Modeling POU_D_EM

Summary:

Modeling POU_D_EM corresponds to the formulation of elements of multifibre beam (heterogeneous beam of section divided into several fibres).

They are usable for three-dimensional problems in linear and nonlinear mechanical analysis.

Contents

1 Discretization

1.1 Degrees of freedom

For the modeling of multifibre beam into three-dimensional the degrees of freedom of discretization are, in each node of the mesh support, the six components of displacement (three translations and three rotations). These nodes are supposed to describe a segment of average fibre of the beam.

Finite element	Degrees of freedom (with each node top)					
POU_D_EM	DX	DY	DZ	DRX	DRY MARTINI	DRZ

1.2 Mesh support of the matrices of rigidity

The meshs support of the finite elements, in displacement formulation, are segments with two nodes SEG2 :

Modeling	Mesh	Finite element	Remarks
POU_D_EM	SEG2	MECA_POU_D_EM	

1.3 Mesh support of the loadings

As for the classical elements of beam (POU_D_E), all the loadings applicable to the elements of multifibre beam are treated by direct discretization on the mesh support of the element in displacement formulation.

No mesh support of loading is thus necessary for the edge of the elements of beam or bar.

1.4 Main features of modeling

Modeling POU_D_EM is based on the resolution of a problem of beam for which each definite section is divided into several fibres.
Each fibre behaves then like a beam of Euler, i.e. the sections remain right and perpendicular to average fibre (assumption of great twinge).
The section can be of an unspecified form.

2 Assignment of the characteristics

For this element of structures 1D, it is necessary to affect geometrical characteristics which are complementary to the data of grid. The definition of these data is carried out with the order AFFE_CARA_ELEM associated with the keywords following factors:

- **AFFE_SECT**
  Allows to associate a grid plan of section with an element beam.
- **AFFE_FIBRE**
  Allows to associate a section made up of one or more specific fibres with an element beam.
- **BEAM**
  Allows to associate a geometrical characteristic of torsion with an element beam.
- **ORIENTATION**
  Allows "to turn" the grid plan of the section around the axis of the beam.

Remarks on the characteristics of modeling:

- Within the framework of modeling of a multifibre type, there are two "levels" of modeling. There is the modeling known as "longitudinal" which will be represented by a beam (geometrical support SEG2) and a modeling planes section (perpendicular to SEG2).
  The operand **AFFE_SECT** allows to associate a grid plan of section with an element beam. The operand **AFFE_FIBRE** allows to associate a section made up of one or more specific fibres (defined by their position and surfaces) with an element beam.
- In general in the plane modeling of the section, several materials cohabit. For example, in a section concrete reinforced, there are at the same time concrete and reinforcements. In this case there, the operator **CREA_MAILLAGE** allows to duplicate support E.F so that there is one material by support.
- The operand **BEAM** is used to affect a geometrical characteristic of torsion (JX) who cannot be calculated starting from the grid plan of the section. If the value is used GENERAL for the keyword SECTION operand BEAM, the characteristics should be given (CARA) With, IY and IZ besides JX because the operator AFFE_CARA_ELEM at least expects these four characteristics for a classical beam.
  Values (VALE) data with With, IY and IZ are not used by the element POU_D_EM, because they are calculated starting from the grid plan of the section. On the other hand a checking of the coherence of information (SURFACE and INERTIA) provided on the one hand by With, IY, IZ and in addition by the keywords AFFE_SECT and AFFE_FIBRE is realized. The criterion of error is based on the error relative and is compared either with the value by default or to that given by the user via the keywords PREC_AIRE and PREC_INERTIE. (Cf orders AFFE_CARA_ELEM keywords PREC_AIRE and PREC_INERTIE operand BEAM).
- The operand **ORIENTATION** is used in general "to turn "the grid plan of the section around the axis of the beam (CARA 'ANGL_VRIL'). Indeed, by default, the axis x (horizontal) of the grid plan of the section is confused with the axis y beam (see [Figure 2-a]).

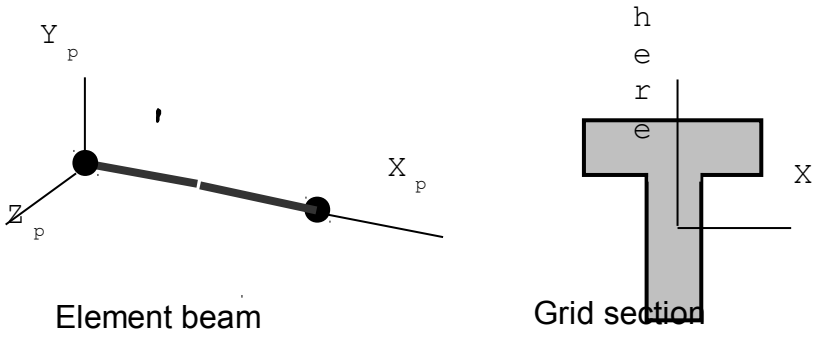

Figure 2-aOrientation by default of the grid plan compared to the element beam

3 Supported loadings

The loadings available are the following:

- **'FORCE_POUTRE'**
  Allows to apply linear forces.
- **'GRAVITY'**
  Allows to apply a loading of type gravity.
- **'ROTATION'**
  Allows to define the number of revolutions and the vector of rotation.

The application of a thermal loading of dilation is carried out by defining the keyword factor AFFE_VARC under AFFE_MATERIAU [U4.43.03].

4 Non-linear possibilities

4.1 Law of behaviors

Laws of behaviors specific to this modeling, usable under BEHAVIOR in STAT_NON_LINE and DYNA_NON_LINE are the following ones, cf [U4.51.11]:

```
/   'CORR_ACIER'
/   ' MAZARS_GC '
/   'PINTO_MENEGOTTO'
/   'VMIS_CINE_LINE'
/   'VMIS_ISOT_LINE'
/   'VMIS_ISOT_TRAC'
/   'GRAN_IRRA_LOG'
```

It is also possible for these modelings using a monodimensional state of stresses to use the behaviors 3D (thanks to the method of Borst [R5.03.03]), provided that one only material is used by finite element.

4.2 Deformations

Only linearized deformations keyword 'SMALL' under DEFORMATION are available in the relations of behavior (cf [U4.51.11]).

5 Examples of implementation: CAS-tests

Linear statics

- SSLL111A [V3.01.111]: Static answer of a reinforced concrete beam (section in T) with a linear behavior subjected to three successive loading cases: a specific force, the actual weight and a rise in temperature.

Non-linear statics

- SSNL119A [V6.02.119]: Deflection test 3 points, static answer of a reinforced concrete beam (rectangular section) with a nonlinear behavior of Borderie.

Linear dynamics

- SDLL130B [V2.02.130]: Seismic response of a reinforced concrete beam (rectangular section) to linear behavior.

Non-linear dynamics

- SDNL130A [V5.02.130]: Seismic response of a reinforced concrete beam (rectangular section) to nonlinear behavior.