

Operator DEFI_GEOM_FIBRE

1 Goal

Within the framework of a modeling of the multifibre type, there are two “levels” of modeling:

- modeling known as “longitudinal” represented by a beam (of support geometrical SEG2)
- modeling planes section (perpendicular to SEG2).

The operator DEFI_GEOM_FIBRE allows to define the geometry of the groups of fibres which will be used in the definition of the cross section using the operator AFFE_CARA_ELEM.

The fibres can be described either using a grid plan read beforehand by the operator LIRE_MALLAGE (keyword SECTION) and/or in the form of surfaces “specific” (keyword FIBRE or ASSEMBLAGE_FIBRE).

Each occurrence of SECTION, FIBRE or ASSEMBLAGE_FIBRE defines a set of fibres which should be named in a single way. A set of fibres will have the same behavior for all its fibres, definite using DEFI_COMPOR.

The order creates a structure of data of the type geom_fibre.

Caution:

With the information given in SECTION or FIBRE, it is possible to calculate the majority of the integrated characteristics of the cross-sections (surface, static and quadratic moments). However, it is not possible to calculate inertias of torsion, of warping as well as the position of the center of torsion. These sizes must be given the operator AFFE_CARA_ELEM.

If the fibres are defined starting from a grid the operator MACR_CARA_POUTRE allows to read this grid and to calculate all the characteristics. The result of this operator is a table which can be as starter of the operator AFFE_CARA_ELEM.

2 Syntax

```
gf [geom_fibre] = DEFI_GEOM_FIBRE (
  ◇ INFORMATION =                                     /1
  [defect]
  /2
  ◆ | SECTION = _F (
    ◆ GROUP_FIBRE = nom_gfs [K24]
    ◆ MAILLAGE_SECT = my [grid]
      ◆/ TOUT_SECT = 'YES'
      /MAILLE_SECT = e-mail [l_maille]
      /GROUP_MA_SECT = g_mail [l_gr_maille]
    ◇ COOR_AXE_POUTRE =/0.0, 0.0 [defect]
      /X , there [l_R]
  ),
  | FIBRE = _F (
    ◆ GROUP_FIBRE = nom_gfp [K24]
    ◇ CARA = /'SURFACE' [defect]
      /'DIAMETER'
    ◆ VALE = l_val [l_R]
    ◇ COOR_AXE_POUTRE =/0.0, 0.0 [defect]
      /X , there [l_R]
  | ASSEMBLAGE_FIBRE = _F (
    ◆ GROUP_ASSE_FIBRE = nom_gfa [K24]
    ◆ GROUP_FIBRE = nom_gf [K24]
    ◆ COOR_GROUP_FIBRE = coord_gf [l_R]
    ◆ GX_GROUP_FIBRE = gx_gf [l_R]
  ),
)
```

3 Operands

3.1 Operand INFORMATION

◇ INFORMATION

If the value of INFORMATION is 2, the operator prints in the file 'MESSAGE', for each occurrence of the keywords SECTION or FIBRE, characteristics of each fibre (position and surface).

Note:

All the characteristics are given compared to the axis defined in the operand COOR_AXE_POUTRE, by calling the coordinates of the section x (horizontal) and y (vertical).

3.2 Keywords SECTION and FIBRE

◆ | SECTION
| FIBRE

The entities of the grid of beams concerned and the sections define which are affected for them.

The keyword SECTION allows to affect a section defined by a grid plan (the elements of this grid are the sections of fibres). The numeration of fibres is completely dependent on the grid and the type of meshes (quadrangle or triangle).

The keyword FIBRE allows to affect a section where the fibres are defined by "points". The fibres are numbered of 1 to N in the order where they were defined.

In the presence of the two key words, the classification of fibres will begin with fibres defined under the keyword `SECTION` then by those definite under the keyword `FIBRE`. To obtain it, it is possible to define `INFO=2` in `DEFI_GEOM_FIBRE`.

Note:

Currently the number of group of fibres on an element beam is limited to 10.

3.2.1 Operands commun runs with `SECTION` and `FIBRE`

◆ `GROUP_FIBRE`

This operand makes it possible to define a name for the group of fibres (24 characters). This name will be useful in the operator `DEFI_COMPOR` to allot a material and a behavior to this group of fibres. It is pointed out that all the fibres defined by an occurrence of `SECTION` or `FIBRE` will have the same behavior.

◇ `COOR_AXE_POUTRE` = (xg, yg)

This operand makes it possible to define the coordinates of the axis of the beam in the reference mark of the cross-section. Integrations (static moments or of inertias) will be made compared to this axis. The position by default is (0.0,0.0).

When one gives the coordinates of fibres is by a grid (word key `SECTION`) maybe by triplets of values (key word `FIBRE`), they are in a reference mark which is not related to the beam. That makes it possible to define complex sections without obligation to define them in the principal reference mark of inertia of the beam.

It is easier to parameterize the grid of a corner section (figure 3.2.1-a) in the reference mark (x_1, y_1) that in the reference mark (x_2, y_2) . The grid of the section is in the plan XY .

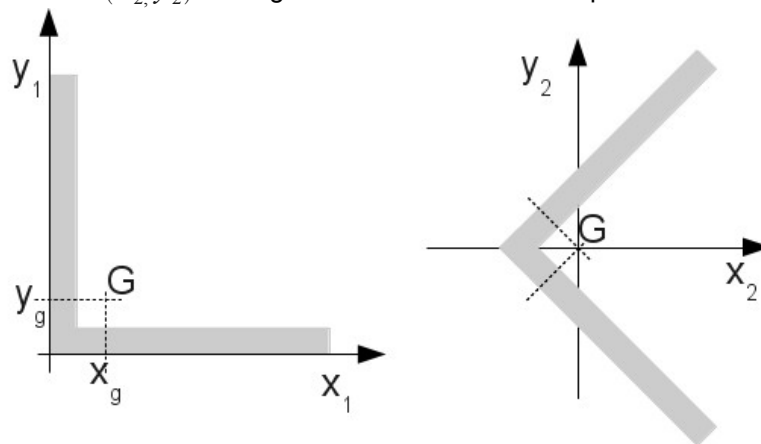


Figure 3.2.1-a : Corner section.

Characteristics of the beams (section, inertias,...) are defined in the principal reference mark of inertia, it is thus necessary to make the relation between the reference mark of definition of fibres and the principal reference mark of inertia of the beam. The key word `COOR_AXE_POUTRE` allows to define the translation of the reference mark related to the grid of the section to make it correspond to that of the beam.

In the example of the angle (figure 3.2.1-a) for:

- the reference mark (x_1, y_1) : `COOR_AXE_POUTRE` = (xg, yg)
- the reference mark (x_2, y_2) : `COOR_AXE_POUTRE` = (0.0, 0.0)

Always for the example of the angle but only in the case of the reference mark (x_1, y_1) it is necessary to turn the reference mark of the section of 45° , that is carried out by the key word `ORIENTATION` of the operator `AFFE_CARA_ELEM`. In the case of the reference mark (x_2, y_2) , the key word `ORIENTATION` of `AFFE_CARA_ELEM` will be 0, the value by default.

3.2.2 Operands specific to SECTION

The group of fibres is defined by a set of “surface” fibres.

◆ MAILLAGE_SECT

Name of “grid” plan which contains the “description of the section”.

By “grid”, one understands a set of triangular meshes with 3 nodes and/or quadrilaterals with 4 nodes. By “description of the section”, one understands part of this “grid” specified by one of the operands TOUT_SECT, MAILLE_SECT or GROUP_MA_SECT. Each mesh represents the section of a fibre.

- ◆ /TOUT_SECT
/MAILLE_SECT
/GROUP_MA_SECT

Operands	Contents/Significance
TOUT_SECT	The section is defined by the totality of the meshes of the grid defined under MAILLAGE_SECT
MAILLE_SECT	The section is defined by a list of meshes
GROUP_MA_SECT	The section is defined by a list of groups of meshes

Note:

*Since it is not used as support with finite elements, the “grid” should not obligatorily have a connectivity, it can be made up of a set of juxtaposed meshes or not.
Coordinates x and y grid plan of the section (x horizontal, y vertical) are defined in a plan perpendicular to the axis of the beam. To define the angle of gimlet, i.e. the angle enters the axis x grid plan of the section and the axis Y element beam, it is necessary to use the keyword ORIENTATION of the operator AFFE_CARA_ELEM.*

3.2.3 Operands specific to FIBRE

The group of fibres is defined by a set of “specific” fibres.

◆ CARA = 'SURFACE' | 'DIAMETER'

Allows to specify if the third value given for each fibre is surface or the diameter (see VALE).

◆ VALE

Each fibre is described by a triplet of values: (x, y, val) It is necessary to give the values according to this sequence, as many triplets as of fibres.

- x and y are the coordinates of the center of fibre in a plan perpendicular to the axis of the beam.
- val is either the surface of a fibre, or the diameter of a cylindrical fibre.

The position of the reference mark can be modified thanks to COOR_AXE_POUTRE. To give an angle of gimlet, should be used the key word ORIENTATION of the operator AFFE_CARA_ELEM.

3.3 Keyword ASSEMBLAGE_FIBRE

This key word makes it possible to define an assembly of group of fibres. Kinematics connecting the various groups of fibres is particular and is used to give an account of phenomena specific to the modeling of the fuel assemblies, cf [R3.08.08] multifibre Element of beam (right).

◆ GROUP_ASSE_FIBRE

This operand makes it possible to define a name for the group of fibre assembly (24 characters). This name will be useful in the operator DEFI_COMPOR to allot a material and a behavior to this group of fibre assembly. It is pointed out that all the fibres defined by an occurrence of GROUP_ASSE_FIBRE will have the same behavior.

◆ GROUP_FIBRE

This operand makes it possible to define the list of the names of the groups of fibres which will constitute the fibre assembly. The names of the groups are defined is by SECTION/GROUP_FIBRE maybe by FIBRE/GROUP_FIBRE.

◆ COOR_GROUP_FIBRE

This operand makes it possible to define the coordinates of the centre of gravity of the groups of fibres defined under the keyword GROUP_FIBRE. It is a list of realities (x1, y1,..., xn, yn) which has 2 times the length of the list given under GROUP_FIBRE. The coordinates of the groups of fibres are relocated of (xi, yi).

◆ GX_GROUP_FIBRE

This operand makes it possible to define the coefficient of torsion of the groups of fibres defined under the keyword GROUP_FIBRE. It is a list of reality (gx1,..., gxn) which has the same length as the list given under GROUP_FIBRE.

4 Example

One wishes to build a section of reinforced concrete beam, with two cylindrical steel reinforcements, of diameter 32.

One creates a structure of data named GF comprising two groups of fibres:

- one is named SBET , and is created with a grid plan,
- the other is named SACI , and is created with two specific fibres for steels.

```
MASEC1 = LIRE_MALLAGE (UNIT = 21)
```

```
GF=DEFI_GEOM_FIBRE (  
  SECTION=_F (GROUP_FIBRE=' SBET', MAILLAGE_SECT=MASEC1,  
    TOUT_SECT=' OUI',  
    COOR_AXE_POUTRE= (0. , 0.),),  
  FIBRE=_F (GROUP_FIBRE=' SACI',  
    CARA=' DIAMETRE',  
    VALE= ( 0.05,-0.2,32.E-3,  
      -0.05,-0.2,32.E-3),  
    COOR_AXE_POUTRE= (0. , 0.),),  
)
```