Workshop Objectives

- Learn to create a geometry with PATRAN,
- Learn to mesh, to create nodes, elements, and connectivity with PATRAN,
- Learn to define properties of the beam elements,
- Learn to define the boundary conditions and the load,
- Learn to create and manage several cases for the load and the BC,
- Learn to read the input file for NASTRAN : *.BDF file
- Learn to read the output file of NASTRAN (file result) : *.F06 file,
- Learn to use NASTRAN directly with a BDF file,
- Learn to use PATRAN as a post processor

Create a shortcut on the desk

Verify on the desk if there is a shortcut for PATRAN and NASTRAN

If they don’t exist create them :

Create a shortcut on the desk for Nastran R2
Démarrer / programme / MSC.Software / MD Nastran / MD R2 Nastran

Create a shortcut on the desk for PATRAN R2
Démarrer / programme / MSC.Software / MD Patran R2 / MD R2 Patran

Verify the resolution of your screen

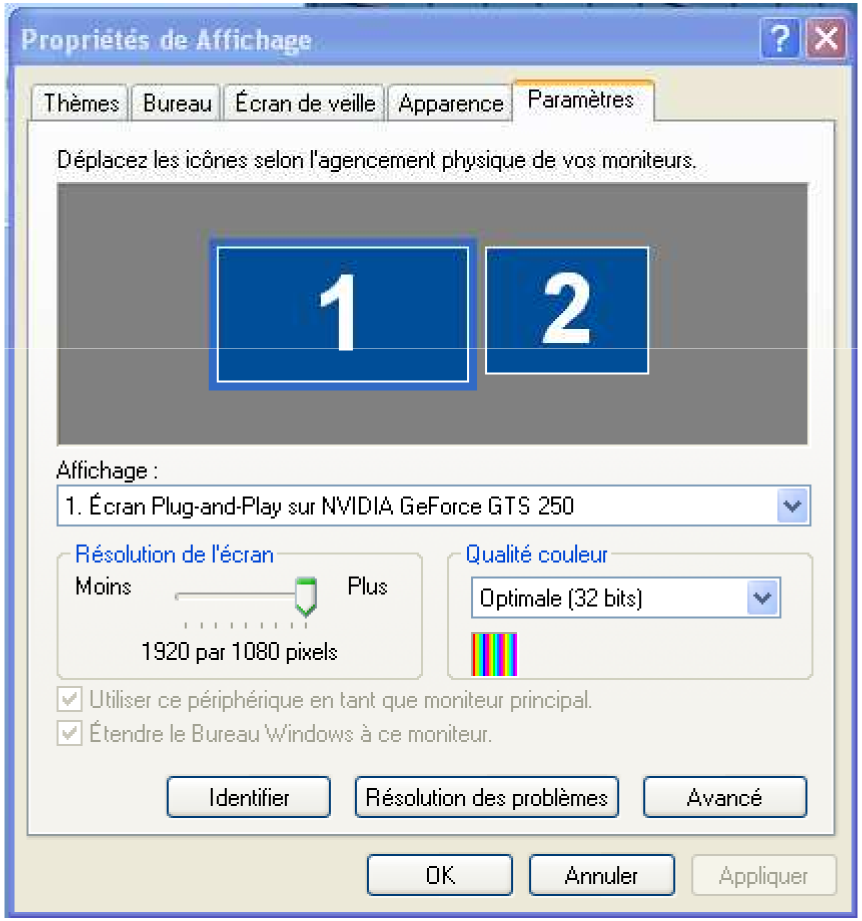
- To work in good conditions you need a resolution of 1280 x 1024 pixel on the screen (or more),

- For that, Right click on the desk and select **Properties**:

- Select **Parameters** and watch the resolution,

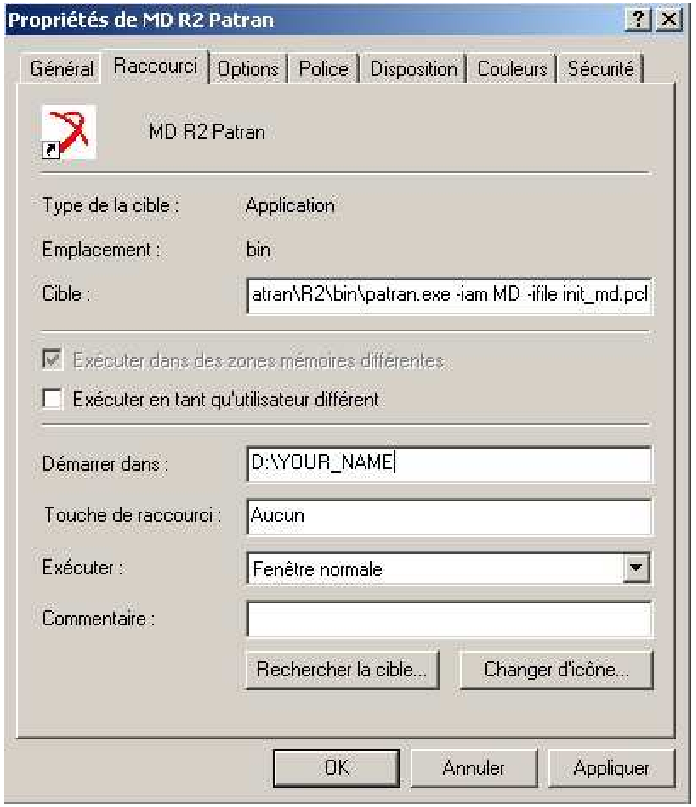
- If the resolution is not the good one, change it with the cursor ,

- Click OK.



Create a directory for your personnal work

- On the hard disk D create a new repertory whose name is your,
- Better solution is to create a repertory in your personal environment,
- On the desk, select **Propriétés** of Patran R2 after a right click mouse on the icon **MD R2 Patran,**
- Write the path of your working directory in the field **Démarrer dans,**
- Click OK,
- Do the same thing for NASTRAN with the Icon MD R2 Nastran,
- Then you can start PATRAN from the icon on the Desk.

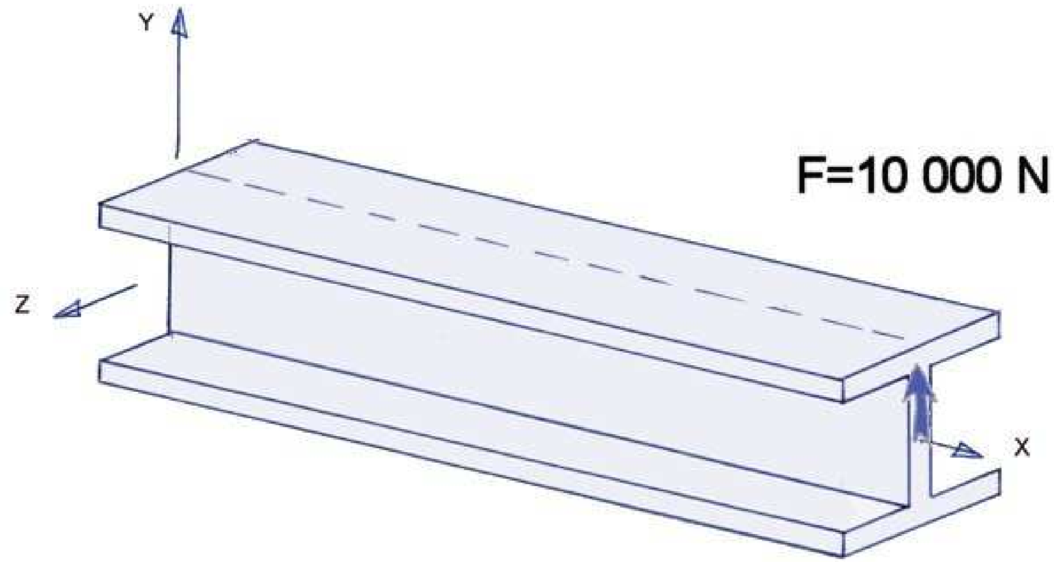


Quick Reference Guide

Open the **Quick Reference Guide in *Nastran_Library.pdf* file**.

C:\MSC.Software\MD_Patran\R2\pdf_patran\Nastran_Library

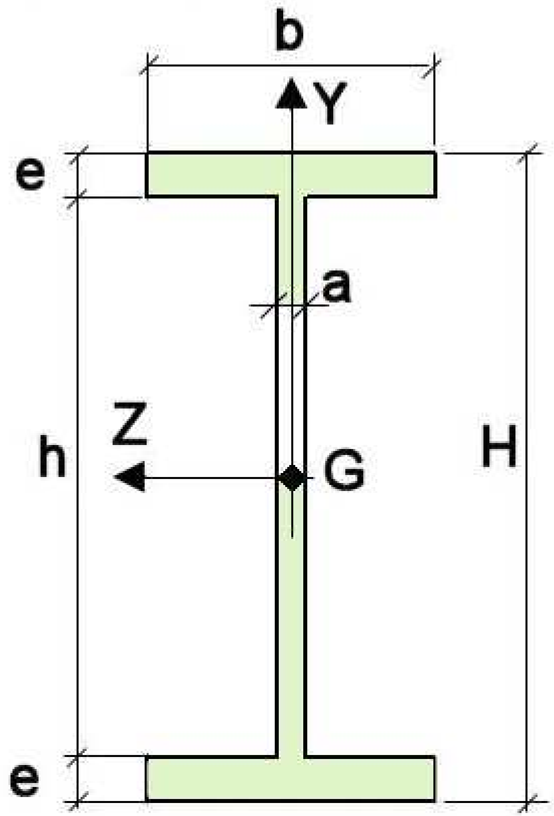
Structure studied : Cantilever with a Punctual Load



The side x = 0 is clamped
The side x = L is free
A punctual force is applied in the centre of gravity of the free side

This force is $\overrightarrow{F} = \left(F_X = 0N \quad F_Y = 10000N \quad F_Z = 0N \right)$

Structure studied : Cross Section

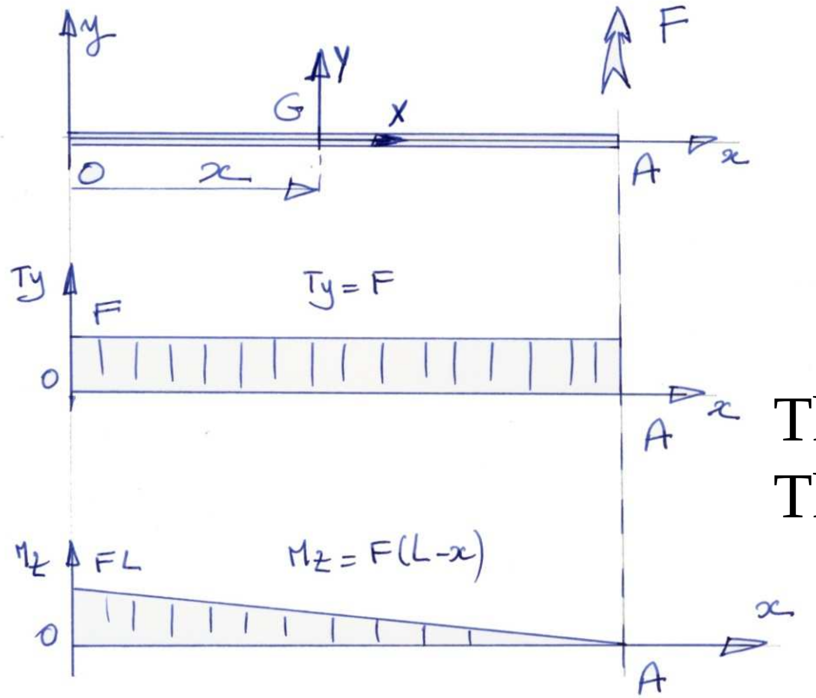


- The length of the cantilever is L = 2 m
- The thickness of each flange is e = 12 mm
- The width of each flange is b = 80 mm
- The web has a thickness a = 8 mm
- The height of the web is h = 136 mm
- The total height is H = 160 mm

The beam is made of steel whose mechanical characteristics are :

- **Young Modulus : 200 000 Mpa**
- **Poisson’s ratio : 0.3**

Internal forces and moments on a cross section



$$M_z = F(l - x)$$

$$T_y = F$$

The shear force is constant versus x

The bending moment is linear versus x

Normal stresses introduced by the bending moment

$$\sigma_X = \frac{M_Z}{I_z} = \frac{F\left(L-x\right)}{I_z} Y$$

The normal stresses are linear versus x AND Y

The maximum normal stress is above and below the cross section clamped.

$$\sigma_X^{Max} = \pm 130.998 \quad MPa$$

Shear Stresses introduced by the shear force
Bredt's Theory

$$\int_C \tau \cdot n \cdot dl = -\frac{T_y}{I_z} \iint_A y.dS$$

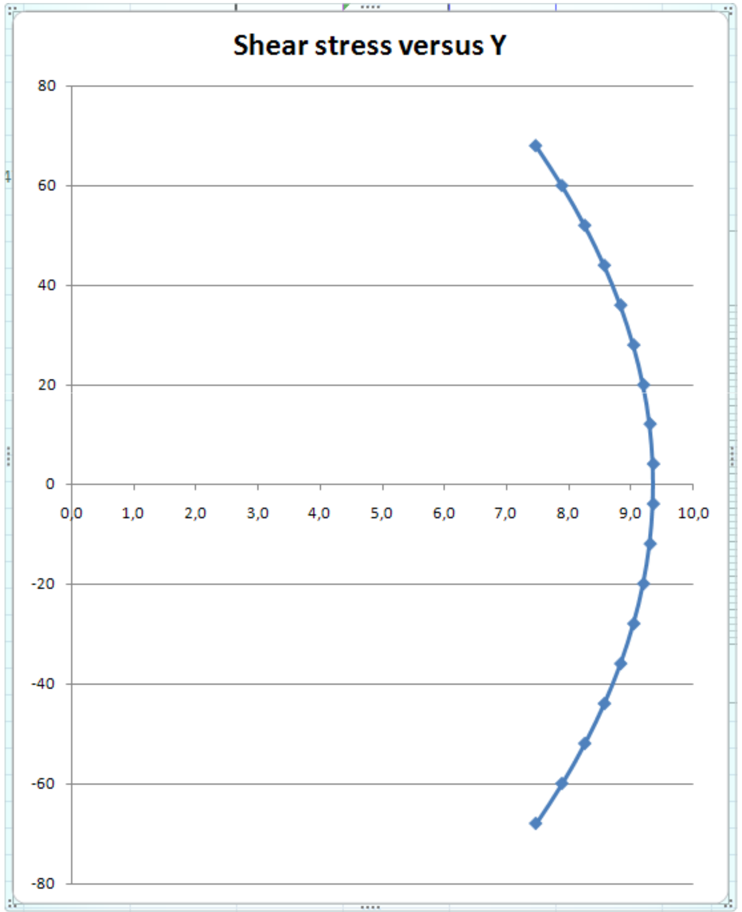
Assumption : The shear stress is constant versus the Z direction in the Web

$$\tau_{xy} = \frac{T_y}{I_z}\left[\frac{eb}{a}\left(\frac{h+H}{4}\right) + \frac{h^2}{8} - \frac{Y^2}{2}\right]$$

The shear stress has a quadratic variation in the Y direction

Shear Stresses introduced by the shear force
Bredt's Theory

Y in mm	Shear stress in MPa
68	7,5
60	7,9
52	8,3
44	8,6
36	8,8
28	9,0
20	9,2
12	9,3
4	9,4
-4	9,4
-12	9,3
-20	9,2
-28	9,0
-36	8,8
-44	8,6
-52	8,3
-60	7,9
-68	7,5



Shear Stresses introduced by the shear force
Simplified theory

$$\tau_{XY} = \frac{T_Y}{S_{Web}} = \frac{F}{ah} = 9.2 \quad MPa$$

Assumptions :

-The shear stress is constant versus the Z direction in the Web
-The shaer force introduces only shear stresses in the web.

Deflexion created by the bending moment Mz

$$\frac{d^2v}{dx^2} = \frac{M_z}{EI_z} = \frac{F\left(L-x\right)}{EI_z}$$

$$\begin{cases} EI_z \dfrac{dv}{dx} = F\left(Lx - \dfrac{x^2}{2}\right) + C_1 \\ EI_z v = F\left(L\dfrac{x^2}{2} - \dfrac{x^3}{6}\right) + C_1 x + C_2 \end{cases}$$

The boundary conditions give $C_1 = 0 \quad , \quad C_2 = 0$

The displacement of the point of the neutral axis of the free side is equal to

$$v_{Mz} = \frac{FL^3}{3 \cdot E \cdot I_z} = 10.92 \quad mm$$

Deflexion created by the Shear Force Ty

$$\frac{dv_{Ty}}{dx} = \frac{T_y}{k_y GS} = \frac{F}{k_y GS} \quad \Rightarrow \quad v = \frac{Fx}{k_y GS} + C_3$$

Boundary Conditions => $C_3 = 0$

The displacement of the point of the neutral axis of the free side is equal to

$$\boxed{v(l) = \frac{FL}{k_y GS}}$$

Two solutions to compute ky

Ky by the simplified theory

$$\frac{dW}{dx} = \frac{1}{2}\frac{T_Y^2}{k_y \cdot GS} = \iint_S \frac{\tau^2}{GS} dS \quad \Rightarrow \quad k_y = \frac{T_Y^2}{S \cdot \iint_S \tau^2 dS}$$

$$\tau_{xy} = \frac{T_Y}{S_{Web}} = \frac{F}{ah} \quad \Rightarrow \quad k_y = \frac{S_{Web}}{S} = \frac{1088}{3008} = 0.362$$

$$\boxed{v_{Ty}(L) = \frac{FL}{k_y GS} = 0.24 \quad mm}$$

Ky by the Bredt's theory

$$\frac{dW}{dx}=\frac{1}{2}\frac{T_Y^2}{k_y\cdot GS}=\iint_S\frac{\tau^2}{GS}dS \quad => \quad k_y=\frac{T_Y^2}{S\cdot\iint_S\tau^2 dS}$$

$$\tau_{xy}=\frac{T_y}{I_z}\left[\frac{eb}{a}\left(\frac{h+H}{4}\right)+\frac{h^2}{8}-\frac{Y^2}{2}\right]$$

$$=> \quad k_y=\frac{5}{3}\frac{a\left(bH^3-h^3b+h^3a\right)^2}{Sh\left(2h^4a^2+10h^2aeb\left(H+h\right)\right)+15e^2b^2\left(h^2+H^2\right)+30e^2b^2hH\Big)}=0.418$$

$$\boxed{v_{Ty}\left(L\right)=\frac{FL}{k_yGS}=0.21 \quad mm}$$

Deflexion created by Ty and Mz

Simplified Theory (NASTRAN)

$$\boxed{v(l) = 10.92 + 0.24 = 11.26 \quad mm}$$

Bredt's Theory

$$\boxed{v(l) = 10.92 + 0.21 = 11.22 \quad mm}$$

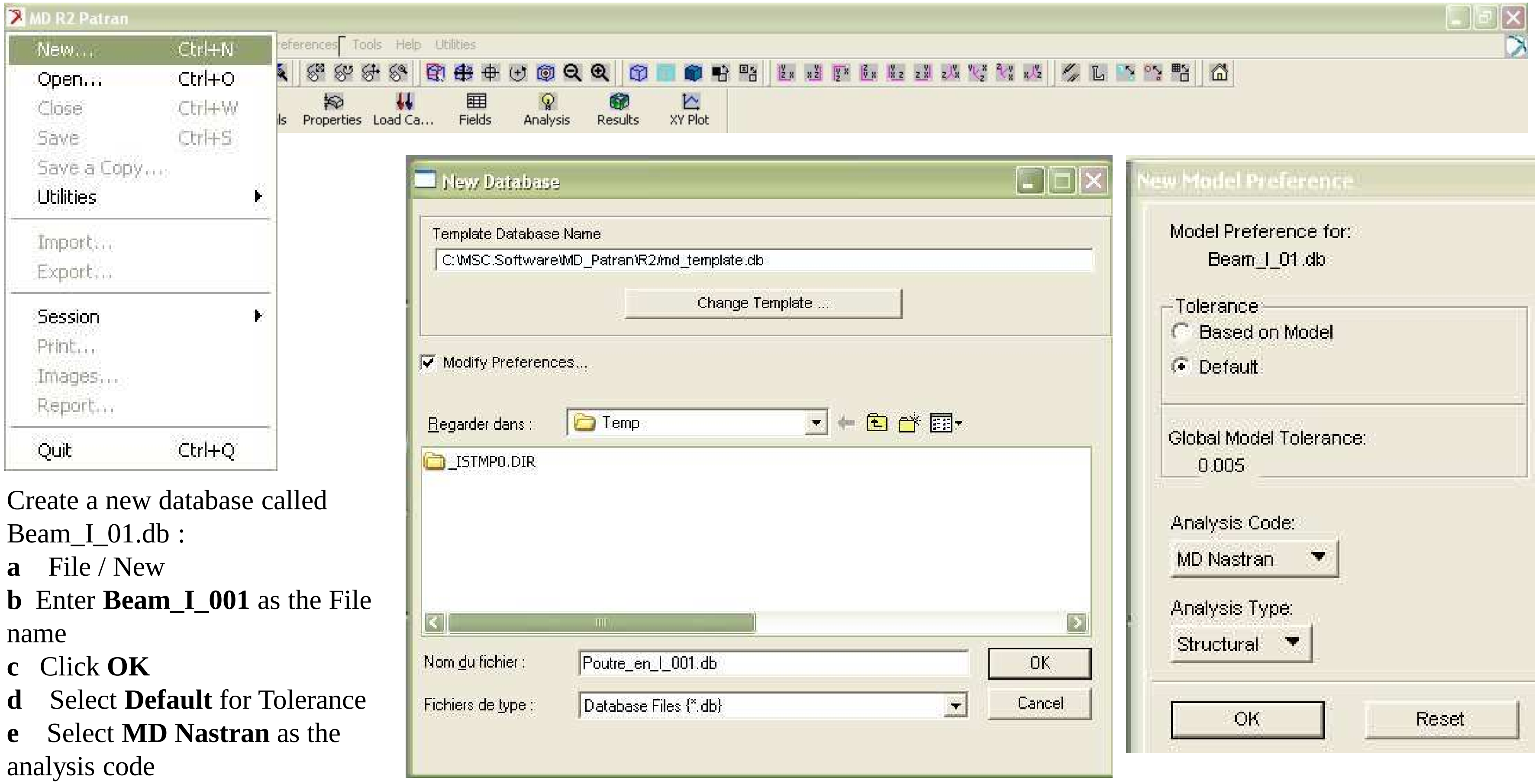
Choose of the units

The choose of the units is free, but these units must be coherent in order to have output results with coherent units.

Input Unit :
Lenght : mm
Force : N
Elastic modulus : MPa
Mass : Tonne
Density : Tonne / mm^3

Output Unit :
Displacement : mm
Force : N
Stresses : Mpa

Step 1 : Creation of a New Database



Create a new database called Beam_I_01.db :

a File / New

b Enter **Beam_I_001** as the File name

c Click **OK**

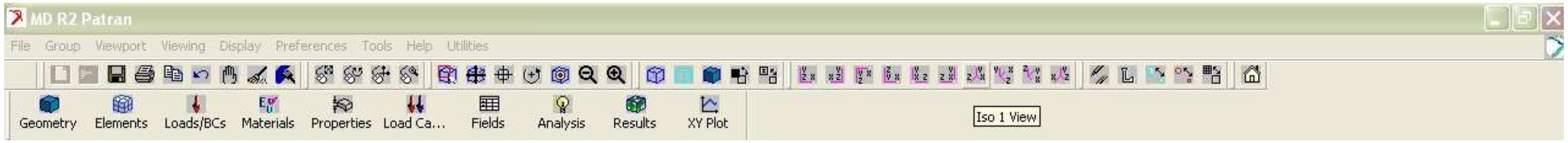
d Select **Default** for Tolerance

e Select **MD Nastran** as the analysis code

f Select **Structural** as the Analysis Type

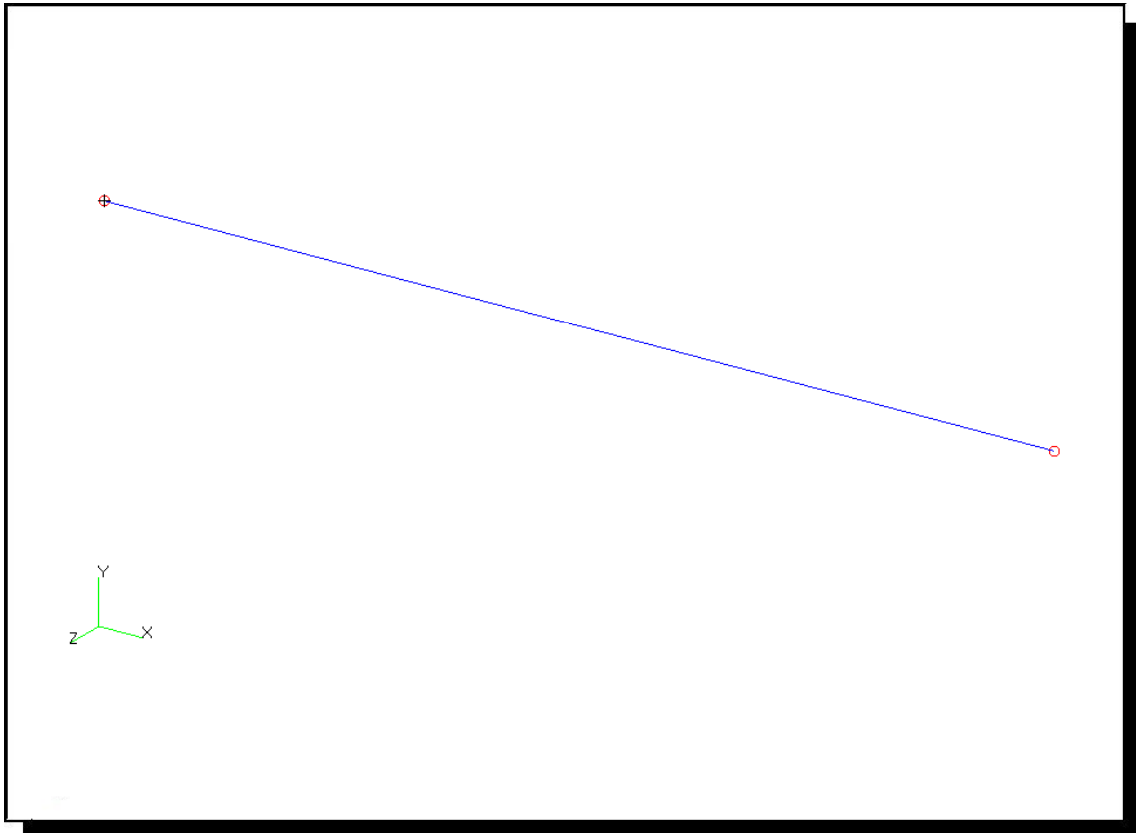
g Click **OK**

Viewing in 3D



To see the Space Work in 3 D

a Select the Icon **Iso 1 View**



Step 2 : Create Geometry

Create the first point :

a Geometry : Create : Point : XYZ

b Unselect **Auto Execute**

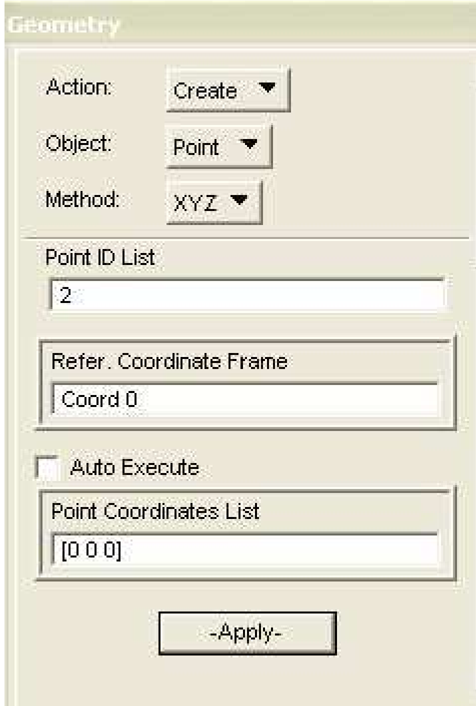
c Enter [0,0,0] for the Point Coordinate List.

d Click **Apply**

Create the second point :

e Enter [2000,0,0] for the Point Coordinate List.

f Click **Apply**



Viewing the Points

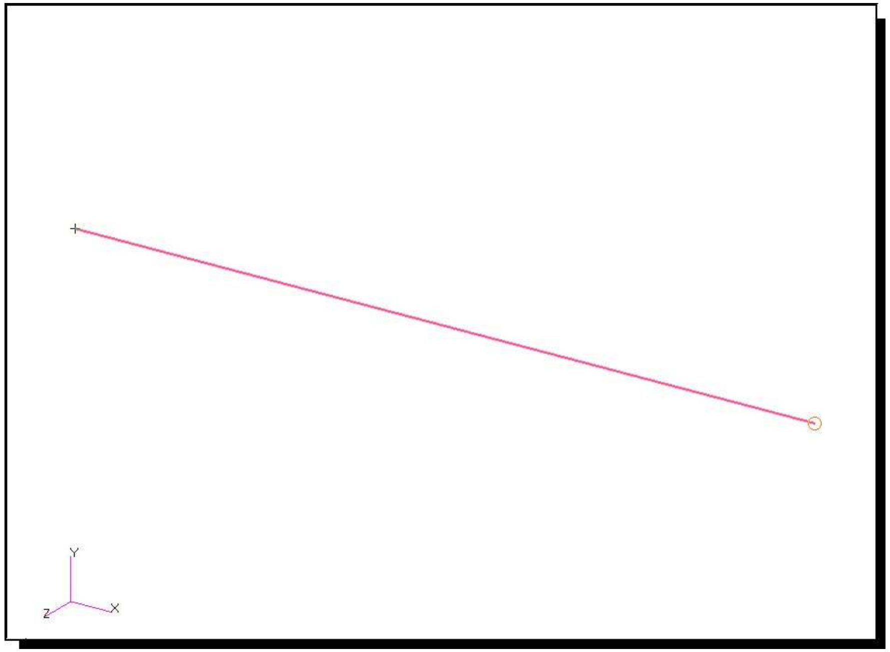
To increase the size of the points

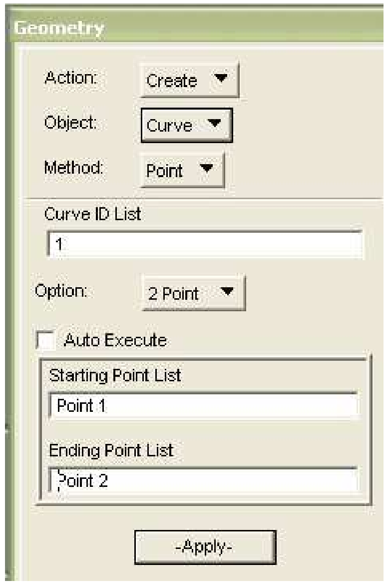
a Select the Icon **Point Size**

Step 2 : Create Geometry (Continue)

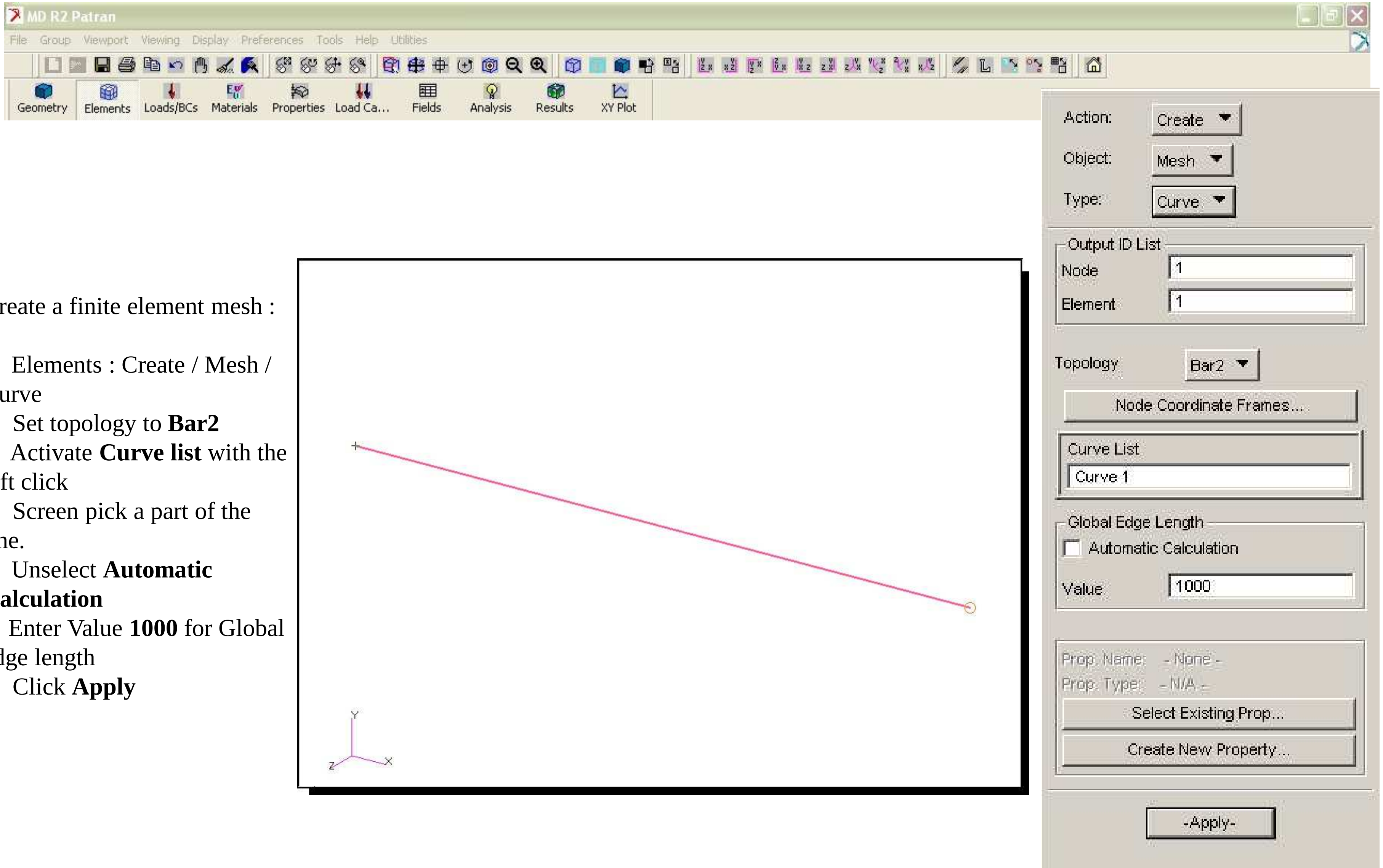
Create a curve to represent the beam :

a Geometry : Create / Curve / Point
b Unselect **Auto Execute**
c Activate **Starting Point list** with the left click
d Screen pick the First Point created.
e Activate **Ending Point list** with the left click
f Screen pick the Second Point created.
g Click **Apply**





Step 3 : Create Mesh



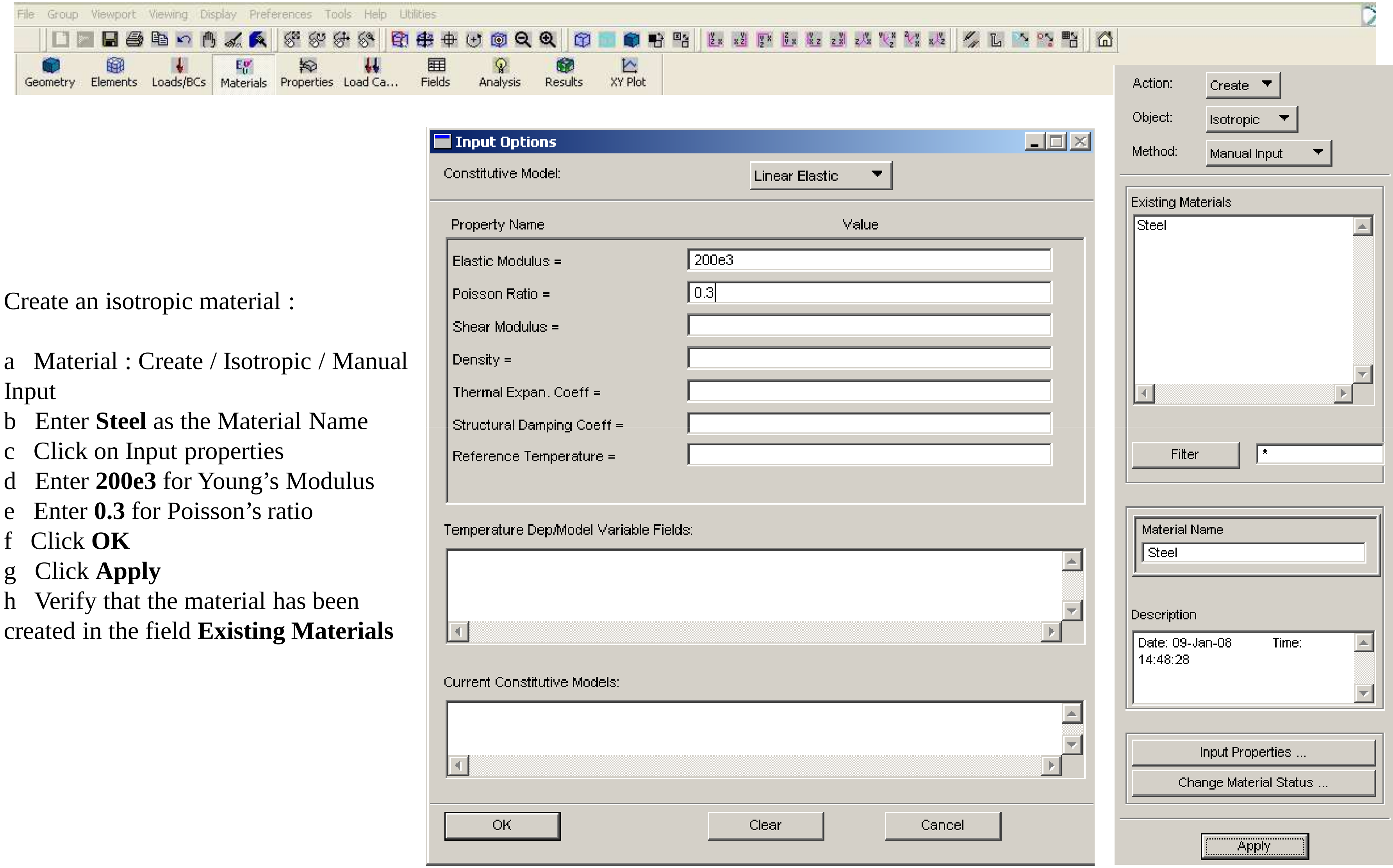
Create a finite element mesh :

a Elements : Create / Mesh / Curve
b Set topology to **Bar2**
c Activate **Curve list** with the left click
d Screen pick a part of the line.
e Unselect **Automatic Calculation**
f Enter Value **1000** for Global edge length
g Click **Apply**

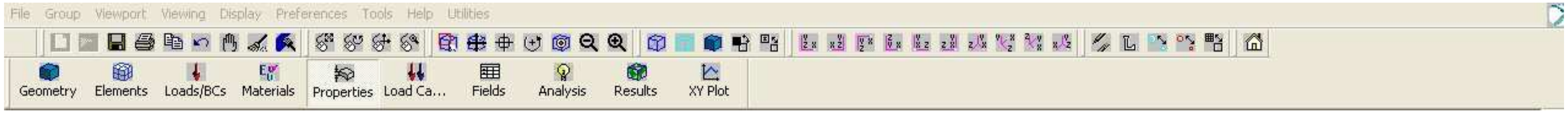
Step 4 : Create Material Properties

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 Young's 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**

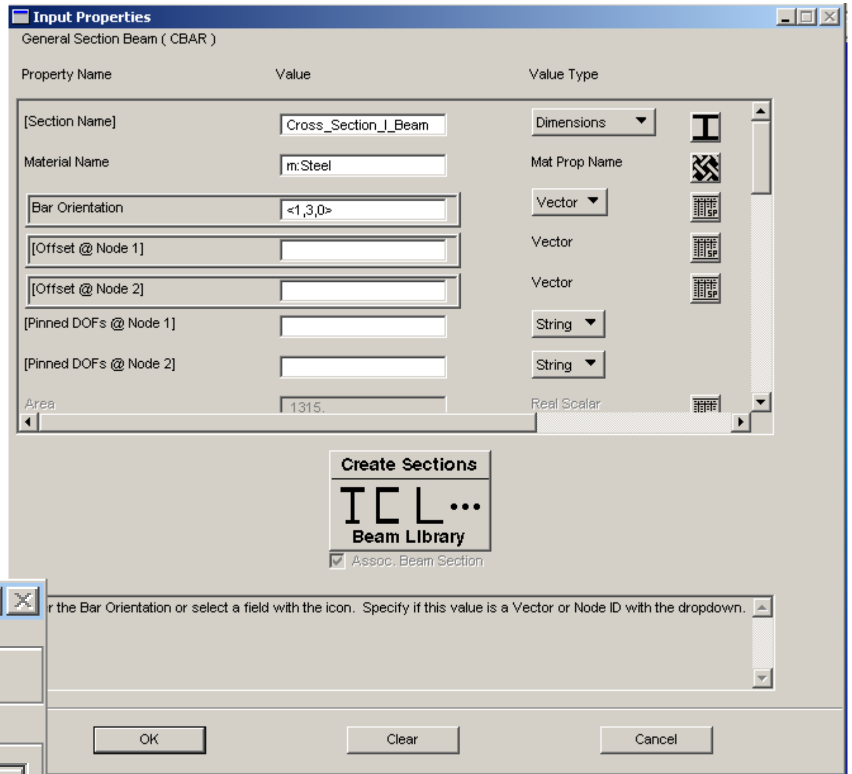


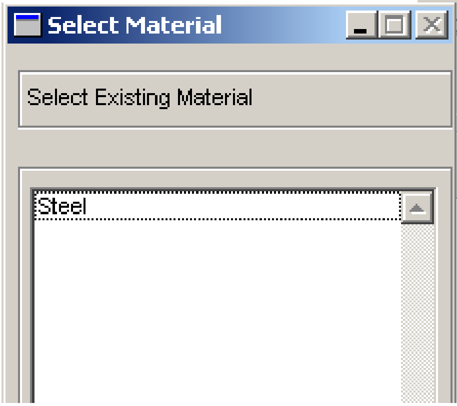
Step 5 : Create Physical properties

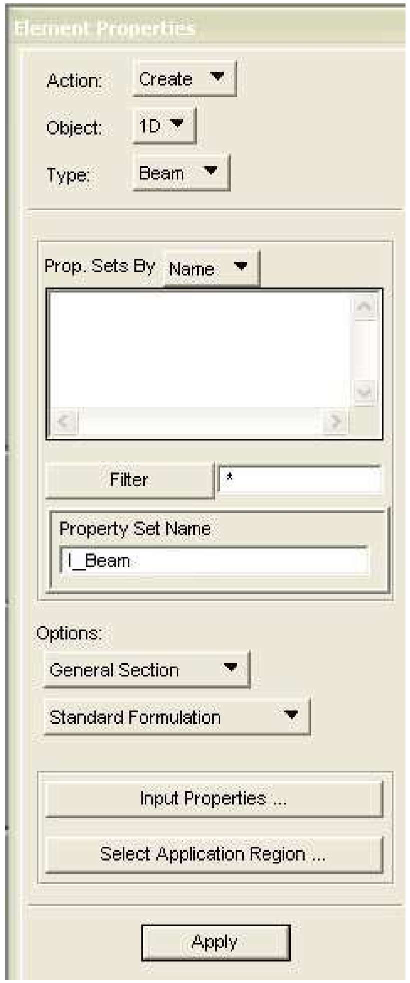


Create element properties :

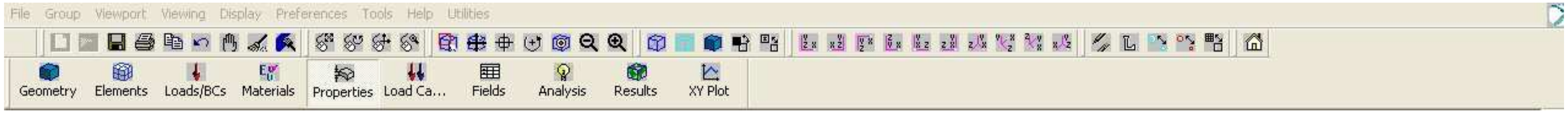
a Properties : Create / 1D / Beam
b Enter **Property_Beam** as the Property set Name.
c Click on **Input properties**
d Click on the **Select Material** icon.
e Select **Steel** as the materiel.
f Click on the **Beam Library** button





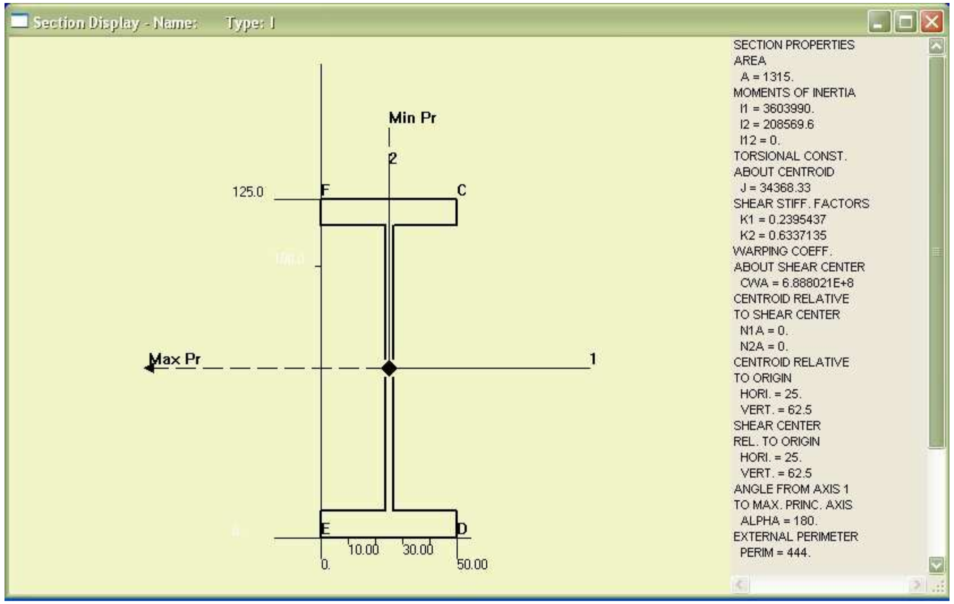


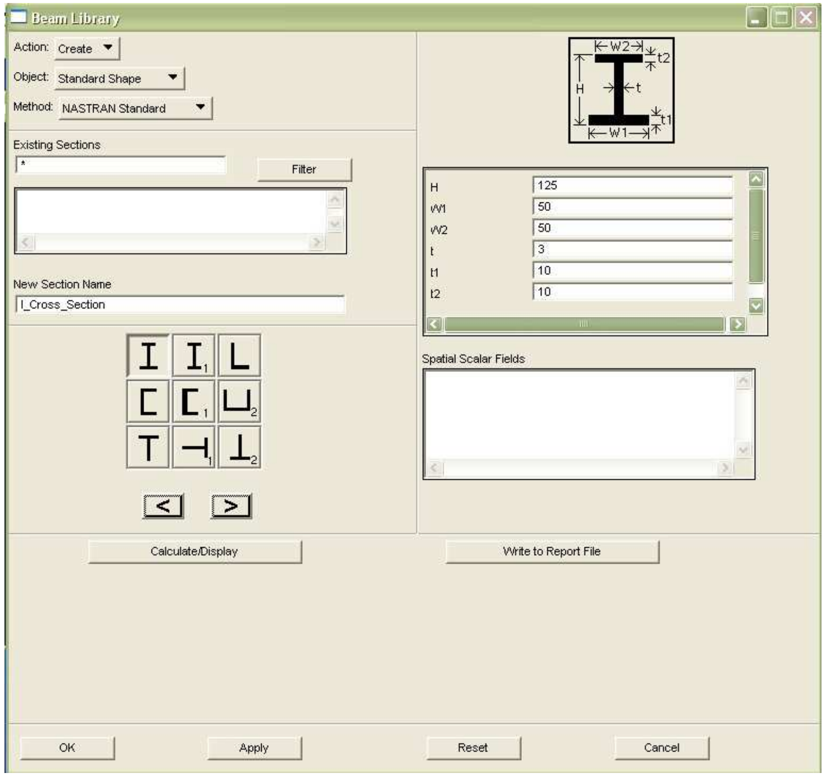
Step 4 : Create Physical Properties



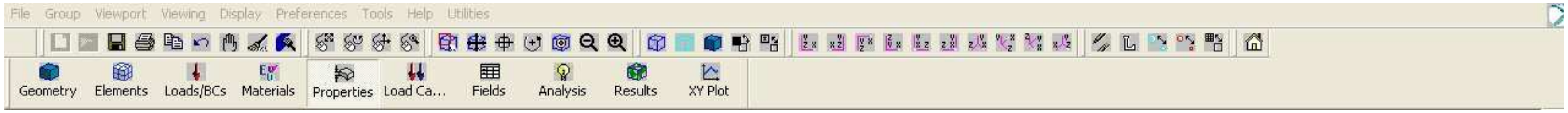
Define the beam section :

a Enter **I_Cross_Section** for the new section Name.
b Enter the appropriate values to define the beam's dimensions**.**
c Click on **Calculate/Display** to view the beam section and its section properties.
d Identify the position of the points C,D,E,F
e After verifying that the section is correct, Click **OK**





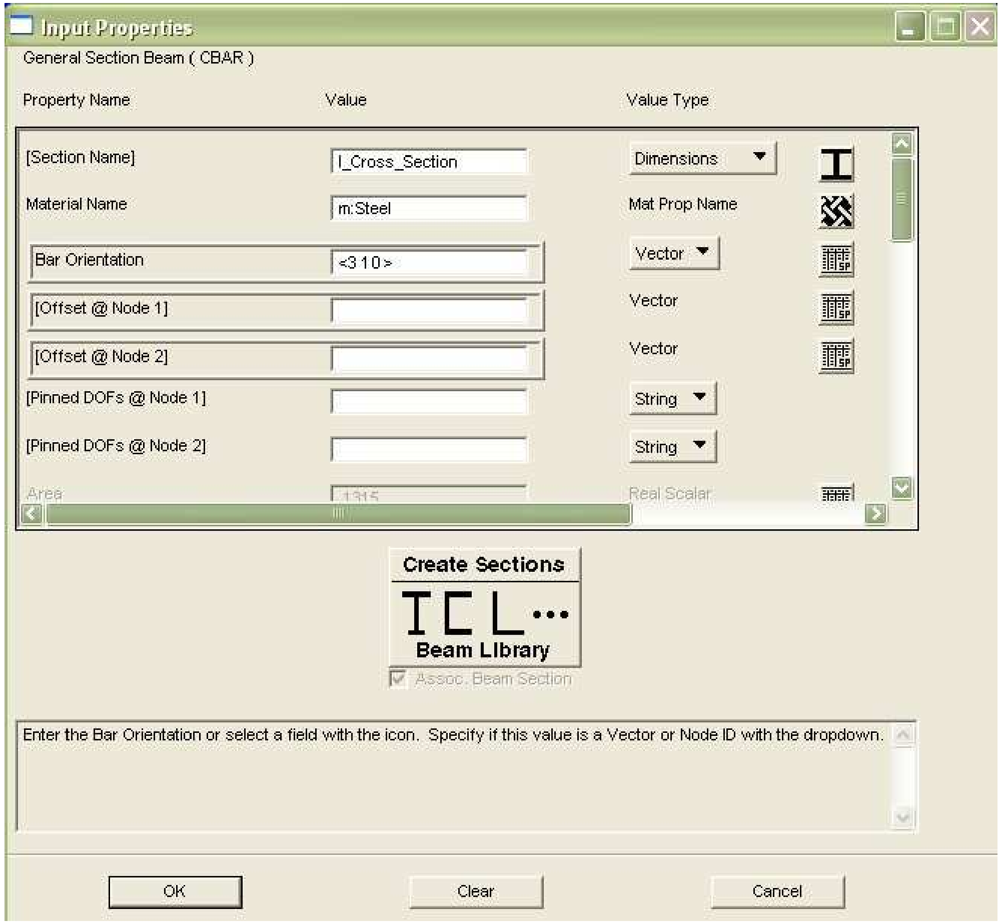
Step 4 : Create Physical Properties



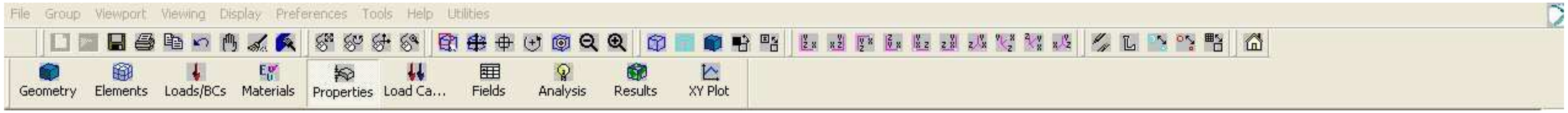
Define the bar Orientation :

a Enter **<3 1 0>** for the bar orientation.
b Click **OK**

REMARK : The cross section has been defined independently of the axis system of the beam. It is necessary to bring the principal axis of the cross section in the good position. For that you have to define an axis system for each beam. The first direction, defined by Patran, is the line of two consecutive nodes. The second direction is the projection in a normal plane of the line of the nodes of a vector you indicate by its components. The third direction is made directly by Patran by the vector product of these two directions.

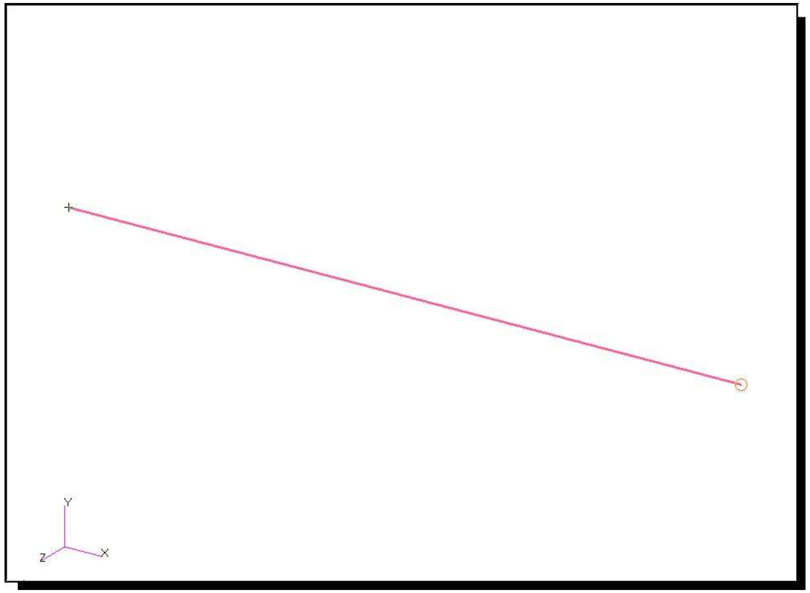


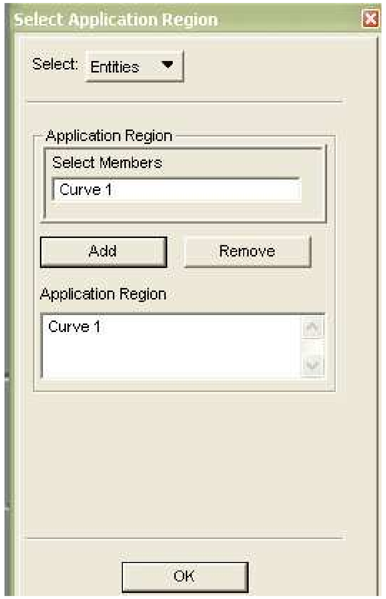
Step 4 : Create Physical Properties

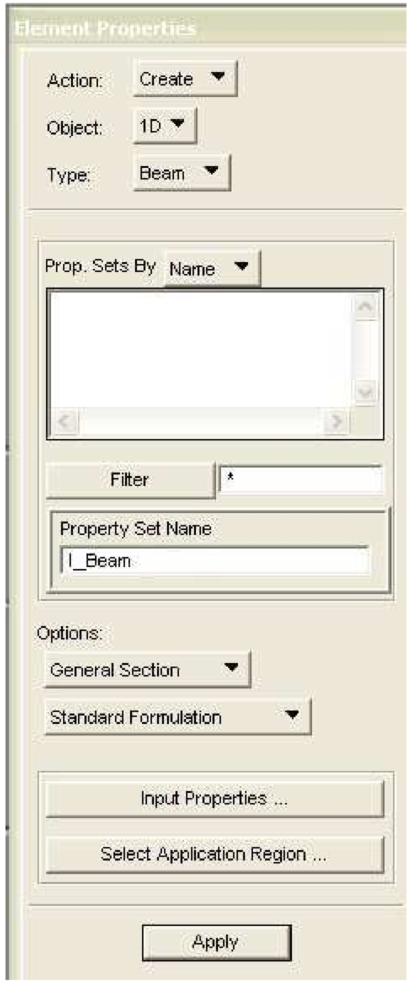


Select application region :

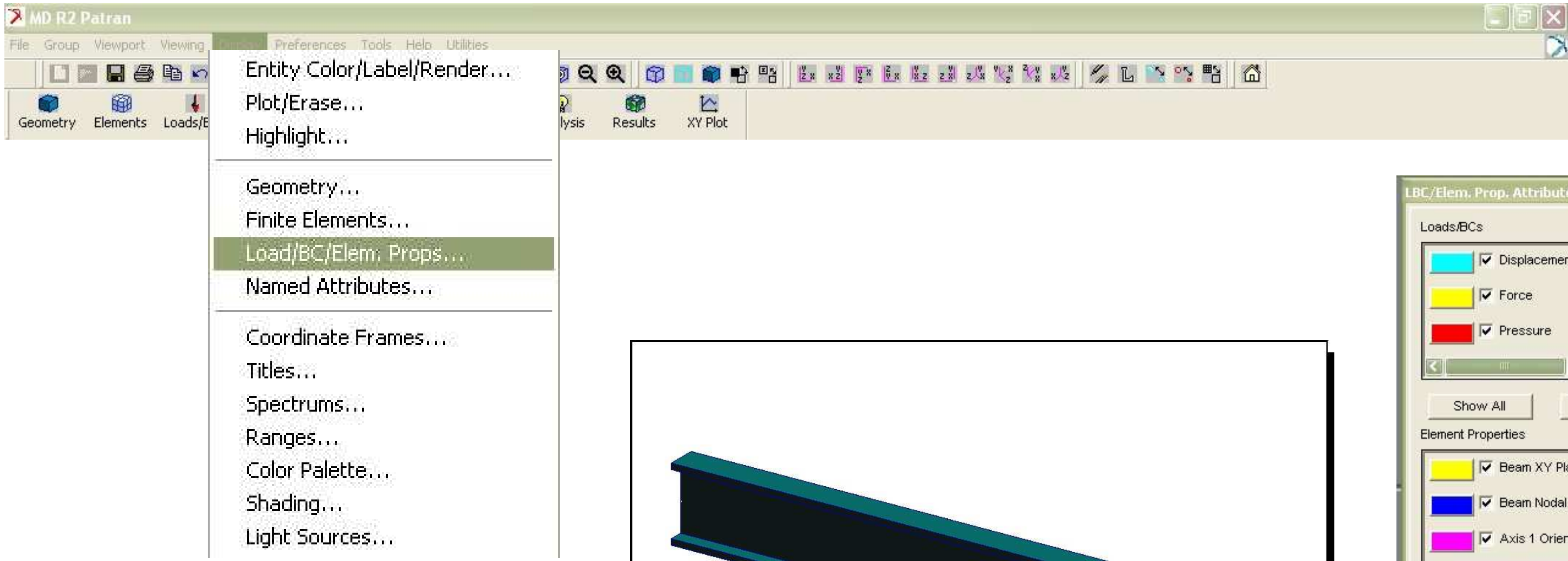
a Click on **Select Application Region**
b Click in the **Select Members** box.
c Select the curve on which you will put the beam element
d Click **Add**
e Click **OK**
f Click **Apply**

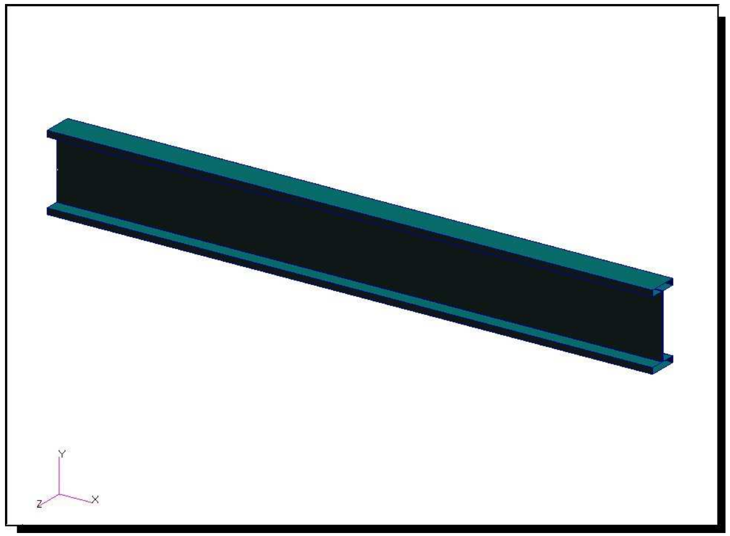


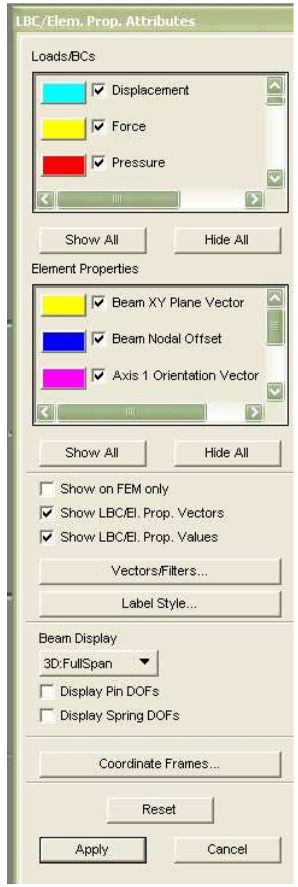




Step 4 : Create Physical Properties



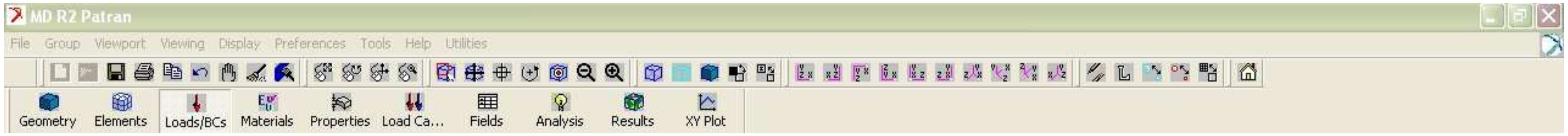




Verify the beam section :

a Click on **Display / Loads/BC/ Elem.Props….**
b Set Beam Display to 3D : Full-Span.
c Click **Apply**.
d Shade the model.
d Rotate the model and zoom to verify that the beam is correctly oriented.
f Return to the front view.
g Set Beam Display back to **1D:Line.**
h Click **Apply.**

Step 5 : Create Boundary Conditions



Create a boundary condition :

a Click on **Loads/BCs : Create / Dispalcement / Nodal**

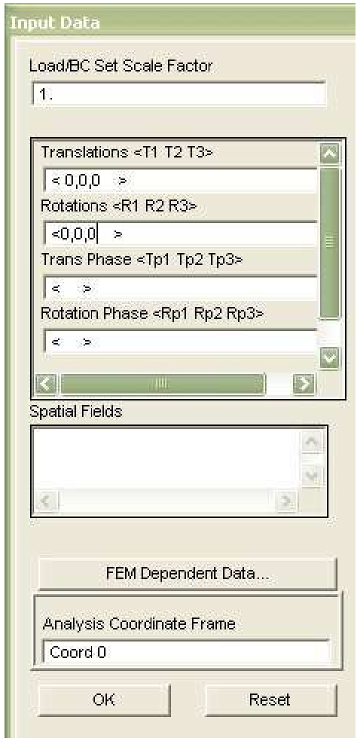
b Enter **Bound_Cond_Left_Side** as the New set Name.

c Click on **INPUT Data**.

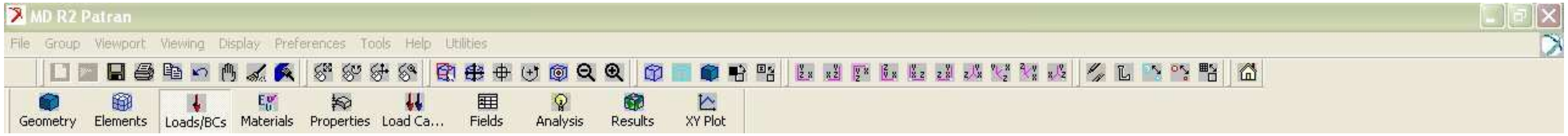
d Enter **<0 0 0>** for Translations.

d Enter **<0 0 0>** for Rotations.

h Click **OK.**

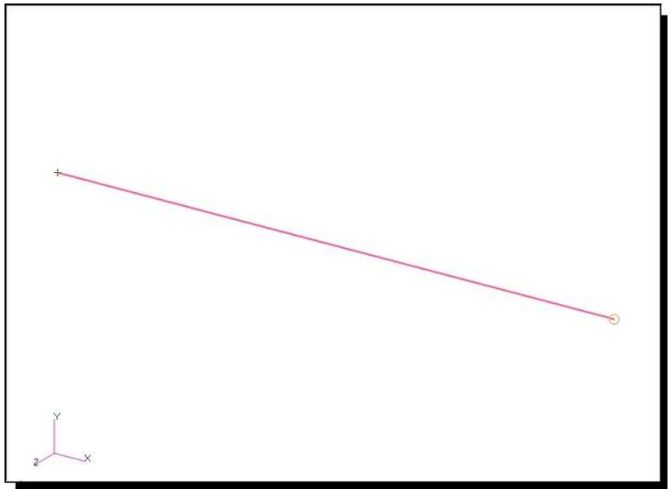


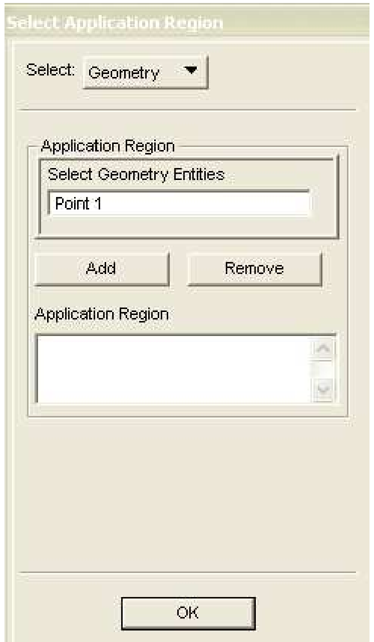
Step 6 : Create Boundaries Conditions (Continue)

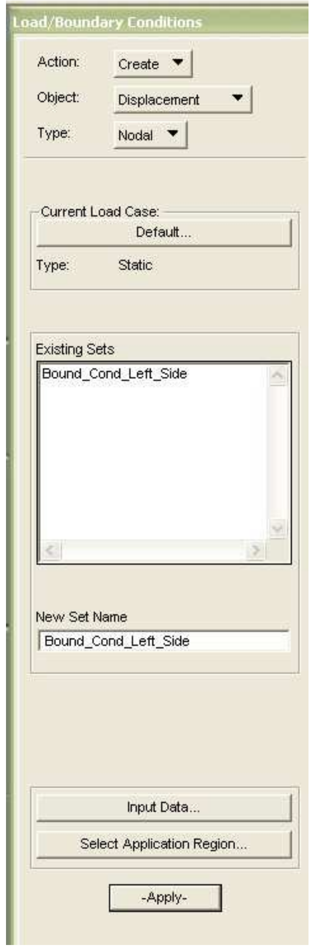


Apply the boundaries conditions :

a Reset graphics.
b Click on **Select Application Region.**
c Select the Point on the left side of the beam.
d Click **Add**
e Click **OK**
f Click **Apply**



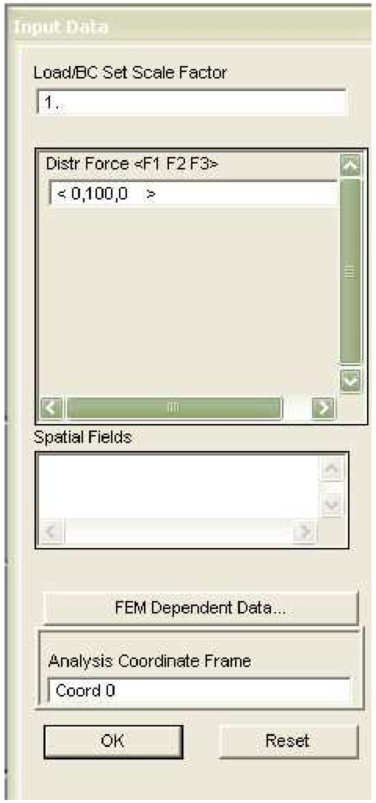


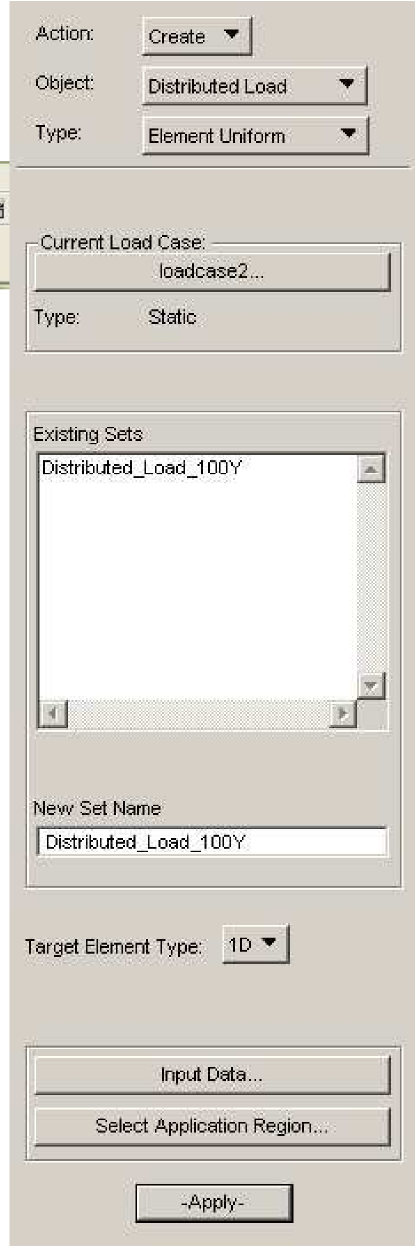


Step 6 : Create Load

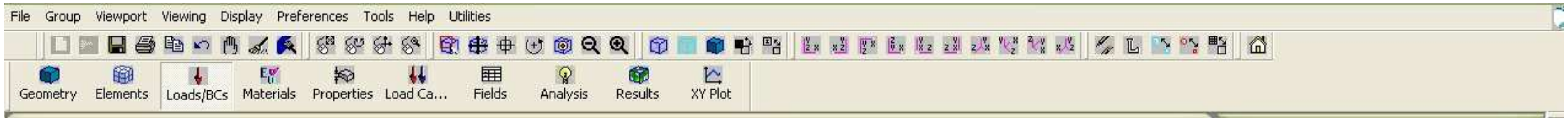
Create a constant uniform load :

a Loads/BCs : Create / Distributed Load / Element Uniform
b Click on **New Set Name**.
c Enter **Distributed_Load_100Y** as name for the load
d Select 1D with icon **Target Element Type**
e Click on **INPUT Data.**
f Enter **<0 100 0>** for components of the distributed load.
g Click **OK**
h Click **Apply**
i Verify that the load has been created in the field **Existing Sets**



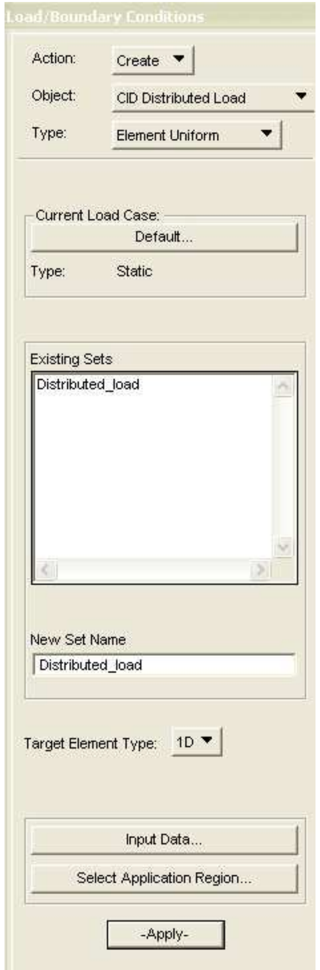


Step 6 : Create Load



Select application region :

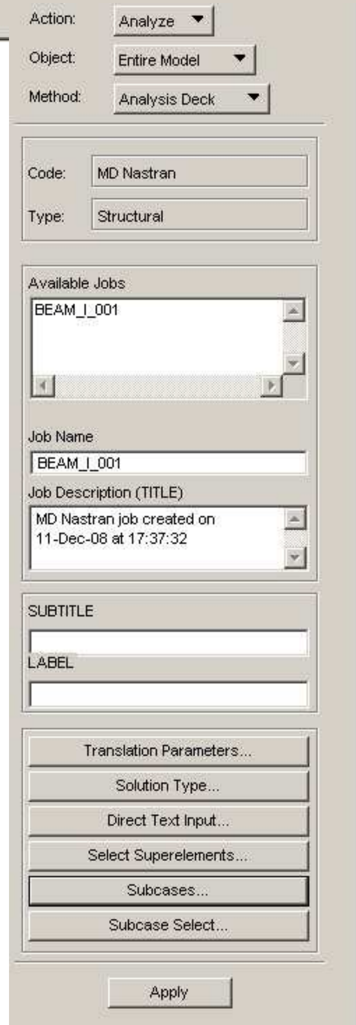
a Click on **Select Application Region**
b Click in the Select Members box.
c Select the curve on which the beam elements have been created
d Click **Add**
e Click **OK**
f Click **Apply**



Create the input file for Nastran *.BDF

Create the BDF input file for Nastran :

a Analysis : Analyse / Entire model / Analysis desk
b Click **Apply**



Read the *.BDF File

In your working directory file open the BDF input file for Nastran with an editor :

Read it and Find :

The nodes (GRID Card)

The elements (CBAR card)

The properties of the elements (PBARL Card)

The boundary conditions (SPC, SPSADD and SPC1 Card)

The load (LOAD, FORCE and PLOAD Cards)

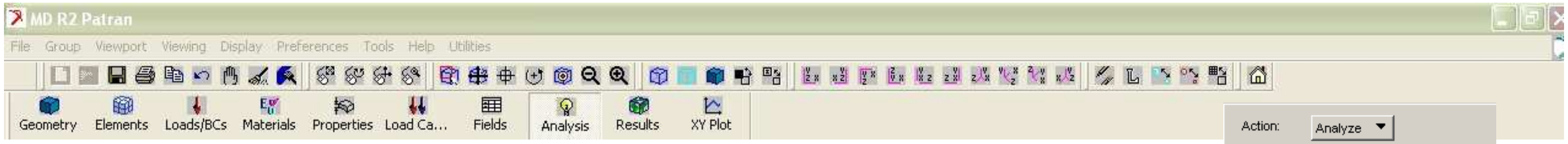
The following page gives you the BDF file of this exercise.

The *.BDF File

-Lines beginning by $ are just comments
-SOL 101 indicate the linear Solver
-SUBCASE 1 begin the description of the first Sub Case

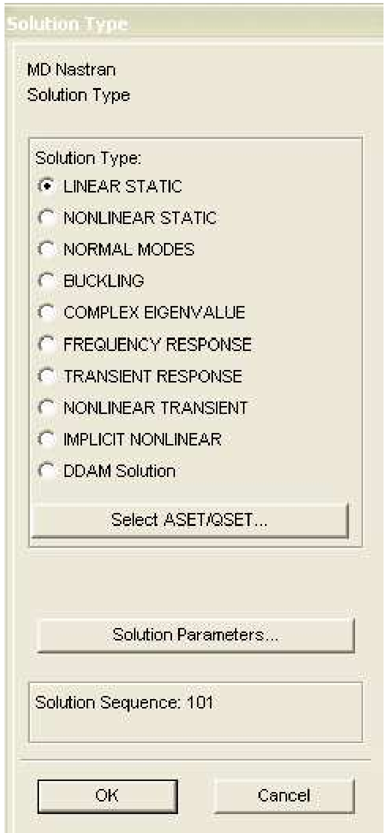
```
$ NASTRAN input file created by the MSC MSC.Nastran input file
$ translator ( MSC.Patran 13.0.053 ) on December 20, 2009 at 12:22:33.
$ Direct Text Input for Nastran System Cell Section
$ Direct Text Input for File Management Section
$ Linear Static Analysis, Database
SOL 101
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC.Nastran job created on 20-Dec-09 at 12:22:19
ECHO = NONE
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM   POST   0
PARAM   AUTOSPC YES
PARAM   PRTMAXIM YES
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : Property_Beam
PBARL   1     1          I
      125.   50.    50.    3.     10.    10.
$ Pset: "Property_Beam" will be imported as: "pbarl.1"
CBAR    1     1     1     2     1.     3.     0.
$ Referenced Material Records
$ Material Record : Steel
$ Description of Material : Date: 20-Dec-09         Time: 12:16:00
MAT1    1     200000.      .3
$ Nodes of the Entire Model
GRID    1           0.     0.     0.
GRID    2           1000.  0.     0.
$ Loads for Load Case : Default
SPCADD  2     1
LOAD    2     1.    1.     1
$ Displacement Constraints of Load Set : Left_Side_Clamped
SPC1    1     123456  1
$ Distributed Loads of Load Set : Distributed_Load_100Y
PLOAD1  1     1     FYE    FR     0.    100.   1.    100.
$ Referenced Coordinate Frames
ENDDATA 740f66ed
```

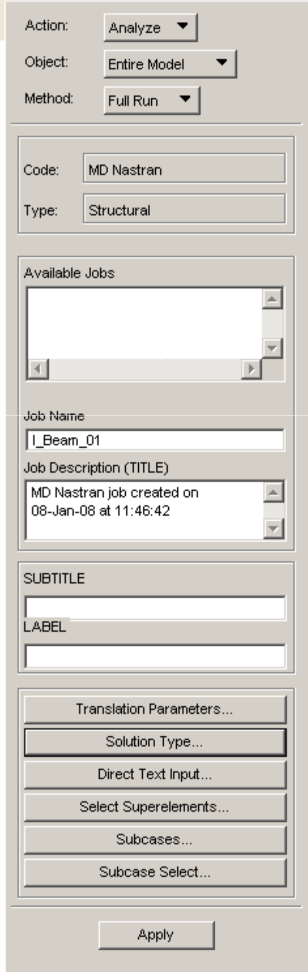
Run Linear Static Analysis



Choose the analysis type :

a Analysis : Analyse / Entire Model / Full Run
b Click on **Solution Type**.
c Select **Linear Static**.
d Click **OK**
e Click **Apply**





Read the *.FO6 File

In your working directory file open the F06 output file of Nastran with an editor :

Read it and Find :

The load applied
The reactions
The value of Epsilon
The displacements of the node. (Compare the good one with the theoretical value of 19.40 mm)
The stresses in the beam elements

The following page gives you and extract of the F06 file of this exercise

Reading the file   *.F06

Open the file Beam_I_001.F06 with Bloc Notes or another file editor

Verification of the load applied

0 **OLOAD** RESULTANT

SUBCASE/ DAREA ID		LOAD	TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	0.0E+00	----	----	----	0.0E+00	0.0E+00	
		FY	----	1.0E+05	----	0.0E+00	----	5.0E+07	
		FZ	----	----	0.0E+00	0.0E+00	0.0E+00	----	
		MX	----	----	----	0.0E+00	----	----	
		MY	----	----	----	----	0.0E+00	----	
		MZ	----	----	----	----	----	0.0E+00	
		TOTALS	0.0E+00	1.0E+05	0.0E+00	0.0E+00	0.0E+00	5.0E+07	

Reading the file *.F06

Verification of the computation

LOAD SEQ. NO.	**EPSILON**	EXTERNAL WORK	EPSILONS LARGER THAN 0.001 ARE FLAGGED WITH ASTERISKS
1	-1.4183939E-15	4.0669391E+05	

Remark : EPSILON must be smaller than 10-6

Reading the file *.F06

Verification of the reactions (SPC Force)

0 **SPCFORCE** RESULTANT

SUBCASE/ DAREA ID		LOAD	TYPE	T1	T2	T3	R1	R2	R3
0	1	FX	0.0E+00	----	----	----	0.0E+00	0.0E+00	
		FY	----	-1.0E+05	----	0.0E+00	----	-5.0E+07	
		FZ	----	----	0.0E+00	0.0E+00	0.0E+00	----	
		MX	----	----	----	0.0E+00	----	----	
		MY	----	----	----	----	0.0E+00	----	
		MZ	----	----	----	----	----	0.0E+00	
		TOTALS	0.0E+00	-1.0E+05	0.0E+00	0.0E+00	0.0E+00	-5.0E+07	

Reading the file *.F06

You have to process to different verifications

MAXIMUM SPCFORCES : Réactions max sur les ddl bloqués

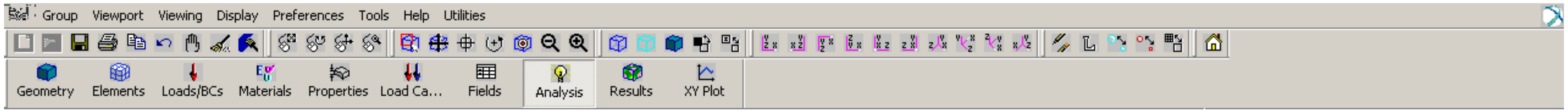
MAXIMUM DISPLACEMENTS :

MAXIMUM APPLIED LOADS :

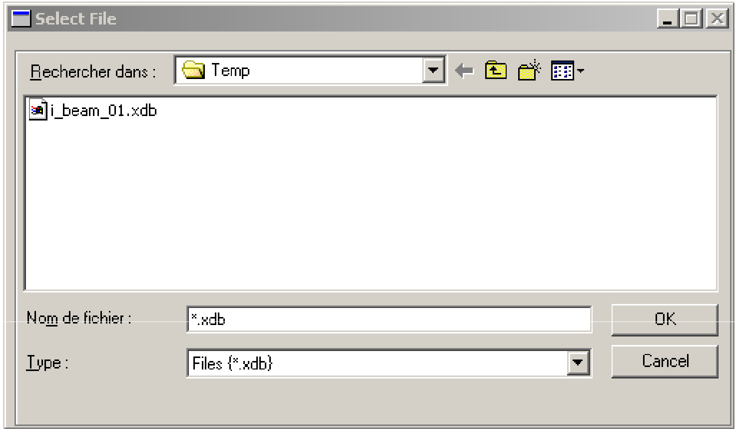
LOAD VECTOR :

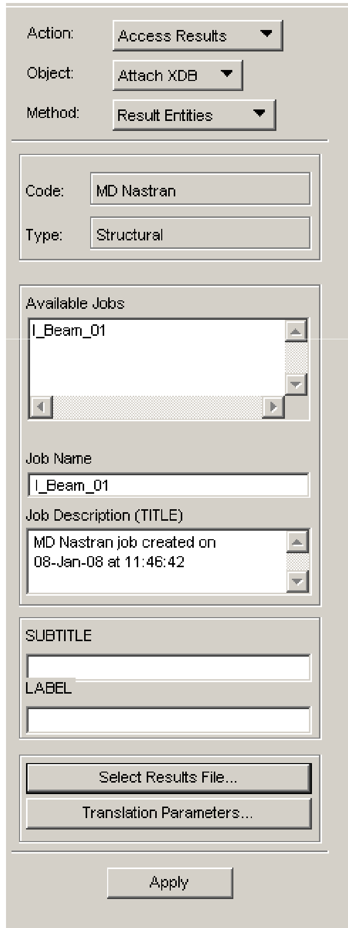
FORCES OF SINGLE POINT CONSTRAINT :

Reading Output Nastran With Patran



Remark : A binary output file (*.XDB or *.OP2) is created by Nastran in which Patran will find the same results than in the F06 file





Choose the analysis type :

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

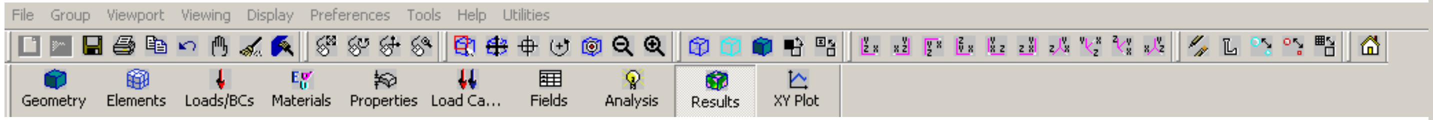
b Select **Select Results File**

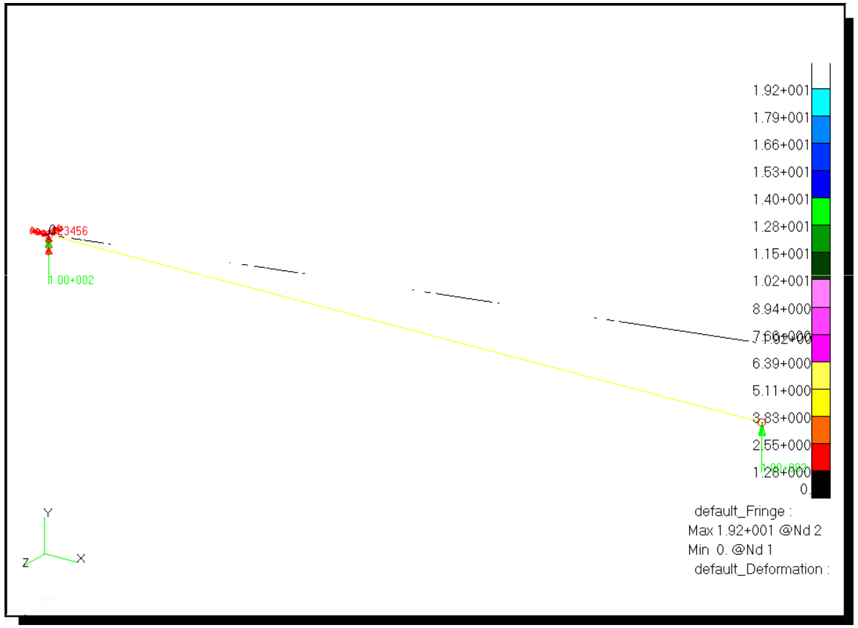
c Select the **job.xbd** File

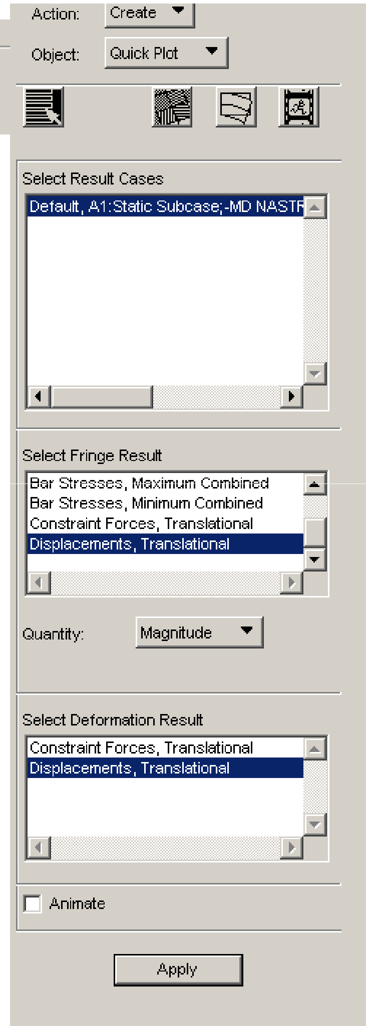
d Click **OK**

E Click **Apply**

Analysis of the Results







Analysis of the results :

a Select Results : Create / Quick Plot
b Select Icon **Select Results**
c Select **Displacements, Translational** in the field **Select Fringe Result**
d Select **Displacements, Translational** in the field **Select Deformation Result**
e Click **Apply**

Management of the Load Cases

Patran can manage several load cases in which you can define for each of us :

- Specific Load ,
- Specific Boundary Conditions,
- Specific options for the output results file

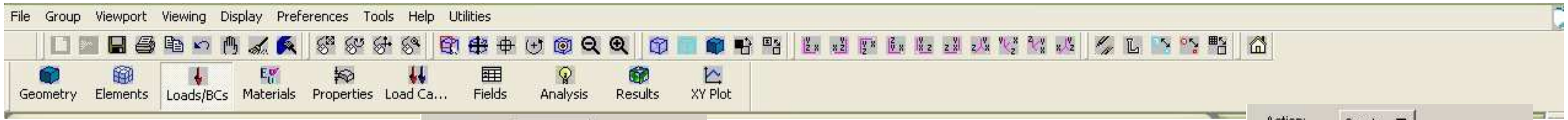
To learn that we shall define :

- a few more loads,
- differents Load Cases,

Then we will manage the Load Cases and see the results.

REMARK : We will create 5 load cases in the following pages

Create two new distributed Loads

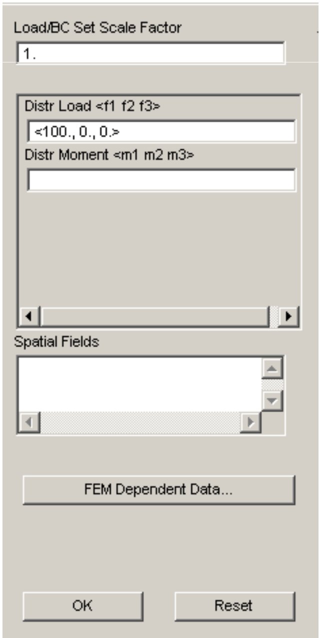


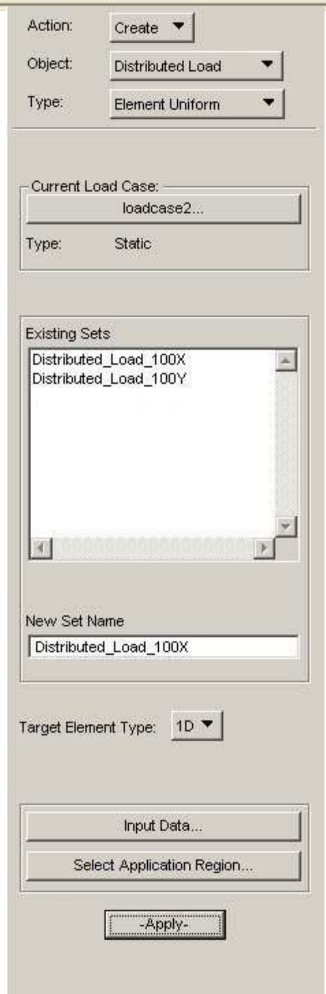
Create a second load whose name is **Distributed_Load_100X**

with the same process :

The load will be

<100 , 0 , 0 >



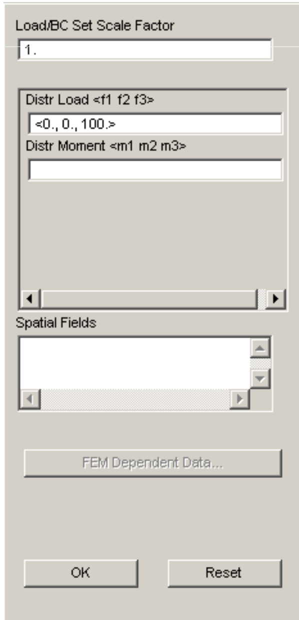


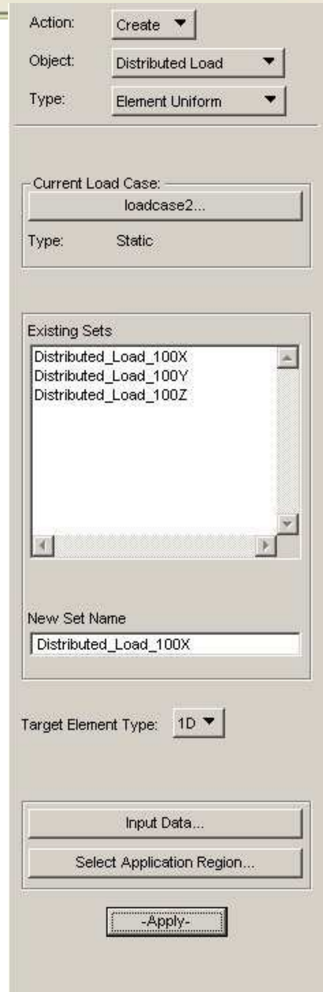
Create a third load whose name is **Distributed_Load_100Z**

with the same process :

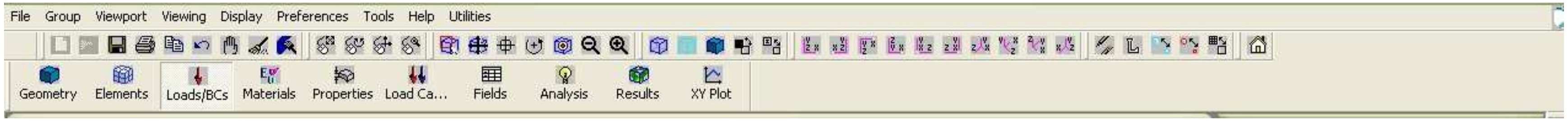
The load will be

<0 , 0 ,100 >



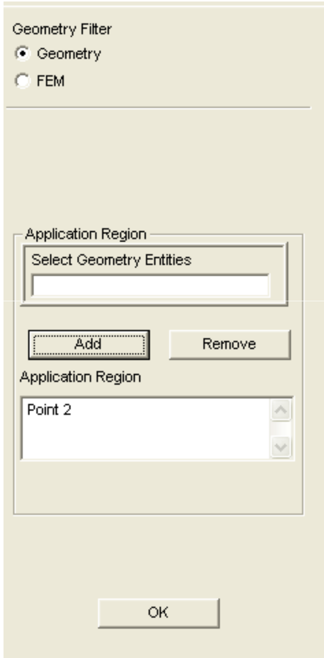


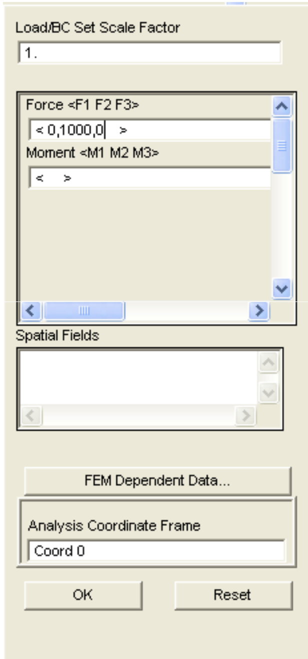
Create a Ponctual Load

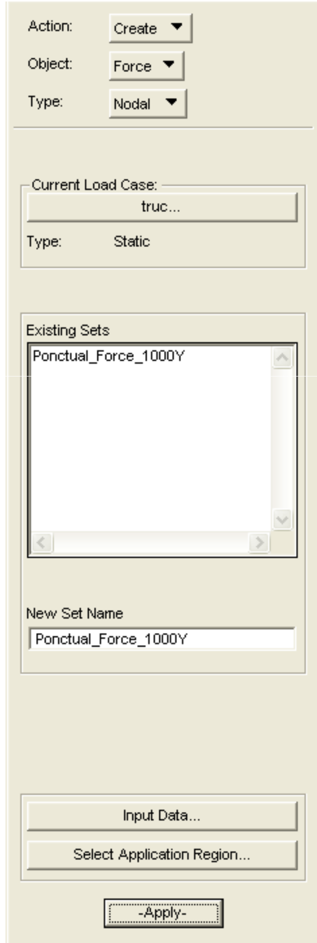


Create a punctual load whose name will be Ponctual_Force_1000Y

a   Loads/BCs : Create / Force/ Nodal
b   Click on **New Set Name**.
c   Enter **Ponctual_Force_1000Y** as name for the load
d   Click on **INPUT Data**.
e   Enter **<0, 1000, 0>** for components of the punctual load.
f   Click **OK**
g Select Application Region
h Select Geometry
i In **Application region** select the Point 2 on the screen
j Click on **Add**
k Click **OK**
l Click **Apply**
m Verify that the load has been created in the field **Existing Sets**







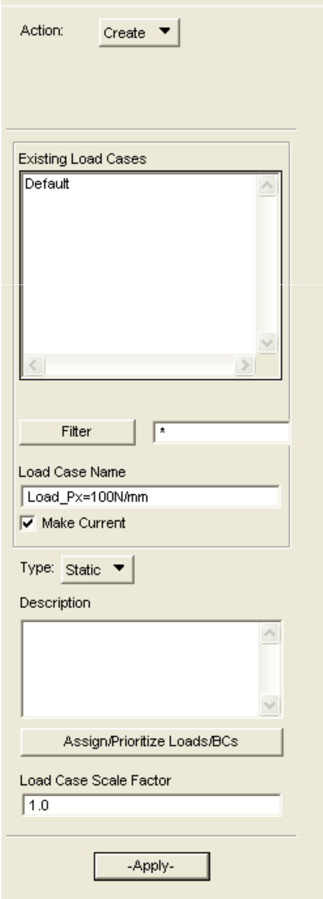
Creation of differents Load Cases

Create different Load Cases, identified by a name you can choose as you want :

a Load Ca....: Create

b In the field **Load Case Name** enter a name for the load case, for exemple **Load_Px=100N/mm**

c Click on **Assign/Prioritize Loads/BCs**



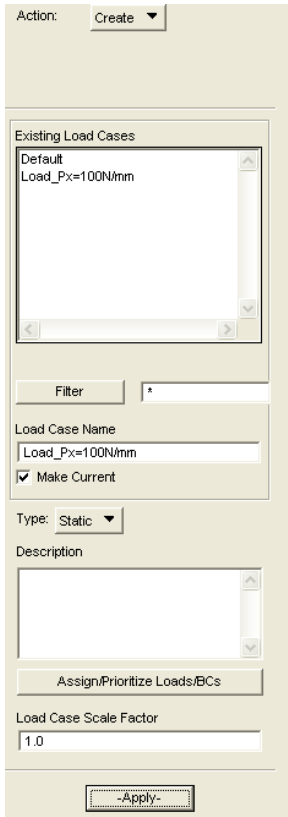
Creation of differents Load Cases

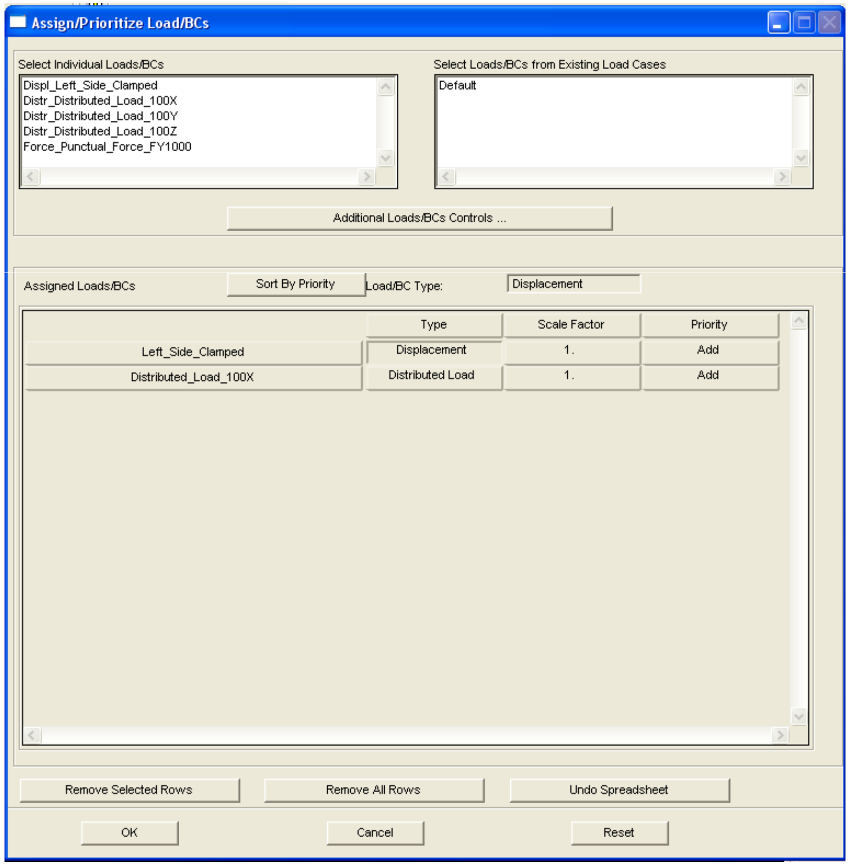
a Select in the windows **Select individual Loads/BCs** the good BCs and the good loads for the Sub Case :
- Disp_Bound_Cond_left_Side
- Dist_Distributed_Load_100X

b Click **OK**

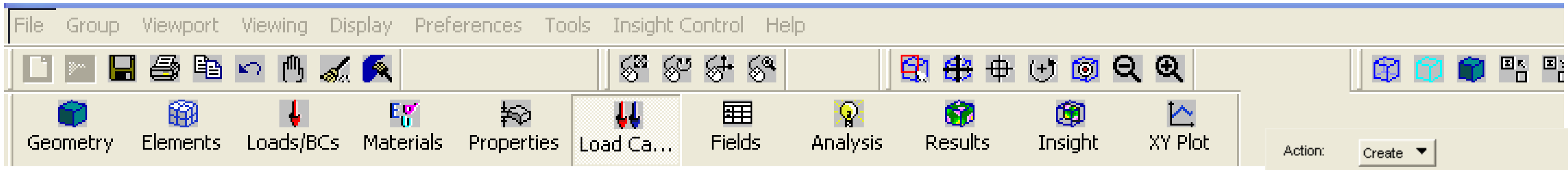
c Click **Apply**

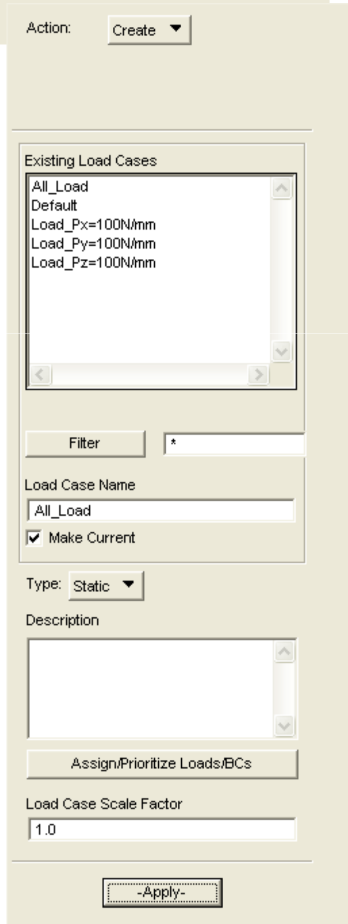
d Verify that the load case have been created in the field **Existing Load cases**





Creation of differents Load Cases





Create 4 others Load Cases with the following names, and put inside each of them the appropriate Loads and Boundary Conditions :

LOAD CASES	BOUNDARY CONDITIONS	LOADS
Load_Py=100N/mm	Bound_Cond_Left_Side	Distributed_Load_100Y
Load_Pz=100N/mm	Bound_Cond_Left_Side	Distributed_Load_100Z
Load_Fy=1000N	Bound_Cond_Left_Side	Punctual_Load_FY1000
All_Load	Bound_Cond_Left_Side	Distributed_Load_100X Distributed_Load_100Y Distributed_Load_100Z Punctual_Force_FY1000

a Verify that the load Cases have been created

Verification of the Load Cases

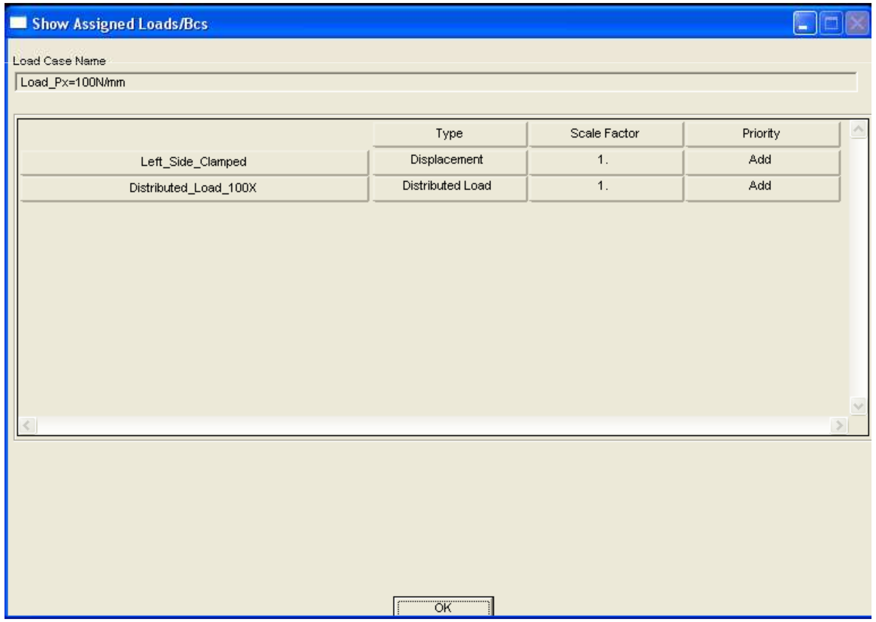
You can verify that the load cases are pretty well for you :

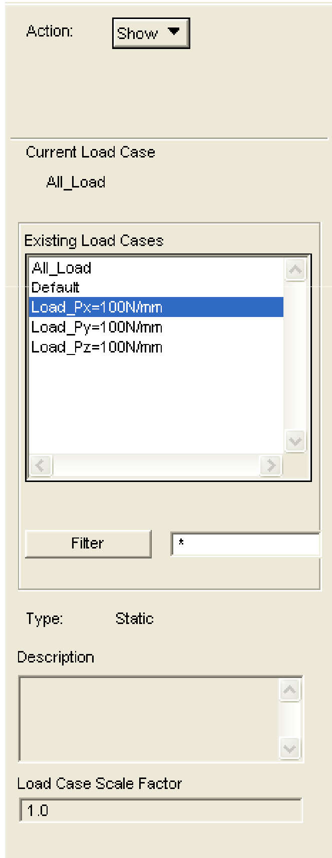
a **Load Ca....: Show**

b Select the Load Case, for exemple **Load_Px=100N/mm**

c Verify the Load and the BCs

d Click **OK**





Modification of the Load Cases

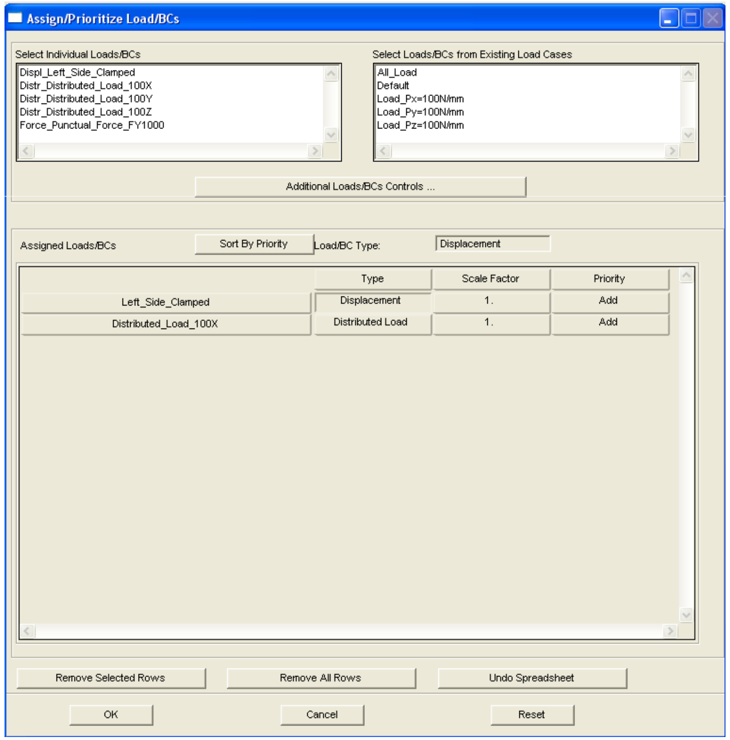
You can remove or add Loads OR (AND) BCs
You also can import Loads OR (AND) BCs from other Load Cases:

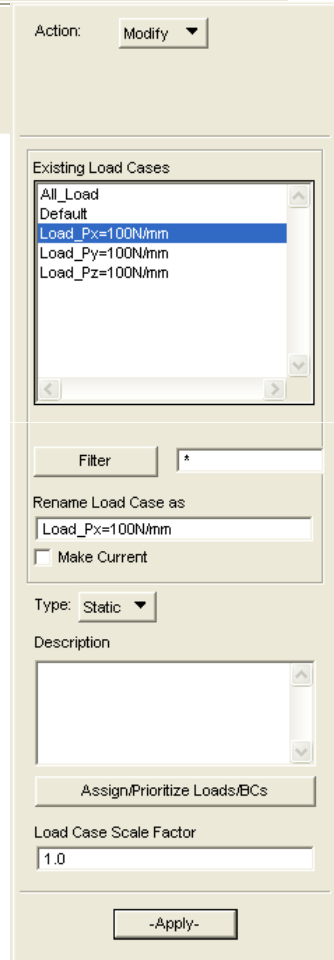
a Load Ca....: Modify

b Select the Load Case, for exemple **Load_Px=100N/mm**

c Verify the Load and the BCs

d Click **OK**





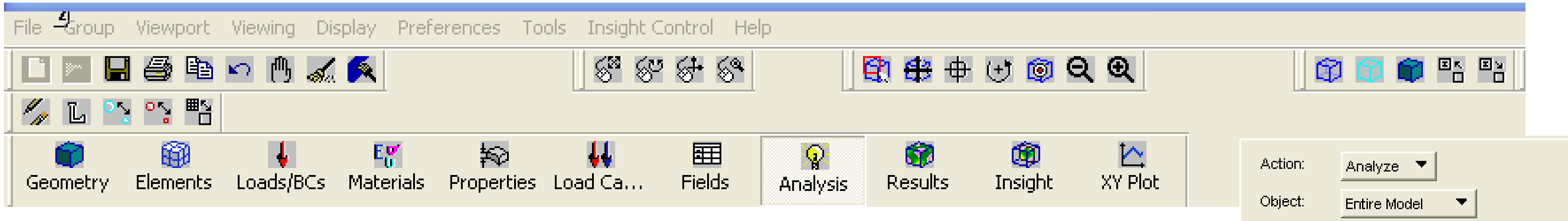
To add Loads or BCs, select them in the field
Select Individual Loads/BCs

To import Load and BCs of other Load Case, select them in the field Select **Loads/BCs from Existing Load Cases**

To save the modifications don’t forget :

e Click OK
f Click Apply

Add output to Each Subcases



You can add specific request for the output file of Nastran, and that for each SubCase

a Analysis/Entire Model/Full Run

b Select **Subcases**

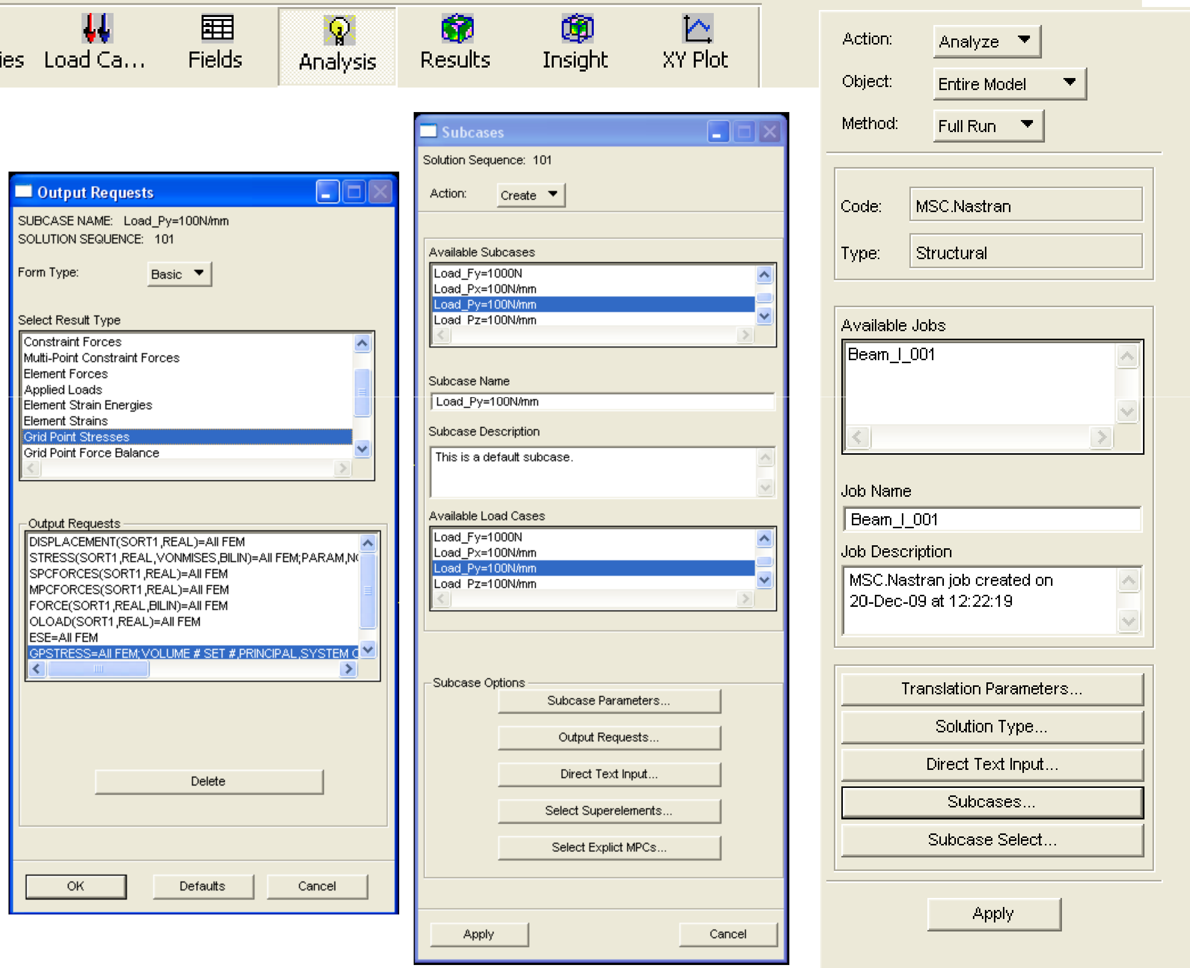
c Select the **Subcase** Load_Py=100N/mm

d Select **Output Request**

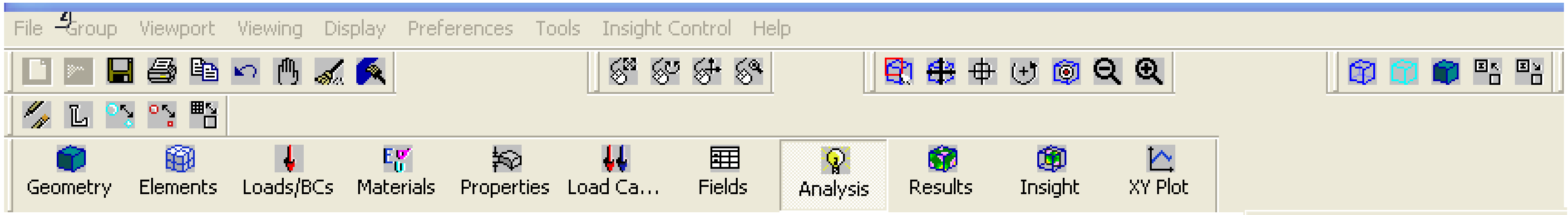
e Add the Output you want by selecting in the windows **Select Result Type**

f Click **OK**

e Click **Apply**



Select the Subcases to execute



You must select the SubCase you want to be computed by Nastran

a Analysis/Entire Model/Full Run

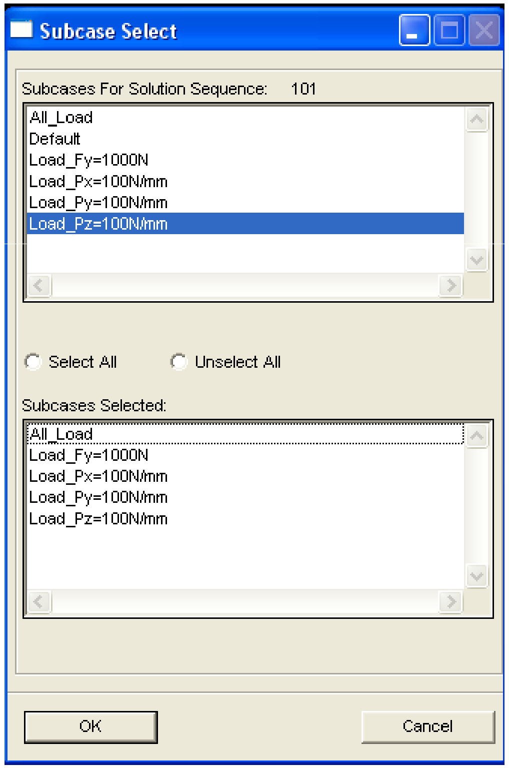
b Select **Subcases Select**

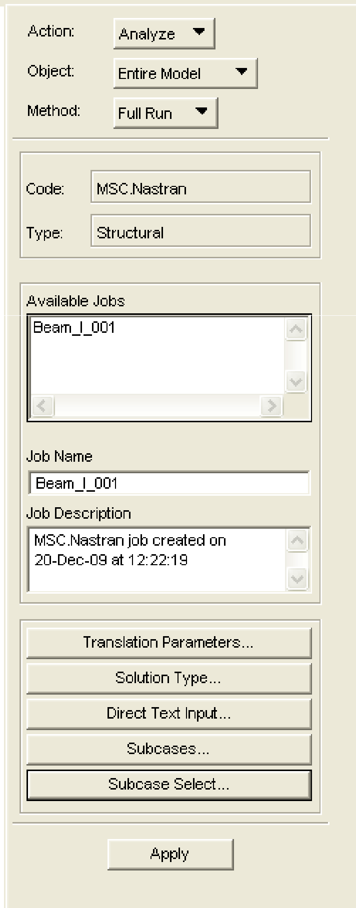
c Select the different Subcases you want to be computed in the window **Subcases for Solution Sequence 101**

d Remove the Subcase Default in the window **Subcase Selected**

e Click **OK**

f Click **Apply**





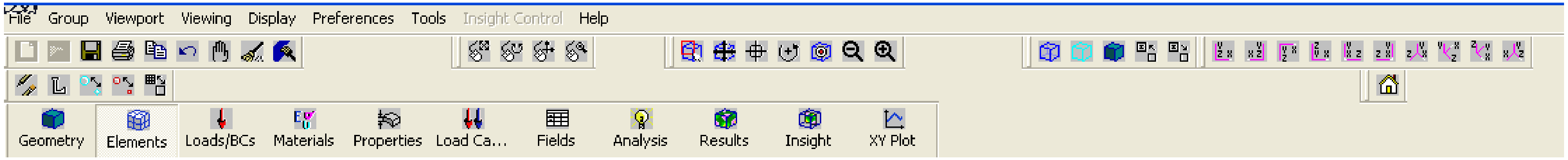
Extract of the BDF File (page 1 of 2)

```
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
SUBCASE 2
$ Subcase name : All_Load
  SUBTITLE=All_Load
  SPC = 2
  LOAD = 7
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
SUBCASE 3
$ Subcase name : Load_Fy=1000N
  SUBTITLE=Load_Fy=1000N
  SPC = 2
  LOAD = 12
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
```

Extract of the BDF File (page 2 of 2)

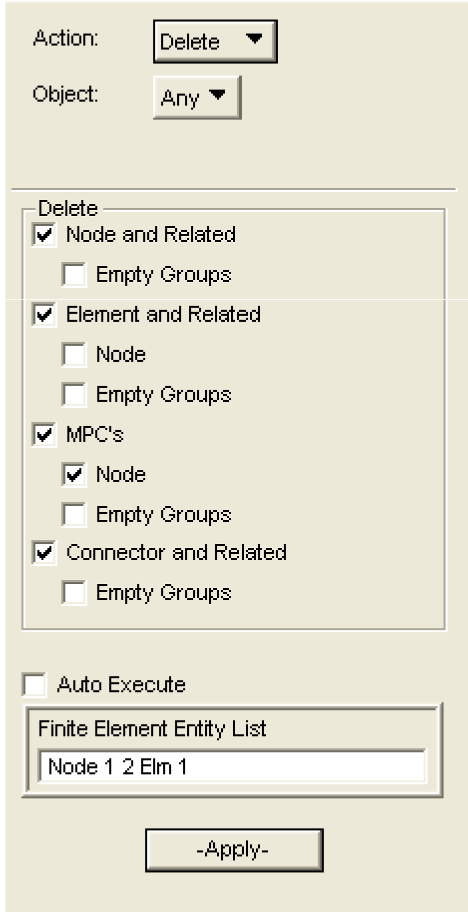
```
SUBCASE 4
$ Subcase name : Load_Px=100N/mm
  SUBTITLE=Load_Px=100N/mm
  SPC = 2
  LOAD = 14
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRAIN(SORT1,REAL,VONMISES,STRCUR,BILIN)=ALL
  SET 1 = 1,2
  GPSTRESS = 1
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
  FORCE(SORT1,REAL,BILIN)=ALL
  STRFIELD = ALL
$ Direct Text Input for this Subcase
SUBCASE 5
$ Subcase name : Load_Py=100N/mm
  SUBTITLE=Load_Py=100N/mm
  SPC = 2
  LOAD = 16
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 6
$ Subcase name : Load_Pz=100N/mm
  SUBTITLE=Load_Pz=100N/mm
  SPC = 2
  LOAD = 18
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
```

Modelling of the beam by several elements



Before creating a new modelling, the best solution is to erase all the nodes and the elements.

As the BCs, the load and the properties are associated to a geometry, these values aren't modified by this operation.

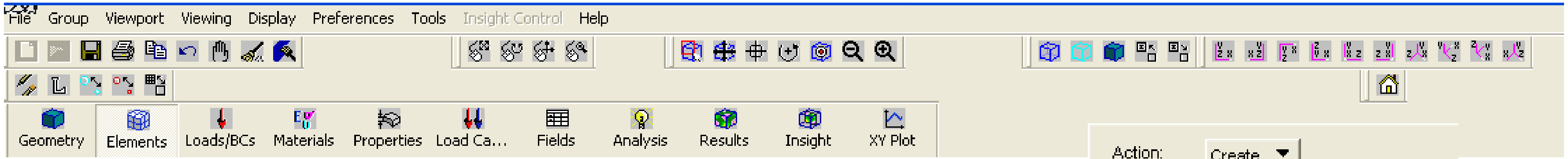


a **Elements/Delete/Any**

b Pick the entire model with the mouse

c Click **Apply**

Modelling of the beam by several elements



Create a new finite element mesh :

a **Elements : Create / Mesh / Curve**
b Set topology to **Bar2**
c Activate **Curve list** with the left click
d Screen pick the line on the screen or write Curve 1.
e Unselect **Automatic Calculation**
f Enter Value **100** for Global edge length
g Click **Apply**
h Do a new analysis, and don’t forget to select the Subcases you want !