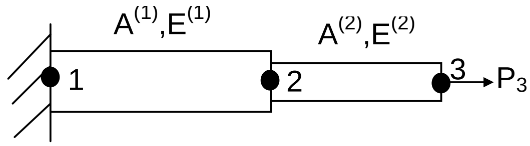
EXERCISES

Table of contents

1. MEF computation : theoretical exercice

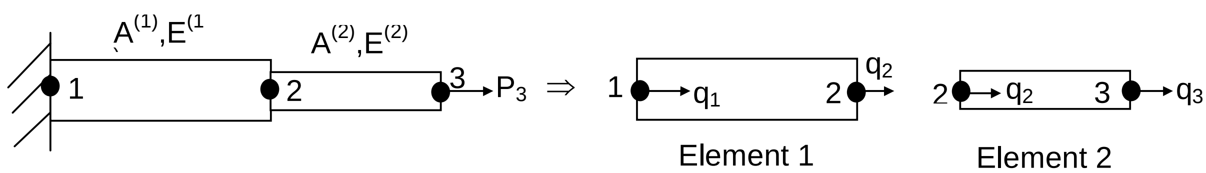


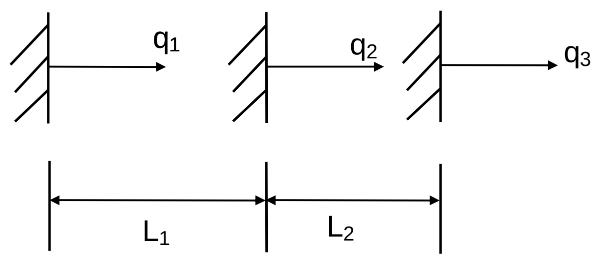
A = area

Compute displacements, strains and stresses in the structure :

- Step 1 : Choice of modelling
- Step 2 : Choice of shape functions
- Step 3 : Matrix
- Step 4 : Assembly
- Step 5 : Nodal displacements
- Step 6 : Strains et stresses in element.

1.1. Step 1 : modelling choice

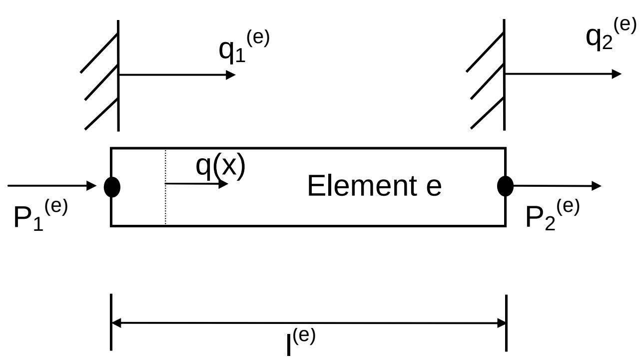




A = section

Axially loaded Structure $\rightarrow$ rod modelling

For one element



1.2. Step 2 : shape function choice

Assumption : linear variation of the axial displacement.

$u(x) = a + bx$

$u(x) = q_1^{(e)}$ at $x = 0$,

$u(x) = q_2^{(e)}$ at $x = l^{(e)}$

d'où

$$u(x) = q_1^{(e)} + \frac{q_2^{(e)} - q_1^{(e)}}{l^{(e)}} x = \left(1 - \frac{x}{l}, \frac{x}{l}\right)\begin{bmatrix} q_1 \\ q_2 \end{bmatrix} = (N)(q)$$

1.3. Step 3 : elementary matrices and minimum energy principle

$\dot{E}$ = strain energy – external forces work

$$\dot{E} = \Pi^{(1)} + \Pi^{(2)} - W_p$$

$$\Pi^{(e)} = A^{(e)} \int_0^{L(e)} \frac{1}{2} \sigma^{(e)} \varepsilon^{(e)} dx = \frac{A^{(e)} E^{(e)}}{2} \int_0^{L(e)} \varepsilon^{(e)^2} dx$$

with :

$$\varepsilon^{(e)} = \frac{\partial q}{\partial x} = \frac{q_2^{(e)} - q_1^{(e)}}{L^{(e)}} \qquad \text{(compatibility)}$$

thus :

$$\Pi^{(e)} = \frac{A^{(e)} E^{(e)}}{2L^{(e)}} \left(q_1^{(e)^2} + q_2^{(e)^2} - 2 q_1^{(e)} q_2^{(e)} \right)$$

Under matrix format :

$$\Pi^{(e)} = \frac{1}{2} \vec{q}^{(e)T} \left[K^{(e)}\right] \vec{q}^{(e)} \qquad \text{où} \qquad \vec{q}^{(e)} = \begin{bmatrix} q_1^{(e)} \\ q_2^{(e)} \end{bmatrix}$$

Applied to each structure element:

$\begin{pmatrix} q_1 \\ q_2 \end{pmatrix}$ for e = 1

$\begin{pmatrix} q_2 \\ q_3 \end{pmatrix}$ for e = 2

$$\left[K^{(e)}\right] = \frac{A^{(e)}E^{(e)}}{L^{(e)}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$ elementary stiffness matrix

1.3.1. External forces work

$W_P = q_1P_1 + q_2P_2 + q_3P_3$

ici : P_1 = fixed node reaction
$P_2 = 0$
P_3 = applied force

1.3.2. Energy minimum

$$\frac{\partial \dot{E}}{\partial q_i} = 0 \qquad i = 1,2,3$$

then

$$\frac{\partial \dot{E}}{\partial q_i} = \frac{\partial}{\partial q_i}\left[\sum_{e=1}^{2}\left(\Pi^{(e)} - W_p\right)\right] = 0 \qquad pour \quad i = 1,2,3$$

$$\sum_{e=1}^{2}\left(\left[K^{(e)}\right]q^{(e)} - P^{(e)}\right) = \vec{0}$$

1.4. Step 4 : Assembly

$$[K]\vec{q} - \vec{P} = \vec{0}$$

où : K = assembled matrix

$$\vec{q} = \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix}$$ nodal displacement vector

$$\vec{P} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix}$$ loading vector

$$[K] = \begin{bmatrix} \frac{E_1A_1}{L_1} & -\frac{E_1A_1}{L_1} & 0 \\ -\frac{E_1A_1}{L_1} & \frac{E_1A_1}{L_1} + \frac{E_2A_2}{L_2} & -\frac{E_2A_2}{L_2} \\ 0 & -\frac{E_2A_2}{L_2} & \frac{E_2A_2}{L_2} \end{bmatrix}$$

Relation to solve :

$$[K]\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} \qquad où \qquad \begin{matrix} q_1 = 0 \\ P_2 = 0 \\ P_3 = F \end{matrix}$$

1.5. Step 5 : Nodal displacement Solving

Introduction of the boundary conditions : $q_1 = 0$.

$$\begin{bmatrix} \frac{E_1A_1}{L_1} + \frac{E_2A_2}{L_2} & -\frac{E_2A_2}{L_2} \\ -\frac{E_2A_2}{L_2} & \frac{E_2A_2}{L_2} \end{bmatrix} \; x \; \begin{pmatrix} q_2 \\ q_3 \end{pmatrix} = \begin{pmatrix} 0 \\ F \end{pmatrix}$$

By suppressing 1st line and 1st column ($q_1 = 0$)

For an unitary load (F = 1), obtention of :

$$q_2 = \frac{L_1}{E_1A_1} \qquad q_3 = \frac{L_1}{E_1A_1} + \frac{L_2}{E_2A_2}$$

1.6. Step 6 : Stress and strain restitution in elements

Compatibility : $$\begin{cases} \varepsilon_1 = \frac{\partial q}{\partial x} forelement1 = \frac{q_2^{(1)} - q_1^{(1)}}{L^{(1)}} = \frac{q_2 - q_1}{L} \\ \varepsilon_2 = \frac{\partial q}{\partial x} forelement2 = \frac{q^{(2)}{}_2 - q_1^{(2)}}{L^{(2)}} = \frac{q_3 - q_2}{L} \end{cases}$$

Behaviour : $$\begin{cases} \sigma^{(1)} = E^{(1)}\varepsilon^{(1)} \\ \sigma^{(2)} = E^{(2)}\varepsilon^{(2)} \end{cases}$$

1.7. Numerical Application

$A^{(1)} = 2\ cm^2$ $\qquad$ $A^{(2)} = 1\ cm^2$ $\qquad$ $L^{(1)} = L^{(2)} = 10\ cm$

$E^{(1)} = E^{(2)} = 2.10^6\ kg/cm^2$ $\qquad$ $P^3 = 1\ kg$

$$[K] = 2.10^5 \begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

$q_2 = 0.25\ 10^{-5}$ cm

$q_3 = 0.75\ 10^{-5}$ cm

$\varepsilon^{(1)} = 0.25\ 10^{-6}$

$\varepsilon^{(2)} = 0.50\ 10^{-6}$

$\sigma^{(1)} = 0.5$ kg/cm^2

$\sigma^{(2)} = 1$ kg/cm^2

2. Units System

2.1. Units

- 1 Pa = 1 N/m^2 = 10^{-6} N/mm^2 = 10^{-5} bar = 1.02 10^{-5}Kgf/cm^2
- 1 Mpa = 10^6 N/m^2 = 1 N/mm^2 = 10 bars = 10.2 Kgf/cm^2
- 1 bar = 10^5 Pa = 0.1 N/mm^2 = 0.1 Mpa = 1.02 Kgf/cm^2

Different units :

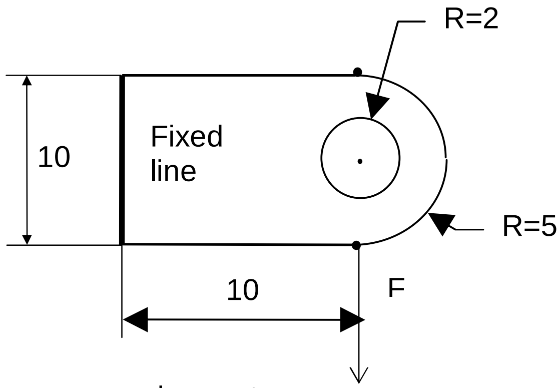
• If Newtons, mm :	M = 7.8E-6 Kg/mm^3,	E = 210 000 Mpa
• if Newtons, mm :	M = 7.8E-6 Kg/mm^3,	E = 210 000 N/mm^2
• If Newtons, meters :	M = 7800 Kg/m^3,	E = 2.1E11 N/m^2
• if Newtons, meters:	M = 7800 Kg/m^3,	E = 2.1E11 Pa
• If daN, mm	M = 7.8E-6 Kg/mm^3,	E= 21 000 daN/mm^2

2.2. Modifying a mesh scale

- From meters to millimeters :

```
.NOE CHARGE
GRAP INIT SX 1000 SY 1000 SZ 1000
EXECUTE
```

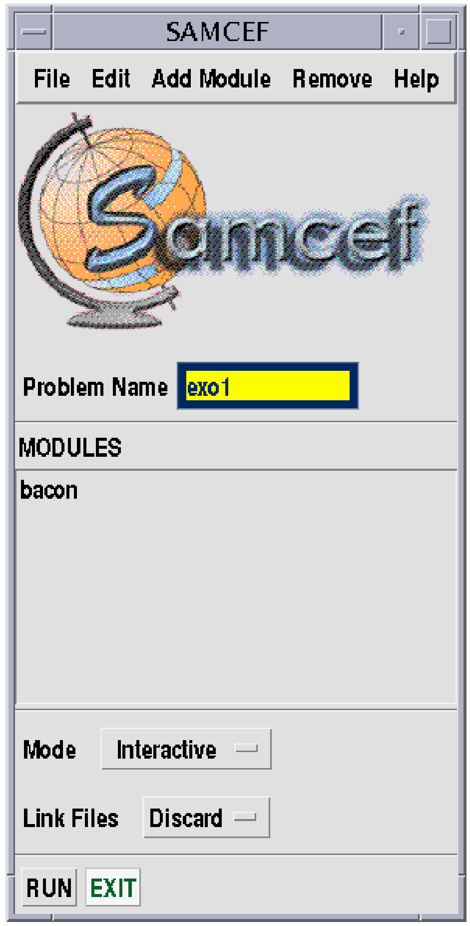
3. Guided example In Menu Mode



Conventions : **<cr>** carriage return
K> keyboard
M> Click with left button of mouse

Note : Left button	"I get"
Mid button	"I validate"
Right button	"I take off the entity from selection"

Pre-processing
Dialogue box : **samcef**



Bacon > **.MENU ON**

Pre processing / Geometry2D / Point 2D / Create / Coord. Rect
K> `0 0; 10 0; 10 10 ; 0 10; 10 5` *create points 1 2 3 4 5*

GEO2D/LINES/Create/Points
M> `2 1 4 3`

GEO2D/Arcs and circles 2D/Create/ Ctr-Radius
M(centre)> 5 K(Radius)> 2

Ctr-radius-Angle
M(centre)> 5
K(radius)> 5
K(angles)> `-90 90`

Structure/Premesh/Line/Contour/Create/Auto

Structure/Premesh/Line/Domain/Create/Auto Creates domain D1

Mesh generation
Pre Processing / Structure / Mesh / Modify / Lines

M> **4**
/ Elements
K> **16**

Exit / Mesh / Free
M> **D1**

Pre Processing/Structure/Post Mesh/Smoothening/Recombine/Medium

Exit / Relocation / Recombine / High / Exit / Relocation

Choice of hypothesis
Preprocessing/Structure/Element Type/Default/Plane stress

Create material
Exit/PreProcessing/ Structure Properties /Material/
Define/New/Isotropic/Elastic

K(name)>	**Steel**
K(Young)>	**2.1E11**
K(Poisson)>	**0.3**
K(Density)>	**<cr>**
K(Thermal expansion coef.)>	**<cr>**

Apply/On all element
M> **Mat. 1 Steel** *Click on name* ***BEFORE*** *OK*
OK
Display/Materials location

Physical Properties /Thickness/Create
K(VALUE)> **0.5**
/Attribute K(Number)> **1**

Boundary conditions & Loading

Loads and BCs /Fixations/All/Line/Create
M> **L2**
Accept /EXIT

Loads and BCs/Nodal loadings/Structural force /Component : **2**
C >**-20**
Point/ Create M> **2**
/Accept /Exit

Computation

/Analysis Click on Asef
select interactive execution
Execute

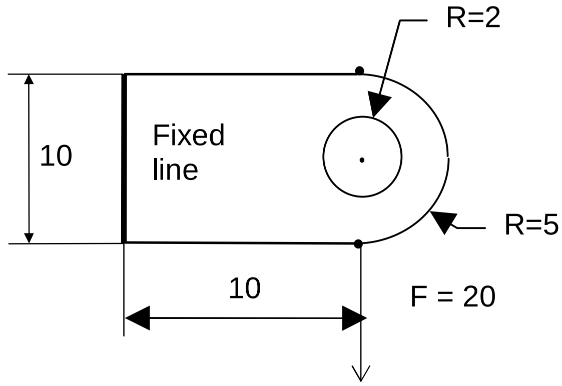
Post processing

/Post processing/Access to results/ EXO1/Asef/ACCESS
/DRAWINGS+LISTS/SELECTION/Displacements / Display/ Displacements/
Deformed shape/Module

4. Exercises on command mode

4.1. Guided exercice in command mode

4.1.1. Model



4.1.2. Bank file Creation

- With Editor text, open a text file named **`exo1b.dat`**.
- At first line, write.

```
.DEL.*
```

- Write lines

4.1.3. Pre-processing

- run samcef
- do not activate the menu
- Enter the following commands :

```
INPUT "exo1b.dat"
```

4.1.4. Chaining modules at a line command (UNIX Station Only)

`samcef   ba,as   exo1b   n   2`
Bank file name: **`exo1b`**

4.1.5. Post-Processing

```
.DOC DB
.DES  CODE  163  REFE  4  1 ;MODU DEPLA ; DEFO 1 ;VI
.STOP
```

4.1.6. Created Bank

```
.DEL.*
.POIN
     I  1  X   0  Y  0
     I  2  X  10  Y  0
     I  3  X  10  Y 10
     I  4  X   0  Y 10
     I  5  X  10  Y  5
     I  6  X  12  Y  5
.DROI
     I  1 POINTS  2  1  4  3
.ARC
     I  4  CENTRE  5  RAYO  2
     I  5  CENTRE  5  RAYO  5  ANGLE  -90  90
.CONTOU AUTO
.DOMAIN AUTO
.GENE
     MODIFIE  LIGNE  4  ELEMENTS  16
     MAILLE  1
.LISSAGE
     RELOC
     RECOMB SEVERITE 3
     RELOC
     RECOMB SEVERITE 2
     RELOC
.SEL GROUPE "ALL" MAILLE TOUT
.HYP MEMBRANE BIDIM
.MAT NOM "STEEL" YT  2.1E11  NT  0.3
.AEL GROUP "ALL" MAT "STEEL"
.PHP GROUP "ALL" THICK  VAL  0.5
.CLM FIX  LIGNE 2
     CHA  POINT  2  NC  1  V  0  -20  0
EXIT
```

4.2. Repetition exercise

It is possible to perform repetitions of command lines with incrementation of the specified parameters values. This can be used in the .NOE, .MAI commands (meshing commands)

R m for "m" repetition of the last line inside a command
Q n for "n" repetition of all lines inside a command.
A p for the repetition of the "p" last lines. (used together with **Q** parameter)

Example :

```
.NOE
 I 1 X 0. Y 0. Z 0.
 I 1 X 1. R 2
 I 3 Y 0.5 Q 2
.MAI
 I 1 N 1 2 5 4
 I 1 N 1 1 1 1 R 1
```

```
 I 2 N 3 3 3 3 Q 1
```

Is equal to :

```
.NOE
 I 1 X 0. Y 0. Z 0.
 I 2 X 1. Y 0. Z 0.
 I 3 X 2. Y 0. Z 0.
 I 4 X 0. Y .5 Z 0.
 I 5 X 1. Y .5 Z 0.
 I 6 X 2. Y .5 Z 0.
 I 7 X 0. Y 1. Z 0.
 I 8 X 1. Y 1. Z 0.
 I 9 X 2. Y 1. Z 0.
.MAI
 I 1 N 1 2 3 4
 I 2 N 2 3 6 5
 I 3 N 4 5 8 7
 I 4 N 5 6 9 8
```

Create a square of 1m with 10X15 elements.

5. Reading an IGES file

5.1. Samcef running

5.2. On menu

File / Import / IGES ;
IGES File Name Filter (default = *): **<cr>**

< name_of_file >

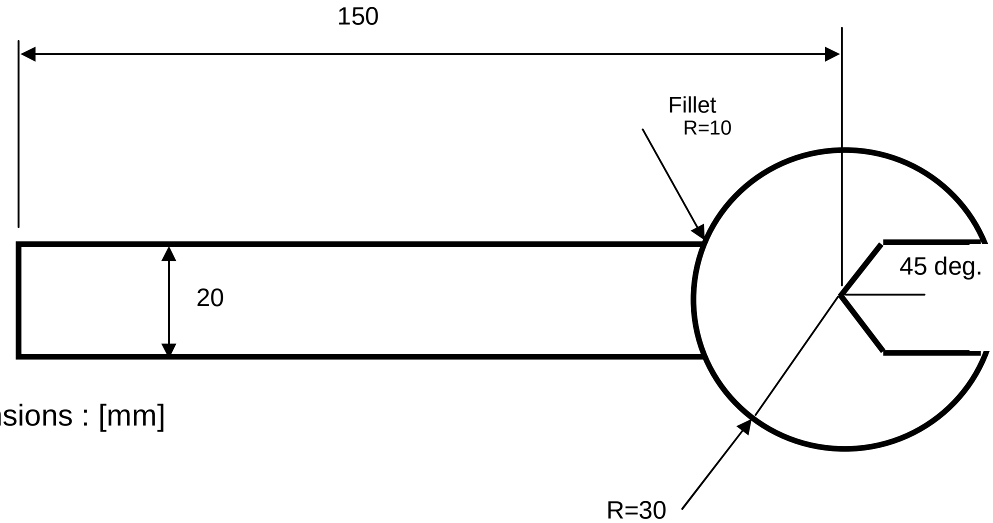
5.3. Keyboard

.INT IGES “name_of_file”

6. bacon.ini

```
ABRE  '/MEN' '.MENU ON'
TITRE DATE 0
GRAP TC  1
.MENU ON
```

7. Mechanical tool 2D



Dimensions : [mm]

Thickness : 5 mm for head, 3 mm for arm , material : Steel.

Create geometry, mesh, define the hypothesis, material, loads and BCs (Fixation of the inner part of the tool, load applied on one corner F=-150, comp 2)
Compute the displacements and stresses.

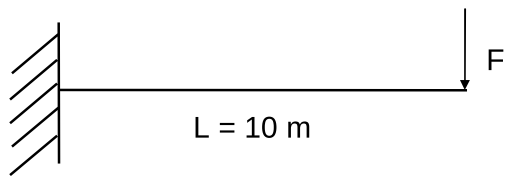
8. Mechanical tool 3D

Retrieve 2D mechanical tool. Delete loads, BCs and thickness.
Create the volume by extrusion. Load, fixation, materials.

9. Beam

Use the command .BPR and .BEAM to transform rod to beam finite element. Study the influence of the orientation of the section.

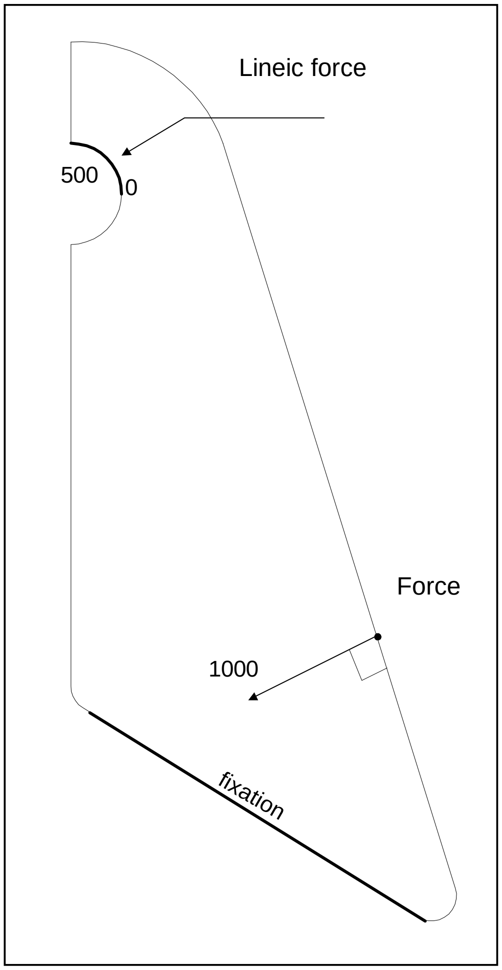
- E = 1.2E11 N/m^2, Nu=0.3
- L = 10 m, width = 1 m, thickness = 0.1 m
- Load : F = 10000 N

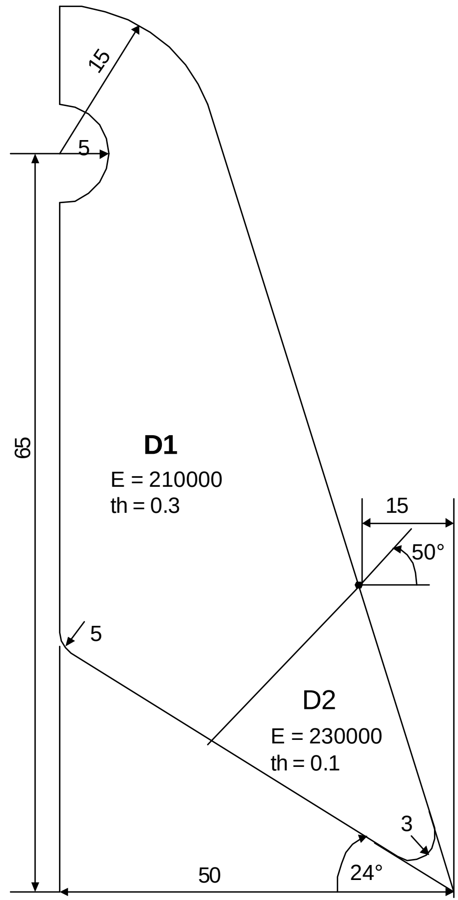


Questions :

- Maximal displacement ?
- Normal effort?

10. Two materials, two thickness plates

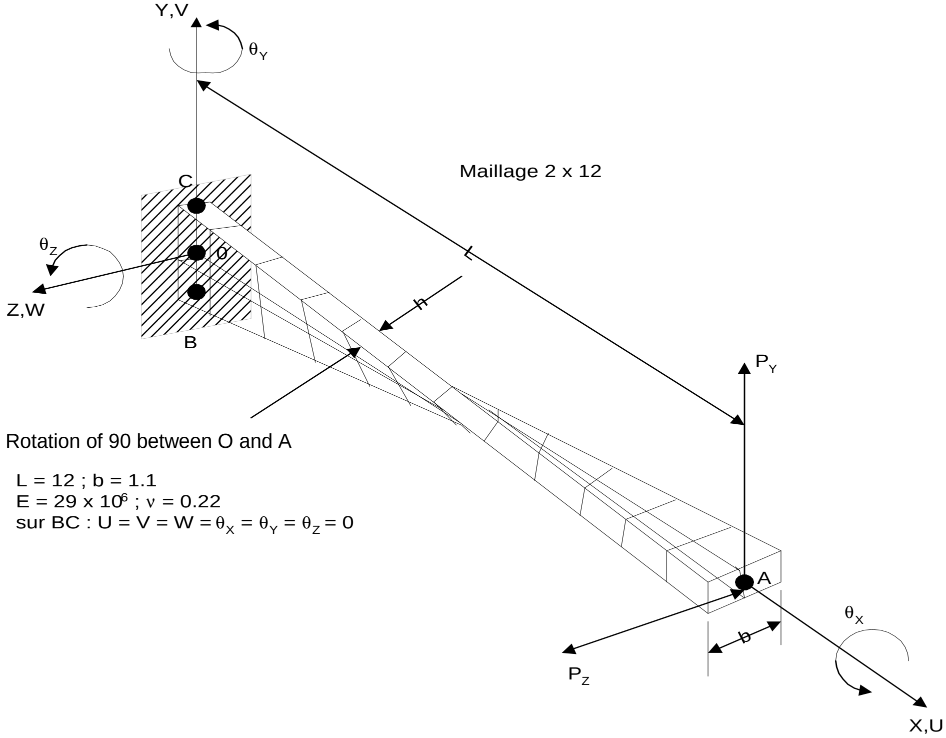




Creation of the geometry taking into account the two material definition and the boundary conditions (loads and fixation required local axis)
Computation of the displacements and stresses. Study the results.

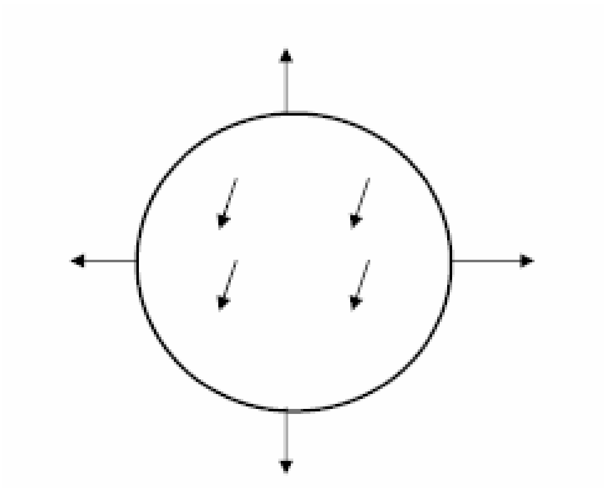
11. Twisted beam

- Length L=12
- Width b=1.1
- Thickness : h = 0.32
- A (1.2,0,0); B (0,-0.55,0); C (0, 0.55,0)



- BC Side fixed
- Point A Load: Fz=1 N, Fy=1 N
- displacements and stresses.
- Use shell elements

12. Circular plate



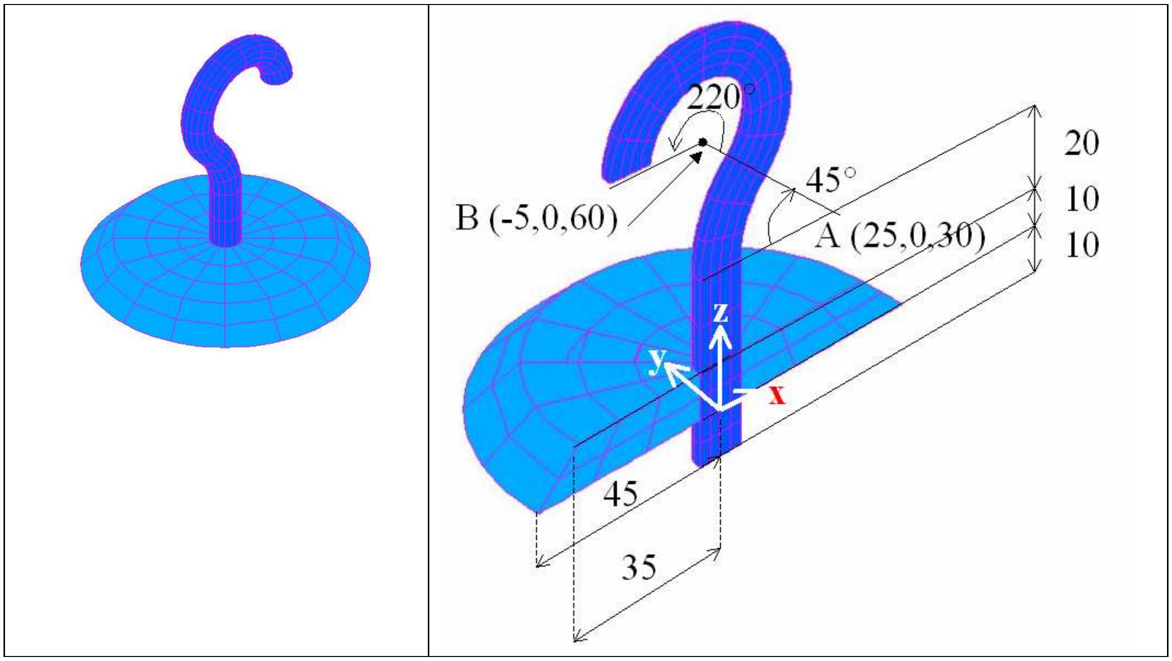
Radius = 100 mm, thickness = 1 mm, E = 70000 MPa, nu= 0.3
Radial loading = 25000 N
Pressure = 0.25e05 Pa
Plate is simply supported on the boundary

The objective is to show the consequence of the pre-stresses induced by the radial loading.
The computation is performed in two steps :

- Step 1 : Apply the radial loading
- Step 2 : Apply the normal pressure but taking into account the radial loading.

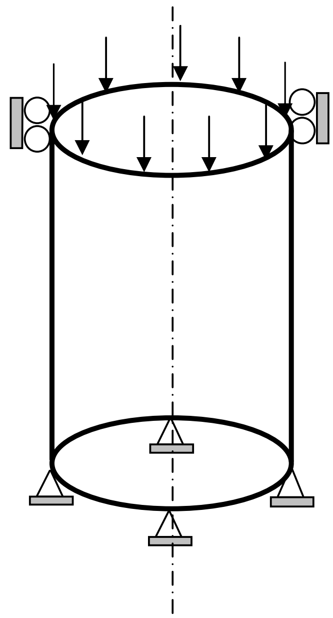
Discuss the choice of the hypothesis.

13. Meshing of a hook by extrusion



Material is steel.
All the basis of the support is clamped.
A pressure of 10 is applied on the basis of the hook.

14. Axi-symmetric buckling of a cylinder.



Analysis Data :

Diameter = 100 mm
Length = 50 mm
Thickness = 0.5 mm
Young's Modulus = 210000 MPa., ν = 0.3

Two different loadcases
1. Axial compression. Apply a total load of **σ** = 1 N/mm2 (F = 2.Π.a.t.**σ**) and perform the analysis
2. External pressure of -1 N/mm2

Theoretical solution for a unit compression stress state: $\lambda = \frac{E.t}{r\sqrt{3(1-\upsilon^2)}}$

Theoretical solution for a external pressure :

- If long cylinder with $L \geq 4.90r\sqrt{\frac{r}{t}}$ we have $\lambda = \frac{1}{12}\frac{E}{(1-\upsilon^2)}\frac{t^3}{r^3}(\upsilon^2 - 1)$
- If short cylinder we have $\lambda = 0.807\frac{Et^2}{r*L}*\sqrt{\sqrt{\left(\frac{1}{(1-\upsilon^2)^3}\right)*\left(\frac{t}{r}\right)^2}}$