

## FORMA02 - Practical works of the formation “Initiation”: pipe bent under thermomechanical and dynamic request

---

### Summary:

This test corresponds to practical works of the basic training to the use of *Code\_Aster*. It is about a bent piping, made up of a linear elastic material, subjected to various loadings: internal pressure, thermal transient, transitory force.

Modelings used are the following ones:

- modeling a: modeling 3D, thermomechanical calculation,
- modeling b: modeling 3D, dynamic, modal calculation then transitory,
- modeling C: elements `hulls DKT`, mechanical calculation under internal loading of pressure,
- modeling D: elements beams (`POU_D_T`), dynamic calculation.

The chapter 1 “Problem of reference” presents the problem to be treated and the data common to all modelings; the statements of Practical works of the formation are included in this document:

- TP2: “thermoelastic 3D” to see modeling A,
- TP2: “dynamic 3D” to see modeling B.

## 1 Problem of reference

### 1.1 Geometry

The study relates to a piping including two right pipes and an elbow [Figure 1.1-a].  
The geometrical data of the problem are the following ones:

- the length  $L_G$  of the two right pipes is of  $3\text{ m}$ ,
- the ray  $R_c$  elbow is of  $0.6\text{ m}$ ,
- the angle  $\theta$  elbow is of  $90^\circ$ ,
- the thickness  $e$  right pipes and elbow is of  $0.02\text{ m}$ ,
- and the external ray  $R_e$  right pipes and elbow is of  $0.2\text{ m}$ .

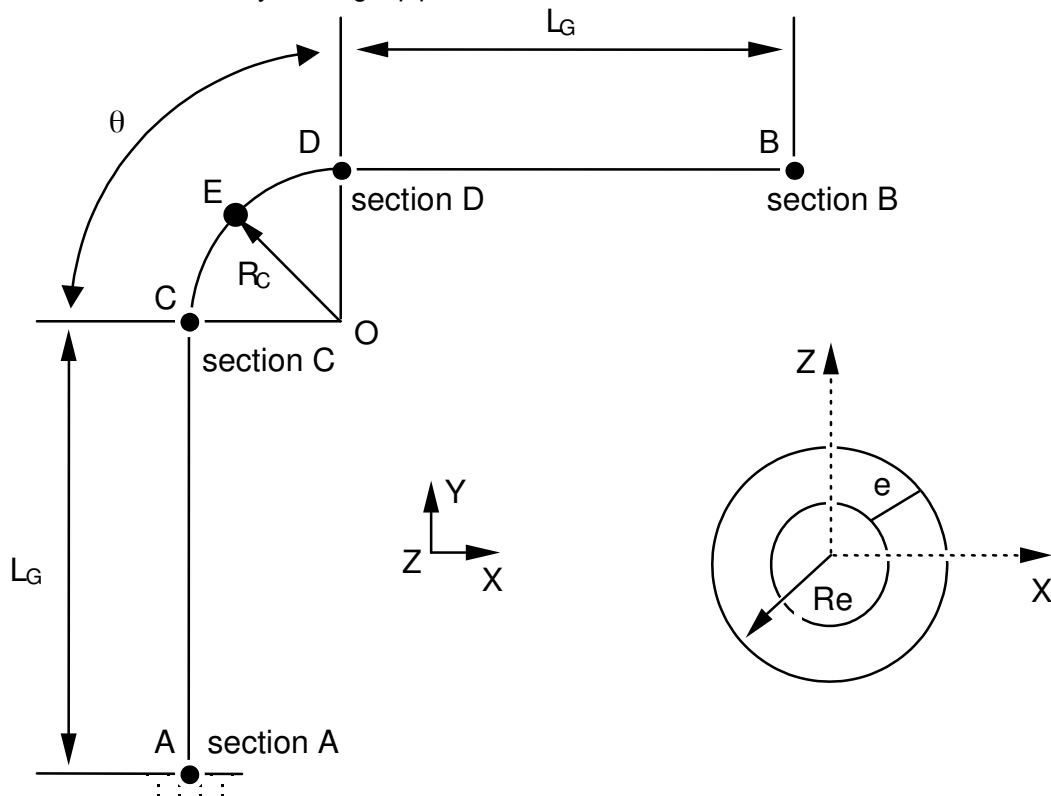


Figure 1.1-a

Note:

The geometry of the problem has a symmetry compared to the plan  $(A, X, Y)$ .

### 1.2 Material properties

For all modelings With, B, C and D :

Isotropic linear elastic material. the properties of material are those of steel A42 with  $20^\circ\text{C}$  :

- the Young modulus  $E = 204\,000 \times 10^6\text{ N/m}^2$ ,
- the Poisson's ratio  $\nu = 0.3$ .

For thermoelastic calculation (modeling A):

- the thermal dilation coefficient  $\alpha = 1.096 \times 10^{-5}/^\circ\text{C}$ ,
- thermal conduction  $\lambda = 54.6\text{ W/m}^\circ\text{C}$ ,
- voluminal heat  $\rho C_p = 3.71 \times 10^6\text{ J/m}^3^\circ\text{C}$ ,

For dynamic calculation (modeling S B, D):

- density  $\rho = 7800 \text{ kg/m}^3$ ,
- the damping of the clean modes will be taken with 5% for the modes.

## 1.3 Boundary conditions and loadings

The boundary conditions for all modelings are the following ones:

- embedding on the level of the section  $A$ ,
- at the time of static loadings, embedding on the level of the section  $A$  and of the section  $B$ .

With regard to static calculations, the loadings applied are of three types:

- internal pressure (on the inner face)  $P = 15 \cdot 10^6 \text{ N/m}^2$  (modeling hulls or 3D),
- thermomechanical loading with a transient of temperature imposed on the inner face of piping (assembled  $20^\circ\text{C}$  with  $70^\circ\text{C}$  in 10s) and a condition of null exchange on the external face of piping (heat insulator) (modeling A only).

With regard to dynamic calculation, the loading applied is a transitory force (in Newton):

$F_Y(t) = 10\,000\,000 \cdot \sin(2\pi \text{Freq}1 \cdot t)$  directed according to the axis  $Y$  and applied to the section  $B$  with  $\text{Freq}1 = 20 \text{ Hz}$ .

## 2 Reference solution

### 2.1 Method of calculating used for the reference solution

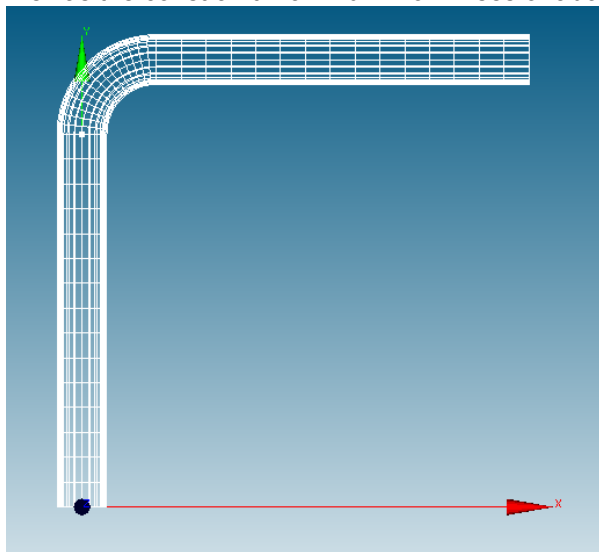
The reference solution is obtained numerically, it thus acts only of the tests of nonregression.

### 2.2 Results of various modelings:

#### 2.2.1 Static calculation

The values tested are:

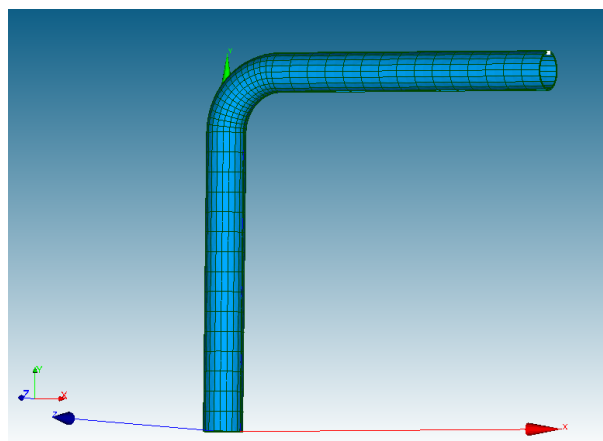
- for loading of pressure (modeling S With and C), its displacement at higher exit of elbow on average surface (as in the figure in lower part in white,  $\Delta X$  with  $-4.52e-4$  m)
- for the loading thermomechanical (modeling A), the temperature in internal wall at the bottom of tube ( $70^{\circ}\text{C}$ ) as well as the constraint maximum Von Mises of about 200 MPa.



#### 2.2.2 Dynamic calculation

The values tested are:

- For modeling 3D (modeling B), the first Eigen frequency of about 9 Hz and Lhas component of displacement  $DY$  point P at the end of piping on the average surface (as in the figure in lower part in white) of the order of  $-0.128\text{m}$  at the moment 1s.
- for modeling 1D (modeling D), the constraints equivalent in foot of tube.



## 3 Modeling A

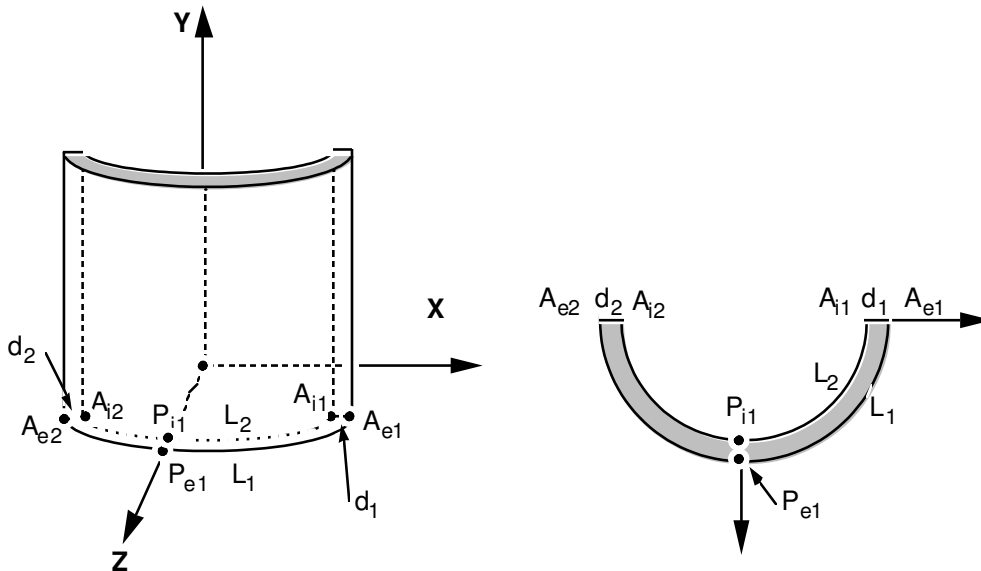
### 3.1 Characteristics of modeling

Glossary: in Salomé one indicates by:

- vertex points useful for the construction of the geometry;
- edge lines (right);
- arc arcs of a circle;
- wire lines made up of edge and arc;
- face the surfaces delimited by one wire;
- Shell a set of face intended to be with a grid or delimiting a volume;
- compound a set of geometrical entities.

The right pipes and the elbow are modelled by quadratic isoparametric solid elements. Piping presents a symmetry plane  $Z=0$ . Only one half volume is netted.

Boundary conditions and loading: to see chapter 1.3 .



### 3.2 Geometry

To launch the module Geometry platform Salomé-Meca.

Two processes to create the geometry (to recommend the deuxième) :

- Extrusion of a basic surface according to a way:
  1. Creation of the points  $A_{e1}$ ,  $A_{i1}$ ,  $A_{e2}$ ,  $A_{i2}$ ,  $P_{e1}$ ,  $P_{i1}$ , then Dbe arcs of a circle  $L_1$  and  $L_2$ , Dbe segments  $d_1$  and  $d_2$  (Menu New Entity → Lowic → Not/arc/Line).
  2. Creation contour (wire) delimited by  $L_1$ ,  $d_1$ ,  $L_2$ ,  $d_2$  and then of surface (object of the type " Face ") starting from this contour (Menu New Entity → Build → wire/Face ).
  3. Creation of the telegraphic way: points  $A$ ,  $C$ ,  $D$ ,  $B$ ,  $O$ , then two lines  $AC$  and  $DB$ , and the arc of a circle  $CD$ , finally the total way  $ACDB$  in wire.
  4. This surface is then extruded by the menu New Entity → Generation → Extrusion Withlength Path.
- Use of basic voluminal objects:

1. To build a right pipe of axis  $Z$  (option by default), using two elementary cylinders height 3 and of rays 0.2 and 0.18 (Menu New Entity → Primitives → Cylinder), and of a Boolean operation (Menu OperationS → Boolean → CC). To re-elect the object created in *TUY1*.
2. To apply a rotation to *TUY1* around  $X$  so that it is of axis  $Y$  (Finely Operations → Transformation → Rotation): Angle =  $-90^\circ$  and to disable Create has Copy.
3. To select the higher face by creating a group: Menu New Entity → Group → Create Group. This face allows to generate the elbow by the menu New Entity → Generation → Revolution (Angle =  $-90^\circ$ ). It is necessary for that to create the axis of revolution (a vector of direction  $Z$  passing by the point  $o(0.6, 3.0, 0.0)$ ): Creation of the points  $(0.6, 3.0, 0.0)$  and  $(0.6, 3.0, 1.0)$  and then creation of the vector (Menu New Entity → Lowic → Vector).
4. Then to create the second right pipe in the same way *TUY2* : to select the higher face of the elbow by creating a group, and to generate the right pipe by the menu New Entity → Generation → Extrusion according to  $X$ .
5. It remains to select half of the piping located in the plan  $Z > 0$ . For that one can initially créer a plan (Menu New Entity → BASIC → Plane, for example, the 5th choice of Plane (creation starting from the system local of the coordinates), with the option OXY and cuts it 8). Then Couper with the assistance of partition (Menu Operations → Partition) by selecting the 3 parts of piping like Objects and the plan like Tool Objects.
6. Créer one group of volume by selecting Lbe partS locatedS in  $Z > 0$  (Finely New Entity → Group → Creat Group). , and finally with this group to create one "compound"(Finely New Entity → Build → Compound).

### **Preparation of the geometry for a grid in hexahedrons :**

The pipe is intended to be with a grid with hexahedrons, i.e. in a regulated way (the number of elements is identical on two side opposite). Before carrying out the grid, it is necessary to check that the geometry is topologically equivalent to a hexahedron (6 faces). For that, it is necessary to use the menu Inspection → Check Compound of Blocks.

According to the method of construction (the 1<sup>er</sup> process), it can be necessary to carry out a partition byux plans of the sections entered and exit of elbow in order to obtain a geometry topologically equivalent to a hexahedron (6 faces). To create the two plans, finely New Entity → Basic → Plane: not + Vector thus Channel + AC / D + COMIC). For the partition, menu Operations → Partition.

If one wants to make a grid with tetrahedrons, this condition is not necessary. On the other hand, if one wants to net in hexahedron, it is essential for the algorithms of the maillieurs.

### **Creation of the groups :**

One will create then the groups of faces where one wishes to put limiting conditions: Base, Symmetry, Efond, Surfint and Surfext (Menu New Entity → Group → Create Group).

One will create also the group of edge with an edge thickness ( $d1$ ) and another group of edge with an edge of circumference ( $L1$ ) to allow a different refinement on these edges.

One will be able to test the other mode of use of module Geometry / Mesh, consisting to produce, possibly to modify, and read again a file python, which contains all the instructions necessary to the construction of the geometry. One creates this file by the menu File → Dump Study. This file python can be read again by the menu File → Load Script. This makes it possible in particular to modify dimensions of the geometry. As example, one will be able to examine the contents of the file forma02a.datg, and to read again it in Salome-Meca.

## **3.3 Maillage**

To launch the module Mesh platform Salomé-Meca.

*Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.*

Copyright 2021 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

The grid is defined by the menu `Mesh` → `Create Mesh`. Sélectionner geometry to be netted, then the algorithm and assumption of discretization per dimension:

- 3D Hexa Hedron (I, J, K).
- 2D Quadrangle: Mapping.
- 1D Wire Discretisation with the assumption basic Number of Segments (15 segments by edge).

Then thatculer grid (Finely Mesh → Compute).

To allow a different refinement according to edges, under-grids will be created by the menu `Mesh` → `Create Sub-mesh`:

- UN sub-mesh defining number segments in the thickness  $d1$ . Prear example, the assumption basic Number of Segments with 2 segments on  $d1$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".
- One sub-mesh defining the number segments on the circumference  $L1$ . Prear example, 10 segments on  $L1$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".

Then thatculer grid (Finely Mesh → Compute).

To create Lbe groups DE e-mailbe geometrical correspondents with the group (Menu `Mesh` → `Create Groups FRomanian Geometry`).

Exporter it grid with the format MED.

To obtain a better precision, one will make pass the grid of linear to quadratic, thanks to the tool "Modification - > Convert to/from quadratic". Most curious can compare the differences in results between the two types of elements.

#### Remarks :

With another smoothness of grid, it is possible to get appreciably different results. In particular, to obtain a correct solution, it is desirable to use quadratic elements.

### 3.4 Creation and launching of the calculation case (via AsterStudy)

To launch the module `AsterStudy` platform Salomé-Meca .

Then in left column, to click on the mitre `View` box, to click right `CurrentCase` to add a new internship where ON definesS orders of the calculation case.

Foot-note: to add orders by the menu `Commands` → `All show`.

The study requires the first transitory thermal calculation followed by a mechanical calculation.

Principal stages for the creation and the launching of the calculation case are the following ones:

- To see the grid with format MED: Order `LIRE_MAILLAGE`.
- To define material: Order `DEFI_MATERIAU` ( $E, \nu, \alpha$  in ELAS,  $\lambda, \rho Cp$  in THER).

#### Thermal calculation:

- To define the finite elements used: Order `AFFE_MODELE` for to affect the phenomenon THERMICS and modeling in 3D with ALL.
- To affect material with all : Order `AFFE_MATERIAU`.
- To affect conditions with limits thermics and the loading :
  - ON has a transient of temperature imposed on surface intern `Surfint` piping. This rise of  $20^\circ C$  with  $70^\circ C$  in 10s is defined with the order `DEFI_FONCTION`: to define the data in `VALE` and `NOM_PARA = INST`.
  - This transient of temperature is imposed with `Commande AFFE_CHAR_THER_F`: option `TEMP_IMPO`, with `Surfint` like `GROUP_MA`, and the function defined as imposed temperature `TEMP`)
  - It is considered that piping is insulated and one does not specify a condition on external surface (what returns to a null flow).

- To define the moments of calculation, 0 S, 5 S, 10 S: order `DEFI_LIST_REEL` .
- To solve thermal problem linear: Order `THER_LINEAIRE`, by defining an initial state of uniform temperature equal to  $20^{\circ}C$  (`ETAT_INIT / VALE`), and while affecting `MODEL`, `CHAM_MATER` (materials), `EXCIT` (loading), `INCREMENT` (list of the moments).

### Thermomechanical calculation:

- To define the finite elements used: Order `AFPE_MODELE` for to affect the phenomenon `MECHANICS` and modeling in 3D.
- To affect material: Order `AFPE_MATERIAU`. The result of thermal calculation will be applied with all as a variable ordering the behavior of mechanical material by `EVOL` in the option `AFPE_VARC` with the temperature of reference `NOM_VARC = TEMP` with  $20^{\circ}$ .
- To affect conditions with limits mechanics and the loading : Order `AFPE_CHAR_MECA` :
  - L piping has is embedded at its two ends and it present a symmetry plane: `DDL_IMPO` .
  - One a pressure has surfaces intern of it of  $15 MPa$  : `PRES_REP` (to check the direction of normal on the surface interns in AsterStudy).
- To solve it problem linear static mechanics : Order `MECA_STATIQUE` with the calculation of the field of temperature defined above and the same list of the moments like thermal calculation.
- To calculate the field : Order `CALC_CHAMP` with `reuse`.
  - for Calcul of the stress field by elements extrapolated with nodes : `CONSTRAINT / SIEF_ELNO` .
  - for Calcul of the equivalent stress field of Von Mises : `CRITERIA / SIEQ_ELNO` .
- Imprimer maximum values of the tensor of constraints `SIEQ_ELNO` with the format `Result (RESULT)` and Imprimer LE fields of temperature, displacements, and stress fields with the format `MED` : Order `IMPR_RESU`.
- For launchR the calculation case, in left column, to click on the mitre `History View`.

## 3.5 Post-treatment of the results

### 3.5.1 With Salomé and Results

To visualize the fields and the grid deformed with some click in the mitre `Results of AsterStudy`. To check some values while comparing with references (see chapter 2.2.1).

One can also launch the module `ParaViS` platform `Salomé-Meca` .

- To open the file of results ( `Menu File` → `Open Para View File` ) container the “mechanical” part. To choose the field `SIEF_ELGA` and then haspliquer the filter `Merge Blocks` by the menu `Filters` .
- To open the file of results container the “thermal” part. To choose the field `TEMP` and then haspliquer the filter `Merge Blocks`.
- To select of the two objects `Merge Blocks` then to apply the filter `Attributes learns` .
- To visualize the deformed grid ( `Menu Filters` → `Common` → `Warp By Vector` ) with `Lhas` temperature and `Lbe` forced at the points of `Gauss` ( `Menu Filters` → `ELGA field To Surface` ).

### 3.5.2 With Aster\_Study by adding one internship

Principal stages of **second** execution with `Aster_Study` Swill eront:

- Extraanger `Lbe` values of the field `SIEF_ELNO` for an azimuth on the level of the entry of elbow (not `C`) in the case of load corresponding to  $F_y$  in a table : Order `MACR_LIGN_COUPE`. With the option of the type `SEGMENT`, L’azimuth is defined by the way `D’` ends ( `0.18,3.0,0.` ) and ( `0.2,3.0,0.` ) , for example with 10 points.
- To print the table produced with the format `XMGRACE` for the component `NOM_PARA=SIYY` with `Commande IMPR_TABLE` : Utiliser `Filter` → for the moment `NOM_PARA=INST` with



10s, and to determine the data to extract by `NOM_PARA`, for example, `COOR_X` (for X-coordinate X) and `SIYY` (for the component of constraint).

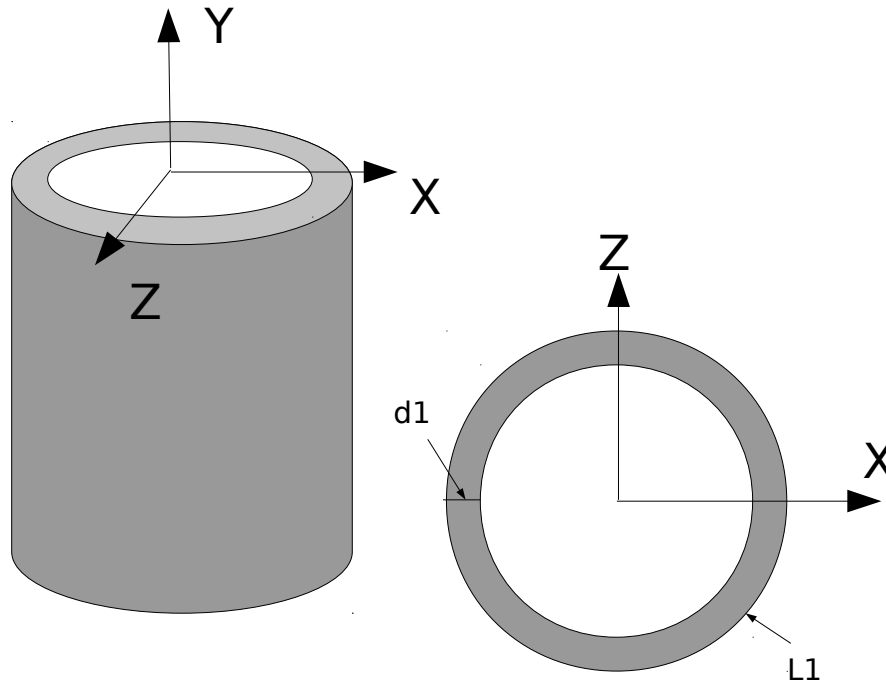
## 4 Modeling B

Glossary: in Salomé one indicates by:

- vertex points useful for the construction of the geometry;
- edge lines (right);
- arc arcs of a circle;
- wire lines made up of edge and arc;
- face the surfaces delimited by one wire;
- Shell a set of face intended to be with a grid or delimiting a volume;
- compound a set of geometrical entities.

The right pipes and the elbow are modelled by quadratic isoparametric solid elements.

Piping presents a symmetry plane  $Z=0$ . But contrary, with the static case, one will not benefit from it. It will be seen indeed, that the modes can leave the symmetry plane. All volume will thus be netted. Boundary conditions and loading : to see chapter 1.3.



### 4.1 Geometry :

To launch the module Geometry platform Salomé-Meca.

Two processes to create the geometry (to recommend the deuxième) :

- Extrusion of a basic surface according to a way:
  1. Creation of the circles interior and external of the base on the plan  $Oxz$  (Menu New Entity  $\rightarrow$  Lowic  $\rightarrow$  Circle), then the surface delimited by the two circles ( Menu New Entity  $\rightarrow$  Build  $\rightarrow$  Face).
  2. Creation of the telegraphic way: points  $A, C, D, B, O$ , then two lines  $AC$  and  $DB$ , and the arc of a circle  $CD$  (Menu New Entity  $\rightarrow$  Lowic  $\rightarrow$  Not/Line/Arc), then it way entier (Menu New Entity  $\rightarrow$  Build  $\rightarrow$  Wanger).
  3. This surface is then extruded by the menu New Entity  $\rightarrow$  Generation  $\rightarrow$  Extrusion Withlength Path.
- Use of basic voluminal objects:

1. To build a right pipe of axis  $Z$ , using two elementary cylinders height 3 and of rays 0.2 and 0.18 (Menu New Entity  $\rightarrow$  Primitives  $\rightarrow$  Cylinder), and of a Boolean operation (Menu OperationS  $\rightarrow$  Boolean  $\rightarrow$  CC). To re-elect the object created in  $TUY1$ .
2. To apply a rotation to  $TUY1$  around  $X$  so that it is of axis  $Y$  (Finely Operations  $\rightarrow$  Transformation  $\rightarrow$  Rotation): Angle =  $-90^\circ$  and to disable Create has Copy.
3. To select the higher face by creating a group: Menu New Entity  $\rightarrow$  Group  $\rightarrow$  Create Group. This face allows to generate the elbow by the menu New Entity  $\rightarrow$  Generation  $\rightarrow$  Revolution (Angle =  $-90^\circ$ ). It is necessary for that to create the axis of revolution (a vector of direction  $Z$  passing by the point  $o(0.6, 3.0, 0.0)$ ): Creation of the points  $(0.6, 3.0, 0.0)$  and  $(0.6, 3.0, 1.0)$  and then creation of the vector (Menu New Entity  $\rightarrow$  Lowic  $\rightarrow$  Vector).
4. Then to create the second right pipe in the same way  $TUY2$ : to select the higher face of the elbow by creating a group, and to generate the right pipe by the menu New Entity  $\rightarrow$  Generation  $\rightarrow$  Extrusion according to  $X$ .
5. Finally to assemble 3 parts of piping with the assistance of partition (Menu Operations  $\rightarrow$  Partition).

### **Preparation of the geometry for a grid in hexahedrons :**

For a grid in hexahedrons, it is necessary to use the algorithm "HexaHedron (I, J, K)" or to use the option "Automatic HexaHedralization".

The pipe is intended to be with a grid with hexahedrons, i.e. in a regulated way (the number of elements is identical on two side opposite). Before carrying out the grid, it is necessary to check that the geometry is topologically equivalent to a hexahedron (6 faces). For that, it is necessary to use the menu Inspection  $\rightarrow$  Check Compound of Blocks.

According to the method of construction (the 1<sup>er</sup> process), it can be necessary to carry out a partition by plans of the sections entered and exit of elbow in order to obtain a geometry topologically equivalent to a hexahedron (6 faces).

For the geometry generated following the first process, several should be created plans for partitionner piping: a plan  $Y = 3$ , a plan  $X = 0.6$  (Menu New Entity  $\rightarrow$  BASIC  $\rightarrow$  Plane : not + Vector thus Channel + AC / D + COMIC), and a surface curved to leave D U way ACDB created (Wire\_1) : menu New Entity  $\rightarrow$  Generation  $\rightarrow$  Extrusion S uivant axis Z, then Operations/Transformation/Translation so that curved surface cut entirely piping .

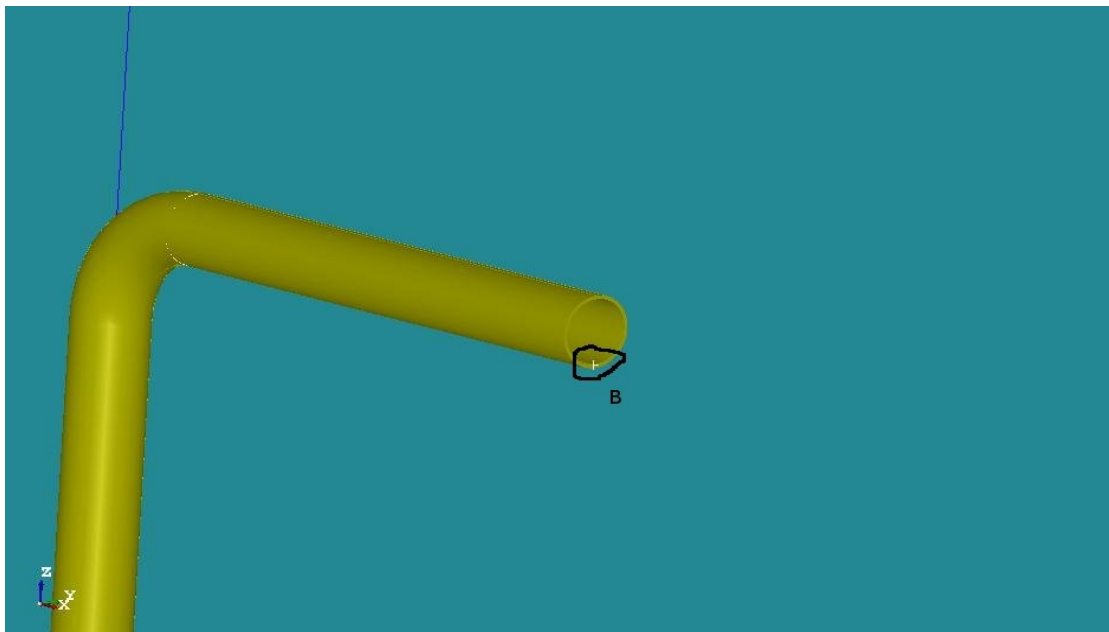
If one wants to make a grid with tetrahedrons, this condition is not necessary. On the other hand, if one wants to net in hexahedron, it is essential for the algorithms of the mailleurs.

### **Creation of the groups :**

One will create then the groups of faces where one wishes to put limiting conditions: Base (surface in bottom to embed), Efond (end of the pipe to apply the dynamic force): Menu New Entity  $\rightarrow$  Group  $\rightarrow$  Create Group.

One will create also the group of edge with an edge thickness ( $dI$ ) and another group of edge with an edge of circumference ( $LI$ ) to allow a different refinement on these edges .

For postprocessing it is practical to create a group on a point where one can follow the evolutions. One will thus create the group not  $B$  located on the section Efond, below, on surface intern pipe.



## 4.2 Grid

To launch the module Mesh platform Salomé-Meca.

The grid is defined by the menu Mesh → Create Mesh. Sélectionner geometry to be netted, then the algorithm and assumption of discretization per dimension:

- 3D Hexa Hedron (I, J, K).
- 2D Quadrangle: Mapping.
- 1D Wire Discretisation with the assumption basic Number of Segment (15 segments by edge).

Then thatculer grid (Finely Mesh → Compute).

To allow a different refinement according to edges, under-grids will be created by the menu Mesh → Create Sub-mesh:

- UN sub-mesh defining number segments in the thickness  $d1$ . Prear example, the assumption basic Number of Segments with 2 segments on  $d1$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".
- One sub-mesh defining the number segments on the circumference  $L1$ . P rear example, 2 0 segments on  $L1$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".

Then thatculer grid (Finely Mesh → Compute).

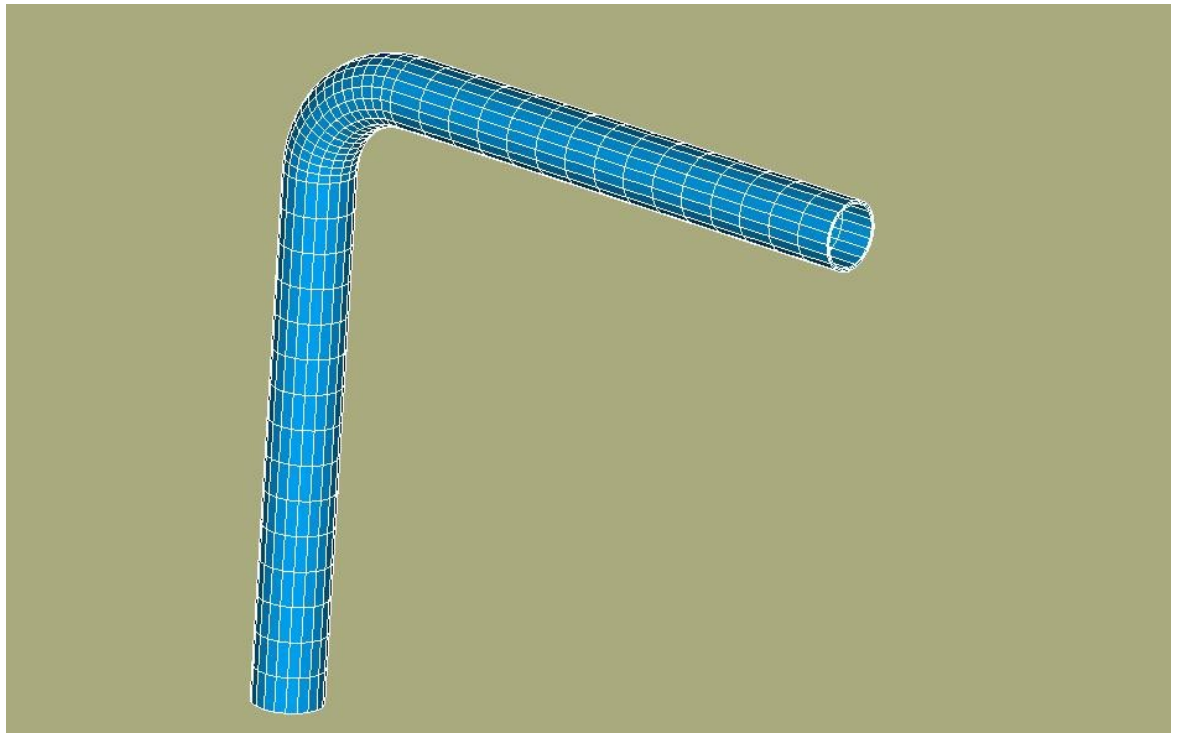
To create Lbe groups DE e-mailbe geometrical correspondents with the group (Menu Mesh → Create Groups FRomanian Geometry).

Exporter it grid with the format MED.

To obtain a better precision, one will make pass the grid of linear to quadratic, thanks to the tool "Modification - > Convert to/from quadratic". Most curious can compare the differences in results between the two types of elements.

### Remarks :

With another smoothness of grid, it is possible to get appreciably different results. In particular, to obtain a correct solution, it is desirable to use quadratic elements.



## 4.3 Creation and launching of the calculation case

Principal stages of calculation with AsterStudy will be:

- **Preparation of the data and analyzes modal:**

- Langer grid by `LIRE_MALLAGE`
- To affect the type of elements and modeling by `AFPE_MODELE`
- To define and affect material by `DEFI_MATERIAU` and `AFPE_MATERIAU`
- To define the boundary conditions by `AFPE_CHAR_MECA` : option `DDL_IMPO` to embed base.
- To assemble the matrices of mass and rigidity of the system by `ASSEMBLY`:

```
ASSEMBLY (MODELE=... ,
          CHAM_MATER=...,
          CHARGE=... ,
          # Classification of the unknown factors of a
system
          # of equations linear
          NUME_DDL=CO ('...'),
          MATR_ASSE= (_F (MATRICE=CO ('...'),
                        OPTION=' RIGI_MECA',),
                    _F (MATRICE=CO ('...'),
                        OPTION=' MASS_MECA',),),);
```

- To calculate them 10 first clean modes by `CALC_MODES`

```
CALC_MODES (MATR_RIGI=...,
            MATR_MASS=...,
            OPTION=' PLUS_PETITE',
            CALC_FREQ= _F (NMAX_FREQ= 10 ,),)
```

- To extract and visualize the clean modes by `IMPR_RESU`.

One will launch then calculation in the module AsterStudy. One recommends to visualize animations in the mitre Results of Asterstudy.

One will be able also to visualize the modes in Paravis by “deforming them”:

- To click right the file of the results in the mitre Box View → Dated Files, and to click Open In Paravis to charge the results by choosing the format mode instead of time, then to click Apply to validate
  - To apply the filter in Filters → Normal Mechanics → modes animation (real), and to validate
  - To display the window Animation view (finely View → Windows → View animation), and to add an animation for the filter above by clicking + beside Normalmodesanimationreal
  - To choose the mode to be visualized (Mode Array Selection) in the window Properties on the left and to validate, then to click the button Play (▶) to visualize the movement of mode
- **Transitory analysis**  
For the transitory analysis starting from the calculated clean modes, one will leave the “wizard” below. One will add to it the orders necessary following the analysis.

- To continue calculation in a new internship by the order CONTINUATION
- Definition of temporal discretization using order DEFI\_LIST\_REEL : of 0s with 2.0s with a step of time of 0.0001s.

### Construction of the specific force

- Definition of the load unit (value with 1) “force on the section B ” (AFFE\_CHAR\_MECA/FORCE\_FACE).
- Calculation of the elementary vectors forces (CALC\_VECT\_ELEM/ OPTION=' CHAR\_MECA').
- Assembly of the vector forces (ASSE\_VECTEUR).
- Definition of the function évoluée with time by FORMULA : VALE for the expression  $FY(t) = 10000000 \cdot \sin(2\pi Freq1 \cdot t)$  directed according to the axis Y and applied to the section B , where  $Freq1 = 20 \text{ Hz}$  . The variables should initially be declared used in the formula.
- To evolve the function by CALC\_FONC\_INTERP according to the definite list of the moments. With PROL\_GAUCHE/DROITE = CONSTANT to define the curve out the framework of the list of the moments.

### Transient on modal basis

- Projection of the problem assembled on the basis of clean mode

```
PROJ_BASE (BASE=...,  
           MATR_ASSE_GENE= (  
                                   # For the matrix of mass  
                                   _F (MATR_ASSE = ... ) ,  
                                   # For the matrix of rigidity  
                                   _F (MATR_ASSE = ... ) ,  
                                   # For the dynamic force  
VECT_ASSE_GENE= _F (TYPE_VECT = 'FORC' ,  
                   VECT_ASSE = ... ) , ) ) .
```

- Transitory calculation by modal recombination (DYNA\_VIBRA)

```
DYNA_VIBRA (TYPE_CALCUL=' TRAN' ,  
           BASE_CALCUL=' GENE' ,  
           SCHEMA_TEMPS= _F (DIAGRAM = 'DIFF_CENTRE' ) ,  
           AMOR_MODAL = ... ,  
           MATR_MASS = ... ,  
           MATR_RIGI = ... ,  
           EXCIT = ... ,  
           INCREMENT = ... ,  
           ... ) .
```

- Recovery of displacements in Y at the point B (RECU\_FONCTION): RESU\_GENE, GROUPE\_NO, NOM\_CHAM=DEPL, NOM\_CMP=DY.

- Impression of these functions to the format `XMGRACE (IMPR_FONCTION)`: `FORMAT`, `UNIT`, `CURVE`, etc.
- To visualize the fields of displacement:
  1. Order `REST_GENE_PHYS` : because the type of the results generated (bases physical) directly by `DYNA_VIBRA` cannot print with the format `MED` by `IMPR_RESU` , it is necessary to restore in the physical base of the results in coordinates generalized while giving `LIST_INST` , `NOM_CHAM` , `RESU_GENE` . Caution: the steps of many times lead very heavy file `MED`, and one recommends to redefine a list of the moments, for example with a step of time 0.01s.
  2. Order `IMPR_RESU` : to print with the format `MED`

## 4.4 Post-treatment of the results

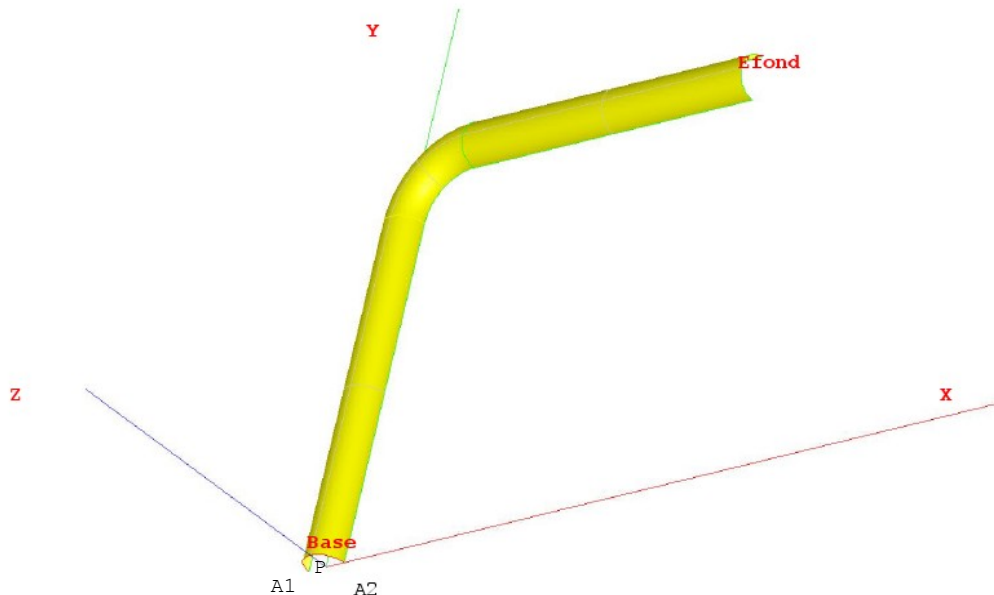
One can visualize directly the curve of displacements `DY` at the point `B` in function time afterwards `IMPR_FONCTION` with format `XMGRACE` .

To import the transitory results in the Results mitre, and to check displacements `DY` on a point at the end of the pipe (see chapter 2.2.2).

## 5 Modeling C

In the case of modeling in elements hulls, the grid consists of the discretization of **average surface** piping. Geometry being symmetrical compared to the plan  $(A, X, Y)$ , only one half-surface is netted.

Boundary conditions and loading: embedding at the two ends of piping, and pressure on the surface intern.



### 5.1 Geometry

One can create this geometry by defining the points  $A1$ ,  $P$  and  $A2$ , then the arc of a circle *Base* (Menu New Entity → Lowic → Not/Arc). It is then enough to create the first right pipe *AC* starting from the arc of a circle *Base* by the menu New Entity → Generation → Extrusion: Vector = OY, Height = 3.

To create the elbow, it is necessary to recover the end of the pipe *AC* while applying the menu New Entity → Explode (Edge), then to create a vector parallel with axis Z and passing the point O  $(0.6, 3.0, 0)$ . Then Générer geometry of Coude by the menu New Entity → Generation → Revolution.

Lastly, to apply the same approach for the pipe *DB* (Explode then Extrusion).

Créer one “compound” (Menu New Entity → Build → Compound) while selecting three parts of piping.

One will create then the groups of S meshes where one wishes to put limiting conditions: Base, Symmetry, Efond and surface of the pipe (Menu New Entity → Group → Create Group).

One will create also the group not *AI*.

### 5.2 Maillage

To launch the module Mesh platform Salomé-Meca.



The grid is defined by the menu `Mesh → Create Mesh`. Sélectionner geometry to be netted, then the algorithm and assumption of discretization per dimension:

- 2D Quadrangle : Mapping.
- 1D Wire Discretisation with the assumption basic Number of Segment ( 15 segments by edge .

Then to calculate the grid Finely ( `Mesh → Compute` ).

To allow a different refinement according to edges, one will create one under-grid (Finely `Mesh → Create Sub-mesh` ) defining the assumption basic Number of Segment on the circumference, for example 10 segments on *base* and the additional assumption " Propagation of 1D assumption one Opposite Edges " .

Then thatculer grid (Finely `Mesh → Compute` ).

To create Lbe groups DE e-mailbe geometrical correspondents with the group ( Menu `Mesh → Create Groups FRomanian Geometry` ) .

Exporter it grid with the format `MED` .

## 5.3 Creation and launching of the calculation case (via AsterStudy )

To launch the module `AsterStudy` platform `Salomé-Meca` .

Then in left column, to click on the mitre `View` box.

One defines the command file of the calculation case.

Foot-note: to add orders by the menu `Commands → All show`.

Principal stages this mechanical calculation for the creation and the launching of the calculation case are the following ones:

- To see the grid with format `MED`: Order `LIRE_MALLAGE`.
- To define the finite elements used: Order `AFFE_MODELE`. Lpiping has serhasmodelledE by elements of hull (`DKT`).
- To direct normals with the elements: Order `MODI_MALLAGE/ ORIE_NORM_COQUE` in the same way to direct all the elements, with a normal turned towards the interior of the pipe (being given the convention of sign on the pressure) in order to give a positive value to the pressure (to use the group `surface`).
- To define material: Order `DEFI_MATERIAU : E, NAKED, ALPHA`
- To affect material: Order `AFFE_MATERIAU`. The mechanical characteristics are identical on all the structure.
- To affect Lbe characteristics elements hulls : Order `AFFE_CARA_ELEM/HULL` to define the thickness.
- To affect conditions with limits mechanics and itS loadingS : Order `AFFE_CHAR_MECA` :
  - There is an embedding on the group of meshes *Base* and *Efond* , and conditions of symmetry (normal displacement *DZ* no one and rotations *DRX* and *DRY* worthless) on the group of meshes *Symetrie* : `DDL_IMPO` .
  - An internal pressure *P* : `PRES_REP`. It is necessary to convert the pressure *P* on the surface intern in the pressure on the average surface.
- To solve it problem rubber band : Order `MECA_STATIQUE : CARA_ELEM, CHAM_MATER, EXCIT, MODEL`.
- To print displacements and the constraints with the format `MED` : Order `IMPR_RESU..`
- For launchR the calculation case, in left column, to click on the mitre `History View` .

## 6 Modeling D

### 6.1 Geometry

One can create the geometry by defining the points  $A, C, D, B$ , then two lines  $AC$  and  $DB$ , and the arc of a circle  $CD$ . This can be done in several ways:

- By defining the coordinates of each point, including the point medium of the arc  $CD$  (Menu New Entity → Lowic →...);
- In a simpler way by using L'tool "Sketcher" (Menu New Entity → Lowic → 2D Sketch).

For having built the whole of the line, it is necessary to define the groups under this "Wire": it will be enough for these practical works to create the groups  $A, B, AC, CD$ , and  $DB$  (Menu New Entity → Group → Create Group).

### 6.2 Maillage

To launch the module Mesh platform Salomé-Meca.

The grid is defined by the menu Mesh → Create Mesh. Sélectionner geometry to be netted, then the algorithm and assumption of discretization per dimension: 1D Wire Discretisation with the assumption of 15 segments by edge.

### 6.3 Creation and launching of the calculation case (via AsterStudy)

To launch the module AsterStudy platform Salomé-Meca.  
Then in left column, to click on the mitre View box.

One defines the command file of the calculation case.

Foot-note: to add orders by the menu Commands → All show.

The study requires the first transitory thermal calculation followed by a mechanical calculation.

Principal stages for the creation and the launching of the calculation case are the following ones:

- To see the grid with format MED: Order LIRE\_MAILLAGE.
- To define the finite elements used: Order AFFE\_MODELE.
- To define material: Order DEFI\_MATERIAU.
- To affect material: Order AFFE\_MATERIAU.
- To affect characteristics of the elements beams: Order AFFE\_CARA\_ELEM.
- To affect conditions with limits mechanics and the loading : Order AFFE\_CHAR\_MECA / Enforce DOF (DDL\_IMPO). The point  $A$  is embedded.
- To define Lbe matrices of the elastic problem : Order ASSEMBLY/ MASS\_MECA and RIGI\_MECA.
- Calculationner Lbe the first 5 clean modes : Order CALC\_MODES.
- Imprimer Lbe clean modes with the format MED : Order IMPR\_RESU. One will print the grid and the modes.

For launchR the calculation case, in left column, to click on the mitre History View.

Transitory analysis:

- Construction of the specific force
  1. To affect Lload has "forces at the point  $B$ " : Order AFFE\_CHAR\_MECA / FORCE\_NODALE.
  2. Calculationner Lbe elementary vectors forces : Order CALC\_VECT\_ELEM.

3. To build a field with the nodes by hassemblage of the vector forces :  
Order ASSE\_VECTEUR.
  4. To define the function evolution of time : Order FORMULA.
- Transient on modal basis
    1. Projection of the problem assembled on the basis of clean mode : Order PROJ\_BASE.
    2. Calculation of it transient by modal recombination : Order DYNA\_VIBRA.
    3. To extract Lbe displacements in  $Y$  in  $B$  : Order RECU\_FONCTION.
    4. Imprimer these functions with the format TABLE : Order IMPR\_FONCTION.