User Manual
Booklet U4.5- : Solution methods
Document : U4.51.01

Operator MECA_STATIQUE

1 Aim

To solve a problem in linear static mechanics.

This operator permits the solution of:

- a linear static mechanics problem with the superposition of different boundary conditions and loads,
- a thermo-mechanical analysis at specified time steps
  - in this case the mechanical properties of the materials may be temperature dependant : the data type cham_mater must be defined using functions (cf. operator DEFI_MATERIAU [U4.43.01] operand ELAS_FO),
  - dilational loading can be specified only if the dilation coefficient and the reference temperature have been defined (cf. operators DEFI_ MATERIAU [U4.43.01] and AFFE_MATERIAU [U4.43.03]).

The data type evol_elast, produced by this operator, contains one or more displacement fields at different calculation times.

In the case of static mechanical analysis, the sequence number 0 (time 0) is assigned to the solution field.

To produce a structure of data type evol_elast.

When a parametric sensitivity analysis is required for the result, the number of evol_elast structures produced equals the number of defined sensitivity parameters.

2 Syntax

```
mestat [evol_elast] = MECA_STATIQUE

             (  ♦  MODELE =            mo  ,                [modele]
                ♦  |  CHAM_MATER =     chmat  ,             [cham_mater]
                   |  CARA_ELEM =      carac  ,             [cara_elem]
                ♦  EXCIT =(_F(  ♦  CHARGE = char,      /    [char_meca]
                                                       /    [char_cine_meca]
                               ◊  FONC_MULT= fmult ,   /    [function]
                                                       /    [formula]
                             ),)
                ◊  /  INST =            /  tps,              [R]
                                        /  0. ,              [Default]
                   /  LIST_INST =       /  litps  ,          [listr8]

                ◊  SOLVEUR = ( ... see [U4.50.01]  ),

                ◊  SENSIBILITE =(   ... see [U4.50.02] ),

                ◊  INFO =    /  1,                           [Default]
                             /  2,

                ◊  TITRE =   titre,                         [l_K80]
             );
```

3 Operands

3.1 Operands MODELE / CHAM_MATER / CARA_ELEM

The arguments for calculating the stiffness matrix (and the second member) are provided as follows:

♦ MODELE = mo,

The name of the model whose elements are the subject of the mechanical calculation.

♦ CHAM_MATER = chmat,

The name of the material field.

◊ CARA_ELEM = carac,

The property names of the structural elements (e.g. beam, shell, mass...) used in the model.

3.2 Keyword EXCIT and operands INST / LIST_INST

The boundary conditions and the loads are defined here.

♦ EXCIT =

This key word factor allows the definition of several *charge* data types, one per case; the solution is calculated by superimposing the effects of the different applied loads.

3.2.1 Operands CHARGE / FONC_MULT

♦ CHARGE = char,

The name of a *char_meca* data type produced by AFFE_CHAR_MECA or AFFE_CHAR_MECA_F [U4.44.01] from the model *mo*. Only one case can make reference to the temperature (*charge* with TEMP_CALCULEE).

The name of a "kinematic load" (type *char_cine_meca*) resulting from operators AFFE_CHAR_CINE and AFFE_CHAR_CINE_ F can also be provided.

◊ FONC_MULT = fmult,

The name of a function (or formula) data type which allows the definition of a multiplying coefficient to be applied to the load *char* at each time step. The temperature field is not multiplied by fmult for a root thermal load (dilation) defined by TEMP_CALCULEE in the command AFFE_CHAR_MECA [U4.44.01],

fmult is a function of time: by default it is a constant function equal to 1.

3.2.2 Operands INST / LIST_INST

◊ / INST = tps,

The key word used to carry out the calculation at only one time step with the temperature corresponding to that instant.

/ LIST_INST = litps,

The list litps produced by DEFI_LIST_REEL [U4.34.01] defines all the time steps for which the calculation of a thermo-mechanical increment is requested.

3.3 Keyword factor SOLVEUR

See [U4.50.01].

3.4 Keyword SENSIBILITE

Activates the derived calculations of the displacement field with respect to a problem parameter.. See [U4.50.02].

3.5 Operand INFO

◊ INFO = 1,

Prints the main characteristics of the system of linear equations to be solved : number of unknowns, size of the matrix.

3.6 Operand TITRE

◊ TITRE = titr,

Title for the result [U4.03.01].

4 Example calculations

4.1 Static calculation with superposition of 2 load cases

```
mest1 = MECA_STATIQUE (    MODELE = mo,  CHAM_MATER = chmat,
                  CARA_ELEM = carac,
                  EXCIT = ( _F( CHARGE = ch1 , FONC_MULT = COS ),
                            _F( CHARGE : ch2 ),),  )
```

4.2 Thermo-elastic calculation at different time steps

```
ch_temp = AFFE_CHAR_MECA (    ... TEMP_CALCULEE = evoth ...) );

mest2 = MECA_STATIQUE (    MODELE = mo , CHAM_MATER = chmat ,
                           EXCIT = (_F( CHARGE = ch_temp ),
                                    _F( CHARGE = bloq ),),
                           LIST_INST = litps )
```

4.3 Sensitivity to an imposed displacement

```
psx= DEFI_PARA_SENSI(VALE=7.0)
psy= DEFI_PARA_SENSI(VALE=3.0)

ch=AFFE_CHAR_MECA_F( MODELE=mo,
       FACE_IMPO=_F(GROUP_MA='BORD_SUP', DX=psx, DY=psy))


mest3 = MECA_STATIQUE (    MODELE = mo , CHAM_MATER = chmat ,
                           EXCIT = _F( CHARGE = ch ),
                           SENSIBILITE=(psx,psy),)
```

This calculation produces the data structure mest3 of type evol_elast, containing the displacement field named 'DEPL'. It produces two other data structures of the type evol_elast. The first one contains the derivative with respect to the parameter psx, with the field name 'DEPL'. The second one contains the derivative with respect to the parameter psy.
The names of these 2 structures are created automatically by the code and remain unknown to the user. Access to their contents (printing, testing, post-processing) is achieved by calling the corresponding command with the name of the principal structure, mest3, and the name of the significant parameter concerned (psx or psy).

5 Remarks

For certain cases in linear elasticity, in which the structural stiffness properties are independent of the thermal history and the other cinematic boundary conditions, the deformations for several loads can be determined using MACRO_ELAS_MULT [U4.51.02].

Page intentionally left blank