Groups, Hypothesis, Material, Functions

Boundary Conditions and Loading

NOTE :
Command list presented in this chapter is not exhaustive. The complete information is available in on line documentation.

Table of content

1. Generalities

1.1. Groups

- Groups (**.SEL**)

1.2. Hypothesis

- Hypothesis **(.HYP)**

1.3. Materials

- Creation **(.MAT)**
- Applying **(.AEL)**

1.4. Physical properties

- Creation, assignation of thickness **(.PHP)**
- Creation, assignation of profiles **(.BPR; .AEL)**

1.5. Boundary conditions

- Displacements, fixations. **(.CLM)**
- Contact structure - rigid foundation **(.JEU)**
- Gaps **(.JER)**
- Linear relations **(.CLI)**
- Rigid bodies **(.RBE)**
- Local axis **(.AXL)**

1.6. Loading

- Nodal forces **(.CLM CHA)**
- Accelerations **(.CLM ACCE + .MAT M)**
- Pressure **(.CLM PRES)**
- Temperature **(.CLT)**
- Centrifugal **(.CLM ROTA, GEL O)**

1.7. Functions

- Functions **(.FCT)**

2. Groups creation (.SEL)

- Group types :

- Groups of nodes
- Groups of element faces or edges
- Groups of elements
- Groups of interfaces

- Use :
  - Boundary conditions
  - Material properties
  - Loading
  - Copies, Symmetries, displacements
  - Post processing, ...

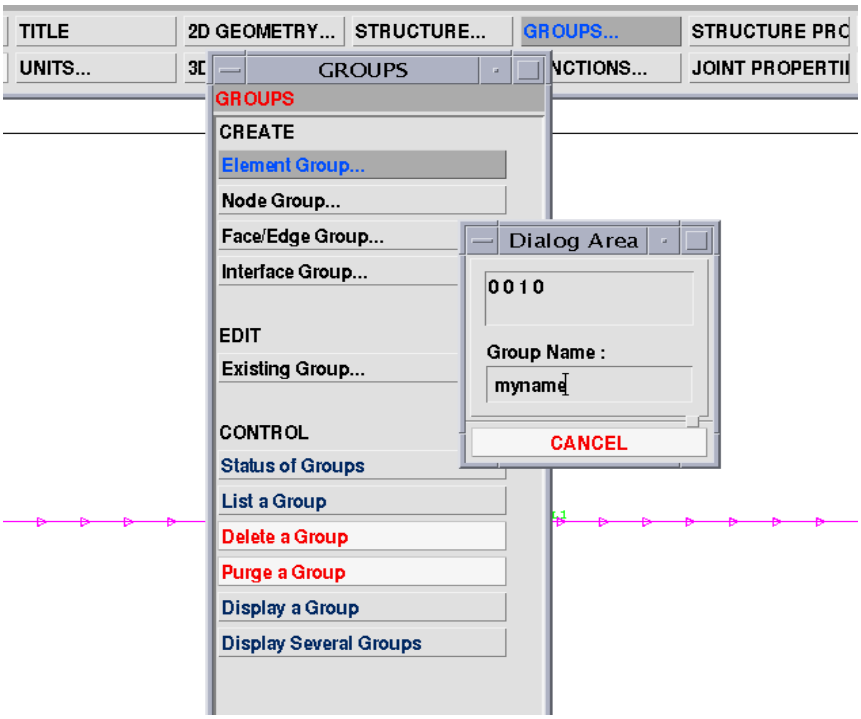
- Designation of a group :
  1. List of entities numbers
  2. Number of group
  3. Or name of group
  4. Type of group entities

Example:

```
.SEL GROUPE 1 NOM "myname" MAILLE
     I …..
     SUPPRIME I …
```

- Select modes :

All, line, point, attribute, digitalisation, list, structural or boxes (structural, cylindrical) , by type of elements (beam, triangle, quadrangle) or by boolean operations on groups (union, intersection, minus, transformation), ...



Select a group of elements using their attribute number
.SEL GROUP "MYNAME" MAILLE **ATTRIBUT 10**
Select a group of nodes using lines numbers
.SEL GROUP "NAME" NOEUDS **LIGNE 20**
Select a group of faces using their location number
.SEL GROUP "name1" FACES **MAILLE ATTRIBUTE n FACE 6**
Transform a group of faces "name1" into a group of nodes
.SEL GROUP "name2" NOEUDS **TRANSFORME "name1"**
Select a group of nodes using attribute number and suppress nodes linked to a line
.SEL GROUP "NAME" NŒUDS **MAILLE ATTRIBUT a** **SUPPRIME LIGNE 40**
Groups of elements by combination of two groups
. SEL GROUP "NAME" Elements **MAILLE ATTRIBUT 20** **GROUP "elem1" MOINS "elem3"**

Select a group of faces using : attribute + structural box + N°of face

```
. SEL GROUP "NAME" FACES
      MAILLE ATTRIBUT 5
      BOITE STRUCTURE XI 0 XS .3
      YI 0.5 YS 1.5 ZI 0.1 ZS 0.6 FACE 4
```

Select a group of faces using numbers of one or more nodes

```
. SEL GROUP "NAME" FACES
  NOEUDS I 5 9 4 7
```

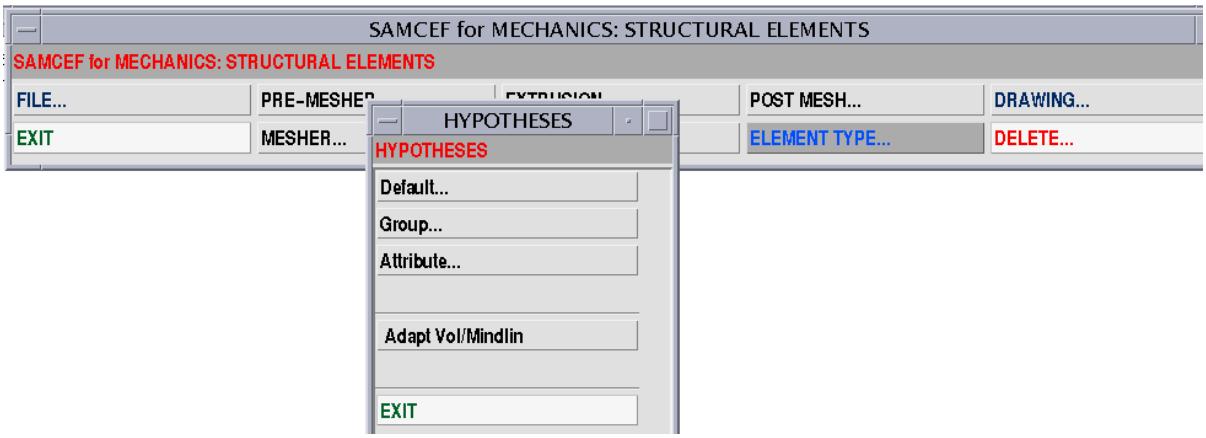
Menu (Filters)	***Commands***
To Select faces	
External Faces All Faces Faces orientation Precision angle : <angle>	**Peau 1 ; Grap remplissage peau** **Peau 0 ; Grap rempl 2 Peau 0** **Orientation x y z Tolerance <angle>**
For groups combination	
Combine groups Add Group Intersection	 **… Union <N° Group>** **Identique < N° Group1> < N° Group2>**
For Box selection	
Rectangular box Click on lower left corner Click on upper right corner	 **… STRUC 0 Box XI xi1 xi2 xs1 xs2…**
Cylindrical Box Select the centre of base Select a point on the circumference	 **… STRUC 0 Box CYLIN xb1 xb2 xc1 xc2**
Cylindrical Box (Radius) Select the centre of base Radius	 **… STRUC 0 Box CYLIN xb1 xb2 Rayon r …**
Other selections	
All	**… Tout …**
Number	**I**
Numbers (Loop) Frst Number : <i> Lst Number : <j> Step : <k>	**… I <i> J <j> K <k> …**
By Elements numbers	**… Elements <numbers > …**
By Node numbers	**… Nodes <numbers > …**
Rods only	**Rods**

Menu (Filters)	*Commands*
Beams only	**Beams**
Triangles only	**Triangles**
Quadrangles only	**Quadrangles**

3. Functional Hypothesis (.HYP)

3.1. Objective

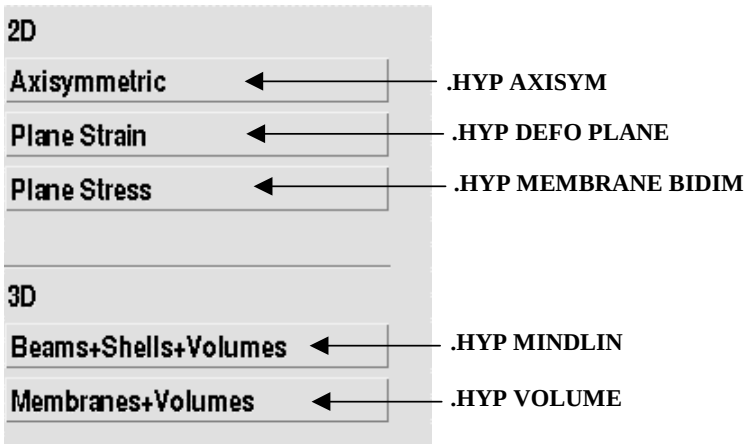
To define functional hypothesis to topological cells. **Mandatory** command.
This command associates a finite element model to a topology.



Hypothesis Mindlin for all elements of group "shell" :
`.HYP GROUP "shell" MINDLIN`

Entities can be selected by :
- Group or list of groups
- Attributes or list of attributes
- Cells or list of cells

3.2. Mainly used hypothesis



Hypothesis	Elements type
Axi-symmetric	48, 26 and 15
Plane strain or Plane stress	30, 26 and 15
Mindlin	21, 22, 28 and 29
Volume	8, 46, 47,22,57,58,151

3.3. Remarks

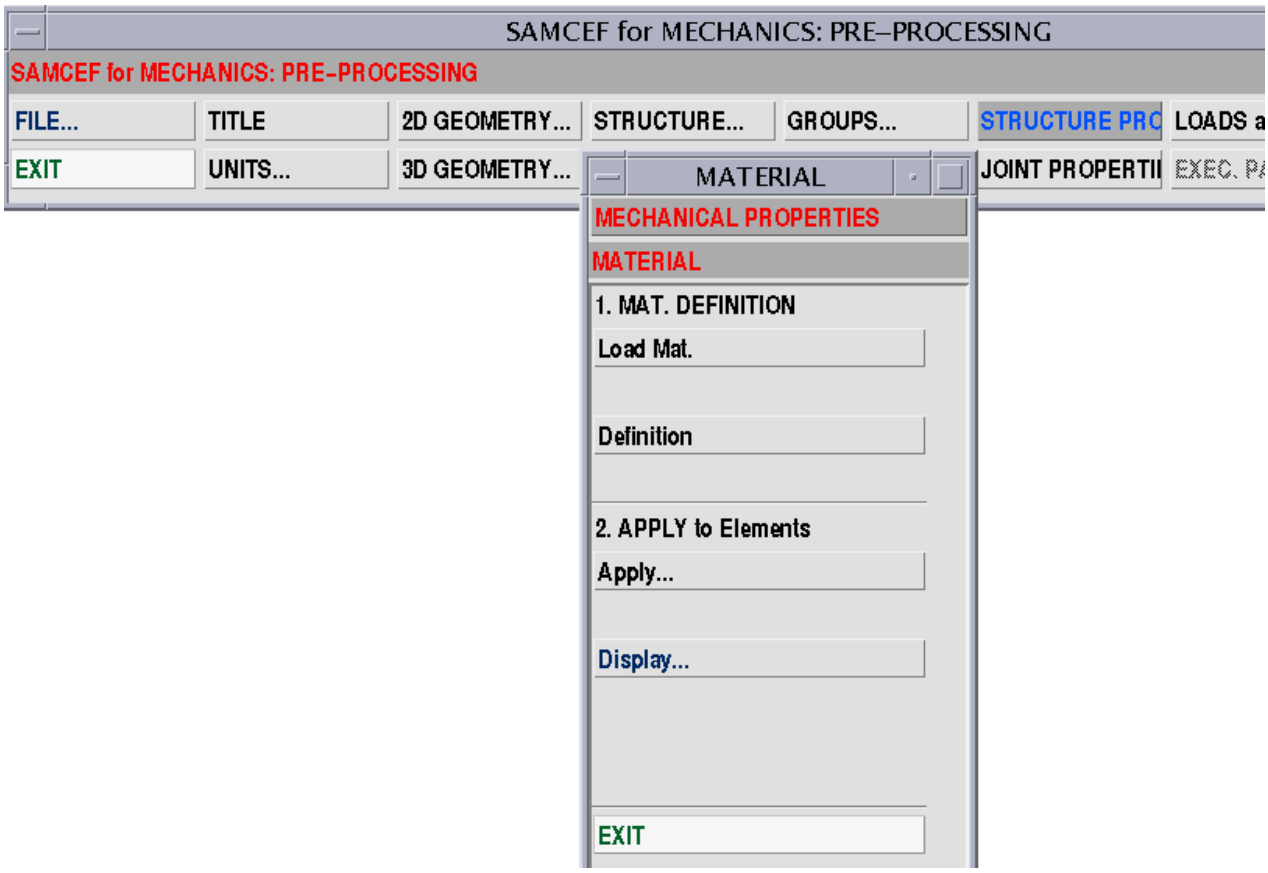
Without selection, this mandatory command is applied to the whole model.

This command has to be applied ***after*** cell generation and ***before*** physical properties and boundary condition application.

NO default hypothesis exists in Samcef.

The option **Adapt Vol/Mindlin** is used to reduce at degree 1 all the common edges used by a solid and a shell element at degree 2.

4. Material properties (.MAT)



Creation of a material

```
.MAT NOM "steel"
     BEHA “Elastic”
     YT 21000
     NT 0.3
     A  24.E-06
     M 7800
```

Meaning of main abbreviations:

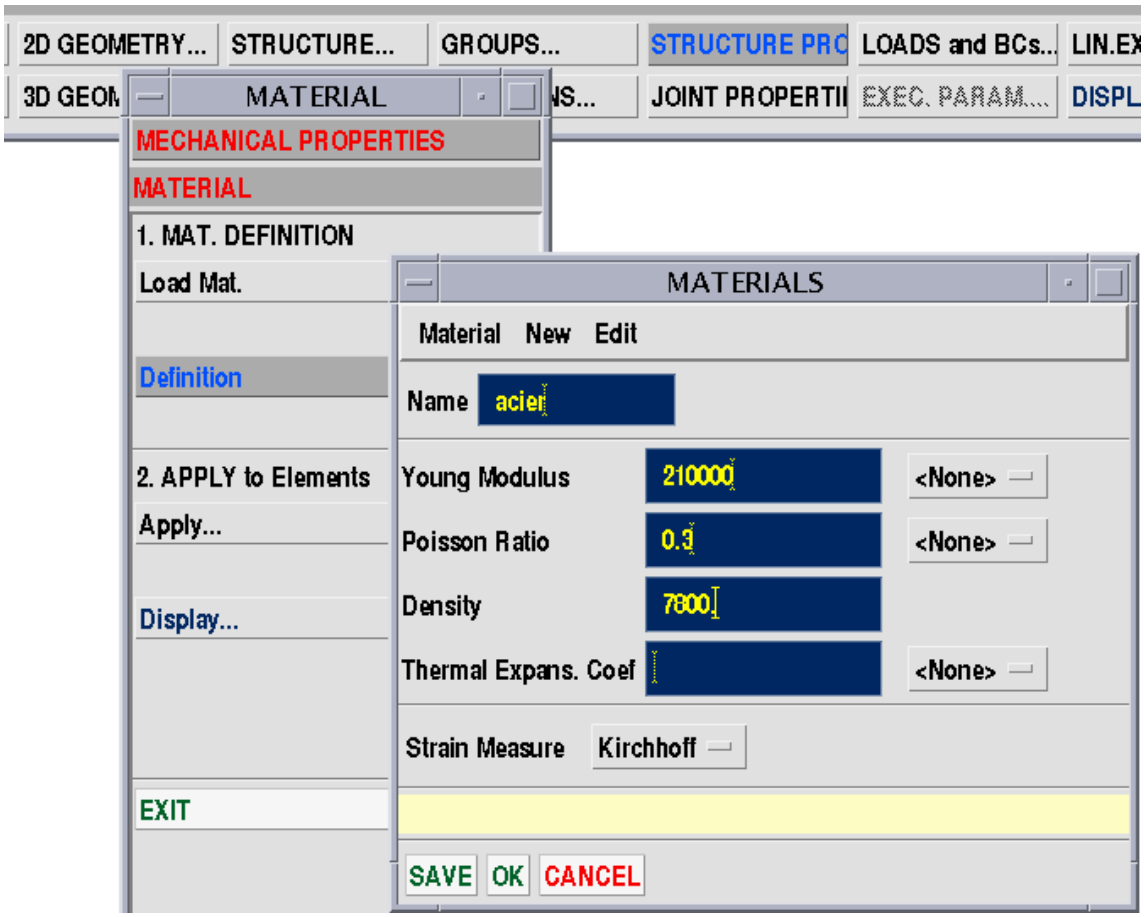
BEHA Behavior description
YT: Young's Modulus
NT: Poisson Ratio
A: Thermal Expansion Coefficient
M: Specific Mass.

If orthotropic properties are defined, the user must defined 3 young moduli, 3 Poisson coefficients and 3 shear moduli.

Entities can be selected by :

- Group or list of groups
- Attributes or list of attributes
- Cells or list of cells

Example:

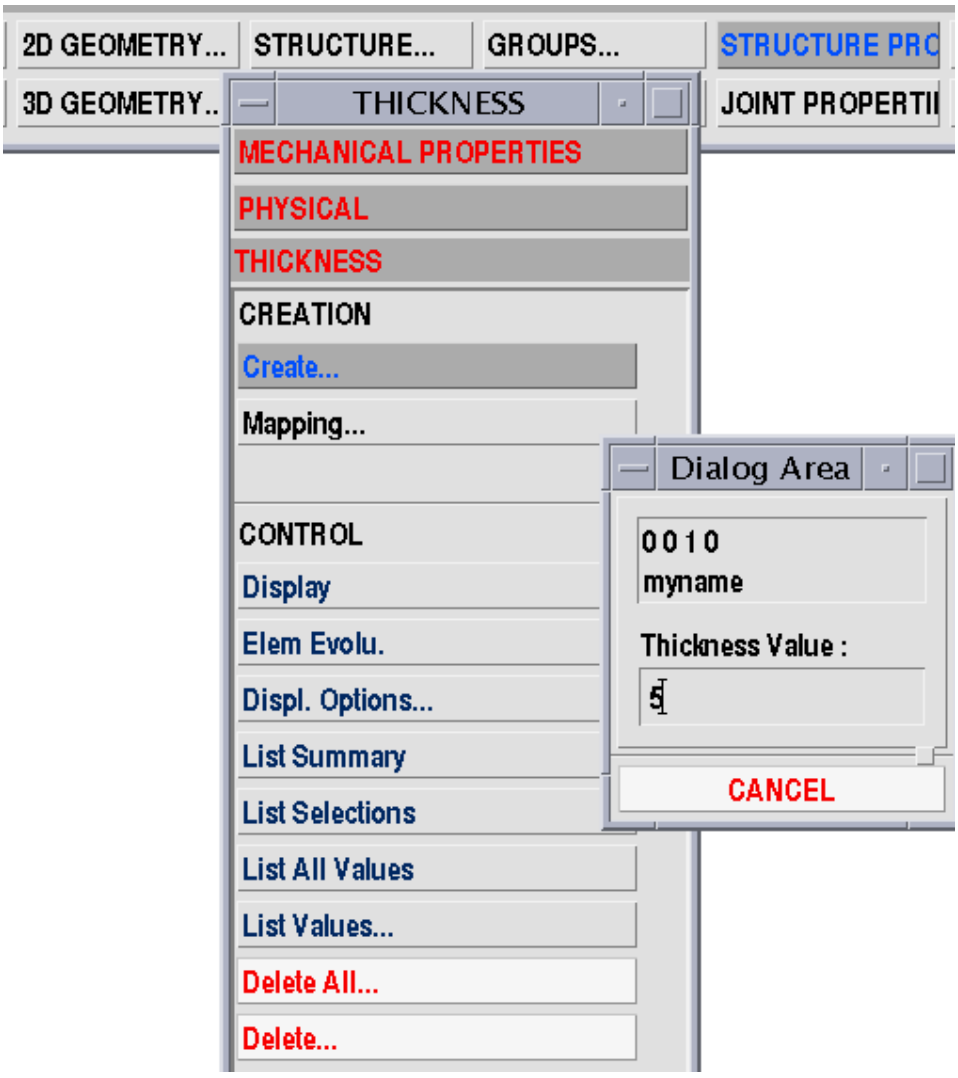


Definition of the material
.MAT NOM "acier" **YT 21000** **NT 0.3** **M 7800**
Apply material "acier" to the group of elements "solid"
.AEL GROUPE "solid" MAT "acier"
Check/Display of the material location
.DES MAT 1;VI
Display of the Young's modulus if it depends on the temperature introduced by .CLT TFX
.DES MAT "Steel" PROP "YT" TEMP 32

5. Physical properties (.PHP - .BPR)

It assigns a thickness to shells and membranes **(.PHP THICK)**, areas for rods and inertia/profile for beams **(. BPR).**

5.1. Shell and Membrane properties(.PHP)

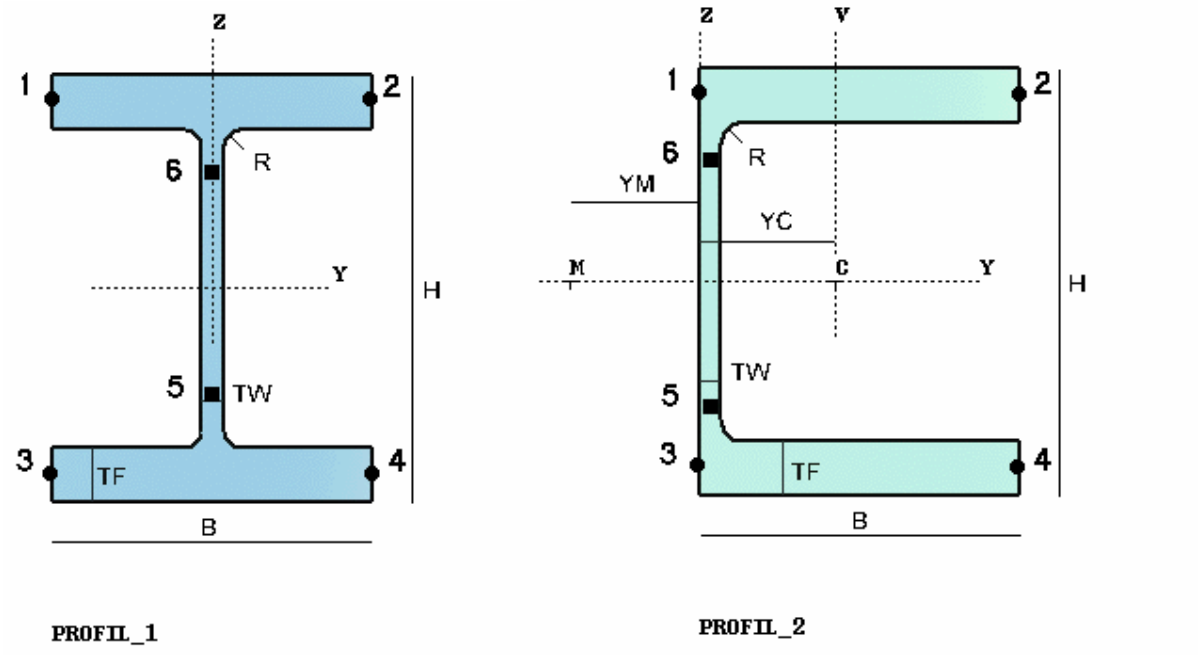


Give a thickness of 5 on element group "shells"
.PHP THICK GROUP "shells" VAL 5
Display of the thickness
.PHP VI THICK

5.2. Beam profiles .BPR

5.2.1. Goal

.BPR defines beams characteristics (areas, inertia) from profile dimensions.
This command can create a profile (or a catalogue of profiles) and select a profile from a catalogue. A general catalogue (Arbed_profiles) with 500 profiles is provided.




```
.BPR NOM "PROFIL_1" UNITE 0.001 TYPE "I"  H 400 B 300 TW 20 TF 50 R 20
.BPR NOM "PROFIL_2" UNITE 0.001 TYPE "U" H 400 B 300 TW 20 TF 50 R 20
```

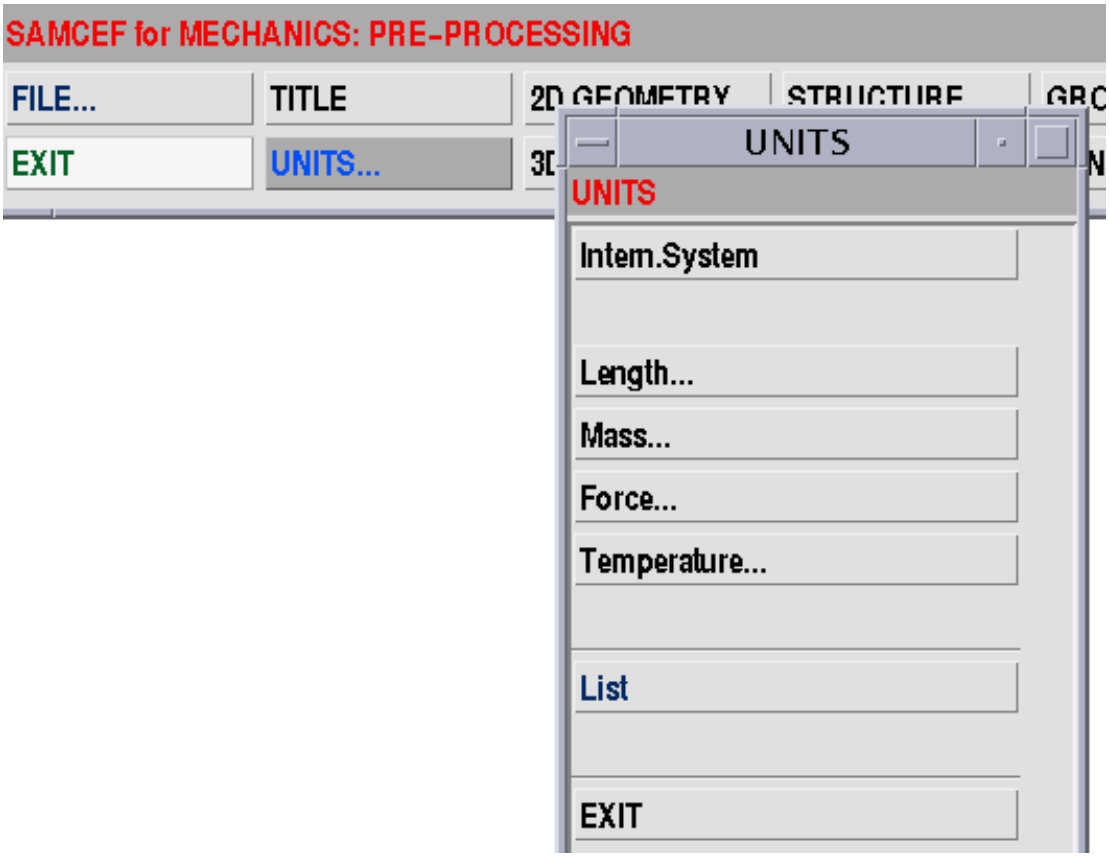
The general way to proceed is the following one :

1. Definition of the unit system (.UNIT)
2. Definition of the beam profile (.BPR)
3. Apply the profile properties to the selection
4. Check the orientation

This process must also be added to the use of the command .BEAM that allows orientating the section of the beam finite element.

5.2.2. Units (**.UNIT**)

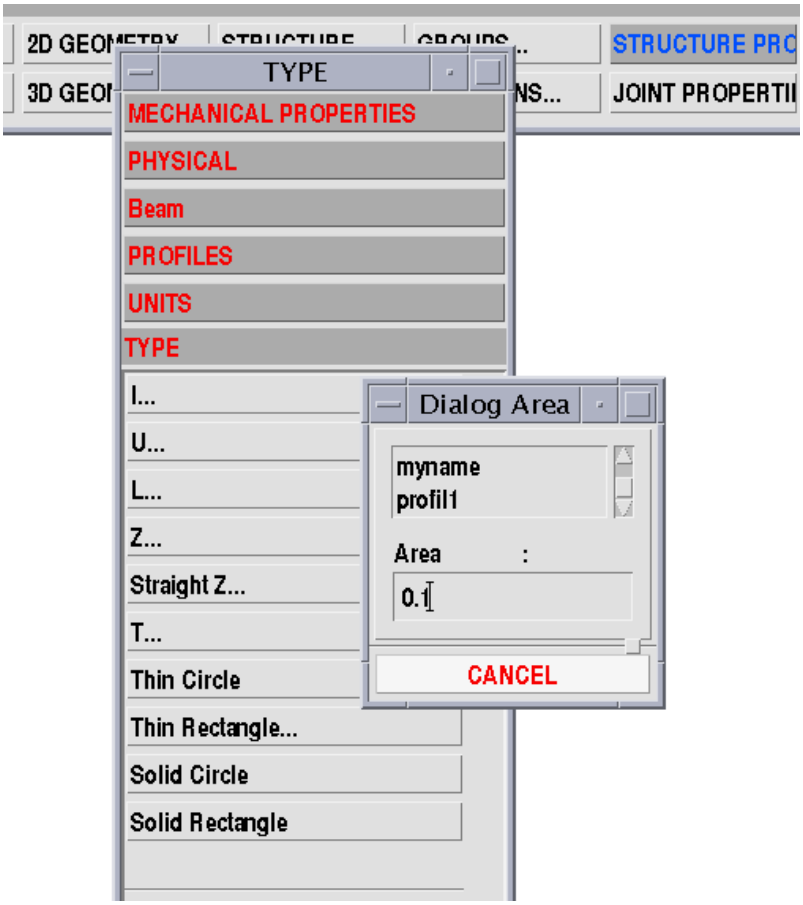
This is only **mandatory** for **.BPR** command (beam profiles).



Specify the international UNIT system (MKS) to the model
.UNIT SI

5.2.3. Syntax

For general properties of beams:



Sections, inertia of beams
.BPR NOM "PROFIL1" UNITE 0.001 **AIRE 0.1 IT (1./12.*1.E-3) IU 8.e-3 IV 8.e-5**
List of the profile
.BPR LIST "PROFIL1"
Apply the profile"profil1" to the group of element "shell"
.AEL GROUP "Shell" PRO "PROFIL1"
Display of the profile
.BPR VI "PROFIL1"
Rods area
.BPR NOM "ROD1" AIRE 0.1

Examples :

.UNIT SI

.BPR NOM "PRO01" UNITE 0.001 TYPE "I" H 300 B 200 TW 5 TF 10 R 0.1
.BPR NOM "PRO02" UNITE 1 TYPE "RECT" UNITE 1 H 0.003 B 0.02

.AEL group 4 PRO "PRO01"

.DES
GRAP ORIENT
VP 2
VI

UNITE is a multiplicative factor with respect to meter (always !) whatever the unit chosen in .UNIT

.des vp 2 display the real beam profile

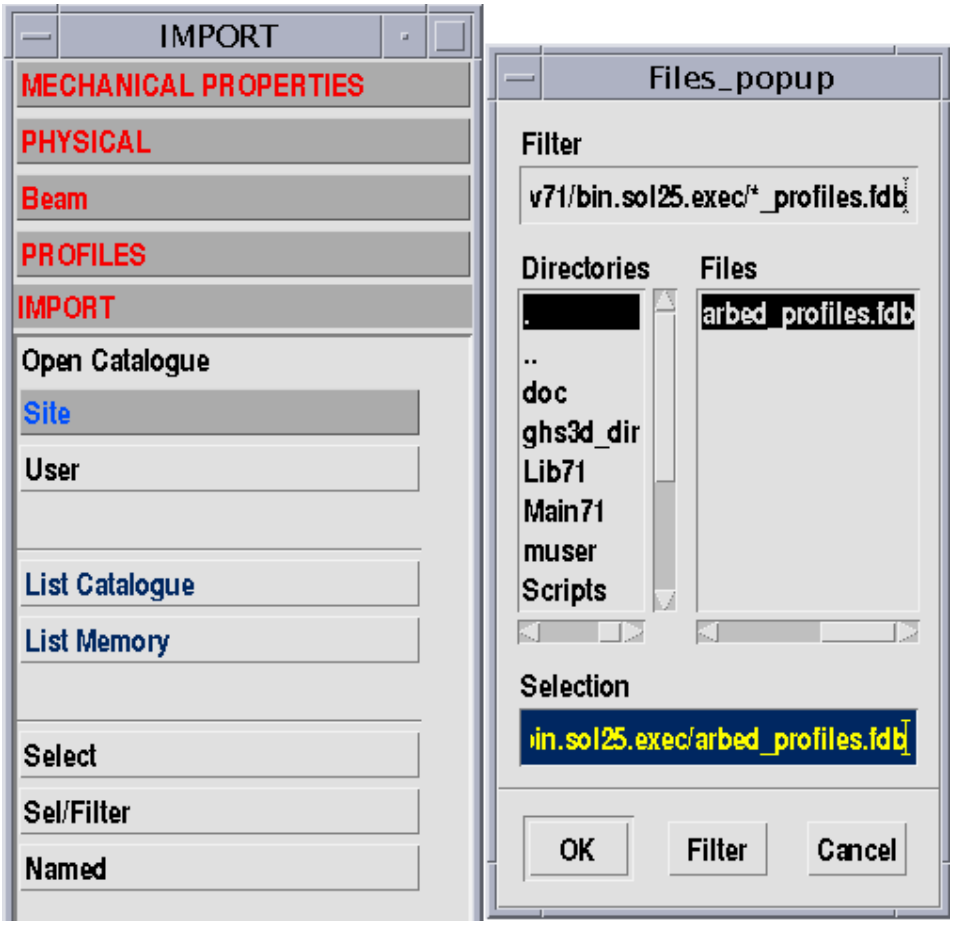
5.2.4. Use of catalogue

The use of a catalogue follows two steps :

1. import the catalogue
2. Select and import the candidates

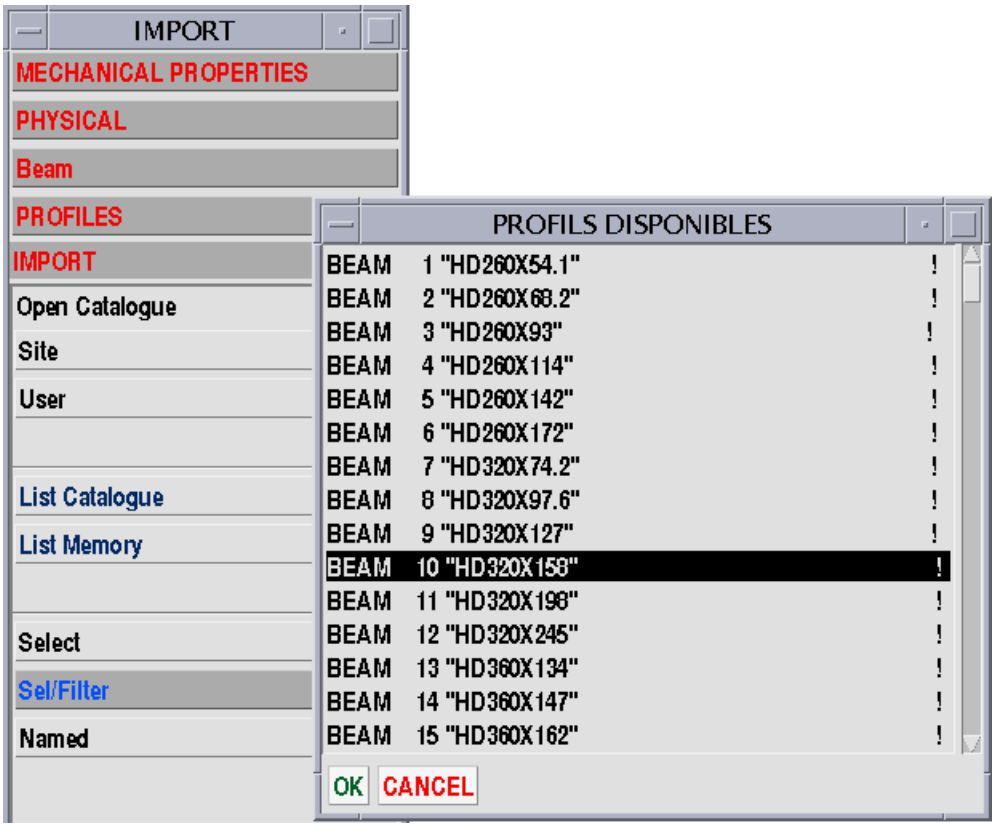
And then the profiles can be assigned to the beam elements.

1. Import of the catalogue.

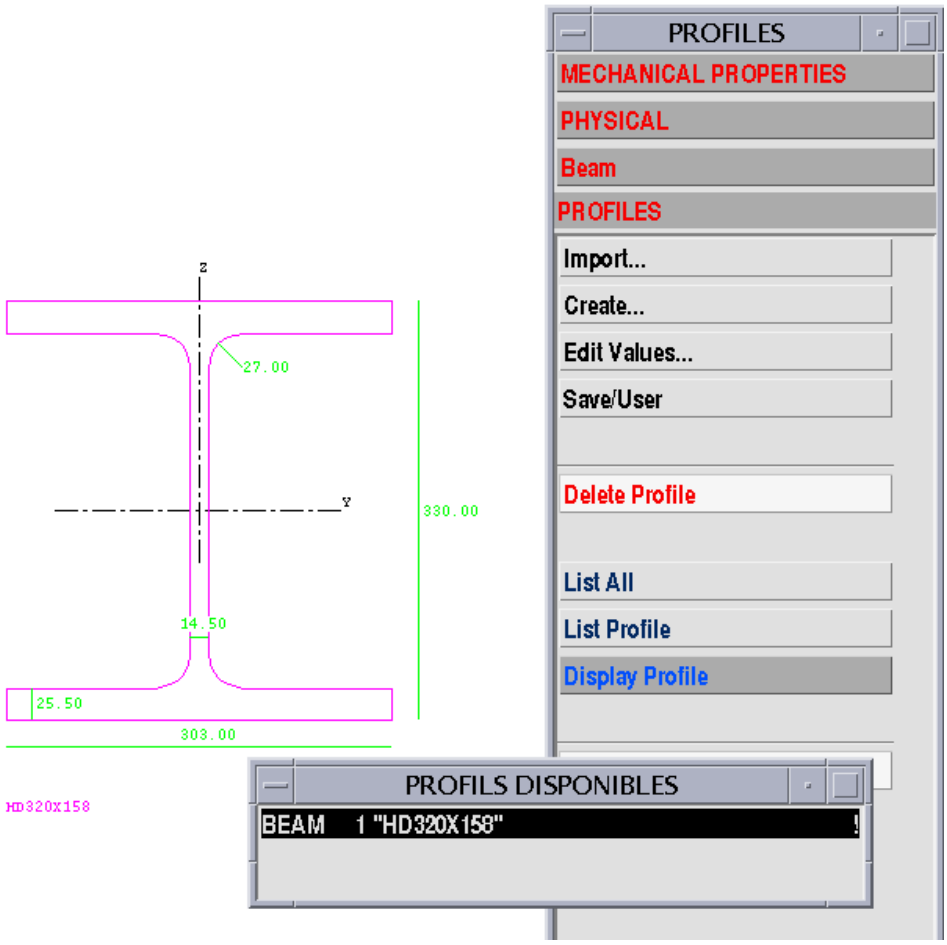


Import of the catalogue
.BPR IMPORT CATALOGUE "&EXE./arbed_profiles" FORMAT
List of the catalogue
.BPR LIST TOUT CATALOGUE

2. Selection and import the candidates

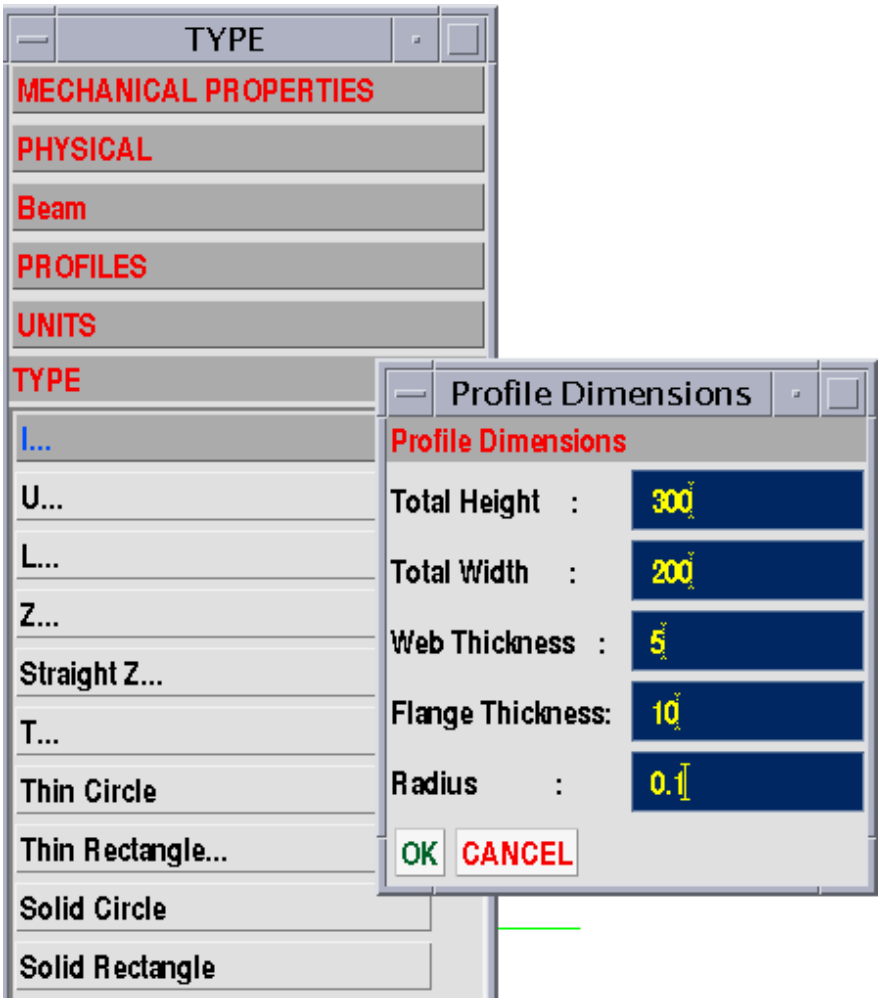


Select by filter "H" and choose "HD320X158" profile
.BPR IMPORT NOM "HD320x158"



Display of the profile
.BPR VI "HD320X158"
List of the profile
.BPR LIST "HD320X158"

5.2.5. Creation of a profile



Definition of the unit system
`.UNIT SI`
Creation of a "I" profile
`.BPR NOM "PRO01" UNITE 0.001` `  TYPE "I" H 300 B 200 TW 5 TF 10 R 0.1`
Assignation to the group "beam"
`.AEL GROUP "beam" PRO "PRO01"`
Display of the profile on the beams (Display Beams & Shapes).

`.DES`
` GRAP ORIEN`
` VP 2 ;VI`

6. The .CLM command

.CLM Support_Nodes Entity_Type Value [History_Variation]

where

- Support_Nodes is the description of the nodes to be prescribed,

  (NOE I node_nr or GROUP "SEL_Node_Group_name" or ...)

- Entity_Type is a keyword describing the type of boundary condition (Fixation, Displacement, ...): DEI, VII, FIX, DEP, POS, VFX, ACC.

- Value is the reference value of the boundary condition to be applied.
  Value_1 (VAL Amplitude COMP Component_nr) is used to apply a constant value on a component of the support.

- History_Variation are optional parameters to describe how varies the value
  NC Loadcase_nr to assign the boundary condition to a load case in linear analysis

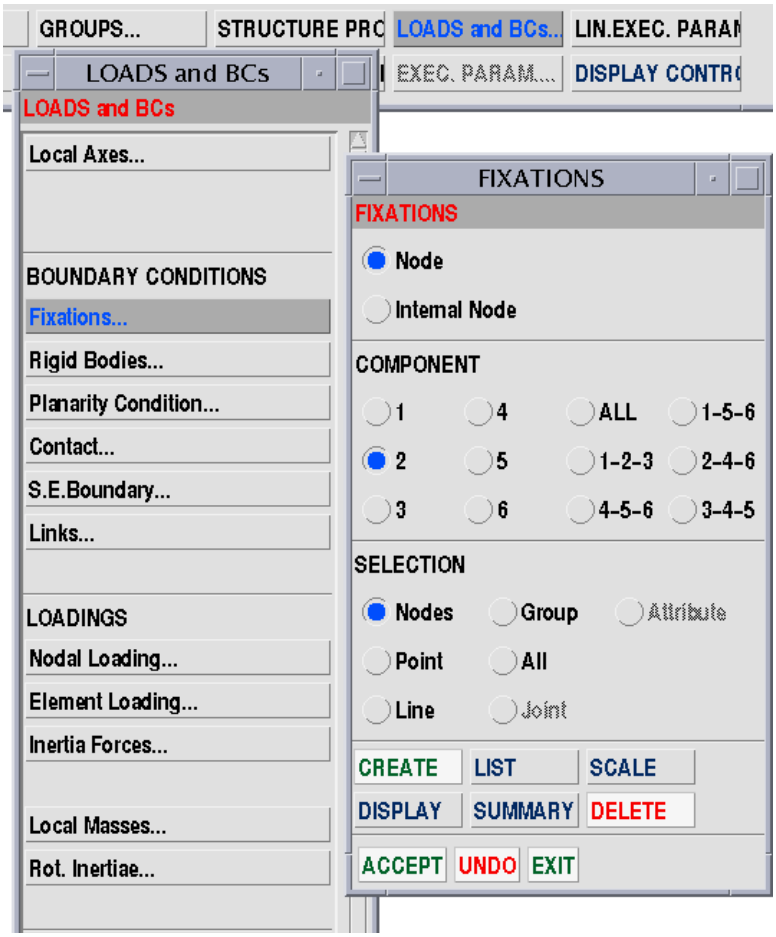
Example :

```
!
! C.1. Initial Conditions
! ----------------------
! not available
!
! C.2. Permanent conditions
! ------------------------
.SEL GROUP "N_Fix_1" NOE ...
.CLM GROUP "N_Fix_1" FIX COMP 1 5 6
        NOE I 3256      FIX COMP 3
!
! C.3. Prescribed conditions
! -------------------------
.CLM DEP NC  1
    NOE I 33142 VAL 0.0 COMP 1
    NOE I 33142 VAL 0.0 COMP 2
    NOE I 33144 VAL 0.0 COMP 1
    ...
.CLM DEP NC  2
    NOE I 33142 VAL 0.1 COMP 1
    NOE I 33142 VAL 0.0 COMP 2
    NOE I 33144 VAL 0.3 COMP 1
    ...
.CLM DEP NC  3
    NOE I 33142 VAL 0.1 COMP 1
    NOE I 33142 VAL 0.0 COMP 2
    NOE I 33144 VAL 0.3 COMP 1
```

7. Fixations (.CLM FIX)

Impose fixation on degrees of freedom of the model

Entities can be selected by :
- Group or list of groups
- Nodes or list of nodes
- Points or lines

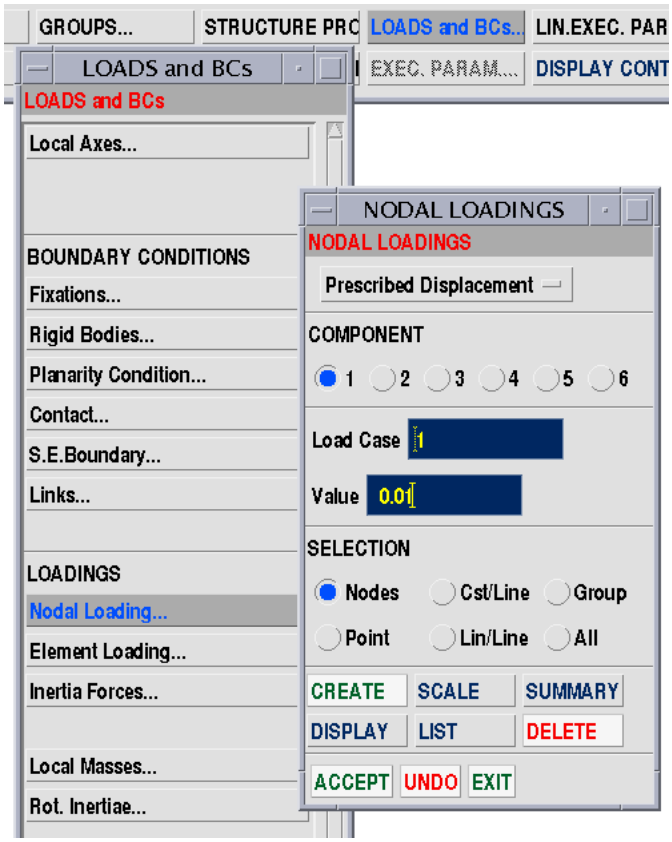


Fixation along Y direction of node 45
.CLM FIX NOEUD 45 COMP 2
Visualization
.CLM VI FIXA

8. Prescribed displacements(.CLM DEP)

Impose displacement to degrees of freedom.

Entities can be selected by :
- Group or list of groups
- Nodes or list of nodes
- Points or lines

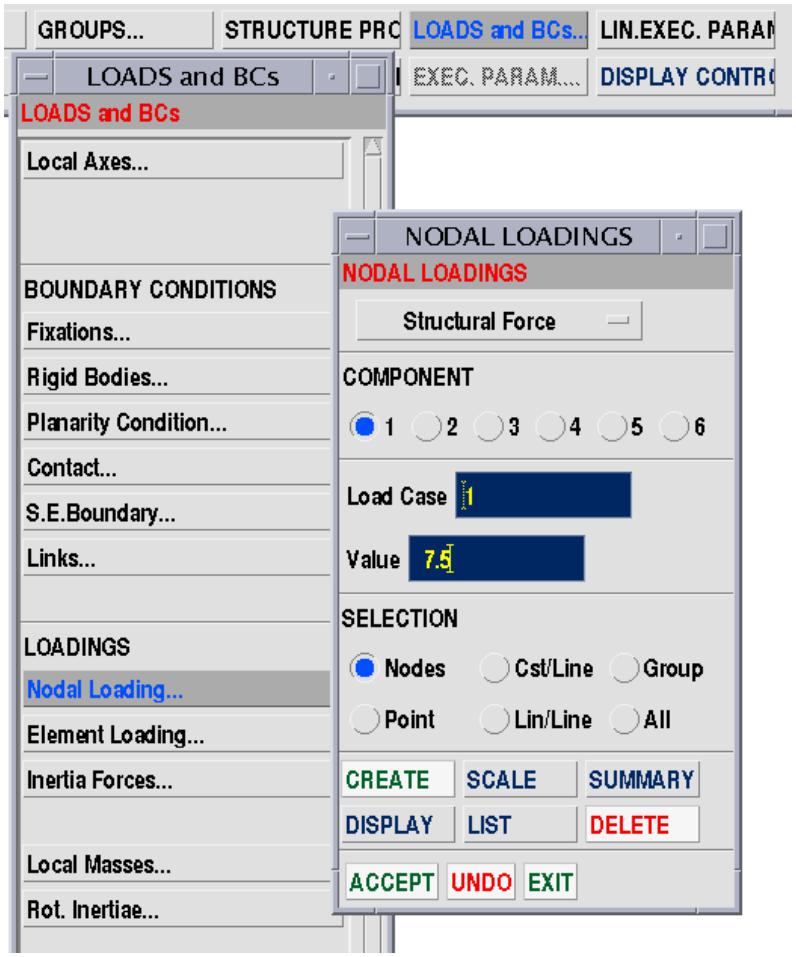


Prescribed Displacements on a list of nodes
.CLM DEPL NOE 6 A 8 COMP 1 VAL 0.01
Visualization
.CLM VI DEPL

9. Nodal loading (.CLM CHA)

Impose loads on degrees of freedom.

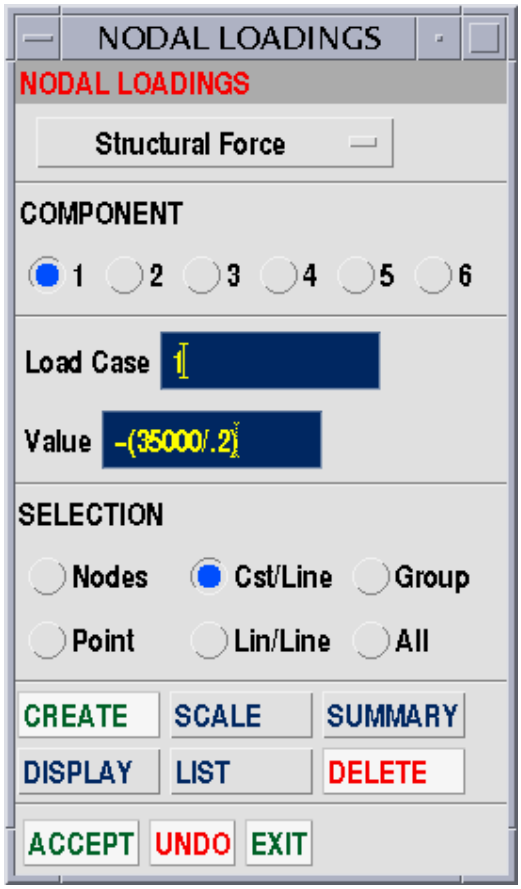
Entities can be selected by :
- Group or list of groups
- Nodes or list of nodes
- Points or lines



Applied load on node 4
.CLM CHA NOEUD 4 COMP 1 VAL 7.5 NC 1
Visualization
.CLM VI CHA NC 1

10. Line loading (.CLM CHA LIGNE)

Impose a distributed constant or linearly variable load on a line.



Loading with constant distribution
`.CLM CHA NOEUD COMP 1 LIGNE 9 CONST -(35000/.2) NC 1`
Loading with linear distribution
`.CLM CHA NOEUD COMP 2 LIGNE 12 LIN -3.5 –100 NC 2`

Remarks : Loading on CAD entities as support is allowed on components 1 to 3.

When distribution is linear, lineic force is -3.5 at the first extremity and -100 and the second extremity.

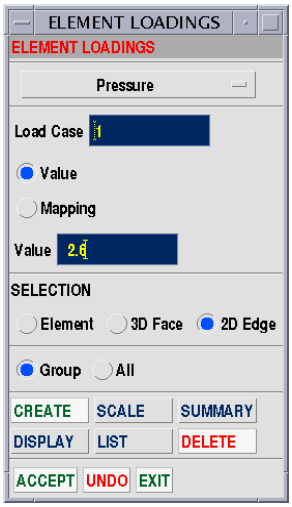
11. Pressure (.CLM PRES)

11.1. Entities

- Beams (according Z local axis)
- Shells selected by number, attribute or group
- 2D edges selected by groups (in 2D, plane strain, axisymetrical….)
- 3D faces selected by groups.

11.2. 2D Elements or Volume

- Pressure perpendicular to 2D edge or face.
- Positive if it compresses the element.
- Applied in the element plane and on edge if 2D.
- No local axis.
- Updated with structure deformation

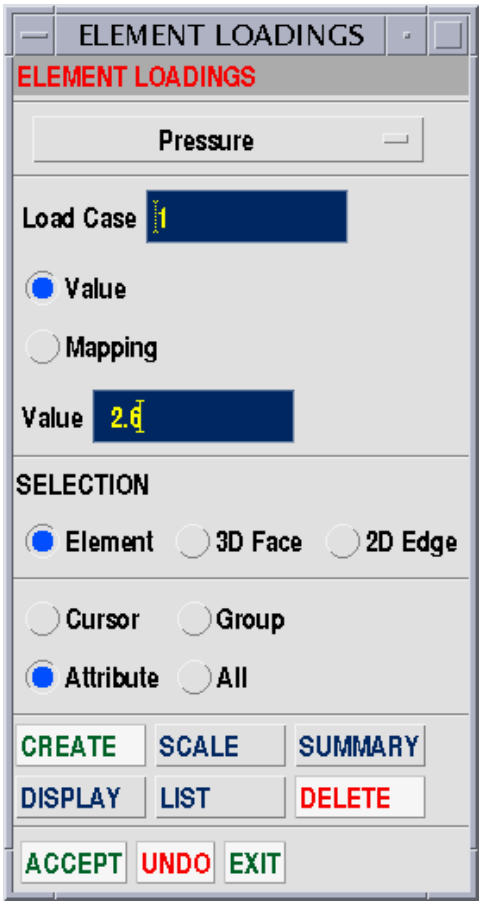


Pressure on a group of 2D faces
.CLM GROUP "FACES" PRESSURE VALUE 2.6 NC 1
Visualisation on 2D faces (edges)
.CLM VI PRES NC 1 ARETES
Visualisation on 3D faces
.CLM VI PRES NC 1 FACES
Isovalues display instead of vectors (PERMANENT COMMAND / Vectors / Modulus intensity)
VECT 0

11.3. Shells

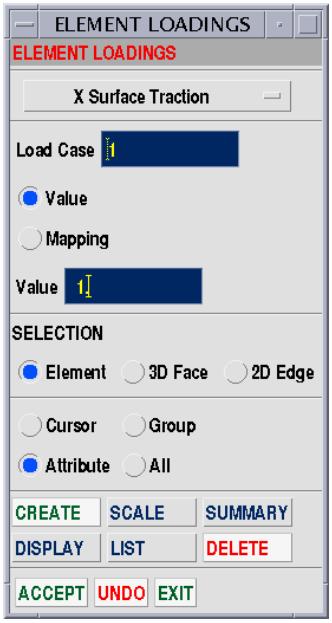
- Pressure perpendicular to shell surface
- Shells are defined by attributes or groups
- Positive according to Z local axis of element.

To display shell orientation: **GRAP ORIENTATION.** To modify a shell orientation, use the command .MOD INVERSE SENS.



Pressure on shell attribute number 2
.CLM ATT 7 PRES VAL 2.6 NC 1
Display
.CLM VI PRES NC 1 ELEMENTS

12. Distributed forces



12.1. Surface forces

Apply a surface force oriented upon structural axis on faces of volumes, shells or membranes.
Constant with structure deformation

```
.CLM GROUP 1 SFX VALUE 1.4 SFY VALUE 0.99 SFZ VALUE 9.01
```

12.2. Line forces on beams

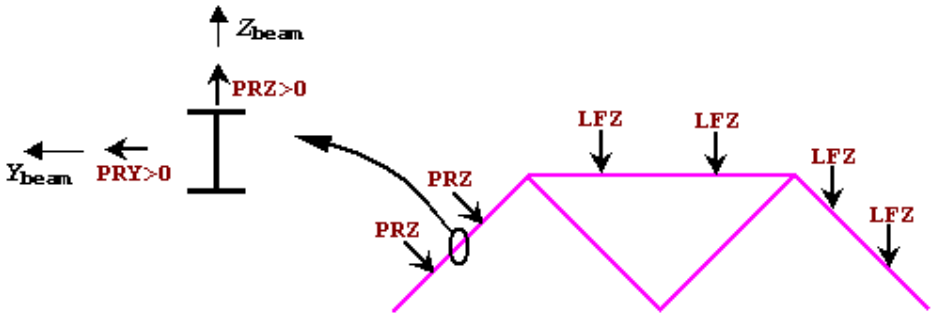
Applies a line force expressed on beam **local** axis.
Updated with structure deformation

```
.CLM  GROUP 1 PRY VALUE 45 PRZ VALUE 121
```

12.3. Line forces on rods or beams

Applies a force expressed on rods or beams in **structural** axis.

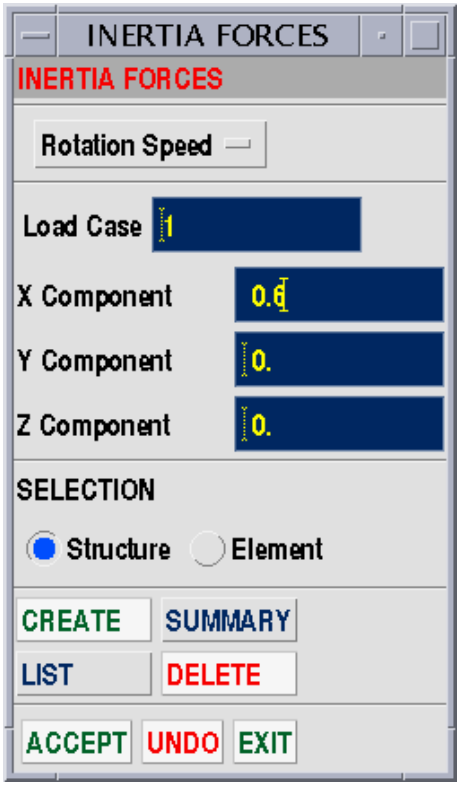
```
.CLM  GROUP 1 LFX VALUE 19.87 LFY VALUE 10.9 LFZ VALUE 12
```



13. Body loads

13.1. Centrifugal force (.CLM ROTA)

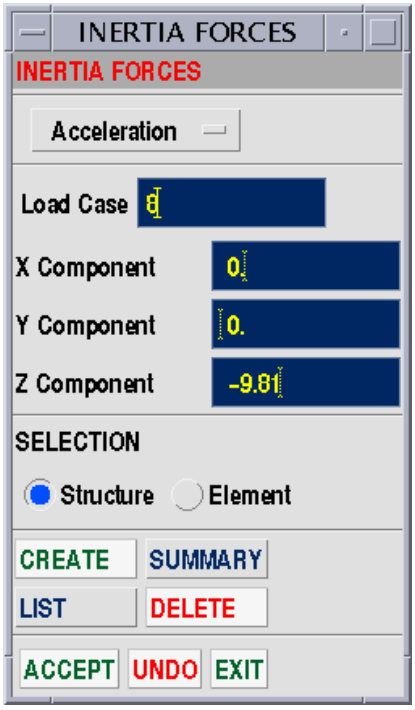
This command applies a rotation speed, which creates a centrifugal force.



Apply a centrifugal force on the whole structure
`.CLM ROTA STRUC VAL .6 0. 0. NC 1`
Visualisation
`VI ROTA NC 1`

- Units : [rad/sec]
- For axisymmetric, harmonic and multi-harmonic model, rotation can be defined only around axial structural axis
- Modification of stiffness matrix due to angular velocity is taken into account with above definition only if the geometrical stiffness is also present (command .SAM IFPR *file_nr* NPR *load_nr*). (See Second Effects).

13.2. Acceleration in translation



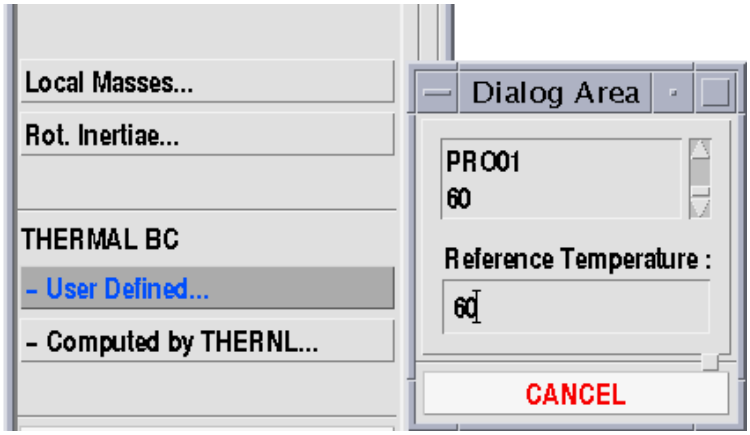
Acceleration of the gravity on the whole structure (Z axis)
.CLM ACCEL STRUC VAL 0 0 -9.8 NC 8
Acceleration of the gravity on part of the structure
.CLM ACCEL GROUP "part1" VAL 0 0 9.8 NC 1

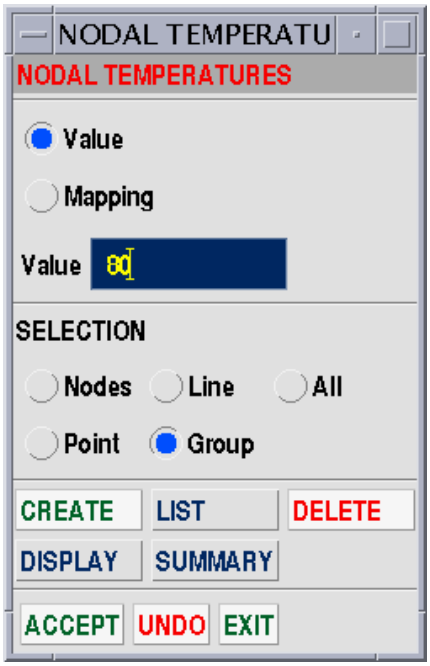
13.3. Acceleration in rotation

Acceleration in rotation on the whole structure
.CLM DROT STRUC VAL 0 0 4 NC 8
Specifying the centre of rotation
.CLM CROT STRUC VAL 0 0.4 1 NC 1

Caution:
density has to be defined in **.MAT**- Caution to be taken wrt used unit ([kg/m3] -> acce. in [m/s2] when mass in [tonne/mm3]->acceleration in [mm/s2].

14. Imposed temperatures (.CLT, .GEL)



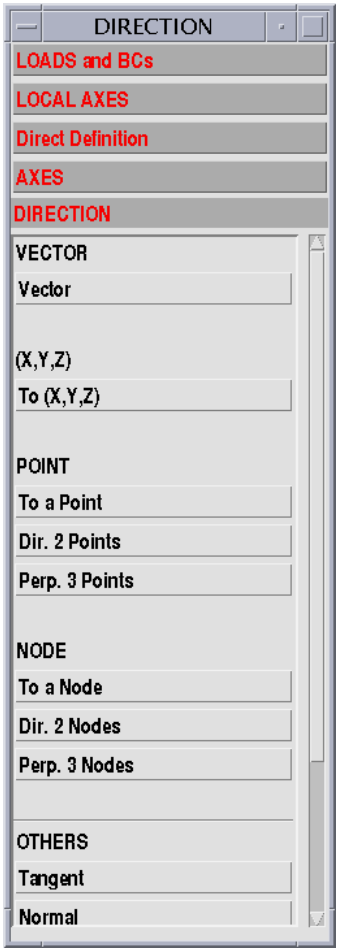


Reference temperature
.GEL TEMP 60
Temperature on a group of nodes
.CLT TFX GROUP 4 VAL 80
Visualisation
.CLT VI TFX

Remarks :
.CLT TFX does not allow **NC** parameter.

15. Local axes (.AXL)

Goal : To apply loads or boundary conditions in local directions when these directions are different from the global axis.
Axis Support : nodes or nodes groups or geometrical entities like points, lines, circles.
Fixation and nodal forces defined on nodes with local axes are interpreted in the local system.



General syntax :

.AXL <selection> AXE I DIRECTION <direction>
<selection> can be nodes, lines, points
<direction> can be nodes, vector, points…

Local axes on a node
.AXL I 34 **AXE 1 DIR NOEUD 1 2** **AXE 2 DIR NOEUD 1 3**
.AXL I 35 frame 4
Visualization
.DES AXL VI

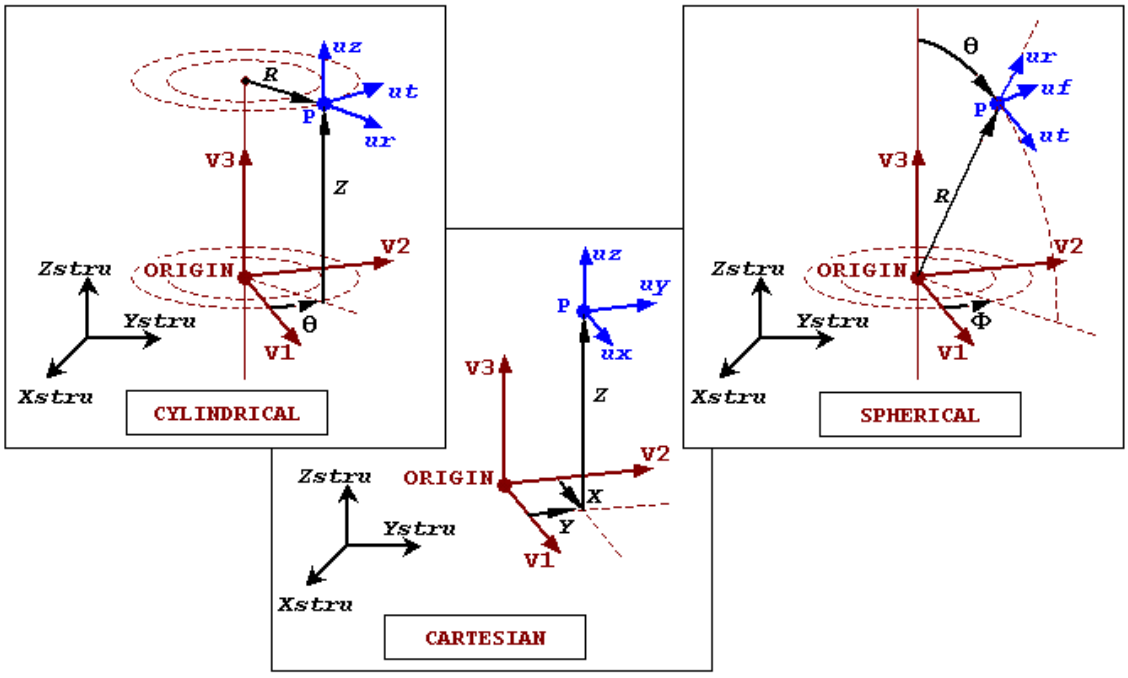
In the first example above, local axis are defined on node 34. First axis is tangent to direction defined by nodes 1 and 2 and second axis is tangent to direction defined by nodes 1 and 3.

In the second example, we use local axis defined by local frame. This local frame is defined by the command .FRAME. This kind of use of these commands can be very useful with a cylindrical symmetry for example.

Exercise :

Create local axes on exercise 10 (2 materials plate)

16. Coordinate System Definition (.FRAME)



This command can be used :

- To define boundary conditions in specific axes
- To define orthotropic material properties (ie composites material)
- Post processing

Syntax :

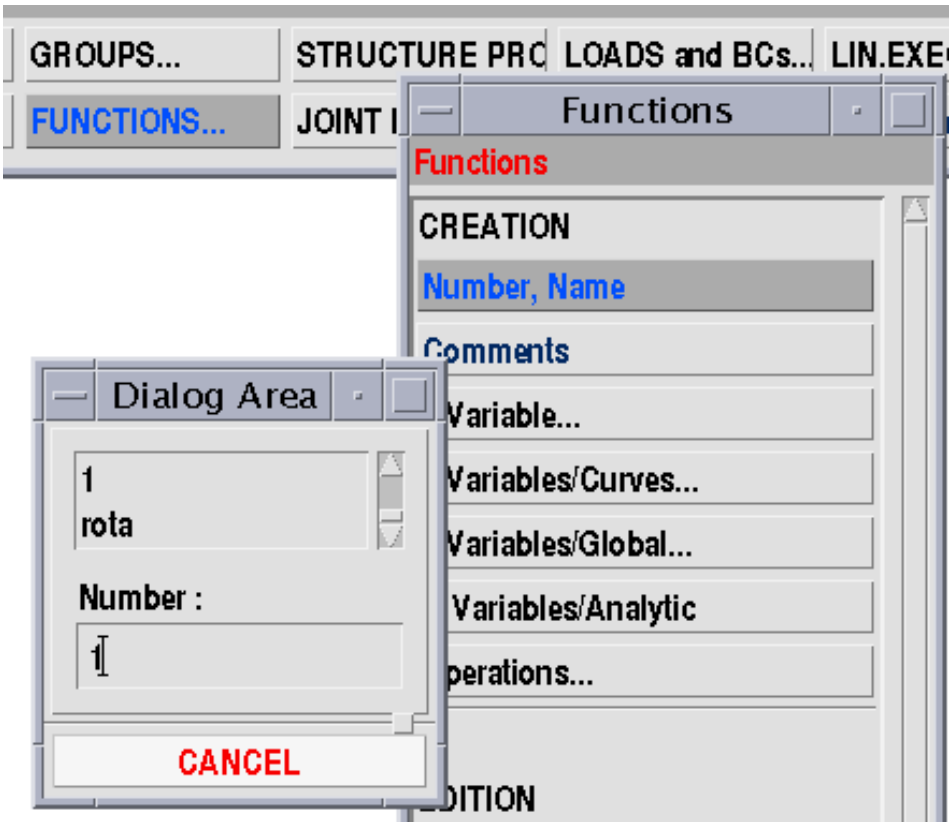
.FRAME I <number> TYPE <type>
ORIGIN <origin>
V1 vx_1 vy_1 vz_1
V2 vx_2 vy_2 vz_2
V3 vx_3 vy_3 vz_3

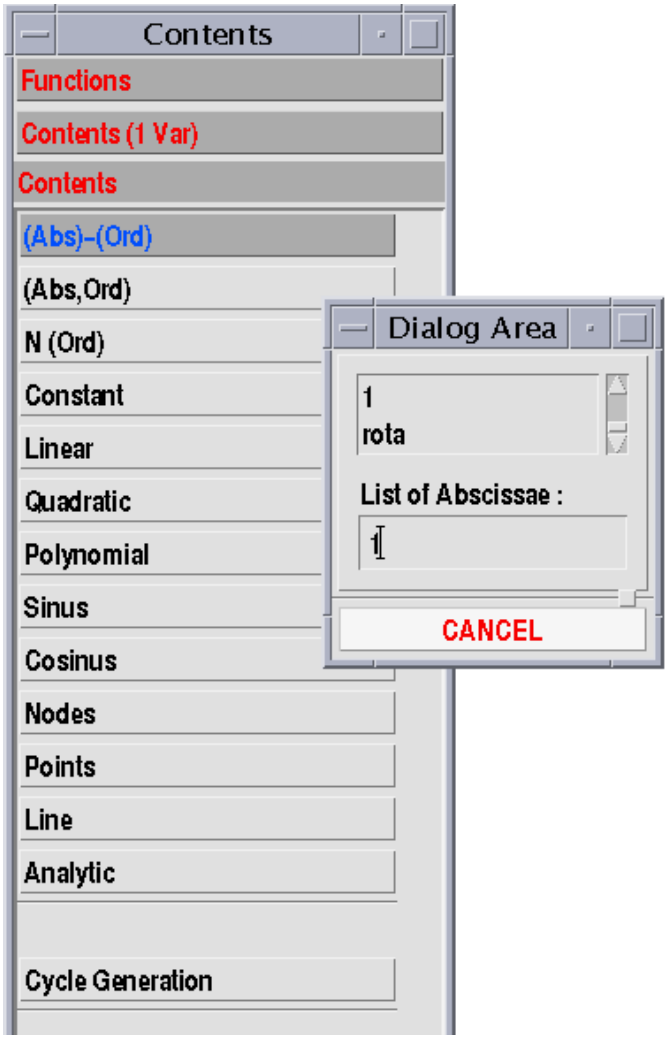
```
.FRAME  VI              ! Display all the frames
.FRAME  VN              ! Display all the frames and nodes
.FRAME  VP              ! Display all the frames and points

.AEL  <element_selection>  FRAME 1
```

17. Creation of functions (.FCT)

- One parameter u : the function describes a curve ;
- Two parameters u, v : the function describes a surface ;
- Three parameters u,v,w : the function describes a volume.

See manual for other possibilities





Creation of the function "rota"

```
.FCT CREE FUNCTION I 1 NOM "ROTA"
     CREE VALE Y U
       ABSCISSES  0  1
       ORDONNEES  0  45
```

Exercises :

- For exercises 7, 8, 11 and 13
  Create groups for fixations, loadings, materials and physical properties

- Exercise 9 for the use of beam profiles.
- Exercise 12s