Objectives of the WorkShop

For the same structure :

Learn to do a modeling with membrane elements,
Learn to do a modeling with 3D elements,
Learn to do a modeling with beam elements,
Learn to do a modeling with shell elements.

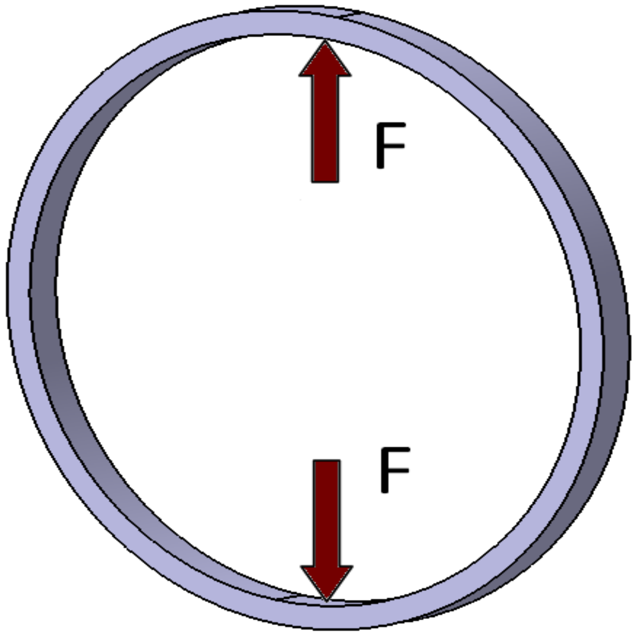
Work to do

Using a 2D modelling with membrane elements verify that
the assumptions of the beam theory are verified for this structure,
Create a modelling with beam elements and compare the results
with the previous one,
Create a modelling with shell elements and compare the results
with the previous ones,
Create a 3D modelling with a cross section such as b=5 mm e=20 mm,
Verify that the shear stress is constant along b and quadratic along e
as the beam theory teach us.

Structure to Design

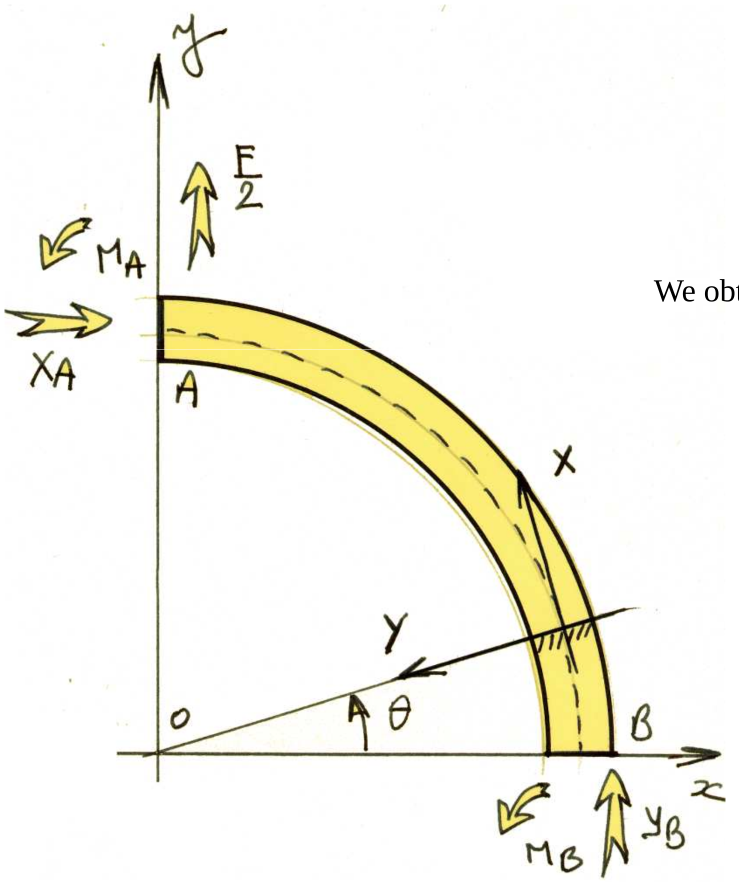
Dynamometer Ring

Width: b = 30 mm
Internal radius: Ri = 240 mm
External radius: Re = 260 mm
Thickness : e = (Re-Ri) = 20 mm
Young Modulus: E = 200 000 MPa
Poisson's ratio: 1/3
F = 2000 N



Analytical Modelling with the theory of beams

The hyperstatic structure is a closed beam, with two planes of symmetry.
Hence, we can study a quarter part of the ring.
If we write the equilibrium of this structure we obtain 3 equations for 4 reactions unknown.



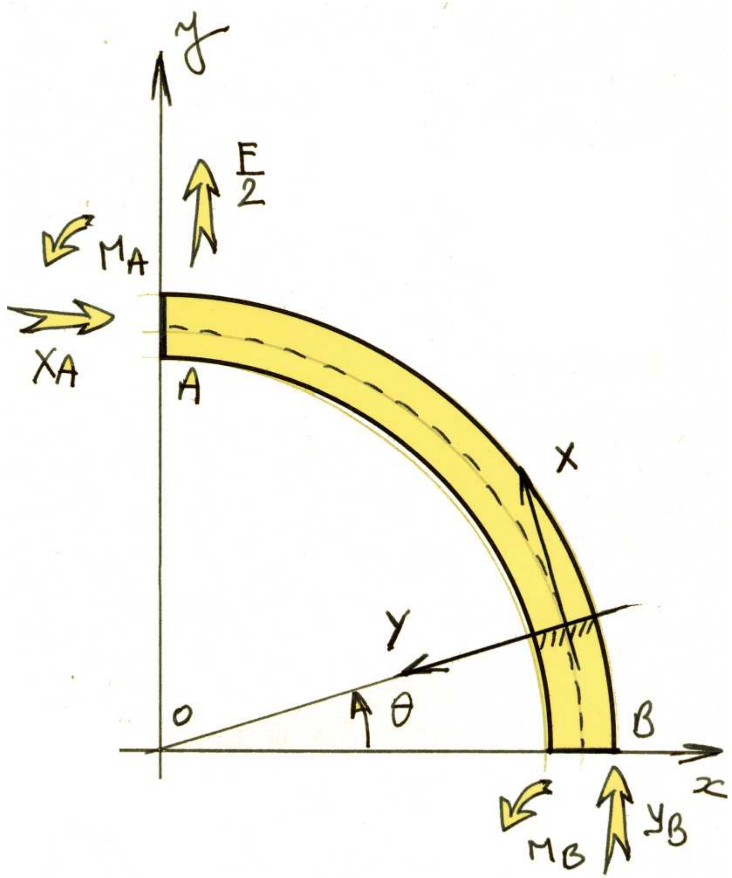
$$\begin{cases} X_A = 0 \\ \dfrac{F}{2} + Y_B = 0 \\ M_A + M_B + RY_B = 0 \end{cases}$$

We obtain a new equation by using the theorem of Castigliano. $\theta_A^z = \dfrac{\partial W}{\partial M_A} = 0$

The internal energy is equal to : $W = \int_0^{\pi/2} \left(\dfrac{1}{2}\dfrac{N_X^2}{ES} + \dfrac{1}{2}\dfrac{T_Y^2}{Gk_Y S} + \dfrac{1}{2}\dfrac{M_Z^2}{EI_Z} \right) r d\theta$

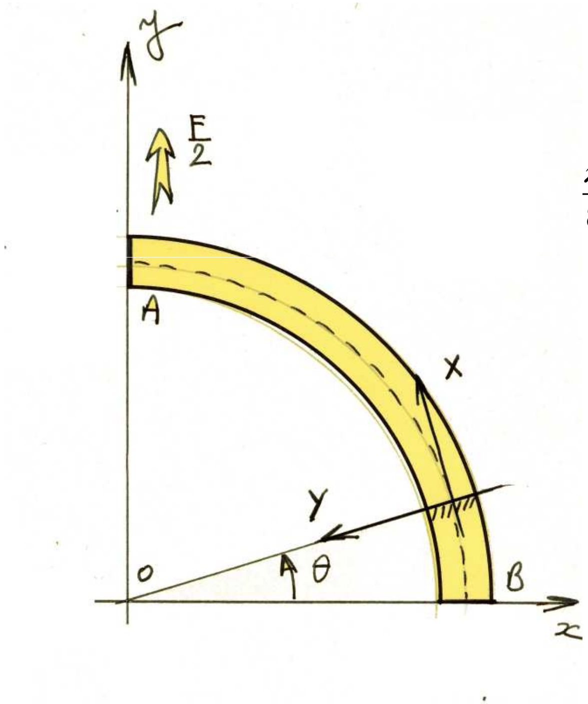
With $\begin{cases} N_X = \dfrac{F\cos\theta}{2} \\ T_Y = -\dfrac{F\sin\theta}{2} \\ M_Z = -\dfrac{FR\cos\theta}{2} + M_A \end{cases}$

Analytical Results : Reactions



$$
\text{Reactions}
\begin{cases}
X_A = 0 \\
Y_B = -\dfrac{F}{2} = -25000 \quad N \\
M_A = \dfrac{FR}{\pi} = 3979 \quad m.N \\
M_B = \dfrac{FR}{2}\left(\dfrac{\pi - 2}{\pi}\right) = 2271 \quad m.N
\end{cases}
$$

Analytical Results : Displacement of the cross section in A



$$\frac{\pi}{8}\frac{R}{ES}F+\frac{\pi}{8}\frac{R}{GK_YS}F+\left(\frac{\pi}{8}-\frac{1}{\pi}\right)\frac{R^3}{EI_Z}F=\begin{cases} v(Nx)=\dfrac{\pi}{8}\dfrac{R}{ES}F=0.04 \quad mm \\ v(Ty)=\dfrac{\pi}{8}\dfrac{R}{GK_YS}F=0.13 \quad mm \\ v(Mz)=\left(\dfrac{\pi}{8}-\dfrac{1}{\pi}\right)\dfrac{R^3}{EI_Z}F=14.53 \quad mm \end{cases}$$

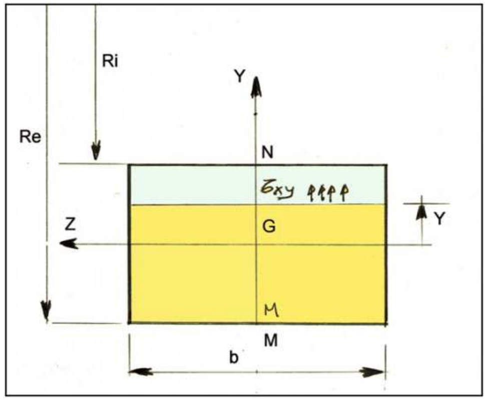
Analytical Results : Stresses

Normal Force
Shear Force
Bending Moment

$$\begin{cases} N_X = \dfrac{F\cos\theta}{2} \\ T_Y = -\dfrac{F\sin\theta}{2} \\ M_Z = FR\left(\dfrac{1}{\pi} - \dfrac{\cos\theta}{2}\right) \end{cases}$$

Normal Stresses in the principal axis : $\sigma_X = \dfrac{N_X}{eb} - \dfrac{6M_Z}{be^3}Y$

Shear Stresses in the principal axis : $\tau_{XY} = \dfrac{12T_Y}{be^3}\left(\dfrac{e^2}{8} - \dfrac{Y^2}{2}\right)$



Results

E	200000	MPa	Young's modulus
ν	0,33		Poisson's ratio
G	75187,96992	MPa	Shear modulus
b	30	mm	Width
R_e	260	mm	External radius
R_i	240	mm	Internal radius
$e=(R_e-R_i)$	20	mm	Thickness
$R=(R_e+R_i)/2$	250	mm	Average Radius
$I_Z=be^3/12$	20000	mm4	Quadratic moment
$S=eb$	600	mm2	Area of the cross section
F	50000	N	Load Applied on the ring
θ	0		Angle in degre
θ	0		Angle in radian
Nx	25000	N	Normal Load

Ty	0	N	Shear Force
M_A	3978873,6	mm.N	Reaction
Mz	-2271126,4	mm.N	Bending moment
Y max	10	mm	Maximum of Y
Y min	-10	mm	Minimum of Y
σ_X	41,7	MPa	Normal Stress by Nx
σ_X	1135,6	MPa	Max normal stress by Mz
σ_X	-1135,6	MPa	Min stresse by Mz
σ_X	1177,2	MPa	MPa
σ_X	-1093,9	MPa	MPa
Ky	0,833333333		Coefficient of reduced Section
σ_X	0,04	mm	Displacement by Nx
v_{Ty}	0,13	mm	Displacement by Ty
v_{Mz}	14,53	mm	Displacement by Mz

Geometry

Points

Point Identifier	x	y
PT 1	0	0
PT 2	0	140
PT 3	0	160
PT 4	160	0
PT 5	140	0

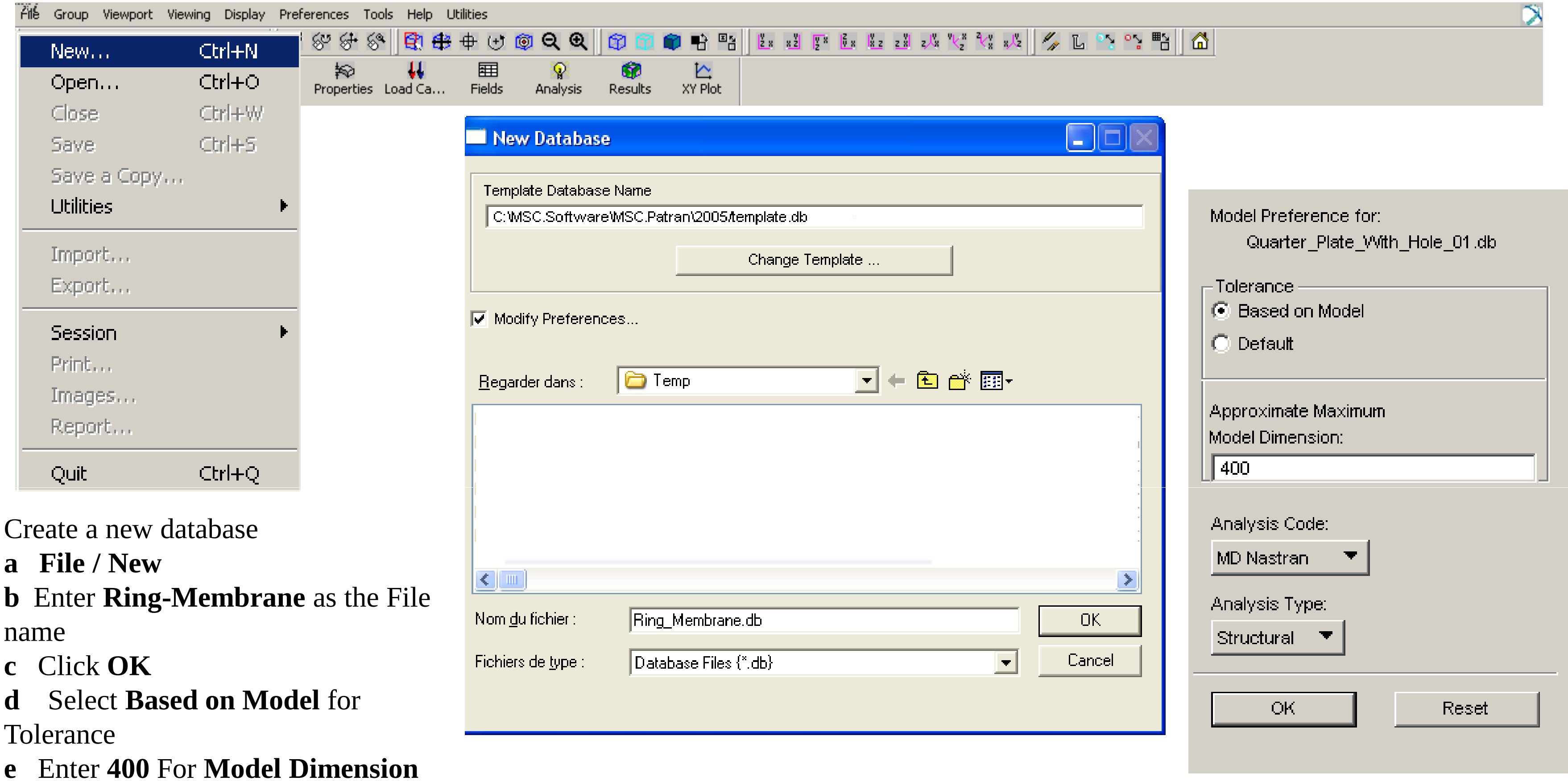
Curves

Curve identifier	Center	1° Point	2° Point
CRV 1	PT 1	PT 2	PT 3
CRV 2	PT 1	PT 5	PT 4

Surfaces

Surface identifier	1° Curve	2° Curve
SUR 1	CRV 1	CRV 2

Step 1 : Creation of a New Database



Create a new database

a **File / New**

b Enter **Ring-Membrane** as the File name

c Click **OK**

d Select **Based on Model** for Tolerance

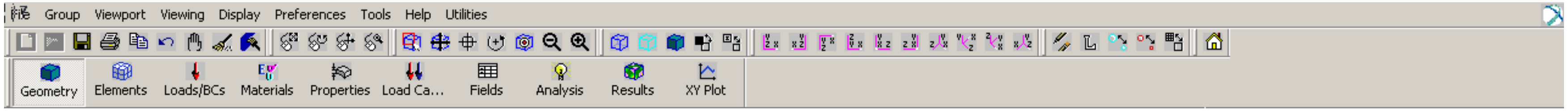
e Enter **400** For **Model Dimension**

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

g Select **Structural** as the Analysis Type

h Click **OK**

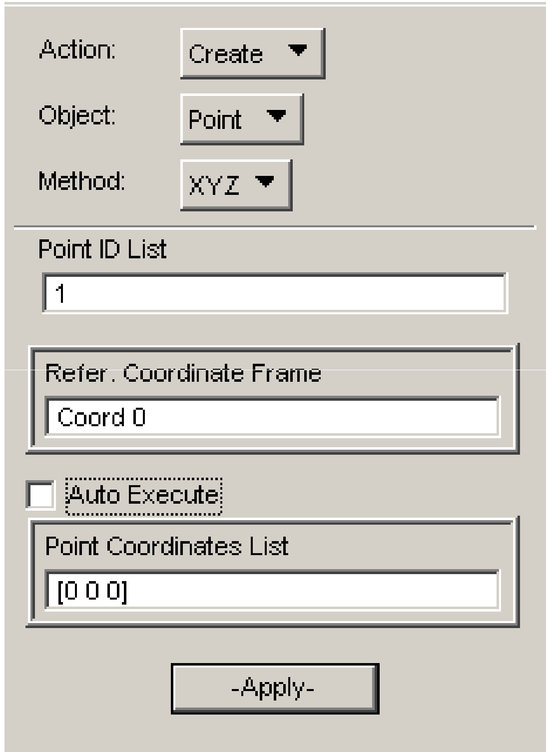
Step 1 : Creation of Geometry - Points



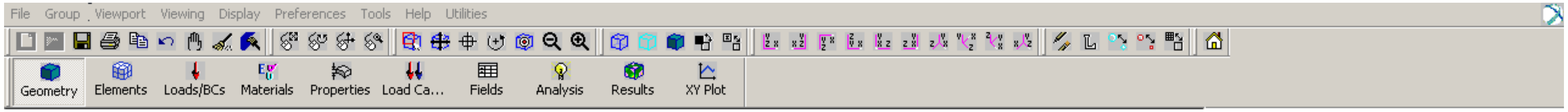
Create the first point :

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 (PT1….PT5) with the same method.



Step 1 : Creation of Geometry – Curves



Create the curve CRV1(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 PT1 for the **Center Point List.**

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

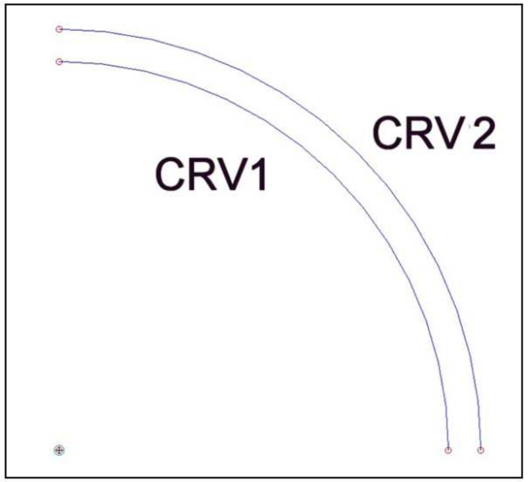
g Screen Pick the pointPT2 for the **Starting Point List.**

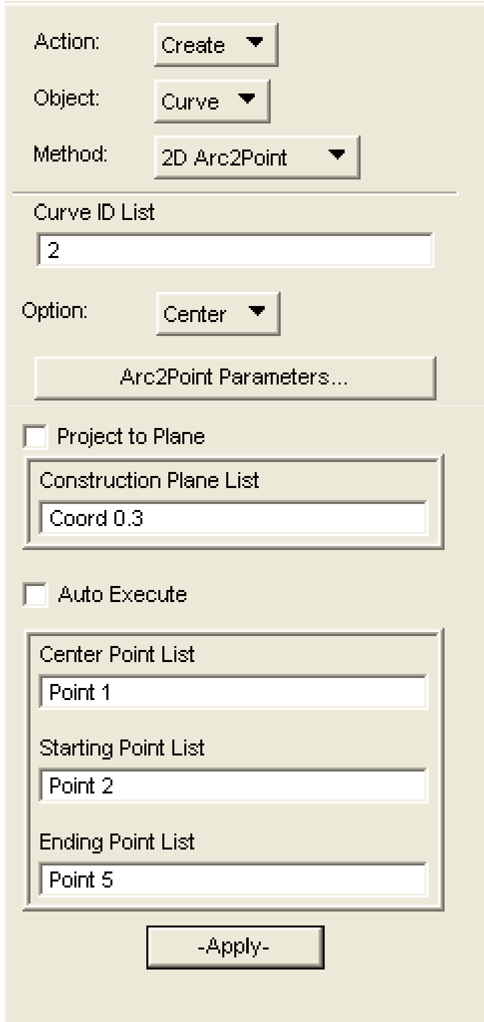
h Activate **Ending Point List** with the left click

i Pick the pointPT5 for the **Ending Point List.**

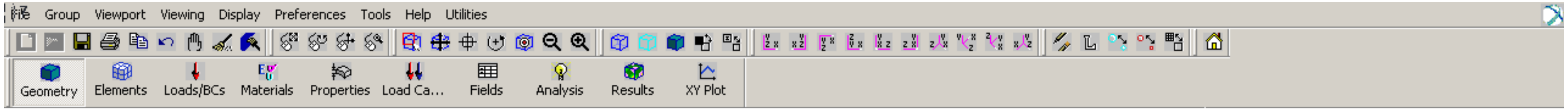
j Click **Apply**

Create the curvez CRV2 with the same process



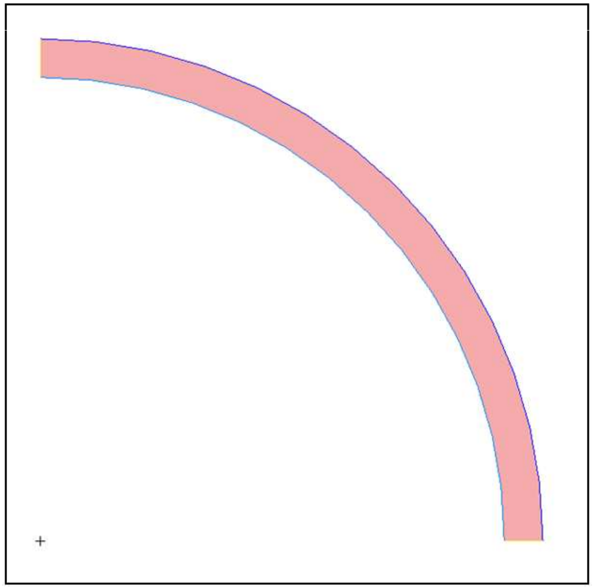


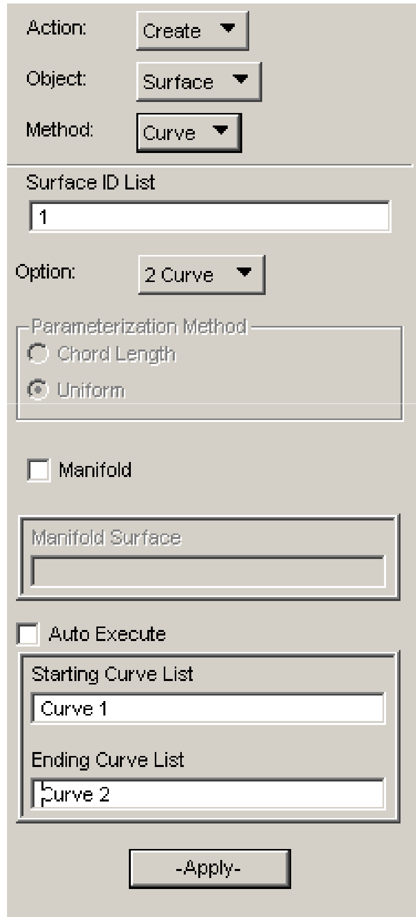
Step 1 : Creation of Geometry - Surface



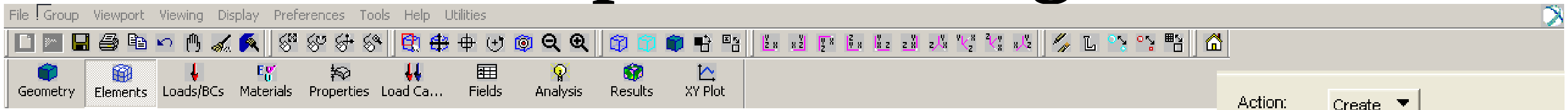
Create Surfaces with 2 curvilinear boundaries :

a Geometry : **Create / Surface / Curve**
b Select Option : **2 Curves**
c Unselect **Auto Execute**
d Screen Pick the Curve 1 CRV1in **Starting Curve List**.
e Screen Pick the Curve 2 CRV2 in **Ending Curve List**.
f Click **Apply**
g Use the shading Icon to visualize the surface





Step 2 : Meshing



Create a mesh seed :

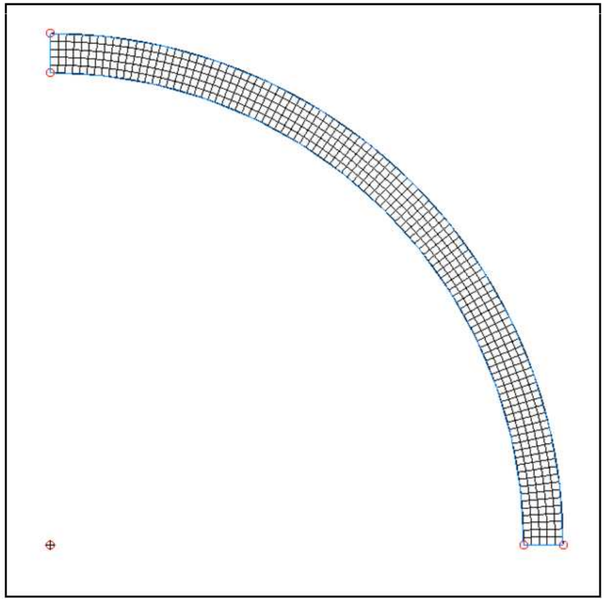
a Elements : **Create / Mesh / Surface**

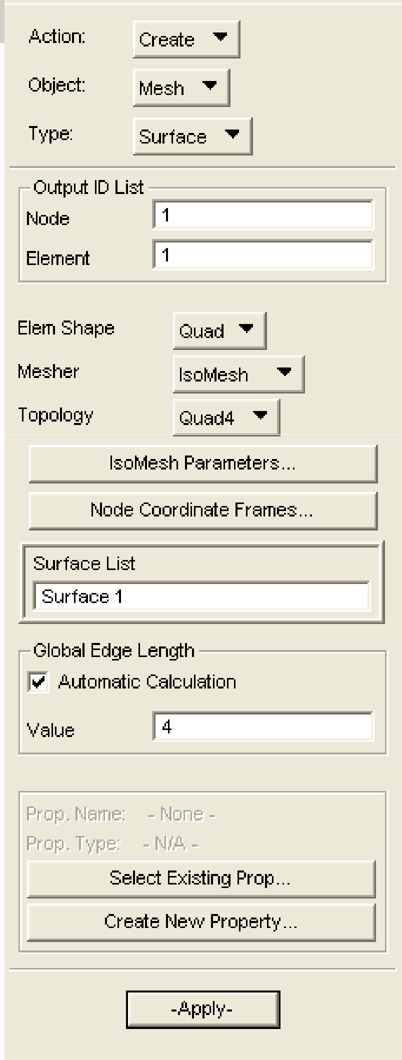
b Select **Quad/IsoMesh/Quad4**

c Grab the Surface1 in **Surface list**

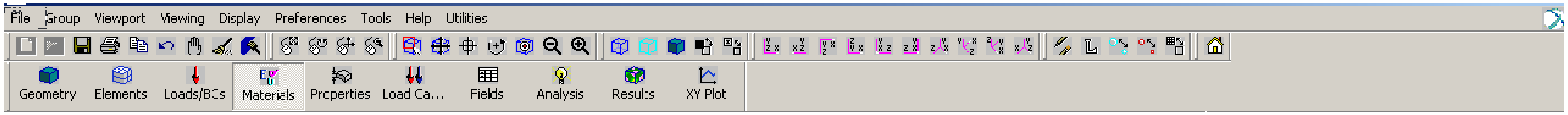
d Enter 4 in **Value** for the size of the edge of the quad4

e Click **Apply**



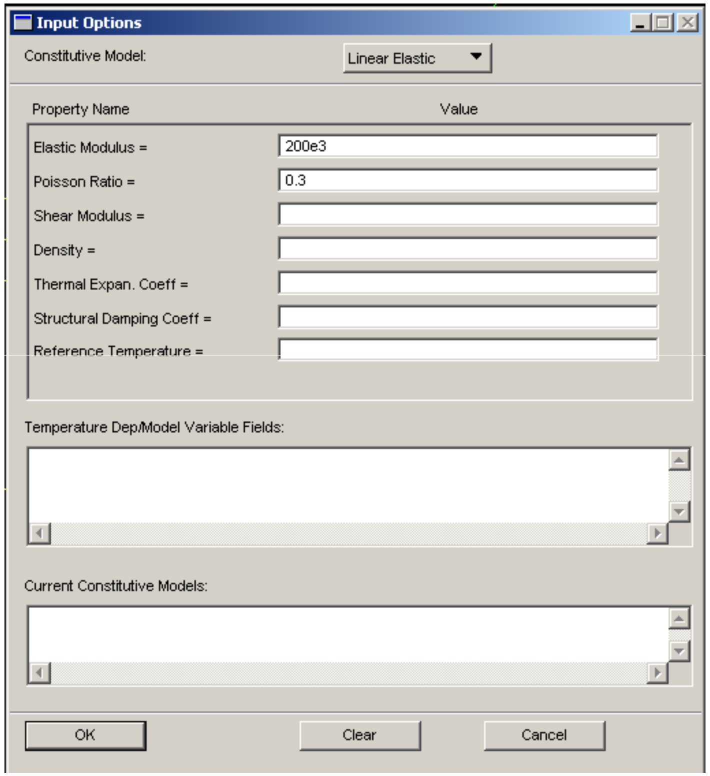


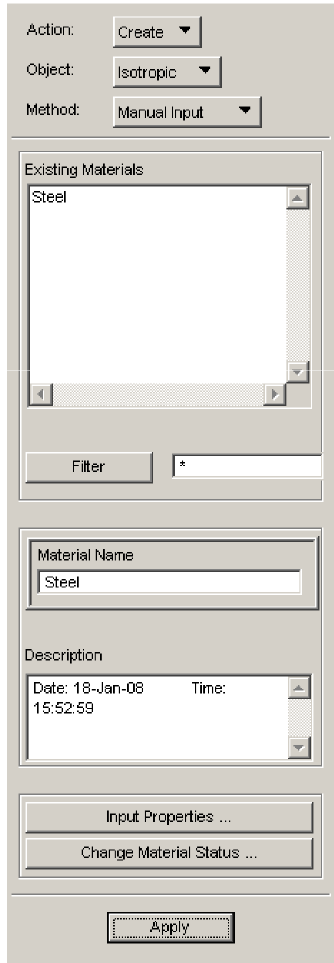
Step 4 Create material



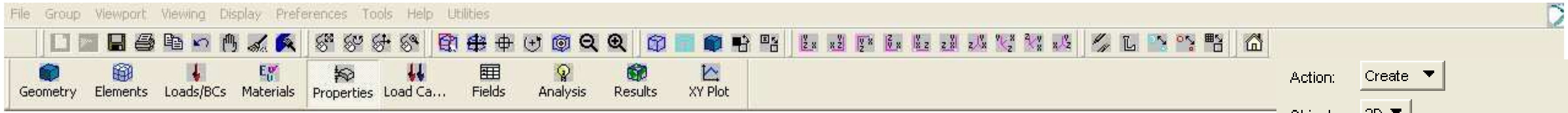
Create an isotropic material :

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**



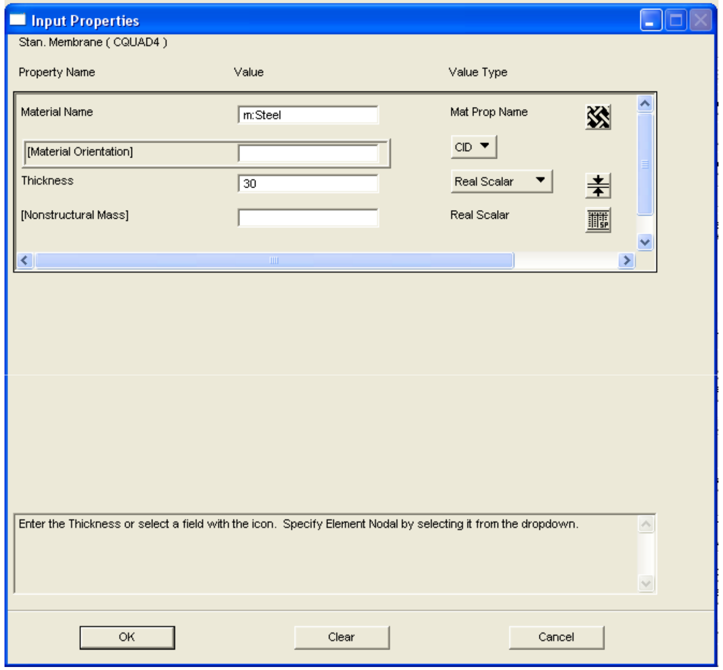


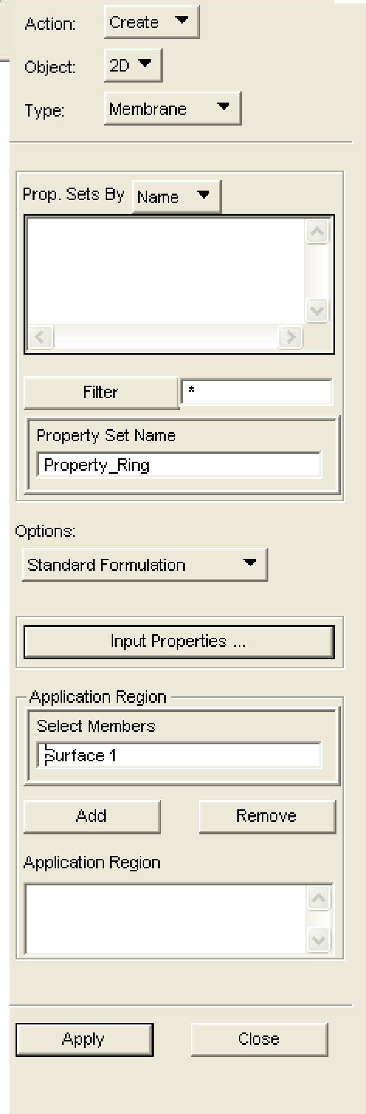
Step 5 : Create Physical Properties



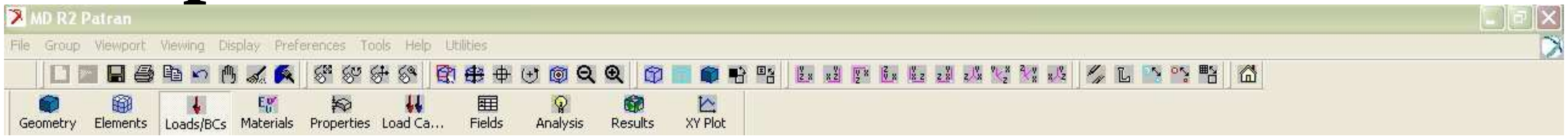
Create element properties :

a Properties : Create / 2D / Membrane
b Enter **Property_Ring** 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 30 as thickness for the plate
g Click **OK**
h Click on **Select Members**
i Drag over the surface to select it
j Click on **Add**
k Click on **Apply**





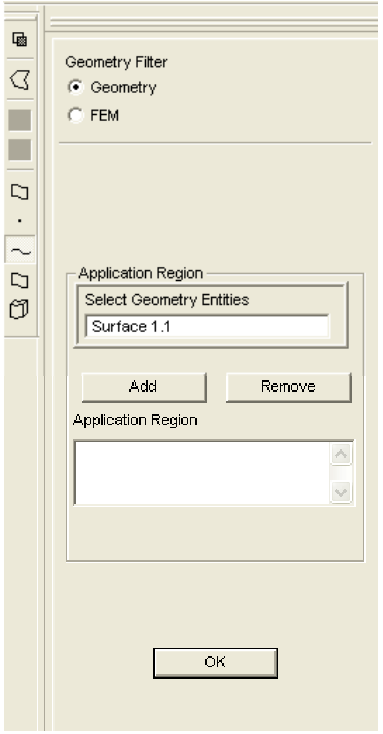
Step 6 :Create Boundaries Conditions

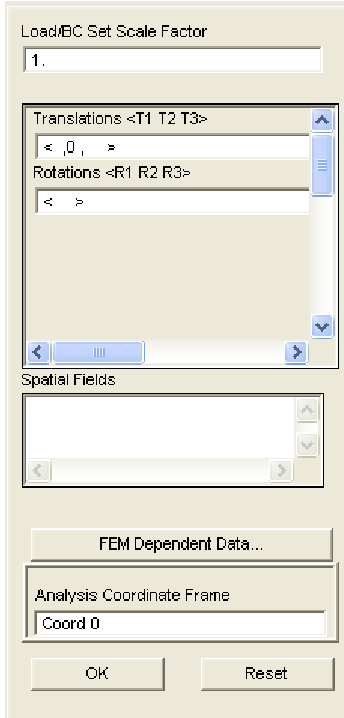


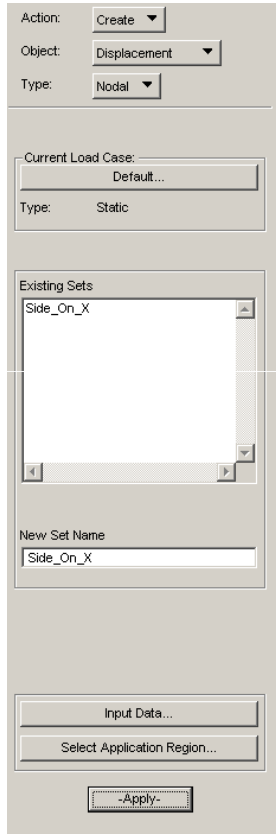
Create boundary conditions :

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 **Geometry**
h Select the icon **Curve or edge** 
i Screen pick the good edge of the Surface 1
j Click **Add**
k Click **OK**
l Click **Apply**
m Verify that the boundaries Conditions have been created

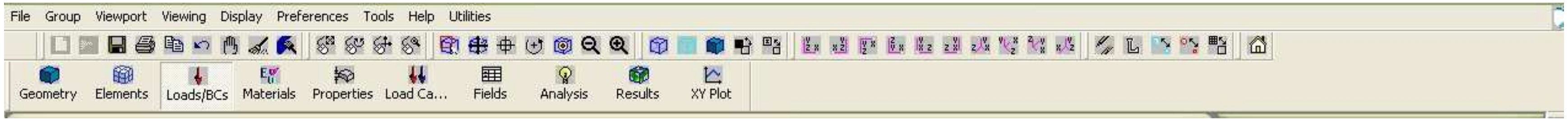
Do the same thing with
a New Set named **Side_On_Y**







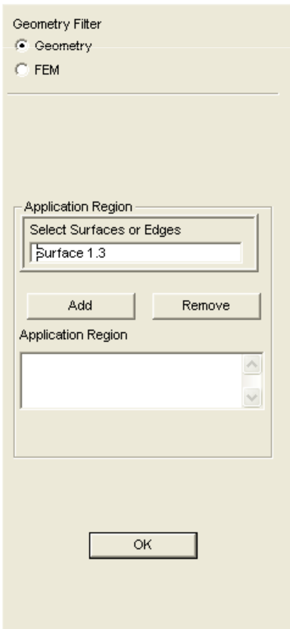
Step 7 : Create Load

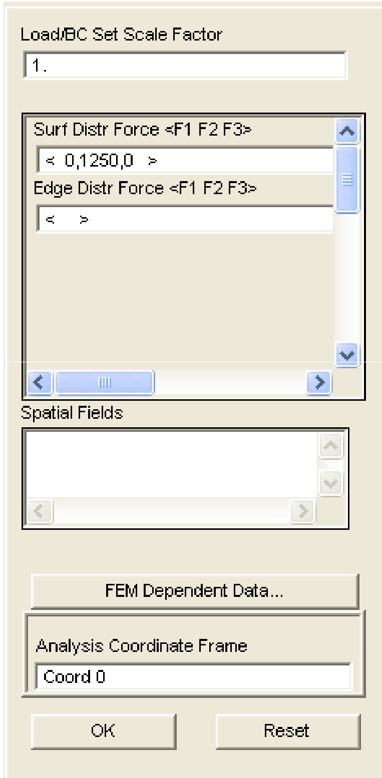


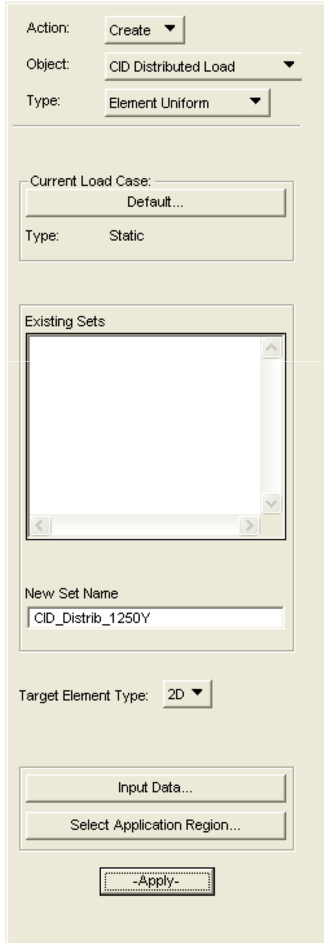
Create a constant uniform load :

a Loads/BCs : Create / CID Distributed Load / Element Uniform
b Enter **CID_Distrib_1250Y** as **New Set Name**
c Select 2D with icon **Target Element Type**
d Click on **INPUT Data**.
e Enter < **, 1250,** > for components of the distributed load.
f Verify that the axis system for the load is the good in the field **Analysis Coordinate Frame**
g Click **OK**
h Click on **Select Application Region**
i Select **Geometry**
j pick up the boundary of the surface
k Click on **Add**
l Click **OK**
m Click **Apply**

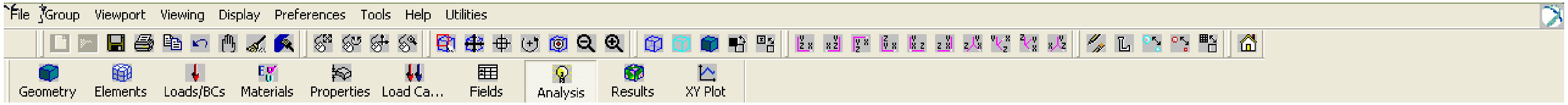
REMARK : With CID Distributed Load you apply a load with components defined in an axis system you choose.







Step 8 : Run Nastran



Computation

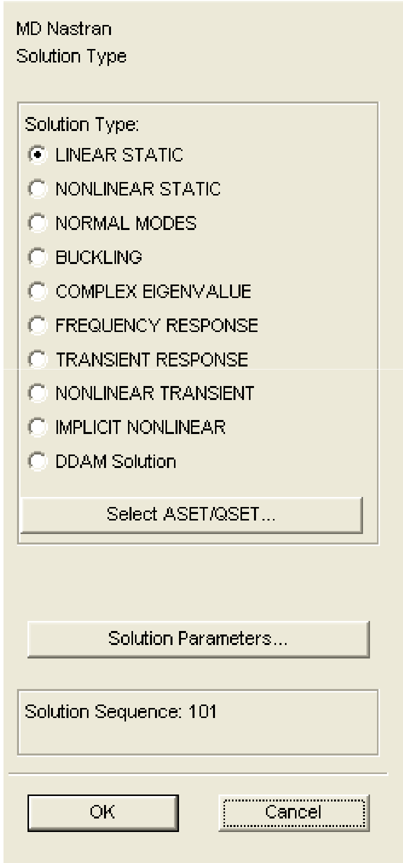
a Analysis : Analyse / Entire Model / Full Run

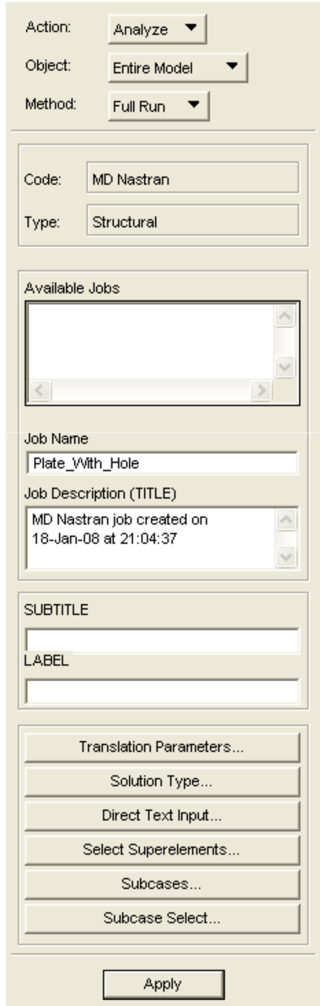
b Select **Solution Type**

c Select **Linear Static**

d Click **OK**

f Click **Apply**





Analysis of the results

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

* Open the BDF file to verify if the nodes, the elements, the material, the properties, the load, the boundary conditions have been defined correctly.

* Open the F06 file to verify if your model has been computed correctly, with the good accuracy. Verify if OLAOD and EPSILON are good. Verify that the model performs.

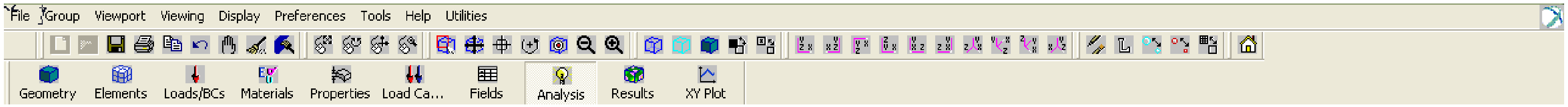
After that you can import a specific output file inside Patran to analyse the results. Two different output files can be generated by NASTRAN :

*. XDB output file (used at ISAE)

*.OP2

To import the output file in Patran see the following slide

Step 8 : Load Nastran Results in Patran



Load the results file of NASTRAN

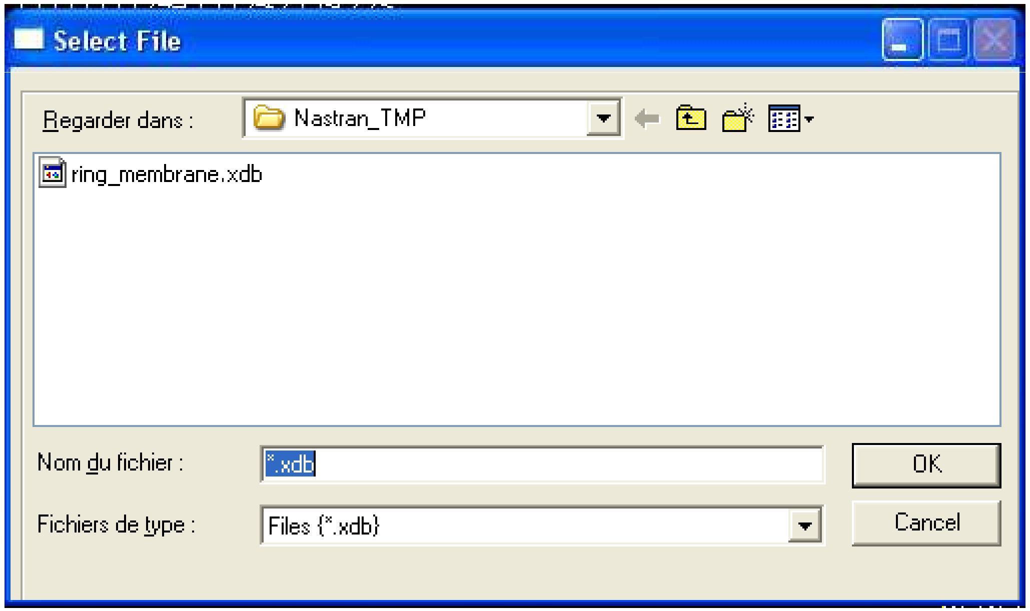
a Analysis : Access Results / Attach XDB / Result Entities

b Click on **Select Results File**

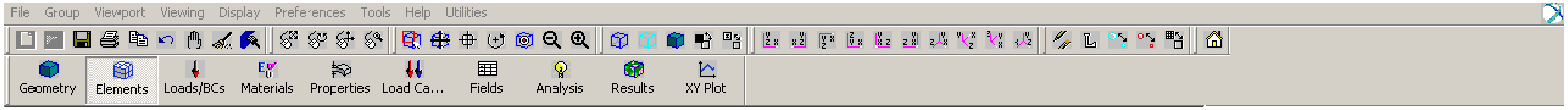
c Select the file results **Ring_membrane.XDB**

d Click **OK**

e Click **Apply**



How to identify a membrane element



If you want to find in the F06 file the results in one or more elements you can find the identifier of this element with the following process

Identify one (or More) element(s)

:

a Elements: Show / Element / Attributes

b Screen Select the element you want to know the identifier

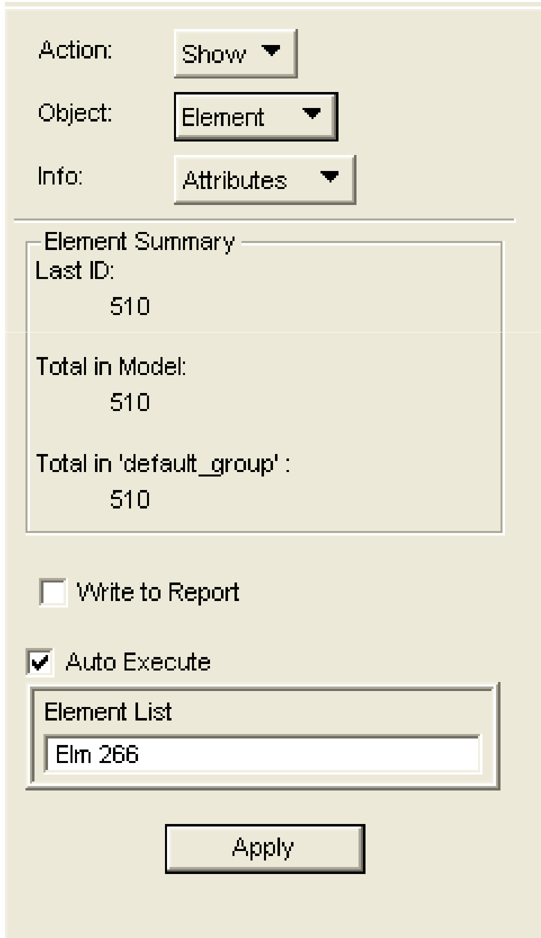
c Click **Apply**

d Read the identification number of the element in the windows that appears.

e Open the file job.F06

f Find the data of the element(s)

g Analyse the results



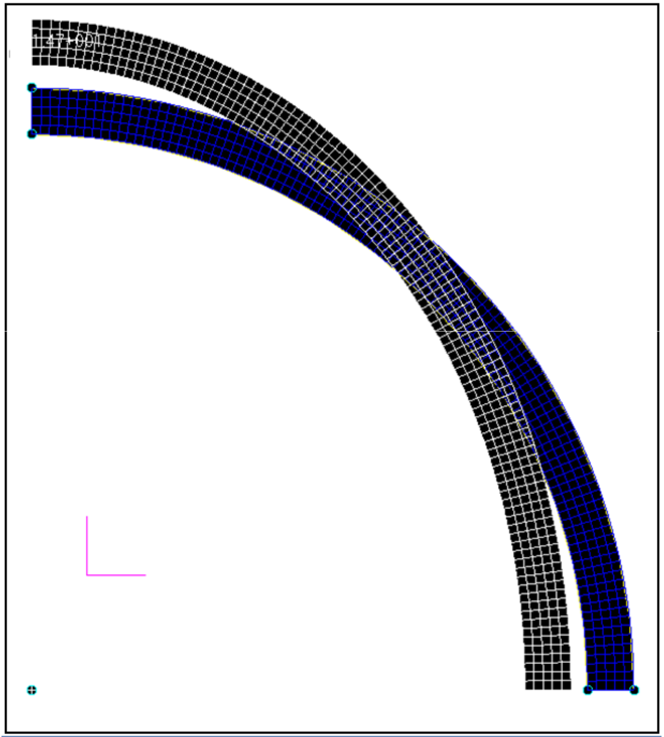
Step 9 : Results - Deformation

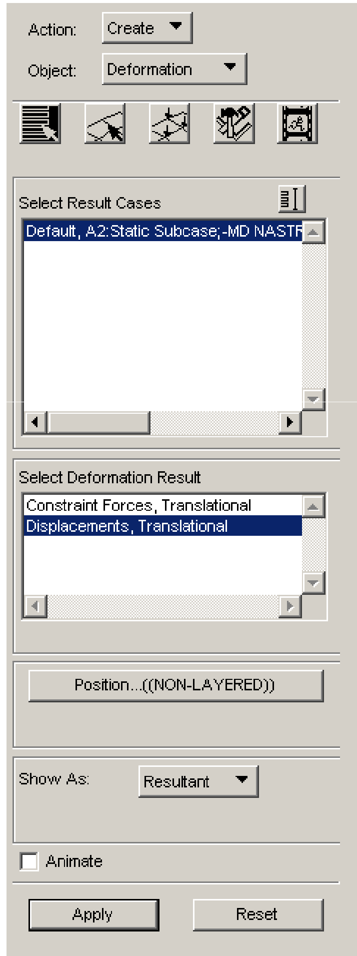
Verify the displacement :

a Results : Create / Deformation

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

c Click **Apply**





Remark
Verify the shape of the displacement,
Verify the boundary conditions.

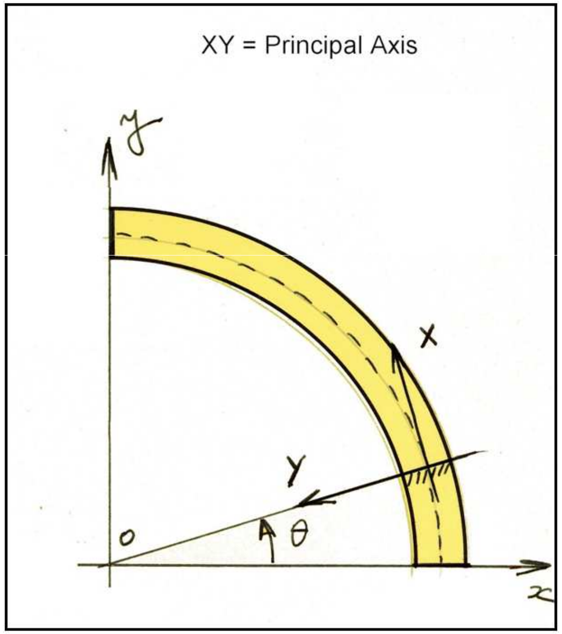
Analysis of the results

In the principal axis of the cross section we know that, according to the beam theory, we have the following results.

$$\sigma_X = -\frac{M_z}{I_z}Y + \frac{N_X}{S}$$

$$\sigma_Y = 0$$

$$\tau_{XY} = \frac{T_Y}{I_Z}\left(\frac{e^2}{8} - \frac{Y^2}{2}\right)$$



To verify if these results are accurate, we need to create an cylindrical axis system and then to create the images described on the following page.

Results : Creation of images

- 1 : Create a Cylindrical axis system to analyse the stresses on the cross section of the ring
- 2 : Create an image of σ_{θ}
- 3 : Create an image of σ_{r}
- 4 : Create an image of $\tau_{r\theta}$

Create a Cylindrical Axis System

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

a Geometry : **Create / Coord / Axis**

b Select **Cylindrical** as Type

c Verify the axis system in witch you will define the vector of the new axis system **(Coordinate Frame)**

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

e With the **KeyBoard** enter the components of a vector for r direction : **[0 0 1]**

f With the **KeyBoard** enter the components of a vector for Theta direction : **[0 1 0]**

g Click **Apply**

h Verify that the axis system has been created

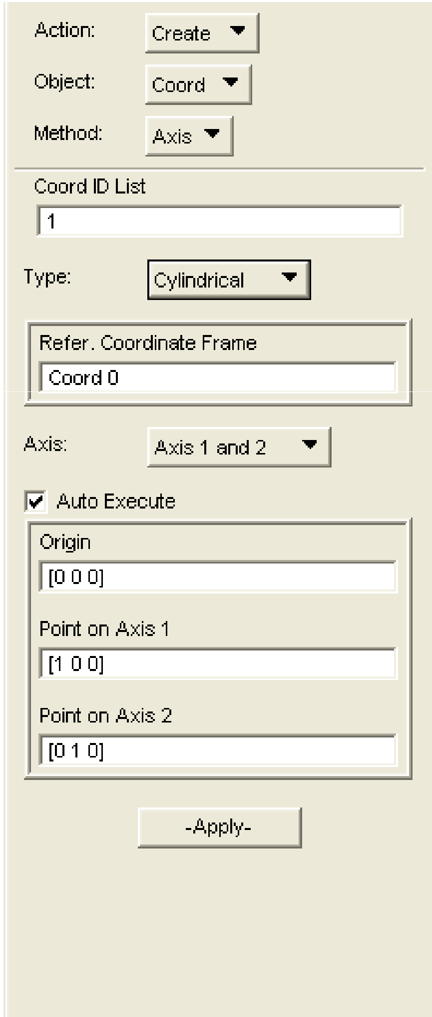


Image for the shear stress

a Results : **Create / Fringe**

b Select the Icon **Select results**

c Select Stress tensor in **Field Fringe Result**

d Select the quantity XY component

e Click **Apply**

f Select the Icon **Target Entities**

g Select Elements as **Target Entity**

h Select the elemenst on the model with the mouse

i Click **Apply**

j Select the Icon **Plot options**

k Select **CID** as Coordinate Transformation

l Select the cylindrical axis system on the model

m Click **Apply**

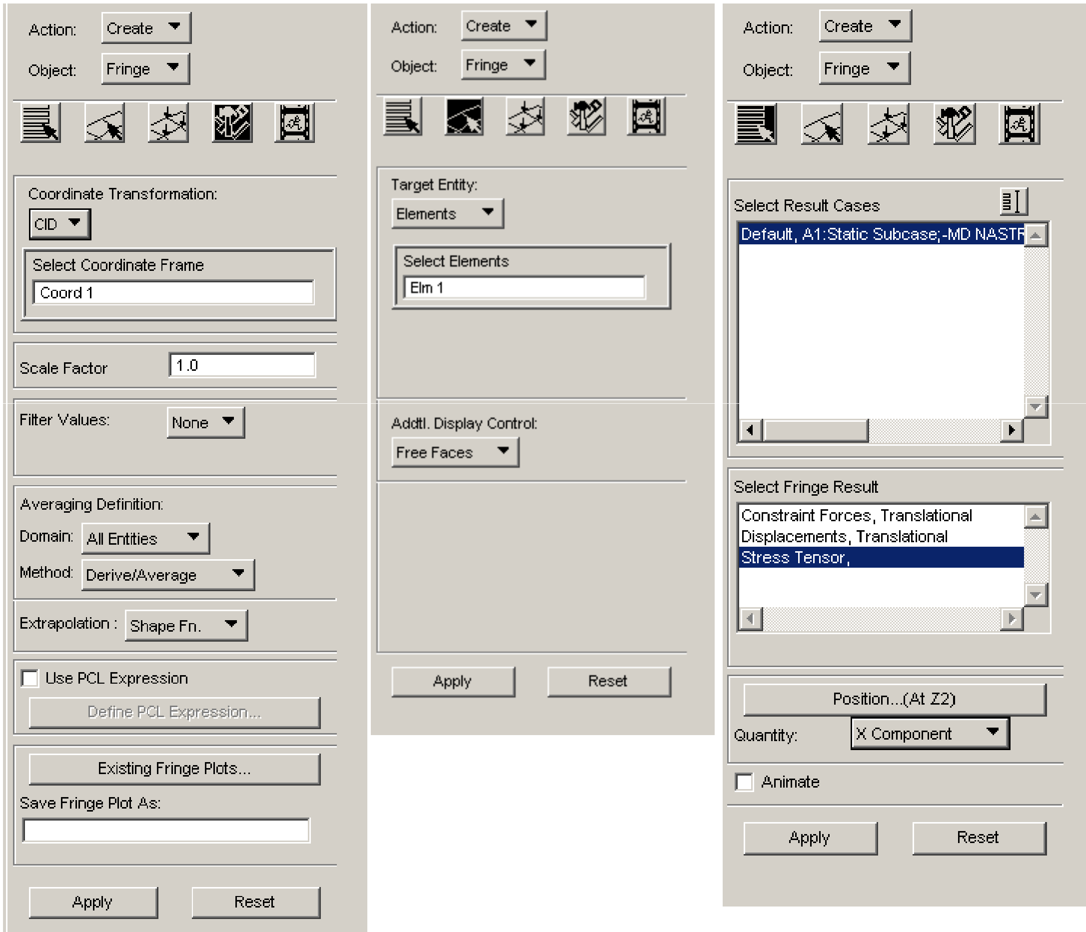


Image for the normal stress in X direction

Remark : The direction X of the principal axis of the cross section is the second direction of the cylindrical axis system (the direction θ)

Principal Axis of the cross section	Cylindrical axis system
X	θ
Y	r
Z	z

Cylindrical axis system	Name in PATRAN for Images
r	X
θ	Y
z	Z

a Results : **Create / Fringe**

b Select the Icon **Select results**

c Select Stress tensor in **Field Fringe Result**

d Select the quantity XY component

e Click **Apply**

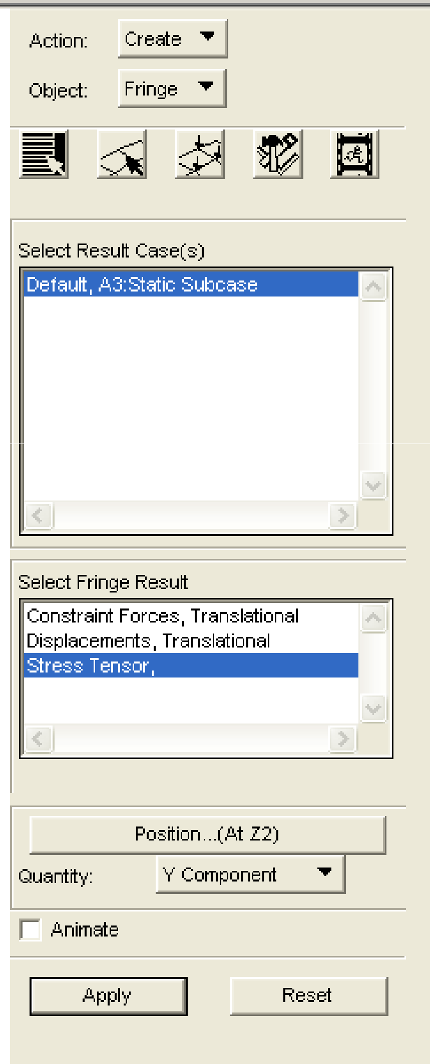


Image for the normal stress in Y direction

Remark : The direction Y of the principal axis of the cross section is the first direction of the cylindrical axis system (the direction r)

Principal Axis of the cross section	Cylindrical axis system
X	θ
Y	r
Z	z

Cylindrical axis system	Name in PATRAN for Images
r	X
θ	Y
z	Z

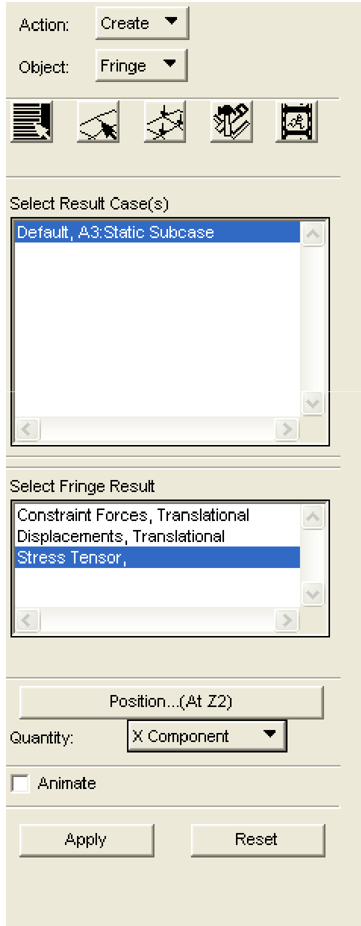
a Results : **Create / Fringe**

b Select the Icon **Select results**

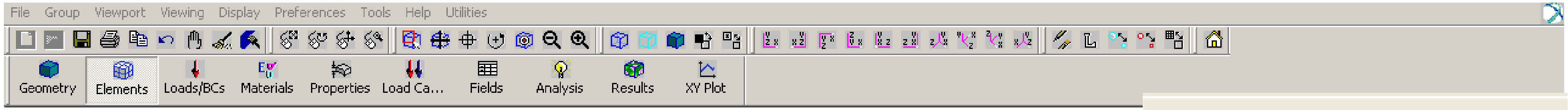
c Select Stress tensor in **Field Fringe Result**

d Select the quantity X component

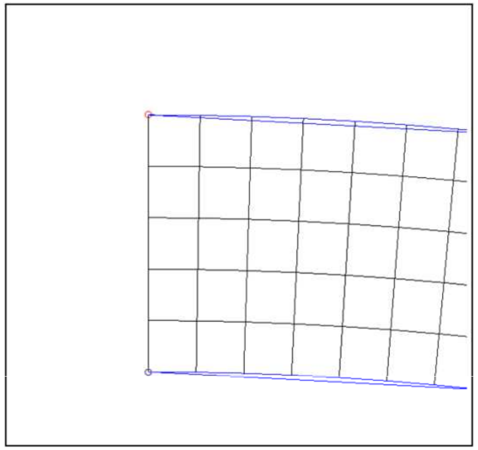
e Click **Apply**



Displacement



Process proposed :
Find the identifier of the nodes
Read the values in the f06 File



a Elements: **Show / Element / Location**

b Screen Select the node you want to know the identifier

c Read the identifier of the nodes in the box **Node List**

d Open the *.F06 File

f Read the displacements of the nodes 103 and 618

g Compare theses values with the analytical one (14.70 mm)

101	G	-7.488257E-02	1.463087E+01	0.0	0.0	0.0	0.0
102	G	-3.793158E-02	1.465303E+01	0.0	0.0	0.0	0.0
103	G	0.0	1.466187E+01	0.0	0.0	0.0	0.0
104	G	-1.337491E+01	0.0	0.0	0.0	0.0	0.0
105	G	-1.337089E+01	1.354034E-02	0.0	0.0	0.0	0.0

616	G	7.571381E-02	1.462391E+01	0.0	0.0	0.0	0.0
617	G	3.886577E-02	1.464955E+01	0.0	0.0	0.0	0.0
618	G	0.0	1.465968E+01	0.0	0.0	0.0	0.0