

## 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  $0.2\text{ m}$ .

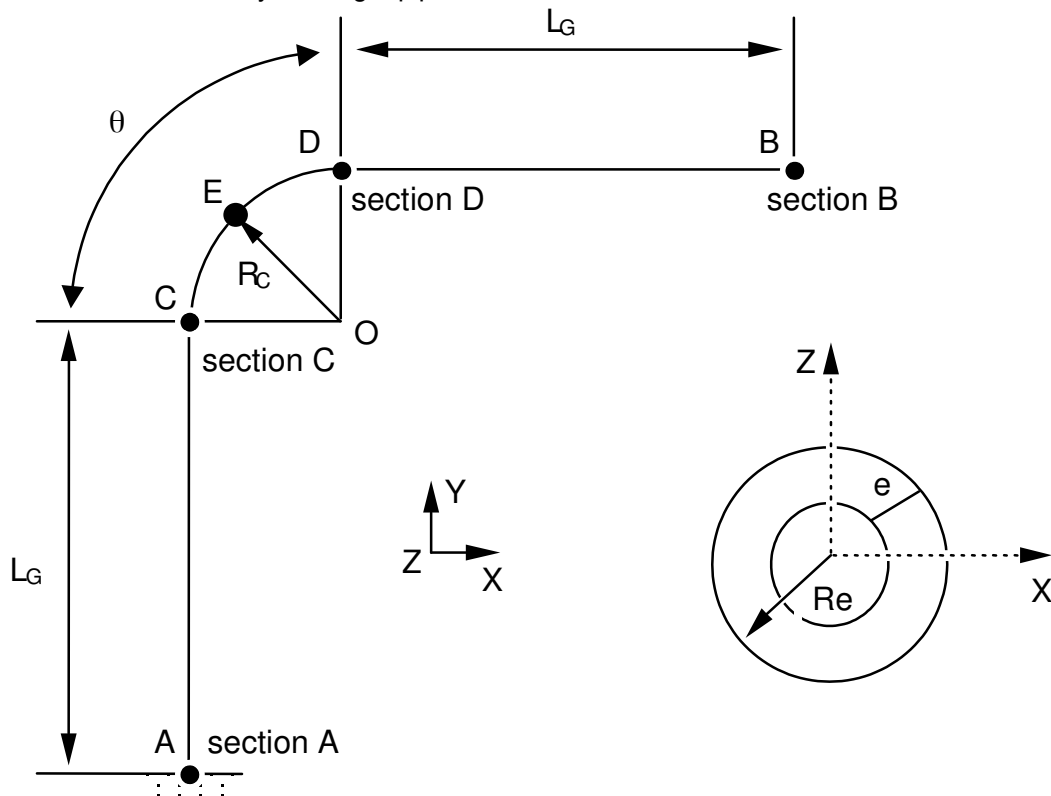


Figure 1.1-a

Note:

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

### 1.2 Material properties

For all modelings:

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

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

For thermoelastic calculation (modeling A):

- the thermal dilation coefficient  $\alpha = 10.92 \cdot 10^{-6}/^\circ\text{C}$ ,
- thermal conduction  $\lambda = 54.6\text{ W/m}^\circ\text{C}$ ,
- voluminal heat  $\rho C_p = 3.71 \cdot 10^6\text{ J/m}^3^\circ\text{C}$ ,

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:

- there is an embedding on the level of the section  $A$ ,
- for the static loadings, there is 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  $P = 15.10^{+6} \text{ N/m}^2$  (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):

$F_Y(t) = 10\,000\,000 \cdot \sin(2\pi \text{Freq1} \cdot t)$  directed according to the axis  $Y$  and applied to the section  $B$ ,

$\text{Freq1}$  such as  $\omega = 2\pi \text{Freq1} = 121 \text{ rad/s}$ .

## 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), it displacement at higher exit of elbow,
- for the loading thermomechanical (modeling A), the temperature in internal wall at the bottom of tube (70°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 displacements at higher exit of elbow,
- for modeling 1D (modeling D), the constraints equivalent in foot of tube.

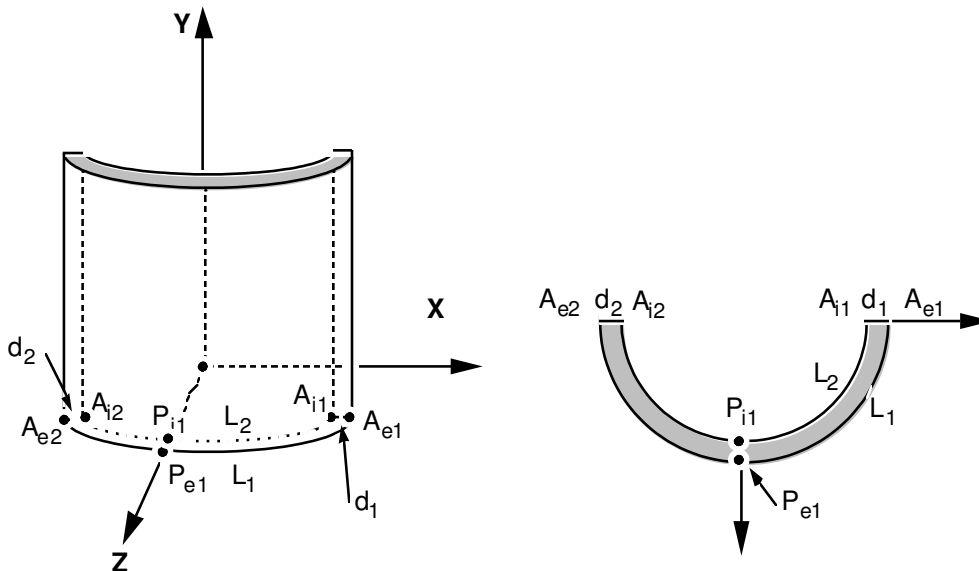
### 3 Modeling A

#### 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.



#### 3.2 Geometry

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

Two processes to create the geometry:

- 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  $L1$  and  $L2$ , Dbe segments  $d1$  and  $d2$  (Menu `New Entity` → `Lowic` →...).
  2. Creation contour (wire) and of surface (object of the type "Face") delimited by  $L1$ ,  $d1$ ,  $L2$ ,  $d2$  (Menu `New Entity` → `Build` →...).
  3. Creation of the telegraphic way: points  $A$ ,  $C$ ,  $D$ ,  $B$ , then two lines  $AC$  and  $DB$ , and the arc of a circle  $CD$ .
  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  $Z$ , using two elementary cylinders height 3 and of rays 0.2 and 0.18 (Menu `New Entity` → `Primitives` → `Cylinder`), and of one Boolean operation (Menu `OperationS` → `Boolean` → `CC`). To re-elect the object created in `TUYI`.

2. To apply a rotation to *TUY1* around *X* so that it is of axis *Y* (Finely Operations → Transformation → Rotation).
3. To select the higher face (by creating for example a group – Menu New Entity → Group → Create Group), and to generate the elbow by the menu New Entity → Generation → Revolution. It is necessary for that to create the axis of revolution of direction *Z* passing by the point  $(0.6, 3, 0)$ .
4. Then to create the second right pipe in the same way *TUY2*.
5. It remains to select half of the piping located in the plan  $Z > 0$ . For that one can cross using a tool “BOX” (Finely New Entity → Primitives → Box), or of partition (Menu Operations → Partition) after having created a plan (Menu New Entity → BASIC → Plane), and to create one “compound” by selecting the part located in  $Z > 0$  (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, 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). 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, Symmetry, Efond, Surfint and Surfext (Menu New Entity → Group → Create Group).

One will create also the group of edge with an edge thickness ( $dl$ ) and another group of edge with an edge of circumference ( $Ll$ ) 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.

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.
- 1D Wire Discretisation with the assumption of 15 segments by edge.

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, for example 2 segments on  $dl$  and the additional assumption “ Propagation of 1D assumption one Opposite Edges ”.
- One sub-mesh defining the number segments on the circumference, for example 10 segments on  $Ll$  and the additional assumption “ Propagation of 1D assumption one Opposite Edges ”.

Then thatculer grid (Finely Mesh → Compute).

To create L be groups D E e-mail be geometrical correspondents with the group ( Menu Mesh → Create Groups F Romanian Geometry ).  
E xport er it grid with the format MED .

#### 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.

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\_MALLAGE.
- To define material: Order DEFI\_MATERIAU .

#### Thermal calculation:

- To define the finite elements used: Order AFFE\_MODELE for modeling in 3D.
- To affect material: Order AFFE\_MATERIAU.
- To affect conditions with limits thermics and the loading : Order AFFE\_CHAR\_THER\_F:
  - ON has a transient of temperature imposed on the interior surface of piping (assembled 20 °C with 70 °C in 10s with the order DEFI\_FONCTION).
  - It is considered that piping is insulated and one does not specify a condition on external surface (what returns to a null flow).
- To solve thermal problem linear: Order THER\_LINEAIRE, by defining an initial state of uniform temperature equal to 20 °C (Initial condition (ETAT\_INIT)). The calculation of the field of temperature is carried out for the moments 0,5 et 10s defined by the order DEFI\_LIST\_REEL.

#### Thermomechanical calculation:

- To define the finite elements used: Order AFFE\_MODELE for modeling in 3D.
- To affect material: Order AFFE\_MATERIAU. The result of thermal calculation will be applied as a variable ordering the behavior of mechanical material (externhasL state variable assignment (AFFE\_VARC)).
- To affect conditions with limits mechanics and the loading : Order AFFE\_CHAR\_MECA :
  - L piping has is embedded at its two ends and it present a symmetry plane: Enforce DOF (DDL\_IMPO) .
  - One a pressure has surfaces intern of it of 15 MPa : PRES\_REP .
- To solve it problem linear static mechanics : Order MECA\_STATIQUE with the calculation of the field of temperature defined above.
- To calculate the field : Order CALC\_CHAMP.
  - for C alcul of the stress field by elements extrapolated with nodes : CONSTRAINT/SIEF\_ELNO .
  - for C alcul 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

To launch the module ParaViS platform Salomé-Meca .

- To open the file of results ( Menu File → Open Para View File ) container the “mechanical” part. To apply the filter Merge Blocks by the menu Filters .
- To open the file of results container the “thermal” part. To apply 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 L has temperature and L be forced at the points of Gauss ( Menu Filters → ELGA field To Surface ).

### 3.5.2 With the explorer of results Stanley

Using this explorer, one will be able to trace:

- deformation, by selecting the field DEPL,
- isovaleurs of the constraints of Von Mises to the nodes,
- change of the temperature according to time at the points *Pil* and *Pel* .

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

Principal stages of **second** execution with Aster\_Study Swill eront:

- Extraanger Lbe values the component SIYY field SIEF\_ELNO for an azimuth on the level of the entry of elbow in the case of load corresponding to  $F_y$  : Order MACR\_LIGN\_COUPE. The azimuth is defined by the way of ends (0.18,3,0.) and (0.2,3,.0.) .
- To print the table produced with the format XMGRACE : Order IMPR\_TABLE.



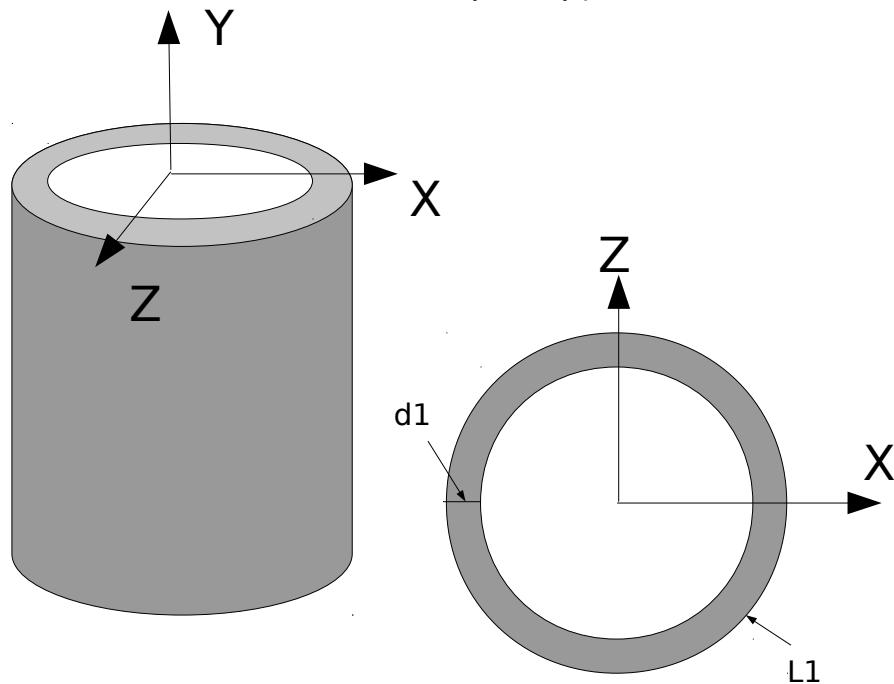
## 4 Modeling B

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.



### 4.1 Geometry :

To launch the module Geometry platform Salomé-Meca.

Two processes to create the geometry:

- Extrusion of a basic surface according to a way:
  1. Creation of the circles interior and external of the base ( Menu New Entity → Low ic → Circle) , then the surface delimited by the two circles ( Menu New Entity → Build → Face ).
  2. Creation of the telegraphic way: points  $A$  ,  $C$  ,  $D$  ,  $B$  , then two lines  $AC$  and  $DB$  , and the arc of a circle  $CD$  (Menu New Entity → Lowic →...), then contour (Menu New Entity → Build → Wanger).
  3. 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  $Z$  , using two elementary cylinders height  $3$  and of rays  $0.2$  and  $0.18$  (Menu New Entity → Primitives → Cylinder), and of a Boolean

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 2019 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

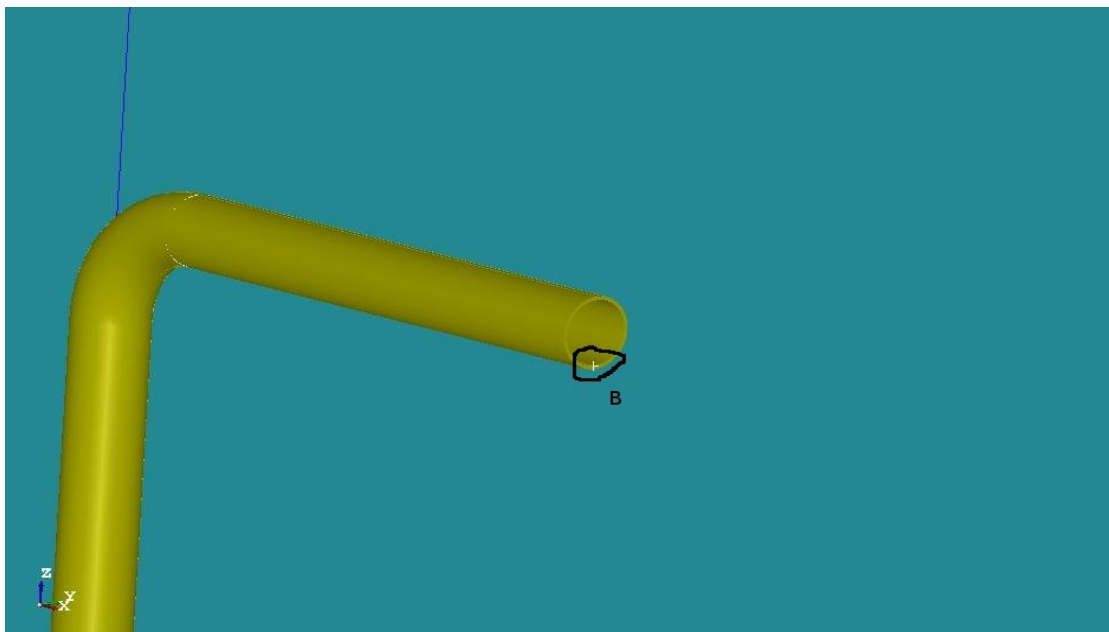
- operation (Menu `OperationS` → `Boolean` → `CC`). To re-elect the object created in `TUY1`.
2. To apply a rotation to `TUY1` around `X` so that it is of axis `Y` (Finely Operations → Transformation → Rotation).
  3. To select the higher face (by creating for example a group – Menu `New Entity` → `Group` → `Create Group`), and to generate the elbow by the menu `New Entity` → `Generation` → `Revolution`. It is necessary for that to create the axis of revolution of direction `Z` passing by the point  $(0.6, 3, 0)$ .
  4. Then to create the second right pipe in the same way `TUY2`.

### 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` → `Check Compound of Blocks`.

According to the method of construction, 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). 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` as well as the section `B` at the end of the pipe to apply the dynamic force (Menu `New Entity` → `Group` → `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 `B`, below, on the interior surface of the 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.
- 1D Wire Discretisation with the assumption of 15 segments by edge.

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, for example 2 segments on  $d$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".
- One sub-mesh defining the number segments on the circumference, for example 10 segments on  $L$  and the additional assumption " Propagation of 1D assumption one Opposite Edges ".

Then thatculer grid (Finely Mesh → Compute).

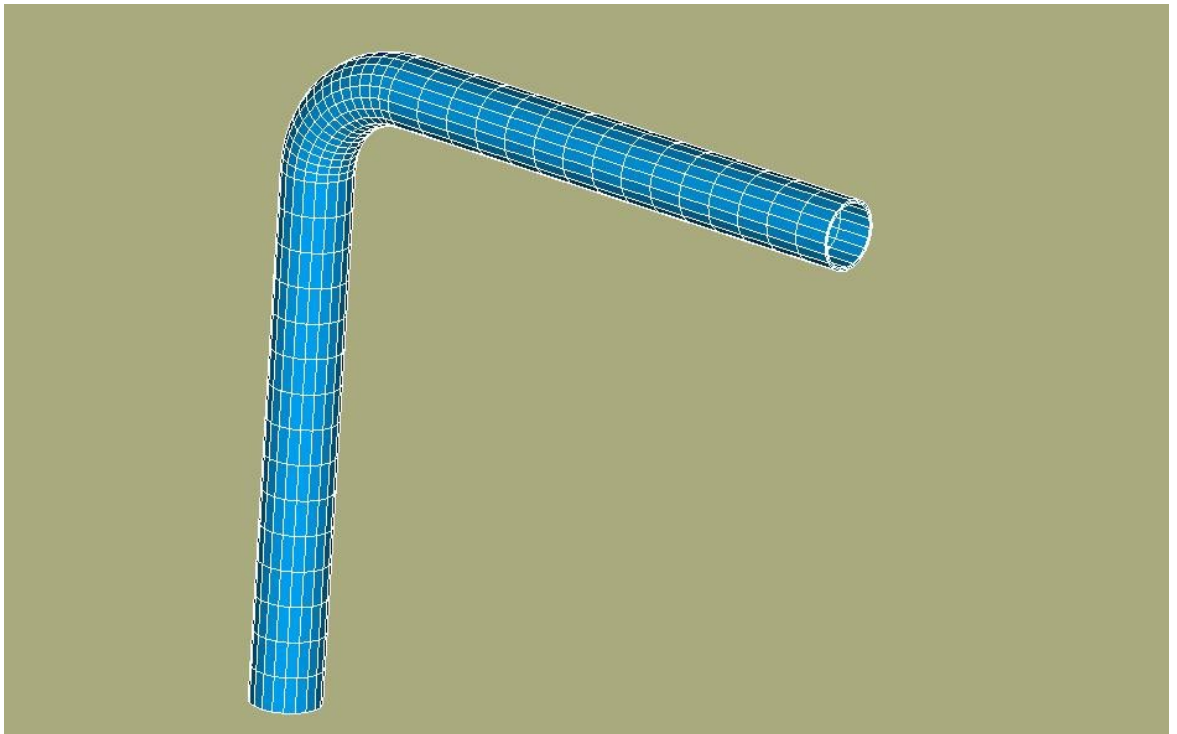
To create L be groups D E e-mail be geometrical correspondents with the group ( Menu Mesh → Create Groups F Romanian Geometry ).

E xport er 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 *Code\_Aster* will be:

- Preparation of the data and analyzes modal

One will use the “wizard” for the modal analysis. It is enough to enter the calculation data while following the displayed windows. One should not forget to specify embedding on the basis. The first five modes of the pipe will be calculated.

One will launch then calculation in the module SALOMÉ *Code\_Aster*. One will be able to visualize the modes in SALOMÉ by “deforming them”. One can also “animate them” thanks to the function `Sweep`. It makes it possible to better understand the modal deformation.

In PARAVIS, this animation is done directly via the macro one `modes`. One selects the mode to be visualized then in the `ScaleVector` object of the tree.

- Transitory analysis

For the transitory analysis, one will leave the command file created by the “wizard” of SALOMÉ. One will add to it the orders necessary following the analysis.

### Construction of the specific force

- Definition of the load “forces on the section  $B$ ” (`AFFE_CHAR_MECA/FORCE_FACE`).
- Calculation of the elementary vectors forces (`CALC_VECT_ELEM`).
- Assembly of the vector forces (`ASSE_VECTEUR`).
- Definition of the function evolution of time (`FORMULA`).

### Transient on modal basis

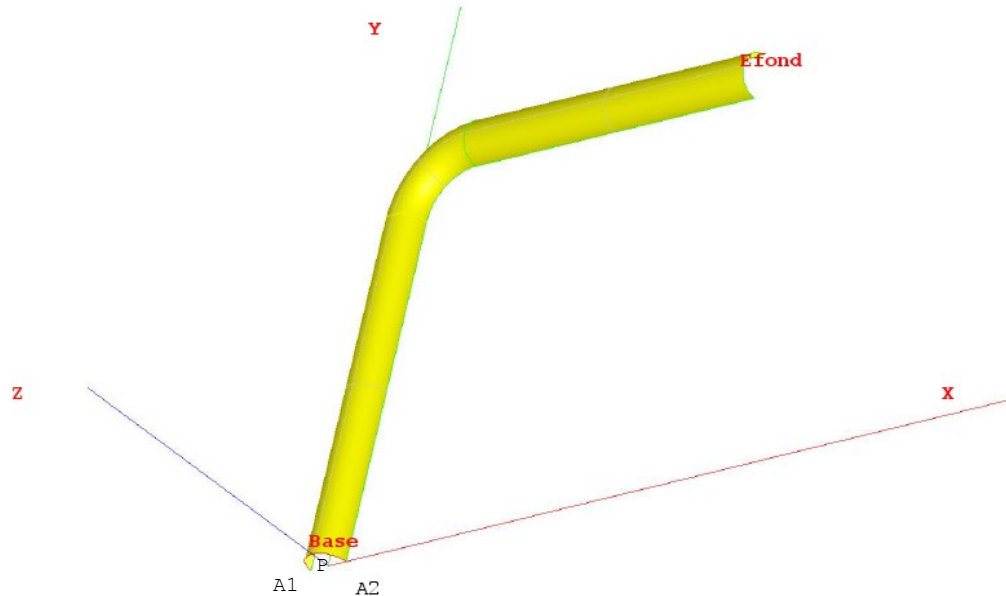
- Projection of the problem assembled on the basis of clean mode (`PROJ_BASE`).
- Transitory calculation by modal recombination (`DYNA_VIBRA (TYPE_CALCUL=' TRAN', BASE_CALCUL=' GENE')`).
- Recovery of displacements in  $Y$  at the point  $B$  (`RECU_FONCTION`).
- Impression of these functions to the format `TABLE (IMPR_FONCTION)`.

## 4.4 Post-treatment of the results

One will import in SALOMÉ the table of displacements according to time and one will make build in SALOMÉ the curve of displacements at the point  $B$ .

## 5 Modeling C

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



### 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 →...).

It is then enough to create the first right pipe  $AC$  by the menu New Entity → Generation → Extrusion.

To create the elbow, it is necessary to recover the end of the pipe  $AC$  while applying the menu New Entity → Explode, then to generate the geometry of Coude by the menu New Entity → Generation → Revolution.

Lastly, to apply the same approach for the pipe  $DB$  and to create one "compound" by selecting the situ part(PRES=PRES,).

ée in  $Z > 0$  (Finely New Entity → Build → Compound).

One will create then the groups of faces 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  $A1$ .

### 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.
- 1D Wire Discretisation with the assumption of 15 segments by edge.

To allow a different refinement according to edges, one will create one under-grid (Finely Mesh → Create Sub-mesh) defining the number segments 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 L be groups D E e-mail be geometrical correspondents with the group ( Menu Mesh → Create Groups F Romanian Geometry ) .

E xport er 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.

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. The right pipes and the elbow will be modelled by elements of hull (DKT).
- To direct normals with the elements: Order MODI\_MAILLAGE/ 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 AI ).
- To define material: Order DEFI\_MATERIAU . The mechanical characteristics are identical on all the structure.
- To affect material: Order AFFE\_MATERIAU.
- 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* conditions of symmetry (normal displacement DZ no one and rotations DRX and DRY worthless) on the group of meshes *Symetrie* : Enforce DOF (DDL\_IMPO) .
  - An internal pressure P : PRES\_REP.
- To solve it problem rubber band : Order MECA\_STATIQUE.
- To print displacements and the constraints with the format MED : Order IMPR\_RESU. One will print in form listing displacement for each result on the section B.
- For launch R 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_CAR_A_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 as of clean modes : Order PROJ\_BASE.
    2. Calculationner 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.