Note of use of the voluminal elements plates, shells, shells SHB, grids and Résumé

membranes:

This document is a note of use for the voluminal modelizations plates, shells, shells SHB, grids and membranes.

The shell elements and of plates play a part in the numerical modelization of thin structures at mean surface, planes (modelization plates) or curve (modelization shells). The modelizations of grids intervene for the numerical modelization of reinforcements and the cables of prestressed in reinforced concrete structures. The modelization of membrane makes it possible to modelize thin structures whose flexural rigidity is negligible.

They are usable in or not linear linear mechanics and thermal.

Contents

1 Introduction4...
2 Mécanique6...
2.1 Capacités of modélisation6...
2.1.1 Rappel of the formulations6..
2.1.1.1 Géométrie of the elements plate and coques6...
2.1.1.2 Géométrie of the voluminal elements shells SHB7..
2.1.2 Formulation of the elements plates, voluminal shells and shells 9
2.1.2.1 Formulation into linear géométrique9...
2.1.2.2 Formulation into nonlinear geometrical, Flambement d' Euler11.............................
2.1.3 Formulation of the voluminal elements shells SHB ....
11.2.1.4 Comparaison between the éléments12...
2.1.4.1 Les differences between the elements plates, shells and shells voluminal SHB . 12
2.1.4.2 Les differences between the elements plaques13...
2.1.4.3 Les differences between the elements coques13...
2.1.4.4 the difference between voluminal shells SHB 14.2.2
Commands with utiliser14..
2.2.1 spatial Discretization and assignment in a modelization: operator AFFE_MODELE 14
2.2.1.1 Degrés of libertés14...
2.2.1.2 Mailles support of the matrixes of rigidité15..
2.2.1.3 Mailles support of the chargements16...
2.2.1.4 Modèle: AFFE_MODELE ....
16.2.2.2 Caractéristiques elementary: AFFE_CARA_ELEM ....
17.2.2.3 Materials: DEFI_MATERIAU ....
19.2.2.4 limiting Loadings and conditions: AFFE_CHAR_MECA and AFFE_CHAR_MECA_F 21
2.2.4.1 Liste of the key words factor of AFFE_CHAR_MECA 21
2.2.4.2 Liste of the key words factor of AFFE_CHAR_MECA_F 23
2.2.4.3 Application of a pressure: key word FORCE_COQUE 23
2.2.4.4 Conditions limiting: key words DDL_IMPO and LIAISON_* 24
2.2.4.5 Raccords shells with other elements mécaniques24...
2.2.4.6 Variables of commandes26...
2.3 Résolution26...
2.3.1 linear Computations: Linear MECA_STATIQUE and other operators ....
26.2.3.2 nonlinear Computations: STAT_NON_LINE and DYNA_NON_LINE 26
2.3.2.1 Comportements and assumptions of strains disponibles26.......................................
2.3.2.2 Détail on the points of intégration28..
2.3.2.3 Comportement nonlinear géométrique28..
2.3.2.4 Flambement linéaire29..
2.4 additional Computations and... post-traitements29

2.4.1 Computations of elementary matrixes: operator CALC_MATR_ELEM ....
29.2.4.2 Computations by elements: operator CALC_CHAMP ....
30.2.4.3 Computations with the nodes: operator CALC_CHAMP ....
32.2.4.4 Computations of quantities on whole or part of structure: operator POST_ELEM ....
32.2.4.5 Valeurs of components of fields of variables: operator POST_RELEVE_T ....
32.2.4.6 Printing of the results: operator IMPR_RESU 33.2.5
Exemples33...
2.5.1 linear Static analysis ....
34.2.5.2 Modal analysis in dynamique37...
2.5.3 nonlinear Static analysis matériau38..
2.5.4 nonlinear Static analysis géométrique39...
2.5.5 Analyse in buckling of Euler41..
2.5.6 Raccords shells and other elements mécaniques42...
3 Thermique44...
3.1 Définition of the problème44...
3.1.1 spatial Discretization and assignment of a modelization: operator AFFE_MODELE 44
3.1.1.1 Degrés of libertés44...
3.1.1.2 Meshes support of the matrixes of rigidité44..
3.1.1.3 Meshes support of the chargements45...
3.1.1.4 Model: AFFE_MODELE ....
45.3.1.2 Caractéristiques elementary: AFFE_CARA_ELEM ....
45.3.1.3 Materials: DEFI_MATERIAU ....
45.3.1.4 limiting Loadings and conditions: AFFE_CHAR_THER and AFFE_CHAR_THER_F . 46
3.1.4.1 Liste of the key words factor of AFFE_CHAR_THER 46
3.1.4.2 Liste of the key words factor of AFFE_CHAR_THER_F 46.3.2
Résolution47...
3.2.1 Transient computations: additional operator ....THER_LINEAIRE
47.3.3 Computations and.. post-traitements47
3.3.1 Computations in... post-traitement47
3.4 Exemples47...
4 Sequence thermomécanique48...
4.1 Formalisme48...
5 Conclusion and advices of utilisation50..
6 Bibliographie52...

1 Introduction

Les shell elements and of plates are particularly used for modelizing thin structures where the relationship between dimensions (thickness/characteristic length) are much lower than $1/10$ (thin shells) or about $1/10$ (thick shells).

These modelizations are usable in linear and nonlinear mechanics, under assumptions of small strains and small displacements or many assumptions of large displacements and large rotations, according to the modelizations. A mean modelization of shell is also available in transitory linear thermal.

Three categories of thin structural elements are described in this document:

- The shell elements , which are plane, therefore the curvature of structure to be represented is not ideally taken into account and it is necessary to use a large number of elements in order to approach correctly the geometry of the structure (aspect breakages).
- The shell elements , which are curved, therefore the geometry of structure is better approximate.
- The voluminal **or three-dimensional shell elements** SHB , which are by way of elements 3D nouveau riches, as well as possible apprehending the phenomena for low thickness.

Concerning the bill of materials and documentations of reference associated with each modelization:

- The plane shell elements triangle and quadrangle are gathered under the modelizations, (documentation of reference [R3.07.03]):
  - DKT : net TRIA3 element DKT, mesh QUAD4 elements DKQ (linear geometrical);
  - DKTG : net TRIA3 element DKT, mesh QUAD4 elements DKQ (linear or not linear geometrical);
  - DST : net TRIA3 element DST, mesh QUAD4 element DSQ (linear geometrical);
  - Q4G : net QUAD4 element Q4G (linear geometrical).
  - Q4GG : net TRIA3 element T3G, mesh QUAD4 elements Q4G (linear geometrical);
- The curved shell elements resulting from models 3D with kinematics of shell are gathered under the modelizations:
  - COQUE_3D : net TRIA7 and QUAD9, structure 3D with unspecified geometry ([R3.07.04] into linear geometrical, [R3.07.05] nonlinear geometrical and [R3.03.07] with following pressures);
  - COQUE_AXIS : net SEG3, shells with symmetry of revolution around the axis 0Y ([R3.07.02] into linear geometrical);
  - COQUE_C_PLAN or COQUE_D_PLAN : net SEG3, shells with invariant geometry along the axis OZ ([R3.07.02] into linear geometrical);
- The voluminal shell elements with isoparametric kinematics like elements 3D standards (into linear and nonlinear geometrical with following pressures) are gathered under the modelization (documentation of reference [R3.07.07a]):
  - SHB8 : net PENTA6 element SHB6 (linear formulation);
    net HEXA8 element SHB8 (linear formulation) ;
    net PENTA15 element SHB15 (quadratic formulation);
    net HEXA20 element SHB20 (quadratic formulation).
- conventional elements of membrane:
  - MEMBRANE : net TRIA3, QUAD4, TRIA6 and QUAD8 (linear geometrical)

  They are elements of membrane with a simple membrane rigidity (not of degree of freedom of rotation). There is no eccentring.
- specific shell elements to represent the three-dimensions functions of reinforcement:
  - GRILLE_EXCENTRE : net TRIA3 or QUAD4 (linear geometrical)
    modelization GRILLE_EXCENTRE corresponds to orthotropic shell elements DKT to 3 or 4 nodes offset compared to the average concrete layer, (only one direction of reinforcements). This last is modelized by shell elements DKT or DST with 3 nodes. The reinforced concrete structure is then represented by the superposition of modelizations GRILLE_EXCENTRE and that used for concrete (DKT or DST).

* GRILLE_MEMBRANE : meshes TRIA3, QUAD4, TRIA6, QUAD8 (Doc. [R3.08.07]): they are elements of reinforcement (only one direction of reinforcements) working only out of membrane (not of degree of freedom of rotation). There is no eccentring.

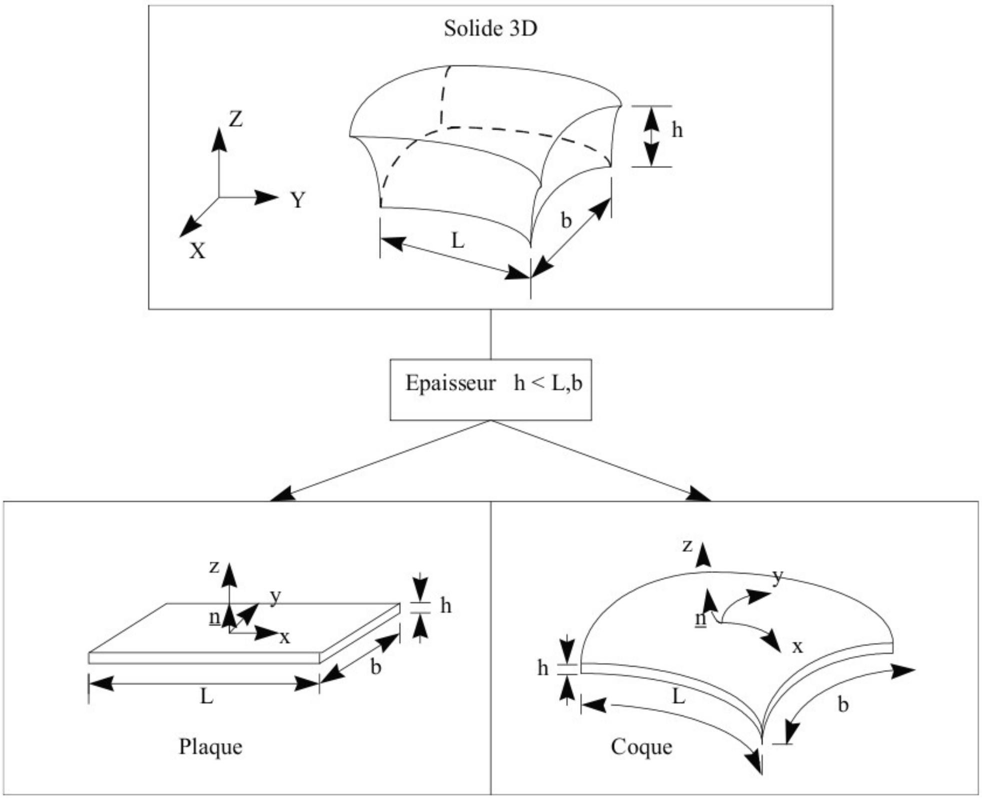
2 Mécanique

2.1 Capacités of Rappel

2.1.1 modelization of the Géométrie

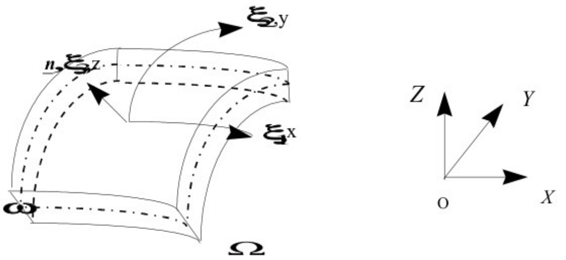
2.1.1.1 formulations of the elements plates and Pour

shells the elements plates and shells one defines a surface of reference, or mean surface, planes (plane xy for example) or curve (x and y define a set of curvilinear coordinates) and a thickness $h(x, y)$. This thickness must be small compared to other dimensions of structure to modelizing. The figure below illustrates these various configurations.



Appear 2.1.1.1 - has: Assumption in theory of the plates and the shells

One attaches to mean surface ω a local coordinate system $Oxyz$ different from the total reference $OXYZ$. The position of the points of the plate or the shell is given by the curvilinear coordinates $(\xi_{1,} \xi_2)$ of mean surface and the front elevation ξ_3 compared to this surface. For the curvilinear plates the coordinate system is a coordinate system Cartesian local.

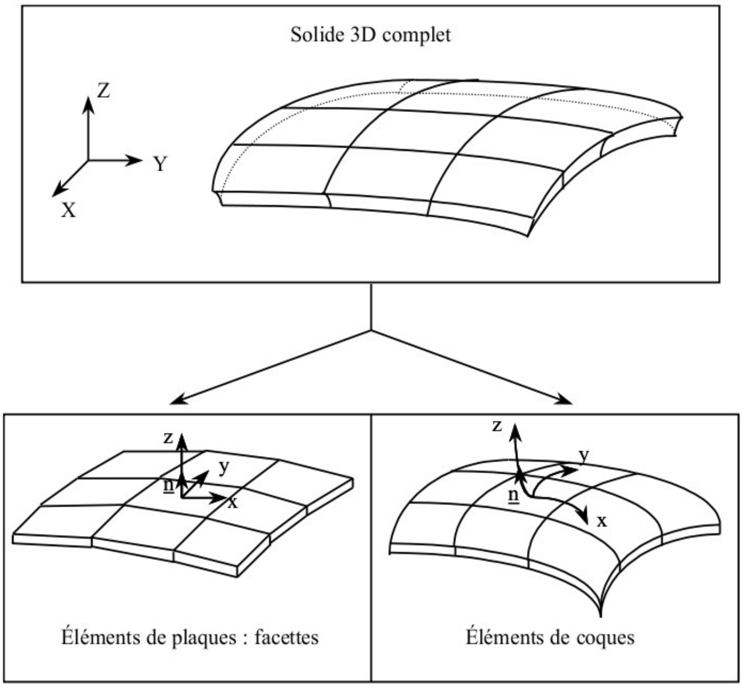


Appear 2.1.1.1 - B: Definition of a Pour

mean surface to represent shells with symmetry of revolution around an axis (COQUE_AXIS) or shells with invariant geometry by translation (COQUE_C_PLAN in plane stresses or COQUE_D_PLAN in plane strains), the knowledge of the section of revolution or the trace of mean surface is sufficient, as the figure [Figure 2.1.2.1 - has] shows it to us. These shells lean on a linear mesh and in a point m of mean surface one defines a local coordinate system (n, t, e_z) by:

$$t=\frac{Om_{,s}}{\|Om_{,s}\|}, \; n\wedge t=e_z$$

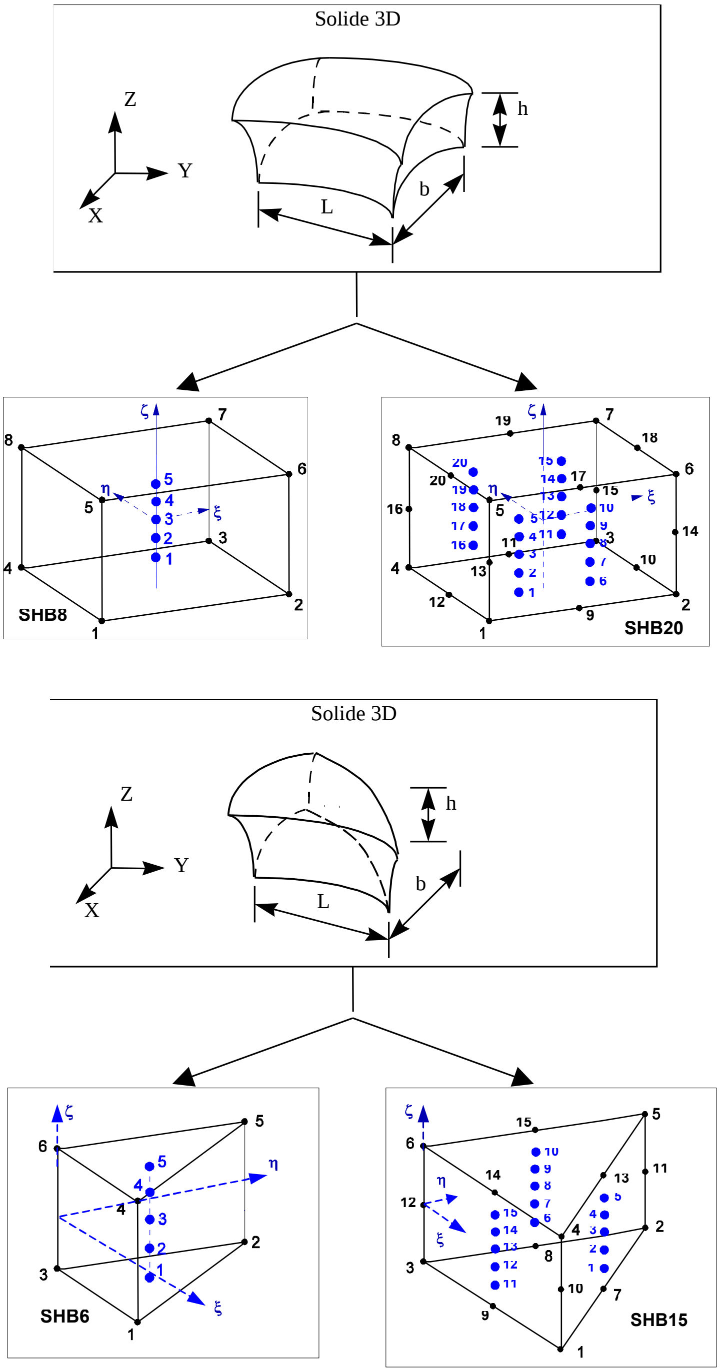
When one wishes to modelize a solid of an unspecified form (not plane), one can use shell elements to account for the curvature, or many shell elements. In this last case, the geometry is approximated by a network of breakages.



Appear 2.1.1.1 - C: Modelization of an unspecified solid 3D by shell elements or Géométrie

2.1.1.2 shells of the voluminal elements shells SHB

Les voluminal elements shells SHB are elements of continuous three-dimensional geometry in which a privileged direction, called thickness, was selected. This thickness is, in general, small compared to other dimensions of structure to modelizing. This direction of the thickness is defined by the way of netting. In general, one needs two surfaces to generate a three-dimensional volume. The direction which is perpendicular to these two surfaces is the direction of the thickness. These elements can thus be used to modelize thin structures and to take into account the phenomena which develop in the thickness in the frame of the three-dimensional mechanics of the continuums. In linear approach, there is the prismatic element with six nodes SHB6 and the hexahedral element with eight nodes SHB8. In quadratic approach, there is the prismatic element with fifteen nodes SHB15 and the hexahedral element with twenty nodes SHB20. The figure below illustrates these various modelizations:



Appear 2.1.1.2 - has: Geometries of the elements of reference and points of Formulation

2.1.2 integration of the elements plates, shells and voluminal shells

2.1.2.1 Formulation into linear geometrical

Dans this formulation, one supposes that displacements are small, one can thus superimpose the initial geometry and the deformed geometry. These elements (except SHB) are based on the theory of the shells according to which:

- the cross-sections which are the sections perpendicular to the surface of reference remain right; the material points located on a norm at not deformed mean surface remain on a line in the deformed configuration. It results from this approach that **the fields of displacement vary linearly in the thickness of the plate or the shell**. If one indicates by u , v and w displacements of a following $q(x,y,z)$ point x , y and z , one has as follows:

$$\begin{Bmatrix} u(x,y,z) \\ v(x,y,z) \\ w(x,y,z) \end{Bmatrix} = \begin{Bmatrix} u(x,y) \\ v(x,y) \\ w(x,y) \end{Bmatrix} + z \begin{Bmatrix} \beta_x(x,y) \\ \beta_y(x,y) \\ 0 \end{Bmatrix}$$

The associated strain tensor is written then: $\boldsymbol{\varepsilon}(x,y,z) = \boldsymbol{e}(x,y) + \boldsymbol{\gamma}(x,y) + z.\boldsymbol{\chi}(x,y)$.

The first term $\boldsymbol{e}$ understands the strains of membrane (for a shell element they are the strains in the plane of the element), the second $\boldsymbol{\gamma}$ those of transverse shears, and the third $z.\boldsymbol{\chi}$ the strains of bending, associated with the tensor of curvature $\boldsymbol{\chi}$. For the thick plates or shells the transverse shears $\boldsymbol{\gamma}$ are taken into account according to the formulation suggested by Reissner, Hencky, Bollé, Mindlin. This formulation includes the approach without transverse shears, where the tensor $\boldsymbol{\gamma}$ null, is developed by Kirchhoff for the thin plates or shells according to which the material points located on a norm $\boldsymbol{n}$ at not deformed mean surface remain on the norm on the deformed surface.

- **The transverse stress σ_{zz} null** because is regarded as negligible compared to the other components of the tensor of the stresses (assumption of the plane stresses).

- One does not describe the variation of the thickness nor that of the transverse strain ε_{zz} which one can compute by using the preceding assumption of plane stresses.

- The taking into account of the transverse shears depends on factors of correction determined a priori by energy equivalences with models 3D, so that rigidity in transverse shears of the model of plate is nearest possible to that defined by the theory of three-dimensional elasticity. For the homogeneous plates, the factor of transverse correction of shears based on this method is $k=5/6$.

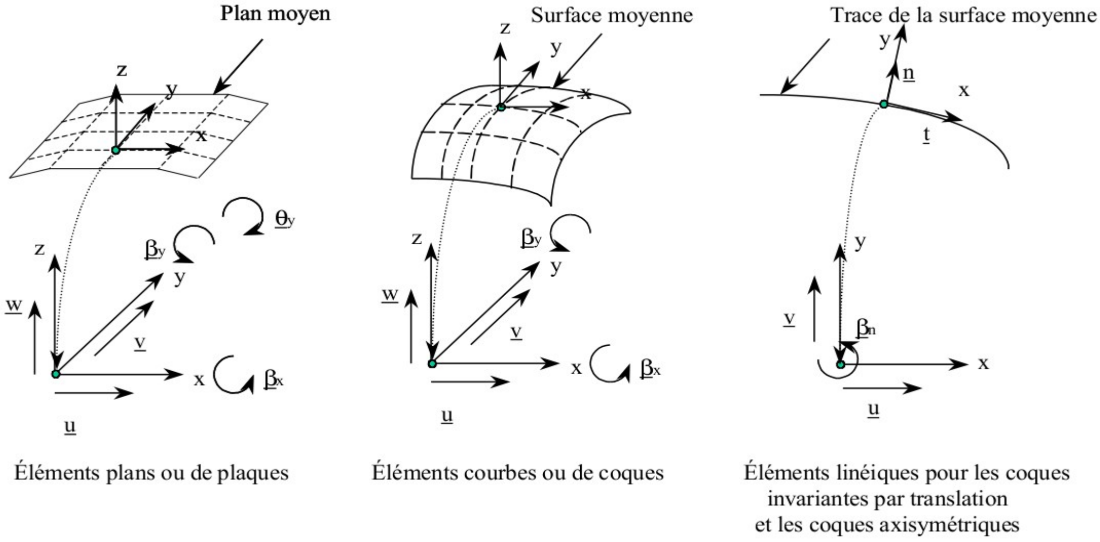
Note:

The determination of the factors of correction rests for Mindlin on equivalences of eigenfrequency associated with the mode with vibration by transverse shears. One obtains then $k=\pi^2/12$, value very close to $5/6$.

These elements utilize locally:

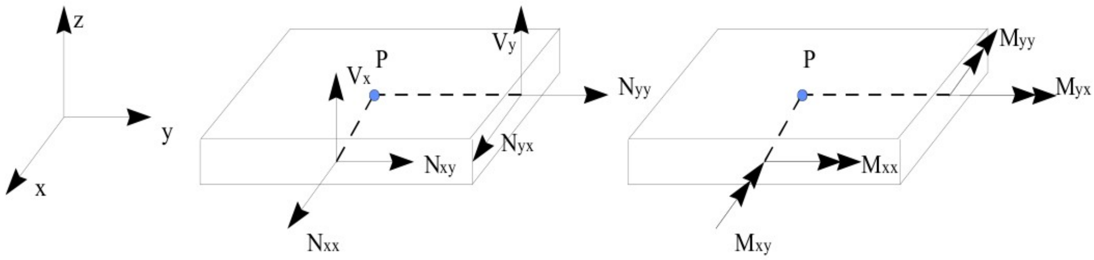
- Five kinematical variables for the unspecified elements plates and shells; displacements of membrane u and v in the datum-line $z=0$, transverse displacement w and rotations β_x and β_y of the norm at mean surface in the planes yz and xz respectively.

- Three kinematical variables for the linear elements; displacements u and v in the datum-line $z=0$ and the rotation β_n of the norm to mean surface in the plane xy .



Appear 2.1.2.1 - has: Kinematical variables for the various shell elements and of shells

- Trois forces resulting from membrane noted Nxx, Nyy, Nxy and three noted bending moments Mxx, Myy, Mxy whatever the shell element or from shell; two noted shearing forces Vx and Vy in the case of shell elements and of unspecified shells.



Appear 2.1.2.1 - B: Resulting forces for a shell element or of Formulation

2.1.2.2 shell into nonlinear geometrical, Flambement d' Euler

Dans the formulation into **nonlinear geometrical**, one is in the presence of large displacements and of large rotations, one cannot superimpose the initial geometry and the deformed geometry.

The formulation, described in documentation of reference [R3.07.05], is based on an approach of continuum 3D, is degenerated by the introduction of the kinematics of shell of the type Hencky - Mindlin - Naghdi in plane stresses into the weak formulation of the equilibrium. The measurement of the strains selected is that of Green-Lagrange, vigorously combined with the Piola-Kirchhoff stresses of second species. The formulation of the equilibrium is thus a total Lagrangian formulation. The transverse shears are treated same manner as in the linear case [R3.07.04].

The element retained into nonlinear is a voluminal shell element (`COQUE_3D`) of curved mean surface as presented to the preceding paragraph, whose meshes supports are `QUAD9` and `TRIA7`.

It is possible to apply to these elements of the following pressures, whose formulation is described in the reference document [R3.03.07]. This loading with the characteristic to follow the geometry of structure during its strain (for example, the hydrostatic pressure remains always perpendicular to the deformed geometry).

Linear buckling also called **buckling of Euler**, described in documentation of reference [R3.07.05], is presented in the form of a typical case of the geometrical nonlinear problem. It is based on a linear dependence of the fields of displacements, strains and stresses compared to the level of load.

The element retained in linear buckling is the voluminal shell element (`COQUE_3D`) of curved mean surface as presented to the preceding paragraph, whose meshes supports are `QUAD9` and `TRIA7`.

2.1.3 Formulation of the voluminal elements shells SHB

Les voluminal elements shells `SHB (SHB6, SHB8, SHB15 and SHB20)` are isoparametric. Their formulations are established by lean on voluminal meshes 3D standards (successively penta6, hexa8, penta15 and hexa20) with only 3 degrees of freedom of displacement to each node of the element.

The formulations as of these elements are described in documentation of reference [R3.07.07] and [R3.07.08]. Four elements `SHB` rest on under-integrated formulations. The basic idea first of all consists in making sure that there are sufficient Gauss points in the thickness to represent the phenomenon of bending correctly, then compute of rigidities of stabilization in an adaptive way according to the plastic state of the element. That represents an unquestionable improvement compared to the conventional formulations for the forces of stabilization, because these last rest on an elastic stabilization which becomes too rigid when the effects of plasticity dominate the response of structure.

The principle of the formulations of elements `SHB` is rather identical, but there exist some differences between the elements:

- element `SHB8` under-is integrated by five Gauss points laid out in the direction of the thickness and is stabilized by the postulated introduction of a strain field "assumed strain";
- element `SHB6` under-is also integrated by five Gauss points laid out in the direction of the thickness. This element not presenting mode of hourglass, stabilization is thus not necessary. To improve his speed of convergence, one introduces amendments of the type "assumed strain" on the operator gradient discretized of the element;
- quadratic elements `SHB15` and `SHB20` under-are integrated respectively by fifteen and twenty Gauss points in the direction of the thickness. They do not need stabilization or amendment of the type "assumed strain".

Into nonlinear geometrical, elements `SHB` are treated in the case of large displacements, weak rotations and small strains. One adopts for that an upgraded Lagrangian formulation. In the case of small displacements, one confuses geometry in beginning and end of pitch, stress of Cauchy and Piola Kirchoff 2, moreover one uses the linear statement of the strains.

A characteristic of elements SHB is that the elastic matrix of behavior C is selected in the following form (written in the local coordinate systems):

$$C=\begin{bmatrix} \bar{\lambda}+2\mu & \bar{\lambda} & 0 & 0 & 0 & 0 \\ \bar{\lambda} & \bar{\lambda}+2\mu & 0 & 0 & 0 & 0 \\ 0 & 0 & E & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

where E is the Young modulus, ν the Poisson's ratio, $\mu=\frac{E}{2(1+\nu)}$ the shear modulus and $\bar{\lambda}=\frac{E\nu}{1-\nu}$ the coefficient of modified Lamé. This model is specific with elements SHB . It resembles that which one would have in the case of the assumption of the plane stresses, put except for the term $(3,3)$. One can note that this choice involves an artificial anisotropic behavior. This choice makes it possible to satisfy all the tests without introducing blocking.

In nonlinear material, a method of particular construction of the tangent matrix C^T is used. It consists in supposing initially that the element is in plane stress state in the local coordinate system of each point of Gauss and the strains except plane are elastic. That involves then immediately that the total deflections except plane are equal to the elastic strain. Let us call C^{CPT} the tangent matrix in plane stresses. The tangent matrix of behavior for the selected behavior is obvious and is written:

$$C^T=\begin{bmatrix} C^{CPT}_{xxxx} & C^{CPT}_{xxyy} & 0 & C^{CPT}_{xxxy} & 0 & 0 \\ C^{CPT}_{xyyx} & C^{CPT}_{yyyy} & 0 & C^{CPT}_{yyxy} & 0 & 0 \\ 0 & 0 & E & 0 & 0 & 0 \\ C^{CPT}_{xyxx} & C^{CPT}_{xyyy} & 0 & C^{CPT}_{xyxy} & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix}$$

Then, the stresses except plane are computed in an elastic way. This method thus makes it possible to connect elements SHB to all the constitutive laws available in *Code Aster* .

2.1.4 Comparison between the elements

2.1.4.1 Les differences between the elements plates, shells and shells voluminal SHB

Les shell elements are curved elements whereas the shell elements are plane. The variation of metric of the geometry (i.e. its radius of curvature) according to its thickness is taken into account for the shell elements but not for the shell elements. This variation of metric implies a coupling between the effects of membrane and bending for nonplane structures which cannot be observed with shell elements plane for a homogeneous material (see [bib1]).

The choice of the shape functions for the discretization of these elements is different because the curved shell elements have a more significant number of degrees of freedom. Thus, the shell elements are linear elements out of membrane whereas the shell elements are quadratic.

The voluminal elements shells SHB lean on meshes 3D. There are no thus problems of connection with solid elements 3D. Therefore, elements SHB are usable for modelizing thin structures which coexist with solid elements 3D. Moreover, it is more practical to use elements SHB for modelizing structures of which the thickness varies continuously. The discretization of elements SHB is either linear for SHB6 and SHB8, or quadratic for SHB15 and SHB20 .

2.1.4.2 The differences between the elements plates

One distinguishes the elements with transverse shears (DST, DSQ and Q4G) from the elements without transverse shears (DKT and DKQ). Elements DST and DKT have triangular meshes support with 3 nodes ($3\times5=15$ degrees of freedom) and elements DKQ, DSQ and Q4G of the meshes quadrangular supports to 4 nodes ($4\times5=20$ degrees of freedom).

Notice important :

> *For the shell elements with 4 nodes (DSQ , DKQ and Q4G), the 4 nodes must be coplanar so that the theory of the plates can be validated. This checking is carried out systematically by Code_Aster, and the user is alarmed if one of the elements of the mesh does not observe this condition.*

In the case of the elements with transverse shears, to avoid the blocking of the elements in transverse shears (overestimation of rigidity for very low thickness), a method consists in building fields of constant shears of substitution on edges of the element, whose value is the integral of the shears on edge in question. In *Code_Aster*, the shell elements and shell with transverse shears use this method in order not to lock in transverse shears. This blocking in shears comes owing to the fact that the elastic strain energy of shears is a term proportional to h (h being the thickness of the plate or the shell) much larger than the term of elastic strain energy of bending which is proportional in h^3 . When the thickness becomes low in front of the characteristic length (the report h/L is lower than $1/20$), for certain shape functions, the minimization of the dominating term in h conduit with a bad representation of the modes of pure bending, for which the deflection is not computed any more correctly (see [bib1] page 295 with $h/L=0.01$).

Element Q4G is a quadrilateral element with four nodes without blocking in transverse shears, with bilinear shape functions in x and y to represent w , β_x and β_y .
The main difference between modelization Q4G and DST comes owing to the fact that one uses for the latter of the shape functions including a quadratic term to discretize rotations β_s where $s=(x, y)$.
The consequence for element Q4G is a constant approximation per pieces of the curvatures which implies to net sufficiently fine in the directions requested in bending.

2.1.4.3 The differences between the elements shells

One distinguishes linear shell elements COQUE_C_PLAN, COQUE_D_PLAN and axisymmetric COQUE_AXIS of the elements of COQUE_3D.

The first are used for modelizing invariant structures according to the axis Oz or of revolution of axis Oy and the seconds in all the other cases. In the case of the invariant shells according to the direction z , one distinguishes the free shells in z (plane stresses COQUE_C_PLAN) from the shells locked in z (plane strains COQUE_D_PLAN). For these shell elements, the meshes supports are linear with 3 nodes. The number of degrees of freedom of these elements is of 9.

Unspecified shell elements COQUE_3D have triangular with 7 nodes or quadrangular meshes support with 9 nodes:

- In the case of triangular meshes, the number of degrees of freedom for the translations is 6 (the unknown factors are displacements with the top nodes and on the mediums on the sides of the triangle) and that of rotations is 7 (the unknown factors are 3 rotations at the preceding points and the center of the triangle). The total number of degrees of freedom of the element is thus of $Nddle=3\times 6+3\times 7=39$.

- In the case of quadrangular meshes with 9 nodes, the number of degrees of freedom for the translations is 8 (the unknown factors are displacements with the top nodes and on the mediums on the sides of the quadrangle) and that of rotations is 9 (the unknown factors are 3 rotations at the preceding points and the center of the quadrangle). The total number of degrees of freedom of the element is thus of $Nddle=3\times 8+3\times 9=51$. These elements thus have about twice more degrees of freedom than the shell elements of family DKT agents. Their cost in time, with equal number, in a computation will be thus more important.

The elements of COQUE_3D automatically take into account the correction of metric between mean surface and the surfaces upper and lower. For the linear elements, this correction must be activated by the user (see paragraph 17). The correction of metric contributes a share in h/L to the stress and $(h/L)^2$ of displacement (see [V7.90.03]). For the plates this correction is without object.

For the shell elements the coefficient of correction of shears k in isotropic behavior can be modified by the user. This coefficient of correction of shears is given in AFFE_CARA_ELEM under key word A_CIS. By default, if the user does not specify anything in AFFE_CARA_ELEM that amounts using the theory with shears of REISSNER; the shear coefficient is then put at $k=5/6$. If the shear coefficient k is worth 1 one places oneself in the frame of the theory of HENCKY-MINDLIN_NAGHDI and if it becomes very large ($k>10^{6}\cdot h/L$) one approaches the theory of LOVE_KIRCHHOFF.

In practice it is advised not to change this coefficient. Indeed, these elements provide a physically correct solution, whether the shell is thick or thin, with the coefficient $k=5/6$.

2.1.4.4 The difference between voluminal shells SHB

Les elements SHB have different meshes support and formulations:
- element SHB6 has as a mesh support the pentahedron with six nodes. Its formulation is linear. Its operator discretized gradient is modified to improve his velocity of convergence;
- element SHB8 has as a mesh support the hexahedron with eight nodes. Its formulation is linear. It is stabilized by method “assumed strain” to have good performances;
- element SHB15 has as a mesh support the pentahedron with fifteen nodes. Its formulation is quadratic. It does not have stabilization;
- element SHB20 has as a mesh support the hexahedron with twenty nodes. Its formulation is quadratic. It does not have stabilization.

2.2 Commands to use

2.2.1 spatial Discretization and assignment of a modelization: operator AFFE_MODELE

Dans this part, one describes the choice and the assignment of one of the modelizations plates or shell as well as the degrees of freedom and the associated meshes. Most described information are extracted from documentations of use of the modelizations ([U3.12.01]: Modelization DKT - DST - Q4G, [U3.12.02]: Modelizations COQUE_C_PLAN, COQUE_D_PLAN, COQUE_AXIS, [U3.12.05]: Modelization SHB8).

2.2.1.1 Degrees of freedom

Les degrees of freedom of discretization are in each node of the mesh support the components of displacement to the nodes of the mesh support, except indication.

Modelization	Degrees of freedom (with each	Remarques

	node)	
COQUE_3D	DX DY DZ DRX DRY DRZ DRX DRY DRZ with the central node	Les nodes belong to the average layer of shell
DKT, DKTG	DX DY DZ DRX DRY DRZ	Les nodes belong to the tangent breakage with the average layer of shell
DST	DX DY DZ DRX DRY DRZ	Les nodes belong to the tangent breakage with the average layer of shell
Q4G	DX DY DZ DRX DRY DRZ	Les nodes belong to the tangent breakage with the average layer of shell
Q4GG	DX DY DZ DRX DRY DRZ	Les nodes belong to the tangent breakage with the average layer of shell
COQUE_C_PLAN	DX DY DRZ	Les nodes belong to the mean surface of shell
COQUE_D_PLAN	DX DY DRZ	Les nodes belong to the mean surface of shell
COQUE_AXIS	DX DY DRZ	Les nodes belong to the mean surface of shell
GRILLE_EXCENTRE	DX DY DZ DRX DRY DRZ	Les nodes belong to the tangent breakage to the average layer of the shell.
SHB8	DX DY DZ	Degree of freedom with all the nodes.
GRILLE_MEMBRANE	DX DY DZ	Degree of freedom with all the nodes.
MEMBRANE	DX DY DZ	Degree of freedom with all the nodes.

2.2.1.2 Meshes support of the Modélisation

stiffness matrixes	Nets	Finite element	Remarques
COQUE_3D	TRIA7 QUAD9	MEC3TR7H MEC3QU9H	Mailles not presumedly plane
DKT, DKTG	TRIA3 QUAD4	MEDKTR3 MEDKQU4	plane Mailles
DST	TRIA3 QUAD4	MEDSTR3 MEDSQU4	plane Mailles
Q4G,	QUAD4	MEQ4QU4	plane Mailles
Q4GG	QUAD4 TRIA3	MEQ4GG4 MET3GG3	plane Mailles
COQUE_C_PLAN	SEG3	METCSE3	Mailles not presumedly plane
COQUE_D_PLAN	SEG3	METDSE3	Mailles not presumedly plane
COQUE_AXIS	SEG3	MECXSE3	Mailles not presumedly plane
GRILLE_EXCENTRE	TRIA3, QUAD4	MEGCTR3 MEGCQU4	plane Mailles
GRILLE_MEMBRANE	TRIA3, QUAD4, TRIA6, QUAD8	MEGMTR3 MEGMQU4 MEGMTR6 MEGMQU8	surface Mailles unspecified
MEMBRANE	TRIA3, QUAD4, TRIA6, QUAD8	MEMBTR3 MEMBQU4 MEMBTR6 MEMBQU8	Mailles surface unspecified
SHB8	PENTA6 HEXA8 PENTA15 HEXA20	MECA_SHB6 MECA_SHB8 MECA_SHB15 MECA_SHB20	Mailles 3D Meshes 3D Meshes 3D Meshes 3D

modelization `GRILLE_EXCENTRE` used to modelize reinforced concrete structures has the same characteristics of mesh as modelization `DKT`

Remarque:

> *In a mesh, to transform meshes `TRIA6` into meshes `TRIA7`, or `QUAD8` in meshes `QUAD9` , one can use operator `MODI_MAILLAGE` .*

2.2.1.3 Meshes support of the Tous

loadings the loadings applicable to the breakages of the elements used here are treated by direct discretization on the mesh support of the element in displacement formulation. The surface pressure and the other forces as well as gravity are examples of loadings applying directly to the breakages. No special mesh of loading is thus necessary for the sides of the shell elements, of shells. Elements `SHB` have the same edge elements as elements `3D` standards.

For the applicable loadings on edges of the elements, one a:

Modelization	Nets	Finite element	Remarques
COQUE_3D	SEG3	MEBOCQ3	
DKT, DKTG	SEG2	MEBODKT	
DST	SEG2	MEBODST	
Q4G	SEG2	MEBOQ4G	
Q4GG	SEG2	MEBOQ4G	
COQUE_C_PLAN	POI1		Mailles supports stub to 1 point
COQUE_D_PLAN	POI1		Mailles supports stub to 1 point
COQUE_AXIS	POI1		Mailles supports stub to 1 point
GRILLE_EXCENTRE, GRILLE_MEMBRANE, Pas			MEMBRANE of affected edge element by these modelizations.
SHB8	QUA4 TRI3 QUA8 TRI6 SEG2 SEG3	MECA_FACE4 MECA_FACE3 MECA_FACE8 MECA_FACE6 MECA_ARETE2 MECA_ARETE3	

Les distributed forces, linear, of tension, shears, the bending moments applied to structure shell edges are included in this category of loadings.

2.2.1.4 Model: `AFFE_MODELE`

the assignment of the modelization passes through operator `AFFE_MODELE` [U4.41.01].

AFFE_MODELE		Remarks
AFFE		
PHENOMENE:	"MECANIQUE"	
MODELISATION	"COQUE_3D"	
	"DKT"	
	"DST"	
	"DKTG"	
	"Q4G"	
	"Q4GG"	
	"COQUE_C_PLAN"	
	"COQUE_D_PLAN"	
	"COQUE_AXIS"	
	"GRILLE_MEMBRANE"	
	"GRILLE_EXCENTRE"	

	"MEMBRANE"	
	"SHB8"	

Note:

It is advisable to check the number of affected elements.

2.2.2 Elementary characteristics: AFFE_CARA_ELEM

Dans this part, the operands characteristic of the shell elements and shells are described. The documentation of use of operator AFFE_CARA_ELEM is [U4.42.01].

AFFE_CARA_ELEM		**COQUE_3D**	**DKT**	**DKTG**	**DST**	**Q4G**	**Q4GG**	**COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS**
SHELL		•	•	•	•	•	•	•
	EPAIS	•	•	•	•	•	•	•
	/ ANGL_REP / VECTOR	•	•	•	•	•	•	
	A_CIS	•						•
	COEF_RIGI_DRZ	•	•	•	•	•	•	•
	ECCENTRING		•		•			
	INER_ROTA		•	•	•	•	•	
	MODI_METRIQUE	•						•
	COQUE_NCOU	•	•					•

AFFE_CARA_ELEM		**GRILLE_EXCENTRE**	**GRILLE_MEMBRANE**
ROASTS		•	•
	SECTION	•	•
	/ ANGL_REP / AXE, ORIG_AXE	•	•
	ECCENTRING	•	
	COEF_RIGI_DRZ		

AFFE_CARA_ELEM		**MEMBRANE**
MEMBRANE		
	/ ANGL_REP / AXE, ORIG_AXE	•

The allocatable characteristics on the shell elements or of shell are:

- Thickness EPAIS constant on each mesh, since the mesh represents only the average layer.
- The coefficient of correction of transverse shears A_CIS for the isotropic curved shells.
- The taking into account of the correction of metric MODI_METRIQUE enters mean surface and the surfaces upper and lower (effective only for the COQUE_C_PLAN, COQUE_D_PLAN, COQUE_AXIS).
- A direction of reference allowing to define a local coordinate system in the tangent plane in any point of a shell. The construction of the local coordinate system is done is using the two

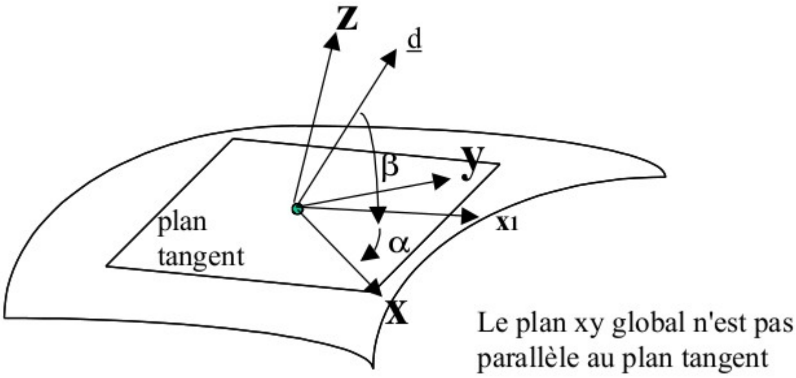
"nautical" angles α and β (provided in degrees) which define a vector v whose projection on the tangent level with the shell fixes the direction x_l . maybe, if the key word VECTOR is present, by the 3 components of the vector v One can define a single V vector for all structure, or one by area (key words GROUP_MA / NET). The construction of the local coordinate system is defined in AFFE_CARA_ELEM [U4.42.01]. One will call this local coordinate system thereafter **identifies user**.
The definition of this reference axis is useful also to lay down the directional sense of fibers of a multi-layer or orthotropic shell (Cf. operator DEFI_COMPOSITE [U4.42.03]).

- The number of layers COQUE_NCOU used for integration in the thickness of the shell, operators STAT_NON_LINE and DYNA_NON_LINE (modelizations DKT, COQUE_3D, COQUE_AXIS, COQUE_C_PLAN, COQUE_D_PLAN).

- A functionality of DEFI_GROUP makes it possible automatically to create a mesh group whose norm is understood in a given solid angle, of axis the direction of reference.

  This command can be used in preprocessing to affect nonisotropic material characteristics or in postprocessing after a computation of shell.

- The eccentring (constant for all the nodes of the mesh) ECCENTRING of each one of them compared to the mesh support. This distance is measured on the norm of the mesh support. In the excentré case inertias of rotation are obligatorily taken into account and INER_ROTA is put at OUI.
- COEF_RIGI_DRZ defines a coefficient of fictitious rigidity (necessarily small) on the degree of freedom of rotation around the norm in the shell. It is necessary to prevent that the stiffness matrix is singular, but must be selected smallest possible. The value by default (1.E-5) is appropriate for most situations (it is a relative value: rigidity around the norm is equal to KRZ time the minor term diagonal of the stiffness matrix of the element).



Appear 2.2.2-a: Total reference and tangent plane

Pour modelizations GRILLE_EXCENTRE and GRILLE_MEMBRANE,
Les given geometrical following are necessary to modelize the three-dimensions function of reinforcements:

- SECTION = S_1 : section of reinforcements in direction 1. The section is given per linear meter. It thus corresponds to the section cumulated over a width of 1 meter. If there is a section s all the 20.0cm, the cumulated section is $5.s$.
- The directional sense of reinforcements is defined either by
  - ANGL_REP, to define a vector project on the element
  - or in the case of a cylindrical shell, by ORIG_AXE, AXE to define the angle of reinforcements, constant in a cylindrical coordinate system.

- The eccentring (constant for all the nodes of the mesh) of the three-dimensions function of reinforcements compared to the mesh support (distance measured on the norm of the mesh support), (modelization GRILLE_EXCENTRE only).
- COEF_RIGI_DRZ defines a coefficient of fictitious rigidity (necessarily small) on the degree of freedom of rotation around the norm in the shell. It is necessary to prevent that the stiffness matrix is singular, but must be selected smallest possible. The value by default (1.E-5) is appropriate for most situations (it is a relative value: rigidity around the norm is equal to KRZ time the minor term diagonal of the stiffness matrix of the element).

To define a grid or if the section of reinforcements in the longitudinal meaning and the transverse one are different, it is necessary to create two layers of elements (command CREA_MAILLAGE, key word CREA_GROUP_MA), a layer of element for the longitudinal direction and a second layer of elements for the transverse direction:

For the modelization MEMBRANE, only the directional sense of the behavior of the membrane is necessary. It is defined is by:

- ANGL_REP, to define a vector project on the element
- is in the case of a cylindrical shell, by ORIG_AXE , AXE to define the angle of reinforcements, constant in a cylindrical coordinate system.

Notice important:

The meaning of the norms to each element is a recurring problem concerning the use of this kind of element, for example when loadings of type pressure are applied, either to define an eccentring or a local coordinate system.

By defect for the surface elements the directional sense is given by the cross product 12^13 for a triangle numbered 123 (DKT,…) or 1234567 (COQUE_3D) and 12^14 for a quadrangle numbered 1234 (DKQ,…) or 123456789 (COQUE_3D). For the linear shells $\boldsymbol{n}$ is given by the formula of the 2.1.1.1 2.1.1.1 with $\boldsymbol{t}$ directed in the meaning of path of the mesh to the level of the mesh.

Generally, these data are accessible while looking in the mesh file, which is not very practical for the user. Moreover, it is necessary that it checks the coherence of its mesh and to make sure that all the meshes have the same directional sense well.

The user can automatically modify the directional sense of the elements of the mesh by imposing a direction of norm, for a mesh or part of using mesh of the modelizations of shell and whatever the type of modelization. The reorientation of the elements is done by the means of operator ORIE_NORM_COQU of command MODI_MAILLAGE [U4.12.05]. The principle is the following: one defines under ORIE_NORM_COQU a direction by the means of a vector and a node pertaining to the mesh group to be reorientated. If the introduced vector is not in the plane of the mesh selected by MODI_MAILLAGE, one of automatically deducted a direction of norm obtained like the vector less given its projection as regards the mesh. All the related meshes of the group to those initially selected will then have the same directional sense of norm automatically. In addition an automatic checking of the same directional sense of the related meshes is carried out by the means of operator VERI_NORM of command AFFE_CHAR_MECA [U4.25.01].

Like the shells, the voluminal elements shells SHB have a privileged direction, called thickness. This direction of the thickness is defined by the way of netting. In general, one needs two surfaces to generate a three-dimensional volume. The direction which is perpendicular to these two surfaces is the direction of the thickness. To check the good directional sense of the sides of the indicated elements (compatibility with the privileged direction), one uses ORIE_SHB of operator MODI_MAILLAGE . If the directional sense of the element is not good, operator MODI_MAILLAGE renumbers the nodes of the element.

2.2.3 Materials: DEFI_MATERIAU

the definition of the behavior of a material is carried out using operator DEFI_MATERIAU [U4.43.01].

DEFI_MATERIAU	**COQUE_3D**	**DKT, DST, Q4G**	**DKTG, Q4GG**	**COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS**	**GRILLE_EXCENTRE, GRILLE_MEMBRANE**	**SHB8**	**MEMBRANE**
ELASTICITY. LINEAIRE							
ELAS	•	•	•	•	•	•	
ELAS_ORTH ELAS_ISTR	•	•					
ELAS_COQUE		•					
ELAS_MEMBRANE							•

DEFI_MATERIAU	**COQUE_3D**	**DKT**	**DST Q4G**	**DKTG**	**Q4GG**	**COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS**	**GRILLE_EXCENTRE GRILLE_MEMBRANE**	**SHB8**	**MEMBRANE**
PAR LAY DOWN									
behaviors available in C_PLAN	•	•				•		•	
behaviors available in 1D							•		
GLRC_DAMAGE				•	•				
GLRC_DM, KIT_DDI				•					

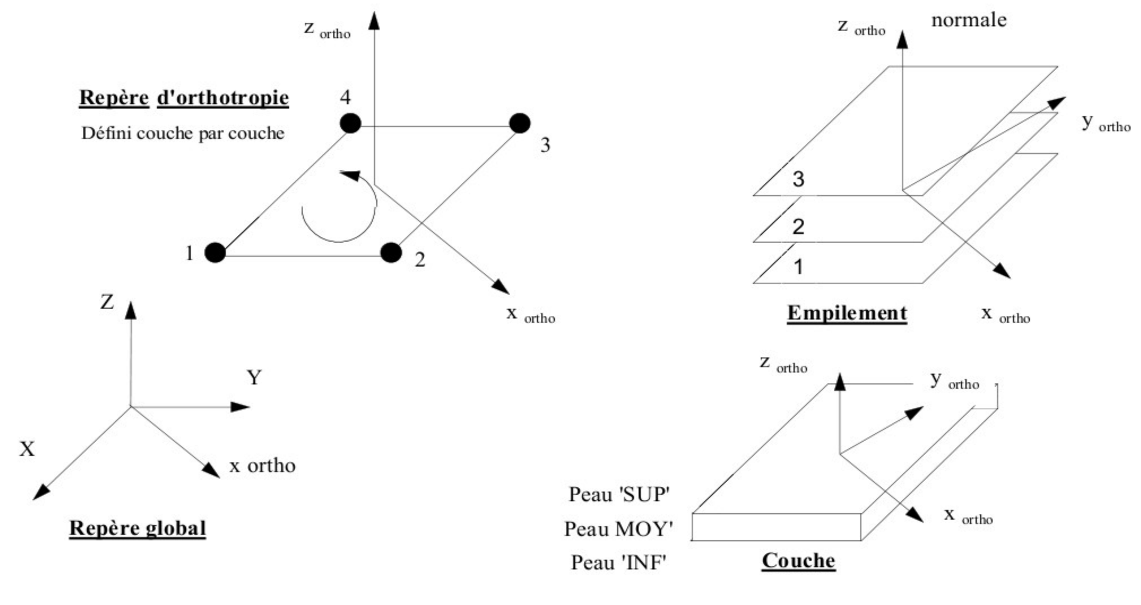
The materials used with all the elements plates or shells can have elastic behaviors in plane stresses whose linear characteristics are constant or functions of the temperature.

All the nonlinear behaviors in plane stresses (either directly, or by the method of Borst [R5.03.03]) are available for the modelizations DKT, SHB8 and shells. For more information on these nonlinearities one can refer to the paragraph [§2626].

All the nonlinear behaviors in 1D (either directly, or by the method of Borst) are available for modelizations GRILLE_EXCENTRE and GRILLE_MEMBRANE.

The composite thin structures can be treated currently only by the modelizations plates, by using DEFI_COMPOSITE with characteristics of homogenized materials. One can also directly introduce the coefficients of rigidity of the matrixes of membrane, bending and shears with ELAS_COQUE. These coefficients are given in the reference user of the element defined by ANGL_REP. It is to be noted that the terms of shears are taken into account with behavior ELAS_COQUE only for elements DST and Q4G. They are not taken into account with elements DKT.

In order to facilitate comprehension, we represented on the figure below the various references used.



Appear 2.2.3-a: References used for the definition of the material

the following example is extracted from benchmark SSLS117B and illustrates the syntax of DEFI_COMPOSITE :

```
MU2=DEFI_COMPOSITE (COUCHE=_F (EPAIS=0.2,
                              MATER=MAT1B,
                              ORIENTATION=0.0,),);
```

In this example, one defines a multi-layer composite of thickness 0.2 , the material being defines by MAT1B, and the angle of the 1st direction of orthotropy (longitudinal meaning or meaning of fibers) being null. One will refer to documentation [U4.42.03] for more details concerning the use of DEFI_COMPOSITE. (see also [R4.01.01].

2.2.4 Limiting loadings and conditions: AFFE_CHAR_MECA and AFFE_CHAR_MECA_F

the assignment of the loadings and the boundary conditions on a mechanical model is carried out using operator AFFE_CHAR_MECA, if the mechanical loadings and boundary conditions 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].

2.2.4.1 List key words factor of AFFE_CHAR_MECA

AFFE_CHAR_MECA	COQUE_3D	DKT, DKTG	DST	Q4G, Q4GG	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	GRILLE_*	SHB8
DDL_IMPO	•	•	•	•	•	•	•
FACE_IMPO	•	•	•	•	•	•	•
LIAISON_DDL	•	•	•	•	•	•	•
LIAISON_OBLIQUE	•	•	•	•	•	•	•
LIAISON_GROUP	•	•	•	•	•	•	•
CONTACT	•	•	•	•	•	•	•
LIAISON_UNIF	•	•	•	•	•	•	•
LIAISON_SOLIDE	•	•	•	•	•	•	•
LIAISON_ELEM	•	•	•	•			
LIAISON_COQUE	•	•	•	•	•	•	
FORCE_NODALE	•	•	•	•	•	•	•

DDL_IMPO	Key word factor usable to impose, with nodes or nodes groups, one or more values of displacement.
FACE_IMPO	Key word factor usable to impose, with all the nodes of a face defined by a mesh or a mesh group, one or more values of displacements (or certain associated quantities).
LIAISON_DDL	Key word factor usable to define a linear relation between degrees of freedom of two or several nodes.
LIAISON_OBLIQUE	Key word factor usable to apply, with nodes or nodes groups, the same component value of displacement definite per component in an unspecified oblique coordinate system.
LIAISON_GROUP	Key word factor usable 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.
CONTACT	Key word factor usable to notify conditions of contact and friction between two assemblies of meshes.
LIAISON_UNIF	Key word factor allowing to impose the same value (unknown) on degrees of freedom of a set of nodes.
LIAISON_SOLIDE	Key word factor allowing to modelize an indeformable part of a structure.
LIAISON_ELEM	Key word factor which makes it possible to modelize the connections of a shell part with a beam part or of a shell part with a pipe part (see 2.2.4.5 2.2.4.5).
LIAISON_COQUE	Key word factor making it possible to represent the connection enters of the shells by means of linear relations.
FORCE_NODALE	Key word factor usable to apply, with nodes or nodes groups, nodal forces, definite component by component in reference GLOBAL or an oblique coordinate system defined by 3 nautical angles.

Particular AFFE_CHAR_MECA	**COQUE_3D**	**DKT, DKTG**	**DST, Q4G**	**Q4GG**	**COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS**	**GRILLE_***	**SHB8**
FORCE_ARETE	•	•	•	•			•
FORCE_COQUE total close tangent room	 • • •	 • • •	 • • •	 • • •	 • • •		
PESANTEUR	•	•	•	•	•	•	•
PRES_REP	•	•	•	•	•		•
ROTATION	•				•		
PRE_EPSI		•	•	•		•	

FORCE_ARETE	Key word factor usable to apply linear forces to an edge of a shell element. For the linear elements the equivalent amounts applying a nodal force to the nodes supports of the element. There is thus no particular dedicated term. On the other hand, it requires edge elements.
FORCE_COQUE	Key word factor usable to apply surface forces (pressure for example) to elements defined on all the mesh or one or more meshes or of the mesh groups. These forces can be given in the total reference or a reference of reference defined on each mesh or mesh group; this reference is built around the norm with the shell element and of a fixed direction (see paragraph

	2.2.2).
PESANTEUR	Key word factor usable for a loading of type gravity.
PRES_REP	Key word factor usable to apply a pressure to one or more meshes, or of the mesh groups.
ROTATION	Key word factor usable to calculate the loading due to the rotation of structure.
PRE_EPSI	Key word factor usable to apply a loading of initial strain.

Note:

> *The forces of pressure being exerted on the shell elements can apply either by FORCE_COQUE (near) or by PRES_REP . The user will have to thus pay attention not to twice apply the loading of pressure for the elements concerned, especially whenever the modelizations of plates would be mixed with other modelizations using PRES_REP .*

In addition it should be noted that forces of pressure, that it is with FORCE_COQUE (near) or PRES_REP are such as a positive pressure acts in the contrary meaning with that of the norm to the element. By default, this norm is dependent on the meaning of path of the nodes of an element, which is not always very easy for the user.

Moreover it is necessary that the aforementioned makes sure that all these elements are directed same manner. One thus advises to impose the directional sense of these elements by the means of operator ORIE_NORM_COQU of command MODI_MAILLAGE (see paragraph [§2.2.2]).

2.2.4.2 List key words factor of **AFFE_CHAR_MECA_F**

Les key word factor generals of operator AFFE_CHAR_MECA_F are identical to those of operator AFFE_CHAR_MECA introduced above.

Particular AFFE_CHAR_MECA_F	**COQUE_3D**	**DKT, DKTG**	**DST**	**Q4G, Q4GG**	**COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS**
FORCE_ARETE	•	•	•	•	
FORCE_COQUE total close tangent room	• • •	• • •	• • •	• • •	• • •

The loadings of pressure functions of the geometry can be indicated by the means of FORCE_COQUE (near).

2.2.4.3 Application of a pressure: key word **FORCE_COQUE**

the key word factor FORCE_COQUE makes it possible to apply surface forces to elements of type shell (DKT, DST, Q4G,…) defined on all the mesh or one or more meshes or of the mesh groups. 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).

	AFFE_CHAR_MECA AFFE_CHAR_MECA_F			**Remarks**
	FORCE_COQUE:		•	

Total reference		TOUT: "OUI" NETS GROUP_MA	•	Place of application of loading
		FX FY FZ MX MY MZ	•	Provided directly for AFFE_CHAR_MECA, in the form of function for PLANE
	AFFE_CHAR_MECA_F	"MOY" "INF" "SUP" "MAIL"	•	Allows to define a load vector force on the average, lower, higher level or of the mesh (elements DKT and DST)
Local coordinate system	PRES	F1 F2 F3 MF1 MF2	•	Provided directly for AFFE_CHAR_MECA, in the form of function for AFFE_CHAR_MECA_F

Nous let us return in the paragraph corresponding to key word FORCE_COQUE of the document of use of operators AFFE_CHAR_MECA and AFFE_CHAR_MECA_F.

2.2.4.4 Limiting conditions: key words DDL_IMPO and LIAISON_*

the key word factor DDL_IMPO makes it possible to impose, with nodes introduced by one (at least) of the key words: TOUT, NODE, GROUP_NO, MESH, GROUP_MA, one or more values of displacement (or certain associated quantities). 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).

The operands available for DDL_IMPO, are listed below:

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

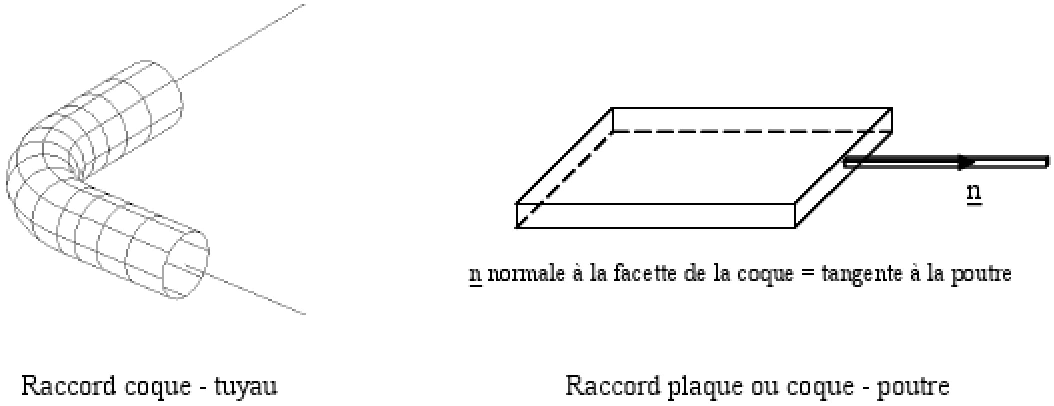
2.2.4.5 Raccords shells with other machine elements

Ces connections must meet the requirements established in [bib4] and that one finds in particular in the connection 3D-BEAM in [R3.03.03].

The connections available with the shell elements and of shells are the following:

- Connection **Beam-Shell**: it is a question of establishing connection between a node end of a beam element and a mesh group of edge of shell elements. The beam theories and of plate know only normal cuts with fiber or mean surface. The connections can take place only according to these fibers or mean surfaces. The connection beam-shell is **realizable for beams whose neutral fiber is orthogonal with the norms with the breakages of the plates or the shells**. To extend to other configurations (a beam arriving perpendicular to the plane of a plate for example) asks a feasibility study because the shell elements or of shell do not have rigidity associated with a rotation in the plane perpendicular to the norm to mean surface. The connection is usable by using key word LIAISON_ELEM: (OPTION: ` COQ_POU') of AFFE_CHAR_MECA.

- Connection **Shell-Pipe** : it is a question of establishing connection between a node end of a pipe section and a group of edge mesh of shell elements. The formulation of the connection shell-pipes is presented in the reference document [R3.08.06]. The theories of pipe and plate, know only normal cuts with fiber or mean surface. The connections can take place only according to these fibers or mean surfaces. The connection shell-pipe is realizable for pipes whose neutral fiber is orthogonal with the norms with the breakages of the plates or the shells. The connection is usable by using key word `LIAISON_ELEM: (OPTION: ` COQ_TUYAU')` of `AFFE_CHAR_MECA`.

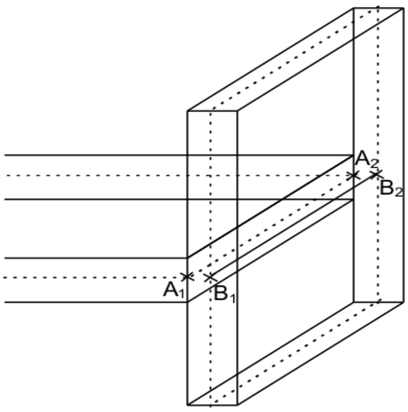


Appear 2.2.4.5 - has: Connections shells with other machine elements

- Raccord **Shell - 3D solid mass**: connection **shell-3D** solid mass is under investigation but it will be restricted initially with the cases where the norm with solid is orthogonal with the norm with the one of the breakages of the shell element or shell (see [bib4]).

- Connection **between shell elements**: to connect two shell elements between them, one uses key word `LIAISON_COQUE` of `AFFE_CHAR_MECA (_F)` (documentation [U4.44.01]). This connection is carried out by means of linear relations. The conventional approach admits that 2 planes with a grid in shells are cut according to a line which belongs to the mesh of structure. In order to prevent that the volume which is the intersection of the two shells is counted twice, one stops the mesh of a shell perpendicular to a shell given to the level of the higher or lower skin of the latter. On [Figure 2.2.4.5 - B], the link between the 2 shells is made by connections of solid body between the nodes in with respect to the segments $A_1 A_2$ and $B_1 B_2$.

- Connection **voluminal Shells `SHB` - 3D solid mass** : this connection is natural because the voluminal elements shells `SHB` have meshes support 3D too. It only should be ensured that all the nodes of the surface of connection must belong to the voluminal element shell SHB and 3D solid mass.
  Consequently, linear elements SHB of formulation (SHB6, SHB8) cannot be connected to elements 3D solid masses of quadratic formulation (PENTA15, HEXA20 for example) and conversely, the SHB15, SHB20 cannot be connected to elements 3D solid masses of linear formulation.



Appear 2.2.4.5 - B: Connection between shell elements

Des benchmark making it possible to validate these connections are available in the section examples.

2.2.4.6 Command variables

Les command variables taken into account by the various modelizations are listed here:

Command variables	COQUE_3D	DKT, DST, Q4G	DKTG	Q4GG	SHB8	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	GRILLE_MEMBRANE GRILLE_EXCENTRE MEMBRANE
TEMP	•	•	•			•	•
others: SECH, HYDR, etc.							

2.3 Linear

2.3.1 Computations resolution: Linear MECA_STATIQUE and other operators

Les linear computations are carried out in small strains. Several linear operators of resolution are available:

MECA_STATIQUE :	resolution of a problem of static mechanics linear ([U4.51.01]);
MACRO_ELAS_MULT :	computes linear static responses for various loading cases or modes of Fourier. ([U4.51.02]).
MODE_ITER_SIMULT :	computation of the values and eigenvectors by methods of under spaces. ([U4.52.03]).
MODE_ITER_INV :	computation of the values and eigenvectors by the opposite iteration method ([U4.52.04]).
MODE_ITER_CYCL :	computation of the eigen modes of a structure with cyclic symmetry ([U4.52.05]);
DYNA_LINE_TRAN :	computation of the response transient dynamics to an unspecified temporal excitation ([U4.53.02]);
DYNA_TRAN_MODAL :	computation is carried out by modal superposition or substructuring ([U4.53.21]);

2.3.2 Nonlinear computations: STAT_NON_LINE and DYNA_NON_LINE

2.3.2.1 Comportements and assumptions of strains available

Les following information are extracted from the documentation of use of operator STAT_NON_LINE : [U4.51.03].

			SHB8	COQUE_3D	DKT	DKTG	DST, Q4G	DKTG	Q4GG	GRILLE_*	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS
COMP_INCR (small strains)	RELATION	Toutes relations available in plane stresses	•	•	•						•
		Relations 3D by using the method of Borst	•	•	•						•
		All relations available in 1D								•	
	DEFORMATION: "GROT_GDEP"	Coque_3D in large displacements and large rotations available with nonlinear incrémentaux behaviors, but in small strains		•							
	DEFORMATION: "PETIT" (or GROT_GDEP)	In small or large displacements available with nonlinear incrémentaux behaviors, but in weak rotations and small strains	•	•						•	•
	DEFORMATION: PETIT or GROT_GDEP	Into small or large displacements				•					
	RELATION	GLRC_DAMAGE				•		•	•		
	RELATION	GLRC_DM, KIT_DDI				•		•			
COMP_ELAS (large displacements, large rotations)	RELATION	ELAS		•							
	DEFORMATION: "GROT_GDEP"			•							
	TYPE_CHARGE: "SUIV"	following Pressure		•							

All the mechanical nonlinear behaviors of plane stresses of the code are accessible. One distinguishes the incremental behavior models (key word factor COMP_INCR) from the nonlinear elastic behavior models (key word factor COMP_ELAS). The behavior model connects strain rates to stress rates.

For modelizations GRILLE_EXCENTRE and GRILLE_MEMBRANE, reinforced concrete structures, nonlinear behaviors 1D correspond to particular incrémentaux behaviors in STAT_NON_LINE (COMP_INCR) :

- GRILLE_ISOT_LINE for plasticity with isotropic hardening,
- GRILLE_ISOT_CINE for plasticity with kinematic hardening linear Bi,
- GRILLE_PINTO_MEN for the behavior of Pinto Menegotto.

Behaviors 3D can also be used using the method of Borst [R5.03.09].

The modelization MEMBRANE is implemented only for one elastic behavior in small strains and small displacements.

The result concept of STAT_NON_LINE/DYNA_NON_LINE contains fields of displacements, stresses and intern variables at the points of integration always computed at the points of gauss:

- DEPL: fields of displacements.
- SIEF_ELGA : Tensor of the stresses by element at the points of integration (COQUE_3D and DKT) in the reference user. For each layer, one stores in the thickness and for each thickness on the points of surface integration. Thus if one wants information on a stress for layer NC, for level NCN (NCN = -1 so the lower, NCN = 0 if medium, NCN = +1 so higher) for the surface point of integration NG, it will be necessary to look at the value given by the point defined in option POINT such as: NP = 3* (NC-1) *NPG+ (NCN+1) *NPG+NG where NPG is the total number of points of surface integration of the element of COQUE_3D (7 for the triangle and 9 for the quadrangle) and of element DKT. For modelizations GRILLE_EXCENTRE, GRILLE_MEMBRANE, one stores simply a value by point of integration: component SIXX in the direction of reinforcements. For modelizations DKTG and Q4GG, SIEF_ELGA contains the 6 forces generalized (membrane forces, bending moments, shearing forces) by point of Gauss. For modelization SHB8, SIEF_ELGA contains the 6 components of the stresses by point of integration, out of local coordinate system, plus 12 components relating to the terms of stabilization.
- VARI_ELGA : Field of intern variables (DKT and COQUE_3D) by element at the points of surface integration. For each point of surface integration, one stores information on the layers while starting with the first, level ` INF'. The number of variables represented is worth thus 2*NCOU*NBVARI where NBVARI represents the number of intern variables.

It can be enriches by the following fields, computed in postprocessing by operator CALC_CHAMP :

- EFGE_ELNO : activate the computation of the tensor of the forces generalized by element with the nodes (membrane forces, bending moments, shearing forces), in the reference user (defined in the paragraph [§2.2.2]).
- VARI_ELNO : activate the computation of the field of intern variables by element to the nodes in the thickness (by layer SUP/MOY/INF in the thickness except indication).

2.3.2.2 Detail on the points of integration

Pour modelizations DKT, COQUE_3D, COQUE_D_PLAN, COQUE_C_PLAN, COQUE_AXIS, in the case of nonlinear computations, the integration method for the shell elements and of shells is an integration method by **layers**, of which the number is defined by the user. For each layer, except modelization ROASTS, one uses a method of Simpson at three points of integration, in the middle of the layer and in skins higher and lower of layer. For N layers the number of points of integration in the thickness is of $2N+1$.

To treat non-linearities material, one advises to use from 3 to 5 layers in the thickness for a number of points of integration being worth 7.9 and 11 respectively. For tangent rigidity, one computes for each layer, in plane stresses, the contribution to the stiffness matrixes of membrane, bending and membrane-flexure coupling. These contributions are added and assembled to obtain the total tangent stiffness matrix. For each layer, one computes the state of the stresses and all the intern variables, in the middle of the layer and in skins higher and lower of layer. This information is available in VARI_ELGA and SIEF_ELGA. The plastic behavior does not understand the transverse terms of shears which are treated in an elastic way, because the transverse shears are uncoupled from the plastic behavior.

For modelizations GRILLE_EXCENTRE and GRILLE_MEMBRANE of reinforced concrete structures, there is only one point of integration per layer.

The elements shells SHB do not have concept of **layer** like the shell elements or COQUE_3D. They are integrated on all their Gauss points. The stresses are computed at the points of integration in the local coordinate system.

2.3.2.3 Nonlinear behavior geometrical

Les computations into nonlinear geometrical (large displacements and large rotations), available with modelization COQUE_3D, are carried out using operator STAT_NON_LINE, while using, under the word - key COMP_ELAS / COMP_INCR, DEFORMATION = "GROT_GDEP"

Les computations into nonlinear geometrical (large displacements and small rotations), available with modelization SHB8 , are carried out using operator STAT_NON_LINE , while using under the word - key COMP_INCR , DEFORMATION = "GROT_GDEP".

Computations into nonlinear geometrical (large displacements and small strains), available with modelization DKTG , are carried out using operator STAT_NON_LINE , while using, under the word - key COMP_INCR , DEFORMATION = "GROT_GDEP".

It is possible to apply to the elements of COQUE_3D and SHB, the following pressures. This loading has the characteristic to follow the geometry of structure during its strain (for example: the hydrostatic pressure remains always perpendicular to the deformed geometry). To take into account this kind of loading, it is necessary to specify in operator STAT_NON_LINE following information:

```
STAT_NON_LINE (
                EXCIT =_F (LOAD = close
                        TYPE_CHARGE = "SUIV")
               )
```

the geometrical nonlinear behavior of structures can have instabilities (buckling, snap - through/snap-back…). The determination and the transition of these limiting points, cannot be obtained by imposing the loading, however the options of control of the loading ` DDL_IMPO' or ` LONG_ARC' of operator STAT_NON_LINE allow to cross these critical points.

2.3.2.4 Linear buckling

Les computations in linear buckling are similar in search of eigen frequencies and of modes of vibration. The problem has to solve is expressed in the form:

To find $(\lambda, X) \in (\mathbb{R}, \mathbb{R}^N)$ such as $AX = \lambda BX$

wh ere	A	is the stiffness matrix
	B	is the geometrical stiffness matrix (calculated with option RIGI_GEOM of CALC_MATR_ELEM), available for modelizations COQUE_3D and SHB8
	λ	is the critical load
	X	is the mode of associated buckling has the critical load

Les operators MODE_ITER_INV [U4.52.04] and MODE_ITER_SIMULT [U4.52.03] are used to determine the critical load and the mode of associated buckling.

2.4 Additional computations and postprocessings

2.4.1 elementary Computations of matrixes: operator CALC_MATR_ELEM

operator CALC_MATR_ELEM (documentation [U4.61.01]) allows of compute of the elementary matrixes, which are then compilable by command ASSE_MATRICE (documentation [U4.61.22]).

The elementary options of operator CALC_MATR_ELEM are described below:

CALC_MATR_ELEM	COQUE_3D	DKT, DKTG	DST	Q4G , Q4GG	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	GRILLE_*	SHB8
` AMOR_MECA'	•	•	•	•	•	•	•
` MASS_MECA'	•	•	•	•	•	•	•
"MASS_INER"	•	•	•	•	•	•	•
` RIGI_GEOM'	•						•
` RIGI_MECA'	•	•	•	•	•	•	•
` RIGI_MECA_HYST'	•	•	•	•	•	•	•

- AMOR_MECA : Damping matrix of the elements computed by linear combination of rigidity and the mass.
- MASS_MECA : Mass matrix.
- MASS_INER: computation of inertial characteristics (mass, center of gravity)
- RIGI_GEOM : Geometrical stiffness matrix (for large displacements).
- RIGI_MECA : Stiffness matrix of the elements.
- RIGI_MECA_HYST : Hysteretic rigidity (complex) computed by the product by a complex coefficient of structural damping of simple rigidity.

2.4.2 Computations by elements: operator CALC_CHAMP

One presents hereafter the options of postprocessing for the shell elements and shells. They correspond to the results which an user can obtain after a thermomechanical computation (stresses, displacements, strains, intern variables, etc…). For structures modelized by shell elements or beams, it is particularly important to know how the results of stresses are presented in order to be able to interpret them correctly. The approach adopted in *Code_Aster* consists with compute the stresses in the reference "user" defined in operator AFFE_CARA_ELEM.
If one wishes to strip his results in another reference which the reference user, should be used command MODI_REPERE .

When a postprocessing relates to only one "subpoint", the user has key words NUME_COUCHE and NIVE_COUCHE of the key word factor REPE_COQUE .

The key words under REPE_COQUE are described in the following table:

OPTIONS	COQUE_3D	DKT	DST, Q4G	DKTG, Q4GG	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	GRILLE_*	SHB8
NUME_COUCHE	•	•			•		
NIVE_COUCHE	•	•	•		•		
PLANE		•					

- More precisely, in the case of a multi-layer material (multi-layer shell defined by DEFI_COMPOSITE), or of a structural element with local nonlinear behavior, integrated by layers, NUME_COUCHE is the whole value ranging between 1 and the number of layers, necessary to specify the layer where one wishes to carry out elementary computation. By convention, layer 1 is the sub-base (in the meaning of the norm) in the case of the shell elements.
- For the numerical layer defined by NUME_COUCHE, allows to specify the Y-coordinate where one wishes to carry out elementary computation: INF / MOY / SUP correspond to the point of integration located in internal skin / average / offsite of the layer.
- PLANE makes it possible to specify the plane of computation of the forces generalized by elements with the nodes starting from displacements (linear elasticity), option EFGE_ELNO. for a model with shell elements (DKT, DST, Q4G, DKTG) by taking account of the possible

eccentring: “MAIL” : plane of the mesh, “MOY” : average plane, “INF” : higher plane (in the meaning of the norm), “SUP” : lower plane (in the meaning of the norm).

The options of postprocessing available are:

OPTIONS	COQUE_3D	DKT	DST, Q4G	DKTG, Q4GG	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	GRILLE_*	SHB8
“ENEL_ELGA” “ENEL_ELNO” “ENEL_ELEM”		•		•			
“ENER_ELAS”		•	•	•			
“EPSI_ELGA”						•	
“SIEQ_ELGA”							•
` DEGE_ELGA'	•	•	•	•	•		
` DEGE_ELNO'	•	•	•	•	•		
` ECIN_ELEM'	•	•	•	•		•	
` EFGE_ELGA'	•	•	•	•	•		
` EFGE_ELNO'	•	•	•	•	•		
` EPOT_ELEM'	•	•	•	•		•	
` EPSI_ELNO'	•	•	•		•	•	
` SIEQ_ELNO'	•	•	•				•
` SIEF_ELGA'	•	•	•	•	•	•	•
` SIEF_ELNO'	•	•	•	•	•	•	•
` SIGM_ELNO'	•	•	•		•	•	
` VARI_ELNO'	•	•		•		•	•

- SIEF_ELGA : Computation of the forces generalized by element at the points of integration of the element starting from displacements (use only in elasticity). Identify user.
- SIGM_ELNO : Computation of the stresses by element to the nodes. Identify user.
- SIEQ_ELNO: Equivalent stresses with the nodes, calculated in a point of the thickness starting from SIGM_ELNO :
  - VMIS : Stresses of Von Mises.
  - VMIS_SG : Stresses of Von Mises signed by the trace of the stresses.
  - PRIN_1, PRIN_2, PRIN_3 : Principal stresses.

  These stresses are independent of the reference.
- EFGE_ELGA : Computation of the forces generalized by element at the points of integration of the element starting from displacements (use only in elasticity). Identify user.
- EFGE_ELNO : Computation of the forces generalized by element with the nodes starting from displacements (use only in elasticity). Identify user.
- EPSI_ELNO : Computation of the strains by element to the nodes starting from displacements, in a point of the thickness (use only in elasticity). Identify user.
- EPSI_ELGA: Computation of the strains by element at the points of integration starting from displacements, in a point of the thickness (use only in elasticity). Intrinsic reference.
- DEGE_ELGA : Computation of the strains generalized by element at the points of integration of the element starting from displacements. Identify user.
- DEGE_ELNO : Computation of the strains generalized by element with the nodes starting from displacements. Identify user.
- EPOT_ELEM : Computation of the linear elastic strain energy of strain per element starting from displacements.
- ENER_TOTALE : computation of the total strain energy integrated on element
- ENER_ELAS : computation of the elastic strain energy integrated on element

- ENEL_ELGA / ENEL_ELNO : elastic strain energy at the points of integration or nodes
- ENEL_ELEM : elastic strain energy on element
- ECIN_ELEM : Computation of kinetic energy by element.
- EFGE_ELNO : Option of activation of the computation of the tensor of the forces generalized (see paragraph [§CalculsNonlinear computations: STAT_NON_LINE and DYNA_NON_LINE]) by element with the nodes, in the reference user, by integration of stresses SIEF_ELGA (into nonlinear).
- EFGE_ELGA : Option of activation of the computation of the tensor of the forces generalized (see paragraph [§ Nonlinear computations: STAT_NON_LINE and DYNA_NON_LINE]) by element at the points of gauss of the element, in the reference user, by integration of stresses SIEF_ELGA (into nonlinear).
- VARI_ELNO : Option of activation of the computation of the field of intern variables by element and layer with the nodes. For each point of surface integration, one stores information on the layers while starting with the first, level ` INF'. The number of variables represented is worth thus 3*NCOU*NBVARI where NBVARI represents the number of intern variables.
- NUME_COUCHE : In the case of a multi-layer material (composite or shell in plasticity), integer value ranging between 1 and the number of layers, necessary to specify the layer where one wants to carry out elementary computation.
- NIVE_COUCHE : For the layer n , one can specify the Y-coordinate where one wishes to carry out elementary computation. A computation in internal skin is indicated by ` INF', in external skin by ` SUP' and on the average layer by ` MOY' (according to the meaning of the norm).
- PLANE : For option EFGE_ELNO one can specify the plane in which one wishes to have computation. This possibility is interesting in the event of eccentring of the shell elements. A computation in the plane of the mesh is indicated by ` MAIL' (defect), a computation in internal skin is indicated by ` INF', in external skin by ` SUP' and on the average layer by ` MOY'.

2.4.3 Computations with the nodes: operator CALC_CHAMP

OPTIONS	COQUE_3D	DKT	DST	Q4G	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	ROASTS	SHB8
` FORC_NODA'	•	•	•	•	•	•	•
` REAC_NODA'	•	•	•	•	•	•	•
_NOEU	•	•	•	•	•	•	•

For the shell elements and shells, operator CALC_CHAMP (documentation [U4.81.04]) allows only the computation of the forces and reactions (computation of the fields to the nodes by moyennation, option _NOEU):

- starting from the stresses, equilibrium: FORC_NODA (computation of the nodal forces starting from the stresses at the points of integration, element by element),
- then by removing the loading applied: REAC_NODA (computation of the nodal forces of reaction to the nodes, the stresses at the points of integration, element per element):
- REAC_NODA = FORC_NODA - loadings applied ,
- useful for checking of the loading and computations of resultants, moments, etc

2.4.4 Computations of quantities on whole or part of structure: operator POST_ELEM

operator POST_ELEM (documentation [U4.81.22]) allows of compute of the quantities on whole or part of structure. The computed quantities correspond to particular computation options of the affected modelization.

OPTIONS	Opérateur	COQUE_3D	DKT	DST	Q4G	COQUE_C_PLAN COQUE_D_PLAN COQUE_AXIS	ROASTS	SHB8

` MASS_INER'	POST_ELEM	•	•	•	•	•		•
` ENER_POT'	POST_ELEM	•	•	•	•	•		
` ENER_CIN'	POST_ELEM	•	•	•	•	•		
` ENER_ELAS'	POST_ELEM		•	•	•			

- `MASS_INER` : computation of the geometrical characteristics (volume, center of gravity, matrix of inertia) for the elements plates and curves.
- `ENER_POT` : computation of the potential energy of deformation due to the equilibrium starting from displacements in linear mechanics of continuums (2D and 3D) and in linear mechanics for the structural elements, or the energy dissipated thermically with the equilibrium in linear thermal starting from the temperatures (`cham_no_TEMP_R`).
- `ENER_CIN` : computation of the kinetic energy starting from a velocity field or a field of displacement and of a frequency (only for the structural elements and elements 3D).
- `ENER_ELAS` : computation of elastic strain energy.

2.4.5 Values of components of fields of variables: operator POST_RELEVE_T

operator `POST_RELEVE_T` (documentation [U4.81.21]) allows, on a nodes group, to extract from the values or to carry out computations:

- to extract from the values of components of fields of variables;
- to carry out computations of averages and invariants:
  - Averages,
  - Resultants and moments of vector fields,
  - Invariants of tensorial fields,
  - directional Trace of fields,
  - statement in references `GLOBAL`, `LOCAL`, `POLAIRE`, `USER` or `CYLINDRIQUE`

the product concept are of type counts.
To use `POST_RELEVE_T`, it is necessary to define three concepts:

- **a place** : the option `NODE` (example: `N01 N045`) or option `GROUP_NO` (example: `BEARING`);
- **an object** : with the choice, option `RESULTAT` (SD result: `EVOL_ELAS`,…) or option `CHAM_GD` (`CHAM_NO` : `DEPL`,… or `CHAM_ELEM` : `SIGM_ELNO`,…) ;
- **a nature** : with the choice, the option `` ` EXTRACTION' `` (value,…) or the option `` ` MOYENNE' `` (average, maximum, mini,…).

Notice important:

> *If one comes from an application interface with a mesh generator), the nodes are arranged by numerical order. It is necessary to reorder the nodes along the line of examination. The solution is to use operator `DEFI_GROUP` with option `NOEU_ORDO` . This option makes it possible to create a `GROUP_NO` ordered containing the nodes of a set of meshes formed by segments (`SEG2` or `SEG3`).*

An example of extraction of component is given in benchmark SSNL503 (see description in the paragraph [§2.5.3] page 38):

```
The purpose of TAB_DRZ=POST_RELEVE_T (ACTION=_F (
                          GROUP_NO = “Of,
                          INTITULE = “TB_DRZ”,
                          RESULTAT = RESUL,
                          NOM_CHAM = “DEPL”,
                          NOM_CMP = “DRZ”,
                          TOUT_ORDRE = “OUI”,
                          OPERATION = “EXTRACTION”
                                 )
                      )
```

Cette syntax is:

to extract:	`OPERATION = "EXTRACTION"`
on the nodes group D :	`GROUP_NO = "Of`
the component DRZ of displacement:	`NOM_CHAM = "DEPL", NOM_CMP = "DRZ",`
for all times of computation:	`TOUT_ORDRE = "OUI"`

2.4.6 Printing of the results: operator IMPR_RESU

operator `IMPR_RESU` allows to write the mesh and/or the results of a computation on listing with the format ` `RESULTAT'` or on a file in a displayable format by offsite tools for postprocessing in Aster: format `RESULTAT` and `ASTER` (documentation [U4.91.01]), format `CASTEM` (documentation [U7.05.11]), format `IDEAS` (documentation [U7.05.01]), format `MED` (documentation [U7.05.21]) or format `GMSH` (documentation [U7.05.32]).

This procedure makes it possible to write with the choice:

- a mesh,
- fields with the nodes (of static displacements, temperatures, eigen modes, modes,…),
- fields by elements with the nodes or the GAUSS points (of stresses, generalized forces, intern variables…).

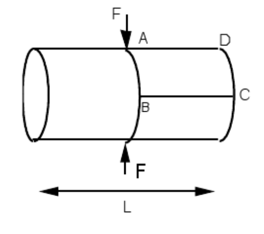
The shell elements and shell being treated same manner that the other finite elements, we return the player to the notes of use corresponding to the format of output which it wishes to use.

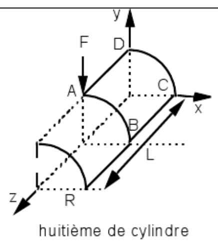
2.5 Exemples

Les benchmarks selected here is conventional benchmarks resulting from the literature and which are usually used to validate this kind of elements.

It is reminded the meeting that modelizations `DKT` correspond to the theory of Coils-Kirchhoff and modelizations `DST`, `Q4G` with the theory with transverse energy of shears (Reissner). The results for modelization `COQUE_3D` are presented only for one theory with transverse energy of shears.

2.5.1 Linear static analysis





Longueur L = 10.35 m
Rayon R = 4.953 m
Epaisseur t = 0.094 m

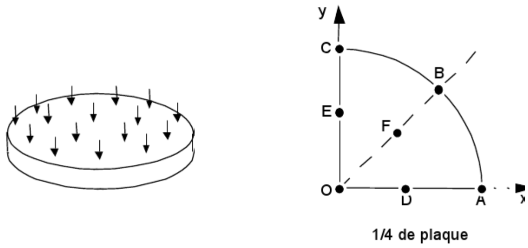
SSLS20

Titrates:: Cylindrical shell pinch on free board

Documentation V: `[V3.03.020]`

Modelizations:

```
SSLS20ADKT
SSLS20BCOQUE_3DMEC          3QU 9H
SSLS20CCOQUE_3DMEC3TR7H
```



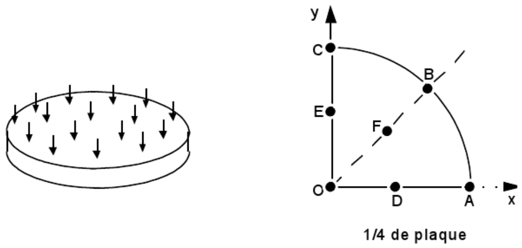
Rayon R = 1 m
Epaisseur t = 0.1 m

SSLS100

Titrates: Plate circular clamped subjected to a uniform pressure.

Documentation V: [V3.03.100]

Modelizations:
SSLS100KCOQUE_3DMEC 3QU 9H
SSLS100LCOQUE_3DMEC3TR7H
SSLS100BDKT
SSLS100EDKQ
SSLS100FDST
SSLS100GDSQ
SSLS100HQ4G
SSLS100I, J COQU_AXIS



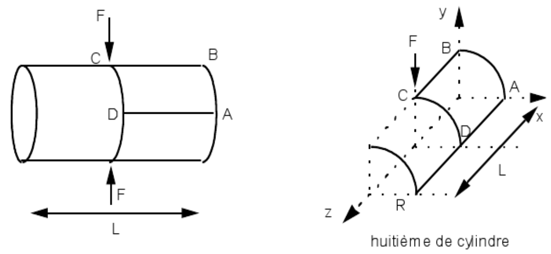
Rayon R = 1 m
Epaisseur t = 0.1 m

SSLS101

Titrates: Plate circular posed subjected to a uniform pressure.

Documentation V: [V3.03.101]

Modelizations:
SSLS101JCOQUE_3DMEC 3QU 9H
SSLS101ICOQUE_3DMEC3TR7H
SSLS101BDKT
SSLS101EDKQ
SSLS101FDST
SSLS101GDSQ
SSLS101HQ4G
SSLS101CSHB8 SHB8
SSLS101D SHB8 SHB20
SSLS101K SHB8 SHB6
SSLS101L SHB8 SHB15



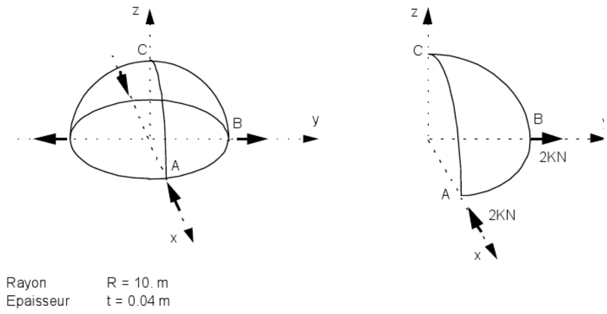
Longueur L = 600
Rayon R = 300
Epaisseur t = 3

SSLS104

Titrates: Cylindrical shell pinch with diaphragm.

Documentation V: [V3.03.104]

Modelizations:
SSLS104BCOQUE_3DMEC 3QU 9H
SSLS104CCOQUE_3DMEC3TR7H
SSLS104ADKT



SSLS105

Titrates: Doubly gripped hemisphere.

Documentation V: [V3.03.105]

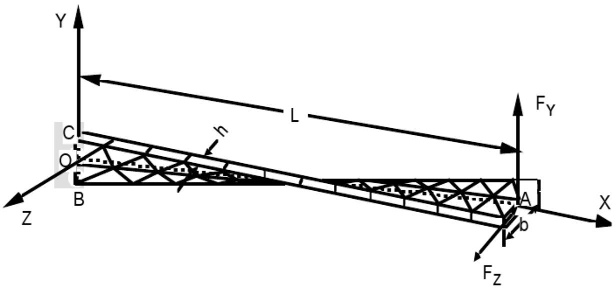
Modelizations:
SSLS105ADKT
SSLS105BCOQUE_3DMEC 3QU 9H
SSLS105CSHB8 SHB8

SSLS107

Titrates: Cylindrical panel subjected to its own weight.

Documentation V: [V3.03.107]

Modelizations:
SSLS107ACOQUE_3DMEC 3QU 9H
SSLS107BCOQUE_3DMEC3TR7H



SSLS108

Titrates: Helicoid shell under concentrated loadings.

Documentation V: [V3.03.108]

Modelizations:
SSLS108ACOQUE_3DMEC 3QU 9H
SSLS108BCOQUE_3DMEC3TR7H
SSLS108CSHB8 SHB8
SSLS108DSHB8 SHB8
SSLS108ESHB8 SHB6
SSLS108FSHB8 SHB6
SSLS108GSHB8 SHB20
SSLS108HSHB8 SHB15

Remarques:
Disadvised use with DKT/DKQ, without transverse shears.

Other benchmarks are more briefly described in the following table:

Nom	**Modélisation**	**Remarques**
hpla100a	2D_AXIS	**Titrates:** Heavy thermo-elastic hollow roll in uniform rotation.
hpla100b	COQUE_AXIS	**Documentation V:** [V7.01.100]
hpla100c	COQUE_3D	the purpose of This test is testing the second members corresponding to the effects of gravity and an acceleration due to a uniform rotation.
hpla100d	COQUE_3D	The analytical solutions for the COQUE_3D include the variation of metric in the thickness of the shell. The analytical solutions for the plates are without correction of metric
hpla100e	SHELL	
hpla100f	SHELL	
hsls01a	DKT/DST/Q4G	**Titrates:** Clamped thin plate subjected to a heat gradient in the thickness. **Documentation V:** [V7.11.001]

hsls01b	COQUE_3D	
hsns100a hsns100b	COQUE_3D/DKT COQUE_3D/DKT	**Titrates:** Plate subjected to a variation in temperature in the thickness. **Documentation V:** [V7.23.100] This benchmark makes it possible to test two ways of imposing the thermal field. The results obtained has some and B must be identical, but the reference solutions obtained are numerical.
ssll102a	COQUE_C_PLAN	**Titrates:** Clamped beam subjected to unit forces **Documentation V:** [V3.01.102]
ssls501a	COQUE_D_PLAN	**Titrates:** Roll infinitely long subjected to two lines of loads. **Documentation V:** [V3.03.501]
ssls114a ssls114b ssls114c ssls114d ssls114i ssls114j ssls114k	COQUE_3D COQUE_3D DKT/DST DKQ/DSQ COQUE_AXI COQUE_C_PLAN COQUE_D_PLAN	**Titrates:** Bets under pressure of a cylindrical quarter of binding ring. **Documentation V:** [V3.03.114] analytical Reference solution. Allows to test the term of pressure and the directional sense of the norms. One tests the results in radial displacement and radial stresses.
ssls124a Ssls124b	SHB8 SHB8	**Titrates:** Beam in bending with various slenderness. **Documentation V:** [V3.03.124] analytical Reference solution. This test makes it possible to mount the limits of the elements in term of slenderness, on the one hand, and to show their good convergence for a very irregular mesh, on the other hand.
ssls123a	SHB8	**Titrates:** Sphere under uniform offsite pressure. **Documentation V:** [V3.03.123] analytical Reference solution. This test makes it possible to evaluate the quality of the modelization of the compressive forces.

2.5.2 Modal analysis in dynamics

Nom	**Modélisation**	**Remarques**
sdls01a	DKT	**Titrates:** Thin square plate free or embedded on an edge **Documentation V:** [V2.03.001] It is of a modal computation and a harmonic computation of response. For modal computation, it is about compute the eigen

sdls01b sdls01c sdls01d sdls01e sdls01f sdls01g sdls01h	DKT DKT DKT COQUE_3D COQUE_3D COQUE_3D COQUE_3D	modes of bending of a thin square plate free or embedded on an edge. - Arêtes of the plate has directed according to the axes of the reference. B - Unspecified directional sense of the plate and harmonic response for the clamped plate. C - Modal computation by conventional and cyclic dynamic substructuring. D - Modal computation following a substructuring of Guyan. E - Arêtes of the plate directed according to the axes of the reference. F - Arêtes of the plate directed according to the axes of the reference. G - Unspecified directional sense of the plate and harmonic response for the clamped plate. H - Unspecified directional sense of the plate and harmonic response for the clamped plate. • For has and B the accuracy on the eigen frequencies is lower than 1% until the sixth mode of bending • Pour C in substructuring, the quality of the results can be improved by the use of a finer substructure mesh. • For D, it is necessary in order to obtain an accuracy of 1% on the eigen frequencies to also condense on the medium nodes of edges. • For E, F, G and H, the accuracy on the eigen frequencies is lower than 1% until the sixth mode of bending for the elements quadrangle and lower than 2% for the elements triangles. Shell element MEC3QU9H is powerful compared to the element DKT which is itself more powerful than element MEC3TR7H.
sdls109a sdls109b, C sdls109d, E sdls109f sdls109h	DKQ (MEDKQU4) and DSQ (MEDSQU4) DKT (MEDKTR3) and DST (MEDSTR3) COQUE_3D (MEC3QU9H and MEC3TR7H) COQUE_C_PLAN (METCSE3) SHB8	**Titrate:** Eigen frequencies of a thick cylindrical ring. **Documentation V:** [V2.03.109] This test is inspired by a vibratory study carried out on collector VVP of sections N4. This collector is thick and present a maximum report thickness on average radius of 0.13. This value, being able to be typical of an industrial structure, is slightly higher than the limiting value of validity usually recognized for the plates and shells. In this study, the modelization of the collector in shells is then evaluated by comparison with a voluminal model on a ring. This test makes it possible to evaluate the algorithm of asset tracking clean MODE_ITER_SIMULT [U4.52.03] with the operators of rigidity and mass.

2.5.3 Nonlinear static analysis material

SSNL501	**Titrates:** Clamped beam subjected to a uniform pressure. **Documentation V:** [V6.02.501] **Modelizations:** SSNL501ECOQUE_3DMEC 3QU 9H SSNL501DCOQUE_3DMEC3TR7H SSNL501BDKT SSNL501CDKQ SSNL501ACOQUE_C_PLAN

Of other benchmarks are more briefly described in the following table:

Nom	Modélisation	Remarques
ssnp15a ssnp15b ssnp15c ssnp15d	3D C_PLAN DKT COQUE_3D	**Titrates:** Square plate in tension-shears - Von Mises (isotropic hardening). **Documentation V:** [V6.03.015] Une plates, made up of a plastic material with linear isotropic hardening, is subjected to a tensile load and a shearing force. Even if the test validates the constitutive law rather that the elements to which it applies, it makes it possible to test the values of the stresses, the forces and the strains in the reference defined by user (ANGL_REP).
ssnv115a ssnv115b ssnv115c ssnv115d ssnv115e ssnv115f	D_PLAN DKT DKT COQUE_3D COQUE_3D COQUE_D_PLAN	**Titrates:** Corrugated iron in nonlinear behavior. **Documentation V:** [V6.04.115] This test validates the nonlinear behaviors in the modelizations of plates or thin shells. Modelization A (2D D_PLAN) is used as reference. The values of displacements are tested. Modelization COQUE_D_PLAN reveals variations on side displacements of sheet of about 13% compared to the other modelizations. This is due to the integration method in the thickness which utilizes only 5 Gauss points for this modelization, compared with 19 points for elements DKT and DKQ and 8 points for modelization D_PLAN.

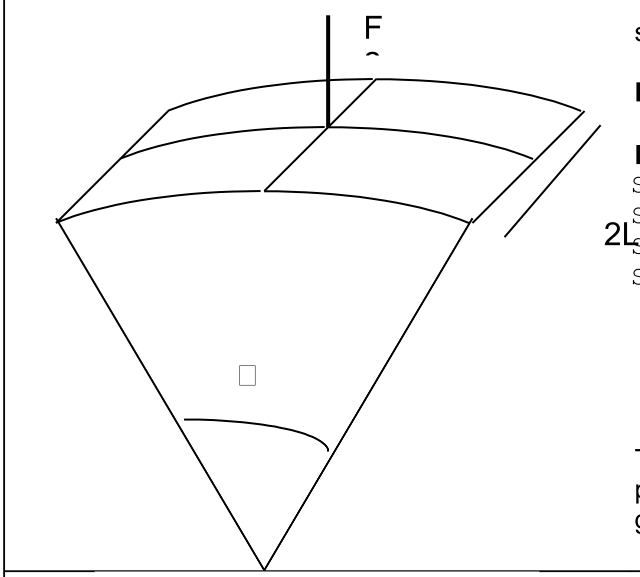
2.5.4 Nonlinear static analysis geometrical

SSNV138	**Titrates:** Plate cantilever in large rotations subjected to one moment. **Documentation V:** [V6.04.138] **Modelizations:** SSNV138 COQUE_3DMEC 3QU 9H SSNV138 COQUE_3DMEC3TR7H **Remarque:** More large rotation reached is slightly lower than . The results obtained are very satisfactory, the maximum change is lower than 0.01%. It is necessary to increase the value of COEF_RIGI_DRZ (10th-5 by defect) with 0,001 in order to be able to increase the value of the swing angle which one can reach.

SSNV139	**Titrates:** Plate skews. **Documentation V:** [V6.04.139] **Modelizations:** SSNV139 COQUE_3DMEC 3QU 9H SSNV139 COQUE_3DMEC3TR7H

SSNL502	**Titrates:** Beam in buckling. **Documentation V:** [V6.02.502] **Modelizations:** SSNL502 COQUE_3DMEC 3QU 9H SSNL502 COQUE_3DMEC3TR7H

SSNS501	**Titrates:** Large displacements of a cylindrical panel. **Documentation V:** [V6.05.501] **Modelizations:** SSNS501 COQUE_3DMEC 3QU 9H SSNS501 COQUE_3DMEC3TR7H



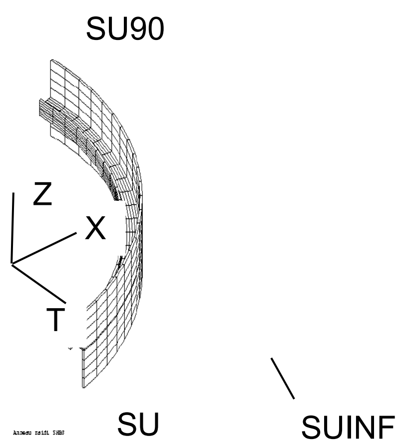
Title: Insulation breakdown of a cylindrical panel under specific force.

Documentation V: `[V6.05.101]`

Modelizations:

SSNS101a, B, C, dSHB8SHB8		
SSNS101e	SHB8	SHB20
SSNS101f	SHB8	SHB6
SSNS101g	SHB8	SHB15

This test of nonlinear quasi-static mechanics makes it possible to validate the elements SHB into nonlinear geometrical and material.



there: Buckling of a cylindrical shell with stiffener.

Documentation V: `[V6.05.102]`

Modelizations:

SSNS102a	SHB8	SHB8
SSNS102b	SHB8	SHB20

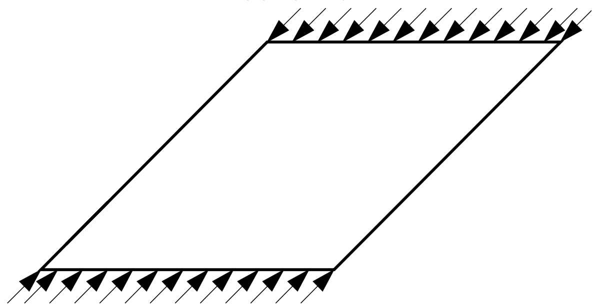
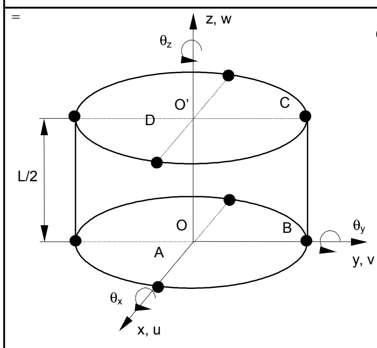
This test of nonlinear quasi-static mechanics makes it possible to validate elements SHB8 into nonlinear geometrical, with or without taking into account of the following pressures and buckling of Euler. It shows the capacitances of this element to dealing with problems of thin shells with stiffener.

Other benchmarks are more briefly described in the following table:

Nom	Modélisation	Remarques
ssnv140a	COQUE_3D	**Titrates:** Clamped cylindrical panel subjected to a surface force. **Documentation V:** `[V6.04.140]`
ssnv140b	COQUE_3D	Cette force is constant for the modelization has and following in the modelization B. The goal of this benchmark is to check geometrical `nonlinear` modelization COQUE_3D by using the algorithm of update of large rotations `3D GROT_GDEP` of `STAT_NON_LINE` and to check the processing of the following pressures. The data of this problem correspond to a thin shell H/L=0.625% what is severe for the finite element triangle `MECQTR7H` (case of blocking to the transverse shears).
ssnv141a	COQUE_3D	**Titrates:** Segment of a sphere pinch. **Documentation V:** `[V6.04.141]` Les given of this problem correspond to a thin shell H/L=0.4% what is severe for the finite element triangle `MECQTR7H` (case of blocking to the

		transverse shears). It is necessary to increase the value of the COEF_RIGI_DRZ which allots a rigidity around the norm of the shell elements which is worth by defect 10-5 smallest flexural rigidity around the directions in the plane of the shell in order to be able to increase the value of the swing angle that one can reach. Values of this coefficient up to 10-3 remain licit.
ssnv144a	COQUE_3D	**Titrates:** Bend in cross-bending, elastic, embedded on with dimensions and subjected to a linear force equivalent to one bending moment. **Documentation V:** [V6.04.144] the goal of this benchmark is to check that, for the elements COQUE_3D, solutions quasi-static into linear geometrical (VMIS_ISOT_LINE in STAT_NON_LINE) and into nonlinear geometrical (GROT_GDEP in STAT_NON_LINE) are close to the linear static solution (MECA_STATIQUE) in the field of the small disturbances.
ssnv145a ssnv145b	COQUE_3D COQUE_3D	**Titrates:** Plate cantilever in large rotations subjected has a following pressure. **Documentation V:** [V6.04.145] the goal of this benchmark is to check modelization COQUE_3D (mesh TRIA7, QUAD9) in the presence of pressure of a following type.

2.5.5 Analyzes in buckling of Euler

SSLS110 	**Titre:** Stability of a compressed square plate. **Documentation V:** [V3.03.110] **Modelizations:** SSLS110 COQUE_3DMEC 3QU 9H SSLS110 COQUE_3DMEC3TR7H
	**Titrates:** Buckling of a free cylinder under offsite pressure. **Documentation V:** [V3.03.125] **Modelizations:** SSLS125aSHB8 SHB8 SSLS125bSHB8 SHB6 SSLS125cSHB8 SHB20 SSLS125dSHB8 SHB15

SDLS504

Titrates: Side buckling of a beam (discharge).

Documentation V: [V2.03.504]

Modelizations:
SDLS504 COQUE_3DMEC 3QU 9H
SDLS504 COQUE_3DMEC3TR7H

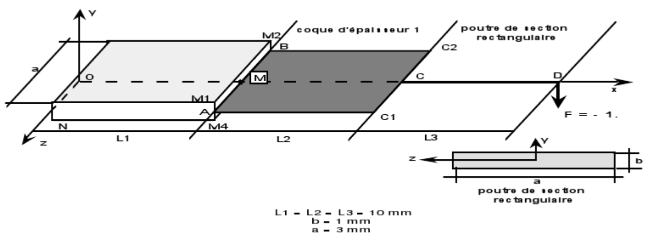
SDLS505

Titrates: Buckling of a cylindrical envelope under offsite pressure.

Documentation V: [V2.03.505]

Modelizations:
SDLS505 COQUE_3DMEC 3QU 9H
SDLS505 COQUE_3DMEC3TR7H

2.5.6 Raccords shells and other machine elements



SSLX100

Titrates: Mix 3D-Shell-Beam in bending.

Documentation V: [V3.05.100]

Modelizations:
SSLX100A3D1
MECA_HEXA20
DKT4 MEDKTR3
POU_D_E2 POU_D_E

SSLX100B3D1
MECA_HEXA20
DKT4 MEDKTR3
POU_D_E2 POU_D_E

SSLX100C3D1
MECA_HEXA20
DKT4 MEDKTR3
COQUE_C_PLAN1 MECPSE3

One tests the axial deflections, stresses, strains and bending moments in 4 points of the axis of the beam.

SSLX102

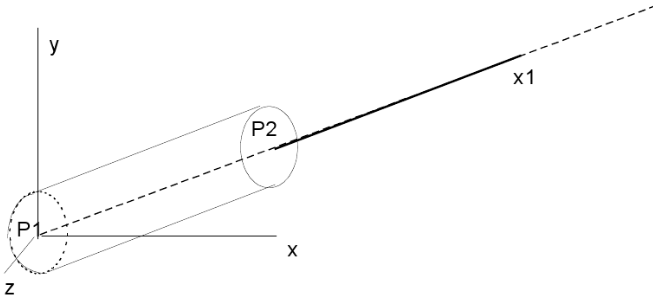
Titrates: Pipework bent in bending.

Documentation V: [V3.05.102]

Modelizations:
SSLX102ADKT and PIPE
(connection COQUE_TUYAU)

SSLX102DCOQUE and BEAM

SSLX101A



Titrate: Pipe right modelized in shells and beams [V3.05.101].

Documentation V: [V3.05.101]

Modelizations:

SSLX101ADIS_TRPOI1
DKTMEDKQU4
POU_D_E2 SEG2

Fixed support of the shell on edge P1. Bending and tension in x1. Variation from 3 to 5% on displacements and rotations in P2 with the analytical solution, due to the mesh shell with plane elements.

SSLX101BDKTMEDKQU4 , METUSEG3
TUYAUMETUSEG3 , MEDKQU4
DIS_TRPOI1

Cette modelization aims to test connection shell pipe in the presence of unit loadings: tension, bending and of torsion. The reference solution is analytical (RDM). The variation with the numerical solution is explained by the fact why the mesh in shells actually consists of plane elements (breakages). The geometry of the pipe is thus itself approximate.

SSLX102ADKTMEDKQU4 , METUSEG3
TUYAUMEDKQU4 , METUSEG3

modelization A utilizes connection coque_tuyau, the solution obtained (2.7% of variation in cross-bending, and 0.4% in bending except plane, compared to the reference: mesh any shells (modelization D) makes it possible to test the good performance of the extender coque_tuyau.

3 Thermal

Pour the resolution of chained thermomechanical problems, one must use for the thermal computation of the finite elements of thermal shell [R3.11.01]. These elements are elements plates, or linear in the case of structures of revolution or invariant structures along an axis. The curvature of structure is not taken into account in itself. The metric one of the tangent plane of each element is computed by supposing that all the tops are coplanar. These elements suppose a distribution a priori parabolic of temperature in the thickness, which results from an asymptotic development in linear thermal for one thickness of weak shell, when the temperature variations are not too important. It is it should be noted that a model based on a development of the richer field of temperature in the thickness sees its terms of a nature higher than two converging towards zero when the shell is thin. One cannot thus deal with the thermal problems of shocks with strong variation of the profile of temperature in the thickness with these shells. The methods of use of these elements are presented in [U1.22.01].

3.1 Definition of the problem

3.1.1 spatial Discretization and assignment of a modelization: operator `AFFE_MODELE`

3.1.1.1 Degrees of freedom

Les degrees of freedom are temperatures `TEMP` (temperature on the mean surface of the shell), `TEMP_INF` (temperature on the lower surface of the shell), and `TEMP_SUP` (temperature on the upper surface of the shell).

3.1.1.2 Meshes support of the Modélisation

stiffness matrixes	**Nets**	**Nature of the Élément**	**mesh finished**	**Remarques**

`plane`	`SHELL`
	`QUAD9`
	`QUAD8`
	`QUAD4`
	`TRIA7`
	`TRIA6`
		TRIA3
		plane
		planes
		planes
		planes
		planes
			`THCOQU9`
			`THCOQU8`
			`THCOQU4`
			`THCOTR7`
			`THCOTR6`
			`THCOTR3`
				nodes with 3 not x, y, z

`supposed`	`coordinates`	COQUE_PLAN SEG3 planes	`THCPSE3`	nodes with 2 not x, y
`supposed`	`coordinates`	COQUE_AXIS SEG3 planes	`THCASE3`	nodes with 2 coordinates x, y

Pour `THCOTRi`, only the three tops are exploited to define the local geometry (tangent plane, norm). For `THCOQUi`, it is considered that the element is plane and its tangent plane is defined by default by 3 of the 4 tops of the element.

3.1.1.3 Meshes support of the Modélisation

loadings	Nets	Finite element	Remarques
SHELL	SEG2	THCOSE2	with TRIA3 and QUAD4
SHELL	SEG3	THCOSE3	with TRIA6,7 and QUAD8, 9

Tous the loadings applicable to the breakages of the shell elements are treated by direct discretization on the mesh support of the element in temperature formulation. No mesh of loading is thus necessary for the sides of the shell elements.

For the applicable loadings on edges of the shell elements, a mesh support of the type SEG2 (element THCOSE2) or SEG3 (element THCOSE3) must be used.

For the imposed temperatures the meshes support are meshes reduced to a point.

3.1.1.4 Model: AFFE_MODELE

the assignment of the modelization passes through operator AFFE_MODELE [U4.41.01].

AFFE_MODELE		Remarks
AFFE		
PHENOMENE:	"THERMAL"	
MODELISATION	elementary "SHELL	
	" "COQUE_PLAN	
	" "COQUE_AXIS	

3.1.2 " Caractéristiques: AFFE_CARA_ELEM

Dans this part, the operands characteristic of the shell elements and shells in thermal are described. The documentation of use of operator AFFE_CARA_ELEM is [U4.42.01].

AFFE_CARA_ELEM	SHELL	COQUE_PLAN	COQUE_AXIS	Remarks
SHELL				
EPAIS	•	•	•	

The characteristics assigned to the materials are the same ones as for a mechanical computation. It is it should be noted that it is not useful to define a particular reference for the analysis of the results of thermal computation because those are limited to the fields of temperature, scalar quantity, independent of the reference frame used.

3.1.3 Materials: DEFI_MATERIAU

DEFI_MATERIAU	SHELL	COQUE_PLAN	COQUE_AXIS	Remarks
THER	•	•	•	
THER_FO	•	•	•	

The materials used with elements plates or shells in thermal can have linear characteristics thermal constant or dependent on the increment of loading.

3.1.4 Limiting loadings and conditions: AFFE_CHAR_THER and AFFE_CHAR_THER_F

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

The documentation of use of AFFE_CHAR_THER and AFFE_CHAR_THER_F is [U4.44.02].

3.1.4.1 List key words factor of AFFE_CHAR_THER

Les affected values of the loadings are real and do not depend on any parameter.

General AFFE_CHAR_THER	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
TEMP_IMPO	•	•	•	

Particular AFFE_CHAR_THER	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
FLUX_REP	•	•	•	on the sides and edges of surface elements
ECHANGE	•	•	•	on the sides and edges of surface elements

- TEMP_IMPO : Key word factor usable to impose, on nodes or nodes groups, a temperature.
- FLUX_REP : Key word factor usable to apply normal flows to a face of thermal shell defined by one or more meshes or the mesh groups of type triangle or quadrangle.
- ECHANGE : Key word factor usable to apply conditions of exchange with an outside air temperature with a face of shell, defined by one or more meshes or mesh groups of type triangle or quadrangle.

3.1.4.2 List key words factor of AFFE_CHAR_THER_F

Les affected values of the loadings can be a function of the total coordinates and time, or temperature in nonlinear thermal (except in shells).

General AFFE_CHAR_THER_F	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
TEMP_IMPO	•	•	•	

Particular AFFE_CHAR_THER_F	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
FLUX_REP	•	•	•	on the sides and edges of surface elements
ECHANGE	•	•	•	on the sides and edges of the surface elements

3.2 Résolution

3.2.1 Transient computations: transitory operator

THER_LINEAIRE Computation option	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
CHAR_THER_EVOL	•			

It is here the processing of the problems of evolutionary thermal.

3.3 Additional computations and Calculs

3.3.1 postprocessings in postprocessing

One presents hereafter the options of postprocessing for the shell elements and elementary

shells OPTIONS	SHELL	COQUE_PLAN	COQUE_AXIS	Remarques
"FLUX_ELNO'	•			
"FLUX_ELGA'	•			

- FLUX_ELNO : This option carries out the heat flux computation to the nodes starting from the temperature.
- FLUX_ELGA : This option carries out the heat flux computation at the points of integration starting from the temperature.

3.4 Examples

One gives here the list of the benchmarks available for the thermal shells. They are steady benchmarks of thermal. The results are correct for the set of these benchmarks, whatever the element used.

Nom	Modélisation	Element	Remarques
tplp301a	SHELL	THCOTR3	**Titrates:** Plate with imposed temperature distributed sinusoïdalement on a side. **Documentation:** [V4.05.301]
tplp302a	SHELL	THCOTR6	**Titrates:** Rectangular plate with temperature imposed on the sides. **Documentation:** [V4.05.302]
tpls100a tpls100b	SHELL COQUE_PLAN	THCOTR6/THCOTR3 THCPSE3	**Titrates:** Infinite plate subjected to a steady skew-symmetric heat flux couple on its two half-sides. **Documentation:** [V4.03.100] conduction is linear, homogeneous and isotropic.
tpls101a tpls101b tpls101c tpls101d tpls101e	SHELL	THCOTR6/THCOSE3 THCOQU4/THCOSE2 THCOQU8/THCOSE3 THCOQU9/THCOSE3 THCOTR7/THCOSE3	**Titrates:** Infinite plate subjected to a couple of thermal conditions with outside, symmetric compared to the average layer. **Documentation:** [V4.03.101]

			conduction is linear, homogeneous and isotropic.
tpls302a tpls302b tpls302c tpls302d	SHELL	THCOQU8/THCOSE3 THCOQU4/THCOSE2 THCOQU9/THCOSE3 THCOTR7/THCOSE3	**Titrates:** Rectangular plate with convection and imposed temperature **Documentation:** [V4.03.302]

4 thermomechanical Sequence

4.1 Formalisme

Pour the resolution of chained thermomechanical problems, one must use for the thermal computation of the finite elements of thermal shell [R3.11.01] whose field of temperature is recovered like input datum of *Code _Aster* for mechanical computation. It is necessary thus that there is compatibility between the thermal field given by the thermal shells and that recovered by the mechanical shells. This last is defined by the knowledge of 3 fields `TEMP_SUP`, `TEMP` and `TEMP_INF` given in skins lower, medium and higher of shell. The table below indicates these compatibilities:

THERMAL modelization	**Nets**	**Element**	**Nets**	**Element**	Modelization MECANIQUE
SHELL	QUAD9	THCOQU9	QUAD9	MEC3QU9H	COQUE_3D
SHELL	QUAD8	THCOQU8			
SHELL	QUAD4	THCOQU4	QUAD4	MEDKQU4 MEDSQU4 MEQ4QU4	DKT DST Q4G
SHELL	TRIA7	THCOTR7	TRIA7	MEC3TR7H	COQUE_3D
SHELL	TRIA6	THCOTR6			
SHELL	TRIA3	THCOTR3	TRIA3	MEDKTR3 MEDSTR3	DKT DST
COQUE_PLAN	SEG3	THCPSE3	SEG3 SEG3	METDSE3 METCSE3	COQUE_D_PLAN COQUE_C_PLAN
COQUE_AXIS	SEG3	THCASE3	SEG3	MECXSE3	COQUE_AXIS

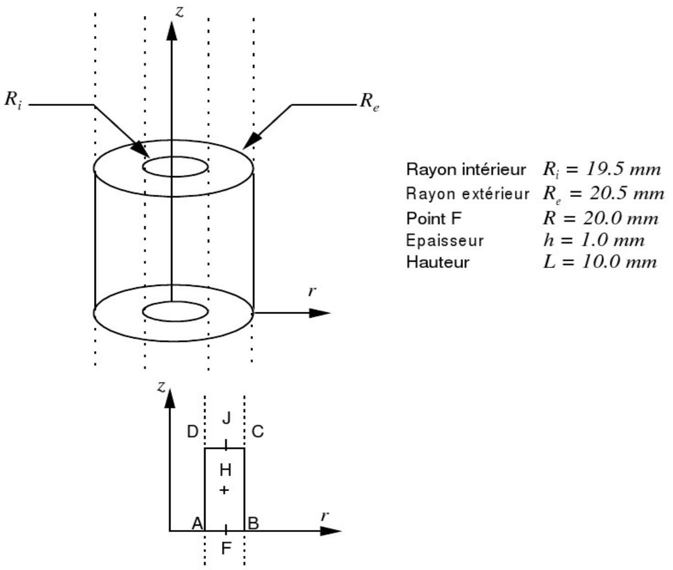
Note:

- *The nodes of the thermal shell elements and plates or mechanical shells must correspond. The meshes for the thermal and the mechanics will thus have the same number and the same type of meshes.*
- *The surface thermal shell elements are treated like plane elements by projection of the initial geometry on the level defined by the first 3 tops. For the sequence of computations with mechanical curved elements it is thus necessary that the geometry of the plate is not too distant from that of the shell. When the structure is curved, that thus requires for thermal computation to net it in a sufficiently fine way in order to have correct results in preparation for the mechanical part. Only the linear elements of thermal are perfectly associated with the corresponding linear elements in mechanics because fascinating of account the curvature of structure with a grid.*
- *The sequence with multi-layer materials is not available for time.*
- *The thermomechanical sequence is also possible if one knows, analytically or by experimental measurements, the variation of the field of temperature in the thickness of structure or certain parts of structure. In this case one works with a card of temperature defined a priori; the field of temperature is not given any more by three values `TEMP_INF` , `TEMP` and `TEMP_SUP` of thermal computation obtained by `EVOL_THER` . Operator `DEFI_NAPPE` allows to create such profiles of temperatures starting from the abundant data by the user. These profiles are affected by command `CREA_CHAMP` and `CREA_RESU` (cf the hsns100b `benchmark`). It will be noted that it is not necessary for mechanical computation that the number of points of integration in the thickness is*

equal to the number of points of discretization of the field of temperature in the thickness. The field of temperature is automatically interpolated at the points of integration in the thickness of the shell elements or of shells.

- *Since the introduction of the temperature as a command variable, in version 9, the thermomechanical sequence is carried out by adding in* `CREA_RESU` *key word* `PREP_VRC1/ PREP_VRC2` *:* The thermal evolution which one can associate with the material field by `AFFE_MATERIAU/AFFE_VARC` must be ready to be used by the finite elements of the mechanical model. A problem arises for the elements of type shell or pipe which use a temperature varying in the thickness on the various layers. For these elements, it is necessary to prepare the computation of the temperature on the layers upstream of command `AFFE_MATERIAU`. For that, the user must use command `CREA_RESU` with one of operations `PREP_VRC1` or `PREP_VRC2` of the Command variables"):
  - `OPERATION='PREP_VRC1"` : computation of the temperature in the layers of a shell on the basis of a temperature `TEMP= F (EPAIS, INST)`
  - `OPERATION='PREP_VRC2`": computation of the temperature in the layers of a shell on the basis of a temperature calculated by aster with a model of shells `(TEMP/TEMP_INF/TEMP_SUP)`.

HPLA100



It acts to study a thermal phenomenon of dilation where the fields of temperature are computed with `THER_LINEAIRE` by a steady computation:

- thermal dilation:

$$T(\rho)-T_{ref}(\rho)=0.5(T_s+T_i)+2.(T_s+Ti)(r-R)/h$$

with: $T_s=0.5\,°C\,, T_i=-0.5\,°C\,, T_{ref}=0\,°C$

$T_s=0.1\,°C\,, T_i=0.1\,°C\,, T_{ref}=0\,°C$

One tests the stresses, the forces and bending moments in L and Mr. Les results of reference is analytical. For modelizations `COQUE_3D` one takes into account the variation of metric with the thickness of the shell. Very good performances whatever the type of element considered.

Titrate: Heavy thermo-elastic hollow roll in uniform rotation

Documentation: `[V7.01.100]`

Modelizations:

```
HPLA100A
ThermiquePLAN32                THPLQU8
MécaniqueAXIS32                MEAXQU8

HPLA100B
          ThermiqueCOQUE_PLAN10
THCPSE3
          MécaniqueCOQUE_AXIS10
MECXSE3

HPLA100C
ThermiqueCOQUE32               THCOQU9
MécaniqueCOQUE_3D32
MEC3QU9H

HPLA100D
ThermiqueCOQUE64               THCOTR7
MécaniqueCOQUE_3D64
MEC3TR7H

HPLA100E
ThermiqueCOQUE200
THCOQU4
MécaniqueCOQUE200
MEDKQU4

HPLA100F
ThermiqueCOQUE400
THCOTR3
MécaniqueCOQUE400
MEDKTR3
```

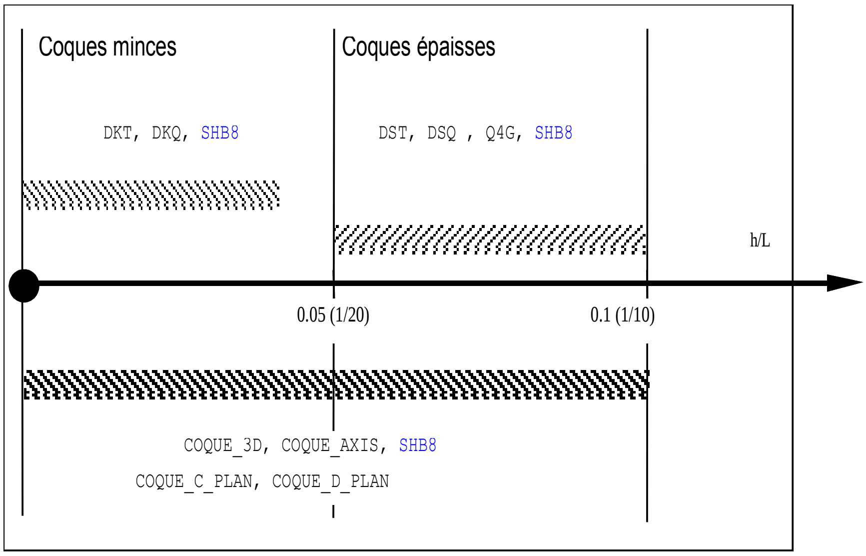
5 Conclusion and advices of Dans

use the following table, a summary of the possibilities offered by the modelizations plates and shells is described.

Modelization	**DKT**	**DST, Q4G**	**DKTG, Q4GG**	**COQUE_3 D**	**COQUE_AXIS, COQUE_D_PLAN COQUE_C_PLAN**	**SHB8**	**GRILLE_***
Domaine of Static							
application linear: Isotropic material	**X**	**X**	**X**	**X**	**X**	**X**	**X**
Material orthotropic, composite	**X**	**X**					
Static nonlinear nonlinear	**materi**		**X**	**X**	**X**	**X**	**X**

	al						
X Static geometrical				**X**		**X**	
Dynamic analysis	**X**	**X**	**X**	**X**	**X**	**X**	**X**
Flambement d' Euler				**X**		**X**	**X**

Sur the figure below the field of application of the plates and the shells is schematized.



Appear 5-a: Fields of application of the plates, the shells and shells SHB

Quelques recommendations concerning the field of application of these elements:

- **Thin structures** : for these structures, of which the report H/L is lower than 1/20, the effects of transverse shears can be neglected and the theory of Kirchhoff applies. One advises to use for this kind of structure of the elements plates `DKT-DKQ` or the shell elements curve (`COQUE_3D`, `_AXIS`, `_C_PLAN`, `_D_PLAN`) or of the voluminal elements shells `SHB8`, `SHB15`, `SHB20`. **It is advised to use preferably elements `DKT` and `DKQ`** which give very good results on displacements and more approximate on the stresses (to be recommended for the vibratory analyses). Even if one must use a large number of these elements, the execution times remain reasonable compared with those of the curved elements. Element `SHB6` is less powerful than other elements `SHB` for thin structures. Nevertheless, element `SHB6` is essential to net unspecified geometries. A mixed modelization `SHB6` and `SHB8` gives reasonable results in accuracy and CPU times. **It is thus advised to net structure to the maximum with elements `SHB8`, and to net the remaining part with elements `SHB6`. It is disadvised using element `SHB6` to net the totality of structure**.

- **Thick structures:** for these structures, one will use shell elements `DST`, `DSQ` and `Q4G` which take account of the transverse shears with a factor of correction of shears $k=5/6$ (theory of Reissner) or **preferably of the shell elements curve**. The factor of correction of the shears makes it possible to pass from a theory of Hencky-Mindlin-Naghdi for $k=1$, with a theory of Reissner for $k=5/6$. The shear coefficient is adjustable only for the shell elements curve but it is advised not to modify its value by default.
  When modelization `Q4G` was privileged, it is necessary to make a small study of sensitivity to the mesh. The tests show indeed that this modelization requires a sufficiently fine mesh in the directions requested in bending to obtain weak errors.
  As for thin structures, the recommendations of use for elements `SHB` for thick structures are the same ones. Elements `SHB8`, `SHB15` and `SHB20` are powerful. It is thus advised to use element `SHB6` to net the complex geometries which one cannot modelize by only element `SHB8`.

Elements DKT, DKQ, DST, DSQ and Q4G are plane elements, they do not take into account the curvature of structures, it is thus necessary to refine the mesh if the curvature is important if one wants to avoid parasitic bendings.

The variation of metric of the geometry (i.e. its radius of curvature) according to its thickness is taken into account:

- automatically for modelization COQUE_3D
- defined by the user for modelizations COQUE_AXIS, COQUE_D_PLAN and COQUE_C_PLAN.

The optimal machine element in statics according to all the benchmarks of the paragraph [§2.5] is the shell element with 9 nodes MEC3QU9H, which makes it possible to obtain good displacements and good stresses thanks to its interpolation P2 out of membrane. It is a general-purpose element which can be at the same time used to represent structures very mean ($h/L\leq 1/100$) or thicker. Like, in addition, **the shell element with 7 nodes MEC3TR7H is less powerful**, it **is advised with the user to net his structure in shells with the greatest potential number of quadrangles**.

- **Non-linearity material:** the nonlinear behaviors (plasticity, etc) in plane stresses are available for the shell elements curve (COQUE_3D, COQUE_AXIS, COQUE_C_PLAN, COQUE_D_PLAN) and the elements plates DKT-DKQ only. The plastic behavior does not take the terms of transverse shears which are treated in an elastic way, because the transverse shears are uncoupled from the plastic behavior. For a good representation of the progression of plasticity through the thickness, one advises to use for numerical integration 3 to 5 layers in the thickness for a number of points of gauss being worth 3.5 and 11 respectively. All the nonlinear behaviors are available for voluminal shell elements SHB (SHB6, SHB8, SHB15 and SHB20).
- **Geometrical non-linearity :** nonthe geometrical lineairities (large displacements, large rotations) in plane stresses are available for the shell elements curve COQUE_3D only. The nonlinear behaviors (large displacements, small rotations) are available for voluminal shell elements SHB.
- **Buckling of Euler :** this kind of analysis is available with the shell elements voluminal curve COQUE_3D and shells SHB.

Elements corresponding to the machine elements exist in thermal; the thermomechanical couplings are thus available at the same time for the shell elements and of shells. For time these couplings are not possible for multi-layer materials.

6 Bibliography

1 J.L. Batoz, G. Dhatt "Modelization of structures by finite elements: beams and plates" Hermes, Paris (1990)

2 J.L. Batoz, G. Dhatt "Modelization of structures by finite elements: shells" Hermès, Paris (1992)

3 D. Bui "Evolution of AFFE_CARA_ELEM " CR MMN/97/004.

4 S. Andrieux "Raccords 3D/beam, 3D/shells and other imaginations". Note to appear.

5 E. Lorentz "plastic Large deformation. Modelization in Aster by PETIT_REAC" .EDF/DER CRMMN 1536/07.

6 P. Nonlinear Kinematical Jettor "of the shells". Report SAMTECH resulting from the PP contract/GC - 134/96.

7 J. Argyris, P. Dunne, C. Malejannakis, E. simple Schelkie "A triangular facet Shell element with application to linear and not linear equilibrium and elastic stability problem". Comp. Meth. Appl. Mech. Eng., flight 11.1977.

8 F. Frey "nonlinear static analysis of structures by the finite element method and its application to the steel construction". Doctorate, Liege, 1978.

9 A.B. Sabir and A.C. Lock "The application of finite elements to the broad deflection geometrically and not linear behavior of cylindrical shells" Variational methods in Engineering, edited by Brebbia and Tottenham, Southampton, 1972.

10 G.S. Dhatt "Instability of thin shells by the finite elements method". Proc. IASS Symp. , Vol1, Vienna 1970, pp1-36.

11 Connection 3D-Beam [R3.03.03].

12 Following pressure for the voluminal shell elements [R3.03.07].

13 Axisymmetric thermo-elastic shells and 1D [R3.07.02].

14 Shell elements DKT, DST, DKQ, DSQ and Q4 [R3.07.03].

15 Voluminal finite elements of shells [R3.07.04].

16 Voluminal shell elements into nonlinear geometrical [R3.07.05].

17 Model of thermal for the thin shells [R3.07.11].

18 Finite elements of right and curved pipe with ovalization, swelling and warping in elastoplasticity [R3.08.06].

19 Model of thermal for the thin shells [R3.11.01].

20 Integration of the elastoplastic relations [R5.03.02].

21 Nonlinear elastic behavior model [R5.03.20].

22 Static and dynamic modelization of the beams in large rotations [R5.03.40].

23 Operator `DEFI_MATERIAU` [U4.23.01].

24 Operator `DEFI_COMPOSITE` [U4.23.03].

25 Operator `AFFE_CARA_ELEM` [U4.24.01].

26 Operator `AFFE_CHAR_MECA` and `AFFE_CHAR_MECA_F` [U4.25.01].

27 Operator `AFFE_CHAR_THER` and `AFFE_CHAR_THER_F` [U4.25.02].

28 Operator `STAT_NON_LINE` [U4.32.01].

29 Operator `CALC_MATR_ELEM` [U4.41.01].

30 Operator `CALC_CHAMP` [U4.81.04].

31 LEGAY A. and COMBESCURE A.: "Elasto-plastic stability analysis of shells using the physically stabilized finite element SHB8PS", International Journal for Numerical Methods and Engineering, Vol. 57,1299-1322, 2003.

32 ABED-MERAIM F. and COMBESCURE A.: "SHB8PS has new adaptive assumed strain continuum mechanics Shell element for impact analysis", Computers and Structures, Vol. 80,791-803, 2002.

33 BELYTSCHKO T. and BINDEMAN L.P.: "Assumed strain stabilization of the eight node hexahedral elements", Computer Methods in Applied Mechanics and Engineering, Vol. 105,225-260, 1993.

34 FLANAGAN D.P. and BELYTSCHKO T.: "A uniform strain hexahedron and equilateral with orthogonal hourglass control", International Journal for Numerical Methods and Engineering, Vol. 17,679-706, 1981.

35 RIKS E.: "Incremental Year approach to the solution of snapping and buckling problems", International Journal of Solids and Structures, Vol. 15,524-551, 1979.