Opérateur CALC_CHAMP

1 Drank

Créer or to supplement a result by computing fields by element or with the nodes (forced, strains, …).

The produced result concept either is created, or modified, i.e. the call to CALC_CHAMP is done in the following way:

```
resu = CALC_CHAMP    ( RESULTAT = resu…,  reuse = resu,…)
```

or

```
resu1 = CALC_CHAMP  (  RESULTAT = resu,…)
```

Table des Matières

2 Syntaxe

```
resu    [*] = CALC_CHAMP

          (  ◊reuse  = resu,
             ◊MODELE    =mo             ,                      [model]
             ◊CHAM_MATER   =chmater        ,                  [cham_mater]
             ◊CARA_ELEM    =carac          ,                     [cara_elem]
             ◊EXCIT  = _F   (     ♦CHARGE    = l_charge,  [l_char_meca]
                                     ◊   / COEF_MULT = cm,    [R]
                                         / FONC_MULT = Fm,    [function]
                                     ◊TYPE_CHARGE  = "FIXE',
                                  )
             ◊#Sélection       of the meshes concerned with computation
                /TOUT   =     'OUI",                          [DEFECT]
                /   |  GROUP_MA =l_grma    ,                  [l_gr_maille]
                    |  NET   =l_mail    ,                [l_maille]

             ◊#Sélection       of the sequence numbers
                /TOUT_ORDRE   =     'OUI',
                /NUME_ORDRE   =l_nuor         ,                [l_I]
                /LIST_ORDRE   =l_nuor         ,                [listis]
                /NUME_MODE     =l_numo        ,                [l_I]
                /NOEUD_CMP     =l_nomo        ,                [l_K16]
                /NOM_CAS      =nocas          ,                [K16]
                /INST   =l_inst         ,                      [l_R]
                /FREQ   =l_freq         ,                      [l_R]
                /LIST_INST    =l_inst     ,                    [listr8]
                /LIST_FREQ    =l_freq     ,                    [listr8]

             ◊  |  CRITERE =    / "RELATIF",                   [DEFECT]
                                / "ABSOLU",
                |  PRECISION =  / prec,
                                / 1.0E-6,                      [DEFAUT]
```

#options for linear mechanical results

```
             ♦RESULTAT  =resu ,
```

#options of computation of the stresses (elements of continuous medium 2D and 3D)

```
             ◊CONTRAINTE   =        |  'EFGE_ELGA'
                                    |  'EFGE_ELNO'
                                    |  'EFGE_NOEU'
                                    |  'SIEF_ELGA'
                                    |  'SIEF_ELNO'
                                    |  'SIEF_NOEU'
                                    |  'SIGM_ELGA'
                                    |  'SIGM_ELNO'
                                    |  'SIGM_NOEU'
                                    |  'SIPM_ELNO '
                                    |  'SIPO_ELNO'
                                    |  'SIPO_NOEU'
                                    |  'SIRO_ELEM'
```

#options of computation of the strains

```
◊DEFORMATION  =        |  'DEGE_ELGA'
                       |  'DEGE_ELNO'
                       |  'DEGE_NOEU'
                       |  'EPME_ELGA'
                       |  'EPME_ELNO'
                       |  'EPME_NOEU'
                       |  'EPSG_ELGA'
                       |  'EPSG_ELNO'
                       |  'EPSG_NOEU'
                       |  'EPSI_ELGA'
                       |  'EPSI_ELNO'
                       |  'EPSI_NOEU'
                       |  'EPVC_ELGA'
                       |  'EPVC_ELNO'
                       |  'EPVC_NOEU'
```

#options of computation of energies

```
◊ENERGIE   =           |  'DISS_ELEM'
                       |  'DISS_ELGA'
                       |  'DISS_ELNO'
                       |  'DISS_NOEU'
                       |  'ECIN_ELEM'
                       |  'ENEL_ELEM'
                       |  'ENEL_ELGA'
                       |  'ENEL_ELNO'
                       |  'ENEL_NOEU'
                       |  'EPOT_ELEM'
                       |  'ETOT_ELEM'
                       |  'ETOT_ELGA'
                       |  'ETOT_ELNO'
                       |  'ETOT_NOEU'
```

#options of computation of ◊CRITERES

```
criteria   =       |  'EPEQ_ELGA'
                   |  'EPEQ_ELNO'
                   |  'EPEQ_NOEU'
                   |  'EPMQ_ELGA'
                   |  'EPMQ_ELNO'
                   |  'EPMQ_NOEU'
                   |  'SIEQ_ELGA'
                   |  'SIEQ_ELNO'
                   |  'SIEQ_NOEU'
```

#options of interpolation and extraction of the intern variables

```
◊VARI_INTERNE =    |  'VARC_ELGA'
```

#options for the nonlinear results (produced by STAT_NON_LINE or DYNA_NON_LINE):

♦RESULTAT =resu , / [evol_noli]

#options of computation of the stresses (elements of continuous medium 2D and 3D)

◊CONTRAINTE = | 'EFGE_ELGA'
| 'EFGE_ELNO'
| 'EFGE_NOEU'
| 'SIEF_ELNO'
| 'SIEF_NOEU'
| "SIGM_ELGA"
| 'SIGM_ELNO'
| 'SIGM_NOEU'
| 'SIPO_ELNO'
| 'SIPO_NOEU'
| 'SIRO_ELEM'

#options of computation of the strains

◊DEFORMATION = | 'DEGE_ELGA'
| 'DEGE_ELNO'
| 'DEGE_NOEU'
| 'EPFD_ELGA'
| 'EPFD_ELNO'
| 'EPFD_NOEU'
| 'EPFP_ELGA'
| 'EPFP_ELNO'
| 'EPFP_NOEU'
| 'EPME_ELGA'
| 'EPME_ELNO'
| 'EPME_NOEU'
| 'EPMG_ELGA'
| 'EPMG_ELNO'
| 'EPMG_NOEU'
| 'EPSG_ELGA'
| 'EPSG_ELNO'
| 'EPSG_NOEU'
| 'EPSI_ELGA'
| 'EPSI_ELNO'
| 'EPSI_NOEU'
| 'EPSP_ELGA'
| 'EPSP_ELNO'
| 'EPSP_NOEU'
| 'EPVC_ELGA'
| 'EPVC_ELNO'
| 'EPVC_NOEU'

#options of computation of energies

◊ENERGIE = | 'DISS_ELEM'
| 'DISS_ELGA'
| 'DISS_ELNO'
| 'DISS_NOEU'
| 'ENEL_ELEM'
| 'ENEL_ELGA'

```
                              |   'ENEL_ELNO'
                              |   'ENEL_NOEU'
                              |   'ETOT_ELEM'
                              |   'ETOT_ELGA'
                              |   'ETOT_ELNO'
                              |   'ETOT_NOEU'
```

#options of computation of ◊CRITERES

```
criteria   =          |   'DERA_ELGA'
                      |   'DERA_ELNO'
                      |   'DERA_NOEU'
                      |   'ENDO_ELGA'
                      |   'ENDO_ELNO'
                      |   'ENDO_NOEU'
                      |   'EPEQ_ELGA'
                      |   'EPEQ_ELNO'
                      |   'EPEQ_NOEU'
                      |   'EPMQ_ELGA'
                      |   'EPMQ_ELNO'
                      |   'EPMQ_NOEU'
                      |   'INDL_ELGA'
                      |   'PDIL_ELGA'
                      |   'SIEQ_ELGA'
                      |   'SIEQ_ELNO'
                      |   'SIEQ_NOEU'
```

#options of interpolation and extraction of the intern variables

```
◊VARI_INTERNE =       |   'VAEX_ELGA'
                          ♦NOM_VARI  = (cf  [#2.4.3.])
                      |   'VAEX_ELNO'
                          ♦NOM_VARI  = (cf  [#2.4.3.])
                      |   'VAEX_NOEU'
                          ♦NOM_VARI  = (cf  [ #2.4.3. ])

                      |   'VARC_ELGA'
                      |   'VARI_ELNO'
                      |   'VARI_NOEU'
```

#options of computation of hydraulic flows (elements THM)

```
◊HYDRAULIQUE  =       |   'FLHN_ELGA'
```

thermal #options

```
♦  RESULTAT =resu  ,                          /  [evol_ther]

◊THERMIQUE=           |   'FLUX_ELGA'
                      |   'FLUX_ELNO'
                      |   'FLUX_NOEU'
                      |   'HYDR_NOEU'
                      |   'SOUR_ELGA'
                      |   'ETHE_ELEM'
```

acoustic #options

```
♦RESULTAT  =resu ,                            /  [acou_harmo]
```

```
                                                        /   [mode_acou]

                ◊ACOUSTIQUE    =         |   'PRAC_ELNO'
                                         |   'PRAC_NOEU'
                                         |   'PRME_ELNO'
                                         |   'INTE_ELNO'
                                         |   'INTE_NOEU'
```

#options for the generalized forces and nodal reactions

```
                ♦RESULTAT  =resu ,

                ◊FORCE  =             |   'FORC_NODA'
                                      |   'REAC_NODA'
```

#calcul of a field user

```
                ◊CHAM_UTIL     = _F (

                   ♦NOM_CHAM  =ncham    ,
                   ♦   /  CRITERE=      |   'VMIS',
                                        |   'INVA_2'
                                        |   'TRACE',
                      /FORMULE     =l_forme  ,                  [formula]
                   ♦NUMERIQUE_CHAM_RESU=numeric         ,                 [I]

                   ),

                ◊TITRE  = title  ,                              [l_Kn]
                ◊INFO   =     /   1,                            [DEFAUT]
                              /2  ,
)
```

2.1 Operands RESULTAT/MODELE/CHAM_MATER/CARA_ELEM/EXCIT

2.1.1 Opérandes RESULTAT

♦RESULTAT =resu

Nom of the data structure result to enrich. This argument can be same as that used for the concept enriched by the operator, or a different name, which will create a new data structure result.

Note:

In the majority of the situations, the data structure resu contains all the necessary information with the computation of the options: the elementary model, material field, characteristics, loadings. Key words MODELE , CHAM_MATER , CARA_ELEM and EXCIT are thus useless.

2.1.2 Operands MODELE / CHAM_MATER / CARA_ELEM.

◊MODELE = Mo

Nom of the model on which the forces are computed, the stresses, the strains, etc
It is optional because it can be extracted the result.

◊CHAM_MATER = chmater

Material field associated with the model Mo. This key word is optional and must be provided only in exceptional cases (voluntary amendment of the material for example).

◊CARA_ELEM = carac

Caractéristiques elementary associated with the model Mo if it contains structural elements or if the isoparametric elements are affected by a local coordinate system of anisotropy.
This key word is optional because it can be extracted the result.

2.1.3 Key word EXCIT

This key word factor (optional) makes it possible to specify the thermal or mechanical loadings to use for the computation of the options, instead of those which were useful in computation of data structure specified under key word RESULTAT.

The definition of this key word is identical to that of the commands which built the data structure resu : see commands MECA_STATIQUE [U4.51.01], STAT_NON_LINE [U4.51.03], DYNA_LINE_HARM [U4.53.11], and DYNA_LINE_TRAN [U4.53.02].

2.2 Selection of the meshes concerned with computation

Les key words TOUT, GROUP_MA and MESH make it possible to the user to choose the meshes on which it wishes to make his elementary computations of postprocessing.

/TOUT = "OUI"

Toutes the meshes (carrying finite elements) will be treated. It is the value by default.

```
/   |      GROUP_MA=l_grma
    |  MAILLE=l_maille
```

Seules the meshes included in l_grma and/or l_maille will be treated.

2.3 Selection of the sequence numbers

the use of key words TOUT_ORDRE, NUM_ORDRE, INST, FREQ is described in the document [U4.71.00].

2.4 Localization of the Dans

fields the continuation of the document one will not explicitly specify the localization of the fields. Indeed, the localization is given in the name of the field (and thus of the option):

- Field by element: *_ELEM
- Field at the Gauss points by element: *_ELGA
- Field with the nodes by element: *_ELNO
- Field with the nodes: *_NOEU

Les fields, for the majority, is computed natively at the Gauss points (*_ELGA).

The fields with the nodes by element (*_ELNO) are obtained by extrapolation starting from the field at the Gauss points (detailed method in [R3.06.03]).

The fields with the nodes (*_NOEU) are obtained starting from the fields with the nodes by element by making a simple arithmetic mean (not balanced by the size of the meshes) of the values recorded on the elements in a given node.

Notice 1:

*For the computation of the equivalents, the fields with the nodes by element (*_ELNO) are not obtained by extrapolation starting from the field at the Gauss points. Extrapolation is made at the stress field or of strain then one computes the field of equivalent.*

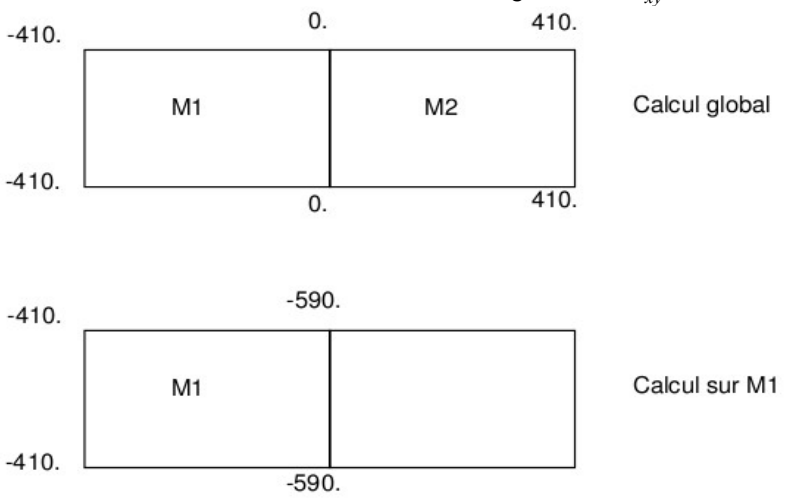
Notice 2:

The averages with the nodes of computed fields in local coordinate systems are licit only if the angles between these references are weak. In the contrary case, they do not have a meaning.

Notice 3:

*When the key words GROUP_MA or MESH are indicated for the computation of an option *_NOEU, the arithmetic mean is made on the selected meshes. Thus this result is different from that obtained by making a total computation then restricted with the only selected meshes.*

Example: One considers a structure whose shearing stress σ_{xy} is worth:



In total computation, σ_{xy} is null on $M1 \cap M2$ like average of two opposite values. These values are far from being null, as computation shows it on $M1$ only. The values on the border of the required field are thus to interpret with precaution.

For the structural elements which have subpoints (multifibre plates, shells, beams, pipes), the fields of the *_ELGA type and *_ELNO are computed on all the subpoints. To obtain a field on only one subpoint (a layer and a level for example), it is necessary to make an extraction *via* operator POST_CHAMP (options EXTR_COQUE, EXTR_PMF and EXTR_TUYAU). Moreover this intermediate operation is essential for compute a field of the *_NOEU type for these structural elements: the fields of the *_NOEU type indeed never have subpoint.

Finally the computation options of energy never produce fields at subpoint. Indeed for the structural elements, the field is integrated in the thickness (integration made on the subpoints).

2.5 Dependence of the fields

the computation of a field can require the preliminary computation of one or more other fields. Thus for example for compute a field *_NOEU it is necessary to have the same field by element for the nodes *_ELNO and the Gauss points *_ELGA.

This dependence is solved by the operator CALC_CHAMP who carries out the computation of the intermediate fields automatically. It is thus not necessary for the user to know the shaft of dependence of the options.
Only the fields explicitly required by the user are saved in the data structure result.

2.6 Operands for the mechanical options

2.6.1 Computation options of the stresses (Opérande FORCED)

Les components of the stress fields and forces generalized are detailed in the document [U2.01.05].

| 'EFGE_ELGA'
| 'EFGE_ELNO'
| 'EFGE_NOEU'

Computation of the generalized forces (structural elements).

It acts is of an extraction of the forces contained in field SIEF_ELGA/STRX_ELGA (case of the beam elements/pipes or of discrete), is of a computation by integration of the stresses (case of the multifibre beam elements or plates and shells).

Notice 1:
Field EFGE_ELNO is not always an extrapolation of field EFGE_ELGA ; in particular for a linear computation where this field is computed directly starting from displacement. This is why certain components are not computed (resettings) into nonlinear.

Notice 2:
For the offset plates, the forces are computed in the "plane" of the mesh. If one wishes these forces in the average "plan" of the plate, command POST_CHAMP should be used / COQUE_EXCENT.

| 'SIEF_ELGA'
| 'SIEF_ELNO'
| 'SIEF_NOEU'

Computation of the stress state (forced or forces generalized according to the modelization) starting from displacements (linear elasticity), see [U2.01.05].

Note:
Field "SIEF_ELGA" is calculated nativement by the nonlinear operators of resolution. It is always present in a data structure result of the evol_noli type.

| 'SIGM_ELGA'

| 'SIGM_ELNO'
| 'SIGM_NOEU'

Computation of the stress state.

It is actually about an extraction of the stresses contained in field SIEF_ELGA, to see [U2.01.05].

| 'SIPO_ELNO'
| 'SIPO_NOEU'

Computation of the stresses in the section of beam broken up into contributions of each generalized force.

List components of the field:

SN	Contribution of the normal force N with σ_{xx} , $\sigma_{xx}=\frac{N}{A}$
SMFY	Contribution of the bending moment MFY with σ_{xx} , $\sigma_{xx}=z\frac{MFY}{I_Y}$
SMFZ	Contribution of the bending moment MFZ with σ_{xx} , $\sigma_{xx}=-y\frac{MFZ}{I_Z}$
SVY	Contribution of the shearing force VY with σ_{xy} , $\sigma_{xy}=\frac{VY\,a_Y}{A}$ a_Y shear coefficient in direction y
SVZ	Contribution of the shearing force VZ with σ_{xz} , $\sigma_{xz}=\frac{VZ\,a_Z}{A}$ a_Z shear coefficient in direction z
SMT	Contribution of the twisting moment MX with σ_{yz} , $\sigma_{yz}=\frac{MX\,R_T}{J_x}$

Les forced above are expressed in the local coordinate system, i.e. the main reference of inertia of the cross-section [R3.08.01].

The values of σ_{xx} due to the two bending moments are the maximum values of those computed in Y_{min} , Y_{max} on the one hand, and in Z_{min} , Z_{max} on the other hand (except for a general section where it is the user who provides the localization of the extremum with the key words RY , RZ and RT *cf* AFFE_CARA_ELEM [U4.42.01]).

For a rectangular section:

- one computes the value of SMFY in $z=HZ/2$,
- one computes the value of SMFZ in $y=HY/2$.

For a circular section, one computes the values of SMFY and SMFZ for y and z being worth R .

| 'SIPM_ELNO'

Computation of the stresses maximum and minimum in the section of beam starting from the generalized forces (linear elasticity).
The same remark that for SIPO_ELNO applies in the case of a general section.

| 'SIRO_ELEM'

Computation of the stresses projected on the skin of a volume (for example on the facings of a hydraulic work.

List components of the field:

Component SIG_NX SIG_NY	SIG_NZ σ_X σ_Y, σ_Z in the total reference of $\vec{\sigma}_n$
SIG_N	Component SIG_N
Valeur SIG_TX SIG_TY	SIG_TZ σ_X σ_Y, σ_Z in the total reference of $\vec{\sigma}_t$
Component SIG_T1X SIG_T1Y	SIG_T1Z σ_X σ_Y, σ_Z in the total reference of $\vec{\sigma}_{t1}$
SIG_T1	Component SIG_{T1}
Eigenvalue SIG_T2X SIG_T2Y	SIG_T2Z σ_X σ_Y, σ_Z in the total reference of $\vec{\sigma}_{t2}$
SIG_T2	Eigenvalue SIG_{T2}

Ces fields are evaluated starting from a stress field calculated on the voluminal meshes (MODELISATION= ` 3D' or "3D SI'):

- Identification of the voluminal meshes corresponding to the surface breakages of the mesh group;
- Recovery of stresses 3D to assign them to the nodes sides;
- Average of each component of the tensor of the stresses in the center of the sides of elements;
- One places oneself in a reference composed by the normal vector $\vec{n}$ at the breakage and the plane of the breakage. A noted tensor is obtained $[\sigma]$.
- One evaluates $[\sigma]\vec{n}=\vec{\sigma}_n+\vec{\sigma}_t$, $\vec{\sigma}_n$ being a vector colinéaire with $\vec{n}$. $\vec{\sigma}_t$ is then a vector representing the shears which are negligible in the case of the faces upstream/downstream of a stopping. It is noted $\vec{\sigma}_n=SIG_N\vec{n}$ and SIG_N indicated the presence of tension if it is positive and of compression if it is negative.
- One thus places oneself on the assumption of negligible shears $[\sigma]=\begin{bmatrix}\sigma_{2D} & 0\\ 0 & SIG_N\end{bmatrix}$

  One seeks the stress vectors main corresponding to σ_{2D} : one thus obtains the vectors $\vec{\sigma}_{t1}$ and $\vec{\sigma}_{t2}$ which are in the plane of the breakage and the eigenvalues SIG_{T1} and SIG_{T2}

Remarque 1:

In the case of breakages plunged in volume, one chooses the voluminal mesh which is side "-" compared to the norm of the breakage. The user has the possibility thanks to command MODI_MAILLAGE / ORIE_PEAU_3D / GROUP_MA_VOLU to reorientate this norm as it wishes it. This convention corresponds to what is made on the facings external of the stopping if the norm with the breakages is "outbound".

Notice 2:

If TOUT= is informed'OUI", the list of the meshes is filtered to keep only the meshes skin.

2.6.2 Computation options of the strains (Opérande DEFORMATION)

Les components of the strain fields are detailed in the document [U2.01.05].

| 'DEGE_ELGA'
| 'DEGE_ELNO'
| 'DEGE_NOEU'

Computation of the strains generalized starting from displacements. This option has meaning only for the structural elements (beam, plate, pipe).
The generalized strains are obtained in the local coordinate system of the element.

| 'EPFD_ELGA'
| 'EPFD_ELNO'
| 'EPFD_NOEU'

Computation of the strains of creep of desiccation, for models BETON_UMLV_FP and BETON_BURGER_FP.

| 'EPFP_ELGA'
| 'EPFP_ELNO'
| 'EPFP_NOEU'

Computation of the strains of clean creep associated with model GRANGER_FP, model BETON_UMLV_FP or model BETON_BURGER_FP.

| 'EPME_ELGA'
| "EPME_ELNO"
| 'EPME_NOEU'

Computation of the "mechanical" strains starting from displacements. This computation is made in theory of "**small displacements** ". The computed strains are equal to the total deflections minus the thermal strains.

$$\varepsilon_{ij}^{m}(u)=\frac{1}{2}\left(u_{i,j}+u_{j,i}\right)-\varepsilon^{th}$$

| 'EPMG_ELGA'
| 'EPMG_ELNO'
| 'EPMG_NOEU'

Computation of the "mechanical" strains starting from displacements. This computation is made in theory of "**large displacements** ". The computed strains are equal to the total deflections minus the thermal strains.

$$E_{ij}^{m}(u)=\frac{1}{2}\left(u_{i,j}+u_{j,i}+u_{k,i}u_{k,j}\right)-\varepsilon^{th}$$

| 'EPSG_ELGA'
| 'EPSG_ELNO'
| 'EPSG_NOEU'

Computation of the strains of Green-Lagrange.

$$E_{ij}(u)=\frac{1}{2}\left(u_{i,j}+u_{j,i}+u_{k,i}u_{k,j}\right)$$

| 'EPSI_ELGA'
| 'EPSI_ELNO'
| 'EPSI_NOEU'

Computation of the strains starting from displacements.

$$\varepsilon_{ij}(u)=\frac{1}{2}\left(u_{i,j}+u_{j,i}\right)$$

For the structural elements, these strains are obtained in the local coordinate system of the element.

| 'EPSP_ELGA'
| 'EPSP_ELNO'
| 'EPSP_NOEU'

Computation of the unelastic strains starting from the field of displacement u, of the stresses σ, the temperature T, the possible unelastic strains ε^a, and the intern variables,

$$\varepsilon^p = \varepsilon(u) - A^{-1}\sigma - \varepsilon^{th}(T) - \varepsilon^a - \varepsilon^{fl}$$

where ε^{fl} is the clean strain of creep of Granger.

| 'EPVC_ELGA'
| 'EPVC_ELNO'
| 'EPVC_NOEU'

Computation of the strains related to the command variables. For time only the following components are defined:

- thermal strains: EPTHER_L, EPTHER_T, EPTHER_N such as: $\varepsilon_i^{th} = \alpha_i (T - T_{ref}) ; i \in \{L, T, N\}$ (if the material is isotropic, the 3 components are equal), T being the temperature and α_i the thermal coefficient of thermal expansion;
- shrinkage of drying EPSECH (used for the models describing the behavior of the concrete) $\varepsilon^{sech} = -K_{dessic}(S_{ref} - S)$, S being the command variable drying and K_{dessic} the coefficient of shrinkage of desiccation;
- shrinkage of hydration EPHYDR (used for the models describing the behavior of the concrete) $\varepsilon^{hydr} = -B_{endog}\, h$, h being the command variable hydration, and B_{endog} being the endogenous coefficient of shrinkage.
- Strain related to the fluid pressure (for the thermo-hydro-mechanics with a resolution by sequence): EPPTOT such as: $\varepsilon^{ptot} = \frac{b}{3\mathrm{K}}\, p_{tot}$, p_{tot} is the command variable stagnation pressure of fluid, b is the coefficient of Biot, K is the elasticity modulus.

2.6.3 Options of extraction of the intern variables (Opérande VARI_INTERNE)

| 'VAEX_ELGA'
| 'VAEX_ELNO'
| 'VAEX_NOEU'

Extraction of **the intern variables in THM only**.

The goal of this option is to be able post-to treat the intern variables in THM in a user-friendly way. The principle of these fields east extracting from field VARI_ELGA (or VARI_ELNO) it (one and only one) intern variable which interests us via a key word without having to know its name in field VARI_*.

List possible components of the field (the field has only one component, that chosen by the user via NOM_VARI):

DPORO	Variation of the porosity of material
DRHOLQ	Variation of the density of material
DPVP	Variation of the steam pressure
SATLIQ	Saturation of voluminal
liquid	EVP cumulated Plastic strain
IND_ETA	mechanical Indicateur of state
D	Valeur of damage
IND_END	Indicateur of damage

TEMP_MAX	maximum Temperature
GAMP	plastic Strain critical déviatoire
cumulated	PCR Pressure
SEUIL_HYD	hydrous Seuil
IND_HYD	Indicateur of irreversibility hydrous
PCOHE	Pressure of cohesion
COMP_ROC	Comportement of isotropic
rock	SEUIL_ISO Seuil
ANG_DEV	Angle of the threshold déviatoire
X11	Components of the tensor of kinematic hardening
X22	Components of the tensor of kinematic hardening
X33	Components of the tensor of kinematic hardening
X12	Components of the tensor of kinematic hardening
X13	Components of the tensor of kinematic hardening
X23	Components of the tensor of kinematic hardening
DIST_DEV	Distance standardized with the threshold déviatoire
DEV_SUR_CRIT	Rapport enters the threshold déviatoire and threshold deviatoric critical
DIST_ISO	Distance standardized to isotropic threshold
NB_ITER	Nombre of internal iterations
ARRET	Valeur of the local test of stop of iterative process
NB_REDE	Nombre of local recutting of time step
SIGNE	Signe of the contracted product of the deviatoric stress by the deviatoric plastic strain

◊NOM_VARI = / nom_vari, [TXM]

internal Variable name.

Notice 1:

When the variable to be extracted does not form part of the intern variables of the models concerned, an alarm is emitted but the field is affected all the same with R8VIDE () (very large real number about 1.0E+308).

Notice 2:

Field VAEX_NOEU is calculated starting from VAEX_ELNO and not by extraction of field VARI_NOEU.

| 'VARC_ELGA'

Computation of the command variables having been used for a mechanical computation.

List components of the field:

TEMP	Cf. documentation of command AFFE_MATERIAU [U4.43.03] for the definition of each component.
HYDR	
SECH	
CORR	
IRRA	
PTOT	
DIVU	
NEUT1	
NEUT2	

Note:

All the components are systematically computed. The variables which were not defined are initialized with value R8VIDE () (very large real number about 1.0E+308).

| 'VARI_ELNO'
| 'VARI_NOEU'

Computation of the intern variables.

List components of the field:

V1	Intern variable 1
...	
VI	Intern variable I
...	
Vn	Intern variable N

the number and the type of these intern variables are specific to each model of behavior (*cf* [U4.51.11]).

Note:

Field "VARI_ELGA" is calculated nativement by the nonlinear operators of resolution. It is always present in a SD result of the evol_noli type.

2.6.4 Computation options of energy (Opérande ENERGIE)

| 'DISS_ELEM'

Computation of the energy dissipated by the damage. The field obtained has only one component of name "ENDO".

List components of the field:

ENDO	Energy dissipated by the Remarque

damage:

Valid only for the elements DKTG and model GLRC_DM. Its statement is given in [R7.01.32].

| 'DISS_ELGA'
| 'DISS_ELNO'
| 'DISS_NOEU'

Computation of the density of energy dissipated by the damage. The field obtained has only one component of name "ENDO".

List components of the field:

ENDO	Energy dissipated by the Remarque

damage:

Valid only for the elements DKTG and model GLRC_DM. Its statement is given in [R7.01.32].

| 'ECIN_ELEM'

Computation of kinetic energy.

$$E_c = \frac{1}{2} m v^2$$

List components of the field:

TOTALE	Kinetic energy
additional Components for the plates and shells:	
MEMBRANE Contributions	BENDING with kinetic energy (cf [R3.07.03])
Component additional for the curved beams:	
PLAN_XY PLAN_XZ	Contributions with kinetic energy (cf [R3.08.01])
Component additional for the discrete ones:	
DX DY DZ DRX DRY DRZ	Contributions with kinetic energy

| 'ENEL_ELEM'

Computation of elastic strain energy.

$$E_p = \frac{1}{2} \sigma \, A^{-1} \sigma$$

List components of the field:

TOTALE	Elastic strain energy
additional Components for the plates and shells:	
MEMBRANE Contributions	BENDING with elastic strain energy (cf [R3.07.03])

| 'ENEL_ELGA'
| 'ENEL_ELNO'
| 'ENEL_NOEU'

Computation of the density of elastic strain energy.

List components of the field:

TOTALE	Elastic strain energy
additional Components for the plates and shells:	
MEMBRANE Contributions	BENDING with elastic strain energy (cf [R3.07.03])

| 'EPOT_ELEM "

Computation of the potential energy of strain, starting from the field of displacement u and of the fields of temperature T :

List components of the field:

TOTALE	Potential energy
additional Components for the plates and shells:	
MEMBRANE Contributions	BENDING with potential energy (cf [R3.07.03])
Component additional for the straight beams:	
TRAC_COM TORSION FLEX_Y FLEX_Z	Contributions with potential energy (cf [R3.08.01])
Component additional for the curved beams:	
PLAN_XY PLAN_XZ	Contributions with potential energy (cf [R3.08.01])
Component additional for the discrete ones:	
DX DY DZ DRX DRY DRZ	Contributions with potential energy

- for elements of continuous medium 2D and 3D:

$$E_{pot}=\frac{1}{2}\int_{element} \varepsilon(u)\, A\, \varepsilon(u)\, dv - \int_{element} \varepsilon(u)\, A\, \varepsilon^{th}(u)\, dv+\frac{1}{2}\int_{element} \varepsilon^{th}(u)\, A\, \varepsilon^{th}(u)\, dv$$

- for the beam elements:

$$E_{pot}=\frac{1}{2}u^T K_e u-u^T B^T A\, \varepsilon^{th}+\frac{1}{2}\varepsilon^{th} A\, \varepsilon^{th}$$

- for the shell elements and shells:

$$E_{pot}=\frac{1}{2}u^T K_e u-u^T B^T A\, \varepsilon^{th}$$

| "ETOT_ELEM "

Computation of the increment of total strain energy enters time running and previous time.

List components of the field:

TOTALE	Incrément of total strain energy

| 'ETOT_ELGA"
| 'ETOT_ELNO'
| 'ETOT _NOEU'

Computation of the increment of density of total strain energy enters time running and previous time.

List components of the field:

TOTALE Incrément of total strain energy

2.6.5 Computation options of criteria (Opérande CRITERES)

| 'DERA_ELGA'
| 'DERA_ELNO'
| 'DERA_NOEU'

Computation of the local indicator of discharge and indicator of loss of radiality [R4.20.01].

List components of the field:

DCHA_V	Indicateur of discharge calculated on the deviative tensor of stresses
DCHA_T	Indicateur of discharge calculated on the total tensor of stresses
IND_DCHA	Indicateur allowing to know if the discharge would remain elastic or if there would be a risk of plasticization if one used a pure kinematic hardening
VAL_DCHA	Indique the proportion of output of the criterion in the case of abusive discharge
X11 X22 X33 X12 X13 X23	Components of the kinematical tensor used for the computation of IND_DCHA
RADI_V	Indicateur of the variation of the direction of the stresses between times t and $t+\Delta t$
ERR_RADI	Erreur η due to the discretization in time, directly connected to the rotation of the norm on the surface of load

DCHA_V and DCHA_T express, in both cases, the variation relative of the norm of the stresses within the meaning of Von Mises: $I_1=\frac{\|\sigma(M,t+\Delta t)\|-\|\boldsymbol{\sigma}(M,t)\|}{\|\boldsymbol{\sigma}(M,t+\Delta t)\|}$, the norm being function of the behavior (isotropic hardening or linear kinematics)

IND_DCHA can take the following values:
- 0 : unconstrained initial value;
- 1 : if elastic load;
- 2 : if plastic load;
- -1 : if licit elastic discharge (whatever the type of hardening);
- -2 : if abusive discharge (one would have plasticized with a kinematic hardening).

RADI_V is given by the following relation:

$$I_2=1-\frac{|\boldsymbol{\sigma}(M,t).\Delta\boldsymbol{\sigma}|}{\|\boldsymbol{\sigma}(M,t)\|\|\Delta\boldsymbol{\sigma}\|}$$

This quantity is null when the radiality is preserved during the increment of time.

ERR_RADI is the angle enters $\boldsymbol{n}^-$, the norm with the plasticity criterion at the beginning of the time step (urgent t^-), and $\boldsymbol{n}^+$, the norm with the plasticity criterion computed at the end of the time step (urgent t^+) in the following way:

$$\eta=\frac{1}{2}\|\Delta\boldsymbol{n}\|=\frac{1}{2}\|\boldsymbol{n}^+-\boldsymbol{n}^-\|=\left|\sin\left(\frac{\alpha}{2}\right)\right|$$

That provides a measurement of the error (also used to refine the time step [U4.51.11]. This criterion is operational for the elastoplastic behaviors of Von Mises with hardening isotropic, kinematical linear and mixed: VMIS_ISOT_LINE , VMIS_ISOT_TRAC , VMIS_ISOT_PUIS , VMIS_CINE_LINE , VMIS_ECMI_LINE , VMIS_ECMI_TRAC, and for the behaviors élasto-visco-plastics of Chaboche: VMIS_CIN1_CHAB, VMIS_CIN2_CHAB, VMIS_CIN2_MEMO, VISC_CIN1_CHAB, VISC_CIN2_CHAB, VISC_CIN2_MEMO.

Note:

The computation of these options requires to compare the stress fields with times t_i and t_{i+1}. The result is arranged with the sequence number associated with time t_i.

The indicator of discharge is computed by: $ID=\frac{\|\sigma_{i+1}\|-\|\sigma_i\|}{\|\sigma_{i+1}\|}$.
By default, computation is done for the sequence numbers 1 *with* $n-1$ *. But if one specifies the list of time (with "holes" possibly), computation will relate to only required times but it will always compare time* t_i *with time* t_{i+1} *in the list of times having been used to make nonlinear computation.*

| 'ENDO_ELGA'
| 'ENDO_ELNO'
| 'ENDO_NOEU'

Computation of the damage d starting from the tensor of the stresses and the cumulated plastic strain p .

List components of the field:

TRIAX	Taux of triaxiality
SI_ENDO	Equivalent stress of damage of Lemaître-Sermage
COENDO	Contrainte of damage of Lemaître-Sermage standardized
DOM_LEM	Dommage of Lemaître-Sermage

the rate of triaxiality α is given by the following relation:

$$\alpha=\frac{\sigma_h}{\sigma_{eq}}$$

and the equivalent stress of damage $\sigma*$:

$$\sigma*=\sigma_{eq}\sqrt{\frac{2}{3}(1+\nu)+3(1-2\nu)\alpha^2}$$

$$s = \sigma-\frac{1}{3}tr(\sigma)\cdot I$$

with: $\sigma_{eq} = \sqrt{\frac{3}{2}s:s}$

$$\sigma_h = \frac{1}{3}tr(\sigma)$$

The kinetics of damage are given by the model of Lemaître-Sermage:

$$\dot{d}=\left[\frac{Y}{S}\right]^s\dot{p} \text{ so } p\geq p_{seuil} \text{ with } Y=\frac{\sigma*^2}{2E(1-D)^2}$$

where S and s are coefficients characteristic of the material and p_{seuil} the threshold of damage related to the energy stored in the material (if $s=1$ one obtains the conventional model of Lemaître).

| 'EPEQ_ELGA'
| 'EPEQ_ELNO'
| 'EPEQ_NOEU'
| 'EPMQ_ELGA'
| 'EPMQ_ELNO'
| 'EPMQ_NOEU'

Computation of the computed strains equivalent starting from fields EPSI_*, or EPME_*).

List components of the field:

INVA_2	equivalent Déformation of Von Mises
PRIN_1 PRIN_2 PRIN_3	Principal strains, arranged in the order ascending
INVA_2SG	equivalent Déformation of Von Mises signed by the trace of ε
VECT_1_X VECT_1_Y VECT_1_Z VECT_2_X VECT_2_Y VECT_2_Z VECT_3_X VECT_3_Y VECT_3_Z	Principal directions

the equivalent strain of Von Mises is given by the following statement:

$$\text{INVA\_2}=\sqrt{\frac{2}{3}dev(\varepsilon)_{ij}dev(\varepsilon)_{ji}} \text{ with } dev(\varepsilon)_{ij}=\varepsilon_{ij}-\frac{1}{3}tr(\varepsilon)\delta_{ij}$$

Remarque:

It is noted that the equivalent strains obtained starting from EPSI_* and EPME_* are identical. Indeed, the difference between the two tensors is a spherical tensor (thermal strain). As the equivalent strain is obtained starting from the second invariant of the deviator, the spherical tensor "disappears" when the deviator is taken.

| 'INDL_ELGA'

Computation of the indicator of localization, based on the acoustic tensor (criterion of RICE).

List components of the field:

INDICE	Indicateur of localization 0 if $det(N.H.N)>0$ 1 if not, which corresponds has the initiation of normal
localization	DIR1 Première at the area of normal
localization	DIR2 Deuxième at the area of normal
localization	DIR3 Troisième at the area of normal
localization	DIR4 Quatrième at the area of indicating

Cet localization defines a state from which the local problem of integration of the behavior loses its character of unicity. It is defined by: $det(N.H.N)\leq 0$, where H appoints the tangent operator and N the norm with the directions of localization.

Note:

The method is developed only in the case $2D$ and for the constitutive laws of the type DRUCK_PRAGER and HUJEUX.

| 'PDIL_ELGA'

Computation of the modulus of rigidity of microcomputer-dilation.

List components of the field:

A1_LC2	Module rigidity of microcomputer-dilation

option PDIL_ELGA provides in the frame of the mediums of second gradient of dilation the value of modulus A1_LC2, making it possible to control the interval of the nontrivial solution of the initially homogeneous problem [R5.04.03].

The computation of A1_LC2 is obtained via the rating of a function depending on the geometrical directional sense of the material tape considered. The angular discretization currently imposed is equal to $0.1°$.

Note:
The method is developed only for the constitutive laws of the type DRUCK_PRAGER and HUJEUX.

| 'SIEQ_ELGA'
| 'SIEQ_ELNO'
| 'SIEQ_NOEU'

Computes equivalent stresses computed starting from the stress fields.

List components of the field:

VMIS	Equivalent stress of Von Mises
TRESCA	Forced of Tresca
PRIN_1 PRIN_2 PRIN_3	Principal stresses, arranged in the order ascending
VMIS_SG	Equivalent stress of Von Mises signed by the trace of σ
VECT_1_X VECT_1_Y VECT_1_Z VECT_2_X VECT_2_Y VECT_2_Z VECT_3_X VECT_3_Y VECT_3_Z	Principal directions
TRSIG	Trace σ
TRIAX	Taux of triaxiality

the equivalent stress of Von Mises is given by the following statement:

$$\text{VMIS}=\sqrt{\frac{3}{2}s_{ij}s_{ji}} \text{ with } s_{ij}=\sigma_{ij}-\frac{1}{3}tr(\sigma)\delta_{ij}$$

the rate of triaxiality is given by the following statement:

$$\text{TRIAX}=\frac{\text{TRSIG}}{\text{VMIS}}$$

2.6.6 Computation option of hydraulic flows (HYDRAULIC Opérande)

| 'FLHN_ELGA'

Computation of hydraulic flows in THM $\Phi_{ij}=M_{ij}.\nu$ on the edge elements (2D or 3D) starting from the vector flow with the nodes.
M_{ij} is the hydraulic vector flow of the component ij.

List components of the field:

FH11	
FH22	
FH12	
FH21	

2.7 Opérandes for the thermal options

2.7.1 THERMAL Opérande

| 'FLUX_ELGA'
| 'FLUX_ELNO'
| 'FLUX_NOEU'

Computation of heat flux starting from the temperature.

List components of the field:

FLUX FLUY FLUZ	Heat flux in the three directions of space (in the average layer for the shells)
Component additional for the shells:	
FLUX_INF FLUY_INF FLUZ_INF	Heat flux in the three directions of space in skin lower
FLUX_INF FLUY_INF FLUZ_INF	Heat flux in the three directions of space in higher skin

| 'HYDR_NOEU'

Computation of the hydration.

List components of the field:

HYDR	Hydratation

Note:

Field "HYDR_ELNO" is calculated nativement by the nonlinear operator of thermal THER_NON_LINE for the modelization of the concrete [R7.01.12].

| 'SOUR_ELGA'

Computation of a heat source.

List components of the field:

SOUR	Heat source

Cette source is calculated starting from an electric potential via the model of Ohm. This electric potential must be computed by operator THER_LINEAIRE [U4.54.01] by making the analogies necessary. This source can be then used in a thermal computation via key word SOUR_CALCULEE of command AFFE_CHAR_THER [U4.44.02].

| 'ETHE_ELEM'

Computation of thermal energy to the equilibrium starting from the field of temperature T.

List components of the field:

Thermal	TOTALE Energy

2.8 Opérandes for the acoustic options

2.8.1 ACOUSTIC Opérande

| 'INTE_ELNO'
| 'INTE_NOEU'

Computation of the acoustic intensity. The definitions are in [R4.02.01].

List components of the field:

Acoustic	INTX_R Intensity, real part according to axis X
INTY_R	acoustic Intensity, real part according to acoustic axis
there	INTZ_R Intensity, real part according to acoustic axis
Z	INTX_I Intensity, imaginary part according to axis X
INTY_I	acoustic Intensity, imaginary part according to acoustic axis
there	INTZ_I Intensity, imaginary part according to axis Z

| 'PRAC_ELNO'
| 'PRAC_NOEU'

Computation of the pressure to the nodes partly real, imaginary part and decibels.

List components of the field:

Acoustic	PRES_I Pressure, real part
PRES_R	acoustic Pressure, imaginary part
dB	acoustic Pressure in decibel

| 'PRME_ELNO'

Computation of the pressure to the nodes for the elements FLUID.

List components of the field:

Acoustic	dB Pressure in Opérande

2.9 decibel for the nodal forces and reactions

2.9.1 Opérande FORCE

| 'FORC_NODA'

Computation of the nodal forces generalized starting from the stresses at the Gauss points.

List components of the field:

Nodal DX DY	DZ Forces
additional Components for the structural elements:	
Nodal DRX DRY	DRZ Forces

computation are done in the following way:

$$\int_{\Omega} \sigma \varepsilon(\boldsymbol{u}) d\Omega = \sum_{K} \int_{K} \sigma^{K} \varepsilon(\boldsymbol{u_K}) dK = \sum_{K} \int_{K} \sigma^{K} \boldsymbol{B} \boldsymbol{u_K} dK$$

with σ_K stresses at the Gauss points of the element K ;

$\boldsymbol{B}$ the operator finite elements of generalized strains;

$\boldsymbol{u_K}$ generalized elementary displacement.

$=\sum_K F_K \boldsymbol{u_K}$ with $F_K=\left\{\int_K {}^t\boldsymbol{B}\,\sigma^K\,dK\right\}$ the generalized nodal forces

where B is the matrix connecting the strains of the 1st order to displacements.

The dimension of the nodal forces is dual of that of $\boldsymbol{u_K}$ to give a work (in Joules).

For the beam elements and the discrete elements, the stresses at the Gauss points are in fact the nodal forces generalized in the reference of the element (obtained by the product of the stiffness matrix of the element by displacement and by taking account of the forces of thermal origin and the forces distributed). The computation of the nodal forces is done by projecting the nodal forces contained in the field of symbolic name "SIEF_ELGA" in the total reference. The summation above on the elements applies then. Components DX, DY and DZ give the forces and DRX, DRY and DRZ the moments.

For the axisymmetric elements, integration in theta is done on a sector of $1\,radian$. If one wants the integral of the surface force on all the disc it is thus necessary to multiply par. 2π

Pour the elements in plane strain, computation is made on a tape of width unit. The computed nodal forces are thus by way of forces per unit length. If one wants to calculate the nodal forces being exerted on a structure of width l, it is necessary to multiply the result in D_PLAN by l, with this close the assumption of plane strain is not valid close to the two sides. There will be thus an approximate result.

For the solid elements (3D, 2D and bars), the FORC_NODA in general have the dimension of a force. It is about a field on the nodes of the mesh where the value in a node is obtained starting from the stresses computed on the convergent elements with this node, thus their values thus vary when the mesh changes. In the absence of distributed loading, the equilibrium imposes their nullity in an interior node, while they correspond to the reaction on the bearings where one imposes a kinematic relation (case of an imposed displacement).

In the case of the shells, components DX, DY and DZ give the FORC_NODA (dimension of a force) in the total reference of the mesh. These components are built with the normal force and cutting-edges in the shell. Components DRX, DRY and DRZ give the FORC_NODA (one moment dimension) in the total reference of the mesh, built with the bending moments in the shell.

In hydraulics, the generalized nodal forces associated with each component correspond to a flow. If one notes $\mathbf{Q}^T\boldsymbol{\sigma}_0$ the result of FORC_NODA, for the hydraulic equations, then for a time step Δt, one a:

$$\int_\Omega \mathbf{Q}^T\boldsymbol{\sigma}_0\,p^*\,d\,\Omega=-\Delta t\int_\Omega \mathbf{M}^-\nabla\,p^*\,d\,\Omega$$

Dans FORC_NODA :

- with degree of freedom PRE1 is associated water flow $-\Delta t\int_\Omega \boldsymbol{M_{vp}}+\boldsymbol{M_w}^-\nabla\,p^*\,d\,\Omega$
- with degree of freedom PRE2 is associated flow with the gas component $-\Delta t\int_\Omega \boldsymbol{M_{ad}}+\boldsymbol{M_{as}}^-\nabla\,p^*\,d\,\Omega$
- with degree of freedom TEMP is associated the heat flux $-\Delta t\int_\Omega \boldsymbol{q}^-\nabla\,T^*\,d\,\Omega$

with q the heat flux and M_w M_{vp}, M_{as} and M_{ad} hydraulic flows of liquid water, the vapor, the air (or very other composing) dry and of the air dissolved in the liquid. These data correspond to the generalized stresses of *Code_Aster* $M_{11}, M_{12}, M_{21}, M_{22}$.

| 'REAC_NODA'

Computation of the nodal forces of reactions generalized with the nodes, the stresses at the Gauss points.

List components of the field:

Nodal DX DY	DZ Forces
additional Components for the structural elements:	
Nodal DRX DRY	DRZ Forces

Pour the result concepts of the evol_elas type, mult_elas, fourier_elas or evol_noli, this computation are done by:

$$\int_{\Omega} \sigma(\varepsilon(\boldsymbol{u})).\varepsilon(\boldsymbol{v}) d\Omega - \boldsymbol{L}(\boldsymbol{v})$$

with $\boldsymbol{L}(\boldsymbol{v}) = \int_{\Omega} \boldsymbol{f}\cdot\boldsymbol{v}\, d\Omega + \int_{\Gamma} \boldsymbol{F}_s\cdot\boldsymbol{v}\, d\Gamma + \sum_i \boldsymbol{F}_i \boldsymbol{v}_i$

where $\boldsymbol{f}$ are the volume forces

$\boldsymbol{F}_s$ the surface generalized forces

$\boldsymbol{F}_i$ the specific forces with the node i

If one notes $\boldsymbol{R}_K$ the vector of the nodal reactions on the element K, one has starting from the generalized nodal forces:

$$\boldsymbol{R}_K = \boldsymbol{F}_K - \int_K \boldsymbol{f}\, dK - \int_{\partial K} \boldsymbol{F}\, \partial K - \sum_i \boldsymbol{F}_i$$

in other words one cuts off with the nodal forces the external forces applied to the element K.

To note that the loading of temperature does not appear in the external forces.

In dynamics, to obtain the nodal reactions, it is advisable to in addition remove the forces of inertia (acceleration) and damping (velocity). Currently in *Code_Aster* the effects of damping on the nodal reactions are neglected.

For the result concepts of the mode_meca type (resulting from modal computations) the formula is:

$$\int_{\Omega} \sigma(\varepsilon(\boldsymbol{u})).\varepsilon(\boldsymbol{v}) d\Omega - \omega^2 \boldsymbol{M}\,\boldsymbol{u}$$

where $\boldsymbol{M}$ is the mass matrix

ω the own pulsation

$\boldsymbol{u}$ the field of displacement

Pour the result concepts of the dyna_trans type resulting from linear transient dynamic computations (DYNA_LINE_TRAN, or DYNA_TRAN_MODAL by the means of REST_GENE_PHYS), of dyna_harmo type resulting from harmonic computations (DYNA_LINE_HARM) or of evol_noli type resulting from computation nonlinear transient dynamics (DYNA_NON_LINE) the statement is:

$$\int_{\Omega} \sigma(\varepsilon(\boldsymbol{u})).\varepsilon(\boldsymbol{v})d\,\Omega + \boldsymbol{M}\,\ddot{\boldsymbol{u}} - \boldsymbol{L}(\boldsymbol{v})$$

where $\boldsymbol{M}$ is the mass matrix;

$\ddot{\boldsymbol{u}}$ the field of acceleration;

$\boldsymbol{L}$ the vector of the external forces applied.

Notice 1:

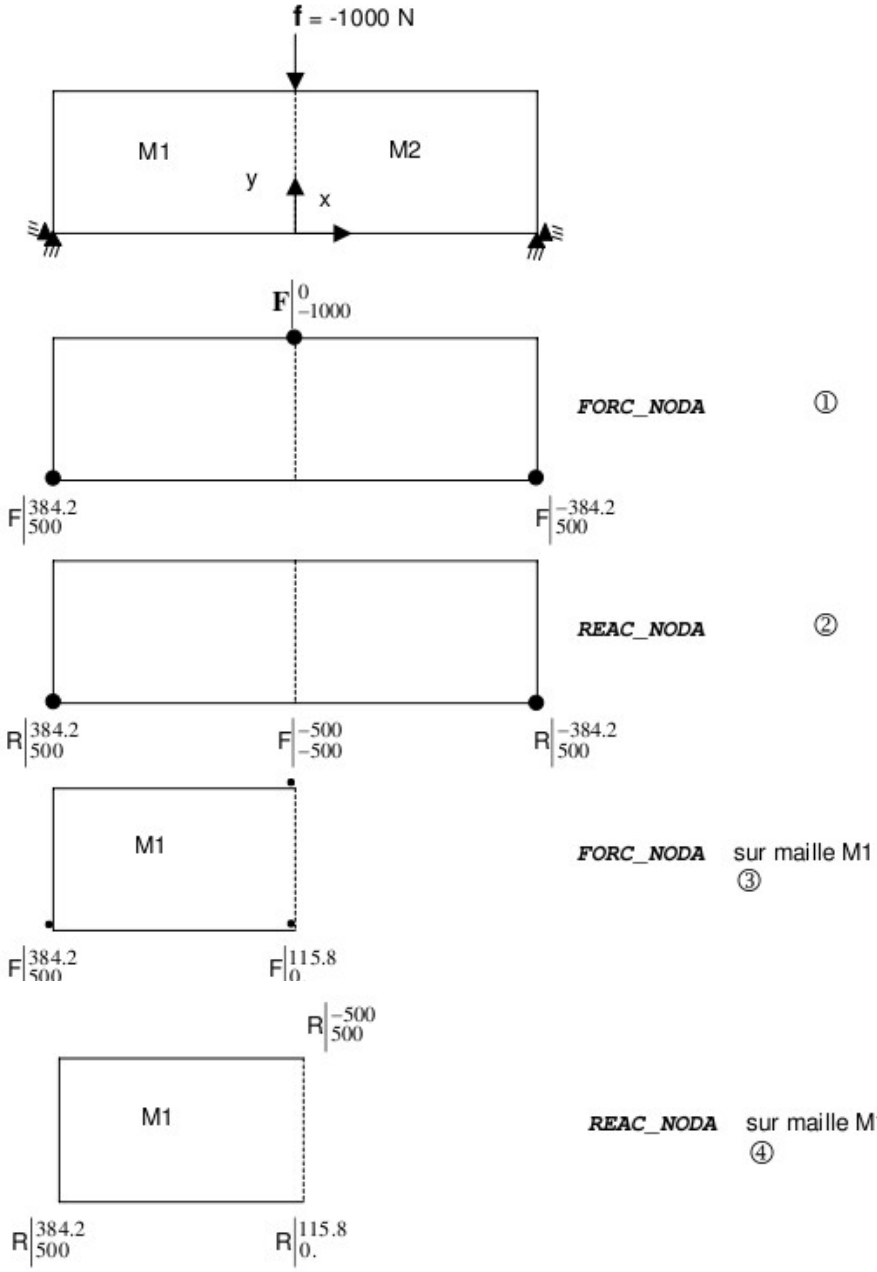
The nodal reactions are null in any interior point of the model and are not null a priori in a point of edge subjected to a kinematical boundary condition (or with forces of contact). However the fact of neglecting the contribution of damping in dynamics can create a light variation compared to the accurate result.

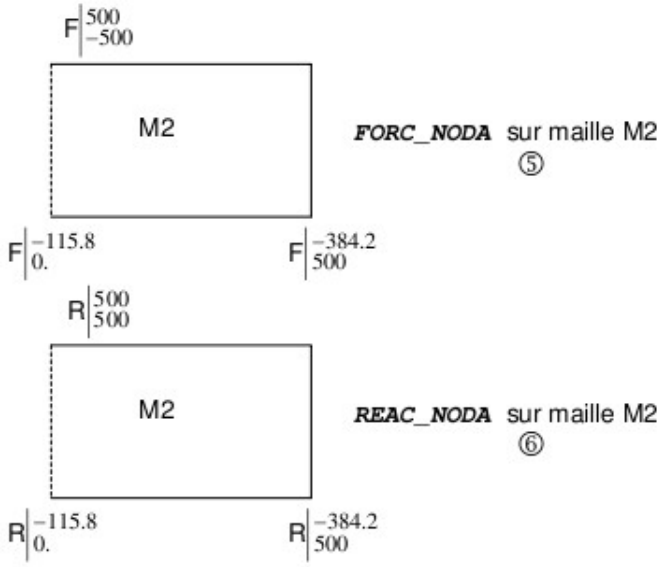
Notice 2:

If key word `GROUP_MA` *is indicated, options* `"FORC_NODA"` *and* `"REAC_NODA"` *are calculated as follows:*

F_K *is computed only on the elements requested then assembled. The result is different from a total computation on all the field then reduced to the required elements. The established method makes it possible to measure the reaction of a piece of model on another.*

2.9.2 Example 1: Structure loaded with nodal force (2 elements QUAD4)





Sur this example, the reactions to the nodes (2) are quite equal to the nodal forces (1) minus the loading. They represent the reactions to the bearings of structure.

If one restricts computation with the mesh $M1$, the forces (3) with the nodes belonging to the border enters $M1$ and $M2$ are different. They represent the reaction of the model formed by $M1$ with the model formed by $M2$. To note that the nodal loading is divided by two because the two meshes contribute to it. The nodal reactions (4) are still equal to the nodal forces minus the loading.

On the computation restricted with the mesh $M2$, the nodal forces (5) according to OX are of contrary sign to the computation restricted with the mesh $M1$, illustrating the principle of the action and the reaction.

2.9.3 Example 2: Structure with loading Données

temperature:

$E=1.10^{9}\,\mathrm{Pa}$
$\nu=0.3$
$\alpha=1.10^{-6}$

Results:

$F_{y}=-3.410^{4}\,N$
$F_{1x}=7.8\,10^{3}\,N$
$F_{2x}=-1.2\,10^{3}\,N$

On this example, the nodal forces and the nodal reactions coincide because the only loading is a loading temperature.

If one restricts computation with the mesh $M2$, the forces according to OY remain the same ones but are different according to OX.

2.10 Computation of a field user

the key word factor CHAM_UTIL allows of compute of the fields unspecified, known as “user” because of the name which will be affected for them in the result concept.
There can be several occurrences of CHAM_UTIL in order to connect the computation of several fields.
Processing being carried out at the end of command CALC_CHAMP, computed fields by the preceding key words (STRESS, DEFORMATION…) are available here.
Either one asks for the computation of a preset criterion, or one applies one or more formulas for compute another field.

2.10.1 Operand NOM_CHAM

It acts of the field from which computations are made. The produced field will have the same type: ELGA, ELNO or NOEU.

2.10.2 Operand CRITERE

Demande the computation of a preset criterion. The criteria are (paragraph 2.6.5 provides the statements of each criterion):

- VMIS (for the stress fields),
- INVA_2 (for the strain fields),
- TRACE (for the stress fields or of strains).

Each one of these criteria produces a component (named X1).
One of the interests is to be able compute INVA_2 of any strain field.

2.10.3 Operand FORMULATES

Ceci makes it possible to calculate any statement function of the components of the field provided for NOM_CHAM.
The produced field will contain as many components as of provided formulas: to the first formula will correspond the X1 component, with second X2, etc Jusqu'à 30 components can be thus created.

Examples of formulas making it possible to find criteria VMIS and INVA_2 can be found in the second part of the test sslv104a.

2.10.4 Opérande NUME_CHAM_RESU

the produced field must be arranged, in a single way, in the result concept. The fields “user” are thus numbered by using NUME_CHAM_RESU and the type of the field.
The name of the field will be thus of type UT01_ELGA, UT22_NOEU, etc

2.10.5 Exemple of computation of a field user

Produit field UT02_ELGA with two components. X1 is the trace of SIGM_ELGA (comparable with component TRSIG of SIEQ_ELGA) and X2 is the equivalent stress of Von Mises (component VMIS of SIEQ_ELGA).

```
fTrace = FORMULA (NOM_PARA= (“SIXX”, “SIYY”, “SIZZ”),
                  VALE= """ SIXX+SIYY+SIZZ """)

fVonMis = FORMULA (NOM_PARA= (“SIXX”, “SIYY”, “SIZZ”, “SIXY”, “SIXZ”,
“SIYZ”),
                   VALE= """ sqrt (3. /2. * (SIXX ** 2 + SIYY ** 2 + SIZZ **
2
                              + 2*SIXY ** 2 + 2*SIXZ ** 2 + 2*SIYZ ** 2)
                              - 1. /2. * fTrace (SIXX, SIYY, SIZZ) ** 2) """)
```

```
LMBO = CALC_CHAMP (reuse=RES,
                   RESULTAT=RES,
                   CHAM_UTIL=_F (NOM_CHAM='SIGM_ELGA',
                                 FORMULE= (fTrace, fVonMis),
                                 NUME_CHAM_RESU=2,),)
```

2.11 Opérande TITRATES

```
◊TITRE  = Titre
```

title that one wants to give to the result of the command [U4.02.01].