

Note of use of the elements TUYAU_*

Summary:

This document is a note of use for modelings TUYAU_3M and TUYAU_6M.

Finite elements TUYAU_3M and TUYAU_6M correspond to linear elements of right piping or curve. The kinematics of the elements PIPE combine at the same time a kinematics of beam, which describes the overall movement of the line of piping, and a kinematics of hull, which brings the description of swelling, the ovalization and the warping of the cross section.

These modelings are usable for problems of relatively thick three-dimensional pipings, in linear mechanical analysis or not linear and small displacements.

Contents

1 Introduction.....	4
2 Capacities of modeling.....	5
2.1 Recall of the formulation.....	5
2.1.1 Geometry of the elements pipes.....	5
2.1.2 Formulation of the elements pipes.....	5
2.2 Comparison with other elements.....	7
2.2.1 Differences between the elements pipes.....	7
2.2.2 Differences between the elements pipes and the elements beams.....	7
3 Description of the command sets.....	9
3.1 Assignment of a modeling and space discretization.....	9
3.1.1 Degrees of freedom.....	9
3.1.2 Mesh support of the matrices of rigidity.....	9
3.1.3 Mesh support of the loadings.....	10
3.1.4 Model: AFFE_MODELE.....	10
3.2 Elementary characteristics: AFFE_CARA_ELEM.....	10
3.2.1 Operand MODI_METRIQUE.....	11
3.2.2 Generator and concept of local reference mark : keyword ORIENTATION.....	11
3.2.3 Example of assignment of characteristic.....	12
3.3 Materials: DEFI_MATERIAU.....	13
3.4 Limiting loadings and conditions: AFFE_CHAR_MECA and AFFE_CHAR_MECA_F.....	13
3.4.1 List of the keyword factors ofAFFE_CHAR_MECA and AFFE_CHAR_MECA_F.....	14
3.4.2 Application of an internal pressure: keyword FORCE TUYAU.....	15
3.4.3 Application of a force distributed: keyword FORCE_POUTRE.....	15
3.4.4 Application of gravity: keyword GRAVITY (AFFE_CHAR_MECA only).....	16
3.4.5 Connections hull-pipes, 3D-pipe and pipe-beams: keyword LIAISON_ELEM.....	16
3.4.6 Limiting conditions: keywords DDL_IMPO and LIAISON_*.....	17
3.5 Application of a thermal dilation.....	18
4 Resolution.....	19
4.1 Linear calculations: MECA_STATIQUE and other linear operators.....	19
4.2 Nonlinear calculations: STAT_NON_LINE and DYNA_NON_LINE.....	19
4.2.1 Behaviors and assumptions of deformations available.....	19
4.2.2 Details on the points of integration.....	20
4.3 Dynamic calculations.....	20
5 Additional calculations and postprocessings.....	22
5.1 Elementary calculations of matrices: operator CALC_MATR_ELEM.....	22
5.2 Calculations by elements: operator CALC_CHAMP.....	22
5.3 Calculations with the nodes: operator CALC_CHAMP.....	23
5.4 Calculations of quantities on whole or part of the structure: operator POST_ELEM.....	23

5.5 Values of components of fields of sizes: operator POST_RELEVE_T.....	23
6 Examples.....	25
6.1 Linear static analysis.....	25
6.2 Static analysis nonlinear material.....	25
6.3 Modal analysis in dynamics.....	26
6.4 Nonlinear dynamic analysis.....	27
7 Bibliographical references.....	28

1 Introduction

Finite elements `TUYAU_3M` and `TUYAU_6M` correspond to linear elements of right piping or curve. They are based on a kinematics of beam of Timoshenko for displacements and rotations of average fibre and on a kinematics of hull for the deformations of the transverse section (ovalization, warping, swelling). These transverse deformations are broken up into Fourier series. Modeling `TUYAU_3M` takes into account 3 modes to the maximum, while modeling `TUYAU_6M` takes into account 6 modes of Fourier.

These modelings are usable for problems of relatively thick three-dimensional pipings, only in linear mechanical analysis or not linear and small displacements. Currently, no calculation of thermics or acoustics is possible.

This document presents the possibilities of modeling `PIPE` available in version 6 of `Code_Aster`. The possibilities of this kind of modeling are initially presented, then one briefly points out the formulation of the finite elements and their differences with modelings beam. One also gives the list of the options available for each element. One finishes by the presentation of some academic CAS-tests and finally some advices of use are given.

The right or curved pipe sections are gathered under modelings `TUYAU_3M` and `TUYAU_6M`. The options of calculations are defined in this document. The current possibilities of these elements pipes are the following ones:

right or curved lines of piping,
linear element with 3 nodes (SEG3) or 4 nodes (SEG4),
relatively thick pipe: $e/R < 0.2$ where e represent the thickness and R the ray of the transverse section,
internal pressure, cross-bendings and anti-plane, torsion and extension,
small displacements,
elastoplastic in constraints plane, or not linear behavior incremental unspecified,
the transverse section can become deformed by: <ul style="list-style-type: none">• swelling due to the internal pressure or the effect Fish,• ovalization due to the inflection,• warping due to the inflections combined in the plan and except plan.

Compared with modeling `TUYAU_3M`, modeling `TUYAU_6M` a better approximation of the behavior of the cross section allows if this one becomes deformed according to a raised mode, for example in the case of thin tubes where the thickness report on ray of the cross section is < 0.1 , and in the case of plasticity.

Modeling `TUYAU_3M` have 21 degrees of freedom per node (6 degrees of freedom of beam and 15 degrees of freedom of hull), while modeling `TUYAU_6M` have 39 degrees of freedom per node (6 degrees of freedom of beam and 33 degrees of freedom of hull).

For modeling `TUYAU_3M`, one can use meshes `SEG3` and `SEG4`.

2 Capacities of modeling

2.1 Recall of the formulation

2.1.1 Geometry of the elements pipes

We point out here the methods and the modelings put in work for the elements pipes and which are presented in the reference document [R3.08.06].

For the elements pipes one defines an average, right fibre or curve (x defines the curvilinear coordinate) and a hollow section of circular type. This section must be small compared to the length of piping. The figure [Figure 2.1.1-a] illustrates the two various configurations. A local reference mark $oxyz$ is associated with average fibre.

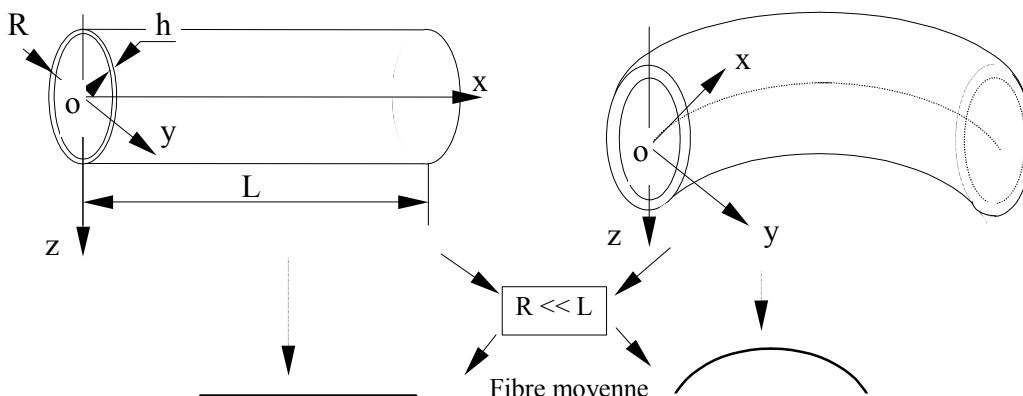


Figure 2.1.1-a : Right pipe or curve.

2.1.2 Formulation of the elements pipes

The kinematics of the pipe [Figure 2.1.1-a] is composed of a kinematics of hull which brings the description of ovalization, swelling and warping, and of a kinematics of beam which describes the overall movement of the line of piping. Displacement U [Figure 2.1.2-b] of a material point of the pipe of a macroscopic part beam is composed (U^P) and of an additional part local hull (U^S):

$$U = U^P + U^S$$

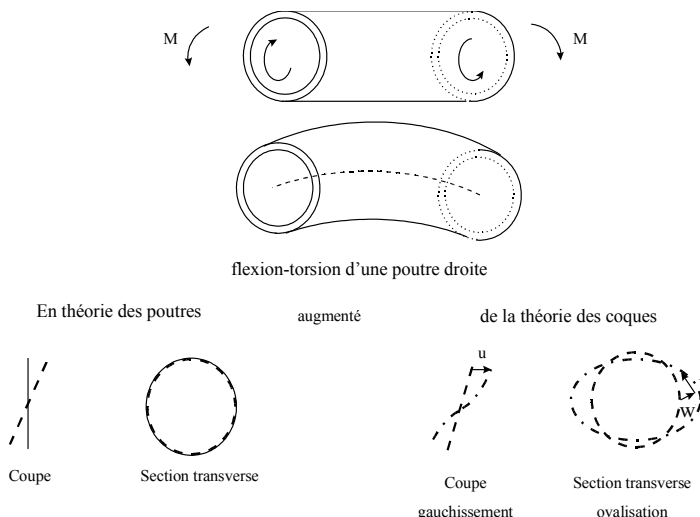


Figure 2.1.2-a : Decomposition of displacement in fields of beam and hull.

The formulation of the elements rests on:

- **theory of the beams** for the kinematics of average fibre. If one makes the complete assumption of the theory of the beams: cross-sections associated with displacements of beam (U^P), which is perpendicular to **average fibre** of reference [Figure 2.1.2-b] remain perpendicular to average fibre after deformation. The cross-section does not become deformed. This will be true in average in

the element PIPE. One uses the theory of the beams only to describe the movement of average fibre: the average fibre of the pipe is equivalent to average fibre of a beam. This kinematics makes it possible to describe the overall movement of the line of piping.

- **theory of the hulls** to describe the transverse deformation of the sections around average fibre. Kinematics of the transverse sections: the cross-sections which are perpendicular to **average surface** of reference remain right. The material points located on the normal at not deformed average surface remain on a line in the deformed configuration. The formulation used is a formulation of the type LOVE-KIRCHHOFF without transverse shearing for the description of the behavior of the transverse sections. The thickness of the hull remains constant. Average surface ω pipe, located at mid thickness, is equivalent to the average surface of a hull. This kinematics of hull brings the description of swelling, the ovalization and the warping of the cross section.

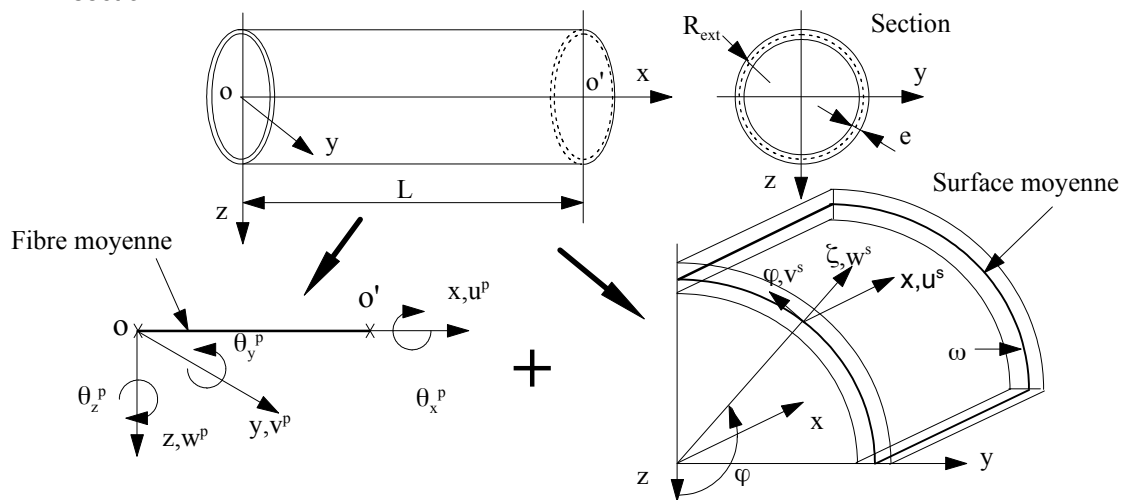


Figure 2.1.2-b : Fibre and average surface in the case of a right pipe .

Additional displacements (U^s) surface of the pipe are approximated by a series of Fourier until the order M ($M=3$ for modeling TUYAU_3M and $M=6$ for modeling TUYAU_6M).

$$u^s(x, \Phi) = \sum_{m=2}^M u_m^i(x) \cos m\Phi + \sum_{m=2}^M u_m^0(x) \sin m\Phi$$

$$v^s(x, \Phi) = w_1^i(x) \sin \Phi + \sum_{m=2}^M v_m^i(x) \sin m\Phi - w_1^0(x) \cos \Phi + \sum_{m=2}^M v_m^0(x) \cos m\Phi$$

$$w^s(x, \Phi) = w^0 + \sum_{m=2}^M w_m^i(x) \cos m\Phi + \sum_{m=2}^M w_m^0(x) \sin m\Phi$$

Where

- u^s : represent the axial displacement of average surface in the local reference mark $x \Phi \zeta$
- v^s : represent the ortho-radial displacement of average surface in the local reference mark $x \Phi \zeta$
- w^s : represent the radial displacement of average surface in the local reference mark $x \Phi \zeta$
- w^0 : represent swelling

These elements thus utilize locally:

- 6 variable kinematics for the beam formulation: displacements u^p , v^p and w^p according to fibre of reference and rotations around the local axes,

- 3 variable kinematics for the hull formulation: additional displacements u^s , v^s and w^s in the reference mark of average surface,
- 4 constraints in the thickness of the pipe noted $SIXX$ (sxx), $SIYY$ (sff), $SIXY$ (sxf), and $SIXZ$ (sxz). The constraint $SIZZ$ (szz) is worthless (assumption of plane constraints). Transverse shear stresses are worthless (assumption of Coils Kirchoff),
- 4 deformations in the thickness of the pipe noted $EPXX$ (exx), $EPYY$ (eff), $EPXY$ (exf), and $EPXZ$ (sxz). Deformation $EPZZ$ (ezz) is worthless for the beam part.

Notice important :

The kinematics of beam is based on the assumption of Timoshenko [R3.08.03]. The element pipe is not "exact" with the nodes for loadings or torques concentrated at the ends, it is necessary to net with several elements to get correct results.

According to average fibre, these elements are of isoparametric type. It results from it that displacements vary like polynomials of order 2 following X for the elements with 3 nodes and of order 3 per 4 nodes.

2.2 Comparison with other elements

2.2.1 Differences between the elements pipes

The elements pipes TUYAU_3M and TUYAU_6M are linear elements:

- TUYAU_3M to three or four nodes.
- TUYAU_6M with four nodes.

These elements are different only on the level from the approximation from the additional field of displacement HULL, which is made by a decomposition in Fourier series:

- TUYAU_3M until order 3.
- TUYAU_6M until order 6.

Consequently the number of degrees of freedom is different:

- TUYAU_3M 21 per node (6 degrees of freedom of beam and 15 degrees of freedom of hull)
- TUYAU_6M 39 per node (6 degrees of freedom of beam and 33 degrees of freedom of hull)

Compared with modeling TUYAU_3M, modeling TUYAU_6M a better approximation of the behavior of the cross section allows if this one becomes deformed according to a raised mode, for example in the case of thin tubes where the thickness report on ray of the cross section is < 0.1 , and in certain cases in plasticity. In the little requested zones, modeling TUYAU_3M is recommended.

Finite elements of right pipe or curve

It appears a difference in behavior on the elbows present in the structure between the grids quadratic curves and rights. The node medium of the elements is not found on the line of curve of the element continuation, for example, with a standard operation CREA_MALLAGE/LINE_QUAD.

This difference strongly modifies the characteristics of stiffness of the element and also the catch in account of the internal pressures (basic effect); it is consequently advised to make so that the grid respect to the maximum the curve of the elbow.

2.2.2 Differences between the elements pipes and the elements beams

Following the example of the finite elements PIPE, finite elements BEAM are also part of the class of the linear finite elements. One compares in this part the applicable formulations and loadings for these two classes of elements.

On the level of formulation :

- **Element BEAM :**

The formulation is based on an exact resolution of the equations of the continuous model carried out for each element of the grid. Several types of elements of beam are available:

- POU_D_E : transverse shearing is neglected, as well as the inertia of rotation. This assumption is checked for strong twinges (Assumption of Euler),
- POU_D_T : transverse shearing and all the terms of inertia are taken into account. This assumption is to be used for weak twinges (Assumption of Timoshenko).

These elements use meshes of the type `SEG2` with 6 degrees of freedom by nodes, 3 displacements and three rotations. The formulation of these elements is presented in the reference document [R3.08.01]. The section is constant, the only possible behavior of the transverse sections is the translation and rotation for the whole of the points of the section. Section perhaps of an unspecified form the constant or variable over the length.

- **Element PIPE :**

The formulation combines at the same time a beam formulation based on the assumption of Timoshenko and a hull formulation based on the assumption of Love_Kirchhoff making it possible to model the phenomena of swelling, ovalization and warping. The hollow section, of circular form, is constant over the entire length of the element. The element is not "exact" with the nodes for loadings or torques concentrated at the ends, it is thus necessary to net with several elements to get correct results, in particular to represent the curve.

These elements use meshes of the type `SEG3` or `SEG4` with, for the kinematics of beam 6 degrees of freedom by nodes, 3 displacements and three rotations, and for the kinematics from hull, 15 or 33 degrees of freedom of type displacement.

On the level of **applicable loadings** :

- **Element BEAM :**

The possible loadings are the loadings of extension, inflection and torsion. The internal loading of pressure for the hollow sections does not exist (the section is indeformable).

- **Element PIPE :**

The element `PIPE` admits the classical loadings of beam as well as the application of an internal pressure.

3 Description of the command sets

3.1 Assignment of a modeling and space discretization

In this part, one describes the choice and the assignment of one of two modelings `PIPE` as well as the degrees of freedom and the associated meshes. Most described information are extracted from the documentation of use [U3.11.06]: Modelings `TUYAU_3M` and `TUYAU_6M`.

3.1.1 Degrees of freedom

The degrees of freedom are, in each node of the mesh support:

- six components of displacement of average fibre (three translations and three rotations),
- three degrees of freedom corresponding to modes 0 and 1,
- for each mode of Fourier, 6 degrees of freedom (U corresponds to warping, V and W with ovalization: V with orthoradial displacement, W with radial displacement, I mean "in plane" and O mean "out of plane").

Element	Degrees of freedom to each node top						Remarks
TUYAU_3M	DX	DY	DZ				Value of the component of displacement in translation imposed on the specified nodes
	DRX	DRY MARTI NI	DRZ				Value of the component of displacement in rotation imposed on the specified nodes
	W0	WI1	WO1				Degree of freedom of swelling and mode 1 on W
	UI2	VI2	WI2	UO2	VO2	WO2	Degrees of freedom related to mode 2
	UI3	VI3	WI3	UO3	VO3	WO3	Degrees of freedom related to mode 3
TUYAU_6M	DX	DY	DZ				Value of the component of displacement in imposed translation
	DRX	DRY MARTI NI	DRZ				Value of the component of displacement in imposed rotation
	W0	WI1	WO1				degrees of freedom of swelling and mode 1 on W
	UI2	VI2	WI2	UO2	VO2	WO2	degrees of freedom related to mode 2
	UI3	VI3	WI3	UO3	VO3	WO3	degrees of freedom related to mode 3
	UI4	VI4	WI4	UO4	VO4	WO4	degrees of freedom related to mode 4
	UI5	VI5	WI5	UO5	VO5	WO5	degrees of freedom related to mode 5
	UI6	VI6	WI6	UO6	VO6	WO6	degrees of freedom related to mode 6

3.1.2 Mesh support of the matrices of rigidity

The meshes support of the finite elements, in displacement formulation, are segments with 3 or 4 nodes.

Modeling	Mesh	Finite element	Remarks
TUYAU_3M	SEG3	METUSEG3	Linear mesh
	SEG4	MET3SEG4	Linear mesh
TUYAU_6M	SEG3	MET6SEG3	Linear mesh

Meshes `SEG4`, which has cubic functions of forms, were developed to solve a simple problem of beam in inflection. For this simple example, the exact solution is obtained using one only element with mesh `SEG4`.

For more complex problems, the experiment shows that one can net much more coarsely with meshes `SEG4`. For example about fifteen elements are needed `SEG3` to obtain a correct solution for an elbow in inflection whereas one needs half with elements of it `SEG4`.

Note:

| One can use the operator `MODI_MALLAGE` to build meshes `SEG4` starting from meshes `SEG3`.

3.1.3 Mesh support of the loadings

All the loadings applicable to the elements used are treated by direct discretization on the mesh support of the element in displacement formulation. The linear pressure and the other forces as well as gravity are examples of loadings applying directly to the element. No special mesh of loading is thus necessary.

3.1.4 Model: `AFFE_MODELE`

The assignment of modeling passes through the operator `AFFE_MODELE` [U4.41.01]. It is pointed out that only the mechanical phenomenon is available with the element `PIPE`.

<code>AFFE_MODELE</code>		<code>TUYAU_3M</code> <code>TUYAU_6M</code>	Remarks
<code>AFFE</code>		•	
<code>PHENOMENON</code>	<code>'MECHANICAL'</code>	•	
<code>MODELING</code>	<code>'TUYAU_3M'</code>	•	
<code>MODELING</code>	<code>'TUYAU_6M'</code>	•	

On the level of the choice of modeling `PIPE`, one can note that the use of a decomposition in Fourier series to order 6 (element `TUYAU_6M`) in the case of improve the approximation of the behavior of the cross section in the presence of local modes, if this one becomes deformed according to a raised mode, for example thin tubes where the thickness report on ray of the cross section is < 0.1 , and in the case of plasticity.

3.2 Elementary characteristics: `AFFE_CARA_ELEM`

In this part, the operands characteristic of the element pipe are described. The documentation of use of the operator `AFFE_CARA_ELEM` is [U4.42.01].

<code>AFFE_CARA_ELEM</code>		<code>TUYAU_3M</code> <code>TUYAU_6M</code>	Remarks
<code>BEAM</code>		•	
<code>SECTION:</code>	<code>'CIRCLE'</code>	•	
<code>% section</code>	<code>constant</code>	•	
<code>% section</code>	<code>Variable</code>	•	
	<code>MODI_METRIQUE</code>	•	
<code>TUYAU_NCOU</code>	<code>tncouch</code>	•	
<code>TUYAU_NSEC</code>	<code>tsec</code>	•	
<code>ORIENTATION</code>	<code>'GENE_TUYAU'</code>	•	Definition of a generator. By default, a generator is created
	<code>PRECISION</code>	•	
	<code>CRITERION</code>	•	

The characteristics which it is possible to affect on the elements `PIPE`, are:

- `SECTION: 'CIRCLE'`

The section is defined by its ray `'R'` external and its thickness `'EP'`, on each mesh since the grid is represented by average fibre of the pipe.

•TUYAU_NCOU: `tncouch`

It is the number of layers to be used for the integration of the nonlinear relations of behavior in the thickness of the right pipe sections. In linear elasticity, one to two layers are enough, into nonlinear one advises to put between 3 and 5 layers. The number of point of Gauss is equal to twice the number of layers plus one ($2 \times tncouch + 1$), with the result that time CPU increases quickly with the number of the layers.

•TUYAU_NSEC: `tnsec`

It is the number of angular sectors to use for the integration of the nonlinear relations of behavior in the circumference of the right pipe sections. By default the number of sectors 16 are worth. One advises to put 32 sectors into nonlinear for precise results (attention with the increase in time CPU with the number of sectors).

•ORIENTATION ('GENE_TUYAU')

One defines from one of the nodes ends of the line of piping a continuous line traced on the pipe. Operands `PRECISION` and `CRITERION` allow to define the precision for the construction of the generator and the limit between a right pipe section and a curved element.

Note:

The directing vector of the line thus defined should not be colinéaire with average fibre of the elbow to the node end considered, by using the keyword `INFO=2` one can check if the definite vector is correct.

3.2.1 Operand `MODI_METRIQUE`

The operand `MODI_METRIQUE` allows to define for the elements `PIPE` the type of integration in the thickness:

- `MODI_METRIQUE = 'NOT'` conduit to assimilate in integrations the ray to the average radius. This is thus valid for the pipes low thickness (compared to ray),
- `MODI_METRIQUE = 'YES'` imply a complete, more precise integration for thick pipings, but being able in certain cases to lead to oscillations of the solution.

3.2.2 Generator and concept of local reference mark : keyword `ORIENTATION`

The generator traced throughout piping makes it possible to define the origin of the angles Φ [Figure 2.1.2-b]. This is used:

- to interpret the degrees of freedom of ovalization;

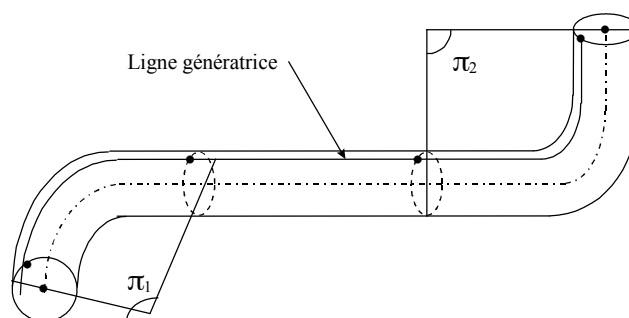


Figure 3.2.2-a : Representation of two noncoplanar elbows connected by a right pipe.

For a transverse section end of the line of piping [Figure 3.2.2-b], the user defines in `AFFE_CARA_ELEM` under the keyword `ORIENTATION` a vector whose projection on the transverse section end defines a vector origin z_1 unit.

Syntax is the following one:

```
AFFE_CARA_ELEM (...
    ORIENTATION = _F (GROUP_NO = 'END',
                     CARA = 'GENE_TUYAU'
                     VALE = (X, there, Z))
```

where: END centers transverse section end is the node.

(x, y, z) contains the 3 components of the vector directing the generator of the pipe, with to project on the transverse section end. This vector must be defined in a node or one group_no end of the pipe. The geometry is then built automatically for all the related elements of PIPE.

The intersection between the direction of this vector and the average surface of the elbow determines the trace of the generator on this section. One calls x_1, y_1, z_1 the direct trihedron associated with this section where x_1 is the unit vector perpendicular to the transverse section. The intersection enters transverse section and line resulting from the center of this section directed by z_k is the trace of a generator represented below. For the whole of the other transverse sections, the trihedron x_k, y_k, z_k is obtained either by rotation of the trihedron

$x_{k-1}, y_{k-1}, z_{k-1}$ in the case of the bent parts, is by translation of the trihedron $x_{k-1}, y_{k-1}, z_{k-1}$ for the right parts of piping.

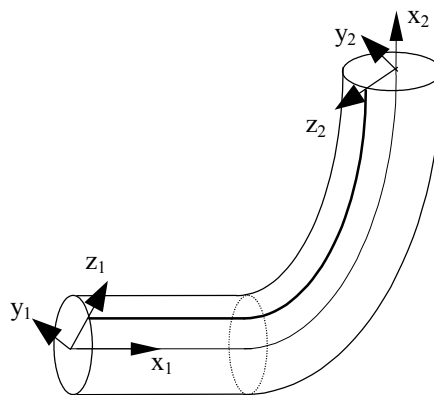


Figure 3.2.2-b : Representation of the generator of reference.

The origin of Φ commune with all the elements is defined compared to the trace of this generator on the transverse section. The angle between the trace of the generator and the current position on the transverse section is located by the angle Φ . The local reference mark of the right and bent pipe is thus defined by the option ORIENTATION ('GENE_TUYAU') order AFFE_CARA_ELEM who allows to define the first vector z_k at an end.

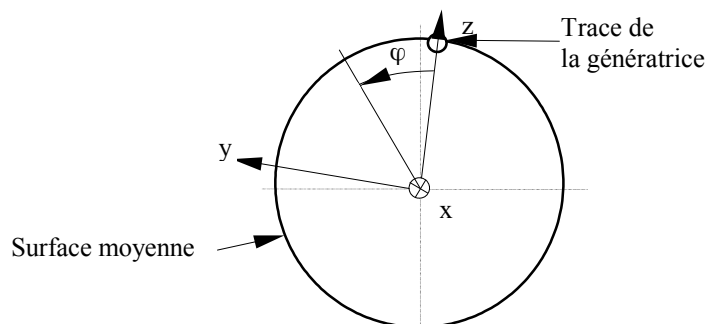
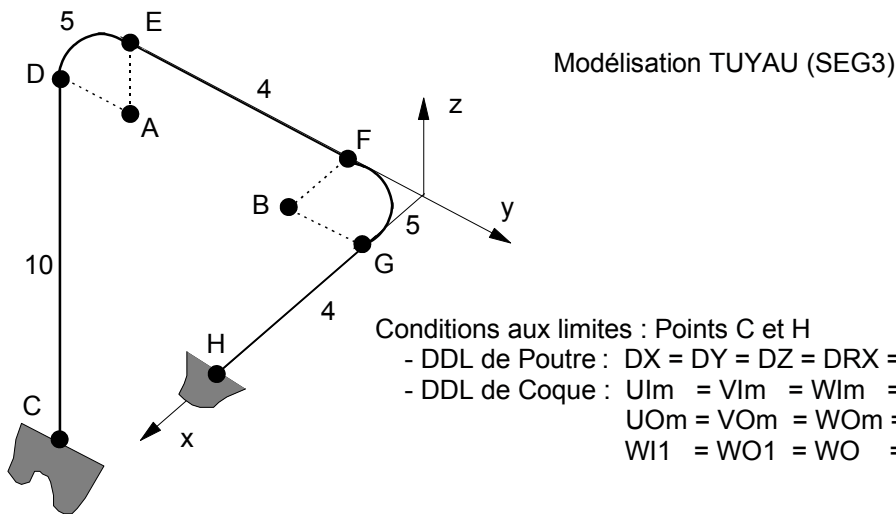


Figure 3.2.2-c : Local reference mark of the element XYZ .

3.2.3 Example of assignment of characteristic

This example is a piping comprising two elbows (problem of Hoovgaard resulting from test SLL101C).



- diameter external of the pipe: 0.185 m
- thickness of the pipe: 6.12 m
- radius of curvature of the elbows: 0.922 m

```

MODELE=AFFE_MODELE (MAILLAGE=MAILLAGE,
                    AFPE=_F (ALL = 'YES',
                              PHENOMENON = 'MECHANICAL',
                              MODELING = 'TUYAU_3M')
                    )

CARELEM=AFPE_CARA_ELEM (MODELE=MODELE,
                        POUTRE=_F (GROUP_MA = 'TOUT_ELT',
                                    SECTION = 'CIRCLE',
                                    CARA = ('R', 'EP',),
                                    VALE = (0.0925, 0.00612,)),
                        ORIENTATION=_F (GROUP_NO = 'IT',
                                         CARA = 'GENE_TUYAU',
                                         VALE = (1., 0., 0.,))
                        )
    
```

3.3 Materials: DEFI MATERIAU

The definition of the behavior of a material is carried out using the operator `DEFI_MATERIAU` [U4.43.01]. There is no particular constraint which had with the use of the elements `PIPE`.

The materials used with the whole of modelings can have elastic behaviors in plane constraints whose linear characteristics are constant or function of the temperature. The nonlinear behaviors in plane constraints are available for modelings pipes. For more information on these nonlinearities one can refer to [§ 2.6].

DEFI MATERIAU	TUYAU_3M	TUYAU_6M	Remarks
ELAS, ELAS_FO, ECRO_LINE, TRACTION,...	•	•	all materials available in C_PLAN

3.4 Limiting loadings and conditions: AFPE_CHAR_MECA and AFPE_CHAR_MECA_F

One points out that it is not possible to carry out thermal calculations, however the assignment of temperature is possible, using the operator `CREA_CHAMP`. (see paragraph [§3.4.4]).

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 assignment of the loadings and the boundary conditions on a mechanical model is carried out using the operator `AFFE_CHAR_MECA`, if the loadings and the boundary conditions mechanical on a system are actual values depending on no parameter, or `AFFE_CHAR_MECA_F`, if these values are functions of the position or the increment of loading.

The documentation of use of `AFFE_CHAR_MECA` and `AFFE_CHAR_MECA_F` is [U4.44.01].

3.4.1 List of the keyword factors of `AFFE_CHAR_MECA` and `AFFE_CHAR_MECA_F`

The keyword factors available for these two operators are gathered in the two following tables.

<code>AFFE_CHAR_MECA</code>	<code>TUYAU_3M</code> <code>TUYAU_6M</code>	Goal, remarks and examples
<code>DDL_IMPO</code>	•	Goal : to impose, with nodes or groups of nodes, one or values of displacement Mode 0 (swelling) and: •modes 1 to 3 for <code>TUYAU_3M</code> •modes 1 to 6 for <code>TUYAU_6M</code> Example : <code>SDLL14</code> , <code>SLL101</code> , <code>SSLX102</code> , <code>SSNL106</code> ,...
<code>LIAISON_DDL</code>	•	Goal : to define a linear relation between degrees of freedom of two or several nodes
<code>LIAISON_OBLIQUE</code>	•	Goal : to apply, with nodes or groups of nodes, the same component value of displacement definite per component in an unspecified oblique reference mark
<code>LIAISON_GROUP</code>	•	Goal : to define linear relations between certain degrees of freedom of couples of nodes, these couples of nodes being obtained while putting in opposite two lists of meshes or nodes
<code>LIAISON_UNIF</code>	•	Goal : to impose the same value (unknown) on degrees of freedom of a set of nodes Example : <code>ELSA01B</code> , <code>ELSA01C</code> and <code>ELSA01D</code>
<code>LIAISON_SOLIDE</code>	•	Goal : to model an indeformable part of a structure Example : <code>ELSA01B</code> , <code>ELSA01C</code> and <code>ELSA01D</code>
<code>LIAISON_ELEM</code>	•	Goal : to model the connections of a massive part 3D with a pipe part or of a hull part with a pipe part Example : <code>SSLX101B</code> , <code>SSLX102A</code> and <code>SSLX102F</code>
<code>LIAISON_CHAMNO</code>	•	Goal : to define a linear relation between all <code>ddl</code> s present in a concept <code>CHAM_NO</code>
<code>GRAVITY</code>	•	Goal : to apply an effect of gravity Example : <code>SLL101</code> , <code>SLL106</code>
<code>FORCE_POUTRE</code>	•	Goal : to apply linear forces, to elements of type beam Example : <code>SLL106</code>
<code>FORCE_NODALE</code>	•	Goal : to apply, with nodes or groups of nodes, nodal forces, definite component by component in the reference mark <code>TOTAL</code> or in an oblique reference mark defined by 3 nautical angles Example : <code>SLL106</code> ,...
<code>FORCE TUYAU</code>	•	Goal : to apply, with elements or groups of elements of type pipe an internal pressure Example : <code>SLL106</code> , <code>SSNL117</code> , <code>SSNL503</code>

AFFE_CHAR_MECA_F	TUYAU_3M TUYAU_6M	Remarks
DDL IMPO	•	See above
LIAISON DDL	•	See above
LIAISON OBLIQUE	•	See above
LIAISON GROUP	•	See above
LIAISON UNIF	•	See above
LIAISON SOLIDE	•	See above
FORCE POUTRE	•	See above
FORCE NODALE	•	See above
FORCE TUYAU	•	See above

3.4.2 Application of an internal pressure: keyword **FORCE TUYAU**

This keyword factor is usable to apply an internal pressure to elements pipe, defined by one or more meshes or of the groups of meshes. The pressure is applied to the level of the internal ray, as in 3D.

Syntax to apply this loading is pointed out below:

```

•AFFE_CHAR_MECA (...
    |   FORCE TUYAU =_F (   ◆ / ALL   = 'YES',
                          / MESH   = lma,           [l_maille]
                          / GROUP_MA = lgma,         [l_gr_maille]
                          ◆ NEAR    = p,             [R]
    )

•AFFE_CHAR_MECA_F (...
    |   FORCE TUYAU =_F (   ◆ / ALL   = 'YES',
                          / MESH   = lma,           [l_maille]
                          / GROUP_MA = lgma,         [l_gr_maille]
                          ◆ NEAR    = PF,           [function]
    )

```

The operand available is:

CLOSE = p (PF)

Value of the imposed pressure (real or function of time or the geometry).

p is positive according to the contrary direction of the normal to the element.

This loading applies to the types of meshes and following modelings:

Mesh	Modeling
SEG3, SEG4	'TUYAU_3M'
SEG3	'TUYAU_6M'

Examples of use are available in the base of tests: CAS-tests ELSA01B, SLL106A, SSNL117A and SSNL503A.

3.4.3 Application of a force distributed: keyword **FORCE POUTRE**

This keyword factor is usable to apply forces **linear**, constants according to X, on elements of type beam defined on all the grid or one or more meshes or of the groups of meshes. The forces are defined component by component, that is to say in the reference mark TOTAL, that is to say in the local reference mark of the element defined by the operator AFPE_CARA_ELEM [U4.42.01].

Syntax is available in the documentation of AFPE_CHAR_MECA/AFPE_CHAR_MECA_F [U4.44.01]. This loading applies to the types of meshes and following modelings:

Mesh	Modeling
------	----------

SEG3, SEG4	TUYAU_3M
SEG3	TUYAU_6M

An example of use is available in the base of tests: CAS-test SSL106.

3.4.4 Application of gravity: keyword GRAVITY (AFFE_CHAR_MECA only)

This keyword is used for applied the effect of gravity on piping.

AFFE_CHAR_MECA	TUYAU_3M TUYAU_6M	Remarks
GRAVITY	(G, ap, LP, CP)	• Acceleration and direction of gravity

Example of use of the operand GRAVITY :

```
POI_PROP = AFFE_CHAR_MECA (MODELE=MODELE,
                             PESANTEUR=_F (GRAVITE=9.81,
                                             DIRECTION= (0. , 0. , - 1.)),
                             )
```

3.4.5 Connections hull-pipes, 3D-pipe and pipe-beams: keyword LIAISON_ELEM

It is a question of establishing the connection between a node end of a pipe section and a group of mesh of edge of elements of hulls or elements 3D. This makes it possible to net part of piping (for example an elbow) in hulls or elements 3D and the rest in right pipes. The formulation of the connection hull-pipes and the connection 3D-Pipe is presented in the reference document [R3.08.06]. This connection makes it possible to transmit warping and ovalization means of the grid hull or 3D to the degrees of freedom corresponding of the pipe.

The connection:

- **Hull-pipe** : it makes it possible to connect elements of edge (SEG2, SEG3) hull part with the node of the pipe to be connected. This connection is currently realizable for pipes whose neutral fibre is perpendicular to the normals with the facets of the plates or the hulls. The connection is usable by using the keyword `LIAISON_ELEM = _F (OPTION = 'COQ_TUYAU')` of `AFFE_CHAR_MECA`.
- **Pipe-3D** : it makes it possible to connect elements of edge (TRIA3, QUAD4, TRIA6,...) part 3D with the node of the pipe to be connected. The connection is usable by using the keyword `LIAISON_ELEM = _F (OPTION = '3D_TUYAU')` of `AFFE_CHAR_MECA`.

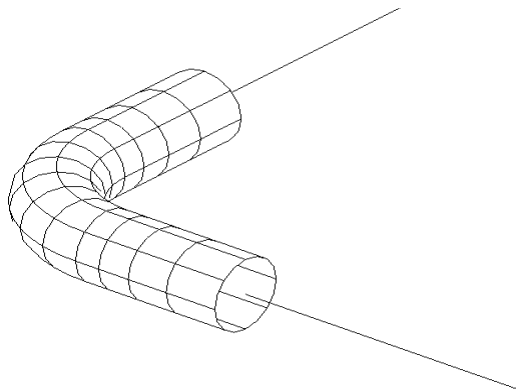


Figure 3.4.5-a : Example of connection between a grid COQUE_3D and PIPE.

The cases which test the connections are presented on the following table.

NAME	MODELING	ELEMENT	Remarks
ELSA01D	POU_D_T DKQ	SEG2	Connection COQ_TUYAU : Definition of the rigid bodies
Doc. V: [V1.10.119]	TUYAU_3M DIS_T DIS_TR	SEG3 POI1	Pipe right and bent in dynamics not inéaire

SSLX101B	DKT PIPE Doc. V: [V3.05.101]	DIS_TR	MEDKQU4 METUSEG3 POI1	Connection COQ_TUYAU : Pipe right modelled in hulls and beams. This test aims to test the connection hull pipe "COQ_TUYAU" in the presence of unit loadings: traction, inflection and of torsion.
SSLX102A	DKT PIPE Doc. V: [V3.05.102]		MEDKQU4 METUSEG3 MEDKQU4 METUSEG3	Connection COQ_TUYAU : Piping bent in inflection.
SSNL106F	COQUE_3D TUYAU_3M Doc. V: [V6.02.106]		QUAD9 SEG3	Connection COQ_TUYAU : at each end of the grid hulls, with an element pipe Right beam in traction and inflection, elastoplastic behavior
SSLX102F	3D PIPE Doc. V: [V3.05.102]		HEXA20 METUSEG3	Connection 3D_TUYAU : Piping bent in inflection: modeling 3D-PIPE, linear relations 3D_PIPE. The elbow is modelled with elements 3D.
ZZZZ112C	COQUE_3D TUYAU_3M Doc. V: [V1.01.112]	POU_D_T	QUAD9 SEG4 SEG3	Connection COQ_TUYAU : a half of the cylinder is modelled in hulls, other half in elements PIPE Cylinder under variable pressure

In all these CAS-tests, the results are satisfactory given that part of the noted variations is ascribable with the fusion of the grid 3D or hulls.

Notice : Connections pipe-beams.

It is a question of establishing a connection between one *node end of a pipe section and a node end of an element of beam*. The pipe formulation comprises a kinematics of type beam identical to the kinematics of the elements beams. There is thus no cut between displacements of type beam (3 displacements and 3 rotations). The average fibre of the beam and the pipe are the same ones. By account, the kinematics of the elements of beam in the case of does not understand kinematics of hull (the section is indeformable) like the elements pipes, there is thus a cut on the level of the deformation of the transverse section.

There does not exist in specific Code_Aster of connection pipe-beam, the connection between these two elements is automatically assured, without intervention of the user, by the node common to the element pipe and the element beam. Nevertheless, some care are to be taken, it is necessary that the transition between the beam and pipe sections is sufficiently distant from all zones "pipe" or the deformation of the transverse section is significant, i.e. that one should connect only when ovalization is deadened.

To model the pipe tees, one will privilege modelings in elements hull or 3D and, therefore, the use of connections of the type COQ_TUYAU or 3D_TUYAU respectively. Concerning the connection hull/pipe, one will privilege his application in the cross-sections instead of the curved sections (one recommends to off-set the connections in regular zones, therefore right).

3.4.6 Limiting conditions: keywords DDL_IMPO and LIAISON_*

The keyword factor DDL_IMPO allows to impose, with nodes introduced by one (at least) of the keywords: ALL, NODE, GROUP_NO, MESH, GROUP_MA, one or more values of displacement (or certain associated sizes). According to the name of the operator called, the values are provided directly (AFFE_CHAR_MECA) or via a concept function (AFFE_CHAR_MECA_F).

Operands available for DDL_IMPO, are listed below:

- DX DY DZ : Blocking on the component of displacement in **translation**
- DRX DRY MARTINI DRZ : Blocking on the component of displacement in **rotation**

If the specified nodes belong to elements 'TUYAU_3M' (these elements have 15 degrees of freedom of hull):

U : warping
I : "in plane"
V , *W* : ovalization
O : "out of planes"

That is to say:

- UI2 VI2 WI2 UO2 VO2 WO2 : degrees of freedom related to mode 2.
- UI3 VI3 WI3 UO3 VO3 WO3 : degrees of freedom related to mode 3.
- WO WI1 WO1 : degrees of freedom of swelling and mode 1 on *W* .

If the specified nodes belong to elements 'TUYAU_6M' (these elements have 33 degrees of freedom of hull), the following degrees of freedom are added:

- UI4 VI4 WI4 UO4 VO4 WO4 : degrees of freedom related to mode 4.
- UI5 VI5 WI5 UO5 VO5 WO5 : degrees of freedom related to mode 5.
- UI6 VI6 WI6 UO6 VO6 WO6 : degrees of freedom related to mode 6.

3.5 Application of a thermal dilation.

No thermal calculation is available with modeling PIPE, it is nevertheless possible to apply a dilation (thermal loading of origin), in the shape of a field of temperature to the nodes, defined by the 3 components TEMP, TEMP_INF, TEMP_SUP, of which one deduces the average temperature.

This field will have been beforehand creates using the operator CREA_CHAMP (documentation [U4.72.04]).

CREA_CHAMP	TUYAU_3M TUYAU_6M	Remarks
TYPE_CHAM	'NOEU_TEMP_R' 'NOEU_TEMP_F'	• Field result of type temperature
OPERATION		
GRID		
MODEL		
AFFE	ALL: 'YES' GROUP_MA MESH NODE GROUP_NO	• The field is manufactured by assignment of values on nodes or meshes
NOM_CMP	'TEMP'	• Names of the components which one wants to affect: temperature
	'TEMP_INF'	• Interior temperature
	'TEMP_SUP'	• Outside temperature

The assignment of thermal dilation is carried out using the operator AFFE_MATERIAU (documentation [U4.43.03]).

AFFE_MATERIAU	TUYAU_3M TUYAU_6M	Remarks
AFFE_VARC		
NOM_VARC	'TEMP'	• Name of the variable of order
NOM_CHAM	'TEMP'	•
EVOL	temple	• temple is the field or the evolution creates by CREA_CHAMP then possibly by CREA_RESU.
VALE_REF		• temperature of reference

Note:

If one wants to apply a temperature defined by a function, one can use the operator `CREA_RESU` (`TYPE_RESU='EVOL_THER',...`), document [U4.44.12] to create a concept of the type `EVOL_THER` usable in `AFFE_MATERIAU`.
Examples of use are available in the base of tests: CAS-tests `ELSA01B`, `SSLL106A`, `SSNL117A` and `SSNL503A`.

The following example is extracted from CAS-test `SSLL101C`.

```
T EMP = CREA_CHAMP (
  GRID = GRID ,
  TYPE_CHAM = 'NOEU_TEMP_R' ,
  OPERATION = 'AFFE' ,
  AFFE= (
    _F ( ALL = 'YES' , NOM_CMP = 'TEMP' , VALE = 472.22 ) ,
  ) ,
)
TEMPER = CREA_RESU (
  OPERATION= 'AFFE' , TYPE_RESU= 'EVOL_THER' ,
  NOM_CHAM= 'TEMP' ,
  AFFE= _F ( INST = 0.0 , CHAM_GD = TEMP )
)
AFF_MAT1 = AFFE_MATERIAU (
  GRID = GRID ,
  AFFE= _F ( ALL = 'YES' , MATER = MATER ) ,
  AFFE_VARC= _F ( ALL = 'YES' , EVOL = TEMPER , NOM_VARC = 'TEMP' ,
    NOM_CHAM = 'TEMP' , VALE_REF = 0.0 ) ,
)
```

4 Resolution

4.1 Linear calculations: `MECA_STATIQUE` and other linear operators

Linear calculations are carried out in small deformations. Several linear operators of resolution are available:

- `MECA_STATIQUE` : resolution of a problem of static mechanics linear [U4.51.01],
- `MACRO_ELAS_MULT` : calculate linear static answers for various loading cases or modes of Fourier [U4.51.02],
- `CALC_MODES` : calculation of the values and clean vectors by methods of subspaces or iterations opposite [U4.52.02],
- `MODE_ITER_CYCL` : calculation of the clean modes of a structure with cyclic symmetry [U4.52.05],
- `DYNA_LINE_TRAN` : calculation of the transitory dynamic response to an unspecified temporal excitation [U4.53.02].

By defaults, the only computed fields are the fields of displacement `DEPL` and of constraint `SIEF_ELGA`. Other fields are available by the operand `OPTION` (see the options available in the paragraph [§5.2] bearing on the use of `CALC_CHAMP`).

4.2 Nonlinear calculations: `STAT_NON_LINE` and `DYNA_NON_LINE`

4.2.1 Behaviors and assumptions of deformations available

Following information is extracted from the documentation of use of the operator `STAT_NON_LINE` : [U4.51.03].

<code>STAT_NON_LINE</code>	<code>TUYAU_3M</code>
<code>DYNA_NON_LINE</code>	<code>TUYAU_6M</code>
<code>BEHAVIOR</code>	<code>RELATION</code> • all behaviors available in <code>C_PLAN</code>
	<code>DEFORMATION SMALL</code> •

Incremental relations of behavior (keyword factor `BEHAVIOR`) according to the assumption of small displacements and small deformations (keyword `DEFORMATION = 'SMALL'`) are only mechanical nonlinear relations of behavior available for modeling `PIPE`. These relations of behavior connect the

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>)

rates of deformation to the rates of constraints. The supported nonlinear behaviors are those already existing in plane constraints defined in the operators `STAT_NON_LINE` and `DYNA_NON_LINE`. Moreover, with the method of Borst [R5.03.03], all behaviors `2D` (`D_PLAN`, `AXIS`) in small deformations are usable.

The concept `RESULT` of `STAT_NON_LINE` contains stress fields, displacement and internal variable at the points of integration always calculated at the points of gauss:

- `SIEF_ELGA` : Tensor of the constraints by element at the points of integration in the local reference mark of the element,
- `VARI_ELGA` : Field of internal variables by element at the points of integration in the local reference mark of the element,
- `DEPL` : fields of displacements.

Moreover, one call to the operator `CALC_CHAMP`, gives access other fields. In particular, one can carry out the passage of the constraints and internal variables of the points of Gauss to the nodes to form the fields `SIEF_ELNO` and `VARI_ELNO` (see the paragraph [§5.2]).

A field `VARI_...` can have several types of components. For example, components of the field `VARI_ELNO` are, for the elements `PIPE` :

$$K \text{ time: } (V1, V2, \dots, Vn)$$

Wher K is the number of points of integration total $K = (2 \times NCOU + 1) \times (2 \times NSEC + 1)$
e: N is the number of internal variables and depends on the behavior.

4.2.2 Details on the points of integration

For linear and non-linear calculations, digital integration is carried out with a method of:

- Gauss along average fibre.

The number of points of integration is fixed at 3. For a mesh whose tops are 1 and 2 and numbered from 1 to 2, the 3 points of gauss are such as first is close to 1, second is at equal distance from 1 and 2 and the third is closer to 2. It is thus necessary to pay attention to the orientation of the meshes when one looks at the results at the points of gauss 1 and 3. Indeed if the orientation of the mesh is changed and that one numbers it from 2 to 1, the first point of gauss is closer to 2.

- Simpson in the thickness and on the circumference:

- Integration in the thickness is an integration of Simpson at 3 points per layer. The number of points of integration per layer is fixed at 3, in the middle of the layer, in higher skin and lower skin of the layer, the two points ends being common with the close layers.

- Integration according to the circumference is an integration of Simpson per sector, each sector being of angle $2\pi/NSEC$. Φ is the angle between the generator and the center of the sector. The number of points of integration per sector is fixed at 3, in the middle of the sector, partly higher ($\Phi + 2\pi/NSEC$) and lower ($\Phi - 2\pi/NSEC$) sector, the two points ends being common with the close sectors.

The number of layers and the number of sectors must be defined by the user starting from the keywords: `TUYAU_NCOU`, `TUYAU_NSEC` of the operator `AFFE_CARA_ELEM`.

For example, with 3 layers and 16 sectors, the number of points of integration per element is $(2 \times NCOU + 1) \times (2 \times NSEC + 1) \times NPG$ what gives 693 points of integration. For each point of gauss over the length of the element, one stores information on the layers and for each layer on all the sectors. If one wants information at the point of gauss NG , on the layer NC level NCN ($NCN = -1$ so lower, $NCN = 0$ if medium, $NCN = +1$ so higher), on the sector NS , level NSN ($NSN = -1$ so lower, $NSN = 0$ if medium, $NSN = +1$ so higher), then one looks at the values searched at the point of integration:

$$NP = (NG - 1) \times (2 \times NCOU + 1) \times (2 \times NSEC + 1) \\ + (2 \times NC + NCN - 1) \times (2 \times NSEC + 1) + (2 \times NS + NSN)$$

4.3 Dynamic calculations

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>)

Concerning dynamic calculations, any specificity due to the finite element PIPE exist.

5 Additional calculations and postprocessings

5.1 Elementary calculations of matrices: operator CALC_MATR_ELEM

The operator `CALC_MATR_ELEM` (documentation [U4.61.01]) allows to calculate elementary matrices assembled by the order `ASSE_MATRICE` (documentation [U4.61.22]).

The only calculable matrices with the elements pipe are the matrices of rigidity and mass of the elements of the model:

CALC_MATR_ELEM	TUYAU_3M	TUYAU_6M	Remarks
'RIGI_MECA'	•	•	
'MASS_MECA'	•	•	

These calculations of elementary matrices for example are used for the determination of the Eigen frequencies of a thick cylindrical ring, in CAS-test SDLS109G.

5.2 Calculations by elements: operator CALC_CHAMP

The operator `CALC_CHAMP` (documentation [U4.81.04]) the calculation of the fields to the elements carries out:

- internal constraints, deformations, variables with the nodes;
- equivalent values (nonavailable for modeling `PIPE`).

One presents hereafter the options of postprocessing for the pipe sections. For the structures modelled by pipe sections, it is particularly important to know how the results of the constraints are presented: approach adopted in *Code_Aster* consist in observing the constraints in a particular reference mark related to the element whose reference axis was defined in the paragraph [§3.2.2]. This approach seems most physical because, for a cylindrical structure, the constraints easiest to interpret are not the constraints in Cartesian reference mark but the constraints in cylindrical coordinates. Moreover this approach allows a greater flexibility in use.

CALC_CHAMP	TUYAU_3M	TUYAU_6M	Remarks
'SIEF_ELGA'	•	•	
'SIEF_ELNO'	•	•	
'SIGM_ELGA'	•	•	
'SIGM_ELNO'	•	•	
'EFGE_ELGA'	•	•	
'EFGE_ELNO'	•	•	
'EPSI_ELGA'	•	•	
'EPSI_ELNO'	•	•	
'VARI_ELNO'	•	•	

- `SIEF_ELGA` : calculation of the constraints by element at the points of integration of the element starting from displacements (Use only in elasticity), in the local reference mark of the element.
- `SIEF_ELNO` : calculation of the constraints by element to the nodes of the element starting from the constraint at the points of Gauss, in the local reference mark of the element.
- `SIGM_ELGA` : calculation of the constraints by element at the points of integration of the element from `SIEF_ELGA` in the local reference mark of the element.
- `SIGM_ELNO` : calculation of the constraints by element to the nodes of the element from `SIGM_ELNO`, in the local reference mark of the element.
- `EFGE_ELGA` : calculation of the efforts generalized by element at the points of Gauss starting from displacements, by integration from `SIEF_ELGA`, in the local reference mark of the element.
- `EFGE_ELNO` : calculation of the efforts generalized by element with the nodes starting from displacements into linear and calculated by integration from `SIEF_ELGA` into non-linear, in the local reference mark of the element.
- `EPSI_ELGA` : calculation of the deformations by element at the points of integration of the element starting from displacements, in the local reference mark with the element (small deformations).

- **EPSI_ELNO** : calculation of the deformations by element to the nodes of the element starting from the deformations at the points of Gauss, in the local reference mark with the element (small deformations).
- **VARI_ELNO** : calculation of the field of internal variables by element with the nodes starting from the points of Gauss, for all the layers (in the thickness *SUP/MOY/INF*) and for all the sectors in the local reference mark of the element.

One obtains then by node of each tensor element only one of constraints (or only one whole of internal variables), which allows the graphic examination (evolution of a component,...).

5.3 Calculations with the nodes: operator **CALC_CHAMP**

CALC_CHAMP	TUYAU_3M	TUYAU_6M	Remarks
'FORC_NODA'	.	.	
'REAC_NODA'	.	.	
'EFGG_NOEU'	.	.	
'SIEF_NOEU'	.	.	
'SIGM_NOEU'	.	.	
'VARI_NOEU'	.	.	

The operator **CALC_CHAMP** (documentation [U4.81.04]) carries out the calculation of the fields to the nodes by average and the calculation of the forces and reactions:

- fields with the nodes: internal constraints, deformations, variables, equivalent values;
 - Name of option: to replace **_ELNO_** by **_NOEU_**
 - One can calculate the fields with the nodes by **CALC_CHAMP**
SIEF_NOEU , VARI_NOEU
- forces and reactions:
 - starting from the constraints, balance: **FORC_NODA** (calculation of the nodal forces starting from the constraints at the points of integration, element per element),
 - then by removing the loading applied: **REAC_NODA** (calculation of the nodal forces of reaction to the nodes, the constraints at the points of integration, element per element):
 $REAC_NODA = FORC_NODA - loadings\ applied,$
 - useful for checking of the loading and calculations of resultants, moments, etc.

5.4 Calculations of quantities on whole or part of the structure: operator **POST_ELEM**

The operator **POST_ELEM** (documentation [U4.81.22]) allows to calculate quantities on whole or part of the structure. The calculated quantities correspond to particular options of calculation of affected modeling.

	TUYAU_3M	TUYAU_6M	Remarks
'MASS_INER'	.	.	

For modeling **PIPE**, the only currently available option is calculation, on each element, of the mass, inertias and the position of the centre of gravity (option **'MASS_INER'**).

5.5 Values of components of fields of sizes: operator **POST_RELEVE_T**

For modeling **PIPE**, the operator **POST_RELEVE_T** (documentation [U4.81.21]) can be used for, on a line, to extract from the values (for example **SIEF_ELNO**). The produced concept is of type counts.

Notice important:

If one comes from an interface with a maillor (**PRE_GIBI , PRE_IDEAS , PRE_GMSH**), nodes of one *groupno* are arranged by digital order. It is necessary to reorder the nodes along the line of

examination. The solution is to use the operator `DEFI_GROUP` with the option `NOEU_ORDO`. This option makes it possible to create one `GROUP_NO` ordered containing the nodes of a set of meshes formed by segments (`SEG2`, `SEG3` or `SEG4`).

An example of extraction of component is given in CAS-test SSNL503 (see description in the paragraph [§6.2]):

```
TAB_DRZ=POST_RELEVE_T (ACTION=_F (
    GROUP_NO = 'OF',
    ENTITLE = 'TB_DRZ',
    RESULT = RESUL,
    NOM_CHAM = 'DEPL',
    NOM_CMP = 'DRZ',
    TOUT_ORDRE = 'YES',
    OPERATION = 'EXTRACTION'
)
```

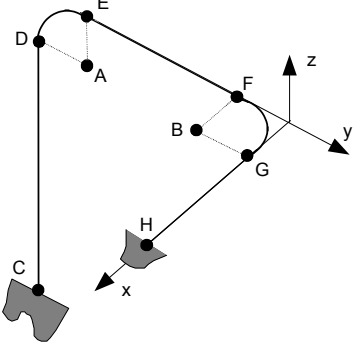
The purpose of this syntax is:

- to extract: OPERATION = 'EXTRACTION'
- on the line (the group of nodes) D : GROUP_NO = 'OF'
- the component DRZ displacement: NOM_CHAM = 'DEPL', NOM_CMP = 'DRZ',
- for every moment of calculation: TOUT_ORDRE = 'YES'

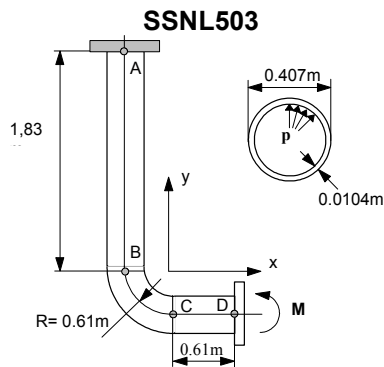
6 Examples

The tables according to describe some CAS-tests using the element PIPE.

6.1 Linear static analysis

<p style="text-align: center;">SSSL101</p> 	<p>Title: Problem of Hovgaard. Static analysis of a three-dimensional piping comprising of the elbows</p> <p>Documentation V: [V3.01.101]</p> <p>Modelings:</p> <table border="0"> <tr> <td>SSSL101D</td> <td>TUYAU_6M</td> <td>SEG3</td> </tr> <tr> <td>SSSL101C</td> <td>TUYAU_3M</td> <td>SEG3</td> </tr> <tr> <td>SSSL101E</td> <td>TUYAU_3M</td> <td>SEG4</td> </tr> </table>	SSSL101D	TUYAU_6M	SEG3	SSSL101C	TUYAU_3M	SEG3	SSSL101E	TUYAU_3M	SEG4
SSSL101D	TUYAU_6M	SEG3								
SSSL101C	TUYAU_3M	SEG3								
SSSL101E	TUYAU_3M	SEG4								
<p style="text-align: center;">SSSL106</p>	<p>Title: Tube right subjected to several loadings.</p> <p>Documentation V: [V3.01.106]</p> <p>Modelings:</p> <table border="0"> <tr> <td>SSSL106B</td> <td>TUYAU_3M</td> <td>SEG3</td> </tr> <tr> <td>SSSL106E</td> <td>TUYAU_3M</td> <td>SEG4</td> </tr> <tr> <td>SSSL106D</td> <td>TUYAU_6M</td> <td>SEG3</td> </tr> </table> <p>Loadings : a traction, 2 efforts cutting-edges, 2 bending moments, a torsion and a pressure. It makes it possible to test displacements, the efforts with the nodes and the constraints and deformations at the points of Gauss, compared to an analytical reference solution. The grid used is the same one for four modelings. Modelings A and C use MECA_STATIQUE, while modelings B and D use STAT_NON_LINE for the resolution.</p>	SSSL106B	TUYAU_3M	SEG3	SSSL106E	TUYAU_3M	SEG4	SSSL106D	TUYAU_6M	SEG3
SSSL106B	TUYAU_3M	SEG3								
SSSL106E	TUYAU_3M	SEG4								
SSSL106D	TUYAU_6M	SEG3								
<p style="text-align: center;">SSLX102</p>	<p>Title: Piping bent in inflection.</p> <p>Documentation V: [V3.05.102]</p> <p>Modelings:</p> <table border="0"> <tr> <td>SSLX102B</td> <td>TUYAU_3M</td> <td>SEG3</td> </tr> <tr> <td>SSLX102C</td> <td>TUYAU_6M</td> <td>SEG3</td> </tr> </table>	SSLX102B	TUYAU_3M	SEG3	SSLX102C	TUYAU_6M	SEG3			
SSLX102B	TUYAU_3M	SEG3								
SSLX102C	TUYAU_6M	SEG3								

6.2 Static analysis nonlinear material



SSNL503

Title: Elastoplastic ruin of a thin bent pipe.

Documentation V: [V6.02.503]

Modelings:

SSNL503A TUYAU_3M SEG3

Loading : thin bent pipe subjected to an inflection in its plan and has an internal pressure with basic effect.

SSNI106

Title: Fixed beam has an end and charged by a traction with linear work hardening or a moment in perfect plasticity.

Documentation V: [V6.02.106]

Modelings:

SSNL106E TUYAU_3M SEG3

SSNL106F TUYAU_3M SEG4

SSNL106G TUYAU_6M SEG3

Loadings : a traction, 2 efforts cutting-edges, 2 bending moments, a torsion and a pressure. Modelings A and C use MECA_STATIQUE, while modelings B and D use STAT_NON_LINE for the resolution.

HSNV100

Title: Thermoplasticity in simple traction of a pipe.

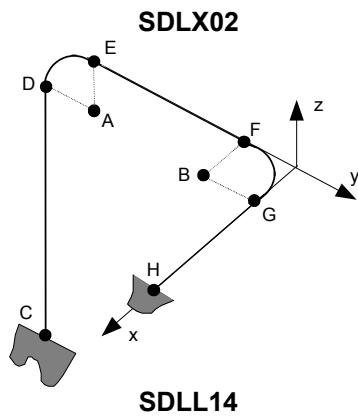
Documentation V: [V7.22.100]

Modelings:

HSNV100C TUYAU_3M SEG3

HSNV100D TUYAU_6M SEG3

6.3 Modal analysis in dynamics



Title: Problem of Hovgaard. Dynamic analysis of a three-dimensional piping comprising of the elbows.

Documentation V: [V2.05.002]

Modelings:

SDLX02D	TUYAU_3M	SEG3
SDLX02F	TUYAU_3M	SEG4
SDLX02E	TUYAU_6M	SEG3

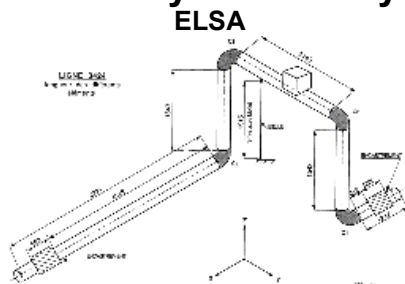
Title: Mode of vibration of a thin elbow of piping.

Documentation V: [V2.02.014]

Modelings:

SDLL14A	TUYAU_3M	SEG3
SDLL14C	TUYAU_3M	SEG4
SDLL14B	TUYAU_6M	SEG3

6.4 Nonlinear dynamic analysis



Title: Nonlinear seismic analysis of a line of piping.

Documentation V: [V1.10.119]

Modelings:

ELSA01B	TUYAU_3M	SEG3
ELSA01C	TUYAU_3M	SEG4

Loadings : a seismic excitation is imposed on the line. This one only involves a partial plasticization (1%) elbows.

7 Bibliographical references

1. Finite elements of right pipe and curve with ovalization, swelling and warping in elastoplasticity, Documentation *Code_Aster*, Manuel de Référence [R3.08.06]
2. Modelings TUYAU_3M and TUYAU_6M, Documentation *Code_Aster*, Instruction manual [U3.11.06]
3. ELSA01 - seismic Analysis nonlinear of a line of piping, Documentation *Code_Aster*, Manuel de Validation [V1.10.119]
4. HSNV100 - Thermoplasticity in simple traction, Documentation *Code_Aster*, Manuel de Validation [V7.22.100]
5. SDLL14 - Modes of vibration of an elbow of piping thin, Documentation *Code_Aster*, Manuel de Validation [V2.02.014]
6. SDLX02 – Piping: Problem of Hovgaard. Spectral analysis, Documentation *Code_Aster*, Manuel de Validation [V2.05.002]
7. SSNL106 - Beam elastoplastic in traction and pure inflection, Documentation *Code_Aster*, Manuel de Validation [V6.02.106]
8. SSLX101 - Pipe right modelled in hulls and beams, Documentation *Code_Aster*, Manuel de Validation [V3.05.101]
9. SSLX102 - Piping bent in inflection, Documentation *Code_Aster*, Manuel de Validation [V3.05.102]
10. SSNL503 - Elastoplastic ruin of a thin bent pipe, Documentation *Code_Aster*, Manuel de Validation [V6.02.503]
11. ZZZZ112 - Cylinder under variable pressure. Validation of LIRE_PLEXUS, Documentation *Code_Aster*, Manuel de Validation [V1.01.112]
12. P. MASSIN, J.M. PROIX, F. WAECKEL, E. CHAMPAIN: Modeling of the behavior not - linear material of the pipings right and bent in statics and dynamics, Note HI - 74/99/013/A, EDF/MTI, 1999.
13. E. CHAMPAIN, Project CACIP: Validation of the developments relating to the taking into account of plasticity in the pipes, HP-52/99/029/B, EDF/AMV, 2000.