

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`, `POU_C_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 m ,
- the ray R_C elbow is of 0.6 m ,
- the angle θ elbow is of 90° ,
- the thickness of the right pipes and the elbow is of 0.02 m ,
- and the external ray R_e right pipes and elbow is of 0.2 m .

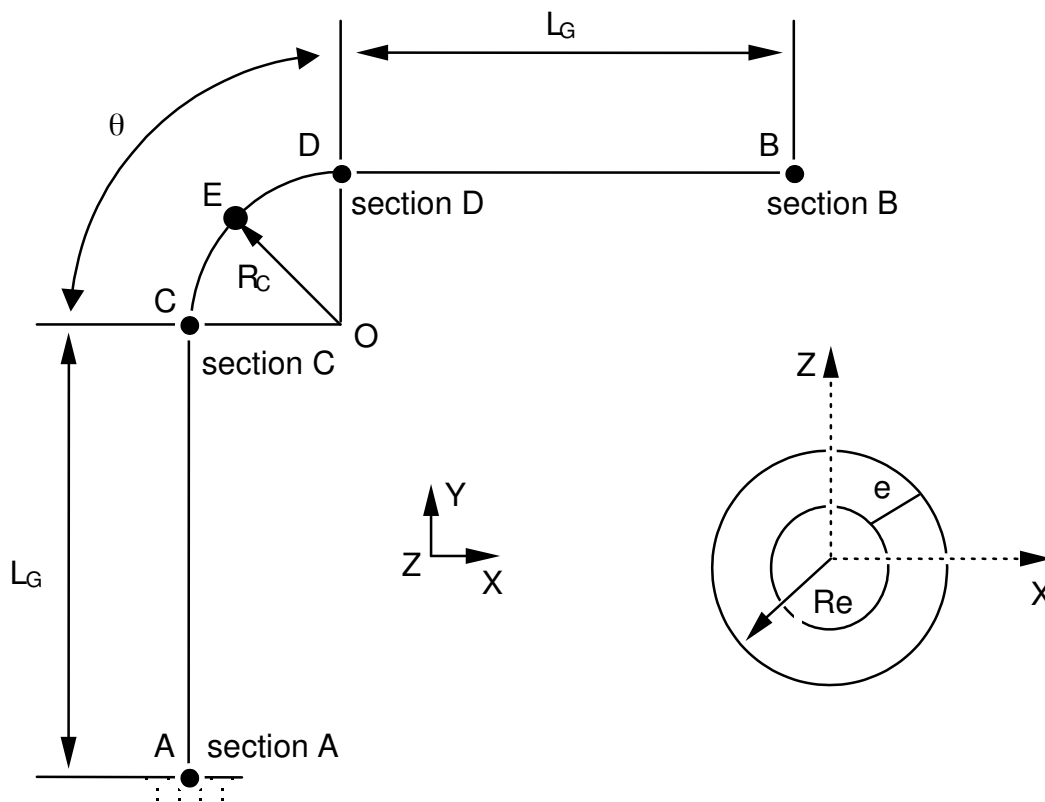


Figure 1.1-a

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 $20^{\circ}C$:

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

For thermoelastic calculation (modelings A)

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

For dynamic calculation (modeling B, D)

- density $\rho = 7800 kg/m^3$,
- 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 loading statics, 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 $P = 15 \cdot E + 6 N/m^2$ (modeling hulls or 3D),
- thermomechanical loading with a transient of temperature imposed on the inner face of piping (assembled $20^{\circ}C$ with $70^{\circ}C$ 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):

$$FY(t) = 10\,000\,000 \cdot \sin(2\pi \text{Freq}l \cdot t)$$

directed according to the axis *Y* and applied to the section *B* ,

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

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 A and C), it displacement at higher exit of elbow,
- for the thermomechanical loading (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

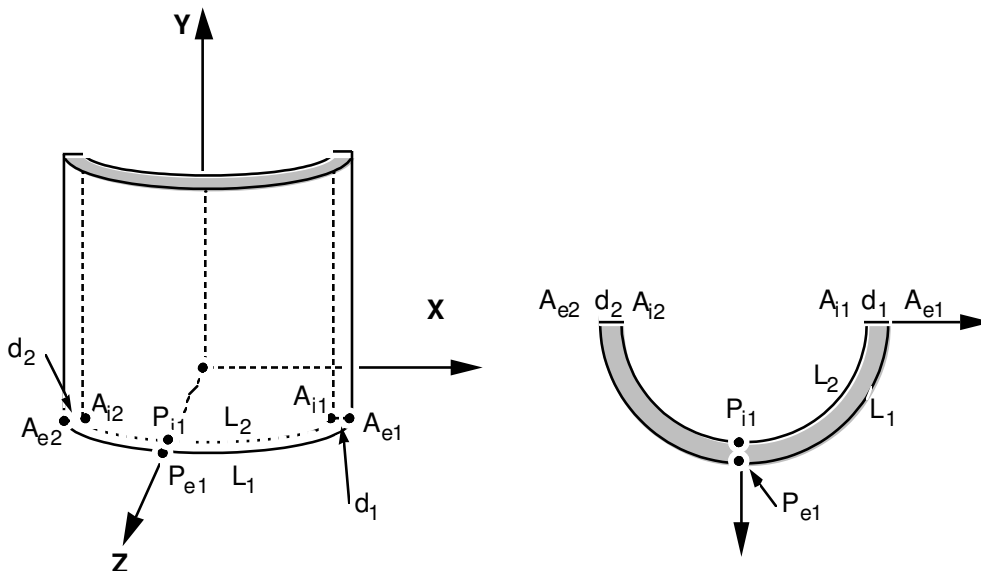
3.1 Characteristics of modeling

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.



Creation of the geometry :

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_{e2} , then arcs of a circle $L1$ and $L2$, segments $d1$ and $d2$, contour (wire) and (object of the type surfaces it "Face") delimited by $L1$, $d1$, $l2$, $d2$,
 2. Creation of the telegraphic way: points A, C, D, B , then two lines AC and DB , and the arc of a circle CD .
 3. This surface is then extruded by Generation/Extrusion along has 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, and of a Boolean operation (operation/boolean/cut). To re-elect the object created in $TUY1$.
 2. To apply a rotation to $TUY1$ around X so that it is of axis Y . To select the higher face (by creating for example a group), and to generate the elbow by Generation/ Revolution. It

is necessary for that to create the axis of revolution of direction Z passing by the point $(0.6, 3, 0)$. Then to create the second right pipe in the same way $TUY2$. It remains to select half of the piping located in the plan $Z > 0$. For that one can cross using a tool "box", or to use `Plane partition/` and to create one "compound" by selecting the part located in $Z > 0$.

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 `Measure/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 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`.

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 refinement different on these edges.

One will be able to test the other mode of use of `GEOM /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 `File /dump Study`. This file python can be read again by `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 Salomé.

3.2 Characteristics of the grid

The grid is defined by `Create Mesh`, selection of the geometry to be netted, then algorithm and assumptions of discretization per dimension:

- 3D Hexaedron (I, J, K)
- 2D Quadrangle
- 1D Wire discretization with the assumption of 15 elements by edge and the additional assumption "Propagation of 1D Hyp one Opposite Edges."

To allow a different refinement according to edges, under-grids will be created:

- one `sub_mesh` defining number of elements in the thickness, for example 2 elements on dl .
- One `sub_mesh` defining the number of elements on the circumference, for example 10 elements on Ll .

Then to calculate the grid by `Compute`.

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.

3.3 Orders Aster

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

The principal stages of calculation are:

- Reading of the grid (LIRE_MALLAGE) with format MED.
- Definition of material (DEFI_MATERIAU)
- Thermal calculation:
 - Definition of the finite elements used (AFFE_MODELE/PHENOMENON/MODELING). Modeling is 3D.
 - Assignment of material (AFFE_MATERIAU).
 - Definition of the boundary conditions thermal (DEFI_FONCTION and AFFE_CHAR_THER_F) : There is a transient of temperature imposed on the interior surface of piping (assembled 20°C with 70°C in 10s). It is considered that piping is insulated and one does not specify a condition on external surface (what returns to a null flow).
 - Resolution of the thermal problem (THER_LINEAIRE), by defining an initial state of uniform temperature equal to 20°C (ETAT_INIT). The calculation of the field of temperature is carried out for the two moments 5s and 10s, defined by: DEFI_LIST_REEL.
- Thermomechanical calculation:
 - Definition of the finite elements used (AFFE_MODELE/PHENOMENON/MODELING). Modeling is 3D.
 - Assignment of material (AFFE_MATERIAU). The result of thermal calculation will be applied as a variable ordering the behavior of mechanical material (AFFE_MATERIAU / AFFE_VARC).
 - Definition of the boundary conditions and mechanical loading (AFFE_CHAR_MECA) : Piping is embedded at its two ends and presents a symmetry plane (DDL_IMPO). There is a pressure surfaces intern of it of 15 MPa (PRES_REP).
 - Resolution of the thermomechanical problem (MECA_STATIQUE).
 - Using CALC_CHAMP, calculation of the constraints by elements extrapolated with the nodes (SIEF_ELNO) and of the equivalent constraints of Von Mises (SIEQ_ELNO).
- One will print with the format RESULT maximum values of the tensor of constraints.
- One will print the fields of temperature, displacements, and the stress fields to the format MED.

One will create a calculation case with this command file and the grid. One will take care to have a base to be able post-to treat the results.

3.4 Postprocessing

3.4.1 With Salomé

One will visualize via the module VISU the deformed grid, isotherms, constraints (component and modulates).

One will visualize via the module PARAVIS grid deformed (filter WarpByVector), the isotherms (filter Contour), the constraints (filter Integrations Points/ELNO Points)

3.4.2 With the explorer of results Stanley

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 2017 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

Using this explorer, one will be able to trace:

- deformation, by selecting the field `DEPL`,
- isovalues of the constraints of Von Mises to the nodes,
- change of the temperature according to time at the points `Pil` and `Pe1`.

3.4.3 With Code_Aster in continuation

Principal stages of **second** execution with *Aster* will be:

- Creation of a new command file starting with `CONTINUATION` ;
- One will extract from the 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 \overline{Fy} . The azimuth is defined by the way of ends `(0.18, 3, 0.)` and `(0.2, 3, .0.)`. One will be able to use for that the order `MACR_LIGN_COUPE`, and impression of the table produced with the format `XMGRACE`.

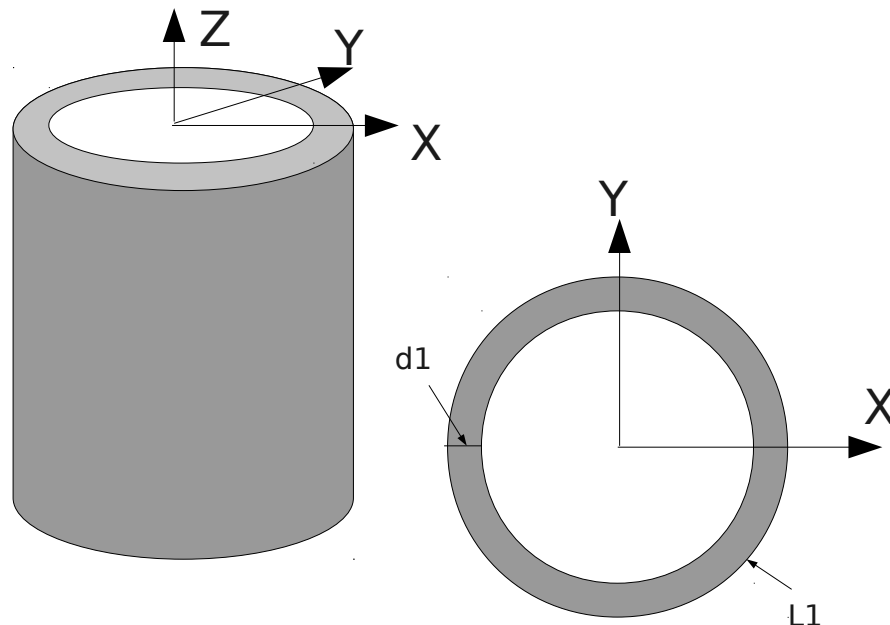
4 Modeling B

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.



Creation of the geometry :

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, then contour (wire) and (object of the type surfaces it " Face ") delimited by the two circles.
 2. Creation of the telegraphic way: points A , C , D , B , then two lines AC and DB , and the arc of a circle CD .
 3. This surface is then extruded by Generation/Extrusion along has path.
- Use of basic voluminal objects:
 1. To build a right pipe of axis Z , using two elementary cylinders height 3 and rays 0.2 and 0.18, and of a Boolean operation (operation/boolean/cut). To re-elect the object created in *TUYI*.

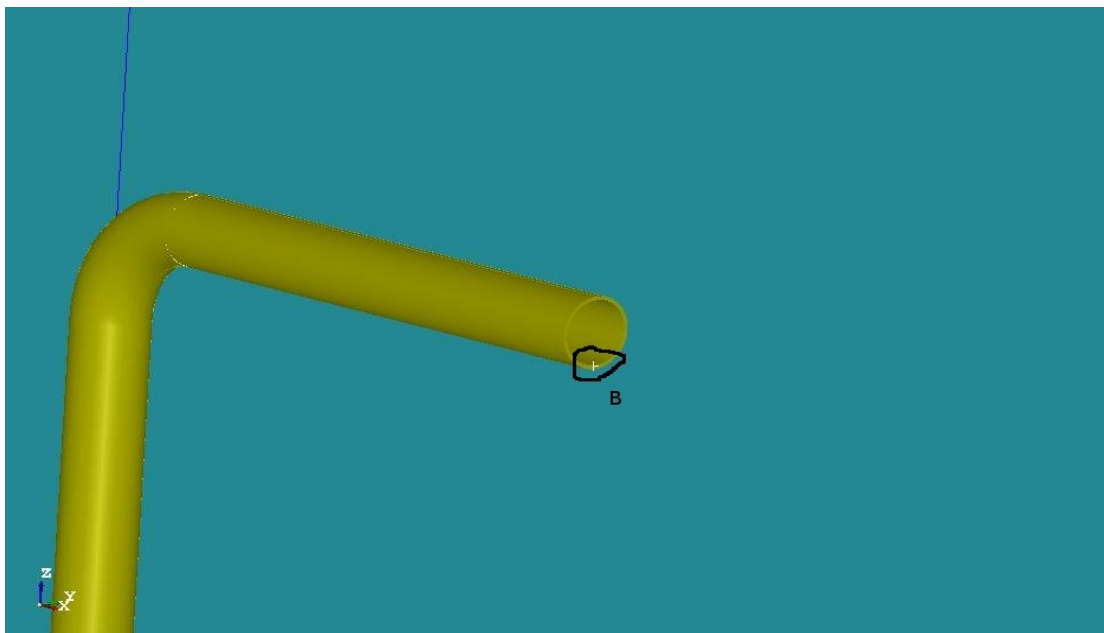
- To apply a rotation to *TUY1* around *X* so that it is of axis *Y*. To select the higher face (by creating for example a group), and to generate the elbow by *Generation/Revolution*. It is necessary for that to create the axis of revolution of direction *Z* passing by the point $(0.6, 3, 0)$. Then 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 "Hexaedron (I, J, K)" or to use the option "Automatic Hexaedrization".

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 *Measure/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 maillieurs.



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.

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 refinement different 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.

4.1 Characteristics of modeling

The grid is defined by *Create Mesh*, selection of the geometry to be netted, then algorithm and assumptions of discretization per dimension:

- 3D Hexaedron

- 2D Quadrangle
- 1D Wire discretization with the assumption of 15 elements by edge and the additional assumption "Propagation of 1D Hyp one Opposite Edges."

To allow a different refinement according to `edges`, under-grids will be created:

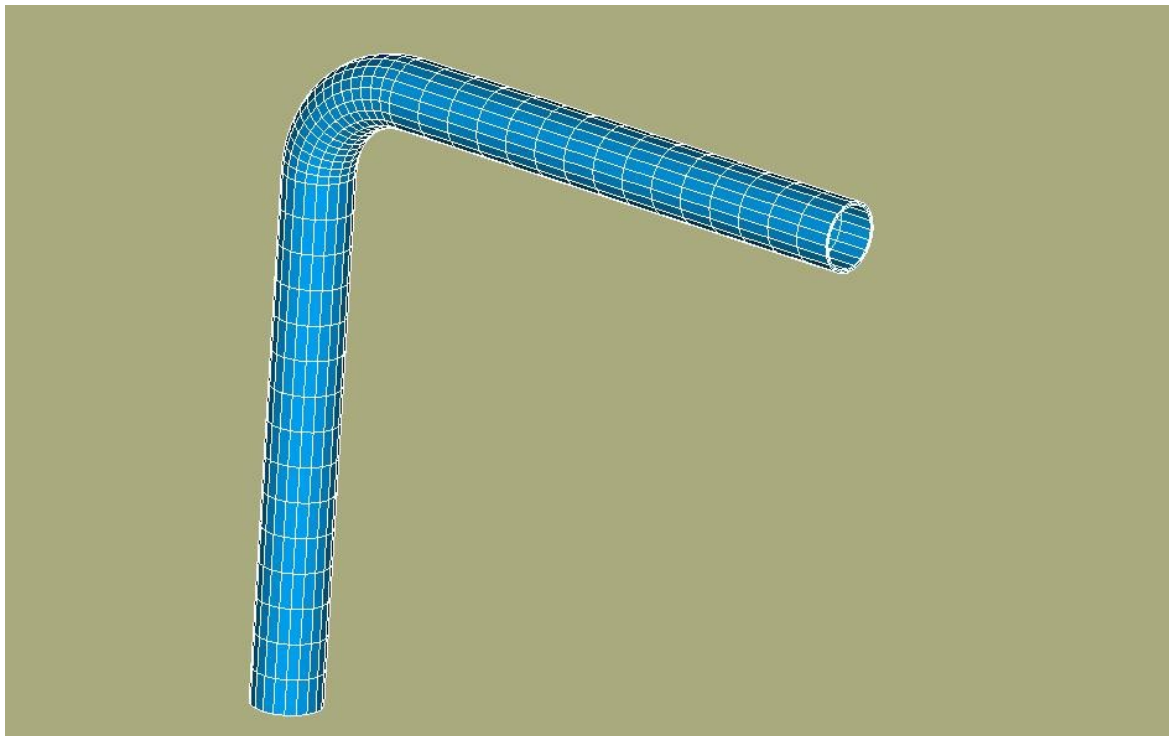
- one `sub_mesh` defining number of elements in the thickness, for example 2 elements on `dl`
- One `sub_meshes` defining the number of elements on the circumference for example 10 elements on `L1`.

To calculate the grid by `compute`.

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.



4.2 Orders Aster

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

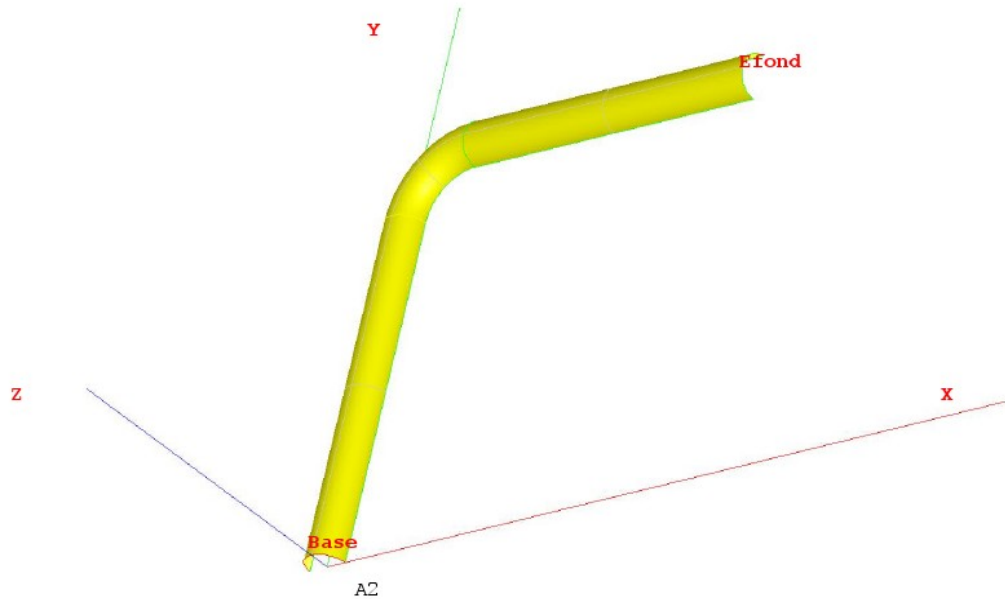
- Projection of the problem assembled on the basis of clean mode (`PROJ_BASE`).
- Transitory calculation by modal recombination (`DYNA_VIBRA (TYPE_CALCUL=' TRAN', BASE_CALCUL=' GENE')`).
- Recovery of displacements in *Y* at the point *B* (`RECU_FONCTION`).
- Impression of these functions to the format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

4.3 Postprocessings

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.



One can create this geometry by defining the two points $A1$ and $A2$, then the arc of a circle $Base$. It is then enough to create the first right pipe AC by *Generation / Extrusion*. To create the elbow, it is necessary to recover the end of the pipe AC while applying *New Entity / Explode*, then to generate the geometry of the elbow by *New Entity / Generation / Revolution*. Lastly, to apply the same approach for the pipe DB .

5.1 Characteristics of the grid

The grid is defined by *Create Mesh*, selection of the geometry to be netted, then algorithm and assumptions of discretization per dimension:

- 2D Quadrangle
- 1D Wire discretization with the assumption of 15 elements by edge and the additional assumption "Propagation of 1D Hyp one Opposite Edges."

To allow a different refinement according to edges, under-grids will be created:

- one *sub_mesh* defining number of elements in the thickness, for example 2 elements on $d1$.
- One *sub_mesh* defining the number of elements on the circumference for example 10 elements on $L1$.

5.2 Orders Code_Aster

Principal stages of calculation with *Code_Aster*, introduced into the command file via *Eficas*, are:

- Reading of the grid (*LIRE_MAILLAGE*) with format MED.

- Definition of the finite elements used (AFFE_MODELE). The right pipes and the elbow will be modelled by elements of hull (DKT).
- Reorientations of the normals to the elements: one will use 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*).
- Definition and assignment of material (DEFI_MATERIAU and AFFE_MATERIAU). The mechanical characteristics are identical on all the structure.
- Assignment of the characteristics of the elements hulls (AFFE_CARA_ELEM/HULL) to define the thickness.
- Definition of the boundary conditions and the loadings (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*.
- an internal pressure *P*.
- Resolution of the elastic problem for each loading case (MECA_STATIQUE).
- One will print in form listing displacement for each result on the section B.
- To print displacements and the constraints with format MED (IMPR_RESU).

6 Modeling D

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 ;
- In a simpler way by using it “ `sketcher` ”

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 .

6.1 Characteristics of the grid

The grid is defined by `Create Mesh`, selection of the geometry to be netted, then algorithm and assumptions of discretization per dimension:

1D `Wire` discretization with the assumption of 15 elements by edge.

6.2 Orders Aster

Principal stages of calculation with *Code_Aster*, introduced into the command file via Eficas, are:

- Reading of the grid to format MED.
- Definition of the finite elements used (`AFFE_MODELE`).
- Definition and assignment of material (`DEFI_MATERIAU` and `AFFE_MATERIAU`).
- Assignment of the characteristics of the elements beams (`AFFE_CARA_ELEM`).
- Definition of the boundary conditions and the loading (`AFFE_CHAR_MECA`). The point A is embedded.
- Definition of the matrices of the elastic problem (`ASSEMBLY`).
- Calculation of the first 5 clean modes (`CALC_MODES`).
- Impression of the clean modes (`IMPR_RESU`): one will print the grid and the modes with format MED
- Transitory analysis

Construction of the specific force

- Definition of the load “forces at the point B ” (`AFFE_CHAR_MECA/FORCE_NODALE`).
- 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`).
- Recovery of displacements in Y in B (`RECU_FONCTION`).
- Impression of these functions to the format `TABLE` (`IMPR_FONCTION`).