Operator CALC_CHAMP

1 Drank

To create or supplement result by calculating 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,…)
```

Contents

2 Syntax

```
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  = "FIXES",
                                  )
             ◊#Sélection       of meshes concerned with /TOUT
                computation   =      ' OUI',
[DEFAULT]
                /   |  GROUP_MA =l_grma    ,                [l_gr_maille]
                    |  NET   =l_mail    ,                [l_maille]

             ◊#Sélection       of the /TOUT_ORDRE
                sequence numbers  =      ' 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",                  [DEFAULT]
                                /"ABSOLU",
                |  accuracy =    prec,
                                /1.0E-6,                      [DEFAULT]
```

#options for mechanical results linear

```
             ♦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 criteria

```
◊CRITERES  =       |  ' 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 local variables

```
◊VARI_INTERN  =    |  ' 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 criteria

```
◊CRITERES  =          |   ' 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 local variables

```
◊VARI_INTERN  =       |   ' 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 flux (elements THM)

```
◊HYDRAULIQUE  =       |   ' Thermal
```

FLHN_ELGA' #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 forces and the generalized 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_formes   ,                  [formula]
                   ♦NUMERIQUE_CHAM_RESU=numeric        ,                 [I]

                   ),

                ◊TITER  = title  ,                                 [l_Kn]
                ◊INFO   =/1          ,                               [DEFAULT]
                             /2 ,
)
```

2.1 Operands RESULTAT/MODELE/CHAM_MATER/CARA_ELEM/EXCIT

2.1.1 Operands RESULTAT

♦RESULTAT =resu

Name of data structure result to enriching. 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 model, the material field, characteristics elementary, loadings. The key words MODELS , CHAM_MATER , CARA_ELEM and EXCIT are thus useless.

2.1.2 Operands MODELS/CHAM_MATER/CARA_ELEM.

◊MODELE = Mo

Name of the model on which the forces are calculated, the stresses, the strains, etc
It is optional because it can be extracted 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 modification of the material for example).

Characteristic ◊CARA_ELEM =

carac 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 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 : to 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 meshes concerned with computation

the key keys TOUT, GROUP_MA and MESH make it possible to the user meshes to choose on whom it wishes to do his elementary computations of postprocessing.

/TOUT all = "OUI'

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

/ | GROUP_MA=l_grma
| MAILLE=l_maille

Only 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 fields

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

the fields, for the majority, are calculated natively with Gauss points (*_ELGA).

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

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

Notice 1:

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

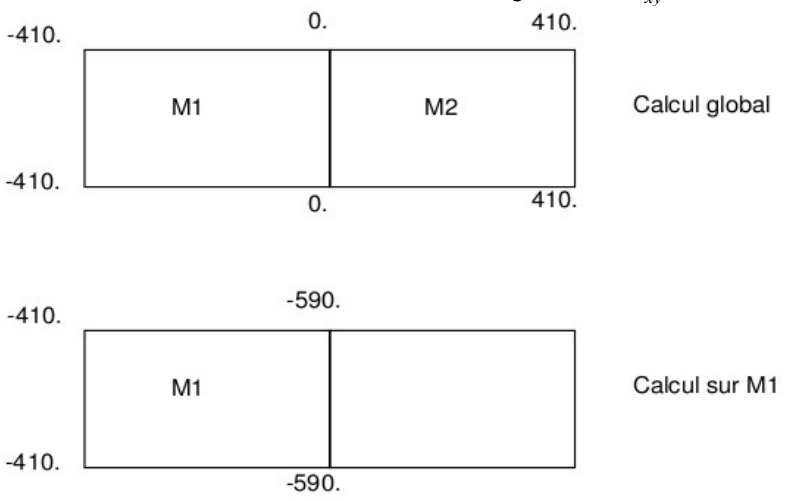
Notice 2:

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

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 type *_ELGA and *_ELNO are calculated 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 to compute: a field of the type *_NOEU for these structural elements: the fields of the type *_NOEU 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 to compute: a field *_NOEU it is necessary to have the same field by element for nodes *_ELNO and Gauss points the *_ELGA.

This dependence is solved by the operator CALC_CHAMP which 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 data structure result.

2.6 Operands for the mechanical options

2.6.1 Computation options of the stresses (FORCED Operand)

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

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

Computation of the generalized forces (structural elements).

It is either about an extraction of the forces contained in field SIEF_ELGA/STRX_ELGA (case of the beam elements/pipes or discrete), or 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 calculated directly starting from displacement. This is why certain components are not calculated (put at zero) into nonlinear.

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

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

Computation of the stress state (forced or forces generalized according to the modelization) starting from displacements (linear elasticity), to 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 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 field of the components:

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

the stresses above are expressed in the local coordinate system, i.e. the principal 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 calculated 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 word RY, RZ and RT *cf* AFFE_CARA_ELEM [U4.42.01]).

For a rectangular section:

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

For a circular section, one calculates 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 field of the components:

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
Value 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}

These fields are evaluated from a stress field calculated on the meshes voluminal ones (MODELISATION= `3D' or "3D_SI"):

- Identification of meshes voluminal corresponding to the surface facets of the mesh group;
- Recovery of the stresses 3D to assign them to the nodes sides;
- Average of each one of the components 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 facet and the plane of the facet. 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 principal corresponding to σ_{2D} : one thus obtains the vectors $\vec{\sigma}_{t1}$ and $\vec{\sigma}_{t2}$ which are in the plane of the facet and the eigenvalues SIG_{T1} and SIG_{T2}

Notices 1:

In the case of facets plunged in volume, one chooses the voluminal mesh which is side "-" compared to the normal of the facet. 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 facets is "outgoing".

Notice 2:

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

2.6.2 Computation options of the strains (Operand DEFORMATION)

the 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 calculation is done in theory of "**small displacements** ". The calculated 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 calculation is done in theory of "**large displacements** ". The calculated 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, the stresses σ, the temperature T, the possible unelastic strains ε^a, and the local 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 fluid stagnation pressure, b is the coefficient of Biot, K is the elasticity modulus.

2.6.3 Options of extraction of the local variables (Operand VARI_INTERNE)

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

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

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

List of the components possible 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
fluid	EVP cumulated Plastic strain
IND_ETA	mechanical Indicator of state
D	Value of damage
IND_END	Indicator of damage

TEMP_MAX	maximum Temperature
GAMP	plastic Strain critical déviatoire
cumulated	PCR Pressure
SEUIL_HYD	hydrous Threshold
IND_HYD	Indicating of hydrous irreversibility
PCOHE	Pressure of cohesion
COMP_ROC	Behavior of isotropic
rock	SEUIL_ISO Threshold
ANG_DEV	Angle of the threshold déviatoire
X11	Component of the tensor of kinematic hardening
X22	Component of the tensor of kinematic hardening
X33	Component of the tensor of kinematic hardening
X12	Component of the tensor of kinematic hardening
X13	Component of the tensor of kinematic hardening
X23	Component of the tensor of kinematic hardening
DIST_DEV	Outdistances standardized with the threshold déviatoire
DEV_SUR_CRIT	Relationship between the threshold déviatoire and the threshold deviatoric criticism
DIST_ISO	Outdistances standardized with isotropic threshold
NB_ITER	Nombre of iterations interns
ARRET	Value of the local test of stop of iterative process
NB_REDE	Number of local recutting of time step
the SIGNE	Signs contracted product of the deviatoric stress by deviatoric plastic strain

◊NOM_VARI =/nom_vari , [TXM]

Name of the local variable.

Notice 1:

When the variable to be extracted is not part of the local 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 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 field of the components:

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

Note:

All the components are systematically calculated. 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 local variables.

List field of the components:

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

the number and the type of these local 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 evol_noli type.

2.6.4 Computation options of energy (Operand ENERGIE)

| ' DISS_ELEM'

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

List field of the components:

ENDO	Energy dissipated by the damage

Note:

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 field of the components:

ENDO	Energy dissipated by the damage

Note:

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 field of the components:

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

| ' ENEL_ELEM'

Computation of elastic strain energy.

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

List field of the components:

TOTAL	Elastic strain energy
additional Components for the plates and shells: Contributions to elastic strain energy (cf [R3.07.03])	
MEMBRANE	Elastic strain energy out of membrane
BENDING	Elastic strain energy in bending
SHEARS	Elastic strain energy in shears
COUPL_MF	Elastic strain energy of membrane-flexure coupling

Note:

Into nonlinear (STAT_NON_LINE, DYNA_NON_LINE,...) the components SHEARS and COUPL_MF are null.

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

Computation of the density of elastic strain energy.

List field of the components:

TOTAL	Elastic strain energy
additional Components for the plates and shells: Contributions to elastic strain energy (cf [R3.07.03])	
MEMBRANE	Elastic strain energy out of membrane
BENDING	Elastic strain energy in bending
SHEARS	Elastic strain energy in shears
COUPL_MF	Elastic strain energy of membrane-flexure coupling

Note:

Into nonlinear (STAT_NON_LINE, DYNA_NON_LINE,...) the components SHEARS and COUPL_MF are null.

| ' EPOT_ELEM "

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

List field of the components:

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

- for the elements of continuums 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 D" strain energy total enters time running and previous time.

List field of the components:

TOTAL	Increment 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 field of the components:

TOTAL Increment of total strain energy

2.6.5 Computation options of criteria (Operand CRITERES)

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

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

List field of the components:

Indicating	DCHA_V of discharge calculated on the deviative tensor of Indicating
stresses	DCHA_T of discharge calculated on the total tensor of Indicating
stresses	IND_DCHA making it possible to know if the discharge would remain elastic or if there would be a risk of plasticization if a pure kinematic hardening VAL_DCHA
were used	X11 Indicates the proportion of output of the criterion in the case of abusive
discharge X22 X33 X12 X13 X23	Component of the kinematical tensor used for the computation of Indicating
IND_DCHA	RADI_V of the variation of the direction of the stresses between times t and $t+\Delta t$
ERR_RADI	Error 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 time step (urgent t^-), and $\boldsymbol{n}^+$, the norm with the plasticity criterion calculated at the end of 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 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}. Result is arranged at the sequence number associated with time t_i.

The indicator of discharge is calculated 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 calculate 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 field of the components:

TRIAX	Rate of triaxiality
SI_ENDO	Equivalent stress of damage of Forced
Lemaître-Sermage	COENDO of damage of Lemaître-Sermage standardized
DOM_LEM	Damage 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 is 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 classical model of Lemaître).

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

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

List field of the components:

INVA_2	equivalent Strain of Von Mises
PRIN_1 PRIN_2 PRIN_3	Principal strains, arranged in the order ascending
INVA_2SG	equivalent Strain 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:

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

Note::

It is noted that the equivalent strains obtained 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 field of the components:

Indicating	INDEX of localization 0 if $det(N.H.N)>0$ 1 if not, which corresponds has the initiation of localization
DIR1	First norm at the zone of localization
DIR2	Second norm at the zone of localization
DIR3	Third norm at the zone of localization
DIR4	Fourth norm at the zone of localization

This indicator 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 stiffness of microphone-thermal expansion.

List field of the components:

A1_LC2	Modulates stiffness of microphone-thermal expansion

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

The computation from A1_LC2 is obtained via the evaluating 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'

Calculates equivalent stresses calculated starting from the stress fields.

List field of the components:

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	Traces σ
TRIAX	Rate 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 flux (Operand HYDRAULIQUE)

| ' FLHN_ELGA'

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

List field of the components:

FH11	
FH22	
FH12	
FH21	

2.7 Operands for the thermal options

2.7.1 THERMAL Operand

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

Computation of heat flux from the temperature.

List field of the components:

FLUX FLUY FLUZ	Heat flux in the three directions of space (in the average average 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 field of the components:

HYDR	Hydration

Note:

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

| ' SOUR_ELGA'

Computation of a heat source.

List field of the components:

SOUR	Heat source

This source is calculated from an electric potential via the model of Ohm. This electric potential must by the operator be calculated 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 the command AFFE_CHAR_THER [U4.44.02].

| ' ETHE_ELEM'

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

List field of the components:

TOTAL	thermal Energy

2.8 Operands for the acoustic options

2.8.1 ACOUSTIC Operand

| ' INTE_ELNO'
| ' INTE_NOEU'

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

List field of the components:

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

List field of the components:

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 elements FLUIDE.

List field of the components:

Acoustic	dB Pressure in decibel

2.9 Operand for the forces and the nodal reactions

2.9.1 Operand FORCE

| ' FORC_NODA'

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

List field of the components:

DX DY DZ	additional Component
nodal Forces for the structural elements:	
DRX DRY DRZ	nodal Forces

The computation is 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 with 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 with 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 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. The 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

For the elements in plane strain, is calculated on a tape of width unit. The calculated 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 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 result approximate.

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 calculated 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, the 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 flux. If one notes $\mathbf{Q}^T\boldsymbol{\sigma}_0$ result FORC_NODA, for the hydraulic equations, then for one time step Δt, one a:

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

In FORC_NODA :

- with degree of freedom PRE1 is associated the water flux $-\Delta t\int_\Omega \boldsymbol{M_{vp}}+\boldsymbol{M_w}^-\nabla\,p^*\,d\,\Omega$
- with degree of freedom PRE2 is associated the flux 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 heat flux and M_w M_{vp}, M_{as} and the M_{ad} hydraulic flux of liquid water, the vapor, the air (or very other composing) dry and of the air dissolved in the fluid. 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 with Gauss points.

List field of the components:

DX DY DZ	additional Component
nodal Forces for the structural elements:	
DRX DRY DRZ	nodal Forces

For the result concepts of the evol_elas `type`, `mult_elas`, `fourier_elas` or `evol_noli`, this computation is 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

For 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:

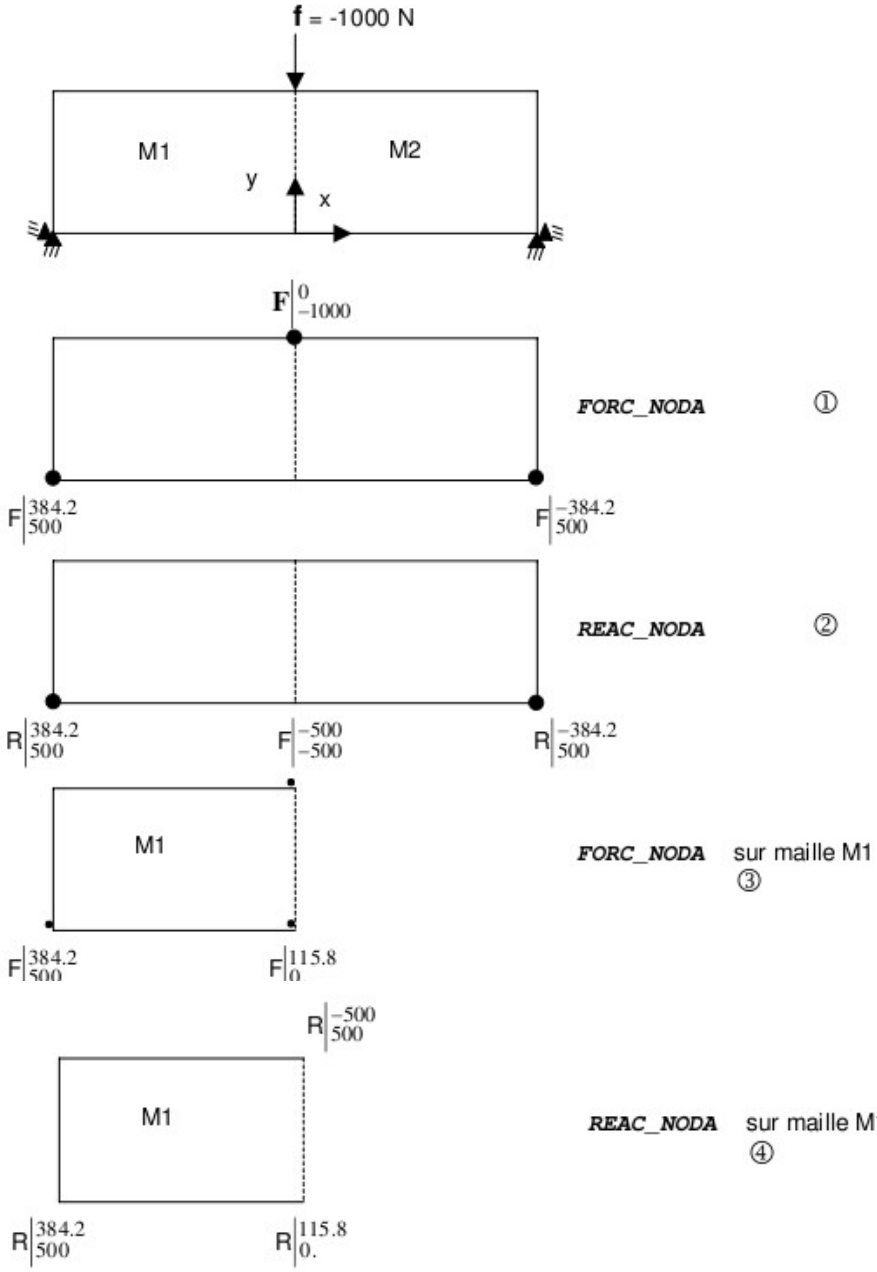
The nodal reactions are null in any interior point of the model and are not null a priori in a point of edge subjected to a kinematical boundary condition (or with forces of contact). However the fact of neglecting the contribution of damping in dynamics can result create a light variation compared to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

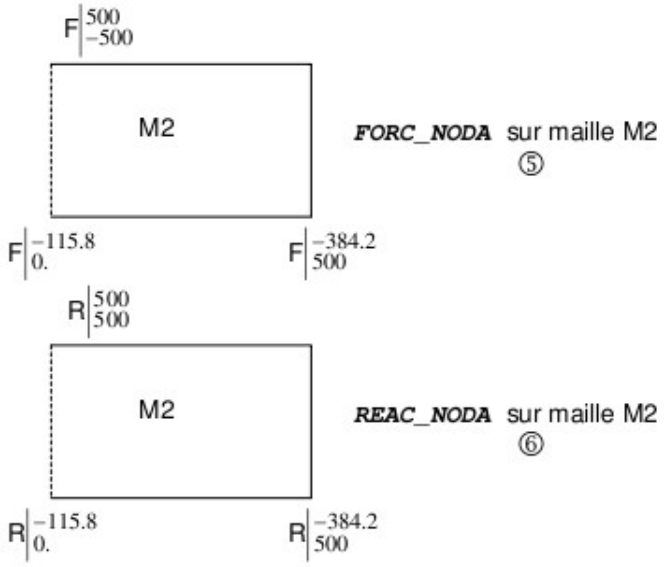
Notice 2:

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

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

2.9.2 Example 1: structure charged with nodal force (2 elements QUAD4)





On 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 both 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 of temperature

Given:

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

Results:

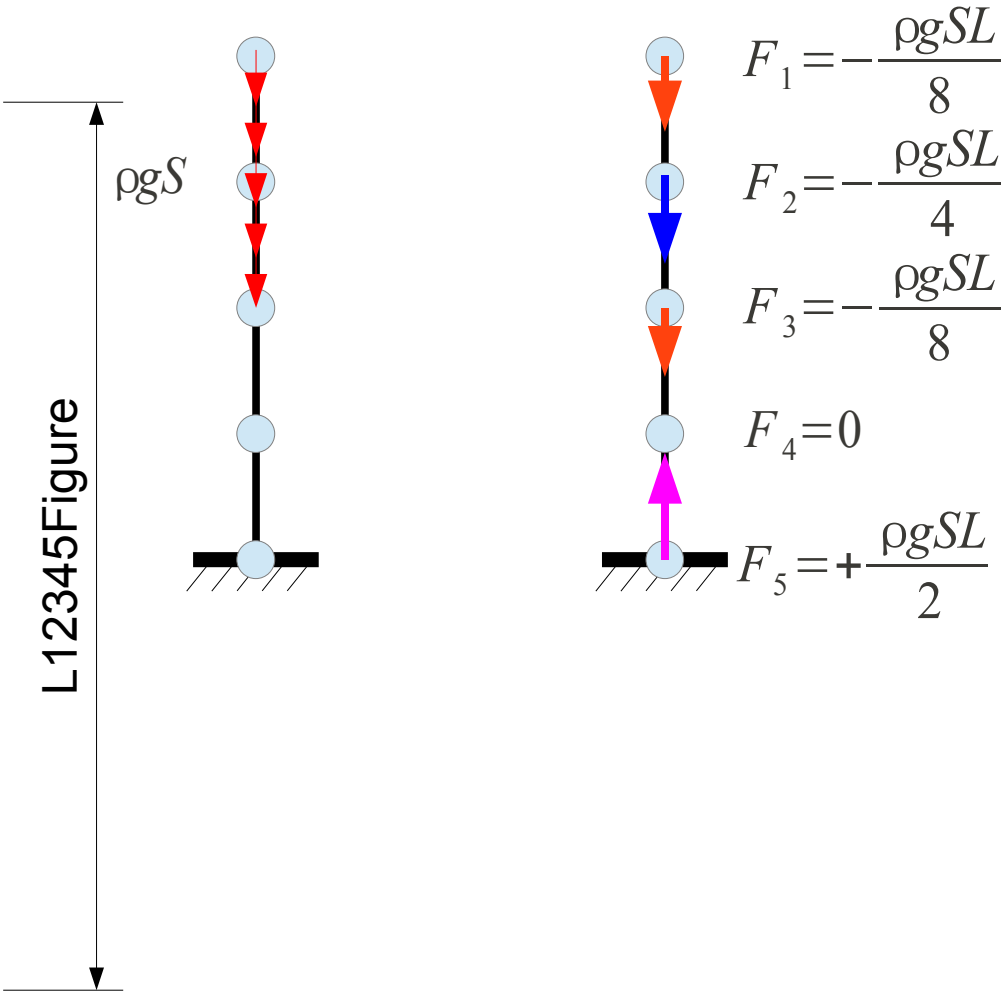
$F_{y}=-3.410^{4}\,N$
$F_{1x}=7.8\,10^{3}\,N$
$F_{2x}=-1.2\,10^{3}N$

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.

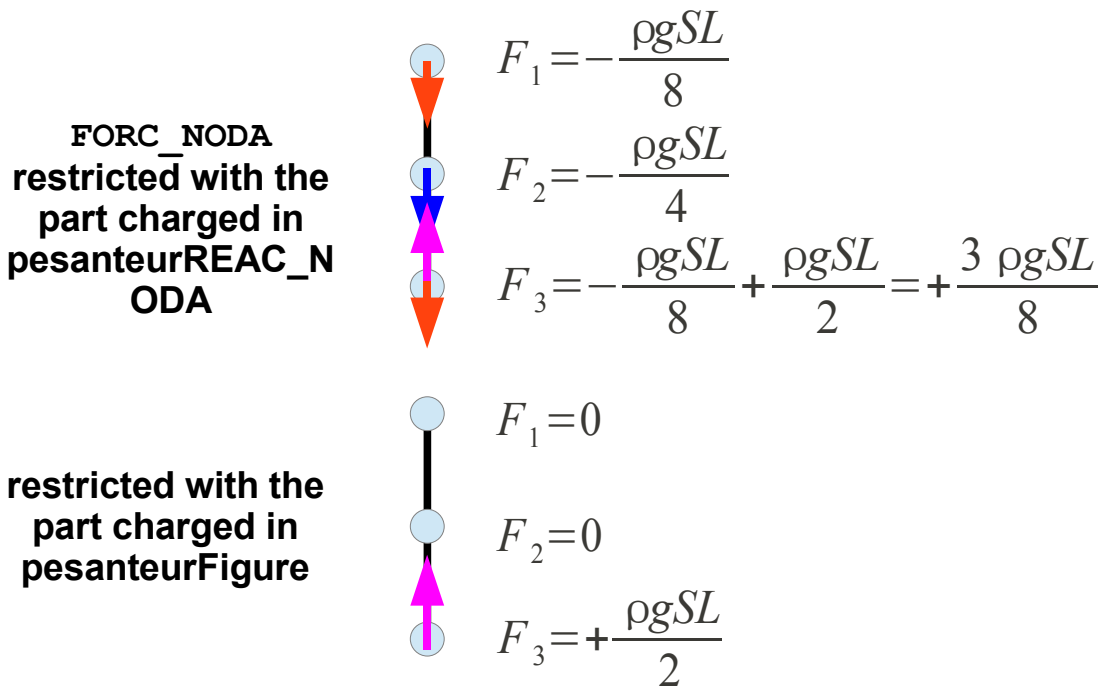
2.9.4 Example 3: structure under distributed loading (beam)

One considers a structure of type beam clamped and subjected to a loading of gravity on its higher half.



2.9.4-1: structure under distributed loading (left), FORC_NODA (right)

On this kind of structure, if one restrict the computation of the forces and the nodal reactions to a under-part of the elements, FORC_NODA and REAC_NODA will not result give the same one on the interface between the part isolated and the rest from structure as shown in the figure 2.9.4-2 (on the force F_3).



2.9.4-2: insulation of part of the structure

2.10 Computation of a field user

The factor key word CHAM_UTIL makes it possible to calculate 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 to compute: another field.

2.10.1 Operand NOM_CHAM

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

2.10.2 Operand CRITERE

Asks for 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 to calculate INVA_2 of any strain field.

2.10.3 Operand FORMULATES

This makes it possible to calculate any statement function of the components field provided for NOM_CHAM.
The produced field will contain as many components as of provided formulas: to the first formula the component X1 will correspond, with the second X2, etc To 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 Operand NUME_CHAM_RESU

the produced field must be arranged, in a single way, in the result concept. The fields “user” are thus numbered by means of 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 Example of computation of a field user

Produces field UT02_ELGA with two components. X1 is the trace of SIGM_ELGA (comparable to 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 Operand TITER

```
◊TITER = title
```

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