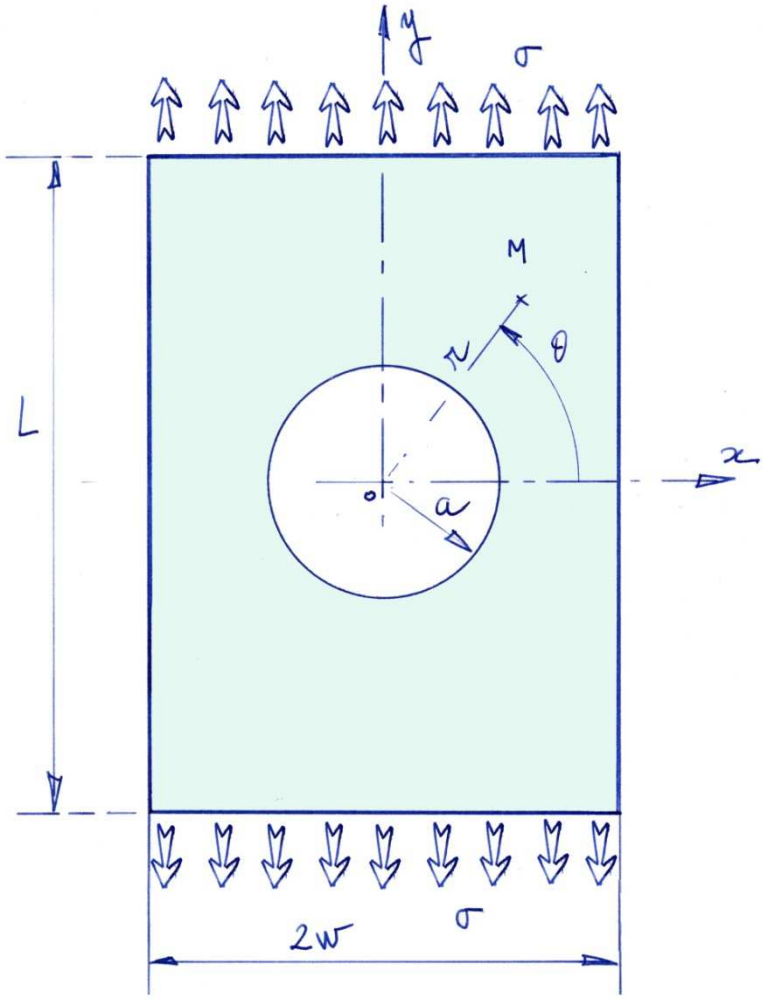
Objectives of the WorkShop

- 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**
- **Lenght L = 600 mm**
- **Width 2W = 200 mm**
- **Drilling a = 50 mm**

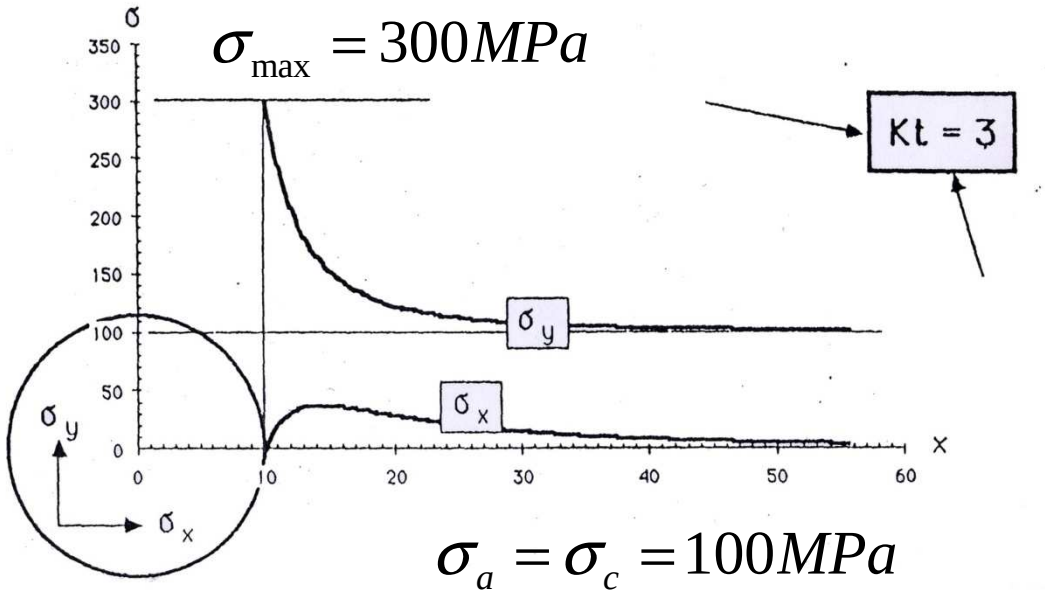
The load is a uniform tensile stress of 100 Mpa on two opposite edges



Particular Case for an infinite plate (r<<W)

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

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



$$\sigma_{\max} = 300MPa$$

$$\sigma_a = \sigma_c = 100MPa$$

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 accurates.

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

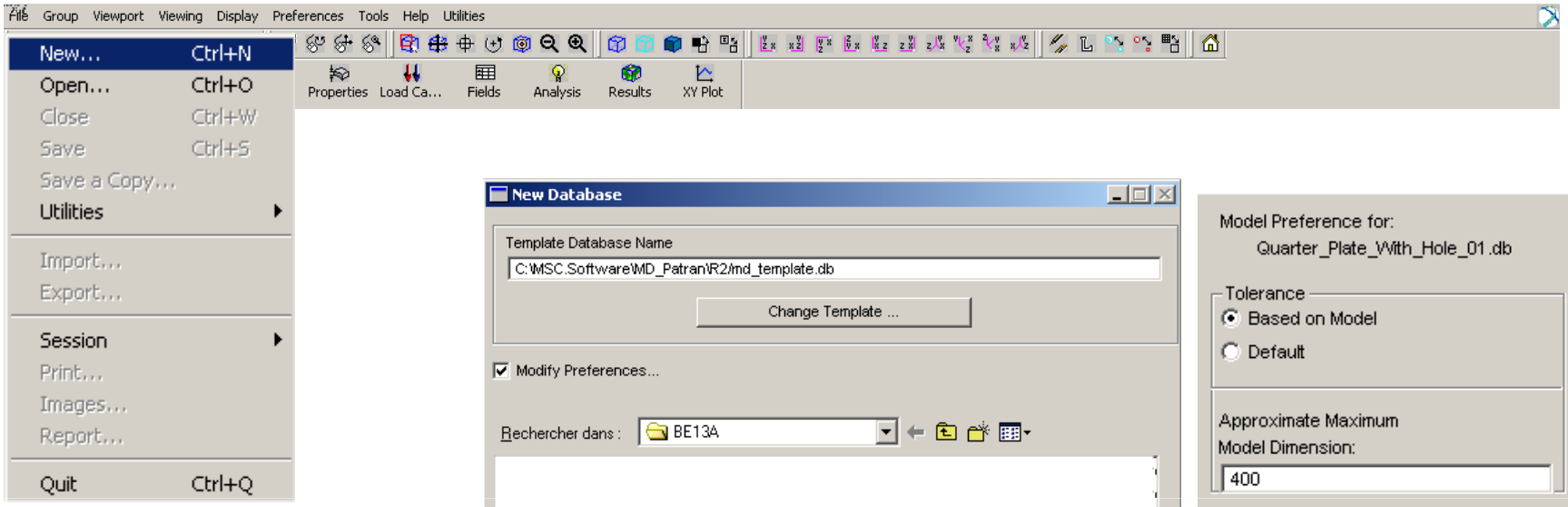
$$\sigma^{Max} = K_t \sigma_{net}$$

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**

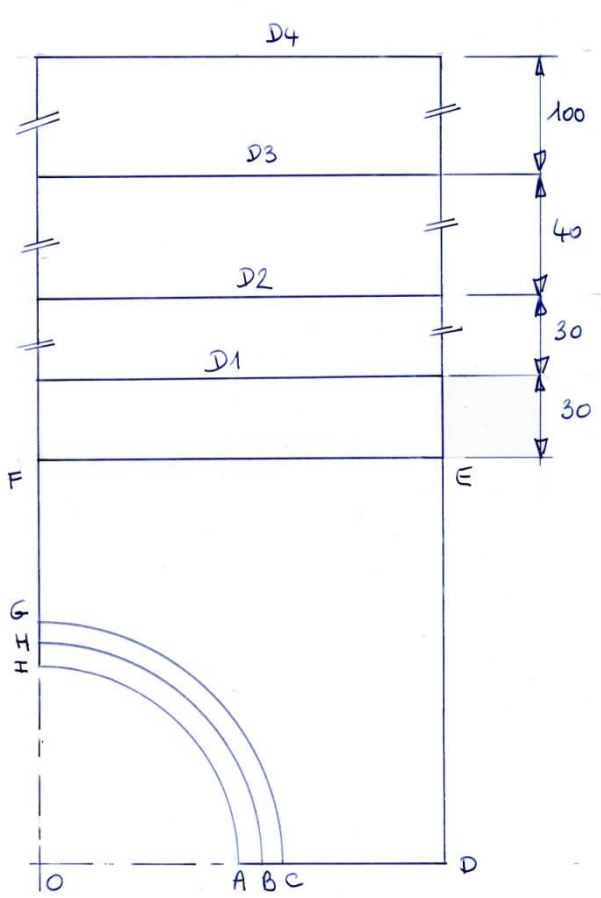
f Select **MD Nastran** as the **Analysis Code**

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.

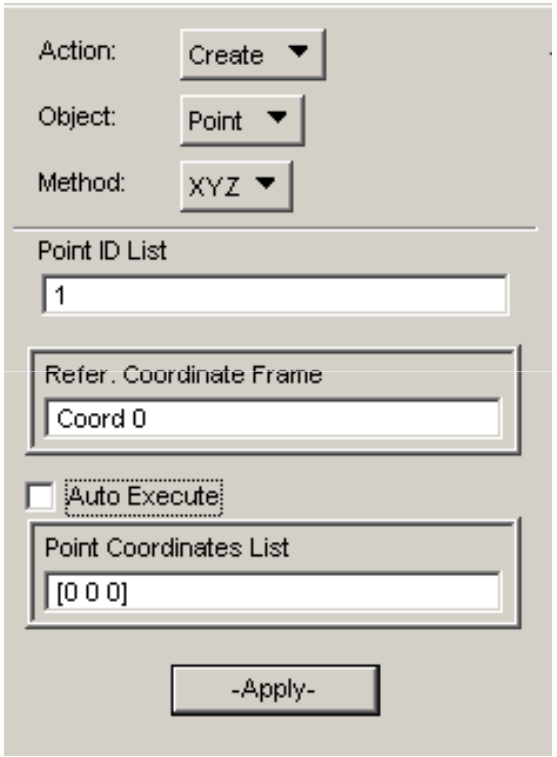


Step 1 : Creation of Geometry - Points

Create the first point :

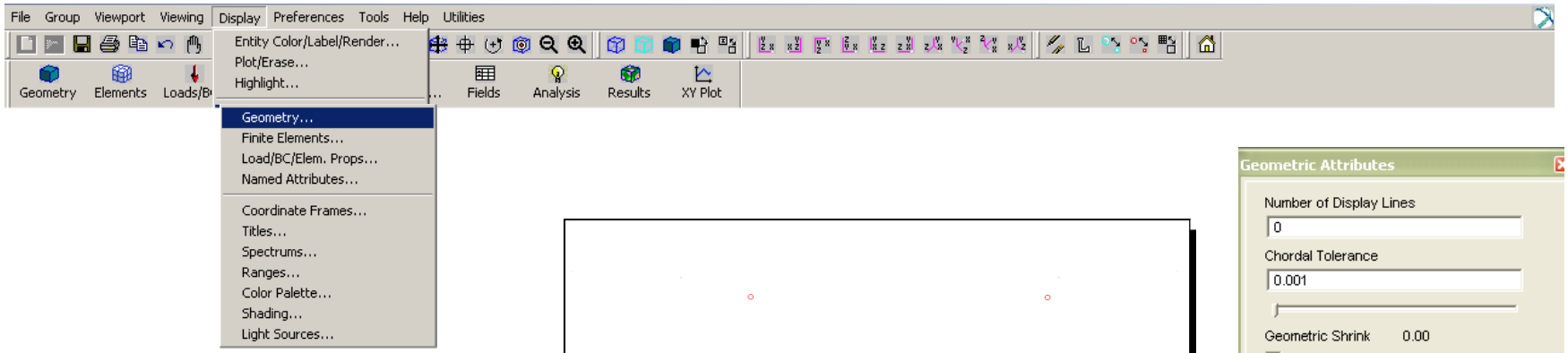
a Geometry : **Create / Point / XYZ**
b Unselect **Auto Execute**
c Enter [0,0,0] for the **Point Coordinate List**.
d Click **Apply**

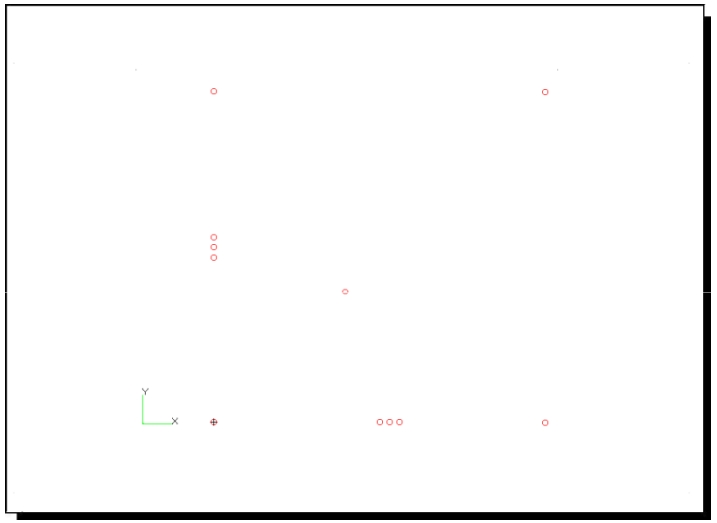
Create the other points (A….I) with the same method.

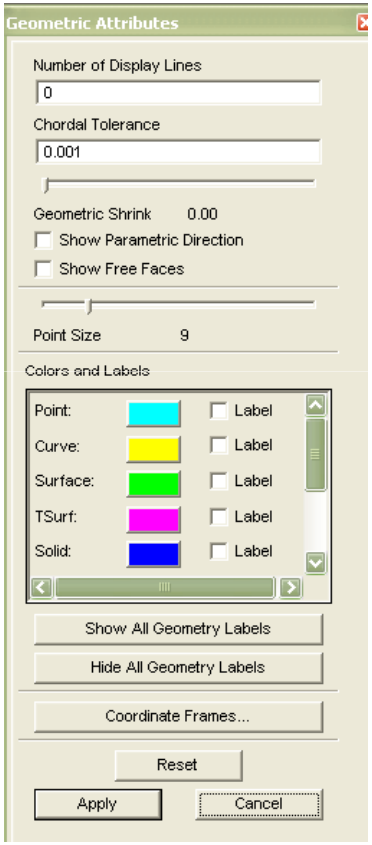


Point	x	y	z	Nastran Id
A	50	0	0	1
B	53	0	0	3
C	56	0	0	4
D	100	0	0	5
E	100	100	0	6
F	0	100	0	7
G	0	56	0	8
H	0	53	0	9
I	0	50	0	10

Viewing of the Points







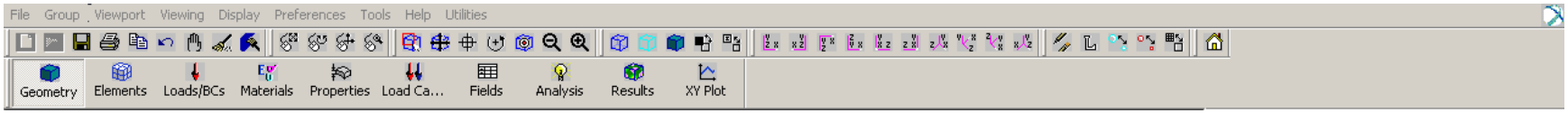
To visualise the points on the screen

a Select the Icon **Point Size**

To increase the size of the points

b **Display / Geometry**
c Modify the position of the cursor **Point Size**
d Click **Apply**
e 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.**

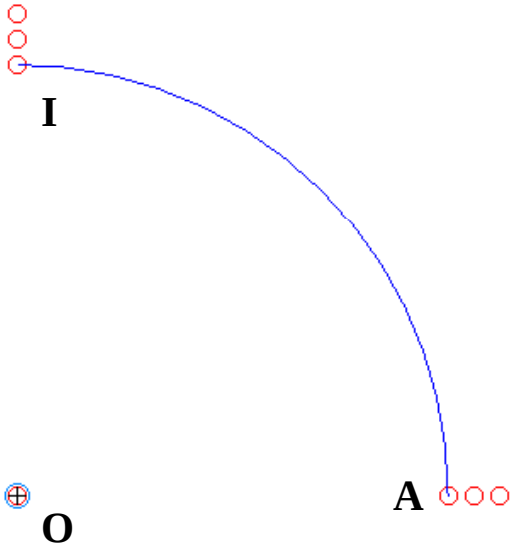
f Activate **Starting Point List** with the left click

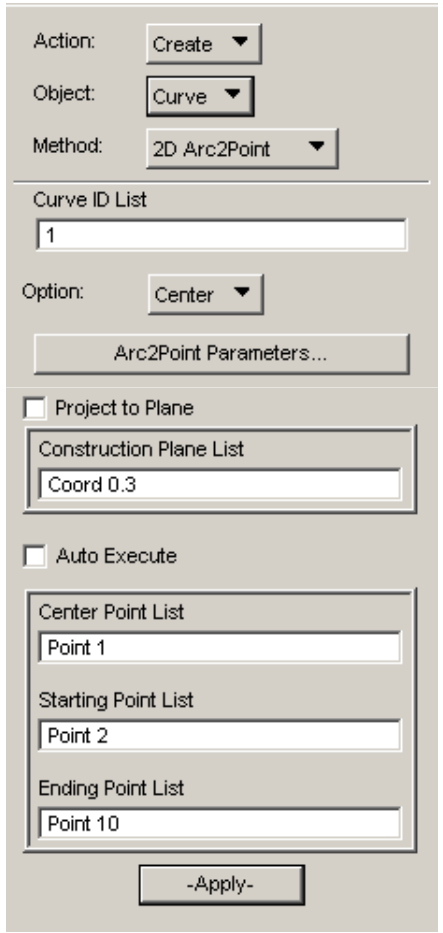
g Screen Pick the point A (Point 2) for the **Starting Point List.**

h Activate **Ending Point List** with the left click

i Pick the point I (Point 10) for the **Ending Point List.**

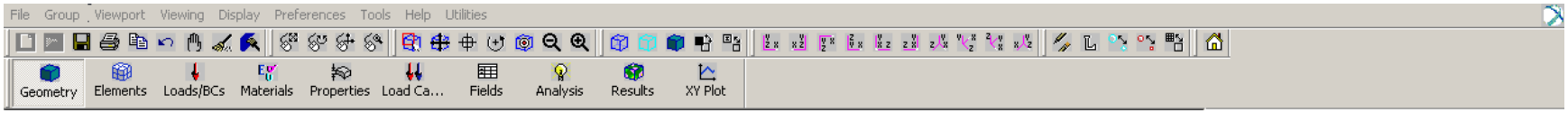
j Click **Apply**





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) :

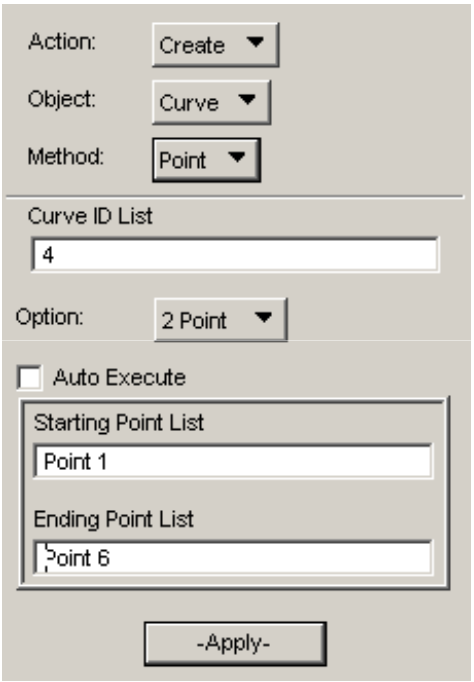
a Geometry : **Create / Curve / Point - Option : 2Point**

b Unselect **Auto Execute**

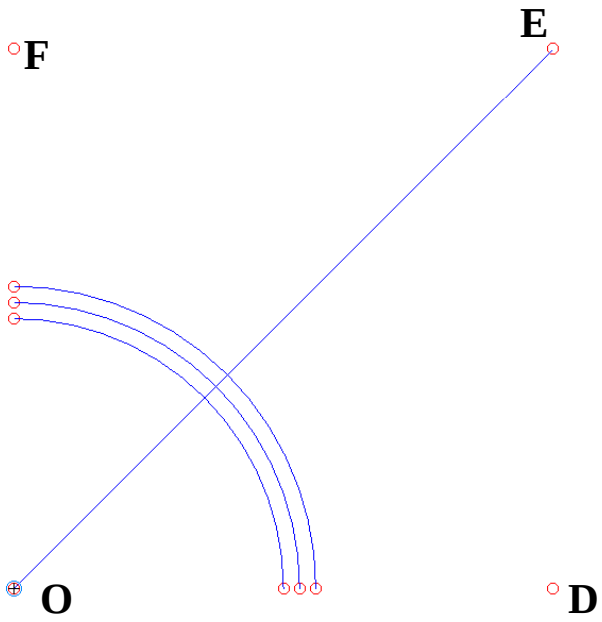
c Screen Pick the point O (Point 1) for the **Starting Point List.**

d Screen Pick the point E (Point 6) for the **Ending Point List.**

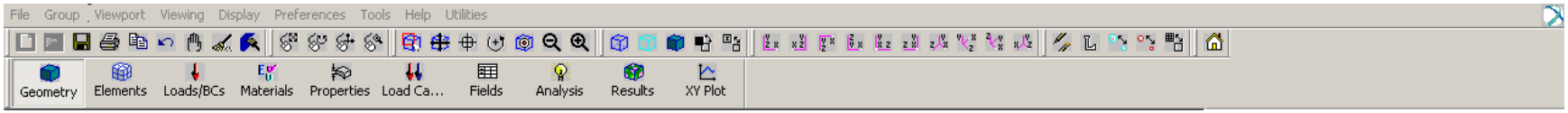
e Click **Apply**



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



Step 1 : Creation of Geometry – Curves



Create the line D1 :

a Geometry : **Create / Curve / XYZ**

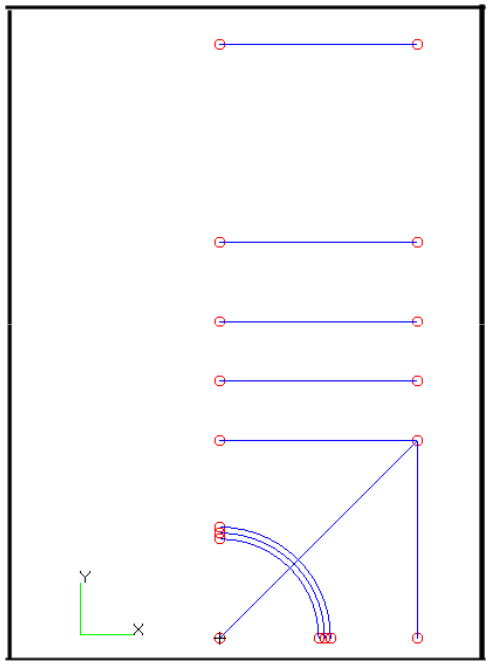
b Unselect **Auto Execute**

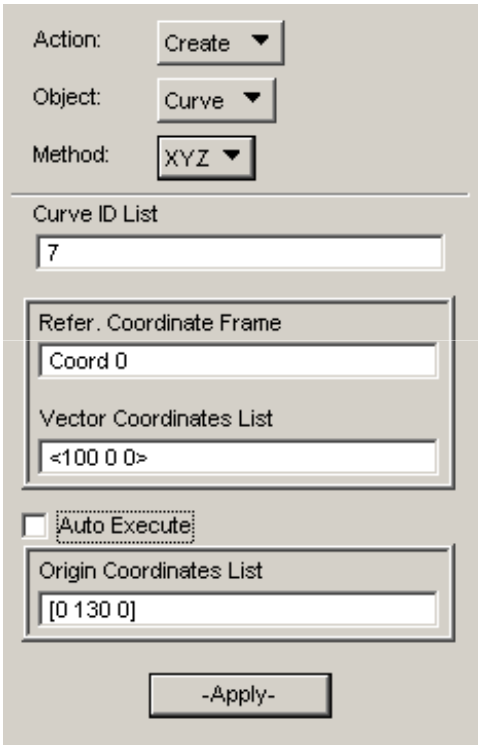
c Enter **[0,130,0],** the coordinates of the beginning point of the line in **Origin Coordinates List**

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

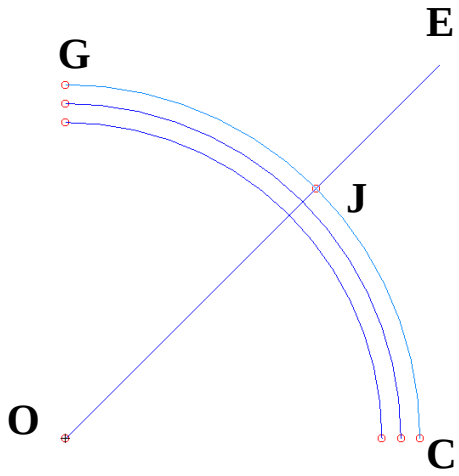


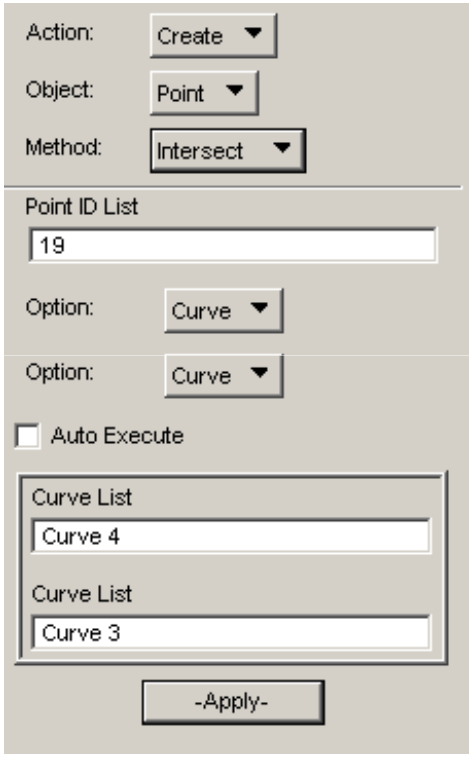


Step 1 : Creation of Geometry

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**

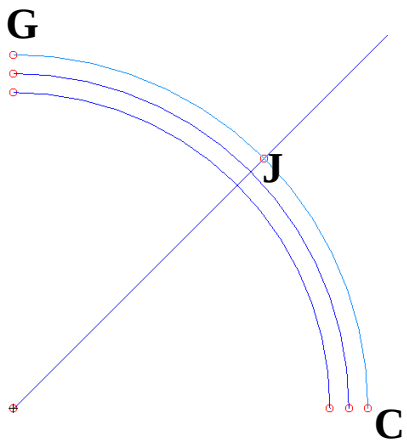


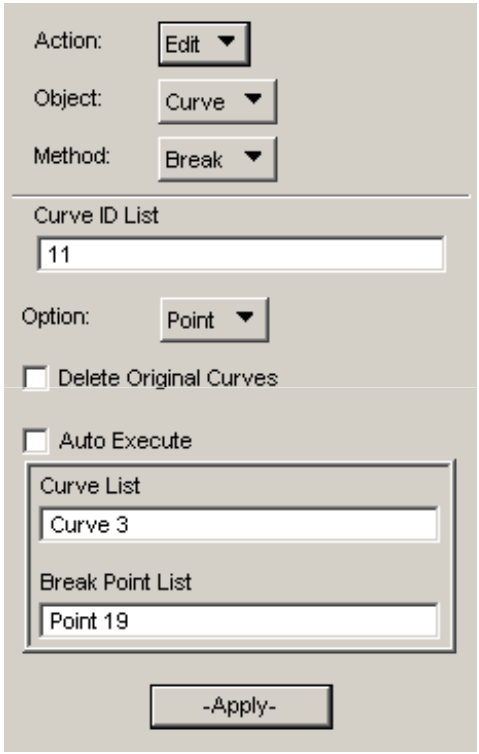


Step 1 : Creation of Geometry

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**



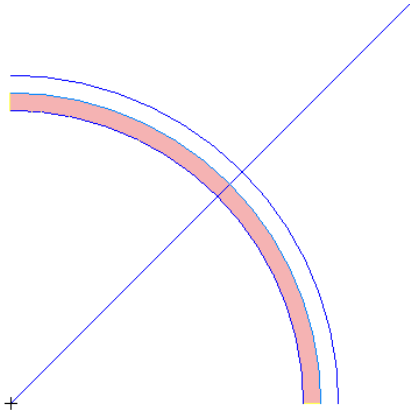


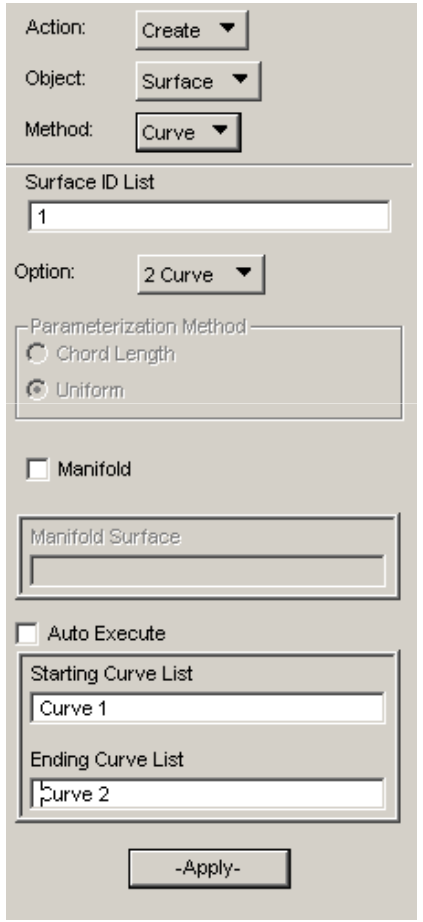
Step 1 : Creation of Geometry

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



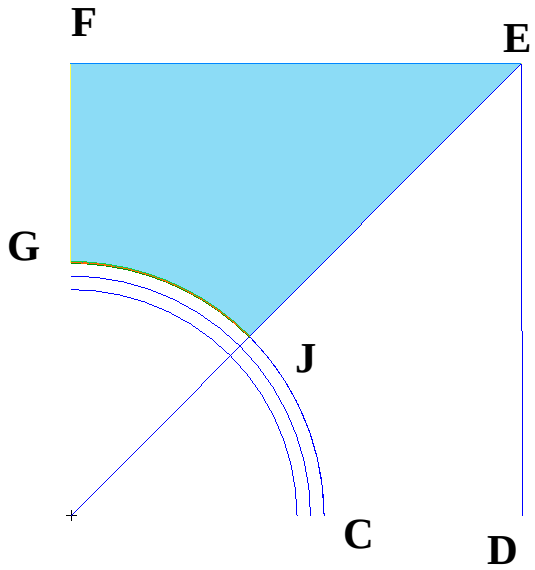


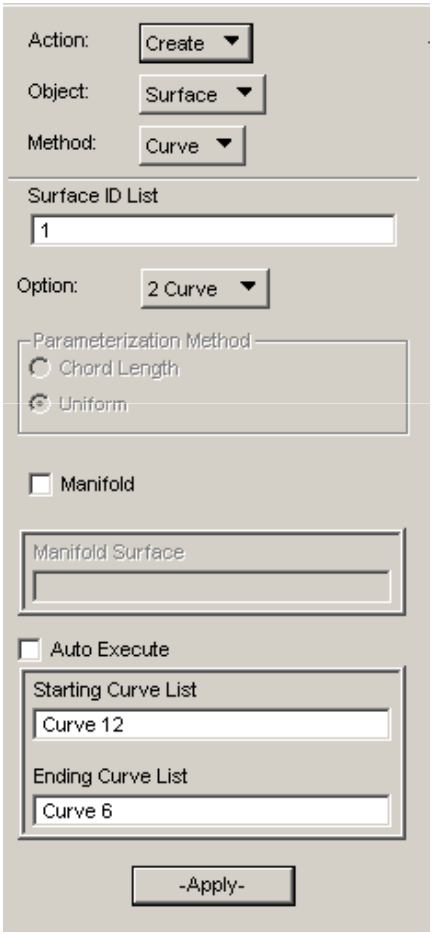
Step 1 : Creation of Geometry

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 :

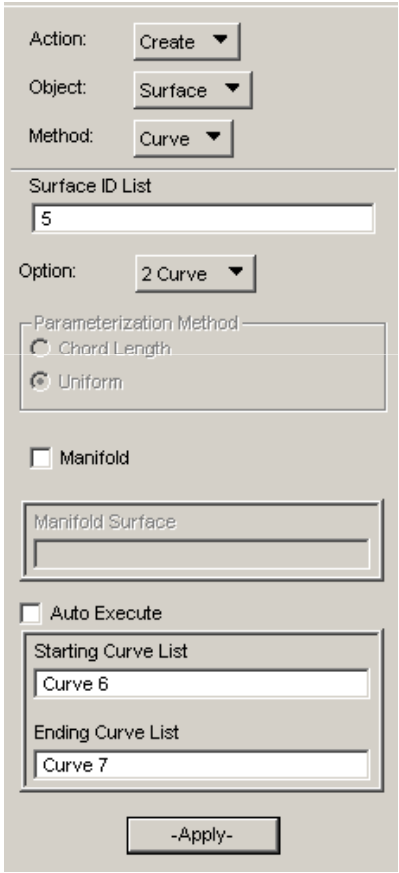
a Geometry : **Create / Surface / Curve**
b Select Option : **2 Curves**
c Unselect **Auto Execute**
d Screen Pick the Curve 6 (Line EF) in **Starting Curve List**.
e Screen Pick the Curve 7 (Line D1) in **Ending Curve List**.
f Click **Apply**

Create the other surfaces :

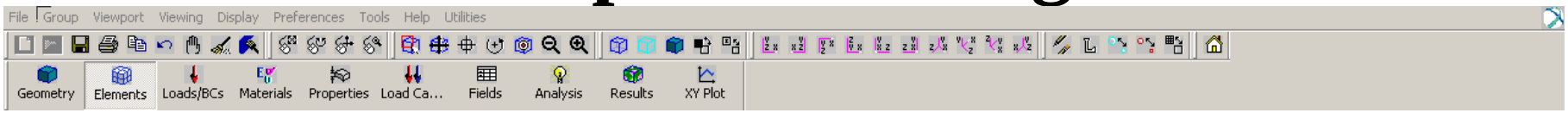
Surface 6 with D1 & D2

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.

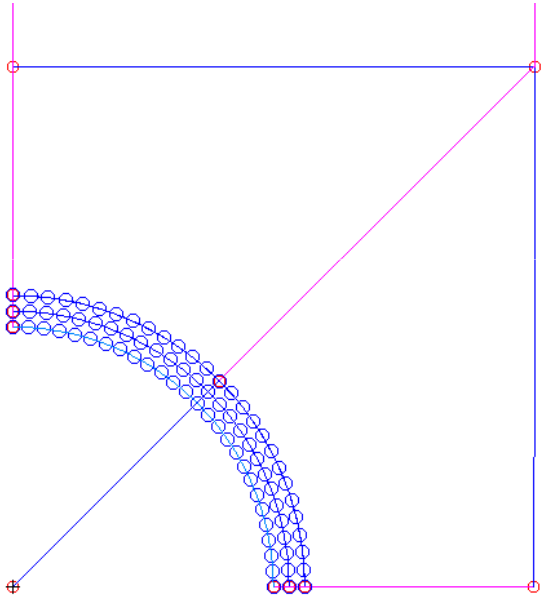
i Screen Pick the Curve 11 for **Curve List List**.

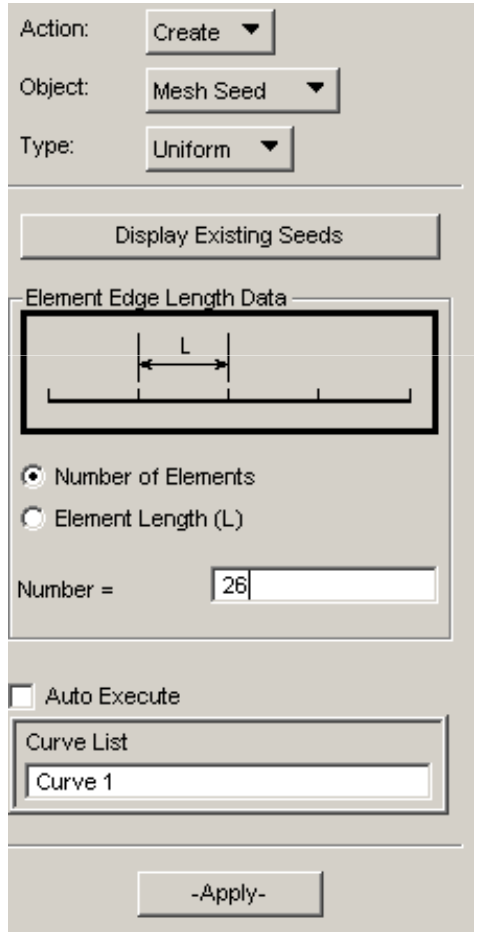
j Click **Apply**

k Enter 13 as the number of elements.

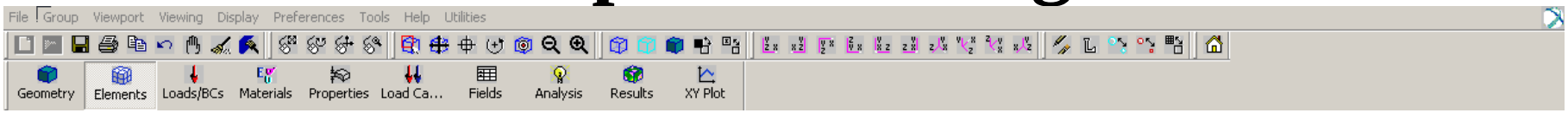
l Screen Pick the Curve 12 for **Curve List List**.

m Click **Apply**





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.

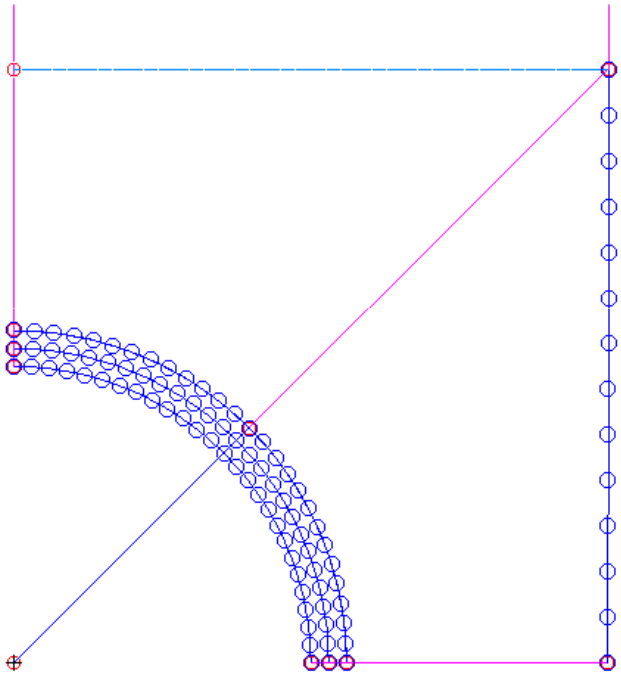
k Screen Pick the Curve 8 for **Curve List**

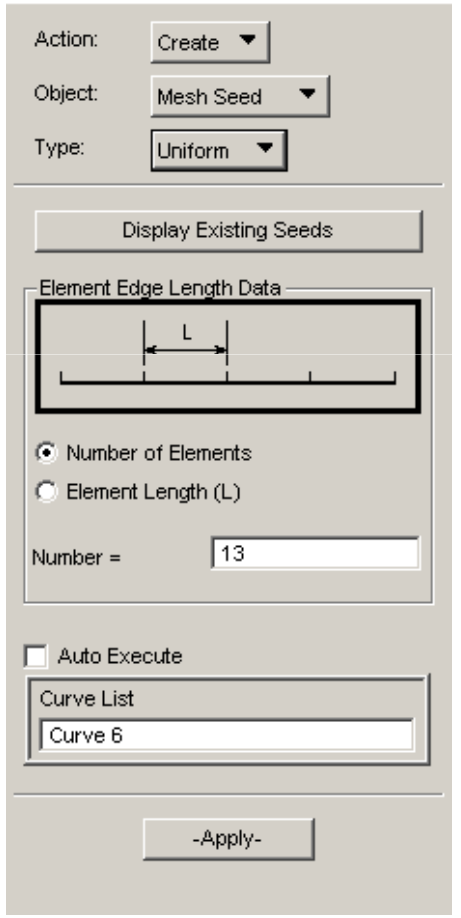
l Click **Apply**

m Enter 7 as the number of elements.

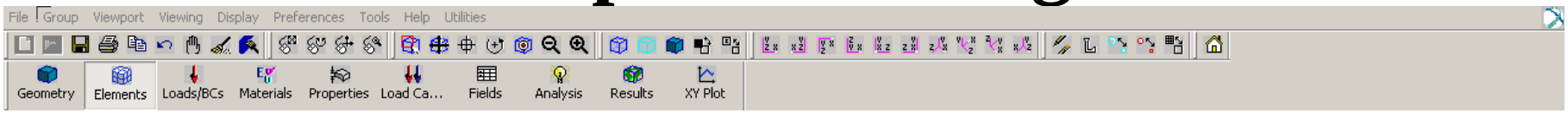
n Screen Pick the Curves 9 & 10 for **Curve List**

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.

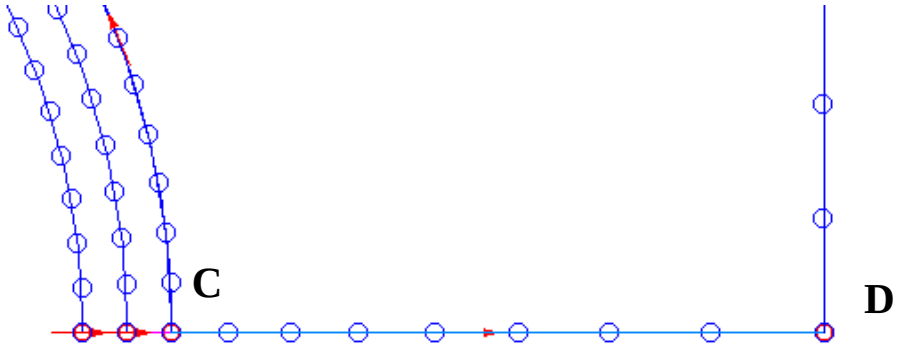
d Enter 2 (or 0.5) as the ratio **L2/L1**

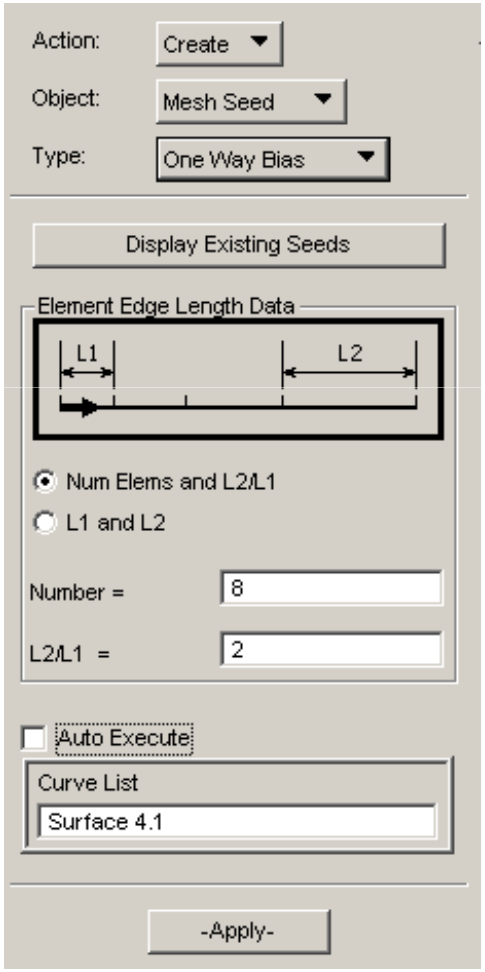
e Screen Pick the Curve between C & D for **Curve List**

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**

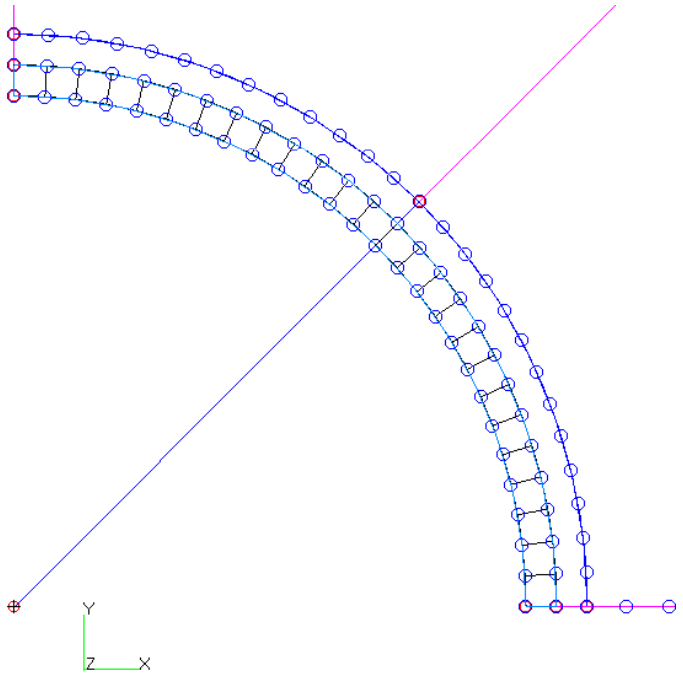
d Unselect **Automatic calculation**

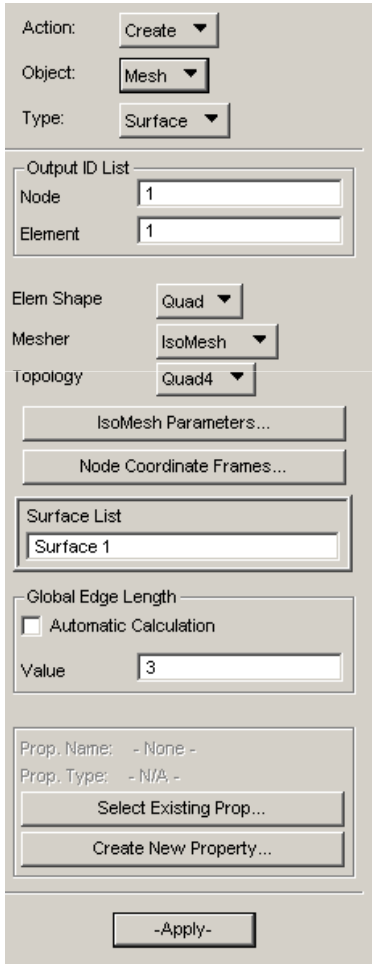
e Enter 3 (Size of the element) for **Value**.

f Click **Apply**

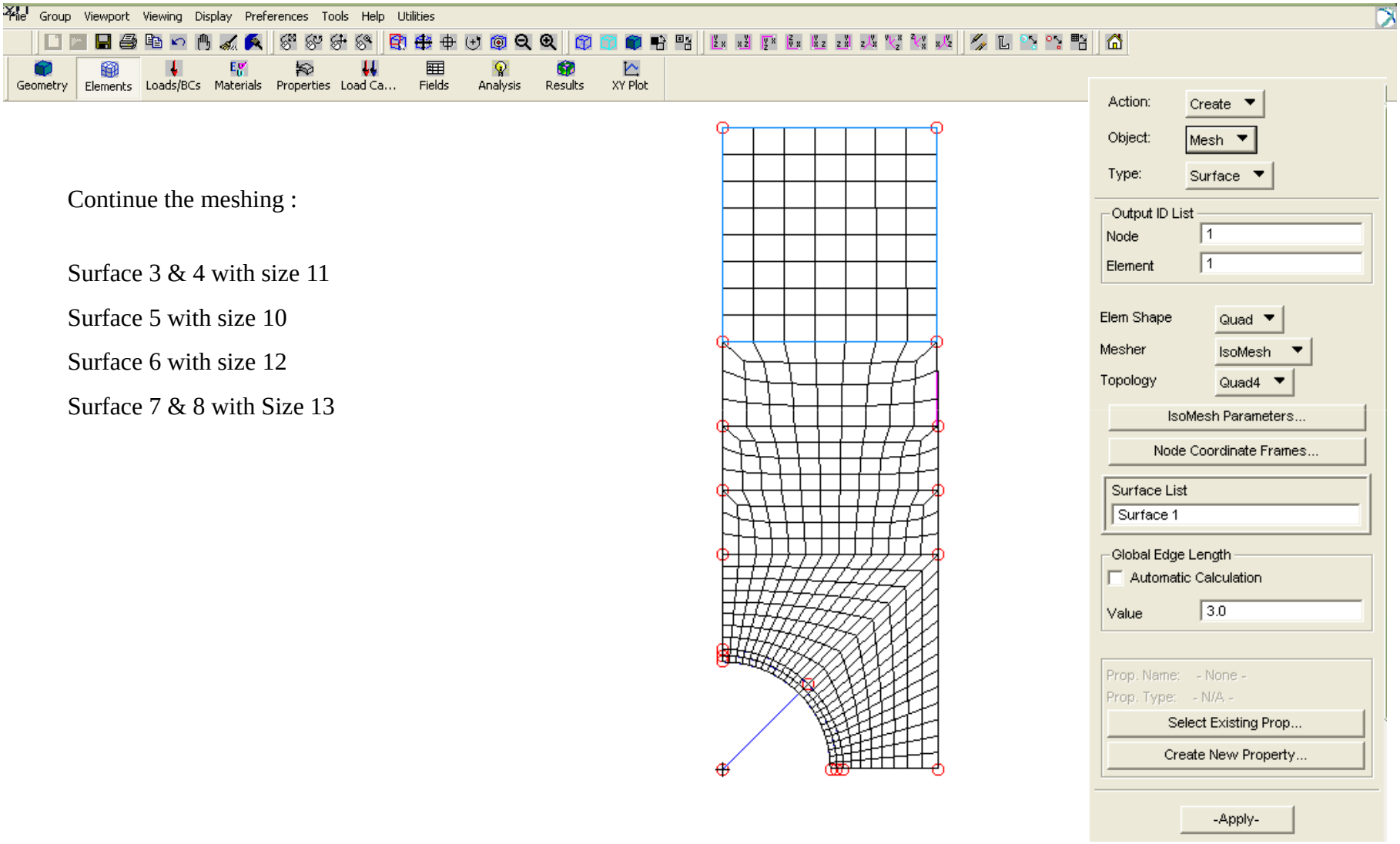
g Screen Pick the Surface 2 for **Surface list**.

h Click **Apply**





Step 3 : Creation of the Nodes & Elements



Continue the meshing :

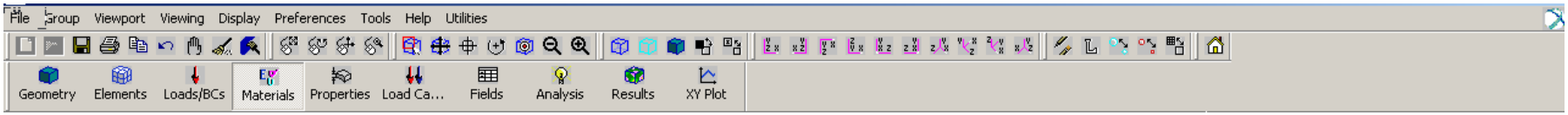
Surface 3 & 4 with size 11

Surface 5 with size 10

Surface 6 with size 12

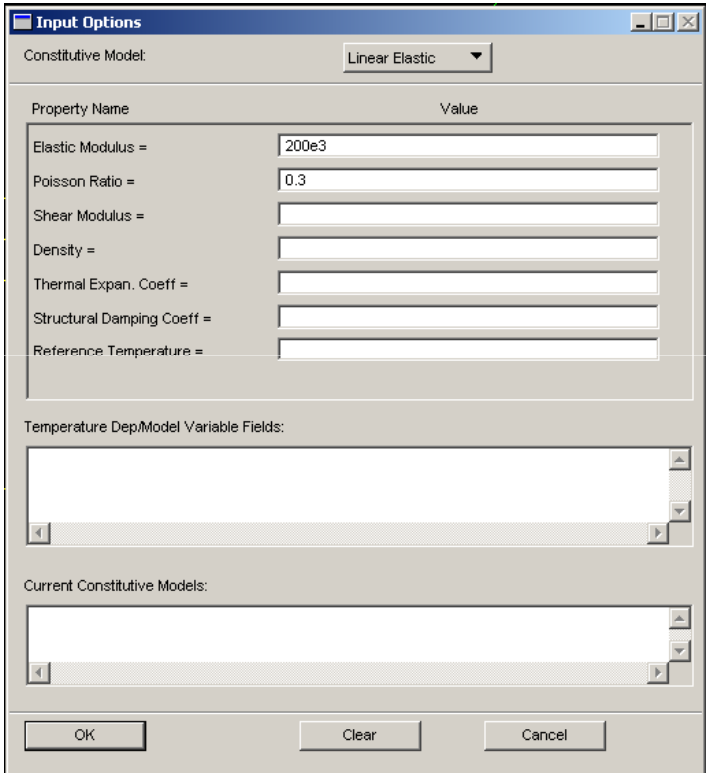
Surface 7 & 8 with Size 13

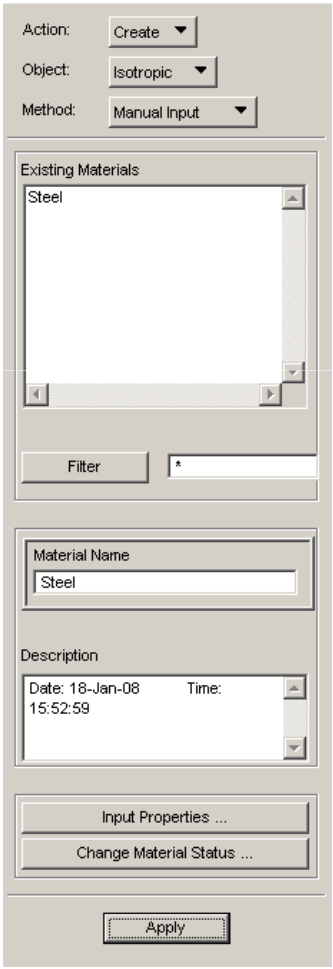
Step 4 Create material



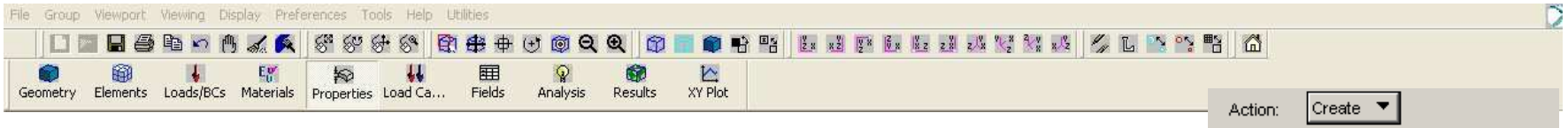
Create an isotropic material :

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**



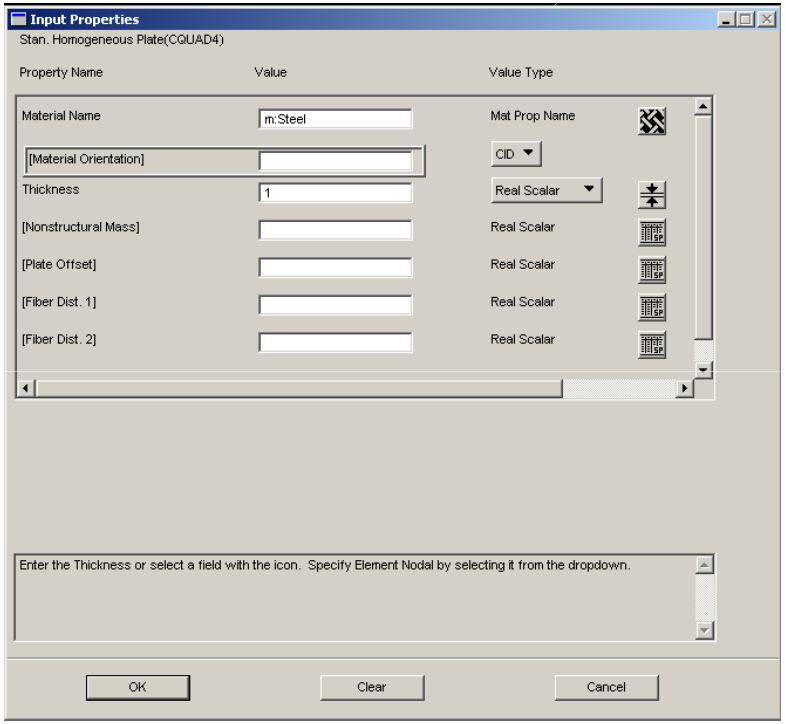


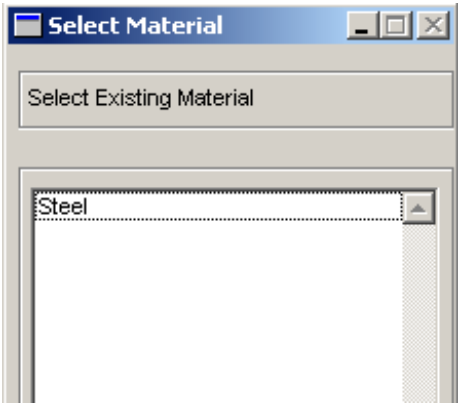
Step 5 : Create Physical Properties

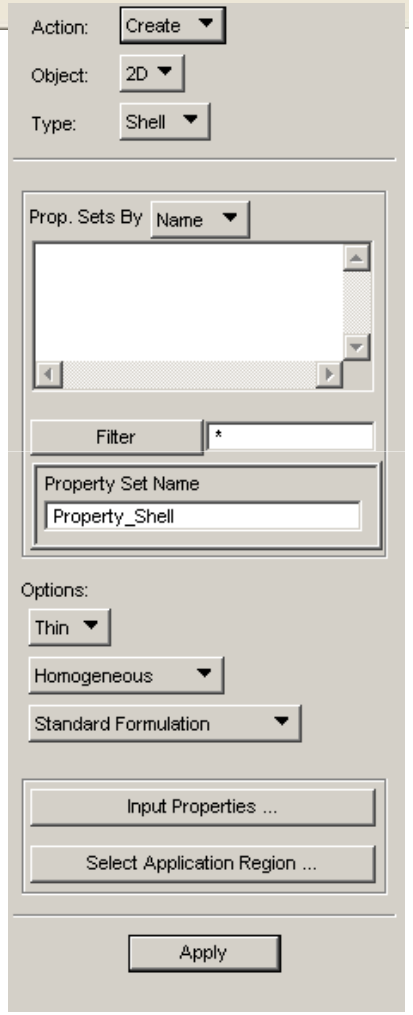


Create element properties :

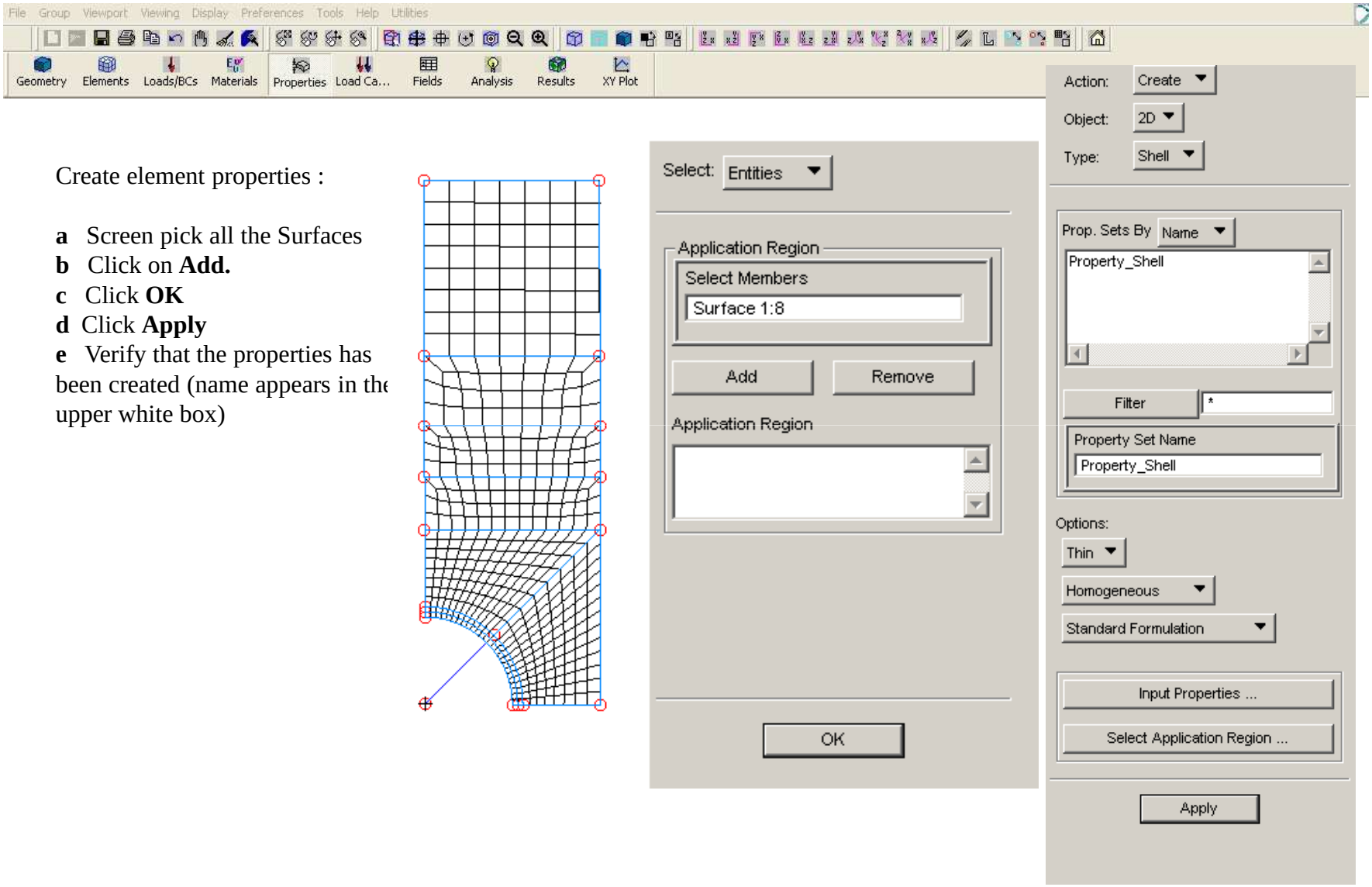
a Properties : **Create / 2D / Shell**
b Enter **Property_Shell** as the Property Set Name.
c Click on **Input properties**
d Click on **Mat Prop Name** Icon
e Select **Steel** as the materiel.
f Enter 1 as thickness for the plate
g Click **OK**
h Click on **Select Application Region**







Step 5 : Create Physical Properties



Create element properties :

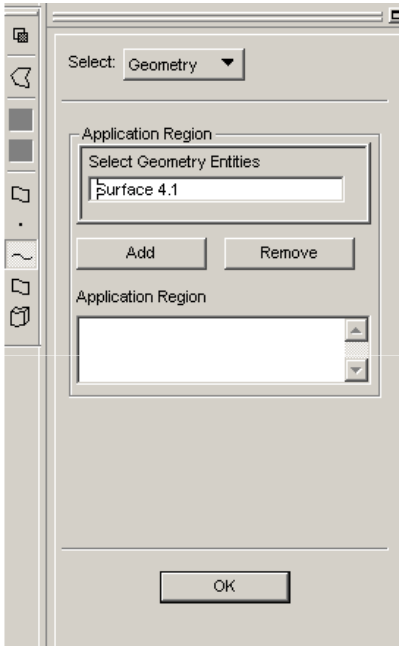
a Screen pick all the Surfaces
b Click on **Add.**
c Click **OK**
d Click **Apply**
e Verify that the properties has been created (name appears in the upper white box)

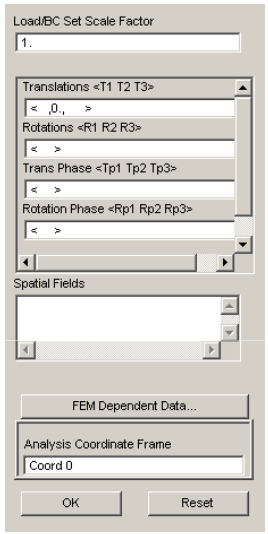
Step 6 :Create Boundaries Conditions

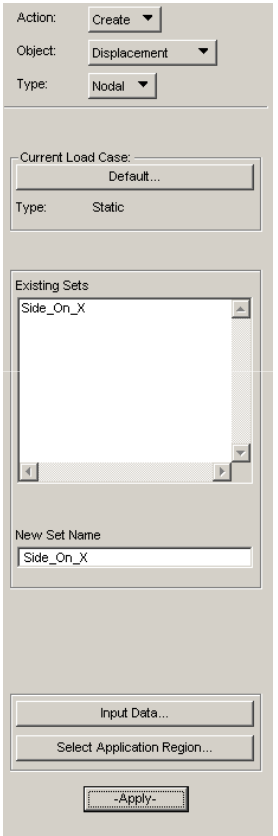
Create boundary conditions :

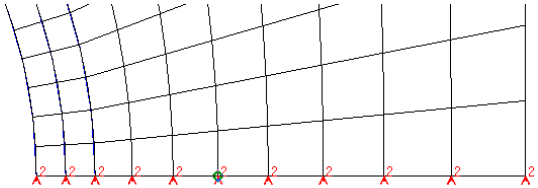
a Click on L**oads/BCs : Create / Displacement / Nodal**
b Enter **Side_On_X** as the New set Name.
c Click on **INPUT Data**.
d Enter **< ,0., >** for Translations.
e Click **OK.**
f Click on **Select Application Region**
g Select **FEM**
h Screen pick with the good option the nodes on the X axis
i 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**

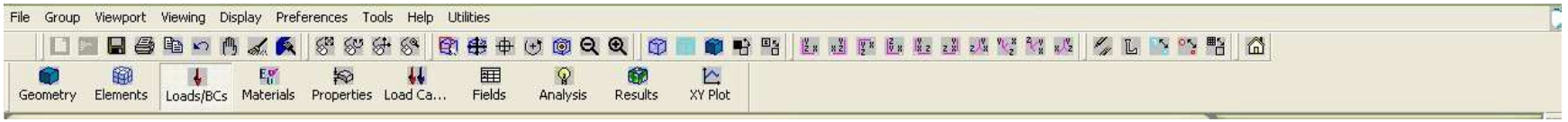






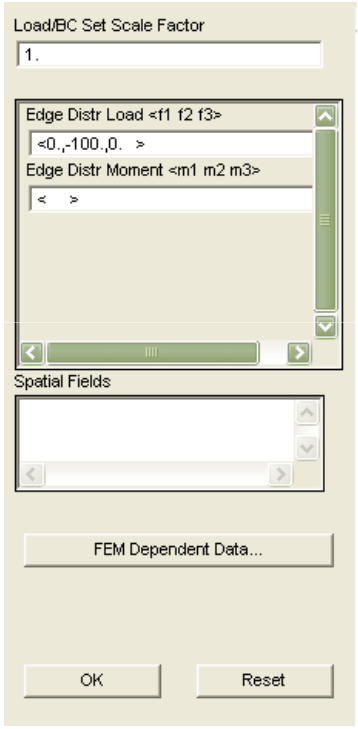


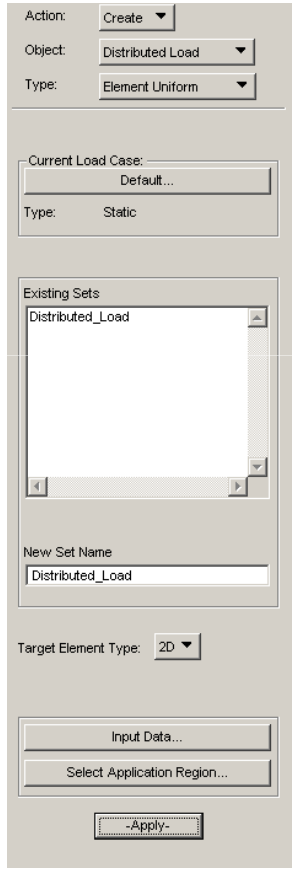
Step 7 : Create Load



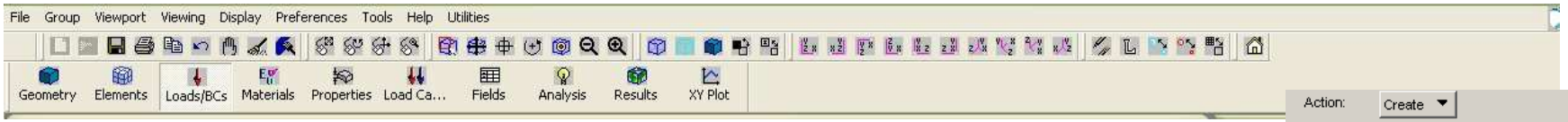
Create a constant uniform load :

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**

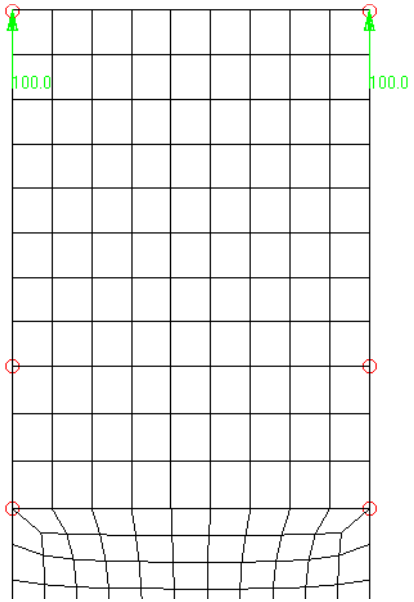


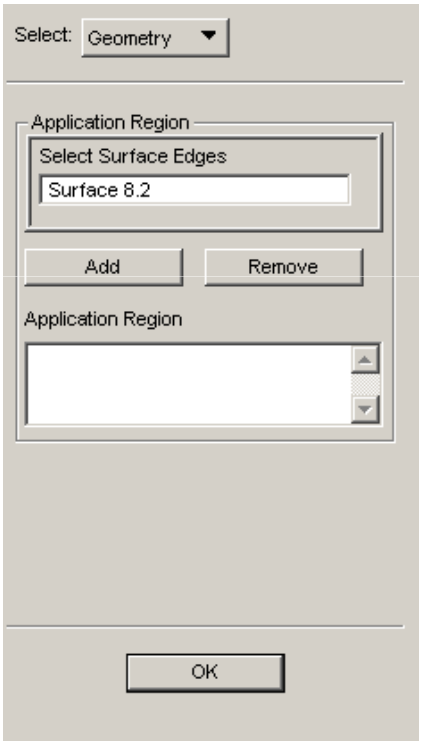


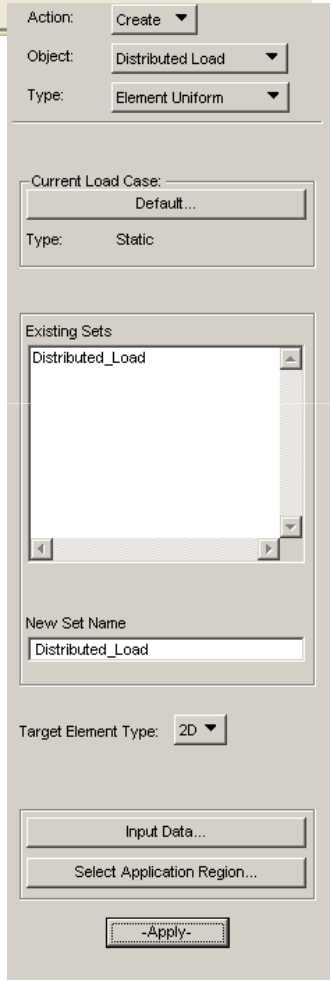
Step 7 : Create Load



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

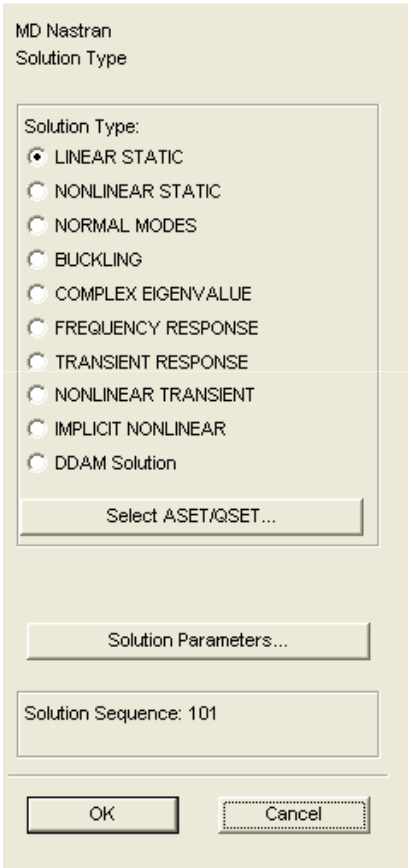
a Analysis : **Analyse / Entire Model / Full Run**

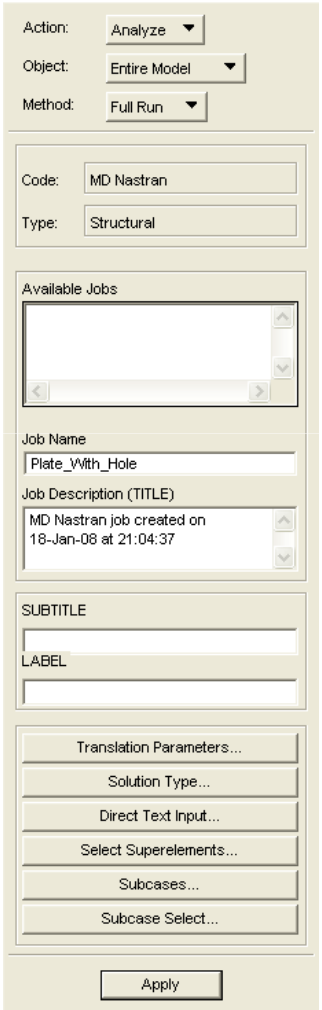
b Select **Solution Type**

c Select **Linear Static**

d Click **OK**

f Click **Apply**





Step 8 : Load Nastran Results in Patran

Load the results file of NASTRAN

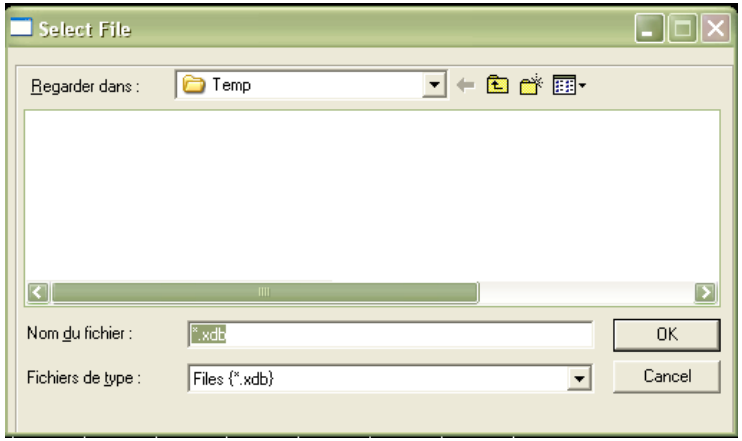
a Analysis : **Access Results / Attach XDB / Result Entities**

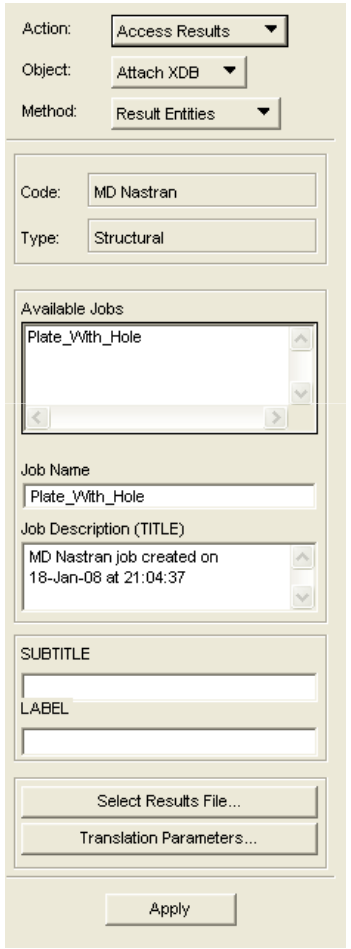
b Click on **Select Results File**

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.

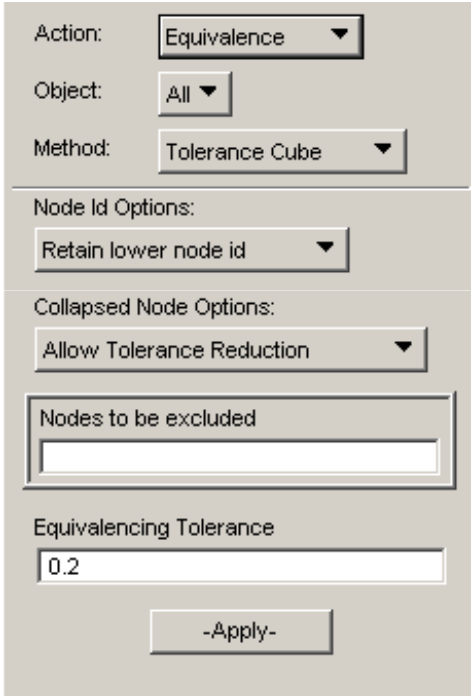
Equivalence

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



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: **Delate / 2D / Shell**

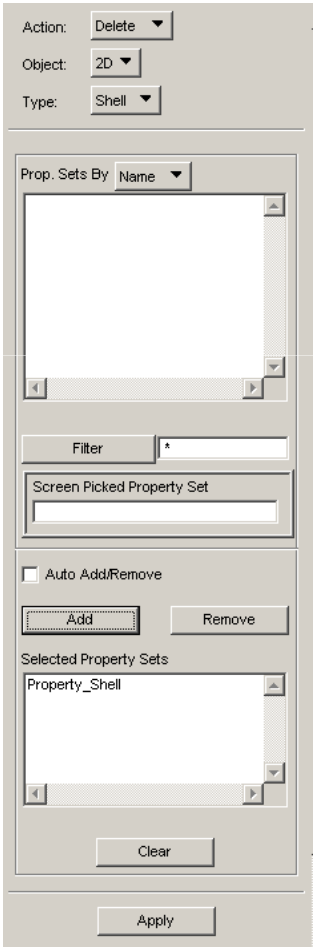
b Select the properties by the name in the good box.

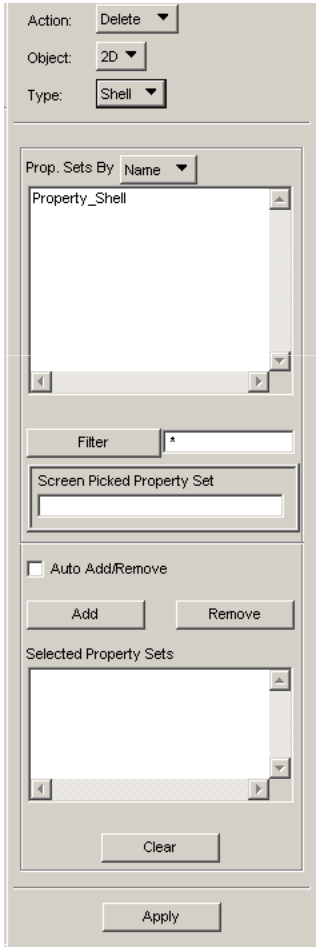
c Click **ADD**

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 you modelling seem good

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

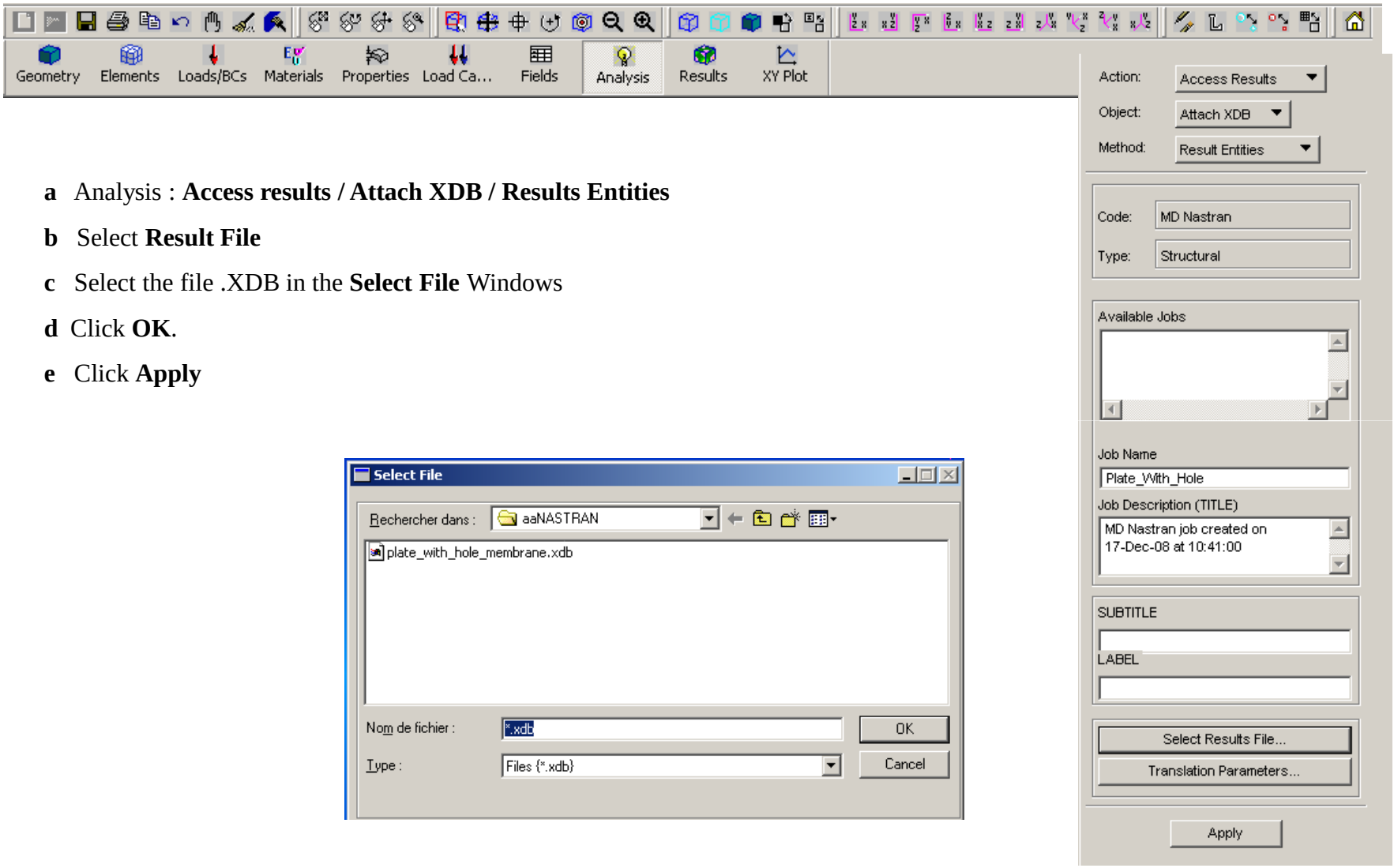
After that you can import a specific output file inside Patran to analyse the results. Two different output files can be generated by NASTRAN :

*. XDB output file (used at ISAE)

*.OP2

To import the output file in Patran see the following slide

Import Output BDF file in PATRAN



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

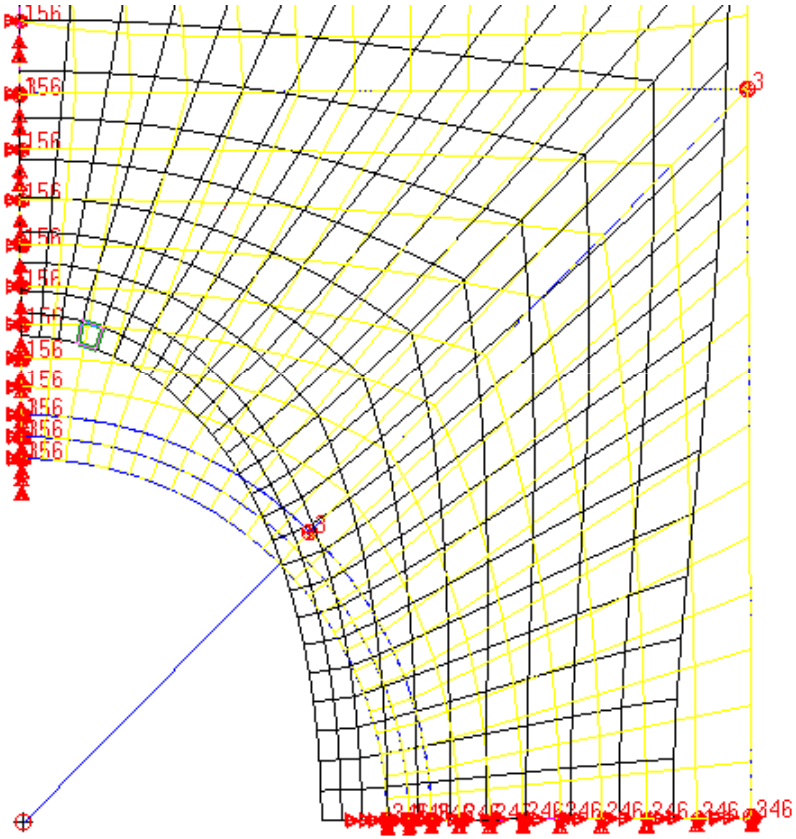
b Select **Result File**

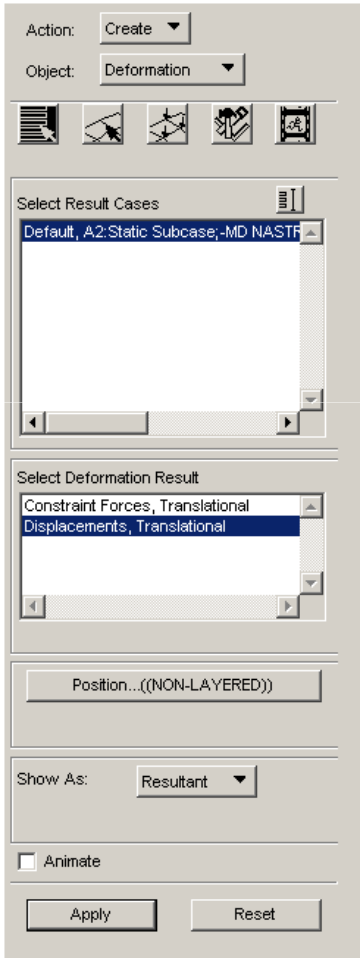
c Select the file .XDB in the **Select File** Windows

d Click **OK**.

e Click **Apply**

Step 9 : Results - Deformation





Verify the displacement :

a Results : Create / Deformation

b Select Displacements, Translational in **Select Deformation Result**

c Click **Apply**

Step 9 : Results - Von Mises

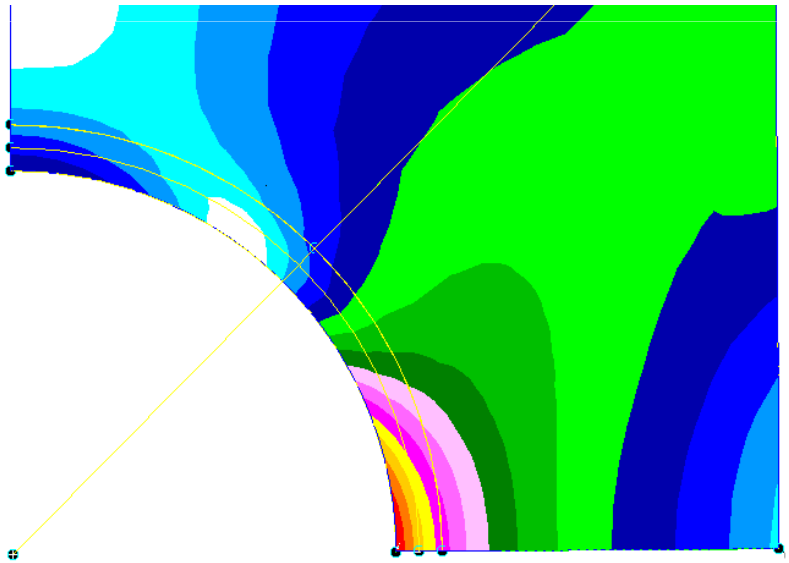
Verify the Von Mises' Criteria :

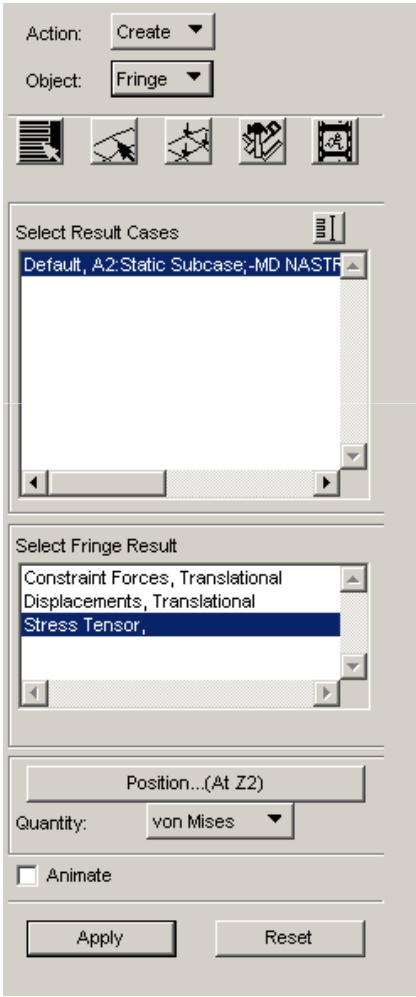
a Results : Create / Fringe

b Select Stress Tensor in **Select Fringe Result**

c Selec**t Von Mises fo Quantity**

d Click **Apply**





Step 9 : Results - σ_x

Verify the Von Mises' Criteria :

a Results : Create / Fringe

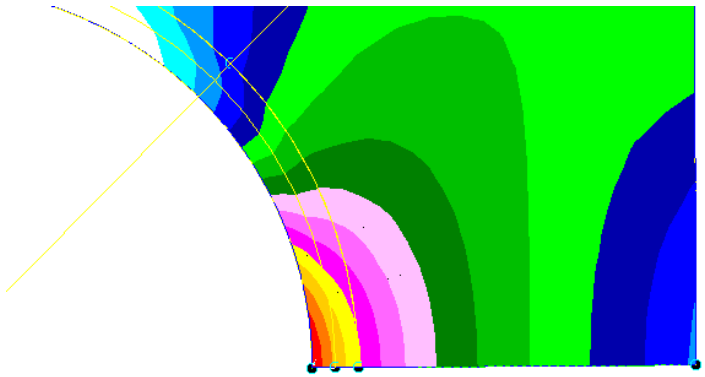
b Select Stress Tensor in **Select Fringe Result**

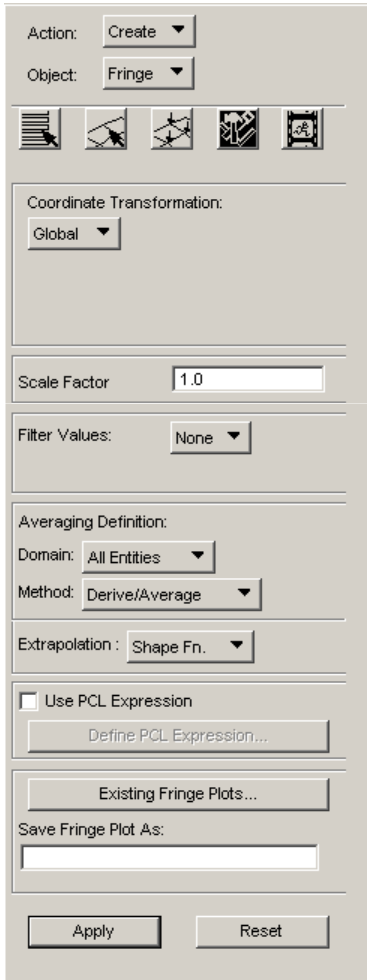
c Select **Y Component** for **Quantity**

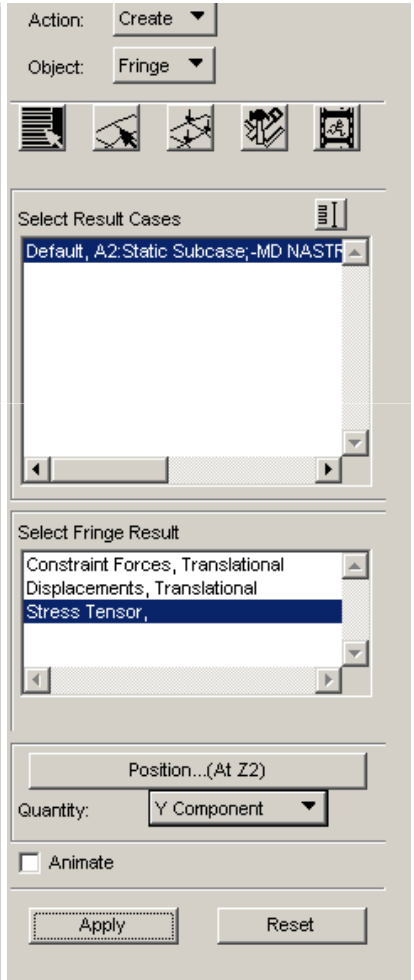
d Select the Icon **Plot Options**

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**

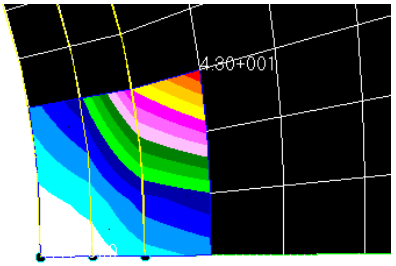
d Select **XY Component**

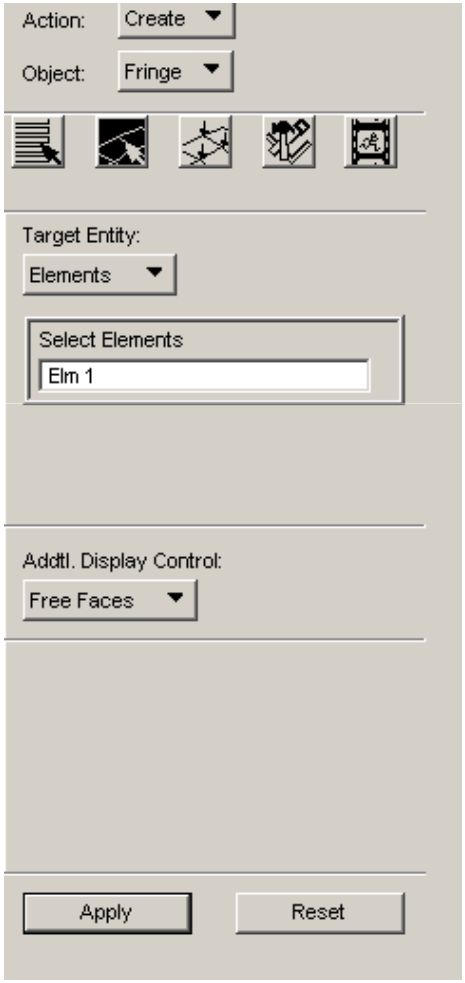
e Select the Icon **Target Entities**

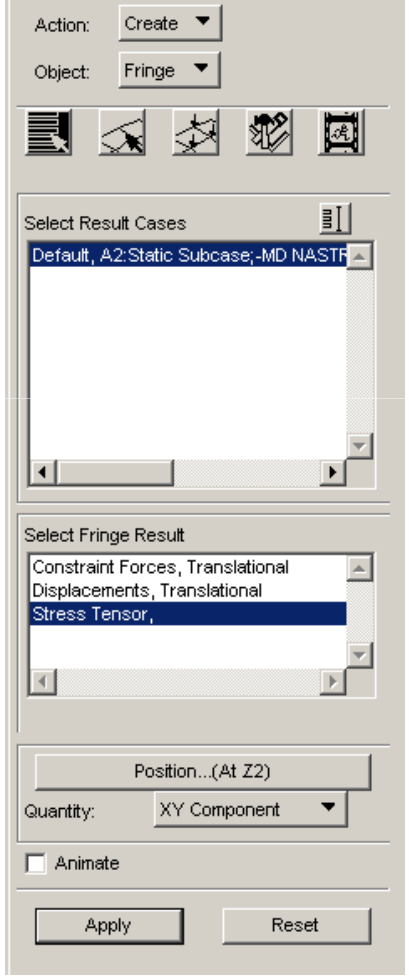
f Select Elements as **Target Entity**

g Select or Screen Pick the elements you want visualize

e Click **Apply**







Results ：Stress in an Element

Identify one (or More) element(s) :

a Elements: Show / Element / Attributes

b Screen Select the element near the hole

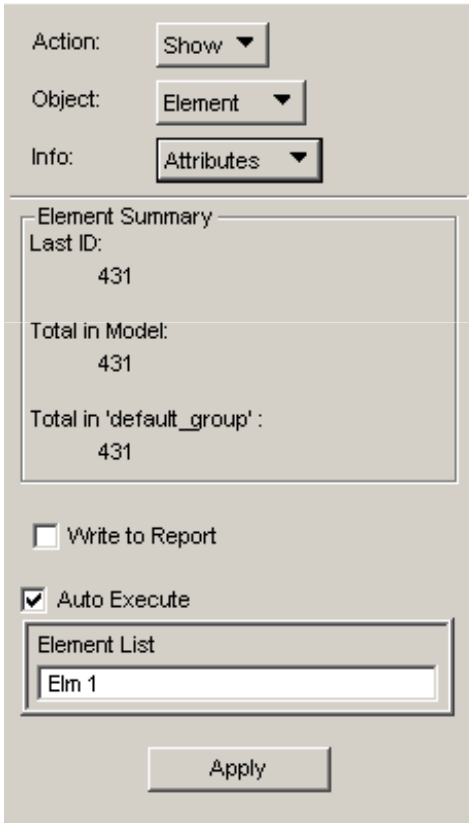
c Click **Apply**

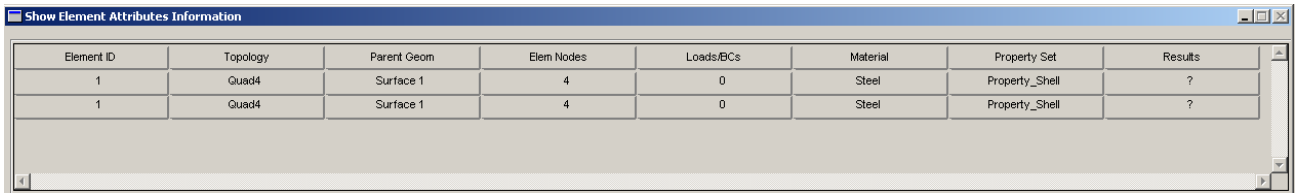
d Read the identification number of the element

e Open the file job.F06

f Find the data of the element(s)

g Analyse the results





Results File Job.F06 (Point A)

ELEMENT		FIBER	STRESSES	IN ELEMENT CO	ORD SYSTEM	PRINCIP	AL STRESSES (Z	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

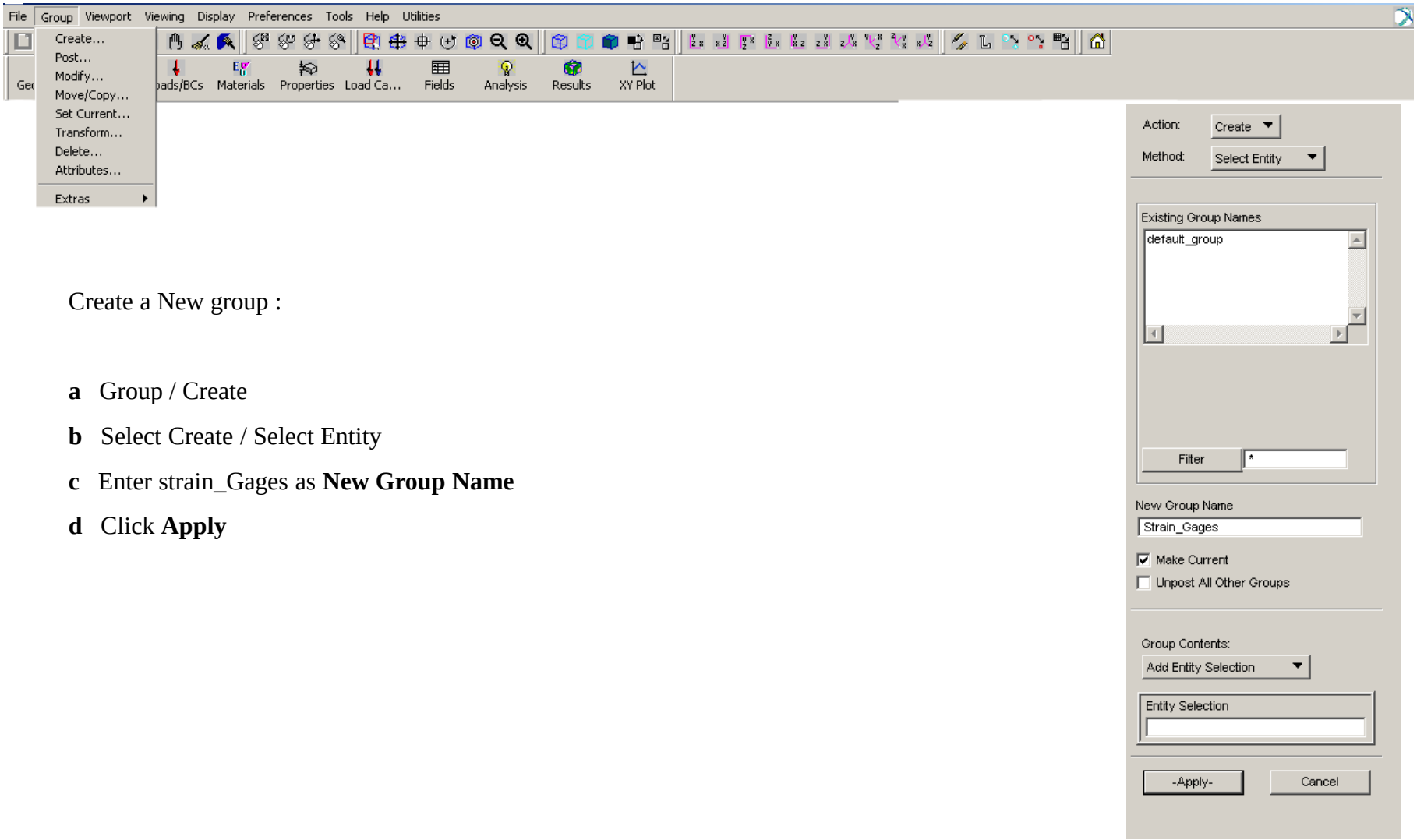
Results File Job.F06 (Point A) (Excell)

ELEMENT		FIBER	STRESSES IN ELEMENT COOR SYSTEM			PRINCIPAL STRESES (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 :

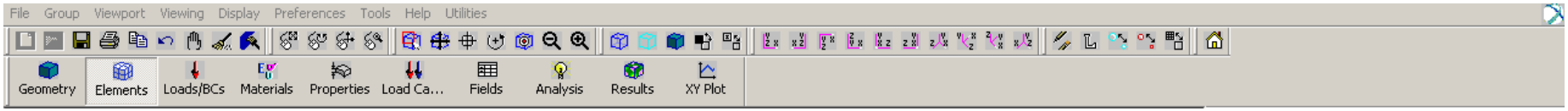
a Group / Create

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 :

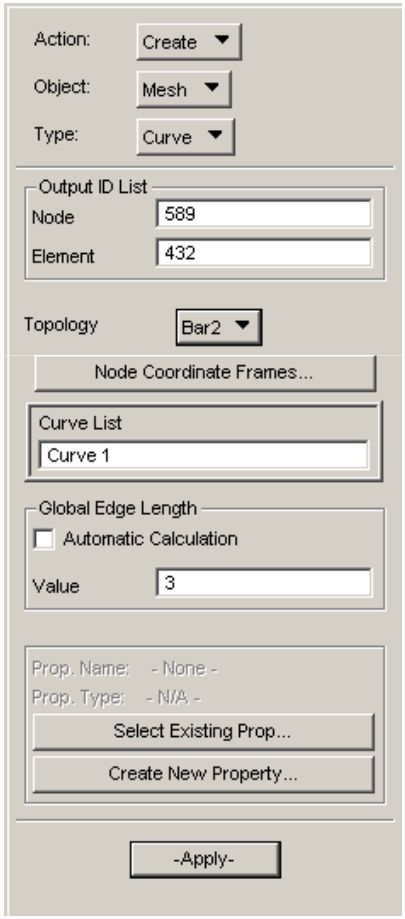
a **Elements :** Create / Mesh / Curve

b Select **Bar2** as **Topology**

c Unselect **Automatic Calculation**

d Enter 3 as V**alue**

e Verify that 26 elements have been created



Create New Properties

Create element properties :

a Properties : Create / 1D / Rod

b Enter Property_Rod as the name of **Property Set Name**

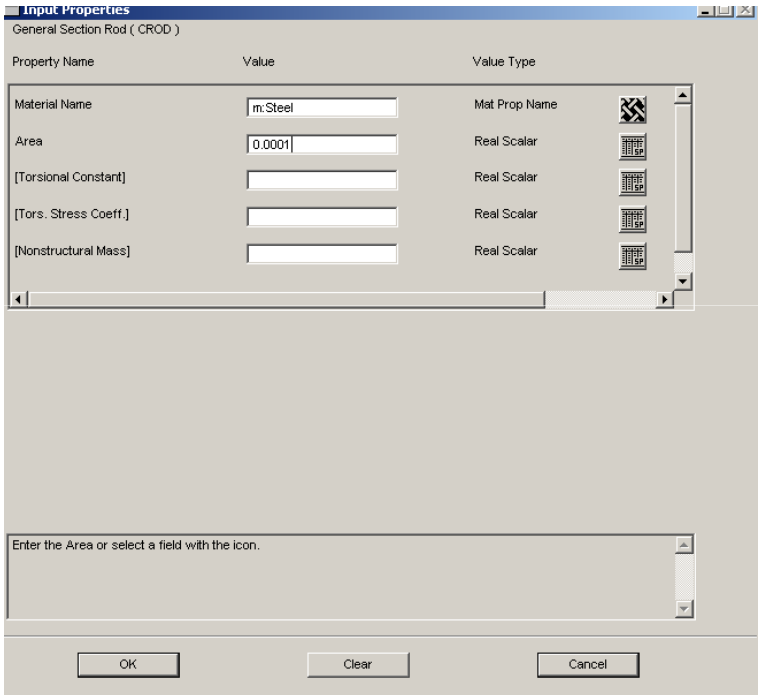
c Select **Input Properties**

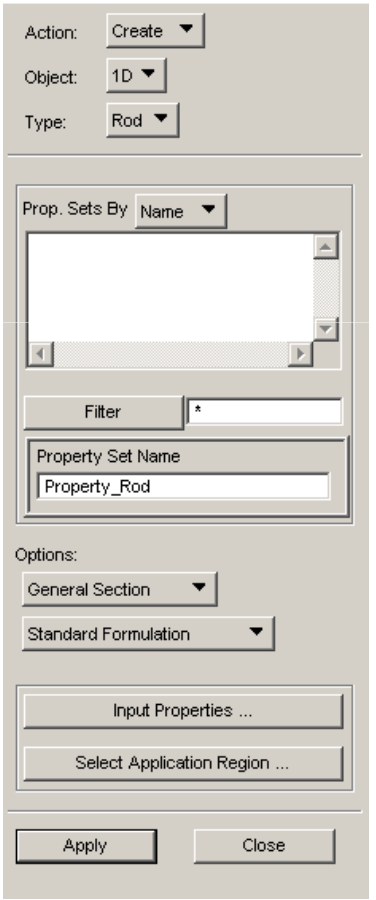
d Select Steel as Material

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**

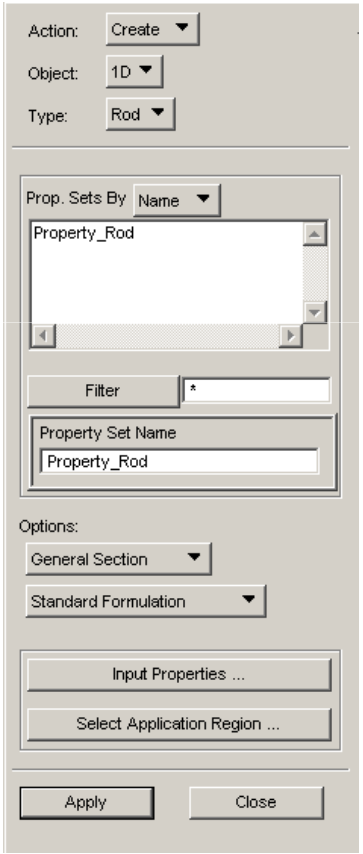
b Screen Select the Curve 1

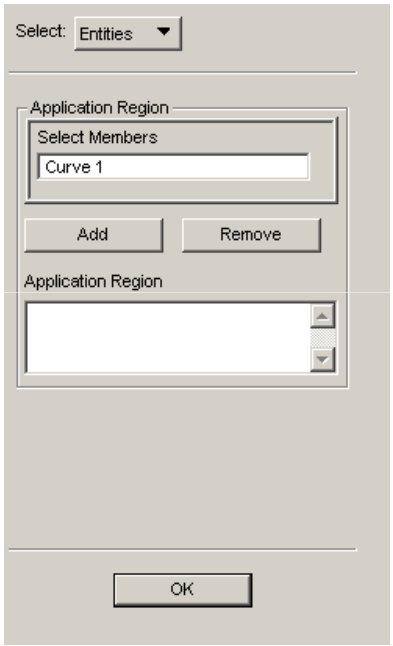
b Click **ADD**

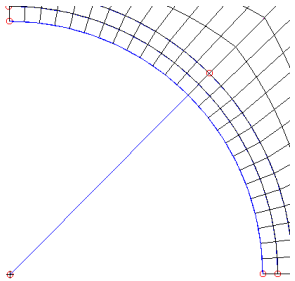
c Click **OK**

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 elemsnt,
- 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

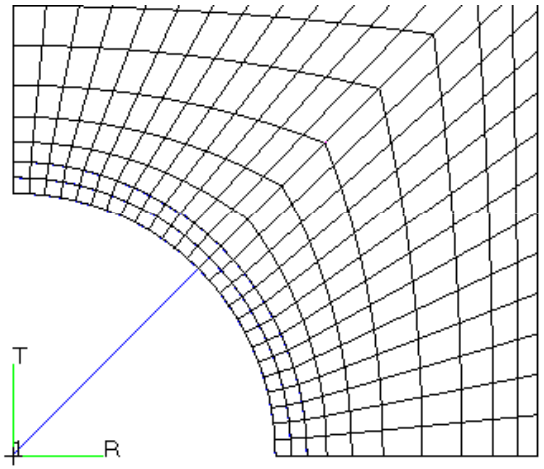
c With the **KeyBoard** enter the coordinates of the center of the axis system : **[0 0 0]**

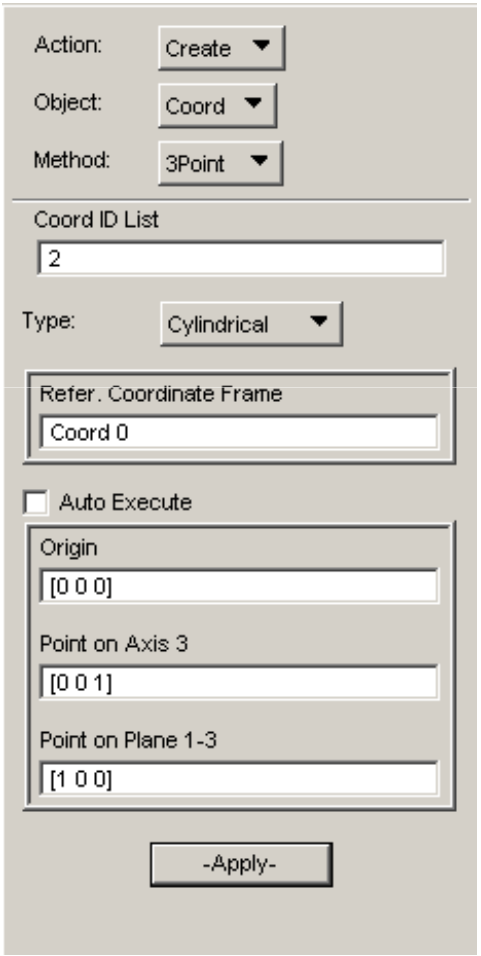
d With the **KeyBoard** enter the coordinates of a p on the third axis : **[0 0 1]**

e With the **KeyBoard** enter the coordinates of a po on the plane 1-3 : **[1 0 0]**

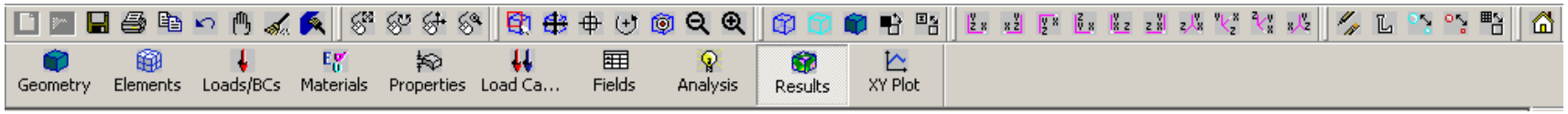
f Click **Apply**

g Verify that the axis system has been created





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**

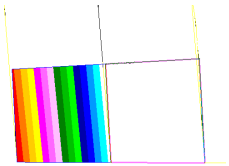
h Select the element on the model

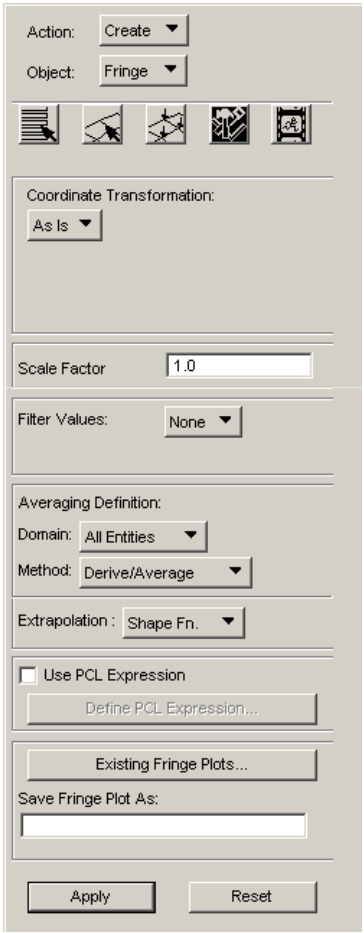
i Click **Apply**

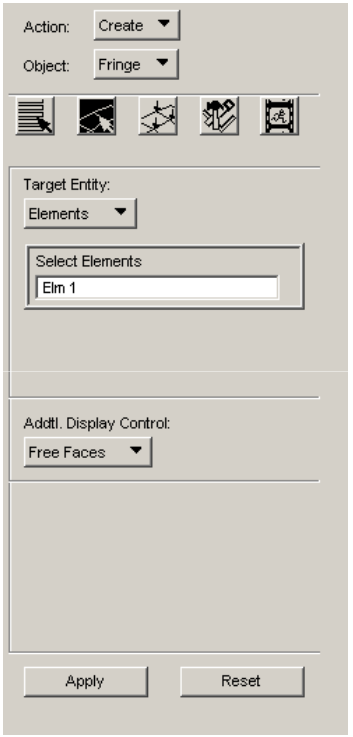
j Select the Icon **Plot options**

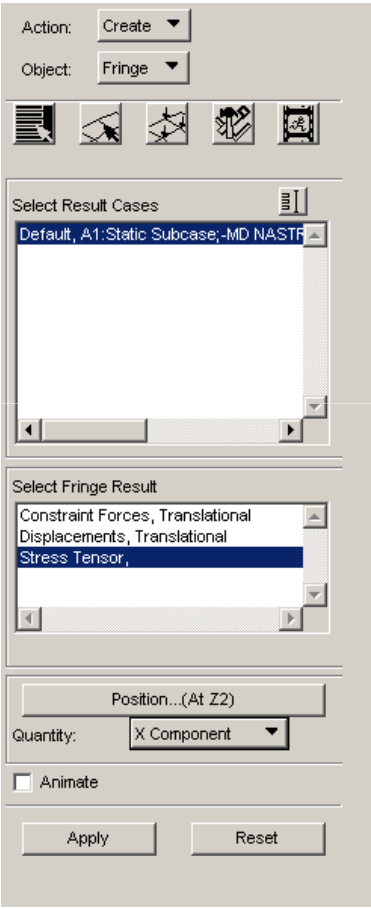
k Select **As is** as Coordinate Transformation

l Click **Apply**

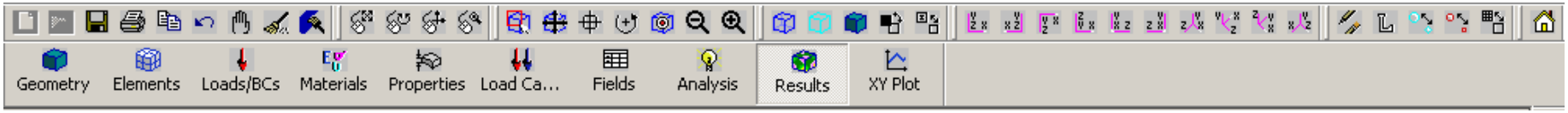








Analysis in the global axis system



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**

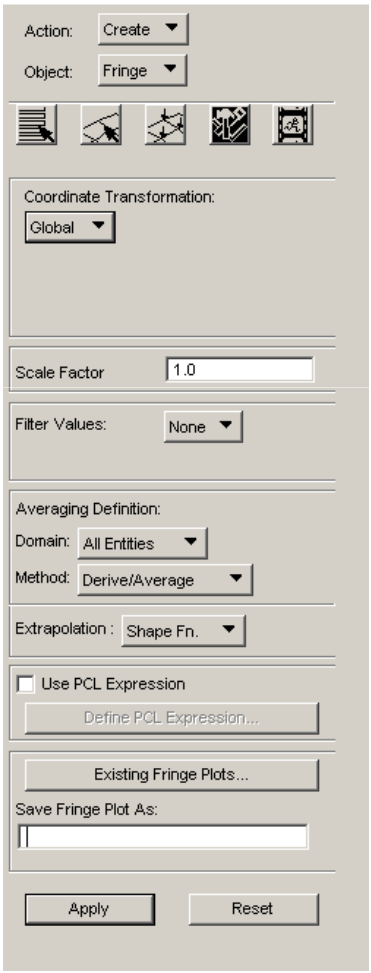
h Select the element on the model

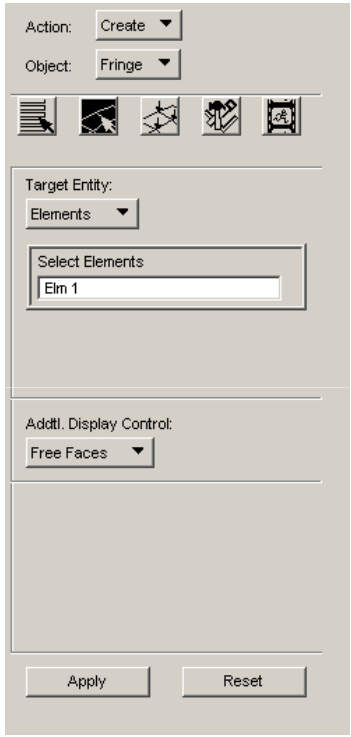
i Click **Apply**

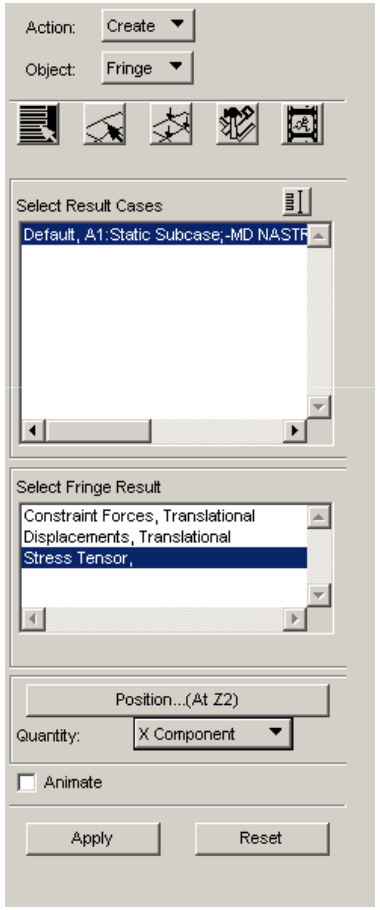
j Select the Icon **Plot options**

k Select **Global** as Coordinate Transformation

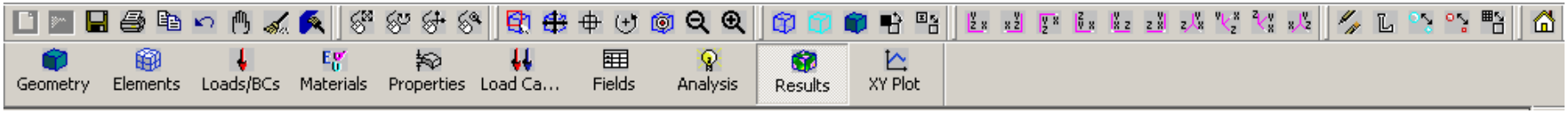
i Click **Apply**







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

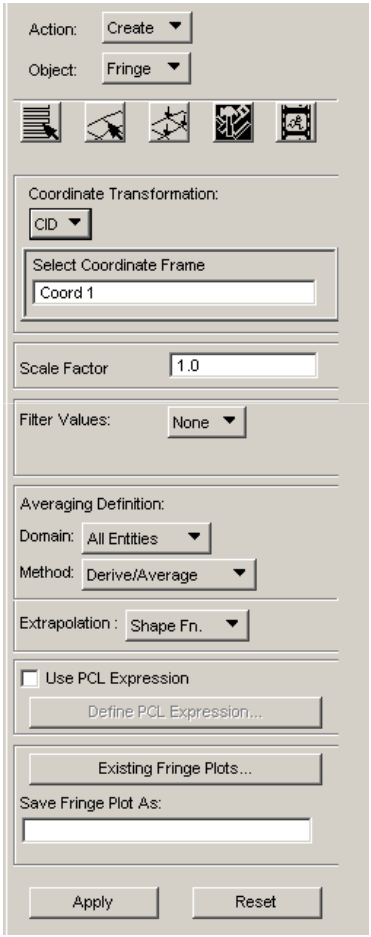
i Click **Apply**

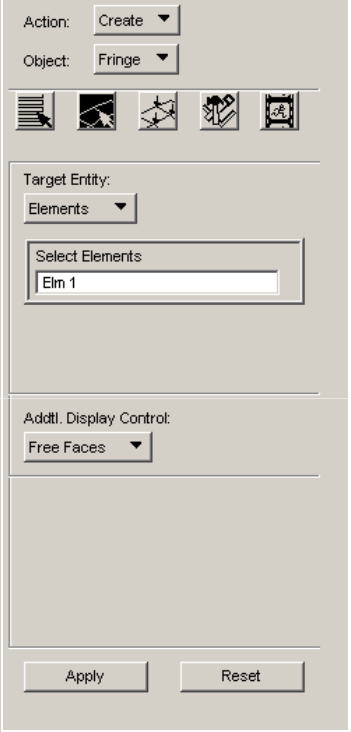
j Select the Icon **Plot options**

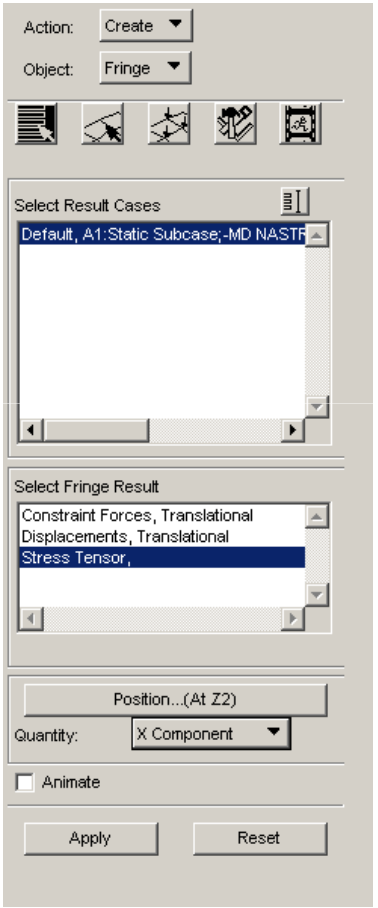
k Select **CID** as Coordinate Transformation

l Select the axis system on the model

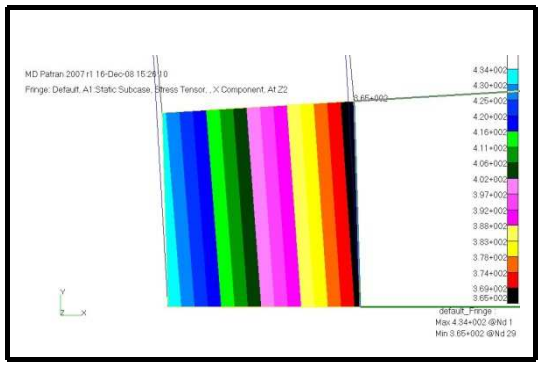
m Click **Apply**



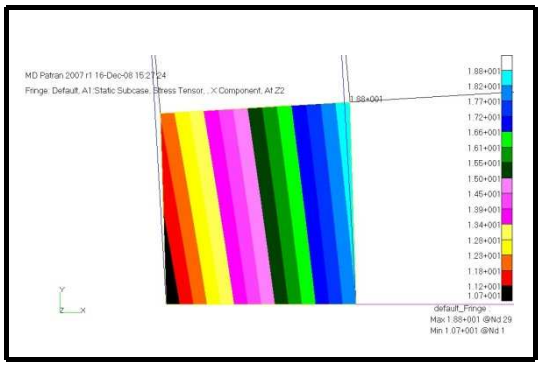




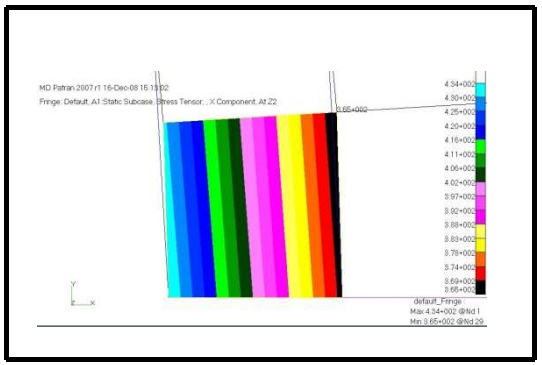
Comparison of the different results



Local Axis System



Global Axis System



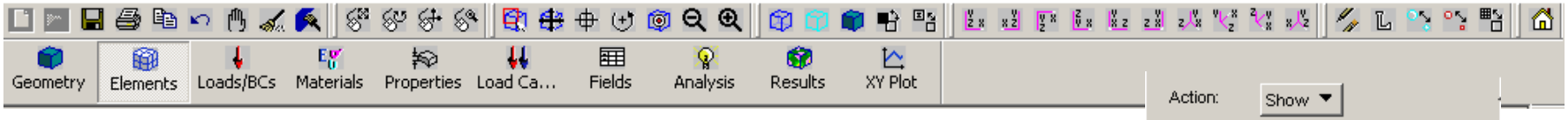
Cylindrical Axis System

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



Create a mesh seed :

a Elements : **Show / Element / Coord. Sys.**

b Unselect **Auto Execute**

c Unselect **use beam Element Offset**.

d Screen Pick the elements for **Select Elements**

e Click **Apply**

f Screen Pick the Curve 2 for **Curve List List**.

g Click **Apply**

h Visualize the result on the screen