POST_T_Q operator

1 Goal

To compute T-stress in 2D and 3D models.

This operator allows computation of T-stress in 2D (modelling 'D_PLAN', 'C_PLAN' and 'AXIS') and 3D ('3D') by extrapolation of displacement on the crack lips, cf [R7.02.08]. This method is appropriate only for homogeneous and isotropic materials.

This operator is only available for a meshed plane crack (method FEM).

Product a concept of type table_sd_aster .

2 Syntax

```
tk [table_sdaster] = POST_T_Q (

      ♦ MODELISATION =       / '3D',
                             / 'AXIS',
                             / 'D_PLAN',
                             / 'C_PLAN',

      ♦   RESULTAT=          =  resu,                  /  [evol_elas]
                                                       /  [evol_noli]
                                                       /  [mode_meca]

       ◊  NB_NOEUD_COUPE    =/  5 ,
                             /  N,                        [I]

# 1) FOND_FISS is indicated for crack
       ♦ /  FOND_FISS       =found ,                     [fond_fiss]

      ◊  ABSC_CURV_MAXI      =dmax,                      [R]

      ◊  /   TOUT           = 'OUI',
            |GROUP_NO       = gr_noeu,                 [l_gr_noeud]
            |SANS_GROUP_NO  = gr_noeu,                 [l_gr_noeud]

      # If modelization is 3D
      ♦  TYPE_MAILLAGE      =   / 'LIBRE',
                                / 'REGLE',              [DEFAULT]

       ◊  EVOL_THER             = evol ,                              [
evol_ther ]

# 2) FISSURE is indicated for crack

      ♦  /  FISSURE       =f ound ,                     [fond_fiss]

      ◊  ABSC_CURV_MAXI      =dmax,                     [R]
      ◊  NUME_FOND           =  /nume,                  [R]
                                /1.
      ◊  NB_POINT_FOND       =  /nbpoint,               [I]

 # Common options

      ♦ MATER= mat,                                     [material]

      ◊   PREC_VIS_A_VIS    =/  1.E-1,                  [DEFAULT]
                            /  epsi,

      ◊  /  TOUT_ORDRE       =  'OUI',
         /   NUME_ORDRE      =  lnuor,                  [L_I]
         /  LISTE_ORDRE      =  lnuor,                  [listis]

         /  TOUT_MODE        =  'OUI',
         /  NUME_ORDRE       =  lnuor,                 [L_I]
         /  LISTE_MODE       =  lnuor,                  [listis]

          / FREQ             =  l_freq,                 [l_R]
```

```
                / LISTE_FREQ       =  l_freq,                    [listr8]
                  ◊ CRITERE        =  /  'RELATIF',              [DEFAULT]
                                      ◊   PRECISION =prec,
                                                    / 1.E-6 ,    [DEFAULT]
                                      /  'ABSOLU',
                                       ♦  PRECISION =prec         [R]
                 / INST              =  [l_R]                     [l_R]
                / LIST_INST          =  [l_R]                    [listr8]
                  ◊ CRITERE          =  /  'RELATIF',            [DEFAULT]
                                      ◊   PRECISION =prec,
                                                    / 1.E-6 ,    [DEFAULT]
                                      /  'ABSOLU',
                                       ♦  PRECISION =prec         [R]

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

3 Keywords

3.1 Keyword MODELISATION

```
♦ MODELISATION =        / '3D',
                        / 'AXIS',
                        / 'D_PLAN',
                        / 'C_PLAN',
```

This option a llows defin i tion of the computation type in 2 D or 3 D. Th e mode l option must be coherent with the model used for the computation of displacements.

3.2 Keyword MATER

```
♦ MATER = mat,                                          [material]
```

The material definition contains the elastic characteristics of the material. It must be homogeneous, isotropic and linear elastic. This material must be **constant** (compulsory use of DEFI_MATERIAU/ELAS). The only tolerated variability is a dependence on **temperature** . If material properties depend on temperature (keyword ELAS_FO of DEFI_MATERIAU), the approch is different according to the type of modelization:

• If EVOL_THER keyword is not given, then the characteristic materials are obtained with reference temperature TEMP_DEF_ALPHA of DEFI_MATERIAU ;

• If EVOL_THER operator is indicated, then the characteristic materials are calculated from the temperature of the crack tip nodes.

3.3 Keyword FOND_FISS

```
◊   FOND_FISS=fond,                                [fond_fiss]
```

It corresponds to computation of a meshed crack, defined for post-processing with operator DEFI_FOND_FISS . By default, this is computed automatically for all the nodes of the crack tip.

3.4 Keywords PREC_VIS_A_VIS/GROUP_NO/SANS_GROUP_NO/TOUT

```
◊  PREC_VIS_A_VIS          =  /  1.D-1,     [DEFAULT]
                              /  epsi,      [R]
◊  /  TOUT                 =  'OUI',
      □  GROUP_NO          =  gr_noeu,      [l_gr_noeud]
       □  SANS_GROUP_NO    =  gr_noeu,      [l_gr_noeud]
```

FOND_FISS keyword makes it possible to input a fond_fiss object (created by the command DEFI_FOND_FISS) in which the necessary information are stored with the automatic search of the nodes of the two lips located on segments normal to the crack tip. It must be noted that the object fond_fiss must be necessarily defined such as the lips of crack are initially stuck (CONFIGURATION_INIT='COLLEE' in DEFI_FOND_FISS [U4.82.01]).

In 3D, by default, the computation of T-stress is calculated on the nodes tips. The user has the following options:

- to select some nodes of crack tips (keyword GROUP_NO);
- to exclude some nodes of the crack tip (keyword SANS_GROUP_NO);
- to compute on all the nodes of the crack tip (keyword TOUT).

◊ ABSC_CURV_MAXI = dmax [R]

is a maximum distance for computation of T-stress starting from the crack tip. In practice, the accuracy of the results is less accurate if it is located very far from the crack tip [R7.02.08]. It is thus advised to choose $dmax$ the smallest as possible (about 4 to 5 elements, or about the radius of the radiant mesh, if necessary). If in a point N of the crack tip, this distance is higher than the distance from the crack tip at this point at the edge of the lips, the value of T-stress at point N are obtained by constant prolongation. The selected value is the point of the crack tip nearest and for which the computation will be carried out.

ABSC_CURV_MAXI operator is optional. When this option is not indicated, the value of ABSC_CURV_MAXI automatically computed is starting from the maximum size h of the meshes connected to the nodes of the crack tip. These size of meshes in each node of the bottom are computed in the DEFI_FOND_FISS command and are present in the concept fond_fiss [D4.10.01]. It was chosen to take ABSC_CURV_MAXI equal to $4h$.

If one chooses the value automatically calculated for ABSC_CURV_MAXI which is advisable, however, needs to make sure that its value (displayed in the file .mess) is coherent with dimensions of the structure.

In automatic search for each node of the crack tip, the operator selects the nodes with the following conditions:

- distance R compared to the crack tip: R < ABSC_CURV_MAXI,
- distance L compared to its opposite on the other lip: L < epsi.ABSC_CURV_MAXI,
- and in 3D distance D from a node of the lips to the line perpendicular to the crack tip: D < epsi_fond. d , where d is the minimal distance between two successive nodes of the crack tip, and where epsi is the value of the provided accuracy (keyword PREC_VIS_A_VIS) and epsi_fond the value of the accuracy provided in keyword PREC_NORM of DEFI_FOND_FISS.

By default epsi is equal to 0.1 . Increasing the value of PREC_VIS_A_VIS (and/or PREC_NORM in DEFI_FOND_FISS) is equivalent to increase the number of potential nodes selected for the computation.

3.5 Keyword RESULTAT

♦ RESULTAT = resu,

resu is a concept of type evol_elas or evol_noli containing the displacement field on all the model.

3.6 Keyword TYPE_MAILLAGE

♦ TYPE_MAILLAGE = / 'LIBRE',
/ 'REGLE', [DEFAULT]

This keyword makes sense only in 3D for meshed cracks defined by FOND_FISS. The principle of computation is then as follows:

1. determination of the normal directions at the crack front for each node localized at the crack front,
2. definition on each one of these directions of NB_NOEUD_COUPE points equally distributed between the crack front and the distance ABSC_CURV_MAXI,
3. projection of the displacement of each lip on these nodes.

3.7 Keyword NB_NOEUD_COUPE

◊ NB_NOEUD_COUPE= / 5, [DEFAULT]
/ N, [I]

It makes it possible to define the number of nodes corresponding to the projection of the the lips displacement on each normal direction. The projection of nodes is equal to distribute between the crack tip and distance ABSC_CURV_MAXI.

3.8 Keywords LIST_INST, INST , FREQ, LIST_FREQ, TOUT_ORDRE, NUME_ORDRE, LIST_ORDRE, TOUT_MODE, NUME_MODE, LIST_MODE

For specific information, refer to [U4.71.00].

3.9 Keyword INFO

◊ INFO = / 1, [DEFAULT]
/ 2,

Level of messages in the message file: if INFO is equal to 2, one gives the list of all the computed values for all the nodes that have been treated.

3.10 Keyword TITRE

◊ TITRE = title,

Title to name the result.

4 Precautions and use advices

4.1 Assumptions relating to the materials used for the computation of displacements

For computation of T-stress by displacement extrapolation method the material needs to be homogeneous and isotropic linear elastic.

This material can be **constant** (use of DEFI_MATERIAU/ELAS) or **function** (use of DEFI_MATERIAU/ELAS_FO). Only the command variables being able to be used for the computation of T-stress in the case of a material function are **"TEMP"** (temperature). These variables must be affected (necessarily in AFFE_MATERIAU/AFFE_VARC) starting from nodal fields (cham_no).

In the case of a material function, the elastic parameters are evaluated at the points of the crack tip. These points are the nodes of the crack tip.

4.2 Count produced

Command POST_T_Q produces a concept of type table_sd_aster. The table can be printed by IMPR_TABLE [U4.91.03]. It contains for each node of the crack tip the value of T-stress: T .

4.3 Additional printings

If INFO is equal to 2, all intermediate computations are displayed in the message file. It is announced that column entitled SAUT_DX (resp. SAUT_DY and SAUT_DZ) in the arrays displayed in the message file corresponding to the displacement step allong axis X1 (resp. X2 and X3), multiplied by a material coefficient, all squared [R7.02.08].

4.4 Precautions and advice

The assumptions necessary to the validity of this method are:

1) the crack must be sufficiently regular (either the crack and the lips must not present a geometric singularity);
2) the material behavior must be elastic, linear, homogeneous and isotropic;
3) the structure must be isothermal (or, the temperature gradients on the lips should be negligable in the area of interpolation).

Computation of interpolation displacement requires having at least 6 nodes on the normal direction of crack tip. If the number of nodes is not sufficient, an alarm is emitted and the lines corresponding to this node of the bottom are set to 0 in the result table. Then the computation continues, if necessary, for the node of the crack tip. In this case one of the following messages are seen:

- either increase the maximum curvilinear abscisse ABSC_CURV_MAXI in order to compute nodes further away from the crack tip;
- either increase the parameter PREC_VIS_A_VIS (and possibly PREC_NORM in DEFI_FOND_FISS), which is less complicated in the selection of nodes for the computation.

Computations are inefficient in time and memory if there are too many points on the crack tip. The use of key word NB_POINT_FOND makes it possible to limit the post-processing to a certain number of points equal ly distribute d along the crack front (for example twenty points is often sufficient).

5 Example

Circular crack in 3D cube body subject to pure tension is presented (testcase SSLV324). Here to call POST_T_Q we need to use :

```
TABT= POST_T_Q(MODELISATION='3D',
               RESULTAT=CHAMDEPL,
               FOND_FISS=FISS,
               MATER=MAT,
                NFO=2,
               TYPE_MAILLAGE='LIBRE',
               NB_NOEUD_COUPE=18,
          )
```