Analysis of the seismic behaviour of the large metal tanks

Summary:

This document aims the description of the various stages to implement in *Code_Aster* to carry out the digital simulation of the seism resistance of large metal tanks. These mean metal structures can then present two privileged modes of ruin: rupture of anchorings or buckling of the rings.

The first approach is based on a lawful method of push-over type. One can, in complement, to use a nonlinear criterion of stability (*cf.* documentation [U2.08.04] and CAS-tests SSLL105 and SSNL126).

The second approach is direct transitory modeling with complete taking into account of the fluid field through a coupled approach fluid-structure in great displacements (modeling similar to that of CAS-test FDNV100). One can also couple this transitory analysis with an analysis of non-linear stability, like CAS-test FDNV100 shows it too.

Contents

1 Introduction

This documentation presents various digital methodologies developed to simulate and analyze the seismic behaviour of large metal tanks. It is thus a question here of presenting the implementation and the sequence of various operators of *Code_Aster* to conduct to good the study of this kind of components, following various assumptions of modeling which are mainly dictated by lawful considerations. These tanks, which are mean metal structures, can present two privileged modes of ruin: rupture of anchorings or buckling of the rings.

The first methodology is based on a lawful method of push-over type ([bib1], [bib2], [bib3], [bib4]). The tank is subjected to an internal pressure imposed, spatially variable, increasing. The resolution is done into quasi-static and the fluid field is not modelled directly: its influence on the wall is transcribed by a particular imposed field of pressure. Nonthe linearities are of type geometrical and behavioral (plasticity). During incremental calculation, one makes grow this pressure until obtaining the ultimate load which corresponds to the buckling of the structure (one can use the keyword CRIT_STAB of STAT_NON_LINE for a non-linear analysis of stability). In order to model anchorings more finely, one can introduce rising on the level of the anchorings bolted with the ground [bib5]. One can also increase the mechanical resistance with buckling by adding a carbon fibre reinforcement on the rings. Its modeling is presented in this document.

The second approach is direct transitory modeling with complete taking into account of the fluid field through a coupled approach fluid-structure in great displacements [bib6]. This modeling, finer than the preceding one, comes to supplement the lawful approaches, in particular while making it possible to better determine the limits of their field of validity, mainly with respect to the great not total linearities like great displacements. However, the practical use of these transitory approaches is limited by the digital overcost which they induce (report of about 10). It is possible to combine the transitory approach with an analysis of non-linear stability via the use of the keyword CRIT_STAB of DYNA_NON_LINE. Contrary to quasi-static calculations, the characteristics of the model fluid-structure requires a particular treatment in CRIT_STAB who will be detailed in this document.

2 Presentation of the problems

2.1 Geometry of the structure

The problem, in the case of a standard cover, can be represented geometrically thus [bib2]:

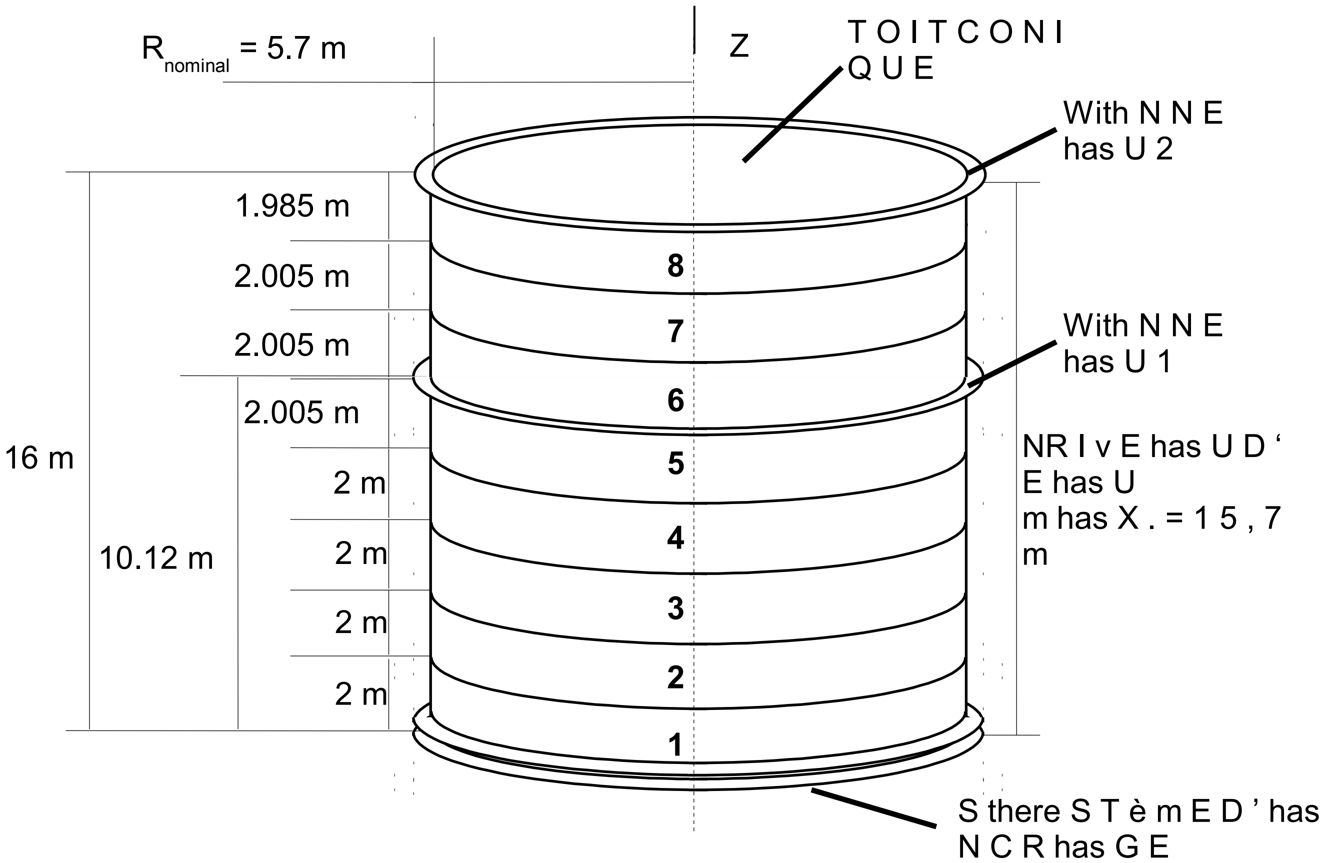

Figure 2.1- has: Schematic representation of a real cover

The cover presented is made up of 8 cylindrical binding rings of approximately $2\,m$ of height each one, which is welded enter they to form the cylindrical wall of the tank. The thickness, constant by ring, are decreasing when one passes from a ring to that which overhangs it. This specific measure makes it possible to optimize the thickness of the rings, according to the hydrostatic pressure exerted by the fluid contained and which varies linearly with the depth.

Being given very low thicknesses of the rings (approximately 4 with $5\,mm$), one chooses to model the metal walls by voluminal thin hulls (`COQUE_3D`).
These elements have as a geometrical support a quadrilateral with 9 nodes.
Certain mailleurs cannot generate this kind of element but propose grids with quadrilaterals with 8 more classical nodes. One thus comes to define the nodes mediums in *Code_Aster* by the following order:

```
MAILLA2=CRÉA_MAILLAGE (MAILLAGE=MAILLA01,
                       MODI_MAILLE=_F (GROUP_MA=' RESERVOIR',
                                       OPTION=' QUAD8_9',
                                       PREF_NOEUD=' NSQ',
                                       PREF_NUME=1,),);
```

One makes sure then that the normals are correctly directed:

```
MAILLA2 = MODI_MAILLAGE (reuse = MAILLA2, GRID = MAILLA2,
                         ORIE_NORM_COQUE = _F (GROUP_MA =
                                                ('RING', 'TFC2'),
                                                  VECT_NORM = (0. , - 1. ,
0.),
                                                GROUP_NO = 'OY',),
                         MODEL = MODELE0, INFORMATION = 2,)
```

One can then define the model `MODEL` final based on the voluminal hulls.

One defines the geometrical characteristics specific to the elements of structures employed (`COQUE_3D`). Ring by ring, one gives the thicknesses and the orientation of the reference marks. It is particularly necessary to be vigilant with coherence between the axes of the reference marks and characteristic materials when one is in an orthotropic case (case of the reinforcement). In the same way for this reinforcement, one can make use of offsetting to take account of his real layout on the outside of the metal ring (its average surface cannot thus be confused with that of the ring supporting it):

```
CARAELEM = AFFE_CARA_ELEM (MODEL = MODEL,
                            HULL = (_F (GROUP_MA = ('SURF0',),
                                          THICK = 7.13E-3,
                                          ANGL_REP = (0. , 90. ),
                                          A_CIS = 0.8333,
                                          COEF_RIGI_DRZ = 1.E-05,
                                          OFFSETTING = 0. ,
                                          INER_ROTA = 'YES',
                                          MODI_METRIQUE = 'YES',),
…
                                    _F (GROUP_MA = ('RING',),
                                          THICK = 1.E-2,
                                          ANGL_REP = (0. , 0. ),
                                          A_CIS = 0.8333,
                                          COEF_RIGI_DRZ = 1.E-05,
                                          OFFSETTING = 0. ,
                                          INER_ROTA = 'YES',
                                          MODI_METRIQUE = 'YES',),
                                    _F (GROUP_MA = ('TFC2',),
                                          THICK = (1. * 1.72E-3),
                                          ANGL_REP = (0. , 90. ),
                                          A_CIS = 0.8333,
                                          OFFSETTING = -4.425E-3,
                                          INER_ROTA = 'YES',
                                          MODI_METRIQUE = 'YES',),),)
```

2.2 Materials employed

The sheet-iron works is out of stainless steel austenitic standard A240 304L and the nuts and bolts will be taken into account as being out of A42 steel.

For A240 steel, various elastoplastic laws with nonlinear isotropic work hardening (in the form of traction diagrams) are proposed. The definition of work hardening starting from traction diagrams, without information on the discharge, is well adapted to the applications where the loading is monotonous growing, which one will suppose in our case (*cf.* § 3.1):

```
A5=DEFI_FONCTION (NOM_PARA=' EPSI',
                 VALE= (0.00097, 191820000. , 0.00116, 221740000. ,
                        0.00138, 247020000. , 0.00167, 267090000. ,
                        0.00207, 285920000. , 0.0029, 304530000. ,
                        0.00365, 314300000. , 0.00468, 322090000. ),
                 PROL_DROITE=' LINEAIRE',
                 PROL_GAUCHE=' LINEAIRE',);
#
MAA240 = DEFI_MATERIAU (ELAS = _F (E = 1.98E11,
                                   NAKED = 0.3,
                                   RHO = 7900. ,),
                         TRACTION = _F (SIGM = A5,),)
```

This relation of behavior is of the type `'VMIS_ISOT_TRAC'` in the nonlinear operator of resolution employed.

The plastic behavior of the A42 nuance will not be taken into account to start. Indeed, the yield stress is higher than for the 304L and the parts made up of this material are more massive, which generates lower levels of constraints. The assumption of purely elastic behavior isotropic is thus acceptable for corresponding material, named `MAA42`.

If one adds a carbon fibre fabric reinforcement (TFC), one models it by an orthotropic elastic material (the threshold of plasticity is never reached for our loadings):

```
MATFC = DEFI_MATERIAU (ELAS_ORTH = _F (E_L = 1.E10,
                                       E_T = 1.E12,
                                       G_LT = 3.E9,
                                       NU_LT = 0.3,
                                       RHO = 2500. ,),)
```

3 Quasi-static approach push-over

3.1 Imposed loadings

The imposed loading is of lawful origin (EC-8), quasi-static [bib1]. One represents the effects of an earthquake on the structure by a variable field of pressure imposed on the inner face of the rings. The value in each point is function of the current coordinate and grows linearly with time. This monotonous evolution in time is characteristic of the methods known as push-over, with the direction EC-8, whose objective is to simulate by a quasi-static calculation the answer to a seismic request, and thus of transitory dynamic physical nature. The dynamic effects, like the inertia and the efforts generated by the fluid in shaking are replaced by this distribution of imposed pressure. The problem to be treated utilizes only one modeling of the structure, without modeling of the fluid field.

The methods push-over obviously were built and justified by making strong assumptions of linearization of the problem (small displacements, elastic behavior, buckling of Euler…). Below the definitions of the imposed fields of pressures are given [bib2]:

```
# ----------------------------------------------------------------------
#    DEFINITION OF THE FIELDS OF PRESSURE  (Pa)
#
#       PH (Z)       : HYDROSTATIC PRESSURE
#       CONK (Z, SUCKED): FLEXIBLE IMPULSIVE PRESSURE
#       PIR (Z, SUCKED): RIGID IMPULSIVE PRESSURE
#       PV (Z)       : VERTICAL PRESSURE
#
#       CONK (Z, SUCKED) = CONK (Z) *COS (SUCKED)
#       PIR (Z, SUCKED) = PIR (Z) *COS (SUCKED)
#
#P (Z, SUCKED) =PH (Z) +INSTANT* (CONK (Z, SUCKED) + PIR (Z, SUCKED) +
0.4*PV (Z))
# ----------------------------------------------------------------------
PHZ = DEFI_FONCTION (  NOM_PARA = ‘Z’,
…
                       VERIF = ‘GROWING’,  )
PIFZ = DEFI_FONCTION (  NOM_PARA = ‘Z’,
…
                         VERIF = ‘GROWING’,  )
PIRZ = DEFI_FONCTION (  NOM_PARA = ‘Z’,
…
                        VERIF = ‘GROWING’,)
PVZ = DEFI_FONCTION (  NOM_PARA = ‘Z’,
…
                       VERIF = ‘GROWING’,)
# ----------------------------------------------------------------------
#       P (X, Y, Z)    : TOTAL PRESSURE
#P (Z, SUCKED) =PH (Z) +INSTANT* (CONK (Z, SUCKED) + PIR (Z, SUCKED) +
0.4*PV (Z))
# ----------------------------------------------------------------------
COSTE=FORMULE (NOM_PARA= (‘X’, ‘Y’), VALE=' X/SQRT ((X*X) + (Y*Y))‘)
PH=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’), VALE=' PHZ (Z) ‘)
PIF=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’), VALE=' PIFZ (Z) *COSTE (X, Y) ‘)
PIR=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’), VALE=' PIRZ (Z) *COSTE (X, Y) ‘)
PV=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’), VALE=' PVZ (Z) ‘)
PS1=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’),
            VALE=' CONK (X, Y, Z) +PIR (X, Y, Z) + (0.4*PV (X, Y, Z))‘)
PS2=FORMULE (NOM_PARA= (‘X’, ‘Y’, ‘Z’)
            VALE=' CONK (X, Y, Z) +PIR (X, Y, Z) - (0.4*PV (X, Y, Z))‘)
```

One can then define the loading mechanics correspondents (following gravity and pressures):

```
= AFFE_CHAR_MECA (MODEL WEIGHED = MODEL,
                        PESANTEUR=_F (GRAVITE=9.81,
                                     DIRECTION= (0.0, 0.0, - 1.0),),);
#
PRESPH = AFFE_CHAR_MECA_F (MODEL = MODEL,
       FORCE_COQUE = _F (GROUP_MA = 'RING',), CLOSE = PH, PLAN = 'INF',),)
#
PRESPS1 = AFFE_CHAR_MECA_F (MODEL = MODEL,
                            FORCE_COQUE = _F (GROUP_MA = 'RING',  ) ,
                                              CLOSE = PS1,
                                              PLAN = 'INF',),)
#
PRESPS2 = AFFE_CHAR_MECA_F (MODEL = MODEL,
                            FORCE_COQUE = _F (GROUP_MA =
                                              ('RING',  ) ,
                                              CLOSE = PS2,
                                              PLAN = 'INF',),)
```

These fields of pressures are multiplied by a linear increasing function of time:

```
FONCMUL = DEFI_FONCTION (NOM_PARA = 'INST', = (0. , 0. , 3. , 3. ),)
```

The connection with the ground is regarded here as being complete (group of nodes `'BASE'`). One defines also the conditions of symmetry (group of nodes `'SYMMETRY'`) since only one half-tank is netted:

```
CONDLIM = AFFE_CHAR_MECA (MODEL = MODEL,
                          DDL_IMPO = (_F (GROUP_NO = 'BASE',
                                          DX = 0. , DY = 0. , DZ = 0. ,
                                           DRX = 0. , DRY MARTINI = 0. , DRZ =
0. ,),
                                         _F (GROUP_NO = 'SYMMETRY',
                                          DY = 0. , DRX = 0. , DRZ = 0. ,),),)
```

3.2 Method of monotonous quasi-static resolution nonlinear

One wants to solve a problem of quasi-static evolution nonlinear. One will thus use the operator `STAT_NON_LINE` ([U4.51.03], [R5.03.01]). The imposed loading will be built with the pressure `PRESPS1` for example:

```
RESU = STAT_NON_LINE (MODEL = MODEL,
                      CHAM_MATER = CHMAT,
                      CARA_ELEM = CARAELEM,
                      EXCIT = (_F (LOAD = CONDLIM,
                                   TYPE_CHARGE = 'FIXE_CSTE',),
                               _F (LOAD = WEIGHED,
                                   TYPE_CHARGE = 'FIXE_CSTE',),
                               _F (LOAD = PRESPH,
                                   FONC_MULT = FONCMUL,
                                   TYPE_CHARGE = 'SUIV',),
                               _F (LOAD = PRESPS1,
                                   FONC_MULT = FONCMUL,
                                   TYPE_CHARGE = 'SUIV',),),
                      BEHAVIOR = (_F (RELATION = 'ELAS',
                                      COQUE_NCOU = 1,
                                      DEFORMATION = 'GROT_GDEP',
                                      GROUP_MA = ('SURF2',
                                      'RING', 'TFC2'),)  ,
```

```
                         _F (RELATION = 'VMIS_ISOT_TRAC',
                              COQUE_NCOU = 1,
                              DEFORMATION = 'GROT_GDEP',
                              GROUP_MA =
                           ('SURF0', 'SURF1',),)  ,),
         INCREMENT = _F (LIST_INST = L_INST1, ) ,

                                )
```

3.3 Remarks on calculations and postprocessings

In the example presented above one uses an algorithm of Newton: one solves with the total tangent operator (stiffness) reactualized with each iteration. If the problem is well posed (sufficient "regular"), it is known that this kind of algorithm offers best convergence. Therefore, in our case, as long as the total tangent operator is "well" defined (far from being singular), calculation well will proceed with a fast convergence. When the level of load imposed will approach the ultimate load, the structure then will become unstable within the meaning of buckling [R7.05.01], which results in the tendency of the total tangent operator to become singular. The loss of stability per boundary point east makes of it the loss of unicity of the solution, that is to say thus the singularity of the operator of resolution. In the vicinity of the boundary point the algorithm of Newton will converge less better, from where the need for imposing increments of smaller times and an increase amongst iterations on the residue in balance.
Generally, more one will approach the ultimate load, more the step of time will have to be small. In spite of that the risks of stop of calculation on nonconvergence are important, from where obligation to carry out calculation according to several successive continuations.

It is nevertheless possible to improve this convergence while changing in the course of calculation of algorithm and to rock on a quasi-Newton. For that, it is enough to solve on the tangent operator whom one reactualizes only with each step (between two iteration one keeps the same one), and if that is insufficient, one can then solve with the elastic operator who, it, cannot become singular. This choice reinforces the robustness of the algorithm in terms of convergence, but it increases, sometimes considerably, the iteration count (and/or of steps) necessary to obtain the solution.

For our type of study one can distinguish two types of quantities of interest for postprocessing. On the one hand a size ready to translate the buckling of the structure and thus to reveal the ultimate load. For that one can trace the pressure level (multiplying coefficient) function of the displacement of a point located at the top of the tank (at the end of the generator most subjected to compression). In addition a more local indicator of the appearance of plasticity: the Iso-value of the cumulated plastic deformation, at every moment of calculation.

These two postprocessings do not present any particular difficulty in *Code_Aster*.

The model finite elements implemented comprised between 55000 and 110000 degrees of freedom. These models were based on a simplified representation of anchorings: the geometry of the reinforcements and brackets is with a grid finely, but boulonnement is not present and it is replaced by a condition of embedding on all the nodes of altitude $0m$. A more realistic modeling of anchorings, with separation would thus have involved a size of total problem appreciably larger.

Method for calculation presented here, whose objective is to study the quasi-static answer in buckling [R7.05.01] of a tank is very close to the framework of the note of calculation to buckling [U2.08.04]. This documentation presents the analyses of linear stability (within the meaning of Euler) and clarification of a nonlinear calculation of push-over type to obtain the ultimate load.

3.4 Fine modeling of anchorings: rising

The type of tanks studied here are bolted on the ground [bib5]. These bolts (or ties) cross reinforced supports welded in base of ring). For more speed of calculation we present thereafter the method put in work on a simplified grid, but with realistic anchorings: the tank is fixed on the ground by 18 ties.

In *Code_Aster*, several modelings of the contact are available.
The modeling of the geometry of the zone of contact can be surface (3D problem), linear (problem 2D) or constituted by discrete elements (keyword `DIS_CHOC` in `STAT_NON_LINE` and `DIS_CONTACT` for the law material).
The contact itself can be treated, either in a nodal way by penalization or the method of Lagrangian, with active constraints or not, or continuously by the method of Lagrangian increased.

The simplest method, in this configuration of study, is that of the penalization, which in this case has also the advantage of being able to take account of the role of the seal without having to net it separately. For a quasi-static calculation, the penalization is not numerically problematic, as in dynamics where this technique can generate disturbances high frequencies (related to the stiffness of penalization). Moreover, for elements of structure, one can fix the stiffness of penalization so as to as well as possible approach the stiffness of contact of the part which is actually massive.

The other methods are however more rigorous because they do not generate interpenetration.
For the study of the separation of the cover, we will thus use a method of penalization, which presents the best report quality of modeling/cost of calculation on our precise case.
Being given the option taken to oblige all the nodes to be able to move only vertically, one can benefit from this total pairing to be satisfied to use discrete elements for the contact. Indeed, it is not necessary to make D-pairing, and thus, master-slave surface general modelings of the zones of contact are unnecessarily heavy. One will thus place a carpet of discrete elements of contact under the low hoop. There will be then a discrete element (`DIS_T` on mesh `SEG2`) under each node of altitude 0.

If the calculated solution proves too far away from the experimental reference solution, it will be necessary to quantify the influence of the condition of only vertical displacement. It will be enough to start again calculations without this kinematic assumption, with, if horizontal displacements are large, a method of management of the contact with D-pairing of the nodes of surfaces concerned.

Remarks

Rigorously, it would be necessary to equip all the nodes with the basic ring of a discrete element of contact. However, if elements of the type are used `COQUE_3D` for the hoops, one cannot place element of contact on the level of the nodes in the center of the elements. Indeed, this node has the effect of carrying only degrees of freedom of type rotation. The condition of contact, which relates to normal displacement to the face, cannot thus be expressed in these nodes. Elements of contact, as well as the kinematic condition of vertical displacement of the basic nodes should thus be carried only by the nodes tops or mediums of the edges.

The modeling of the contact by discrete elements allows, in *Code_Aster*, to directly define the preload of these elements. One can thus represent the prestressing of the ties, without having pre-to deform these elements, as one should make it if one had chosen another modeling of the contact being based on surfaces in opposite.

Below, we present the position of the discrete elements of contact penalized (in clear gray) and of the 9 springs modelling the ties (in black). The reinforcement in base of ring is indicated in dotted lines:

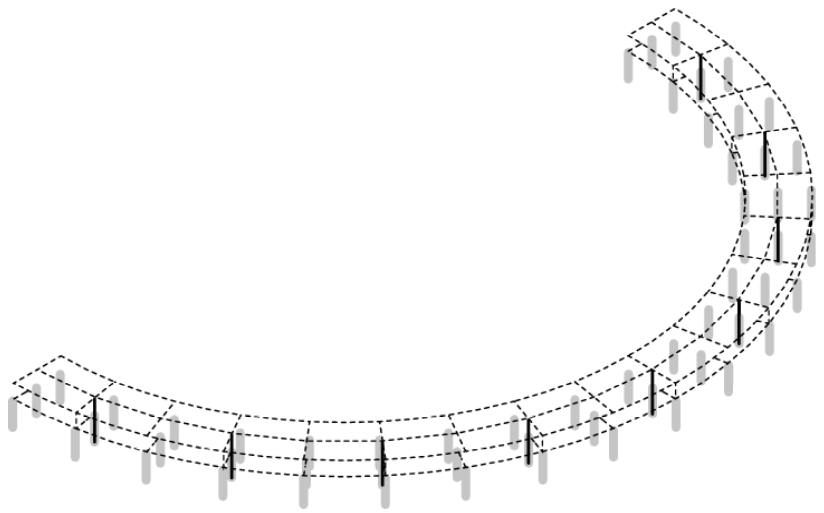

Figure 3.4- has: Provision of the discrete elements for the connection with the flagstone

These discrete elements for anchorings are grouped in 2 groups of meshs:

- RESSC : elements of contact in clear gray (an element SEG2 by node of the low support, that is to say 57 SEG2),
- LMBO : ties in black (9 springs by elements SEG2).

One thus introduces an additional material (MATRES) who corresponds to the elements of contact and who allows to take into account prestressing by the tightening of the nuts on the ties (one defines also the distance associated with the height between supports with DIST_1 and DIST_2):

```
Eboulon = 2.E11;
Dboulon = 0,026;
Sboulon = 3.14159 * Dboulon * Dboulon/4;
kressort = Eboulon * Sboulon;
kpenal = kressort * 100.
MATRES = DEFI_MATERIAU (DIS_CONTACT = _F (RIGI_NOR = kpenal,
                                     DIST_1 = 0.05, DIST_2 = 0.05,),);
```

One supplements the assignment material with these discrete elements on the group of meshs RESSC :

```
CHMAT = AFFE_MATERIAU (GRID = MAILLA2,
                       AFFE = (_F (GROUP_MA = 'RING'  ,),
                                    MATER = MAA240,),
                                _F (GROUP_MA = ('ANNET',),
                              MATER = MABID,)  ,
                                _F (GROUP_MA = ('ANNEA',),
                                    MATER = MAA240,)  ,
                                _F (GROUP_MA = ('RESSC',),
                                    MATER = MATRES,),),)
```

The elementary characteristics of the new discrete elements are as follows defined:

```
CARAELEM = AFFE_CARA_ELEM (MODEL = MODEL,
                           DISCRETE = (_F (GROUP_MA = 'RESSC',
                                          REFERENCE MARK = 'LOCAL', CARA =
'K_T_D_L',
                                        VALE = (10. , 0.0, 0.0,),),
                                      _F (GROUP_MA = 'LMBO',
                                          REFERENCE MARK = 'LOCAL', CARA =
       'K_T_D_L',
                                        VALE = (kressort, 0. , 0. ,),),),)
```

The springs which model the ties (GROUP_MA LMBO) have the equivalent stiffness well kressort.

It remains to modify the boundary conditions in displacement for anchorings: one does not authorize that displacements according to the vertical of the nodes of low the lower flange facing (nodes initially in contact with the ground).

The operator STAT_NON_LINE ([U4.51.03], [R5.03.01]) sees also its arguments impacted by the introduction of rising in base (keyword DIS_CHOC for the group of mesh RESSC):

```
RESU = STAT_NON_LINE (MODEL = MODEL, _MATER = CHMAT,
…
                      C OMP_INCR = (_F (RELATION = 'ELAS',
                                        DEFORMATION = 'GROT_GDEP',
                                        GROUP_MA = 'RING',)   ,
                                    _F (RELATION = 'ELAS',
                                        GROUP_MA = 'LMBO',)   ,
                                    _F (RELATION = 'DIS_CHOC',
                                        GROUP_MA = 'RESSC',),
                              _F (RELATION = 'VMIS_ISOT_TRAC',
                                        DEFORMATION = ' GROT_GDEP ',
                                        GROUP_MA =  ('SURF0', 'ANNEA',
```

```
                    'SURF1', 'SURF2'),),),
        …
        );
```

The rest of the command file is unchanged.

3.5 Use of the nonlinear criterion of stability

One can also use a criterion of stability based on the tangent matrix: there is an instability if the matrix of tangent stiffness becomes singular, that is to say so at least one of its eigenvalues is cancelled. One then solves the problem with the eigenvalues according to, written in great displacements (writing into Lagrangian with the tensor of deformation of Green-Lagrange) [R7.05.01]:

$(K_T+\lambda I_d)x=0 \Leftrightarrow K_T x=\lambda I_d x$ with:

$K_T=K+K^L(\underline{u})+K^Q(\underline{u})+K(\pi)$	Matrix of tangent rigidity
$K^L(\boldsymbol{u})$	Linear part in $\underline{u}$ matrix K_T
$K^Q(\boldsymbol{u})$	Quadratic part in $\underline{u}$ matrix K_T
$K(\pi)$	Geometrical matrix of stiffness
π	Tensor of Piola-Kirchhoff II
I_d	Matrix identity
λ	Eigenvalue

Documentation [R7.05.01] presents these analyses of stability more in detail. Documentation [U2.08.04] into present the use.

Remarks

When displacements are small, one has simply $K(\pi)=K(\sigma)=K_G$ and matrices K^L and K^Q can be neglected.
The linear analysis within the meaning of Euler in Code_Aster does not make it possible to take account of the following aspect of the field of pressure: it is obligatorily necessary to pass the approach presented here and basedE on a non-linear resolution.

If one wants to make use of the criterion of stability, it is enough to add the following line among the arguments of `STAT_NON_LINE` ([U4.51.03], [R5.03.01]):

```
CRIT_STAB=_F (NMAX_CHAR_CRIT=1,),
```

One then seeks the first eigenvalue of the tangent total operator of our system.
So during calculation one observes that this eigenvalue decreases, even changes sign, that means that one approached the first critical load and that then it was even exceeded.
The number of eigenvalues to determine can be imposed by the keyword `NMAX_CHAR_CRIT` (3 by defaults).
It is also possible, by using the order `CHAR_CRIT`, to choose the band in which it is necessary to seek these eigenvalues (– 10 to 10 by defaults).

Notice

The indication of a waveband is useful especially for calculations in small disturbances where a test of Sturm is carried out for the provided waveband. One can thus save time by calculating the eigenvalues only if there is in the band indicated. The test of Sturm is not made in great deformations and the eigenvalues are calculated with each step of time.

Mode of buckling as well as the eigenvalues given (load factor criticizes) can be recovered by using the order `IMPR_RESU` :

```
IMPR_RESU (MODEL = MODEL, FORMAT = 'RESULT', RESU= (_F (RESULTAT=RESU,
                 NOM_PARA=' CHAR_CRIT', 'MODE_FLAMB',),),)
```

CAS-test SSLL105 [V3.01.105] proposes an example of use of this criterion of stability for a linear case and CAS-test SSNL126 [V6.02.126] for a nonlinear case (elastoplastic beam).

3.6 Piloting of the loading

In order to facilitate the convergence of incremental calculation when one is close to the level of ultimate load, or in order to be able to exceed this critical point, it can be judicious to be placed more in loading forced to privilege a piloting in displacement or a piloting by length of arc (its use for the post-critical one is briefly recalled in documentation [U2.08.04]). Piloting cannot be used with the contact [U4.51.03].

4 Coupled transitory approach

4.1 Problem of reference

One leaves here the regulation framework and one will exploit all the opportunities of modeling given by *Code_Aster*. The model of the tank itself remains unchanged (elastoplastic voluminal hulls). On the other hand, one will represent the fluid field by a massive grid. Moreover, the resolution will be done in transitory dynamics with the operator `DYNA_NON_LINE` ([U4.53.01], [R5.05.05]), the external request being of the seismic type.

The fluid field is modelled in linear acoustics (barotrope, compressible, nonviscous and with free surface). The problem coupled fluid-structure is solved in *Code_Aster* by a symmetrical formulation (u, p, φ) ([R4.02.02], [bib7]), in reactualized Lagrangian writing. The loading is of the accélérogramme type imposed in base of cover.

The discretized problem arises then thus [bib6]:

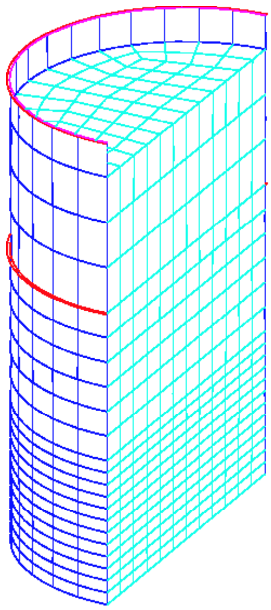

Figure 4.1- has: Complete grid with the fluid field and the structure

4.2 Coupled modeling fluid-structure in *Code_Aster*

In order to be able to carry out a calculation coupled by formulation (u, p, φ) [bib7] in *Code_Aster* , for free surfaces and slip with the interface (not adherence between the nonviscous fluid and the internal wall of the tank), it is necessary to respect a certain construction of the grid and corresponding models.
One must thus define the fluid grid, the grid structure and the interface fluid-structure.

In the maillor one thus generates two distinct grids (different meshs but common nodes) for the fluid field and the structure. For practical reasons, it can be simpler to generate them distinct meshs separately, therefore with duplicated nodes and then to eliminate these double nodes not to have but simple nodes more.

Then, in *Code_Aster*, one as follows will generate the grid support of the interface, starting from the groups of meshs SURF0 , SURF1 and SURF2 who are the rings of the tank:

```
MAILLA01=CRÉA_MAILLAGE (MAILLAGE=MAILLA1,
                        CREA_GROUP_MA=_F (NOM=' IFLUSTRU',
                                          GROUP_MA= ('SURF0', 'SURF1',
       'SURF2',),
                                          PREF_MAILLE=' I',),);
```

A new group of meshs is thus created IFLUSTRU . The nodes of this grid are confused with those of the structure, but the meshs are different.

Remarks

The structure is with a grid in elements COQUE_3D , of which the geometrical support is a quadrilateral with 9 nodes. The node medium has the effect of carrying only degrees of freedom of rotation.
If one wants to couple such an element with a solid element of fluid, one cannot thus write kinematic condition of coupling for this node medium since it does not comprise a degree of freedom of translation, contrary to the node corresponding which comes from the fluid field and which, carries to him only degrees of freedom of translation.
To circumvent this problem, one will write the coupling fluid-structure only on the nodes of the grid hull comprising the degrees of freedom of translation: the nodes tops and the nodes mediums of the edges of the elements.
It is thus necessary to have in Code_Aster the grid structure comprising only quadrilateral elements with 8 nodes (they are well the nodes tops and the nodes mediums of the edges), on which one defines the interface. The grid structure for COQUE_3D being defined only after, starting from this grid, by adding the nodes mediums.
The fluid grid, to be in conformity with the interface, will be him composed of parallelepipeds with 20 nodes.

Moreover, even if one used a grid structure with elements of hull whose geometrical support would be a quadrilateral with 9 nodes, and where all nodes, even the exchange carried degrees of freedom of translation, the coupling fluid-structure could pose a problem. Indeed, the massive fluid grid conforms should be carried out with parallelepipedic solid elements with 27 nodes. However, certain mailleurs do not offer this kind of complete elements which are employed rather little in structural analysis, contrary to the classical quadratic elements which are the parallelepipeds with 20 nodes.

It is imperative that the normal external with the fluid field is always directed in the same direction. It is strongly advised to keep the convention of orientation of the structure towards the fluid for all modelings of interface fluid-structure, in particular FLUI_STRU .

Once the definition of the interface, which is thus with a grid in quadrilateral elements with 8 nodes (which are equivalent to the faces of the solid elements employed for the fluid field: elements parallelepiped with 20 nodes), one can thus make the modification of the grid structure (group of meshs RESERVOI) in elements with 9 nodes, geometrical support of COQUE_3D :

```
MAILLA2=CRÉA_MAILLAGE (MAILLAGE=MAILLA01,
                        MODI_MAILLE=_F (GROUP_MA=' RESERVOIR',
                                        OPTION=' QUAD8_9',
                                        PREF_NOEUD=' NSQ',
                                        PREF_NUME=1,),);
```

One can then define the models necessary to coupled calculation:

```
MODELE=AFFE_MODELE (MAILLAGE=MAILLA2, INFO=1, VERIF=' MAILLE',
                     AFFE= (_F (GROUP_MA=' SURFLIBR',
                               PHENOMENE=' MECANIQUE',
                               MODELISATION=' 2D_FLUI_PESA',),
                            _F (GROUP_MA= ('FLUID0', 'BOTTOM', 'PLANCENT',),
                               PHENOMENE=' MECANIQUE',
                               MODELISATION=' 3D_FLUIDE',),
                            _F (GROUP_MA=' IFLUSTRU',
                               PHENOMENE=' MECANIQUE',
                               MODELISATION=' FLUI_STRU',),
                            _F (GROUP_MA=' RESERVOI',
                               PHENOMENE=' MECANIQUE',
                               MODELISATION=' COQUE_3D',),),),);
```

The group of meshs SURFLIBR (meshs of edge of the fluid field, located on its higher face) a model of free surface [R4.02.04] carries: 2D_FLUID_PESA .
Groups of meshs FLUID0 (massive fluid field), BOTTOM (meshs of edge of FLUID0 defining the bottom) and PLANCENT (meshs of edge of FLUID0 defining the symmetry plane) define the total fluid field: 3D_FLUIDE .
The interface fluid-structure (FLUI_STRU) is carried by the group of meshs IFLUSTRU .
Lastly, grid of the tank (RESERVOI) is the support of the model structure in voluminal hulls (COQUE_3D).

One defines also fluid compressible material:

```
EAU=DEFI_MATERIAU (FLUIDE=_F (RHO=1000.0, CELE_R=1500.0,),);
```

that one will assign to the fluid field (massive model and its edges) like with the interface IFLUSTRU and at free surface SURFLIBR :

```
CHMAT=AFFE_MATERIAU (MAILLAGE=MAILLA2,
             AFFE=  (_F  (GROUP_MA=  ('FLUID0',  'BOTTOM',  'PLANCENT',
'IFLUSTRU', 'SURFLIBR',),
      MATER=EAU,),
             …
               );
```

4.3 Boundary conditions

The boundary conditions kinematics relate to embedding in base of tank (SYMETRI2), on the generators in the vertical plan of symmetry (SYMMETRY) and on nonthe penetration (worthless normal speed) at the bottom of the fluid field (BOTTOM), like in its vertical plan of symmetry (PLANCENT).

```
CONDLIM=AFFE_CHAR_MECA (MODELE=MODELE,
                    DDL_IMPO= (_F (GROUP_NO=' SYMETRI2',
                                  DX=0.0, DY=0.0, DZ=0.0,
                                  DRX=0.0, DRY=0.0, DRZ =0.0,),
```

```
                                    _F (GROUP_NO=' SYMETRIE',
                                       DY=0.0, DRX=0.0, DRZ=0.0,),),
                                  VITE_FACE=_F  (GROUP_MA=  ('BOTTOM',  'PLANCENT',),
VNOR=0.0,),);
```

The effects of gravity are taken into account:

```
PESA=AFFE_CHAR_MECA (  MODELE=MODELE,
                       PESANTEUR=_F (   GRAVITE=9.81,
                                    DIRECTION= (0.0, 0.0, - 1.0),),);
```

However this order is insufficient because in modeling (u, p, φ) [bib7] the effects of gravity in the fluid field cannot be taken into account. If nothing moreover were done, gravity thus would be truly imposed only on the structure.
To approach the effects of gravity of the fluid on the wall, one will force an equivalent hydrostatic pressure (but which cannot take account of the variation height of the fluid during calculation when shaking will start):

```
PHZ=DEFI_FONCTION (NOM_PARA=' Z',
…
                  VERIF=' CROISSANT',);
#
PH = FORMULA (NOM_PARA= ('X', 'Y', 'Z'), VALE=' PHZ (Z) ',),
#
PRESSHYD=AFFE_CHAR_MECA_F (MODELE=MODELE,
                           FORCE_COQUE=_F (GROUP_MA=' VIROLE', PRES= PH,
                                   PLAN=' INF',),);
```

The earthquake is imposed as being a accélérogramme (function GASDM_X1) imposed in base according to the direction X. It is thus a request of type classical mono-support:

```
      ACCELERX=CALC_FONCTION (COMB= (_F (FONCTION=GASDM_X1, COEF=0.5,),),);
      #
      MULT_X=CALC_CHAR_SEISME (MATR_MASS=MATMAS,
                              DIRECTION= (1.0, 0.0, 0.0,), _APPUI=' OUI',);
      #
CHARG_SE=AFFE_CHAR_MECA (MODELE=MODELE, VECT_ASSE=MULT_X,);
```

The matrix of mass used (MATMAS) is the matrix of mass of the total coupled system.

4.4 Initial conditions

The initial state of transitory calculation must correspond to the balance of the total system when it is not subjected to the earthquake. This state of static balance thus corresponds to the loading of gravity and the hydrostatic effects.
If one began dynamic calculation with an initial state not respecting this balance, then that would generate oscillations of the transitory solution since it would not be initially with balance (the seismic level being however then null). One can attenuate these oscillations by adding a structural damping "large" and while waiting for that the solution is stabilized before imposing the earthquake, but this technique is rather not very elegant…
To calculate this statically balanced initial state, one can thus solve a static problem (which one supposes in linear) of balance under the action of the forces of gravity and hydrostatic.
With this intention, one calculates and assembles beforehand the total matrices K and M with the hydrostatic loading and gravity:

```
ASSEMBLY (MODEL = MODEL, CHAM_MATER=CHMAT,
                       CARA_ELEM=CARAELEM,  LOAD       = (CONDLIM,  WEIGHED,
PRESSHYD,),
```

```
                    NUME_DDL   = CO ('NUMSTA'),
                    SOLVEUR=_F (METHODE=' MULT_FRONT', RENUM=' METIS'),
                    MATR_ASSE= (
                     _F (MATRICE= CO ('RIGSTA'), OPTION= 'RIGI_MECA'),
                     _F (MATRICE= CO ('MASSTA'), OPTION= 'MASS_MECA'),),);
```

The matrix of stiffness assembled being singular because of fluid field (the formulation (U, p, φ) this matrix makes singular at worthless frequency [bib8]), one modifies the problem slightly by considering the matrix of stiffness $K_{cor}=K+\varepsilon M \quad \varepsilon \ll 1$ who is not singular any more (one names it RIGICOMB).
One can, for example, to take $\varepsilon=0{,}001$ as below:

```
RIGICOMB = COMB_MATR_ASSE (COMB_R= (_F (MATR_ASSE= RIGSTA, COEF_R= 1.),
                                    _F (MATR_ASSE= MASSTA, COEF_R= -0,001),);
```

One assembles then the vector loading F_0 (second member):

```
E_0 = CALC_VECT_ELEM (CARA_ELEM=CARAELEM, CHAM_MATER=CHMAT,
                             OPTION=' CHAR_MECA', CHARGE= (CONDLIM, WEIGHED,
PRESSHYD,),);
F_0      = ASSE_VECTEUR (VECT_ELEM= E_0, NUME_DDL= NUMSTA);
```

One can then solve the linear problem of statics $K_{cor}\underline{U}=\underline{F_0}$, while using, for example, a factorization of the type LDL^T :

```
RIGICOMB = TO FACTORIZE (reuse=RIGICOMB, MATR_ASSE= RIGICOMB,
                       STOP_SINGULIER= 'NOT');
DEP0     = TO SOLVE (MATR_FACT= RIGICOMB, CHAM_NO=F_0);
```

The field solution DEP0 will be thus the initial state of the transitory dynamic calculation which follows.

4.5 Transitory resolution

The operator is used DYNA_NON_LINE ([U4.53.01], [R5.05.05]) as follows:

```
RESU=DYNA_NON_LINE (MODELE=MODELE,
                   CHAM_MATER=CHMAT,
                   CARA_ELEM=CARAELEM,
                   EXCIT= (_F (CHARGE=CONDLIM,),
                          _F (CHARGE=PESA,),
                          _F (CHARGE=PRESSHYD,
                             FONC_MULT=FONCMUL0,
                             TYPE_CHARGE=' SUIV',),
                          _F (CHARGE=CHARG_SE,
                             FONC_MULT=ACCELERX,),),
                   COMPORTEMENT= (_F (RELATION=' ELAS',
                             DEFORMATION=' PETIT_REAC',
                             GROUP_MA= ('FLUID0', 'BOTTOM', 'PLANCENT',
                                       'IFLUSTRU', 'SURFLIBR',),),
                                        _F (RELATION=' ELAS', DEFORMATION='
PETIT_REAC',
                                 GROUP_MA=' ANNEAU',),
                              _F (RELATION=' VMIS_ISOT_TRAC',
                                 DEFORMATION=' PETIT_REAC',
                                 GROUP_MA= ('SURF0', 'SURF1',
                                           'SURF2', 'SURF3',),),),),
                   ETAT_INIT=_F (INST_ETAT_INIT=0.0, DEPL=DEP0,),
```

```
                    INCREMENT=_F (LIST_INST=LINST,),
                    SCHEMA_TEMPS=_F (SCHEMA=' HHT', ALPHA=-0.1,
                                    FORMULATION=' DEPLACEMENT',),
                              NEWTON=_F  (REAC_INCR=1,  MATRICE='  TANGENTE',
REAC_ITER=1,),
                    SOLVEUR=_F (STOP_SINGULIER=' NON',),
                    CONVERGENCE=_F (RESI_GLOB_RELA=1.e-05, ITER_GLOB_MAXI=20,
                                   ARRET=' OUI',),
                    ARCHIVAGE=_F (LIST_INST=LARCH),);
```

The resolution is done into Lagrangian reactualized (option DEFORMATION=' PETIT_REAC') because the fluid field is in small disturbances on each step. It thus should be checked that the step of time is sufficiently small so that this assumption is checked.

One uses a diagram of integration in times of type modified average acceleration (SCHEMA=' HHT', ALPHA=-0.1) with digital damping in order to stabilize the solution and to facilitate convergence.

CAS-test FDNV100 [V8.03.100] presents the calculation of a rectangular tank full of water with a flexible wall. The modeling put in work is very close to that used here for the large tanks.

4.6 Use of the nonlinear criterion of stability

One can also use a criterion of stability just like into quasi-static. The presence of the fluid requires some specific options however. Indeed, the matrix of total stiffness assembled of the problem coupled fluid-structure is intrinsically singular (cf. documentation [R4.02.02]) on the level of the fluid degrees of freedom. It is thus advisable to exclude these degrees of freedom from the analysis of stability, but also to modify the matrix of stiffness (as well as the geometrical matrix of stiffness when it is used). For that, it is necessary to inform the keyword following, under CRIT_STAB :

- MODI_RIGI = 'YES' ,
- DDL_EXCLUS= ('PHI', 'NEAR', 'DH',) .

The list of the excluded degrees of freedom must comprise all the types of degrees of freedom related to the fluid model: in the example of CAS-test FDNV100 there is thus the potential PHI, pressure NEAR and vertical displacement on the level of free surface **DH**. If this treatment is not made, then the call to CRIT_STAB will plant due to singular matrix and no strategy of shift could surmount that.

Into quasi-static, this problem does not arise because modeling coupled fluid-structure then does not have a direction.

5 Bibliography

1) S. Goubet: *Seismic design/checking of the large flat-bottomed tanks*. Note EDF SEPTEN ENGSDS030109.

2) NR. Greffet, A. Assire, J.Pigat, J. - M. Proix : *Prototype study of metal tanks to the earthquake - nonlinear Analysis of stability*. Note EDF R & D HT-62/02/011/B.

3) NR. GREFFET: *Digital study of a model of cover: comparison with the tests Freyssinet*. Note EDF R & D HT-62/03/009/A.

4) NR. GREFFET: *Prototype study of metal tanks to the earthquake - nonlinear Analysis of stability*, Acts of the VI^{E} National conference AFPS, vol. II, p. 225-232, July 1st-3rd, 2003.

5) NR. GREFFET: *Modeling of separation for the simulation of a cover under lawful seismic loading*. Note EDF R & D HT-62/04/026/A.

6) NR. GREFFET: *Nonlinear instability of large tanks under earthquake - lawful approach towards the transient coupled fluid-structure*, Acts of the 7^E National conference in Calculation of the Structures, vol. 1, p. 265-270, Giens, May 17th-20th, 2005

7) H.J. - P. Morand, R. Ohayon: *Interactions fluid-structures*, Masson Editions, 1992.

8) NR. Greffet: V*hear of improvement of the coupled formulation fluid-structure in Code_Aster*, Notes EDF R & D HT-62/02/023/A.