

FORMA02 - Practical works of training “Initiation”: pipe elbow under thermomechanical request and dynamics

Summarized:

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

The modelizations used are the following ones:

- modelization a: modelization 3D, thermomechanical computation,
- modelization b: modelization 3D, dynamic computation, modal then transitory,
- modelization C: elements `shells DKT`, mechanical computation under loading of pressure interns
- modelization D: elements beams (`POU_D_T`, `POU_C_T`), dynamic computation.

The chapter 1 “Problem of reference” presents the problem to treating and the given common to all the modelizations; the statements of Practical works of training are included in this document:

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

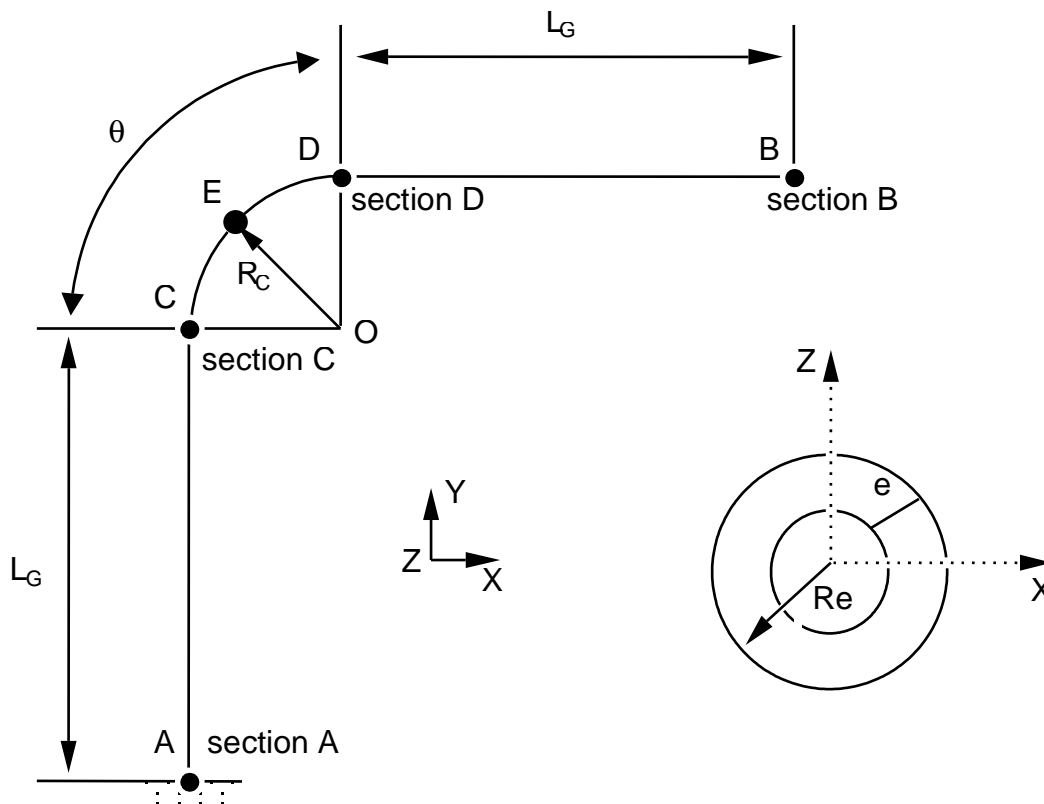
1 Problem of reference

1.1 Geometry

the study relates to a pipework 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 radius R_C of the elbow is of 0.6 m ,
- the angle θ of the elbow is of 90° ,
- the thickness of the right pipes and the elbow is of 0.02 m ,
- and the radius external R_e of the right pipes and the elbow is of 0.2 m .



Appear 1.1-a

Note::

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

1.2 Material properties

For all the modelizations:

Isotropic linear elastic material. the properties of the material are those of steel *A42* with 20°C :

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

For thermoelastic computation (modelizations A)

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

For dynamic computation (modelization B, D)

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

1.3 Boundary conditions and loadings

the boundary conditions for all the modelizations are the following ones:

- there is a fixed support on the level of the section *A*,
- for the loading statics, there is fixed support on the level of the section *A* and the section *B*.

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

- internal pressure $P = 15 \cdot E + 6 \text{ N/m}^2$ (modelization shells or 3D),
- thermomechanical loading with a transient of temperature imposed on the internal face of the pipework (gone up of 20°C to 70°C in 10s) and a condition of null exchange on the external face of the pipework (heat insulator) (modelization A only).

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

$$FY(t) = 10\,000\,000 \cdot \sin(2 \pi Freq1 \cdot t)$$

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

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

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 non regression.

2.2 Results of the various modelizations:

2.2.1 Static computation

For **the loading of pressure**, displacement in higher output of elbow for the various modelizations is worth:

Time t=0	
Modelization	DX
A : 3D (mesh Salome, 900 HEXA20)	- 4,51718E-04
C : Shell (mesh Salomé, 450 QUAD4)	4,67223E-04

For **the thermomechanical loading** :

Time t=10	
Modelization	
A : 3D Temperature as in point AII	70
A : 3D maximum Von Mises	1,8961E+08

2.2.2 Dynamic computation

the first eigenfrequency of the pipe elbow for a mesh Salome 3D, made up by 2250 HEXA20 (quadratic) is the following one:

Modelization	b: HEXA20
mode 1	the 8.91505

results in section B are the following for the modelization 3D:

Modelization	harmonic $t = 1s$	transient computation computation
		$f = 10$
B : 3D Displacement DY	-0,132956	the 0,934

results in section A at time $2s$ are the following for the modelization 1D:

Modelization	SN	SVY	$SMFZ$
D : 1D Equivalent stress	-4,8538255E+08	2,26529763E+0	-2,578787993E+10

3 Modelization A

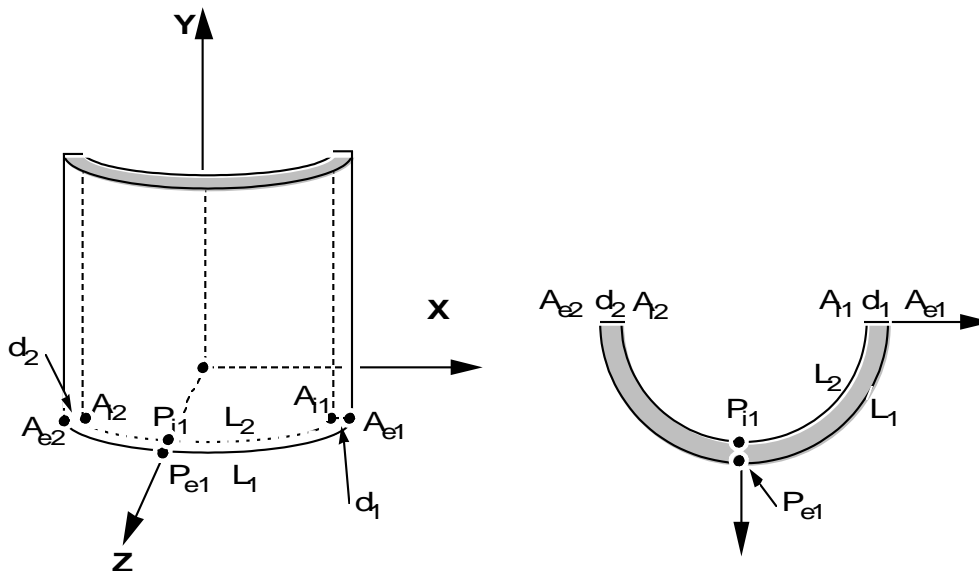
3.1 Characteristic of the modelization

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 a 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.

The pipework 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 path:
 1. Creation of the points A_{e1} A_{i1} A_{e2} A_{i2} P_{e1} P_{e2} , then the arcs of a circle L_1 and L_2 , the segments d_1 and d_2 , contour (wire) and surfaces it (object of the type "Face") delimited by L_1 d_1 L_2 d_2 ,
 2. Creation of the telegraphic path: 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 radius 0.2 and 0.18, and of a Boolean operation (operation/boolean/cut). To re-elect the object created in *TUYI*.
 2. To apply a rotation to *TUYI* around X so that it is of axis Y . To select the upper face (by creating for example a group), and to generate the elbow by Generation/ Revolution. It

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.

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 pipework located in the plane $Z > 0$. For that one can cut using tools "box", or use `Plane Partition/` and create a "compound" by selecting the part located in $Z > 0$.

Preparation of the geometry for a mesh 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 mesh, it is necessary to check that the geometry is topologically equivalent to a hexahedron (6 sides). 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 planes of the sections entered and output of elbow in order to obtain a geometry topologically equivalent to a hexahedron (6 sides). If one wants to make a mesh 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 mesh generators.

Creation of the groups :

One will create then the groups of sides where one wishes to put limiting conditions: `Base`, `Symmetry`, `Efond`, `Surfint` and `Surfext`.

One will create also the group of edge with an edge of the 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 Salome. Characteristics

3.2 of the mesh The mesh

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 the edges, under-meshes will be created:

- a sub_mesh defining number of elements in the thickness, for example 2 elements on. dl
- A sub_mesh defining the number of elements on the circumference, for example 10 elements on. Ll Then

to calculate the mesh by `Compute`. Note:

With

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

3.3

the study requires the first transitory thermal computation followed by a mechanical computation.

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.

The main steps of computation are: Reading

- of mesh (LIRE_MALLAGE) to med format. Definition
- of material (DEFI_MATERIAU) Thermal computation
- : Definition
 - of the finite elements used (AFFE_MODELE/PHENOMENE/MODELISATION) . The modelization is 3D. Assignment
 - of material (AFFE_MATERIAU) . Definition
 - of the thermal boundary conditions (DEFI_FONCTION and AFFE_CHAR_THER_F) : There is a transient of temperature imposed on the interior surface of the pipework (gone up of to $20^{\circ}C$ in $70^{\circ}C$). 10s It is considered that the pipework is insulated and one does not specify a condition on external surface (what returns to a null flux). Resolution
 - of the thermal problem (THER_LINEAIRE) , by defining an initial state of uniform temperature equal to ($ETAT_INIT\ 20^{\circ}C$) . The computation field of temperature is carried out for the two times and 5s , 10s defined by: DEFI_LISTE_REEL . Thermomechanical computation
- : Definition
 - of the finite elements used (AFFE_MODELE/PHENOMENE/MODELISATION) . The modelization is 3D. Assignment
 - of material (AFFE_MATERIAU) . Result of thermal computation will be applied as a variable ordering the behavior of the mechanical material (AFFE_MATERIAU/AFFE_VARC) . Definition
 - of the boundary conditions and mechanical loading (AFFE_CHAR_MECA) : The pipework is embedded at its two ends and presents a symmetry plane (DDL_IMPO) . There is a pressure surfaces of it intern of (PRES_REP 15 MPa) . Resolution
 - of the thermomechanical problem (MECA_STATIQUE) . Using
 - CALC_CHAMP , computation of the stresses by elements extrapolated with nodes (SIEF_ELNO) and of the equivalent stresses of Von Mises (SIEQ_ELNO) . One
- will print with result format the maximum values of the stress tensor. One
- will print the fields of temperature, displacements, and the stress fields with med format .
One

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

3.4 With

3.4.1 Salomé One

will visualize via modulus VISU the deformed mesh, the isotherms, the stresses (component and modulates). One

will visualize via modulus PARAVIS the deformed mesh (WarpByVector filter) , the isotherms (filter Contour) , the stresses (filter Integrations Points/ELNO Points) With

3.4.2 the explorer of results Stanley Using

this explorer, one will be able to trace:

- deformed shape, by selecting field DEPL ,
- the isovaleurs of the von Mises stresses to the nodes,
- the change of the temperature according to time at the points and P_{i1} . P_{e1} With

3.4.3 Code_Aster in poursuite

the main steps of the second **execution** with Aster will be : Creation

- of a new command file starting with POURSUITE ; One
- will extract from the values component SIYY of field SIEF_ELNO for an azimuth on the level of the entry of elbow for the loading case corresponding to. F_y The azimuth is defined by the path of ends and $(0.18, 3, 0.)$. $(0.2, 3, .0.)$ One will be able to use for that command MACR_LIGN_COUPE, and the printing of the array produced with format XMGRACE .
Modelization

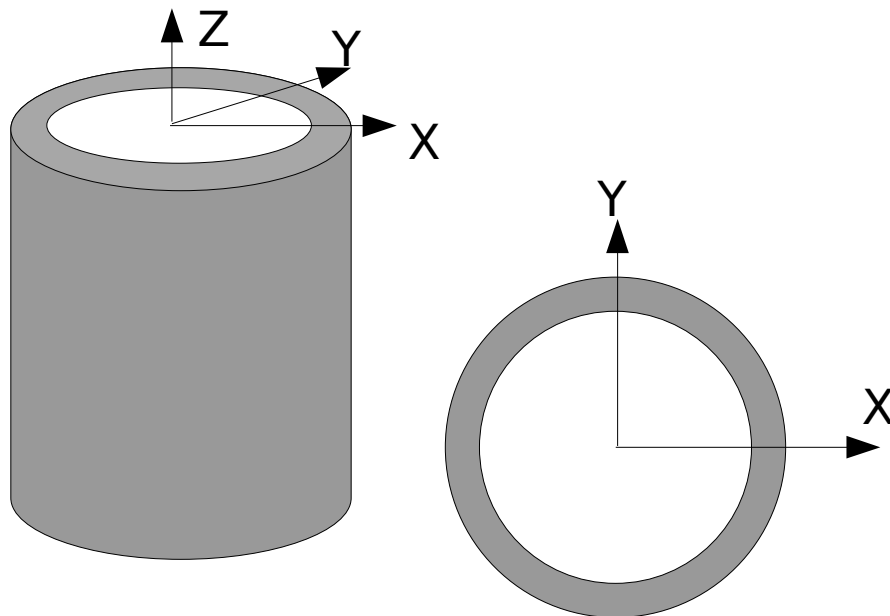
4 B Glossary

: 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 a 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.

The pipework 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

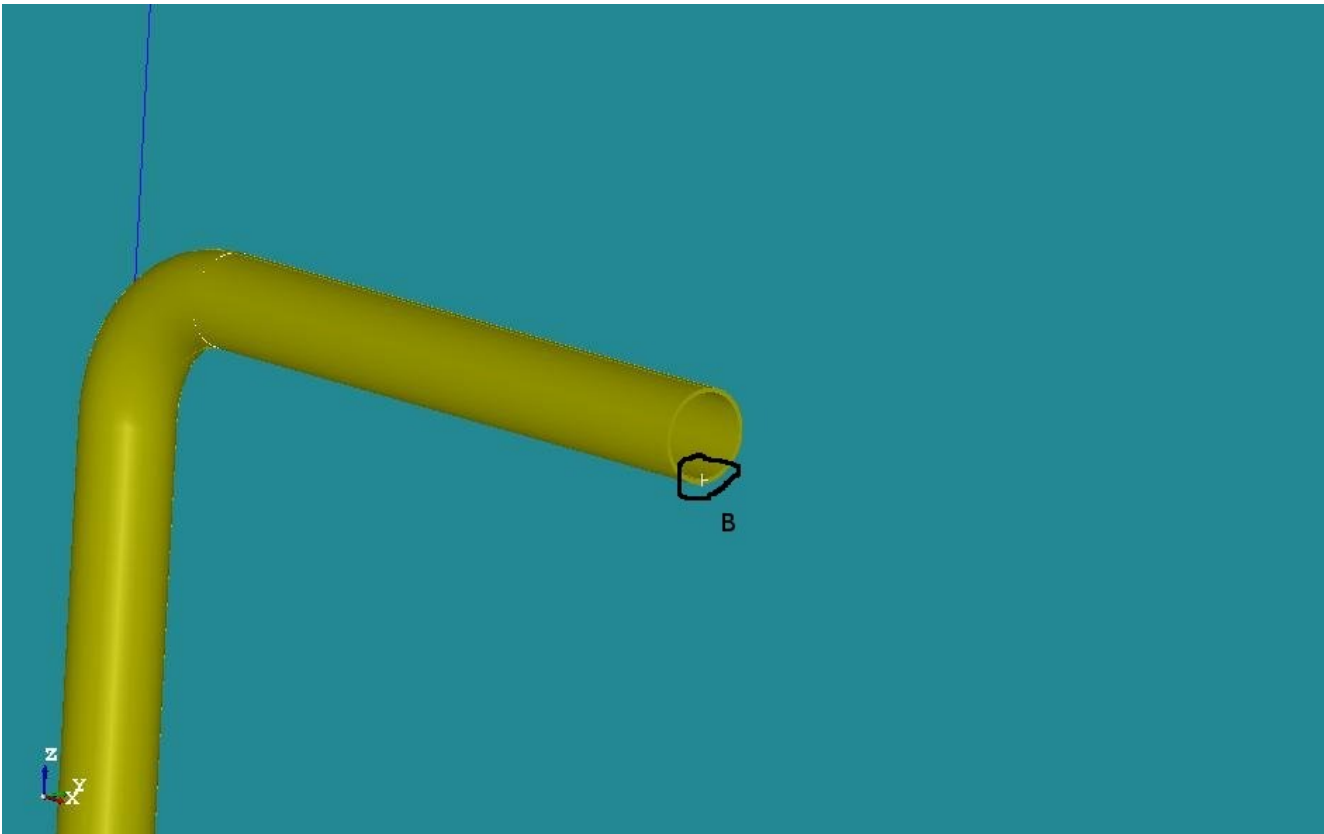
- of a basic surface according to a path: Creation
 1. of the circles interior and external of the base, then contour (wire) and surfaces it (object of the type "Face ") delimited by the two circles. Creation
 2. of the telegraphic path: points A C D , B then two lines and AC , DB and the arc of a circle. CD This
 3. surface is then extruded by Generation /Extrusion along has path. Use
- of basic voluminal objects: To build

- 1.a right pipe of axis, Z using two elementary cylinders height 3 and radius and 0.2 , 0.18 and of a Boolean operation (operation/boolean/cut) . To re-elect the object created in. $TUY1$ To apply
- 2.a rotation to around $TUY1$ so that X it is of axis. Y To select the upper 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 passing Z 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 mesh 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 mesh, it is necessary to check that the geometry is topologically equivalent to a hexahedron (6 sides). 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 planes of the sections entered and output of elbow in order to obtain a geometry topologically equivalent to a hexahedron (6 sides). If one wants to make a mesh 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 mesh generators. Basepression



of these functions to format TABLEAU (IMPR_FONCTION) .EfondX

of the groups: One

will create then the groups of sides 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 of the thickness () dl and another group of edge with an edge of circumference () $L1$ 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 located B on the section, B below, on the interior surface of the pipe. Characteristics

4.1 of the modelization The mesh

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 the edges , under-meshes will be created:

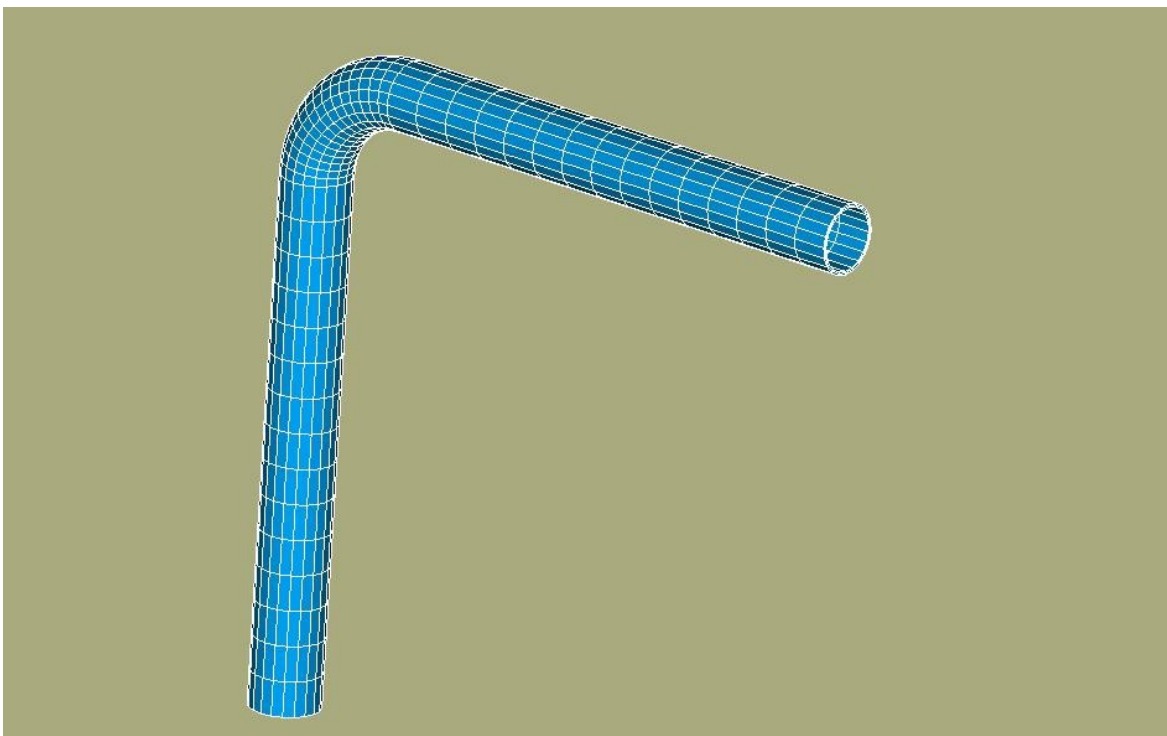
- a sub_mesh defining number of elements in the thickness, for example 2 elements on. dI
- A sub_meshes defining the number of elements on the circumference for example 10 elements on. LI To calculate

the mesh by compute .

To obtain a better accuracy, one will make pass the mesh of linear to quadratic, thanks to the tools "Modification - > Convert to/from quadratic". Most curious can of results compare the differences between the two element types. Note:

With

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



4.2

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.

the main steps of computation with Code_Aster will be : Preparation

•of the data and modal analysis 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 the fixed support on the basis. The first five modes of the pipe will be calculated. One

will launch then computation in modulus SALOME Code_Aster . One will be able to visualize the modes in SALOME by “deforming them”. One can also “animate them” thanks to the Sweep function . It makes it possible to better understand the modal deformed shape. In PARAVIS, this animation is done directly via the macro `fashions` . One selects the mode to be visualized then in the ScaleVector object of the shaft. Transient analysis

•For

the transient analysis, one will leave the command file created by the “wizard” of SALOME. One will add to it the commands necessary following the analysis. Construction

of the specific force Definition

- of the load “forces on the section” B (`AFFE_CHAR_MECA/FORCE_FACE`) . Computation
- 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 Projection modal base

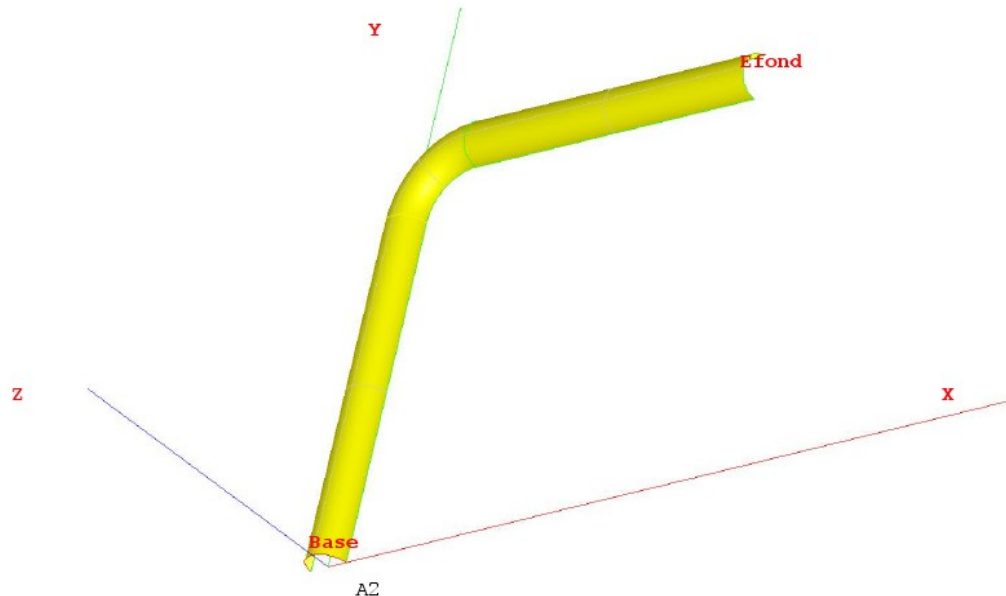
- of the problem assembled on the basis as of eigen modes (`PROJ_BASE`) . Transient computation
- by modal recombination (`DYNA_VIBRA` (`TYPE_CALCUL=' TRAN', BASE_CALCUL=' GENE'`) . Recovery
- of displacements in at Y the point (`RECEIVED B_FONCTION`) . Printing
- of these functions to format `TABLEAU` (`IMPR_FONCTION`) . Post

4.3 - processing One

will import in SALOME the table of displacements according to time and one will make build with SALOME the curve of displacements at the point. B Modelization

5 C In

the case of the modelization in elements shells, the mesh consists of the discretization of the mean surface of the pipework. The geometry being symmetric compared to the plane, (A, X, Y) only one half-surface is netted. One



can create this geometry by defining the two points and $A1$, $A2$ then the arc of a circle. *Base* It is then enough to create the first right pipe by *AC* Generation / Extrusion . To create the elbow, it is necessary to recover the end of the pipe by *AC* 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* Characteristics

5.1 of the mesh The mesh

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 the edges , under-meshes will be created:

- a sub _mesh defining number of elements in the thickness, for example 2 elements on. $d\bar{l}$
- A sub _mesh defining the number of elements on the circumference for example 10 elements on. *LI* Commands

5.2 Code_Aster

the main steps of computation with Code_Aster , introduced into the command file via Eficas, are:
Reading

- of mesh (LIRE_MALLAGE) to med format. Definition
- of the finite elements used (AFFE_MODELE) . The right pipes and the elbow will be modelled by shell elements (DKT) . Reorientations
- of the norms to the elements: one will use MODI _MALLAGE/ORIE_NORM_COQUE in the same way to direct all the elements, with a norm turned towards the interior of the pipe (being given sign convention 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 structure. Assignment
- of the characteristics of the elements shells (AFFE_CARA_ELEM/COQUE) to define the thickness. Definition
- of the boundary conditions and loadings (AFFE_CHAR_MECA) . There is a fixed support on the mesh group and *Base* of *Efond* the conditions of symmetry (normal displacement no one *DZ* and rotations and *DRX* null *DRY*) on the mesh group. *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. Printing
- displacements and the forced with med format (IMPR_RESU) . Modelization

6 D One

can create the geometry by defining the points, A, C, D, B then the two lines and AC, DB and the arc of 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
- way simpler by means of the “sketcher ” After

having built the group of 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 Characteristics

6.1 of the mesh The mesh

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 . Aster commands

6.2

the main steps of computation with Code_Aster , introduced into the command file via Efficas, are:
Reading

- of the mesh to med format. 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 loading (AFFE_CHAR_MECA) . The point is A clamped. Definition
- of the matrixes of the elastic problem (ASSEMBLY) . Computation
- of the first 5 eigen modes (MODE_ITER_SIMULT) . Printing
- of eigen modes (IMPR_RESU) : one will print the mesh and the modes with med format the Transient analysis
- Construction

of the specific force Definition

- of the load “forces at the point” B (AFFE_CHAR_MECA/FORCE_NODALE) . Computation
- 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 Projection modal base

- of the problem assembled on the basis as of eigen modes (PROJ_BASE) . Transient computation
- by modal recombination (DYNA_VIBRA) . Recovery
- of displacements in of Y (RECEIVED B _FONCTION) . Printing
- of these functions to format TABLEAU (IMPR_FONCTION) .