Coordinates List**
- e Click **Apply**

Create the other lines D2, D3 & D4



Action:

Object:

Method:

Curve ID List

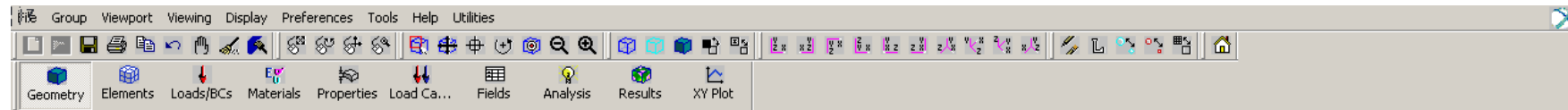
Refer. Coordinate Frame

Vector Coordinates List

☐ Auto Execute

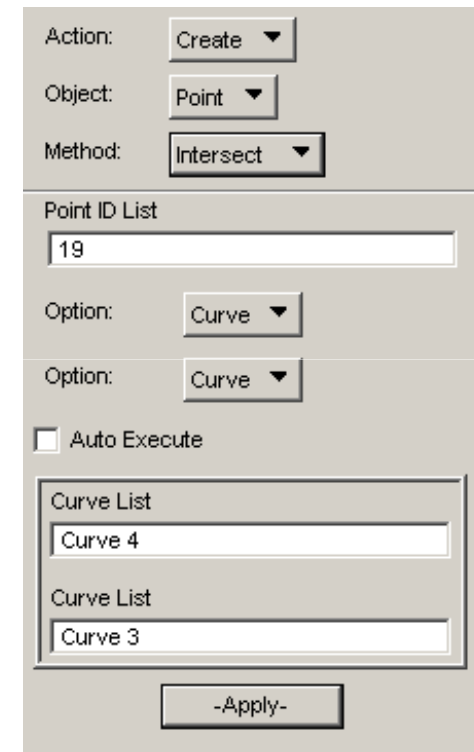
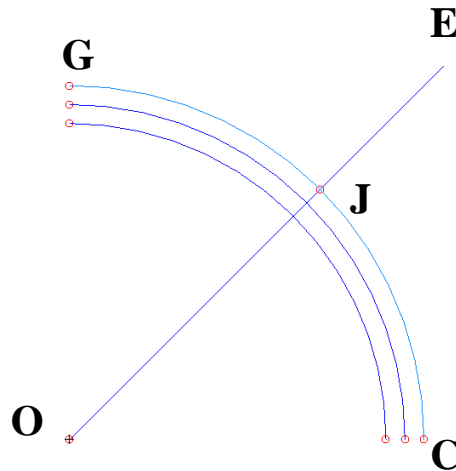
Origin Coordinates List

Step 1 : Creation of Geometry

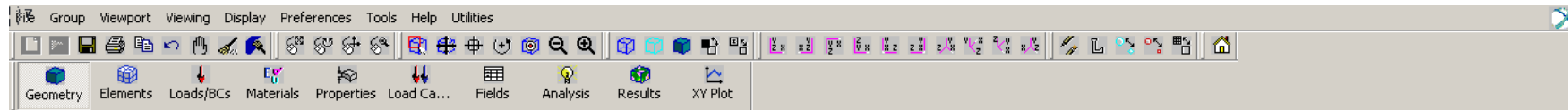


Create the point J :

- a Geometry : Create / Point / Intersect -
- b Select Option 1 : **Curve** – Option 2 : **Curve**
- c Unselect **Auto Execute**
- d Pick the Curve 4 (Line OE) in **Curve List**.
- e Pick the Curve 3 (Cercle CG) in **Curve List**.
- g Click **Apply**

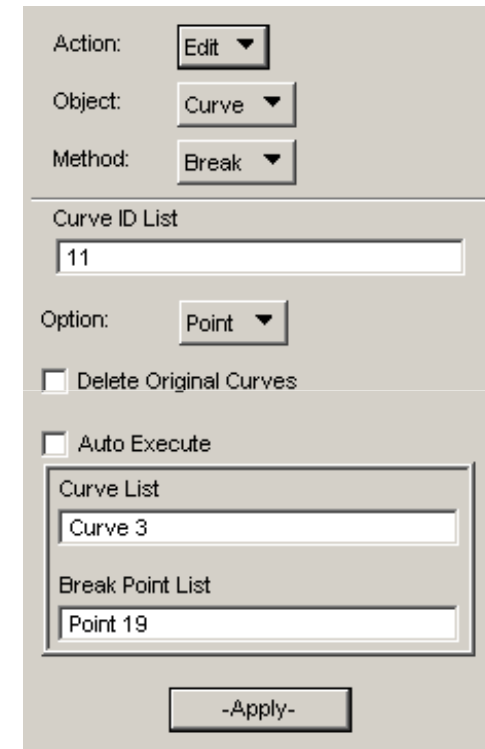
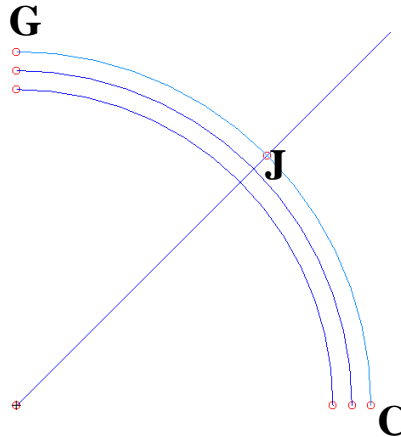


Step 1 : Creation of Geometry

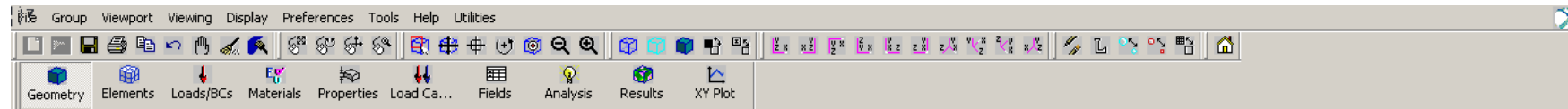


Break the circle on the point J :

- a Geometry : **Edit / Curve / Break**
- b Select Option : **Point**
- c Unselect **Delete Original Curve**
- d Unselect **Auto Execute**
- e Screen Pick the Curve 3 (Cercle CG) in **Curve List**.
- f Screen Pick the Point J in **Break Point List**.
- g Click **Apply**



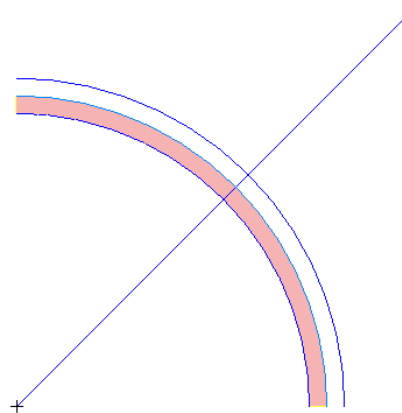
Step 1 : Creation of Geometry



Create Surfaces with 2 curvilinear boundaries :

- a Geometry : **Create / Surface / Curve**
- b Select Option : **2 Curves**
- c Unselect **Auto Execute**
- d Screen Pick the Curve 1 (Cercle AI) in **Starting Curve List**.
- e Screen Pick the Curve 2 (Cercle BH) in **Ending Curve List**.
- f Click **Apply**
- g Use the shading Icon to visualize the surface

Create the surface 2 with curvilinear boundaries



Action:

Object:

Method:

Surface ID List

Option:

Parameterization Method

☐ Chord Length

☒ Uniform

☐ Manifold

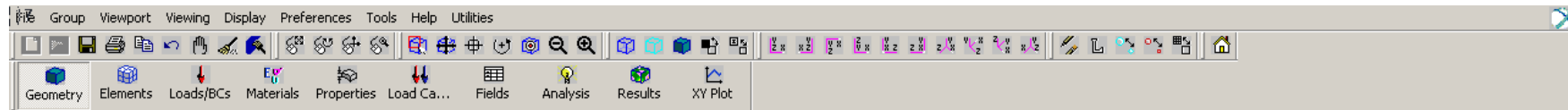
Manifold Surface

☐ Auto Execute

Starting Curve List

Ending Curve List

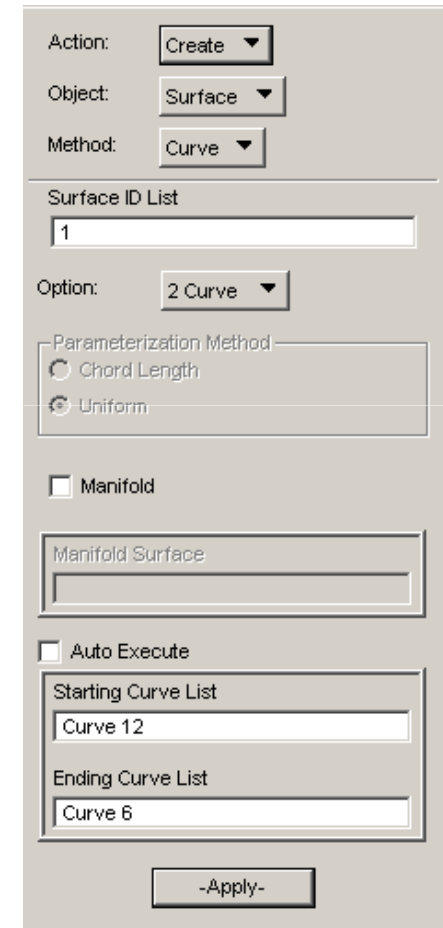
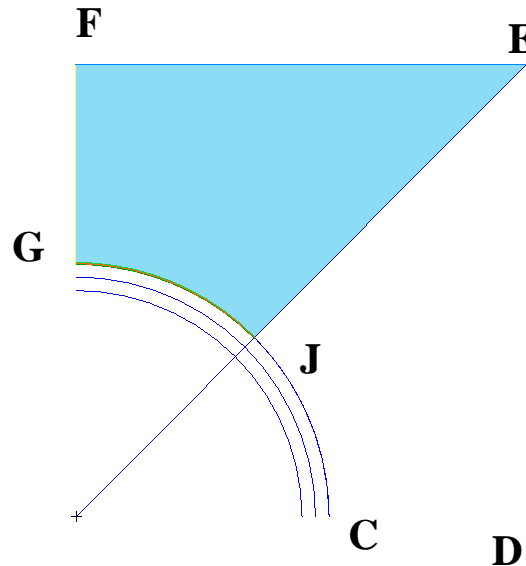
Step 1 : Creation of Geometry



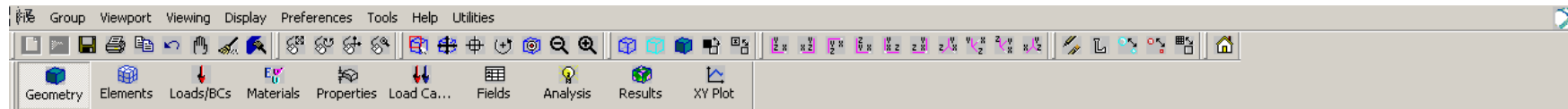
Create 2 New Surfaces as indicated below :

- a Geometry : **Create / Surface / Curve**
- b Select Option : **2 Curves**
- c Unselect **Auto Execute**
- d Pick the Curve 12 (Circle JG) in **Starting Curve List**.
- e Pick the Curve 6 (Line EF) in **Ending Curve List**.
- f Click **Apply**

Create the surface 4 with Circle CJ and line DE



Step 1 : Creation of Geometry



Create 4 New Surfaces with the different lines as indicated below :

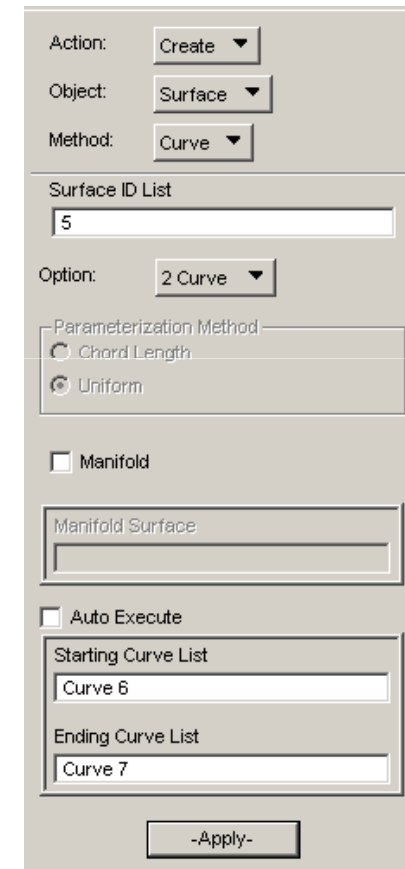
- a Geometry : **Create / Surface / Curve**
- b Select Option : **2 Curves**
- c Unselect **Auto Execute**
- d Screen Pick the Curve 6 (Line EF) in **Starting Curve List**.
- e Screen Pick the Curve 7 (Line D1) in **Ending Curve List**.
- f Click **Apply**

Create the other surfaces :

Surface 6 with D1 & D2

Surface 7 with D2 & D3

Surface 8 with D3 & D4

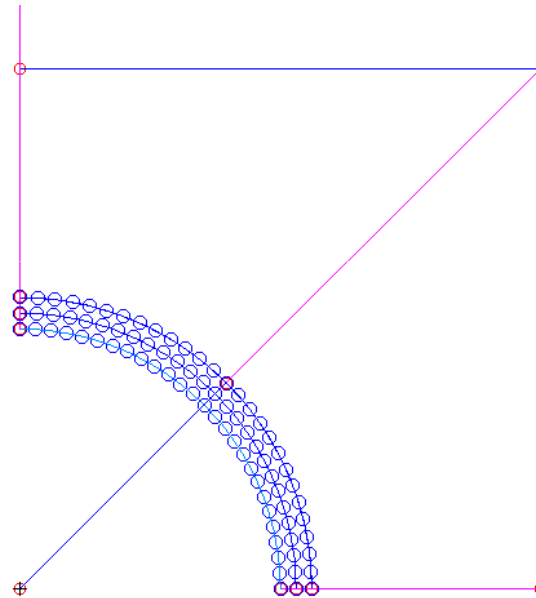


Step 2 : Meshing



Create a mesh seed :

- a Elements : **Create / Mesh Seed / Uniform**
- b Unselect **Auto Execute**
- c Enter 26 as the number of elements.
- d Screen Pick the Curve 1 for **Curve List List**.
- e Click **Apply**
- f Screen Pick the Curve 2 for **Curve List List**.
- g Click **Apply**
- h Enter 13 as the number of elements.
- i Screen Pick the Curve 11 for **Curve List List**.
- j Click **Apply**
- k Enter 13 as the number of elements.
- l Screen Pick the Curve 12 for **Curve List List**.
- m Click **Apply**



Action: Create

Object: Mesh Seed

Type: Uniform

Display Existing Seeds

Element Edge Length Data

☒ Number of Elements

☐ Element Length (L)

Number = 26

☐ Auto Execute

Curve List

Curve 1

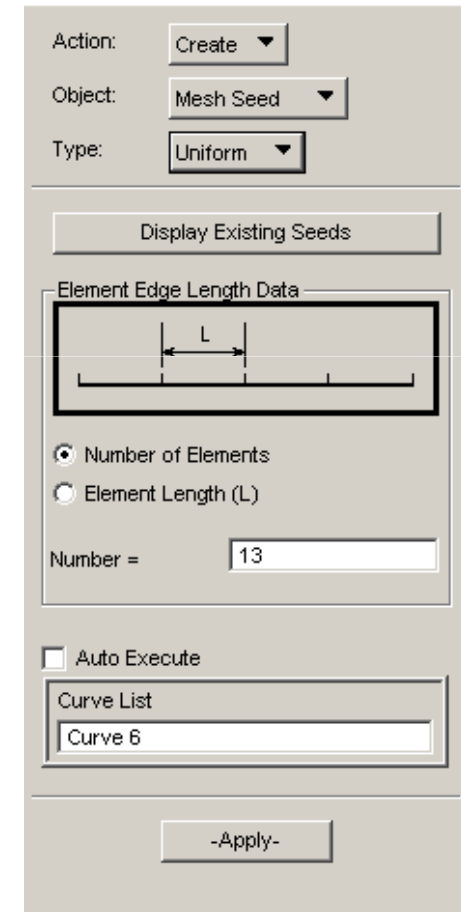
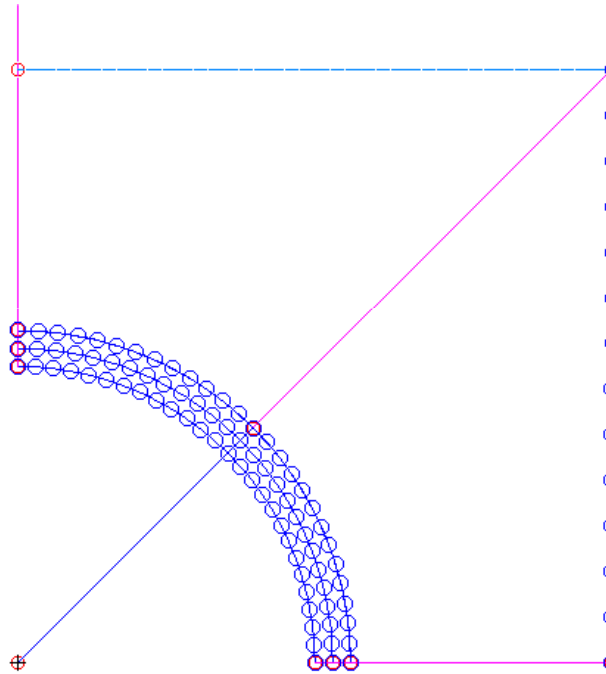
-Apply-

Step 2 : Meshing



Continue the mesh seed :

- a Elements: **Create / Mesh Seed / Uniform**
- b Enter 13 as the number of elements.
- c Screen Pick the Curve 5 for **Curve List**
- d Click **Apply**
- e Screen Pick the Curve 6 for **Curve List**
- f Click **Apply**
- g Enter 11 as the number of elements.
- h Screen Pick the Curve 7 for **Curve List**
- i Click **Apply**
- j Enter 9 as the number of elements.
- k Screen Pick the Curve 8 for **Curve List**
- l Click **Apply**
- m Enter 7 as the number of elements.
- n Screen Pick the Curves 9 & 10 for **Curve List**
- o Click **Apply**

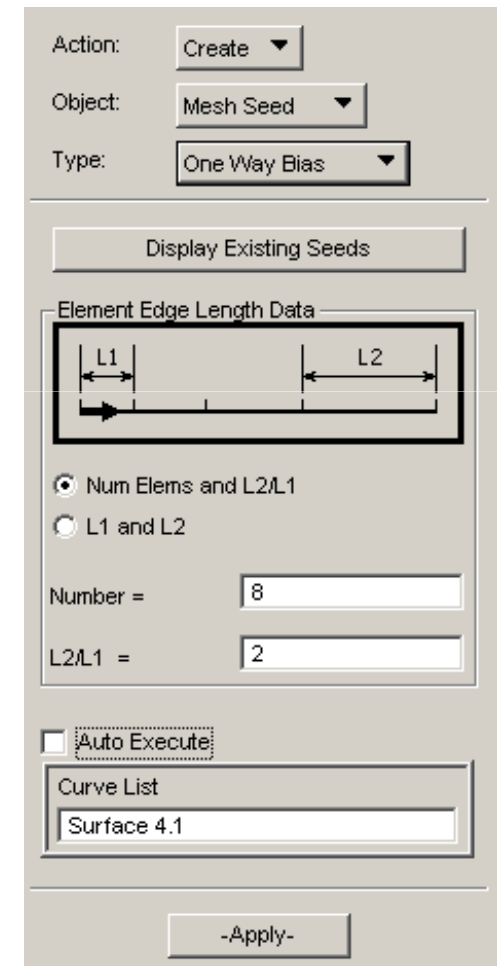
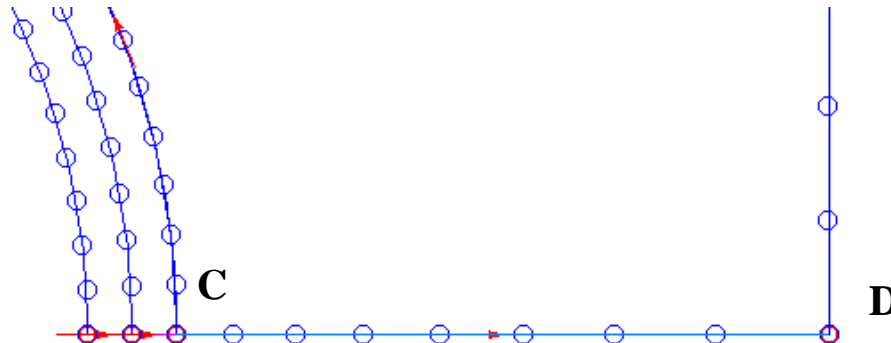


Step 2 : Meshing

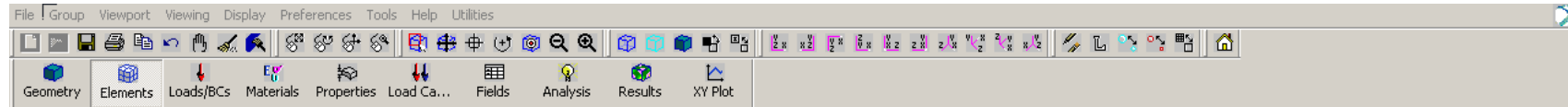


Continue the mesh seed :

- a Elements : **Create / Mesh Seed / One Way Bias**
- b Unselect **Auto execute**
- c Enter 8 as the number of elements.
- d Enter 2 (or 0.5) as the ratio **L2/L1**
- e Screen Pick the Curve between C & D for **Curve List**
- f Click **Apply**
- g Screen Pick the Curve between G & F for **Curve List**
- h Click **Apply**



Step 2 : Meshing



Create the mesh (nodes & elements) :

a Elements: **Create / Mesh / Surface**

b Select **Quad – Isomesh – Quad4**

c Screen pick the surface 1 for **Surface list**

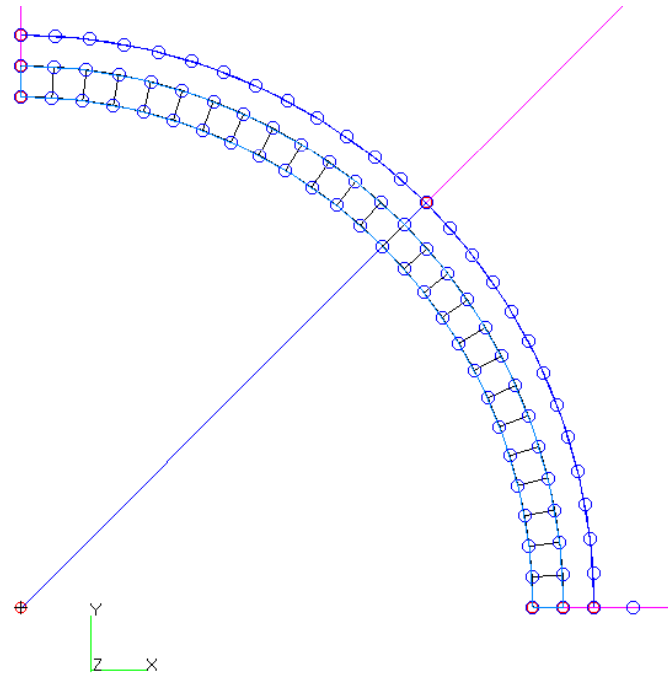
d Unselect **Automatic calculation**

e Enter 3 (Size of the element) for **Value**.

f Click **Apply**

g Screen Pick the Surface 2 for **Surface list**.

h Click **Apply**



Action: **Create** ▼

Object: **Mesh** ▼

Type: **Surface** ▼

Output ID List

Node

Element

Elem Shape **Quad** ▼

Mesher **IsoMesh** ▼

Topology **Quad4** ▼

IsoMesh Parameters...

Node Coordinate Frames...

Surface List

Global Edge Length

☐ Automatic Calculation

Value

Prop. Name: - None -

Prop. Type: - N/A -

Select Existing Prop...

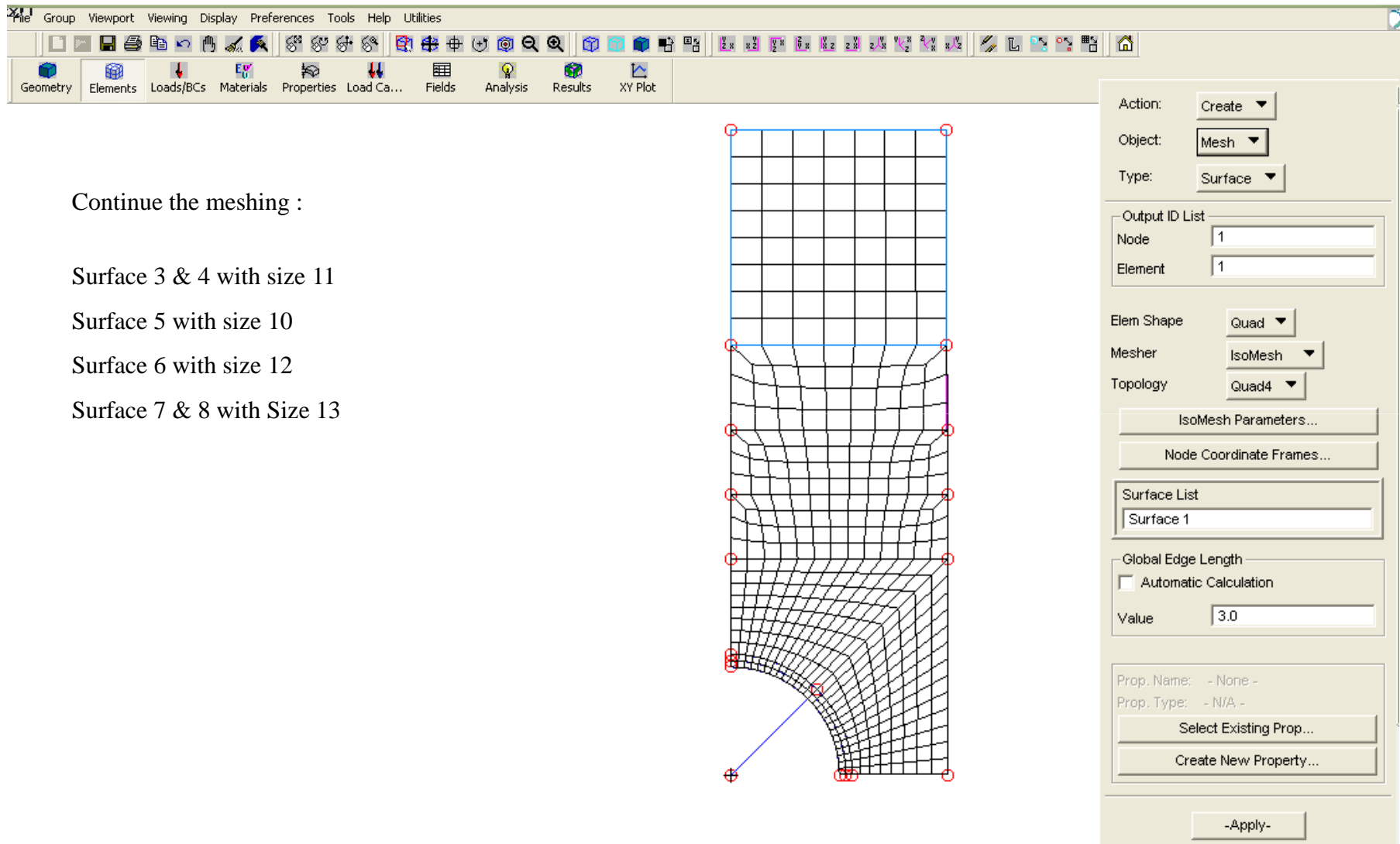
Create New Property...

-Apply-

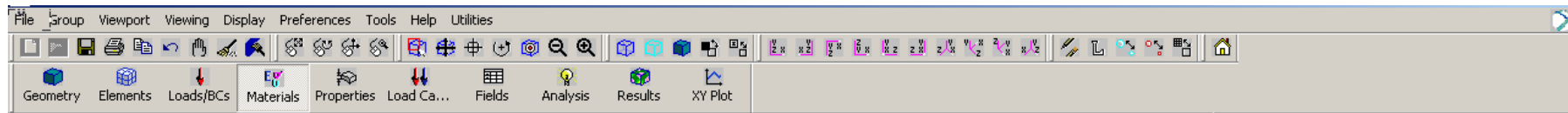
Step 3 : Creation of the Nodes & Elements

Continue the meshing :

- Surface 3 & 4 with size 11
- Surface 5 with size 10
- Surface 6 with size 12
- Surface 7 & 8 with Size 13



Step 4 Create material



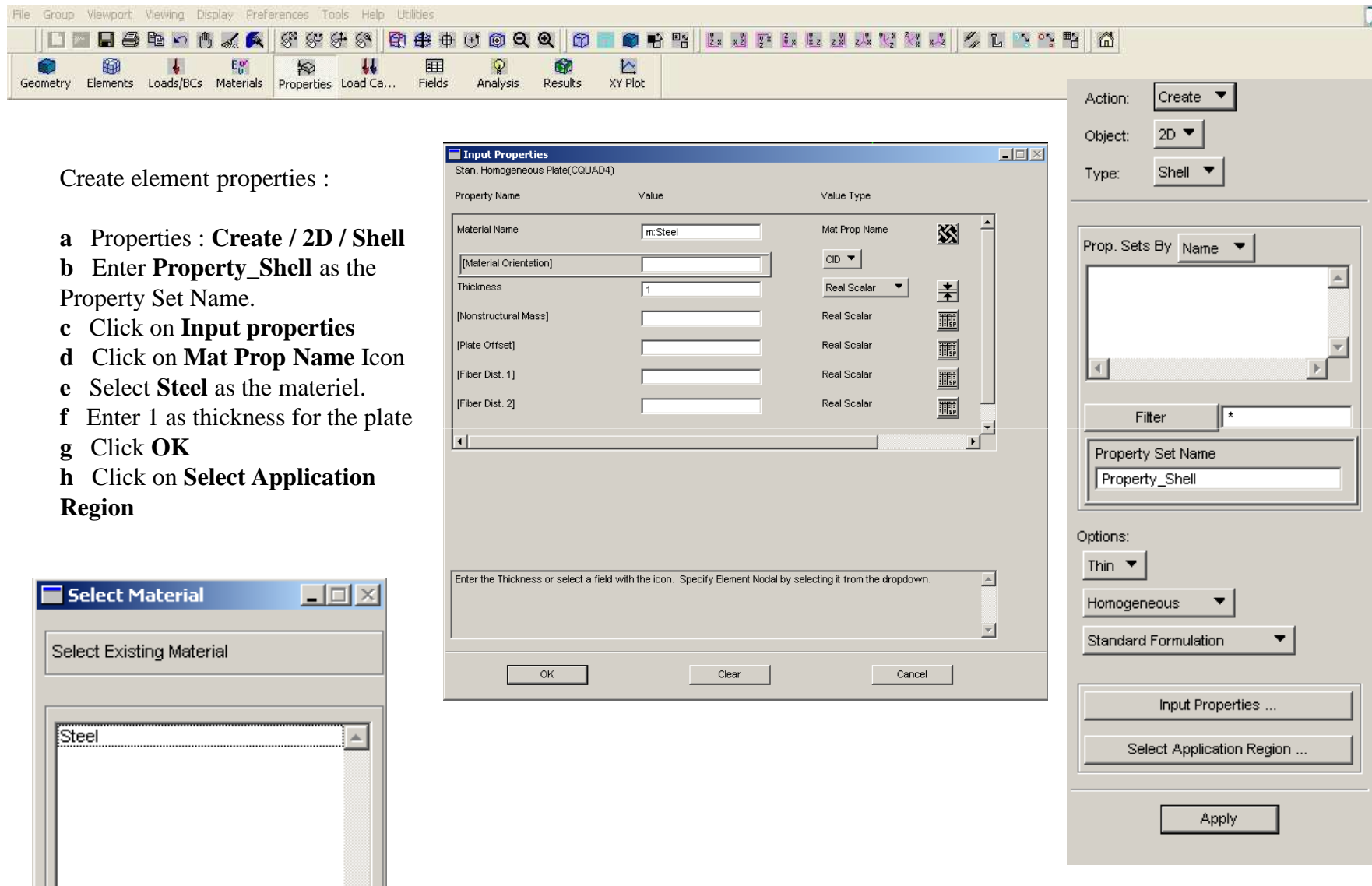
Create an isotropic material :

- a Material : **Create / Isotropic / Manual Input**
- b Enter **Steel** as the Material Name
- c Click on **Input properties**
- d Enter **200e3** for Elastic Modulus
- e Enter **0.3** for Poisson's Ratio
- f Click **OK**
- g Click **Apply**
- h Verify that the material has been created in the field **Existing Materials**

The 'Input Options' dialog box is shown. The 'Constitutive Model' is set to 'Linear Elastic'. The 'Property Name' and 'Value' columns are visible. The 'Elastic Modulus' is entered as '200e3' and the 'Poisson Ratio' is entered as '0.3'. Other properties like 'Shear Modulus', 'Density', 'Thermal Expan. Coeff', 'Structural Damping Coeff', and 'Reference Temperature' are empty. The 'Temperature Dep/Model Variable Fields' section is empty. The 'Current Constitutive Models' section is also empty. At the bottom are 'OK', 'Clear', and 'Cancel' buttons.

The 'Material Properties' dialog box is shown. The 'Action' is set to 'Create', the 'Object' is 'Isotropic', and the 'Method' is 'Manual Input'. The 'Existing Materials' list is empty. The 'Material Name' field contains 'Steel'. The 'Description' field shows 'Date: 18-Jan-08' and 'Time: 15:52:59'. At the bottom are 'Input Properties ...', 'Change Material Status ...', and 'Apply' buttons.

Step 5 : Create Physical Properties

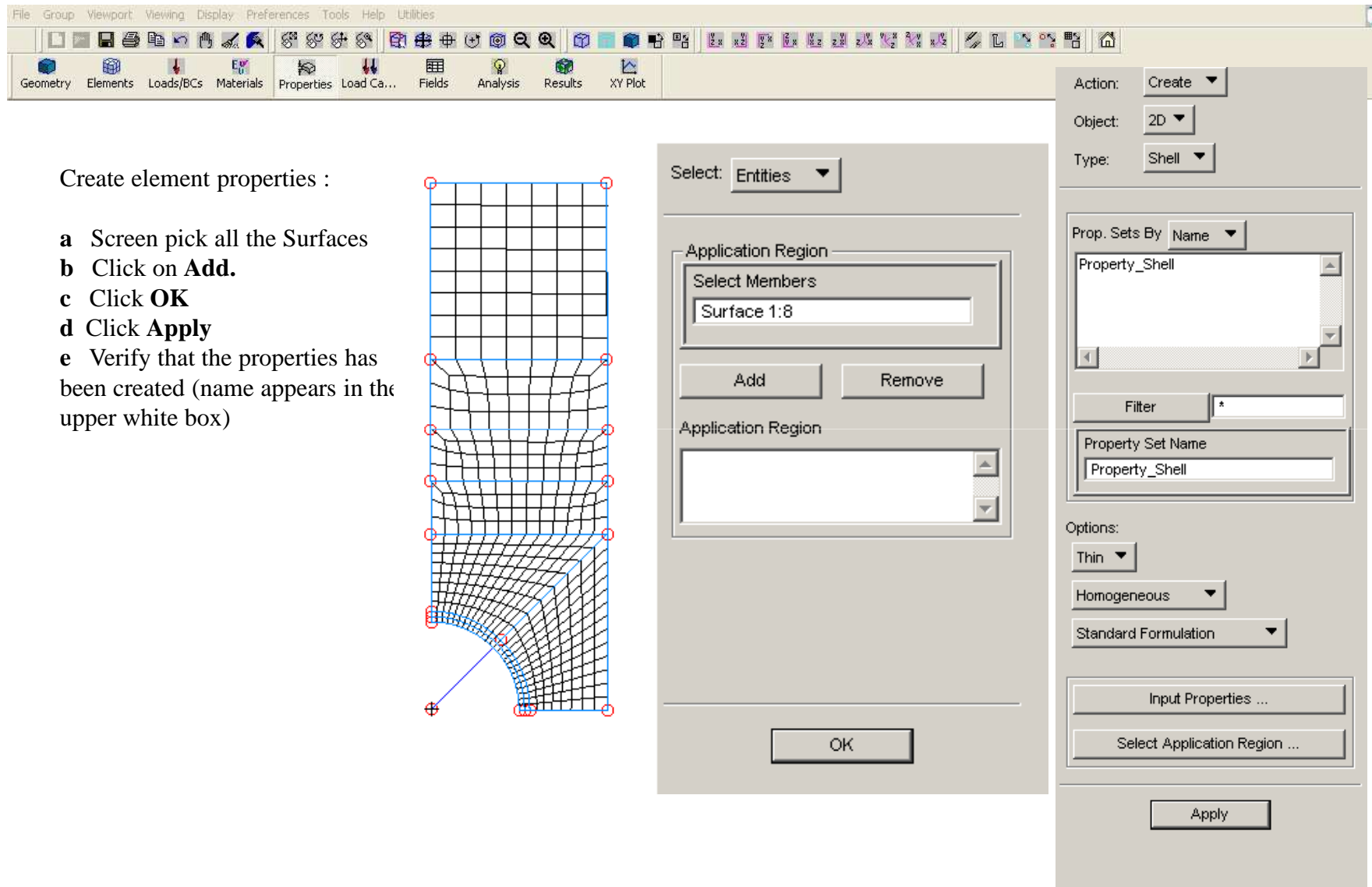


Create element properties :

- Properties : **Create / 2D / Shell**
- Enter **Property_Shell** as the Property Set Name.
- Click on **Input properties**
- Click on **Mat Prop Name** Icon
- Select **Steel** as the material.
- Enter 1 as thickness for the plate
- Click **OK**
- Click on **Select Application Region**

Region

Step 5 : Create Physical Properties

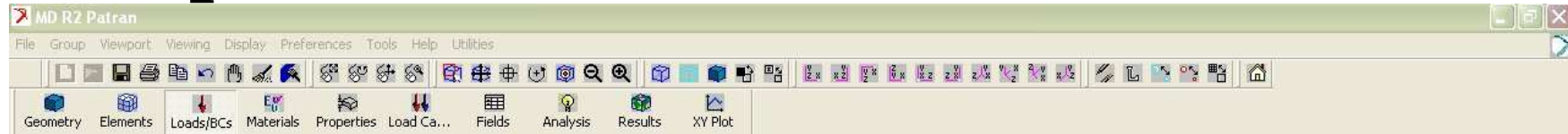


Create element properties :

- a Screen pick all the Surfaces
- b Click on **Add**.
- c Click **OK**
- d Click **Apply**
- e Verify that the properties has been created (name appears in the upper white box)

The screenshot shows the software interface with the 'Create element properties' dialog box open. The dialog box has a 'Select' dropdown set to 'Entities'. Below this, there are two 'Application Region' sections. The first section, 'Select Members', contains a list box with 'Surface 1:8'. Below this list box are 'Add' and 'Remove' buttons. The second 'Application Region' section is empty. At the bottom of the dialog box is an 'OK' button. To the right of the dialog box, there is a 'Property Sets By' section with a 'Name' dropdown. Below this is a list box containing 'Property_Shell'. Below the list box is a 'Filter' field with an asterisk. Below the filter is a 'Property Set Name' section with a text box containing 'Property_Shell'. Below this is an 'Options' section with three dropdowns: 'Thin', 'Homogeneous', and 'Standard Formulation'. Below the options are two buttons: 'Input Properties ...' and 'Select Application Region ...'. At the bottom of the entire interface is an 'Apply' button.

Step 6 : Create Boundaries Conditions

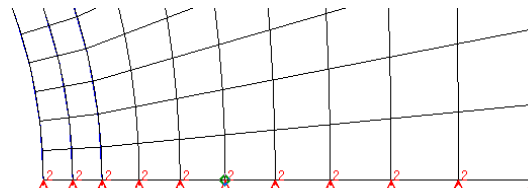
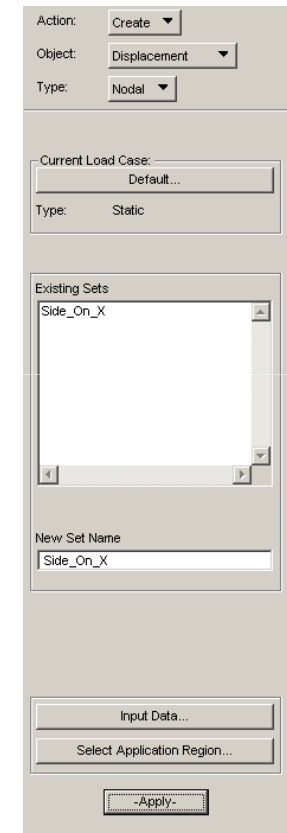
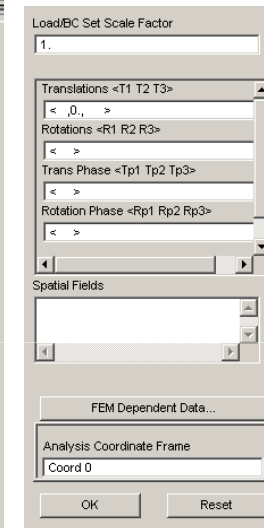
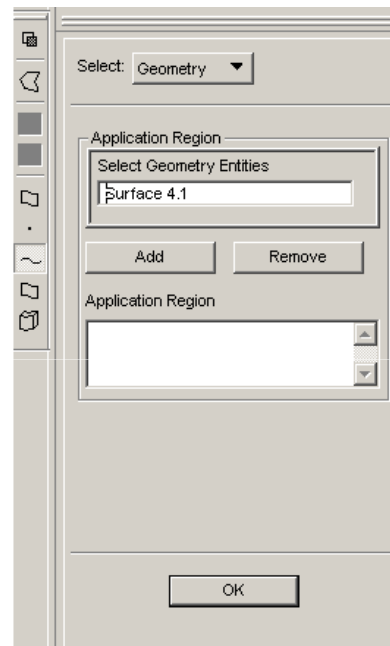


Create boundary conditions :

- a Click on **Loads/BCs : Create / Displacement / Nodal**
- b Enter **Side_On_X** as the New set Name.
- c Click on **INPUT Data**.
- d Enter **< ,0., >** for Translations.
- e Click **OK**.
- f Click on **Select Application Region**
- g Select **FEM**
- h Screen pick with the good option the nodes on the X axis
- i Click **Add**
- k Click **OK**
- l Click **Apply**
- m Verify that the boundaries Conditions have been created

Do the same thing with

a New Set named **Side_On_Y**

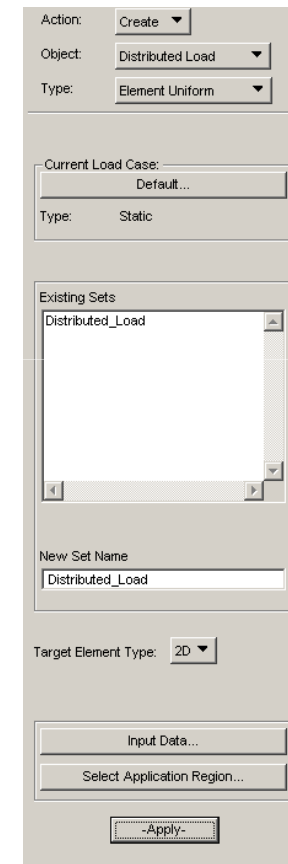
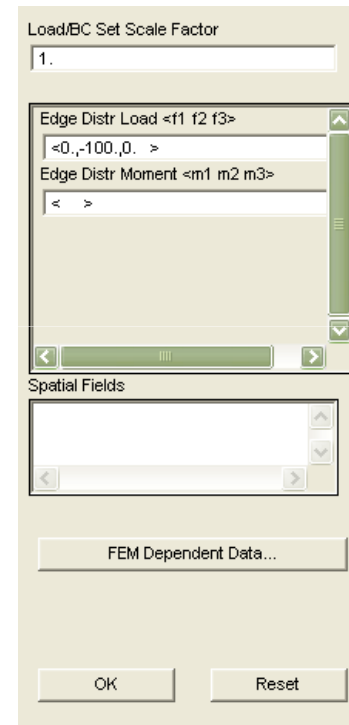


Step 7 : Create Load

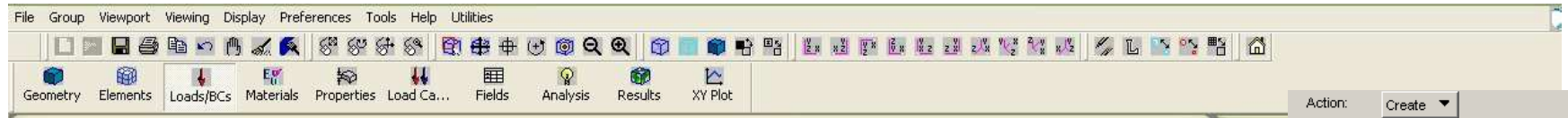


Create a constant uniform load :

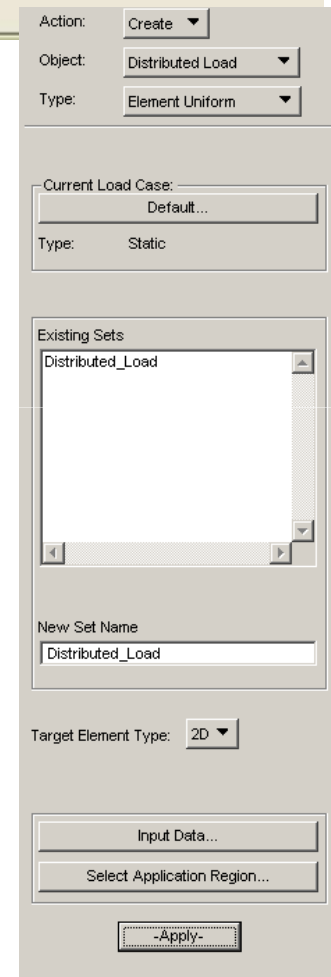
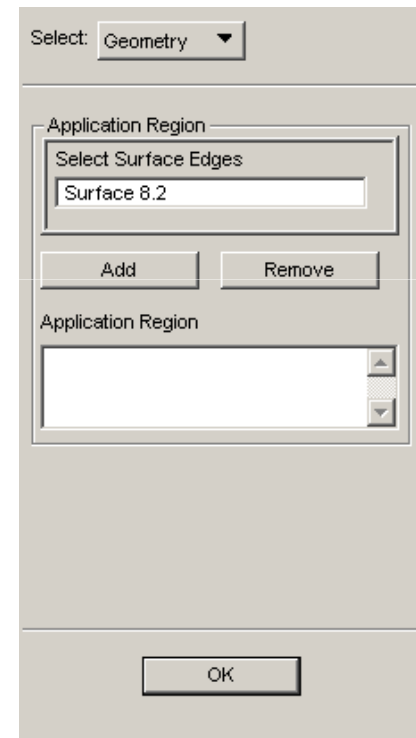
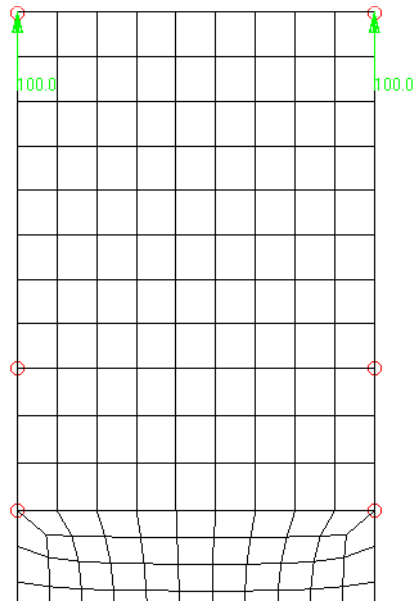
- a Loads/BCs : Create / Distributed Load / Element Uniform
- b Enter **Distributed_Load** as **New Set Name**
- c Select 2D with icon **Target Element Type**
- d Click on **INPUT Data**.
- e Enter **<0., -100., 0.>** for components of the distributed load.
- f Click **OK**
- g Click on **Select Application Region**



Step 7 : Create Load



- a Select **Geometry**
- b Select the good edge of the Surface 8 for the Load
- c Click **Add**
- d Click **OK**
- e Click **Apply**
- f Verify that the load has been created in the field **Existing Sets**

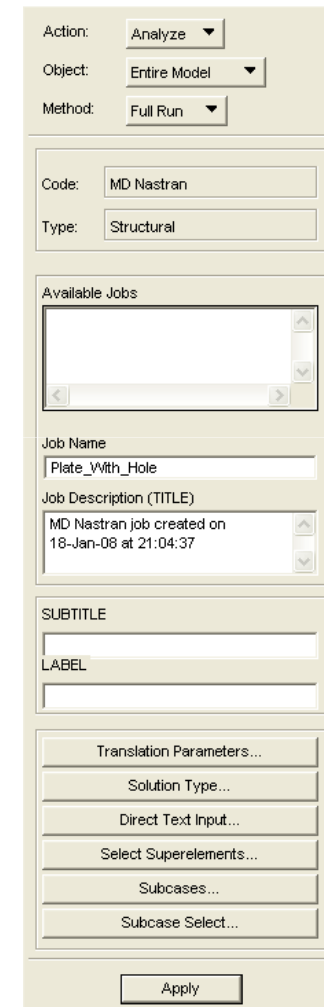
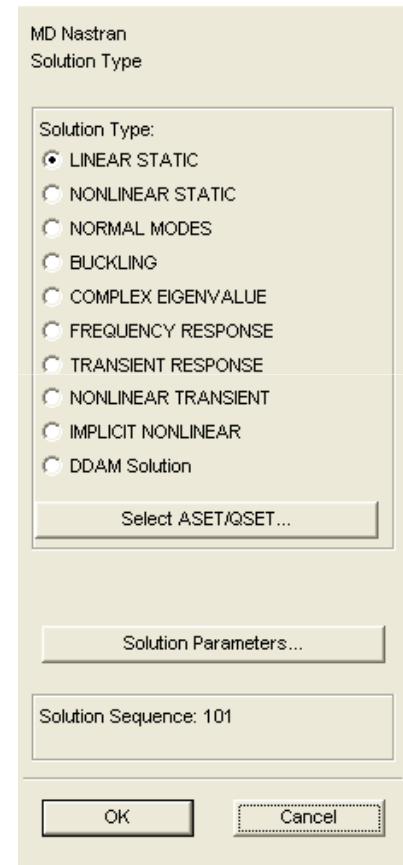


Step 8 : Run Nastran



Computation

- a Analysis : **Analyze / Entire Model / Full Run**
- b Select **Solution Type**
- c Select **Linear Static**
- d Click **OK**
- f Click **Apply**

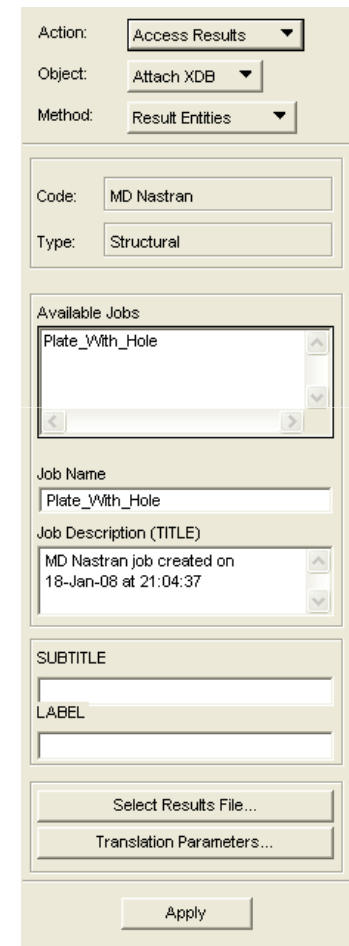
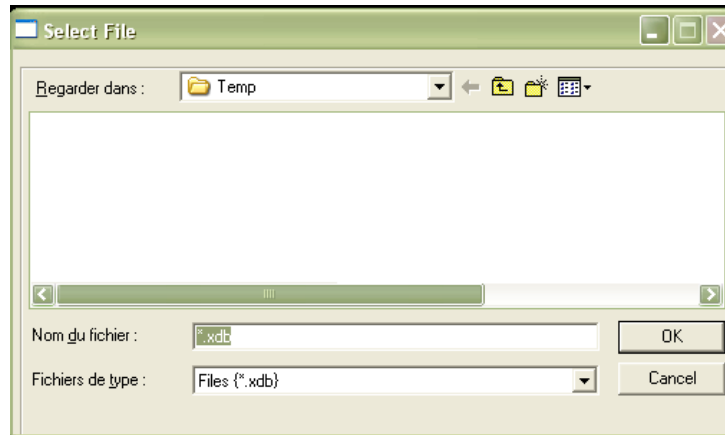


Step 8 : Load Nastran Results in Patran



Load the results file of NASTRAN

- a Analysis : **Access Results / Attach XDB / Result Entities**
- b Click on **Select Results File**
- c Select the file results **Plate_With_Hole.XDB**
- d Click **OK**
- e Click **Apply**



Analysys of Files *.F06 and *.BDF

- Open the file Plate_With_Hole.F06
- Verify the run and the results
- Open the file Plate_With_Hole.XDB
- Verify the NASTRAN Input file

Verification

The model doesn't perform !!

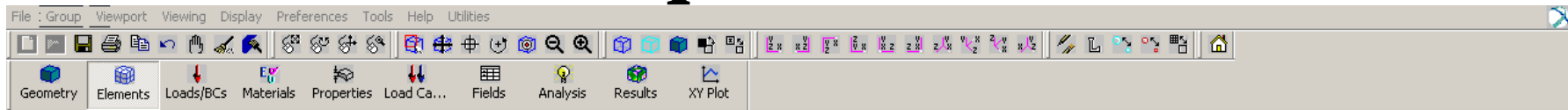
You have to find a solution to solve the problem !!

Modify the modelling and run it again.

Where did the errors come from

- Verify if the equivalence has been made,
- Verify if there are rigid body movements, and removed them if necessary,
- Verify if the properties of the elements are the good one,
- Verify if the load has been applied,
- Verify if the boundary conditions are sufficient for the elements choosed for the modelling.

Equivalence



Never forget to equivalence the nodes of your model :

a Elements : Equivalence / All / Tolerance Cube

b Click **Apply**

**Run the model again, Load the file results *.XDB, and verify the
*.F06 and *.BDF files**

A screenshot of the 'Equivalence' dialog box in ANSYS Workbench. The dialog has a light gray background and contains the following controls:

- Action:** A dropdown menu with 'Equivalence' selected.
- Object:** A dropdown menu with 'All' selected.
- Method:** A dropdown menu with 'Tolerance Cube' selected.
- Node Id Options:** A dropdown menu with 'Retain lower node id' selected.
- Collapsed Node Options:** A dropdown menu with 'Allow Tolerance Reduction' selected.
- Nodes to be excluded:** A text input field that is currently empty.
- Equivalencing Tolerance:** A text input field with the value '0.2' entered.
- Apply-:** A button at the bottom right of the dialog.

Rigid Body movement

Remove all the rigid body movements:

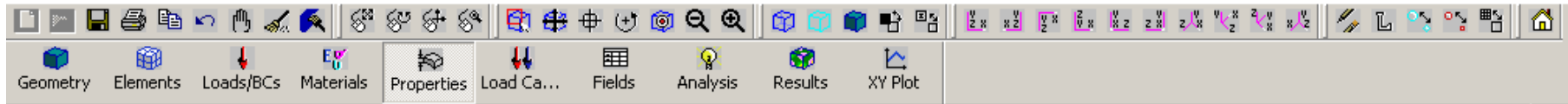
If you have a plane problem, like the one studied, don't forget that NASTRAN solve a 3 dimensional problem

Boundary Conditions

Don't forget the degrees of freedom of the element choosed :

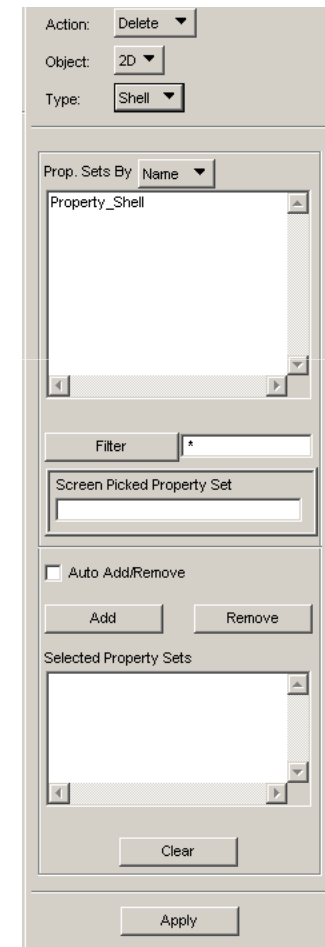
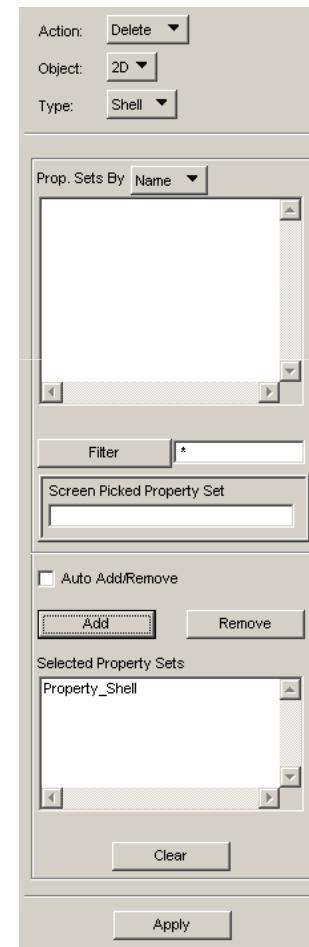
	Membrane Element	Shell Element
Local Axis of the element	3 DOF	5 DOF
Global Axis	3 DOF	6 DOF
DOF for Translation	TX, TY, TZ	TX, TY, TZ
DOF for Rotation	RX, RY, RZ	RX, RY, RZ

Shell and Membrane Elements



When the model will be good, change the Shell elements by Membrane elements and compare the boundary conditions needed for each type of elements. Before creating new properties (see previous pages) you must erase the old, like this :

- a Properties: **Delete / 2D / Shell**
- b Select the properties by the name in the good box.
- c Click **ADD**
- d Verify the name properties is in the box Selected property Sets
- e Click **Apply**
- f Create Membrane properties as described in previous pages



Analysis of the results

Before creating images, graph and data files to store the results you must :

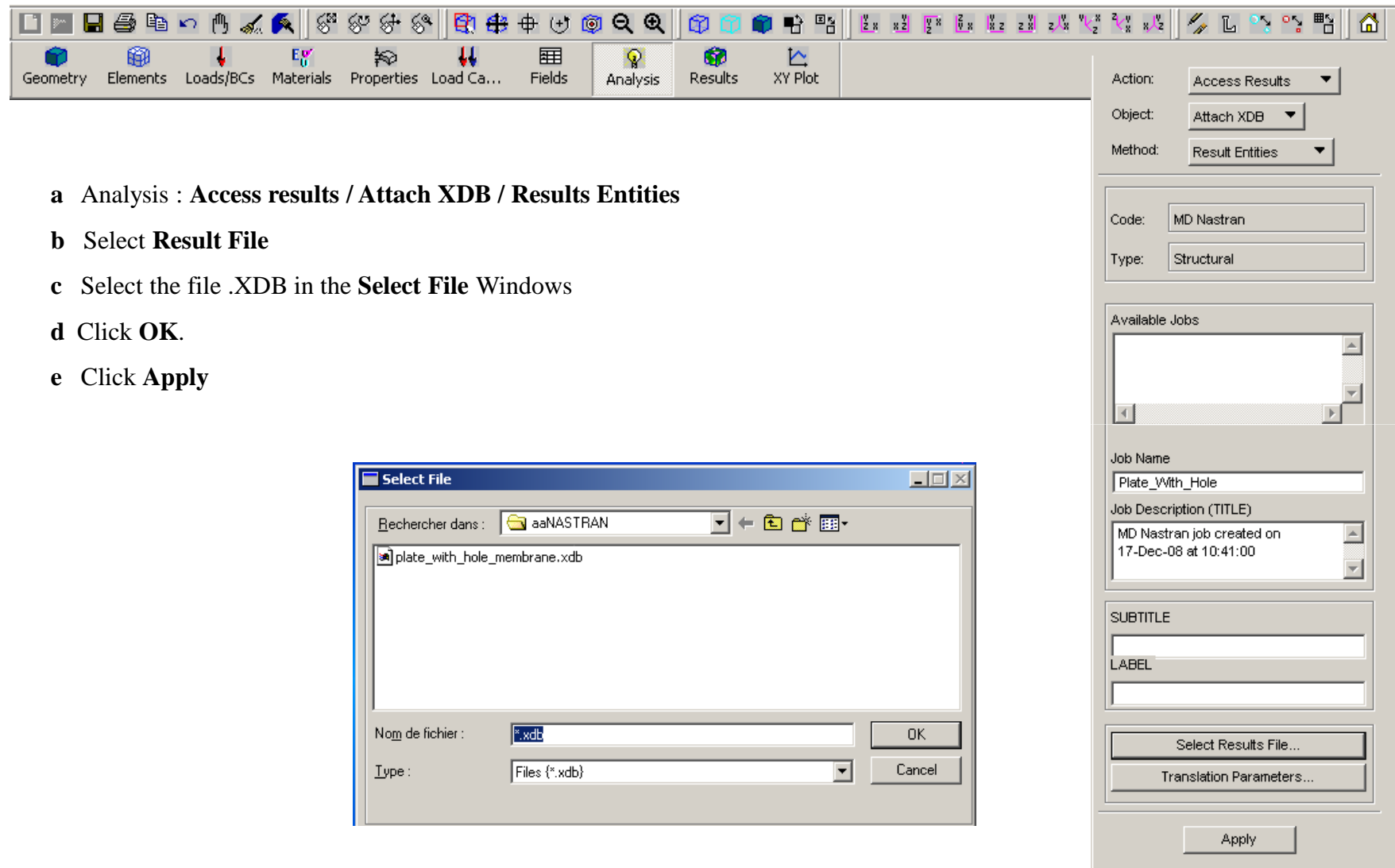
- * Open the BDF file to verify if your modelling seem good
- * Open the F06 file to verify if your model has been computed correctly, with the good accuracy.

After that you can import a specific output file inside Patran to analyse the results. Two different output files can be generated by NASTRAN :

- *. XDB output file (used at ISAE)
- *.OP2

To import the output file in Patran see the following slide

Import Output BDF file in PATRAN



The image displays the PATRAN software interface. The top toolbar includes icons for Geometry, Elements, Loads/BCs, Materials, Properties, Load Ca..., Fields, Analysis, Results, and XY Plot. The Analysis menu is open, showing options: Access Results, Attach XDB, Result Entities, Code: MD Nastran, and Type: Structural. Below the menu, the 'Available Jobs' list is empty. The 'Job Name' field contains 'Plate_With_Hole', and the 'Job Description (TITLE)' field contains 'MD Nastran job created on 17-Dec-08 at 10:41:00'. The 'SUBTITLE' and 'LABEL' fields are empty. At the bottom, there are buttons for 'Select Results File...', 'Translation Parameters...', and 'Apply'.

a Analysis : **Access results / Attach XDB / Results Entities**

b Select **Result File**

c Select the file .XDB in the **Select File** Windows

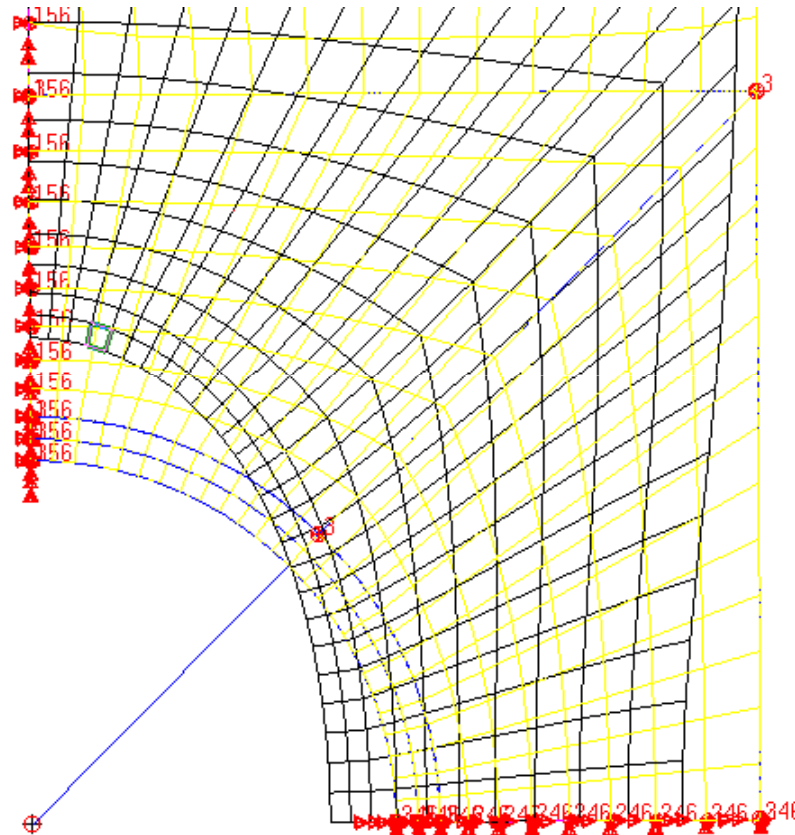
d Click **OK**.

e Click **Apply**

The **Select File** dialog box is shown, displaying the file **plate_with_hole_membrane.xdb** in the **Rechercher dans :** **aaNASTRAN** folder. The **Nom de fichier :** field shows ***.xdb**, and the **Type :** dropdown is set to **Files (*.xdb)**. The **OK** and **Cancel** buttons are visible.

File Group Viewport Viewing Display Preferences Tools Help Utilities

Geometry Elements Loads/BCs Materials Properties Load Ca... Fields Analysis Results XY Plot



- a Results : Create / Deformation**
- b Select Displacements, Translational in **Select Deformation Result****
- c Click **Apply****

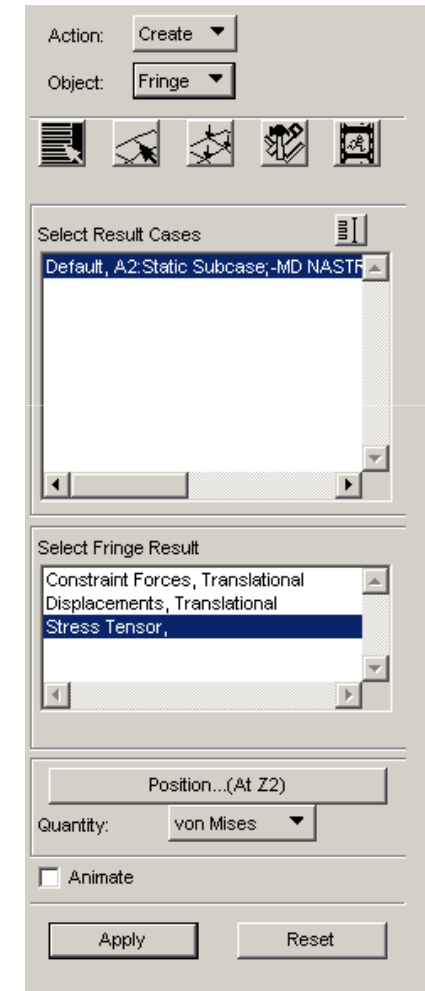
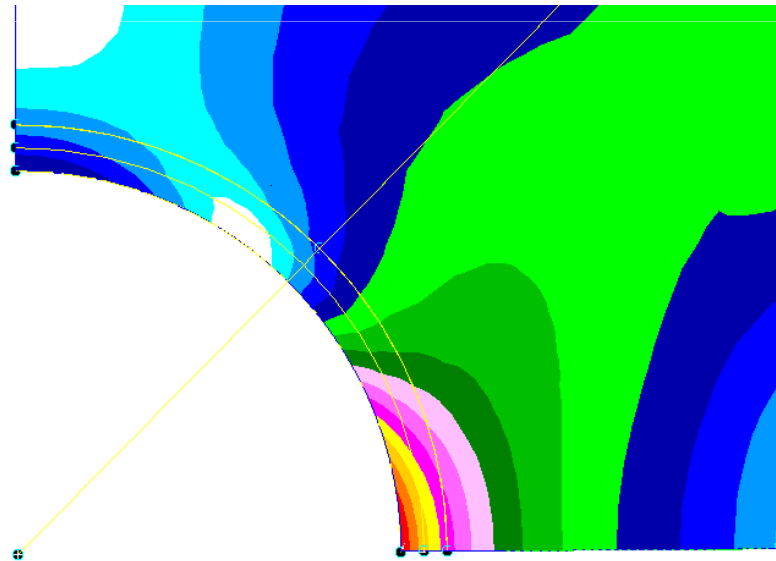


Step 9 : Results - Von Mises



Verify the Von Mises' Criteria :

- a Results : Create / Fringe
- b Select Stress Tensor in **Select Fringe Result**
- c Select **Von Mises** fo Quantity
- d Click **Apply**

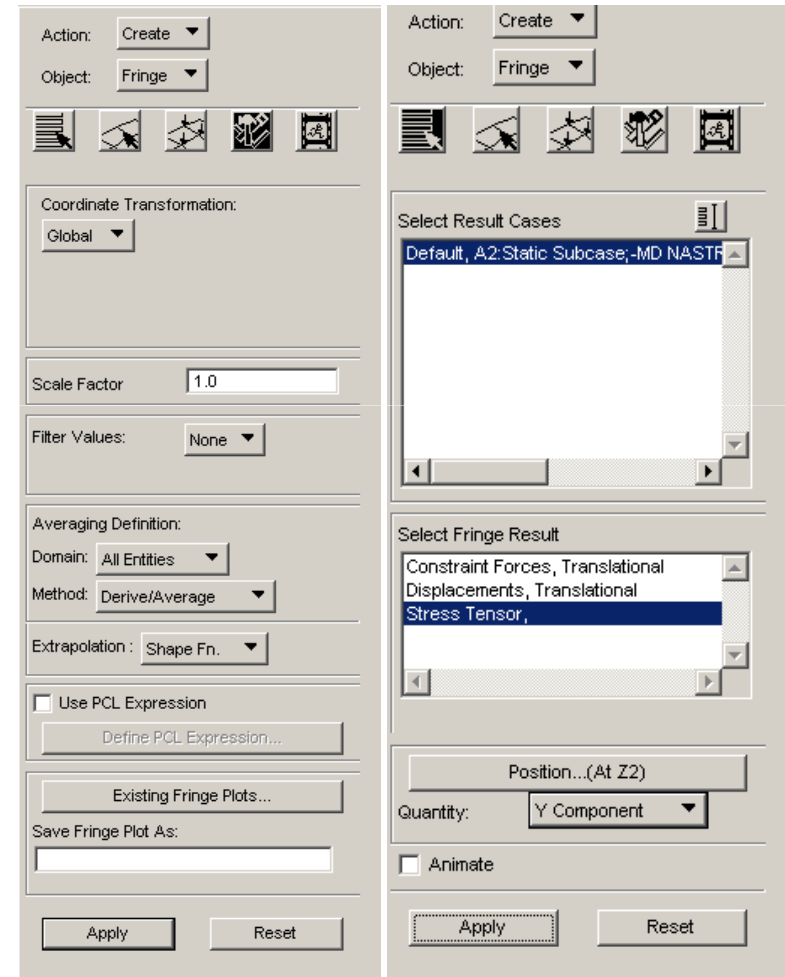
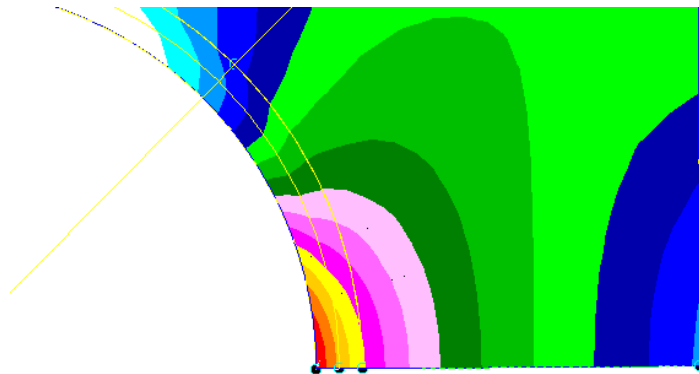


Step 9 : Results - σ_x



Verify the Von Mises' Criteria :

- a Results : Create / Fringe
- b Select Stress Tensor in **Select Fringe Result**
- c Select **Y Component** for Quantity
- d Select the Icon **Plot Options**
- e Select **Global** as **Coordinate Transformation**
- d Click **Apply**

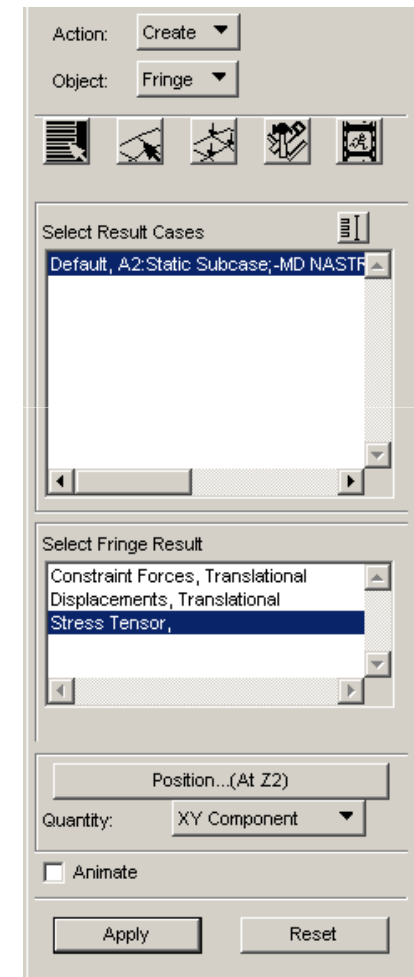
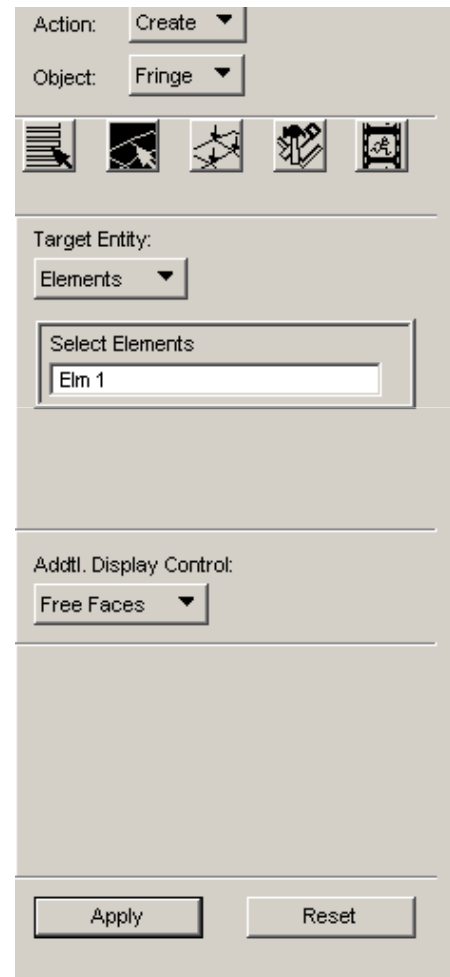
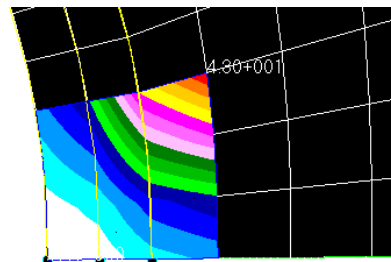


Step 9 : Local Results – σ_x

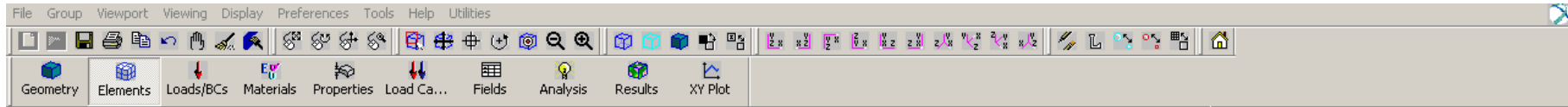


Display a local area for the shear stress :

- a Results : Create / Fringe
- b Select icon **Select Results**
- c Select Stress Tensor in **Select Fringe Result**
- d Select **XY Component**
- e Select the Icon **Target Entities**
- f Select Elements as **Target Entity**
- g Select or Screen Pick the elements you want visualize
- e Click **Apply**



Results : Stress in an Element



Identify one (or More) element(s) :

- Elements: Show / Element / Attributes
- Screen Select the element near the hole
- Click **Apply**
- Read the identification number of the element
- Open the file job.F06
- Find the data of the element(s)
- Analyse the results

Show Element Attributes Information

Element ID	Topology	Parent Geom	Elem Nodes	Loads/BCs	Material	Property Set	Results
1	Quad4	Surface 1	4	0	Steel	Property_Shell	?
1	Quad4	Surface 1	4	0	Steel	Property_Shell	?

Action: Show ▼

Object: Element ▼

Info: Attributes ▼

Element Summary

Last ID: 431

Total in Model: 431

Total in 'default_group': 431

☐ Write to Report

☒ Auto Execute

Element List

Elm 1

Apply

Results File Job.F06 (Point A)

ELEMENT		FIBER	STRESSES	IN ELEMENT CO	ORD SYSTEM	PRINCIP	AL STRESSES (2	ERO SHEAR)	
ID	GRID-ID	DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-01	4.013723E+02	1.076666E+01	1.475449E+00	0.2164	4.013778E+02	1.076109E+01	3.961069E+02
		5.000000E-01	4.013723E+02	1.076666E+01	1.475449E+00	0.2164	4.013778E+02	1.076109E+01	3.961069E+02
1		-5.000000E-01	4.366468E+02	1.044953E+01	1.475600E+00	0.1984	4.366519E+02	1.044442E+01	4.315245E+02
		5.000000E-01	4.366468E+02	1.044953E+01	1.475600E+00	0.1984	4.366519E+02	1.044442E+01	4.315245E+02
2		-5.000000E-01	4.366471E+02	1.108379E+01	1.475603E+00	0.1987	4.366523E+02	1.107867E+01	4.312196E+02
		5.000000E-01	4.366471E+02	1.108379E+01	1.475603E+00	0.1987	4.366523E+02	1.107867E+01	4.312196E+02
29		-5.000000E-01	3.667758E+02	1.106584E+01	1.475300E+00	0.2376	3.667819E+02	1.105972E+01	3.613790E+02
		5.000000E-01	3.667758E+02	1.106584E+01	1.475300E+00	0.2376	3.667819E+02	1.105972E+01	3.613790E+02
28		-5.000000E-01	3.667761E+02	1.046748E+01	1.475297E+00	0.2372	3.667822E+02	1.046138E+01	3.616650E+02
		5.000000E-01	3.667761E+02	1.046748E+01	1.475297E+00	0.2372	3.667822E+02	1.046138E+01	3.616650E+02

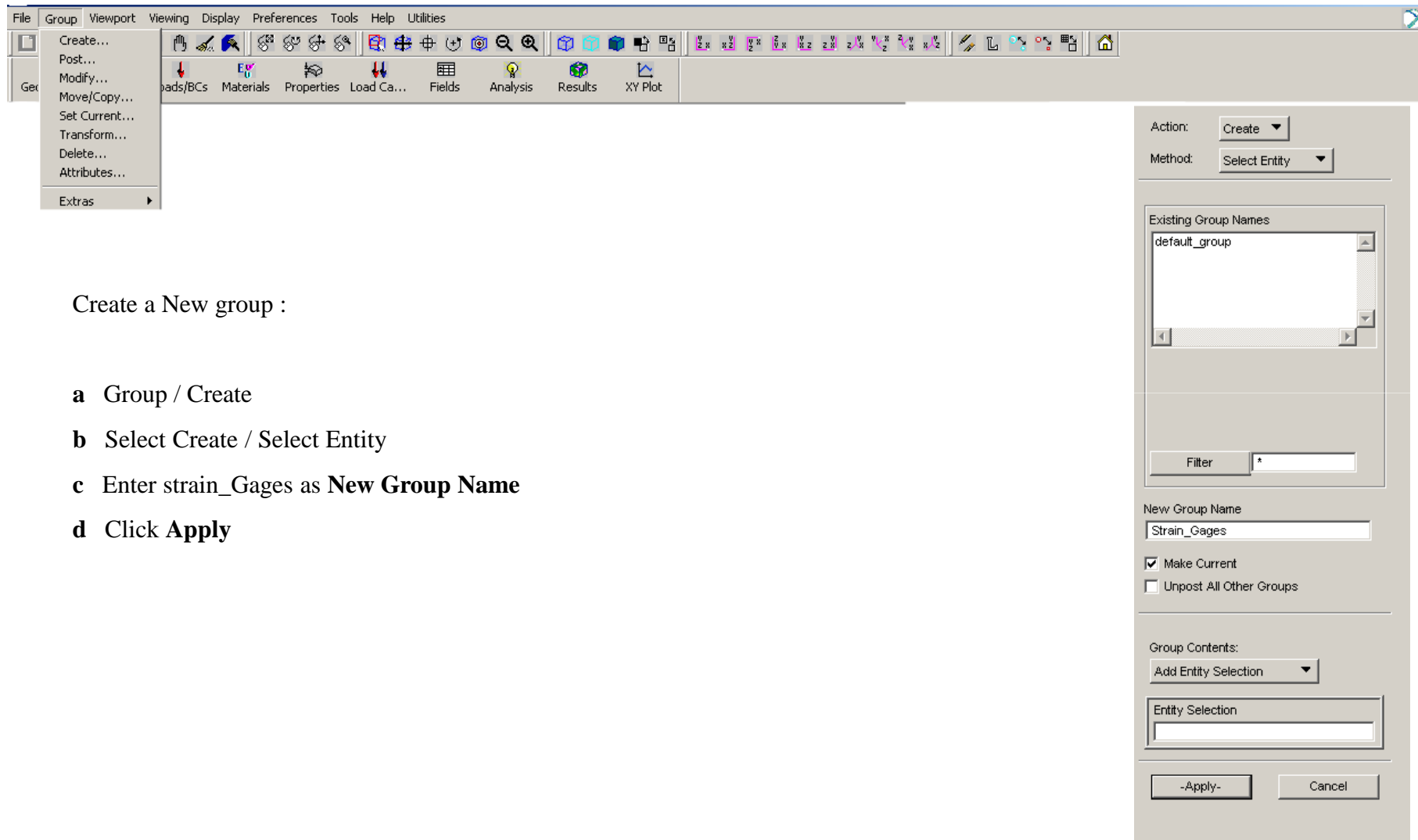
Results File Job.F06 (Point A) (Excell)

ELEMENT		FIBER	STRESSES IN ELEMENT COOR SYSTEM			PRINCIPAL STRESSES (ZERO SHEAR)			
ID	GRID-ID	DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5,00E-01	401,4	11	1	0,2	401	11	396
		5,00E-01	401	11	1	0,2	401	11	396
1		-5,00E-01	437	10	1	0,2	437	10	432
		5,00E-01	437	10	1	0,2	437	10	432
2		-5,00E-01	437	11	1	0,2	437	11	431
		5,00E-01	437	11	1	0,2	437	11	431
29		-5,00E-01	367	11	1	0,2	367	11	361
		5,00E-01	367	11	1	0,2	367	11	361
28		-5,00E-01	367	10	1	0,2	367	10	362
		5,00E-01	367	10	1	0,2	367	10	362

Strain gauges on the edge of the hole

- Create a new group,
- Create ROD elements on the edge of the hole,
- Create properties for these ROD elements,
- Run a new Job,
- Load the results and analyse the results in the ROD elements,
- Draw the curve of σ_{θ} on the edge of the hole

Creation of a New Group



Create a New group :

- a Group / Create
- b Select Create / Select Entity
- c Enter strain_Gages as **New Group Name**
- d Click **Apply**

Creation of a New Mesh

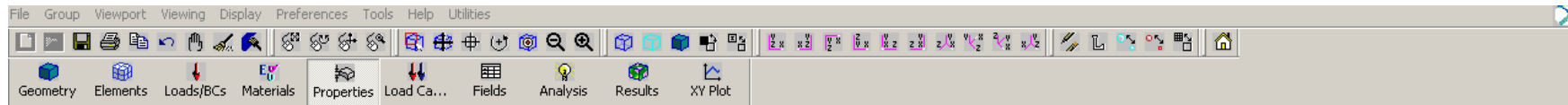


Create a new Mesh :

- a **Elements** : Create / Mesh / Curve
- b Select **Bar2** as **Topology**
- c Unselect **Automatic Calculation**
- d Enter 3 as **Value**
- e Verify that 26 elements have been created

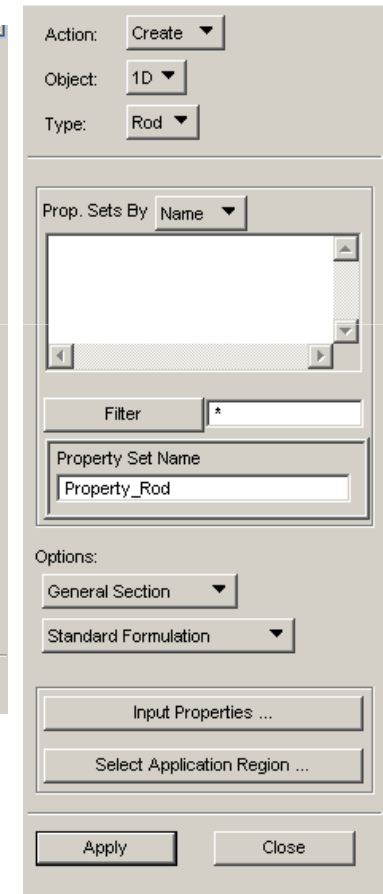
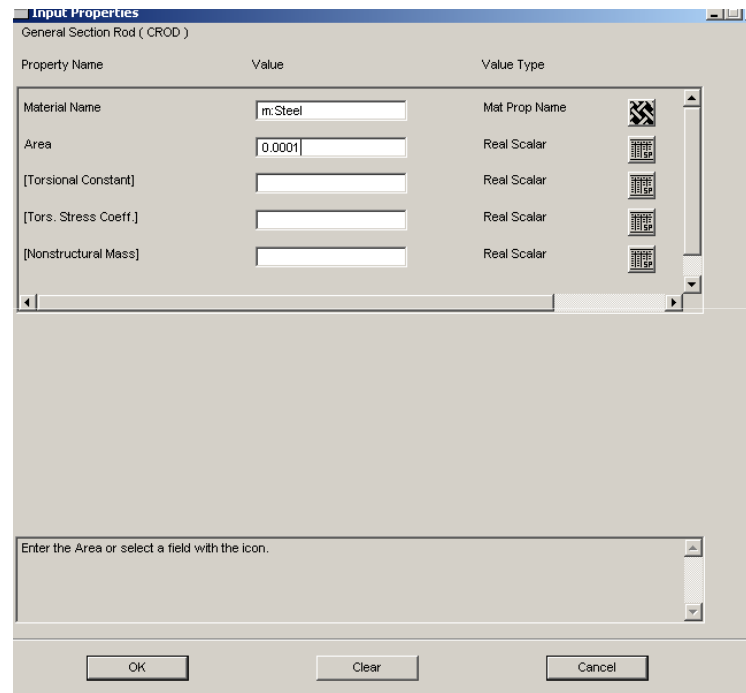
The image shows the 'Create Mesh' dialog box in ANSYS Workbench. It has a light gray background and a standard Windows-style layout. At the top, there are three dropdown menus: 'Action' set to 'Create', 'Object' set to 'Mesh', and 'Type' set to 'Curve'. Below these is a section for 'Output ID List' with two input fields: 'Node' containing '589' and 'Element' containing '432'. Underneath is a 'Topology' dropdown menu set to 'Bar2'. A button labeled 'Node Coordinate Frames...' is located below the topology menu. The next section is 'Curve List' with a text box containing 'Curve 1'. Below that is a 'Global Edge Length' section with an unchecked checkbox for 'Automatic Calculation' and a 'Value' input field containing '3'. At the bottom of the dialog are two buttons: 'Select Existing Prop...' and 'Create New Property...'. A large '-Apply-' button is at the very bottom.

Create New Properties

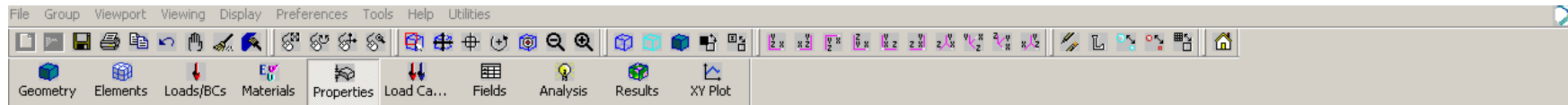


Create element properties :

- a Properties : Create / 1D / Rod
- b Enter Property_Rod as the name of **Property Set Name**
- c Select **Input Properties**
- d Select Steel as Material
- e Enter 0.0001 for **Area**
- f Click **OK**
- g Select **Application Region**

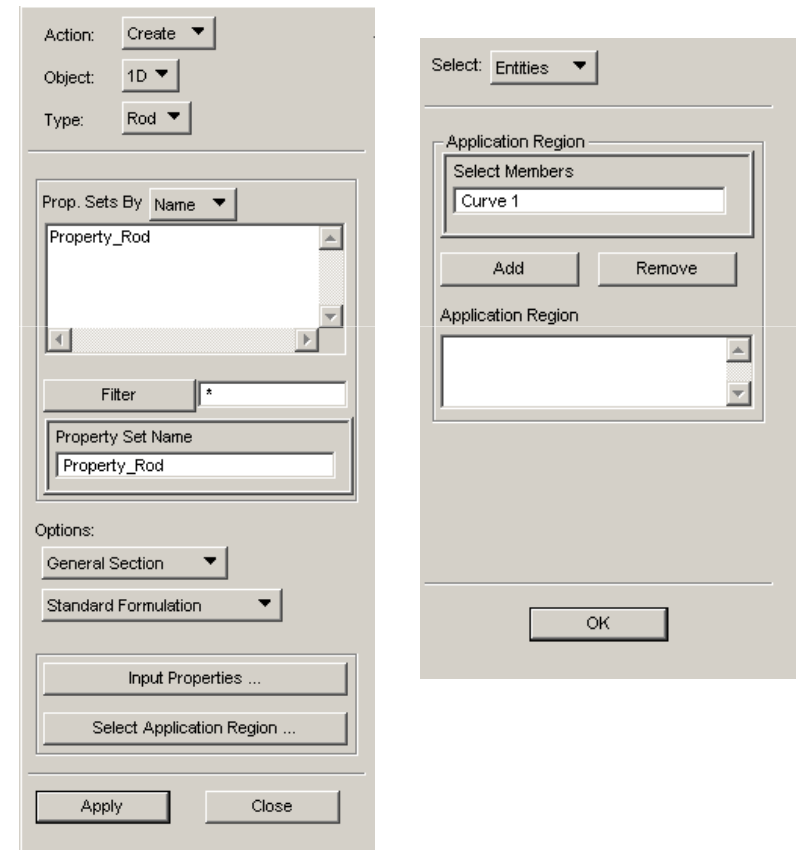
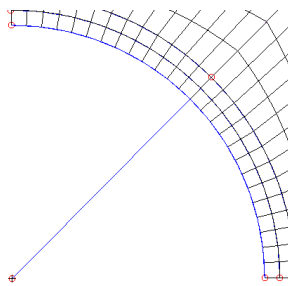


Create New Properties



Create element properties :

- a Activate the Field **Select Member**
- b Screen Select the Curve 1
- b Click **ADD**
- c Click **OK**
- d Click **Apply**
- e Verify that the Property has been created



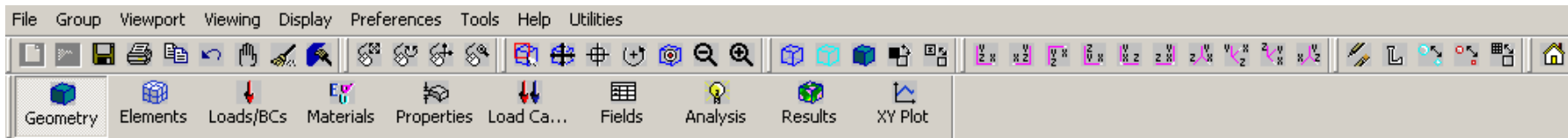
Results : Creation of images

- Create a Cylindrical axis system to analyse the stresses on the edge of the hole,
- Create an image of σ_{θ} near the hole,
- Create an image of σ_r near the hole,
- Create an image of $\tau_{r\theta}$ near the hole,
- Refine the mesh near the hole to increase the accuracy

Results : Creation of images

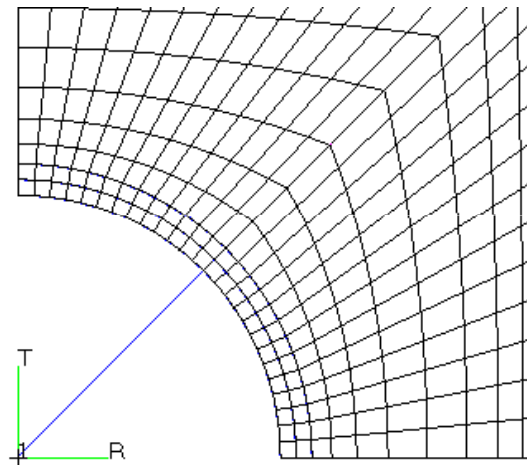
- Analyse the stresses in an element with :
- The local axis of the element,
- The global axis of the plate
- The cylindrical axis,

Create a Cylindrical Axis System



There are several solutions to create a cylindrical axis system, for instance :

- a Geometry : **Create / Curve / Coord / 3Point**
- b Select **Cylindrical** as Type
- c With the **KeyBoard** enter the coordinates of the center of the axis system : **[0 0 0]**
- d With the **KeyBoard** enter the coordinates of a p on the third axis : **[0 0 1]**
- e With the **KeyBoard** enter the coordinates of a po on the plane 1-3 : **[1 0 0]**
- f Click **Apply**
- g Verify that the axis system has been created



Action: **Create** ▼

Object: **Coord** ▼

Method: **3Point** ▼

Coord ID List
2

Type: **Cylindrical** ▼

Refer. Coordinate Frame
Coord 0

☐ Auto Execute

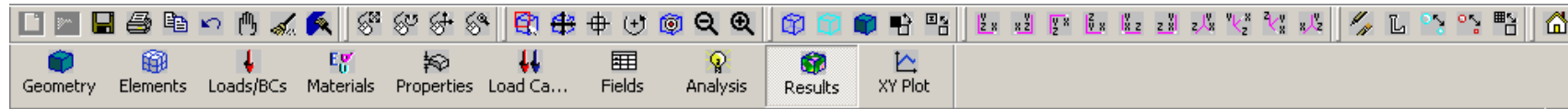
Origin
[0 0 0]

Point on Axis 3
[0 0 1]

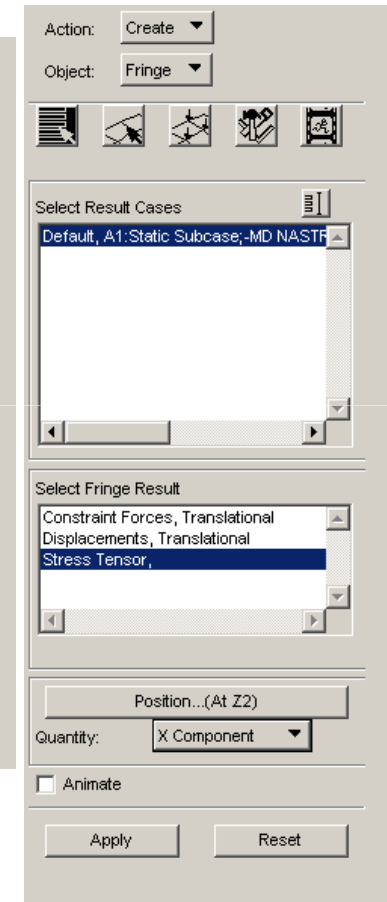
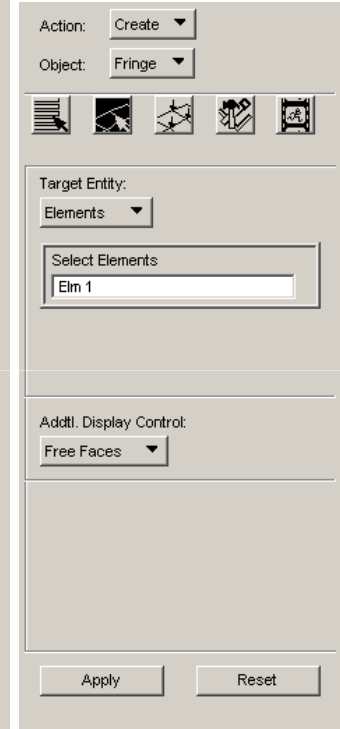
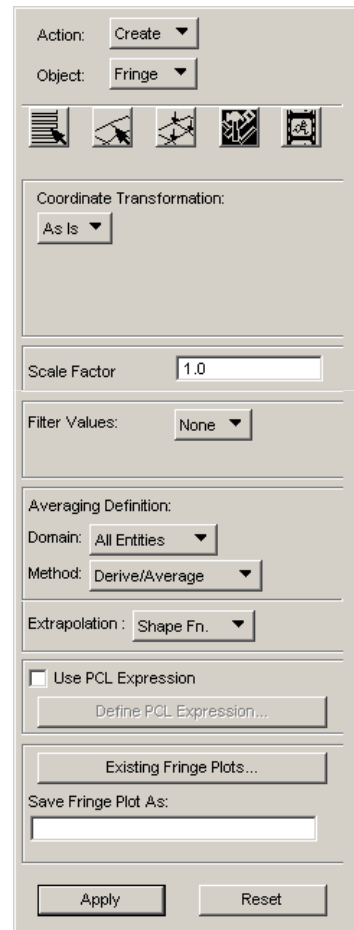
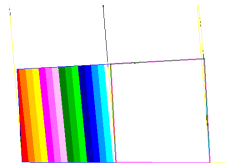
Point on Plane 1-3
[1 0 0]

-Apply-

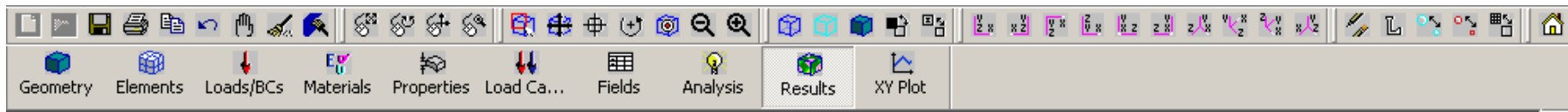
Analysis in the local axis system



- a Results : **Create / Fringe**
- b Select the Icon **Select results**
- c Select Stress tensor in **Field Fringe Result**
- d Select the quantity X component for instance
- e Click **Apply**
- f Select the Icon **Target Entities**
- g Select Elements as **Target Entity**
- h Select the element on the model
- i Click **Apply**
- j Select the Icon **Plot options**
- k Select **As is** as Coordinate Transformation
- l Click **Apply**

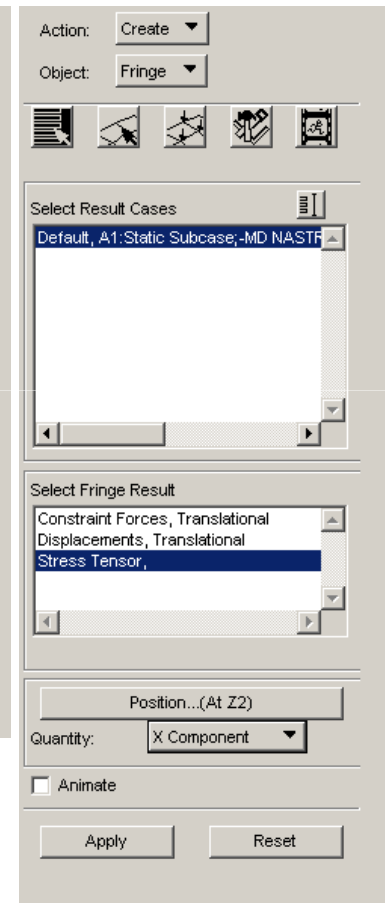
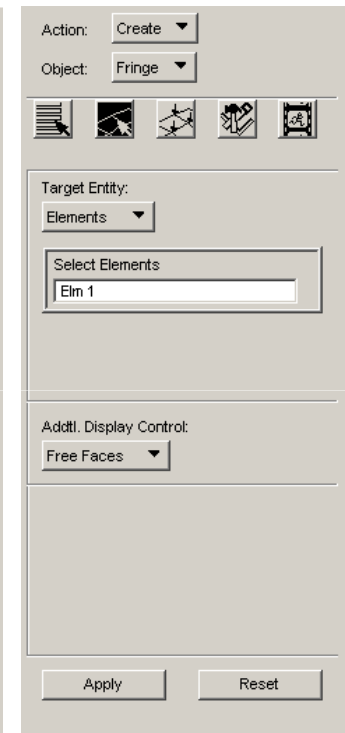
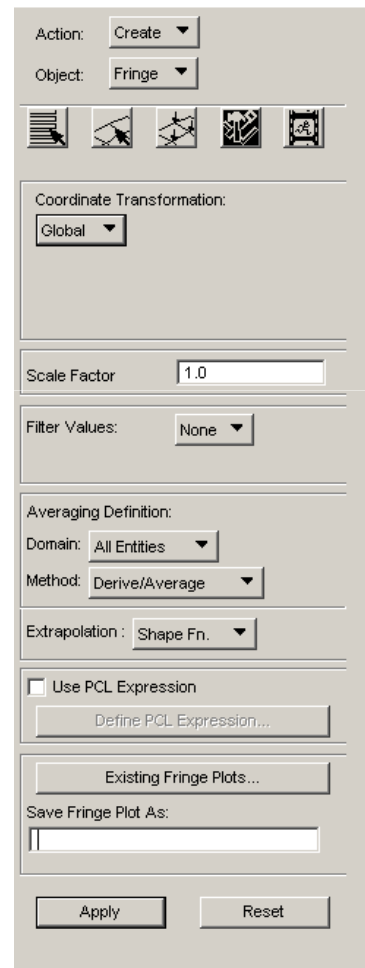


Analysis in the global axis system

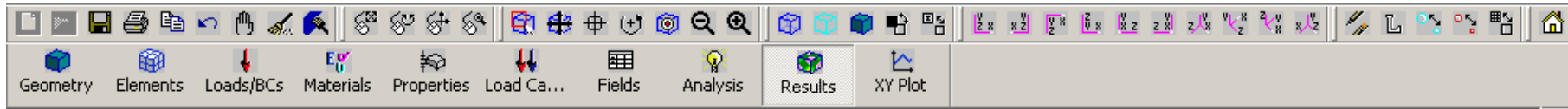


REMARK : The steps a to j have already been done in the previous slide. Go directly to the point k.

- a Results : **Create / Fringe**
- b Select the Icon **Select results**
- c Select Stress tensor in **Field Fringe Result**
- d Select the quantity X component for instance
- e Click **Apply**
- f Select the Icon **Target Entities**
- g Select Elements as **Target Entity**
- h Select the element on the model
- i Click **Apply**
- j Select the Icon **Plot options**
- k Select **Global** as Coordinate Transformation
- i Click **Apply**

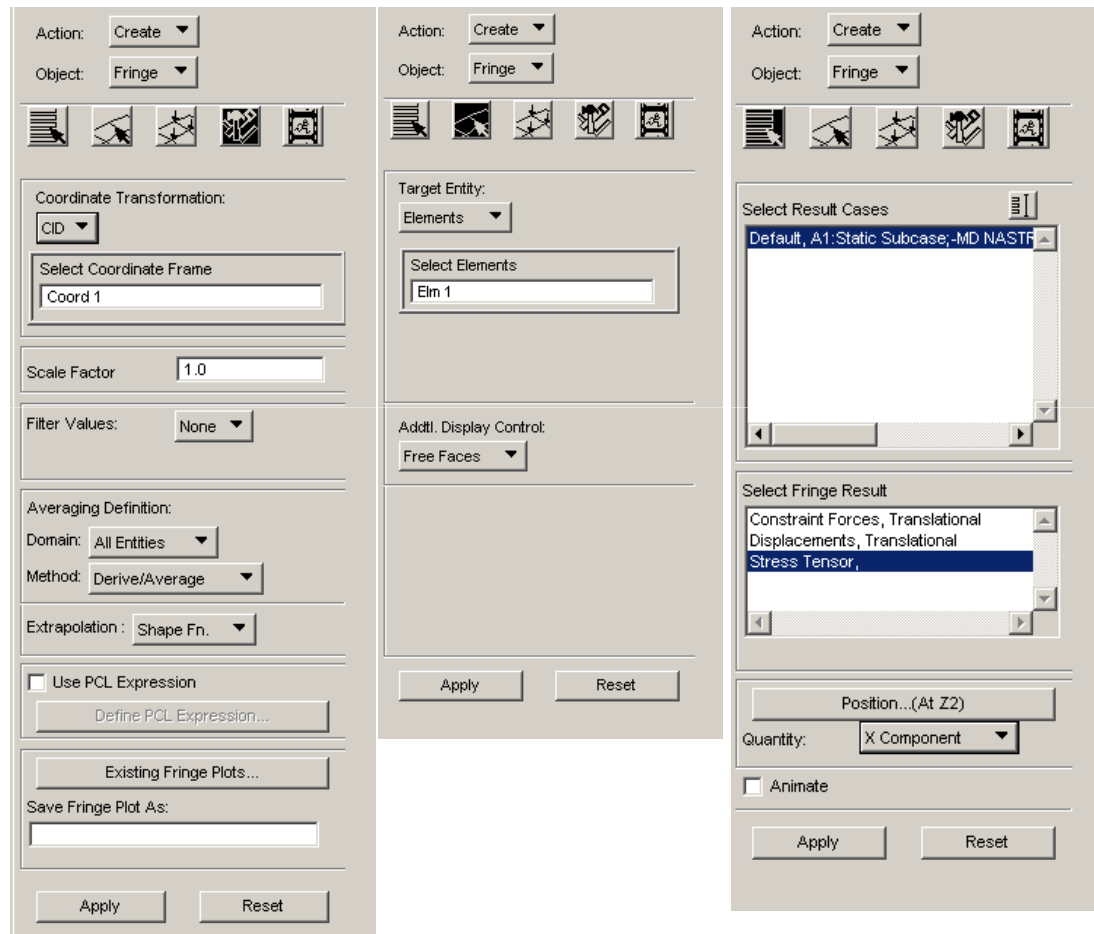


Analysis in the Cylindrical axis system

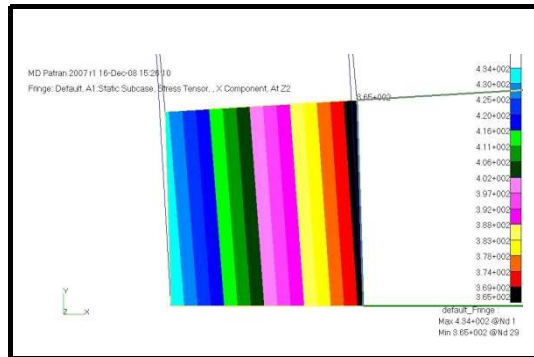


REMARK : The steps a to j have already been done in the previous slide. Go directly to the point k.

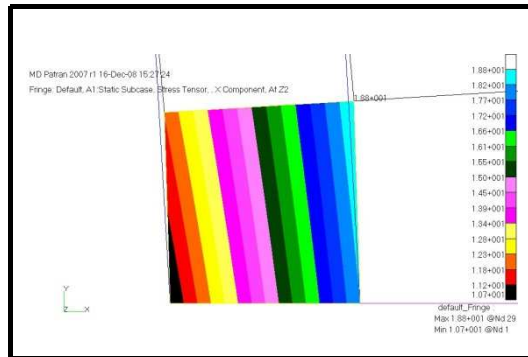
- a Results : **Create / Fringe**
- b Select the Icon **Select results**
- c Select Stress tensor in **Field Fringe Result**
- d Select the quantity X component for instance
- e Click **Apply**
- f Select the Icon **Target Entities**
- g Select Elements as **Target Entity**
- h Select the element on the model
- i Click **Apply**
- j Select the Icon **Plot options**
- k Select **CID** as Coordinate Transformation
- l Select the axis system on the model
- m Click **Apply**



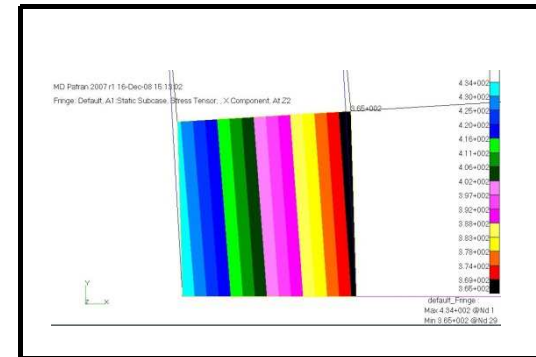
Comparison of the different results



Local Axis System



Global Axis System



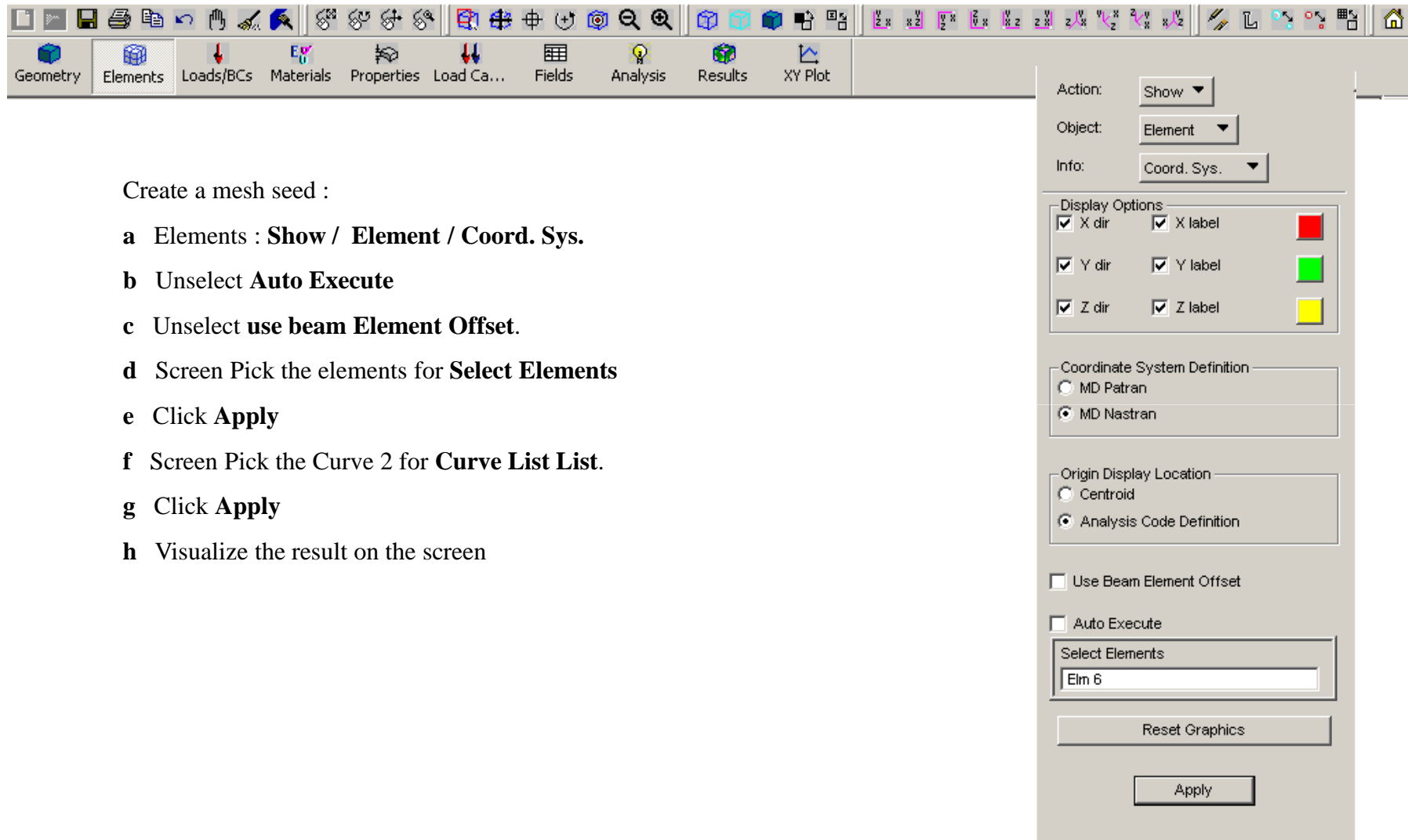
Cylindrical Axis System

Local Axis of the shell elements

In the output file of Nastran all the results are calculated and printed in the local axis system of the element. It's very hard to visualize the field of stresses from these results. The best solution is to ask the computer output result in a specific axis system.

To visualize the local axis system of each element process like explained in the following pages

Local axis of the Shell Elements



The screenshot shows the Patran software interface. The 'Elements' menu is open, and the 'Show / Element / Coord. Sys.' option is selected. The dialog box for 'Show / Element / Coord. Sys.' is displayed, showing the following settings:

- Action: Show
- Object: Element
- Info: Coord. Sys.
- Display Options:
 - ☒ X dir, ☒ X label (Red)
 - ☒ Y dir, ☒ Y label (Green)
 - ☒ Z dir, ☒ Z label (Yellow)
- Coordinate System Definition:
 - ☐ MD Patran
 - ☒ MD Nastran
- Origin Display Location:
 - ☐ Centroid
 - ☒ Analysis Code Definition
- ☐ Use Beam Element Offset
- ☐ Auto Execute
- Select Elements: Elm 6
- Reset Graphics
- Apply

Create a mesh seed :

- Elements : **Show / Element / Coord. Sys.**
- Unselect **Auto Execute**
- Unselect **use beam Element Offset.**
- Screen Pick the elements for **Select Elements**
- Click **Apply**
- Screen Pick the Curve 2 for **Curve List List.**
- Click **Apply**
- Visualize the result on the screen