Operator CALC_BT

Summary:

Order CALC_BT allows to create in an automatic way and ofto optimize a structure Rod-Ties with to leave the results of the type `evol_elas` or `evol_noli` resulting from a modeling 2D in constraint or deformation planes (`C_PLAN, D_PLAN`) . The goal is the calculation of the reinforcement of a concrete structure.

For more details on the method one can consultR documentation [R7.04.06] " Data processing method of the rod-ties for the calculation of reinforcement.

Contents

1 Syntax

```
CALC_BT (

    # Results of basic calculation and geometry
    ♦ RESULT  =  RESULT,                          [evol_elas, evol_noli]
    ♦ INST      = inst,                                                [R]

    # interior and external Contour
    ♦  GROUP_MA_EXT = gr_ma_ext                                        [TXM]
    ◊  GROUP_MA_INT = gr_ma_int                                        [l_TXM]

    # Boundary conditions
    ♦  DDL_IMPO = _F (  ♦  GROUP_NO = gr_nod,                          [TXM]
                        ♦   |  DX = dx,                                [R]
                            |  DY = Dy,                                [R]
                    ),

    ♦  FORCE_NODALE = _F (♦  GROUP_NO = gr_nof,                           [TXM]
                          ♦  | FX = fx,                                [R]
                             | FY = fy,                                [R]
                        ),

    # Materials
    ♦ CONCRETE = CONCRETE
       [mater_sdaster]

    ♦ STEEL = STEEL                                       [mater_sdaster]

    # Parameters of optimization
    ◊  DIAGRAM =/'SECTION',                                            [DEFECT]
               /'REPORT',
    ♦  SIGMA_C =     FC,                                               [R]

    ♦  SIGMA_Y =     fy,                                               [R]

    ◊  PAS_X = pas_x,                                                  [R]

    ◊  PAS_Y = pas_y,                                                  [R]

    ◊  TOLE_BASE =/0.01,                                               [DEFECT]
                  / tole_1,                                            [R]

    ◊  TOLE_BT =/0.01,                                              [DEFECT]
                / tole_2,                                              [R]

    ◊  NMAX_ITER =/150,                                             [DEFECT]
                  / maxiter,                                           [R]

    ◊  RESI_RELA_SECTION=/1E-5,                                        [DEFECT]
                         / conv_1,                                     [R]

    ◊  RESI_RELA_TOPO=/1E-6,                                           [DEFECT]
                      / conv_2,                                        [R]

    ◊  CRIT_SECTION =/0.5,                                          [DEFECT]
                     / sevol,                                          [R]
```

```
◊  CRIT_ELIM =/0.1,                                      [DEFECT]
             / melim,                                        [R]


♦  SECTION_MINI = minsec,                                    [R]

♦  LONGUEUR_MAX = maxlon,                                    [R]
◊ INIT_ALEA = seed                                           [I]
                   ),
)
```

2 Operands

2.1 Operand RESULT

♦ RESULT = RESULT

Name of the field evol_elas or evol_noli resulting from the basic calculation (on the geometry for which one wants to calculate the reinforcement).

2.2 OpéRand INST

♦ INST = inst

Reality specifying the moment over which the restitution takes place.

2.3 Operands CONCRETE and STEEL

♦ CONCRETE = CONCRETE

Structure of data of the type mater_sdaster defining them properties rubber bands material concrete (Young Modulus, E , and Poisson's ratio, ν), as defined by DEFI_MATERIAU/ELAS.

♦ STEEL = STEEL

Structure of data of the type mater_sdaster defining the characteristics of the material steel (Young Modulus, E , and Poisson's ratio, ν), as defined by DEFI_MATERIAU/ELAS.

2.4 Operand GROUP_MA_EXT

♦ GROUP_MA_EXT =/gr_ma_ext

Name of the group of meshs of the type seg2 defining contour external of the geometry. The connectivity of these groups must define closed geometries.

2.5 Operand GROUP_MA_EXT

◊ GROUP_MA_INT =/gr_ma_int

List of groups of meshs of the type seg2 defining internal contours of the geometry. The connectivity of these groups must define closed geometries.

2.6 Operand DIAGRAM

♦ DIAGRAM =/'SECTION',
/'REPORT',

One of the two possible diagrams of optimization must be selected by the user. In the absence of the operand, the diagram of optimization 'SECTION' by default will be selected.

2.7 Operands SIGMA_C and SIGMA_Y

♦ SIGMA_C = FC,

♦ SIGMA_Y = fy,

Limiting values of resistance considered in optimization. SIGMA_C, resistance represents with compression of the concrete and SIGMA_Y the resistance of steel in traction.

2.8 Operands PAS_X and PAS_Y

◊ PAS_X = pas_x,

◊ PAS_Y = pas_y,

The user can define the step of mesh usedE for the interpolation of the values in the stress fields. Two values are necessary: a value for the step according to direction X and one for the step according to direction Y.

2.9 Operand TOLE_BASE

```
◊ TOLE_BASE =/0.1,
            / tole_1,
```

The value of tolerance TOLE_BASE allows to define the distance from fusion between the peaks of constraint during the interpolation of the stress fields, on the basic model, i.e. on the result as starter the macro one (geometry for which one wants to calculate the reinforcements). The provided value represents a percentage of the longest distance in X, there or Z between the nodes of the basic grid.

2.10 Operand TOLE_BT

```
◊ TOLE_BT =/0.1,
          / tole_2,
```

The value of tolerance TOLE_BT allows to define the distance from fusion between two or several nodes of the model lattice. The provided value represents a percentage of the longest distance in X, there or Z between the nodes of the basic grid.

2.11 Operand MAX_ITER

```
◊ NMAX_ITER =/150,
            / maxiter,
```

This keyword makes it possible to the user to define the maximum number of iterations carried out by the selected diagram of optimization.

2.12 OperandS RESI_RELA_TOPO and RESI_RELA_SECTION

```
◊ RESI_RELA_SECTION =/1E-6,
                    / conv_1,

◊ RESI_RELA_TOPO =/1E-5,
                 / conv_2,
```

Precision of convergence of the procedure of optimization perhaps defined by the user. The value of convergence RESI_RELA_SECTION control the stop of the procedure of optimization, that it maybe of type SECTION or REPORT. The value RESI_RELA_TOPO start the topological procedure of optimization and is used only for the diagram REPORT.

For an optimization carried out exclusively with the option DIAGRAM = 'SECTION', the paramètrE RESI_RELA_TOPO is ignored.

2.13 Operand CRIT_SECTION

```
◊  CRIT_SECTION =/0.5,
                 / sevol
```

The operand CRIT_SECTION allows to define the maximum evolution of the sections of the elements at the time of the algorithm of optimization. A value of 0.5, equivalent to an acceptable maximum evolution of ±50%, is imposed by default.

2.14 Operand SECTION_MINI

```
♦  SECTION_MINI = /minsec,
```

The operand SECTION_MINI allows the user to define the minimal section for the elements whose thrust load tends towards zero. This parameter is used at the same time for the diagram SECTION and REPORT.

2.15 Operand CRIT_ELIM

```
◊  CRIT_ELIM =/0.1,
              / melim,
```

The operand CRIT_ELIM allows to define the maximum rate of elimination of the elements. This value represents a percentage of the totality of the elements in the initial system. This parameter is used only for the diagram REPORT and is ignored if the diagram SECTION is selected.

2.16 Operand LONGUEUR_MAX

```
♦  LONGUEUR_MAX = maxlon,
```

The value LONGUEUR_MAX allows to limit the generation of elements in the BT model. The elements, rods or ties, have like maximum length the value given .

2.17 Operand INIT_ALEA

```
◊  INIT_ALEA = seed,
```

The keyword INIT_ALEA initialize the germ of the random continuations used for a random pulling. If this operand is indicated, two calculations of reinforcement with same initialization produce the same result then.

2.18 Keyword factor DDL_IMPO

```
♦   DDL_IMPO = _F (♦  GROUP_NO = gr_nod,
                 ♦   |  DX = dx,
                     |  DY = Dy,
                  )
```

This keyword factor makes it possible to apply displacements imposed to a group of nodes. It is responsibility for the user to inform the same boundary conditions usedbe for calculation on the starting geometry (basic calculation). Syntax is the same one as for the operator `AFFE_CHAR_MECA` [U4.44.01].

2.18.1 Operand GROUP_NO

♦ GROUP_NO = gr_no

Group of nodes associated with the boundary conditions.

2.18.2 Operands DX and DY

```
♦   |  DX = dx,
    |  DY = Dy,
```

Values of the nodal displacements imposed defined in the reference mark total of definition of the maillagE.

2.19 Keyword factor FORCE_NODALE

```
♦   FORCE_NODALE = _F (♦  GROUP_NO = gr_nof,
                     ♦   |    FX = fx,
                         |    FY = fy,
                    )
```

Keyword factor usable to apply, with groups of nodes, nodal forces, definite component by component in the total reference mark. It is responsibility for the user to inform the same boundary conditions used for calculation about the starting geometry (basic calculation). Syntax is the same one as for the operator `AFFE_CHAR_MECA` [U4.44.01].

2.19.1 Operand GROUP_NO

```
♦  GROUP_NO = gr_no
```

Group of nodes associated with the conditions with the nodal loadings.

2.19.2 Operands FX and FY

```
♦   |  FX = fx,
    |  FY = fy,
```

Values of the nodal forces imposed defined in the reference mark total of definition of the grid.

3 Example of use

This section provides the description of the principal stages of a calculation of reinforcement, for modeling in plane constraint.

For other examples, vto oir CAS-tests ssnp105 [V6.03.105] and ssnp106 [V6.03.105].

3.1 Before using the macro-order

It is nécEssaire of to carry out a linear elastic design with a modeling planes (`D_PLAN, C_PLAN`).

The totality of the elements is associated with the material concrete.

3.1.1 Grid and model

```
E-MAIL = LIRE_MAILLAGE (FORMAT = 'ASTER',
                       INFORMATION = 2,
                       UNIT = 20)

E-MAIL = DEFI_GROUP (reuse = E-MAIL,
                  CREA_GROUP_MA = _F (NAME = 'all',
                                     TOUT=' OUI'),
                  MAILLAGE=E-MAIL)

MODEL = AFFE_MODELE (AFFE = _F (GROUP_MA = 'all',
                               MODELING = ('C_PLAN',),
                               PHENOMENON = 'MECHANICAL'),
                      GRID = E-MAIL)
```

3.1.2 Mytériaux

The order `CALC_BT` find a lattice made up of concrete rods and steel ties (reinforcements). The two materials are here defined. However, for basic calculation one uses only material concrete, the structure to be reinforced being considered homogeneous.

```
CONCRETE = DEFI_MATERIAU (ELAS = _F (E = 30E9,
                                    NAKED = 0.2))

STEEL = DEFI_MATERIAU (ELAS = _F (
                                 E = 210E9,
                                 NAKED = 0.3))

materi = AFFE_MATERIAU (AFFE = _F (MATER = (CONCRETE,),
                                  ALL = 'YES'),
                         MODEL = MODEL)
```

3.1.3 Boundary conditions

Boundary conditions static are defined in the structure with the nodes.

These same boundary conditions will be then used in the calculation of the rods and ties.

```
bc = AFFE_CHAR_MECA (DDL_IMPO = (_F ( GROUP_NO = ('SUP_1',),
                                    CONNECTION = 'EMBEDS'),
                                 _F (DY = 0.0,
                                    GROUP_NO = ('SUP_2',))),
                      FORCE_NODALE = (_F (FY = -300000.0,
                                        GROUP_NO = 'LOAD_1'),
                                    ),
```

```
MODELE=MODEL)
```

3.1.4 Calculation to préliminairE

It is a question here of carrying out the mechanical calculation which will provide the starting state for BT optimization, starting from the loadings and blockings defined before.

For the moment, the only boundary conditions which it is possible to apply are nodal forces and imposed degrees of freedom.

```
RESU = MECA_STATIQUE (CHAM_MATER = materi,
                        EXCIT = _F (LOAD = bc,
                                    TYPE_CHARGE = 'FIXE_CSTE'),
                        MODEL = MODEL,
                        TITLE = 'SYSTEM')
```

3.2 Use of the macro-order

Like already mentioned, in the keywords DDL_IMPO and FORCE_NODALE, groups of nodes and forces/ displacements must be exactly the same ones as those used for calculation basic.

The material CONCRETE must be the same one used for basic calculation. The material STEEL is used only in the body of the macro-order.

```
STM_TOPO = CALC_BT (RESULT = RESU,
              RESU_BT = CO ('RES_TOPO'),
              INST = 1. ,
              DDL_IMPO= (_F (GROUP_NO = 'SUP_1',
                             DX = 0,
                             DY = 0),
                    _F (GROUP_NO= 'SUP_2',
                             DY = 0)
                  ),
              FORCE_NODALE=_F (
                        GROUP_NO = 'LOAD_1',
                        FY=-3000000.0
                         ),
              CONCRETE = 'CONCRETE',
              STEEL = 'STEEL',
              GROUP_MA_EXT = 'SKIN1',
              GROUP_MA_INT = 'SKIN2',
              DIAGRAM = 'REPORT',
              SIGMA_C = 35000000,
              SIGMA_Y = 500000000,
              PAS_X = 0.05,
              PAS_Y = 0.05,
              TOLE_BASE = 0,026,
              NMAX_ITER = 200,
              RESI_RELA_TOPO = 0.00001,
              RESI_RELA_SECTION = 0.000001,
              CRIT_SECTION = 0.5,
              CRIT_ELIM = 0.5,
              SECTION_MINI = 1e-6,
              LONGUEUR_MAX = 7,
              INIT_ALEA = 0,
              )
```