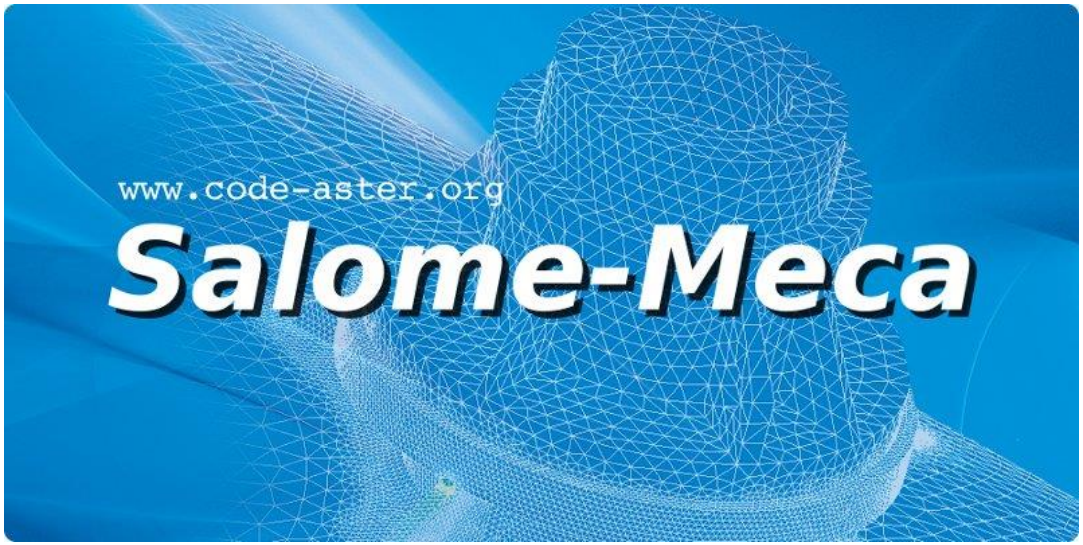
Overview on structural elements



Code_Aster, Salome-Meca course material

Definitions and description

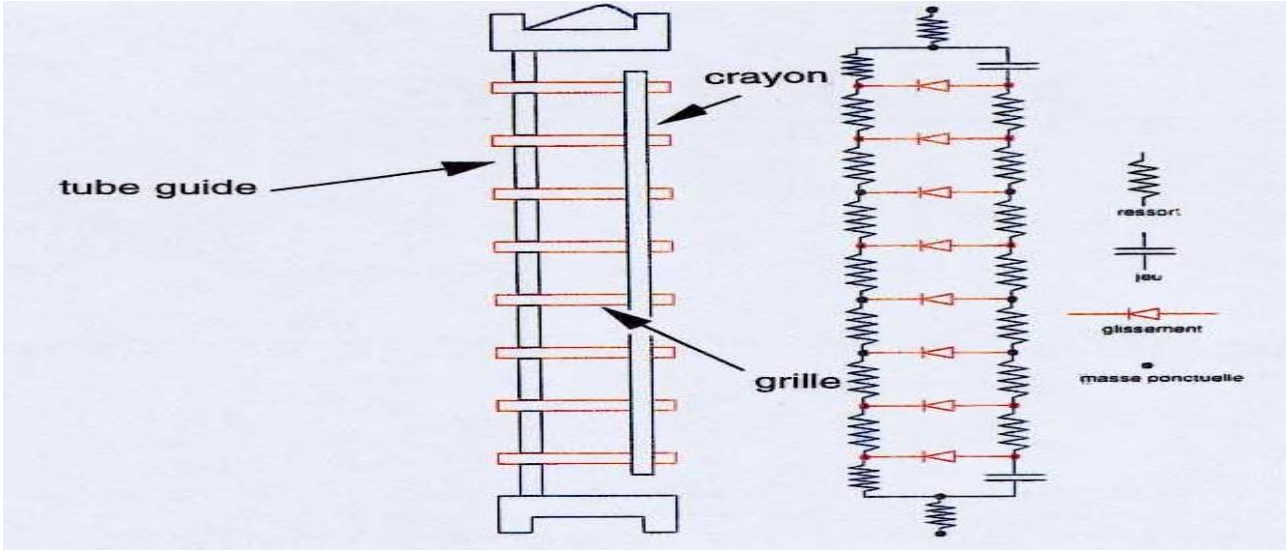
Definitions

Modeling of structures with neutral axis or neutral surface
Strong assumptions useful to reduce the problem size
Some advanced models are dedicated for civil or mechanical structures.

- **Discrete elements (0D)**
  - Spring, mass / inertia, damping
- **Beams, bars and cables (1D)**
  - Straight or curved beam, ball joint rods
  - Dimensions : L1,L2<<L
- **Pipes (1D)**
  - Straight or curved pipes
  - Both beam and shell theory combined in one model
  - For circular sections only
- **Plates and shells (2D)**
  - thin structures with plane neutral surface (plate model) or curved (shell model)
  - e<<Weight,Lenght

Discrete elements

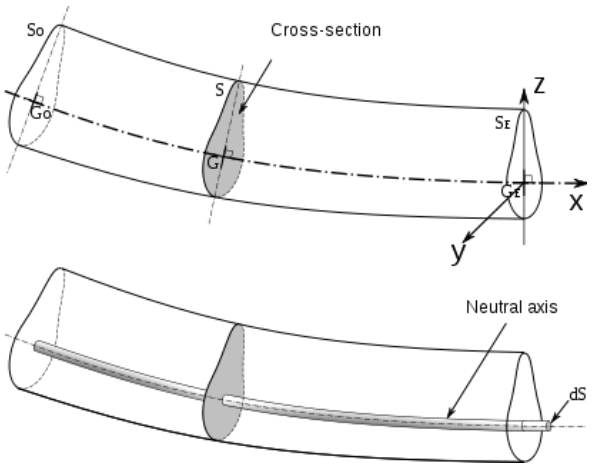
- **Modeling of mass-points, springs and dampers**
- **Cell support**
  - `POI1` or `SEG2`
- **Modeling in *Code_Aster***
  - `DIS_T`, `DIS_TR`, `2D_DIS_T`, `2D_DIS_TR`
- **Usage**
  - Spring, mass / inertia, damping



Eg :Use of discrete elements for fuel rod

Beam, bar and cable elements (1/2)

Modelling of slender structures with neutral fiber



Cell support

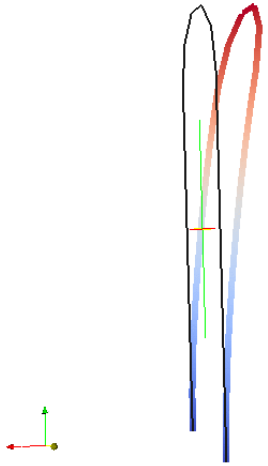
- **SEG2, SEG3**

Modeling in *Code_Aster*

- Straight beams : **POU_D_E**, **POU_D_T**, **POU_D_TG**
- Multi-fibers beams : **POU_D_EM**, **POU_D_TGM**
- Bars : **2D_BARRE**, **BARRE**
- Cables : **CABLE**

Usage

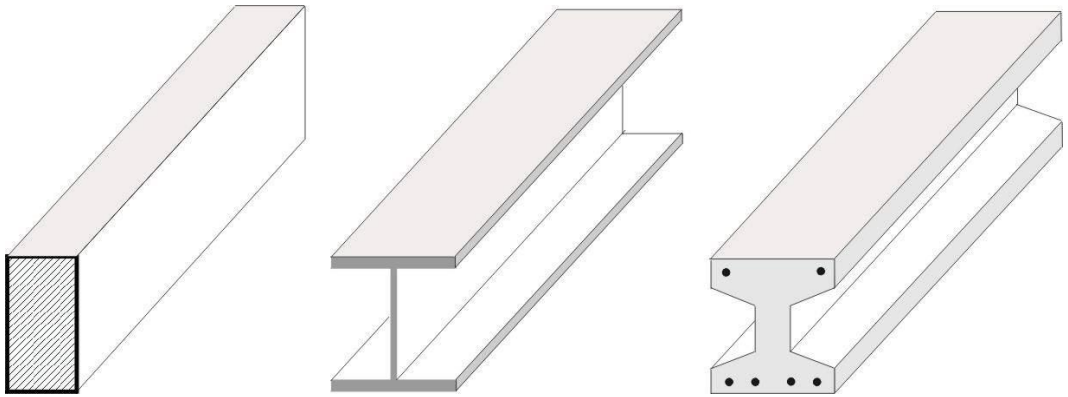
- Use of beam model for non linear modal analysis of steam water tubes



Beam, bar and cable elements (2/2)

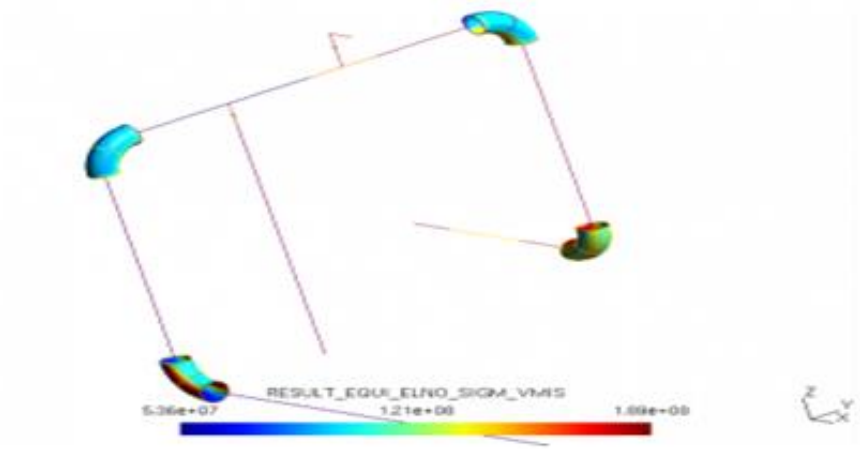
How to choose the right model?

- **Euler beams (`POU_D_E*`) :** significant slenderness
- **Timoshenko (`POU_D_T*`) :** small slenderness
- **Warping (`POU_D_TG*`) :** thin-walled section (sections I, H, L)
- **Multi-fiber (`POU_D_*M`) :** for non-linear materials
- **Bars :** reinforcement, tension / compression stress
- **Cable :** tensile load



Pipe elements (1/2)

- **Modeling of straight or bended pipes**
- **Cell support**
  - **SEG3** ou **SEG4** (bended pipes)
- **Modeling in *Code_Aster***
  - **TUYAU_3M** (straight and bended), **TUYAU_6M** (straight)
- **Usage**
  - Straight or bended pipes
  - Mixed models : Euler-Bernoulli hypotheses for the deformation of the fiber coupled with Love-Kirchoff hypotheses for the deformation of the cross section



Eg :Use of Pipe elements mixed with shell elements

Pipe elements (2/2)

How to choose the right model?

- **Mixed formulation beam + Shell**
- **Displacement of the shell is decomposed into Fourier series**
  - **`TUYAU_3M` : 3 Fourier modes**
  - **`TUYAU_6M` : 6 Fourier modes**
- **Thin pipe : thickness on the radius of the cross-section smaller than < 0.1 : `TUYAU_6M`**
- **Plasticity : `TUYAU_6M`**

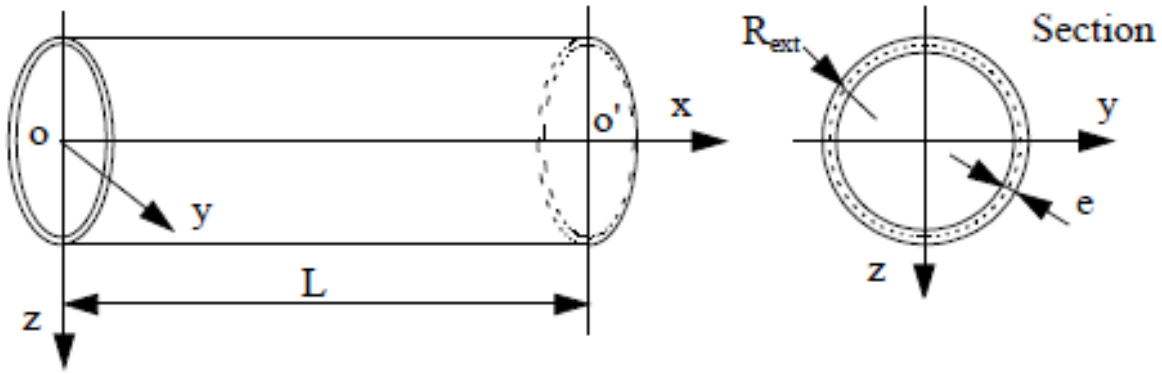
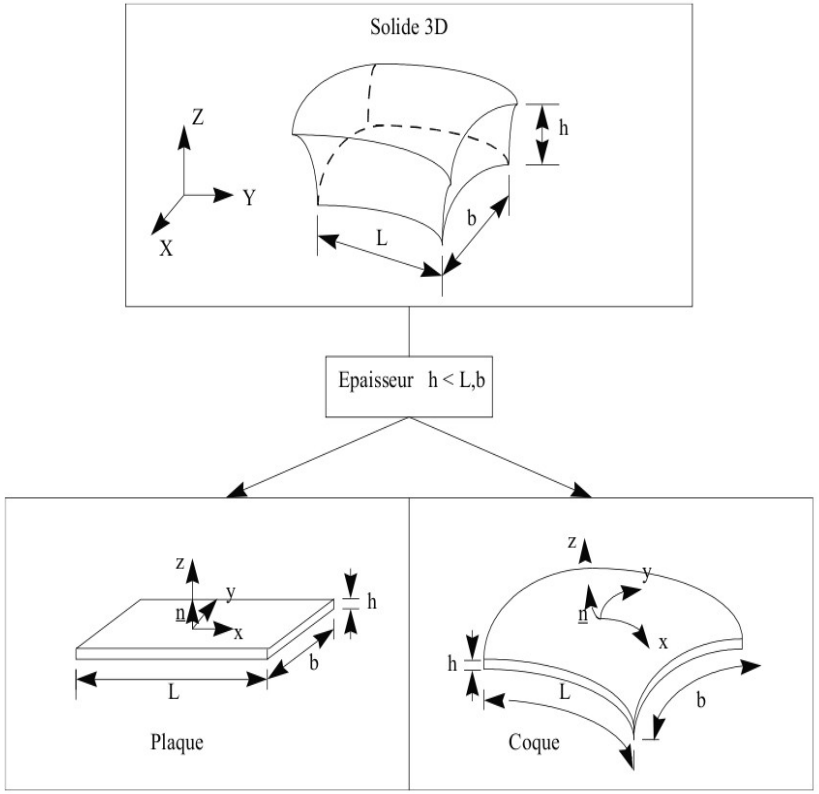
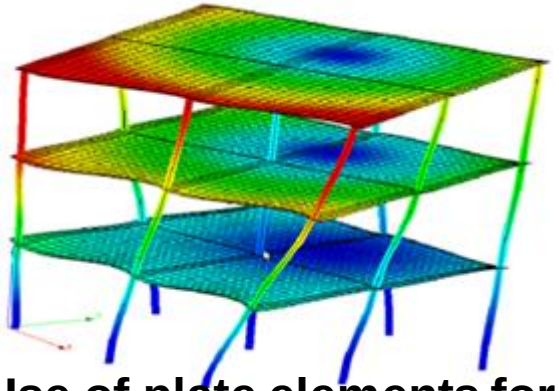


Plate and shell elements (1/2)

- **Modelling of slender structures with neutral surface**



- **Cell support**
  - `TRIA3`, `QUAD4`
  - `TRIA7`, `QUAD9` (`COQUE_3D`)
- **Modeling in *Code_Aster***
  - Plates : `DKT`, `DST`, `Q4G`, `DKTG`, `Q4GG`
  - Shells : `COQUE_3D`
- **Usage**

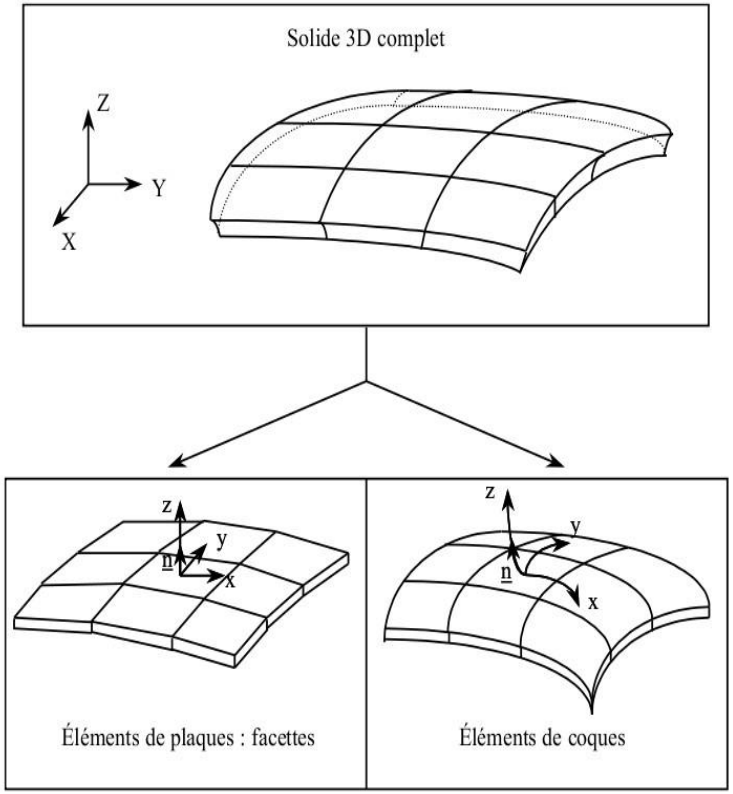


Eg :Use of plate elements for a seismic analysis

Plate and shell elements (2/2)

How to choose the right model?

- **Planar structures:**
  - **DKT**, **DST**, **Q4G**
- **Curved structures :**
  - **COQUE_3D**
- **Thick structures (thickness/length ratio is about 1/10) :**
  - **DST**, **Q4G**
- **Non-linear material :**
  - **DKT**, **COQUE_3D**
- **Large displacements:**
  - **COQUE_3D**



The characteristics of structural elements

General information

- **The characteristics are defined in the command `AFFE_CARA_ELEM` [u4.42.01]**

- **Geometrical information not given by the mesh**

  - **Discrete elements** (spring, mass / inertia, damping) : values of stiffness, mass or damping matrices; orientation

  - **Beams** : cross-section; orientation of the principal axes of inertia about the neutral axis; curvature of the curved elements; general characteristics

  - **Bars or cables** : area of the cross-section

  - **Pipes** : cross section, number of angular sectors and layers

  - **Shells** : thickness ; reference frame in the tangent plane ; number of layers ; orientation

Characteristics for discrete elements

- **Defined with the command AFFE_CARA_ELEM / DISCRET**

```
cara=AFFE_CARA_ELEM(MODELE=model,
                    DISCRET=(_F(REPERE='LOCAL',
                                CARA='K_T_D_L',
                                GROUP_MA=('AB','FG',),
                                VALE=(1.E5,0.,1.E5,),),
                             _F(REPERE='LOCAL',
                                CARA='M_T_L',
                                GROUP_MA=('AB','FG',),
                                VALE=(0.,0.,0.,0.,0.,
                                      0.,0.,0.,0.,0.,
                                      0.,0.,0.,0.,0.,
                                      0.,0.,0.,0.,0.,),),),
                    ORIENTATION=_F(GROUP_MA=('AB','FG',),
                                   CARA='VECT_Y',
                                   VALE=(-1.,0.,0.,),),);
```

Choosing the reference frame

Choosing the type of discrete : spring, mass, damping

Choosing the entities: nodes, group of nodes, cells, group of cells

Coefficients of the matrix

Choosing the orientation

Characteristics for beams

- **Defined with the command AFFE_CARA_ELEM / POUTRE**

```
cara=AFFE_CARA_ELEM(MODELE=model,
                    POUTRE=_F(GROUP_MA=('CD','DE'),
                              SECTION='GENERALE',

                              CARA=('A','IY','IZ','AY','AZ','JX',),
                              VALE=(0.11780973E-2,0.125172834E-6,
                                    0.125172834E-6,1.3096094,
                                    1.3096094,0.250345668E-6,),),

                              ORIENTATION=_F(GROUP_MA=('CD','DE'),
                                             CARA='ANGL_VRIL',
                                             VALE=30.,),

);
```

Choosing the shape of the cross-section

Cross-section caracteristics

Choosing the orientation

Characteristics for bars and cables

- **Defined with the command AFFE_CARA_ELEM / BARRE**

```
cara=AFFE_CARA_ELEM(MODELE=model,
                    BARRE=_F(GROUP_MA='TOUT',
                             SECTION='GENERALE',
                             CARA='A',
                             VALE=2.827e-05,),);
```

Choosing the shape of the cross-section

Cross-section caracteristics

- **Defined with the command AFFE_CARA_ELEM / CABLE**

```
cara=AFFE_CARA_ELEM(MODELE=model,
                    CABLE=_F(GROUP_MA='TOUT',
                             SECTION=1.3,
                             N_INIT=3000.,),);
```

Area of the cross-section

Initial tension of the cable

Characteristics for pipes

- **Defined with the command AFFE_CARA_ELEM / POUTRE**

```
cara=AFFE_CARA_ELEM(MODELE=model,
                    POUTRE=_F(GROUP_MA='TOUT',
                              SECTION='CERCLE',
                              CARA=('R','EP',),
                              VALE=(REXT,EP,),
                              TUYAU_NSEC=16,
                              TUYAU_NCOU=3,),
                    ORIENTATION=_F(GROUP_NO='A',
                                   CARA='GENE_TUYAU',
                                   VALE=(0.0,0.0,1.0,),),);
```

Choosing the entities: nodes, group of nodes, cells, group of cells

Choosing of the cross-section, always CERCLE for pipes

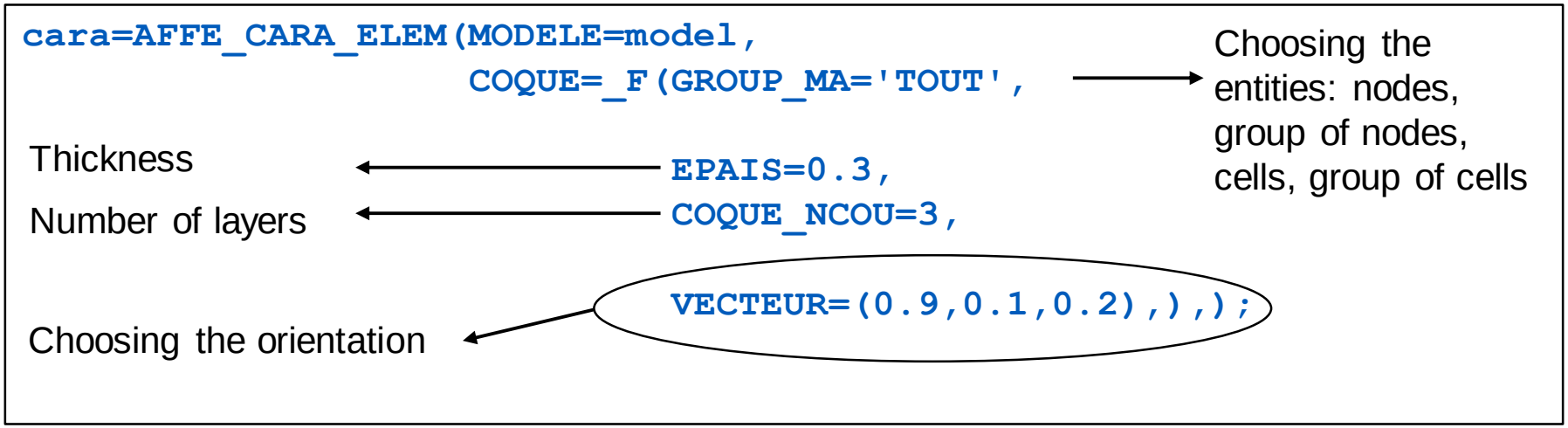
Radius and thickness

Number of angular sectors and layers (thickness)

Determining the direction of the generatrix of the pipe

Characteristics for plates and shells

- **Defined in the command `AFFE_CARA_ELEM / COQUE`**



BEWARE !

The multi-layers concept is different from the multi-material concept !

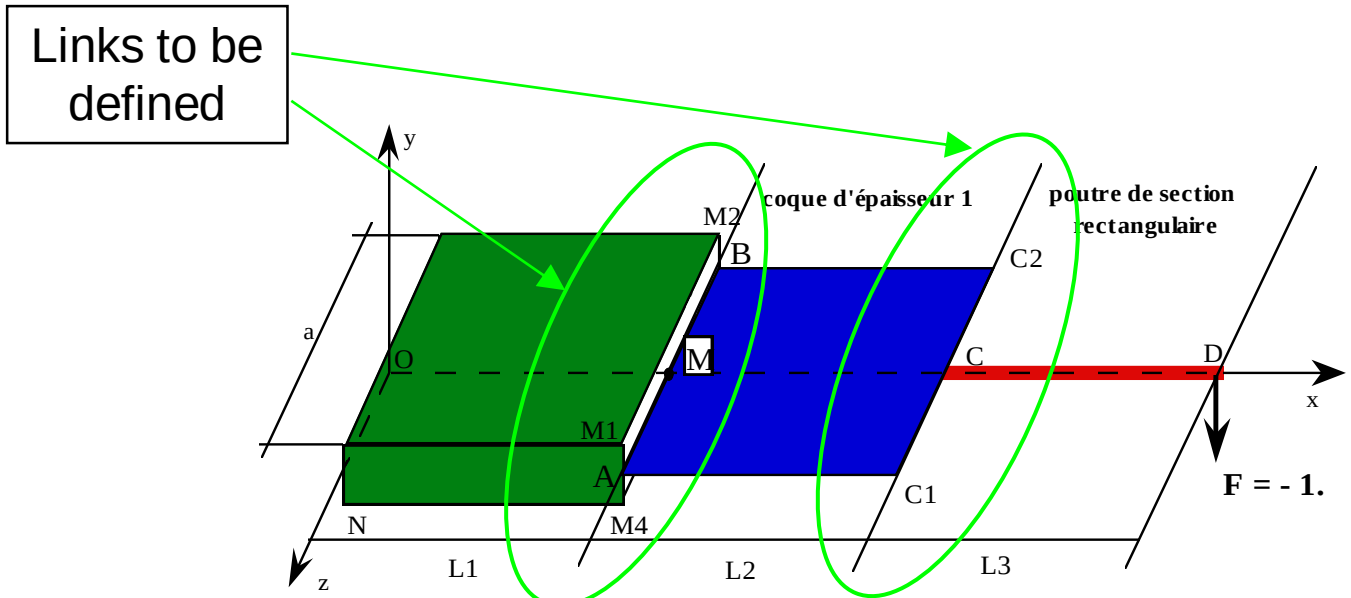
multi-layers == more integration points in the thickness

multi-material == composite (1 material by layer)

Connecting models

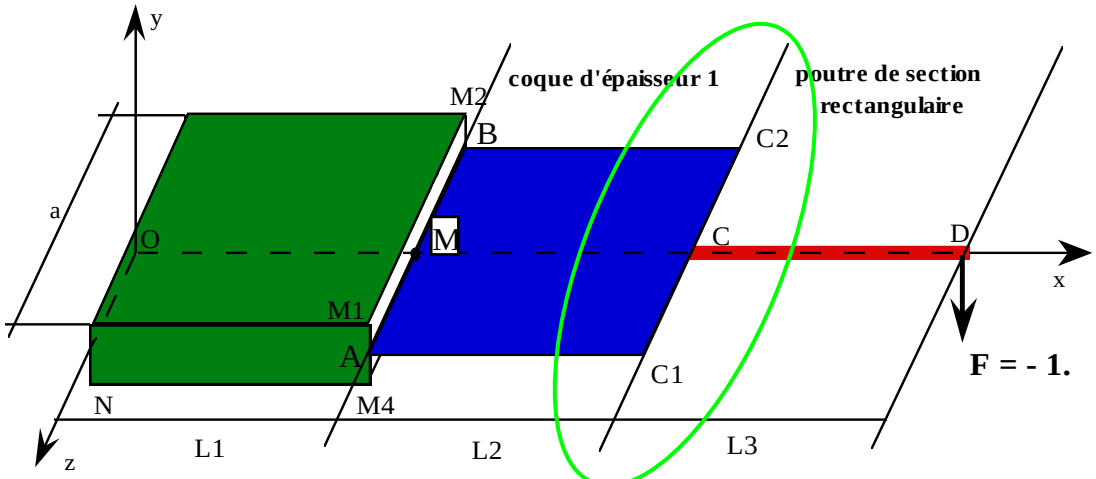
Mixtures of models

- **For saving time and memory**
- **Need to link the different parts**
- **Definition of linear relations**
- **Defined in the command `AFFE_CHAR_MECA` [u4.44.01]**



Connection of beam models

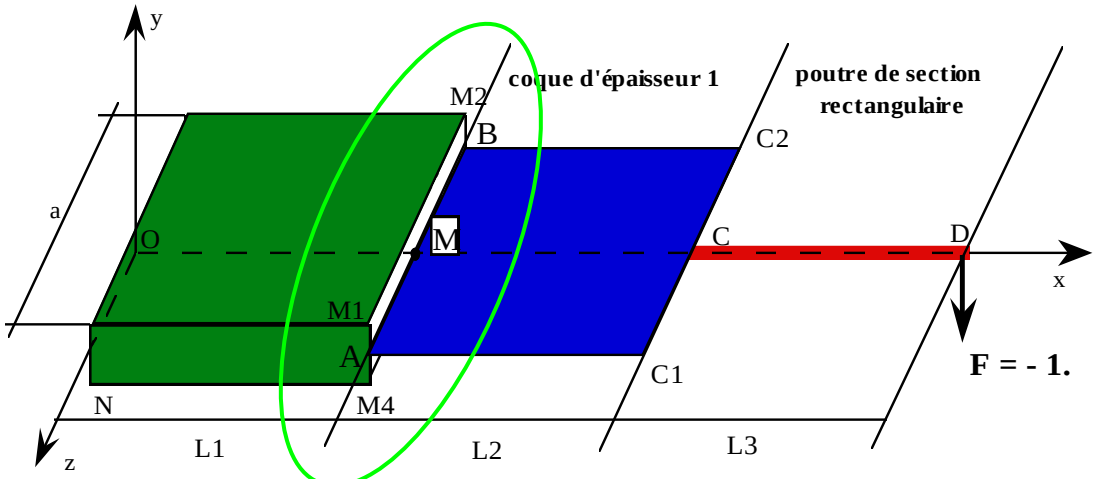
- **Defined in the command AFFE_CHAR_MECA / LIAISON_ELEM**
  - 3D_POU / 2D_POU
  - COQ_POU




```
char=AFFE_CHAR_MECA(
          MODELE=model,
          LIAISON_ELEM=_F(OPTION='COQ_POU',
                          CARA_ELEM=CAREL,
                          AXE_POUTRE=(1.,0.,0.,),
                          GROUP_MA_1='C1C2',
                          GROUP_NO_2='C',),);
```

Connection of shell models

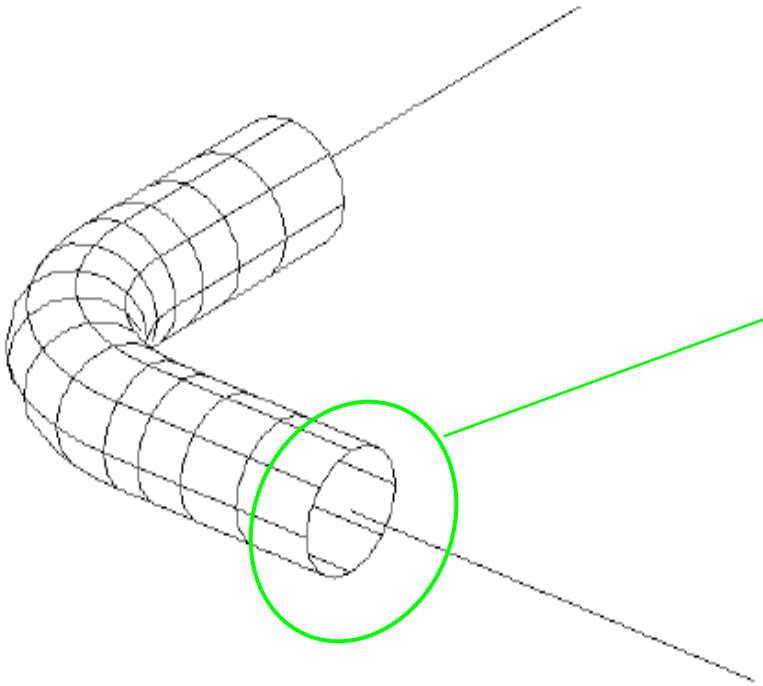
- **Defined in the command AFFE_CHAR_MECA / LIAISON_MAIL**
  - COQUE
  - MASSIF_COQUE
  - COQUE_MASSIF




```
char=AFFE_CHAR_MECA(MODELE=model,
          LIAISON_MAIL=_F(
                        TYPE_RACCORD='MASSIF_COQUE',
                        GROUP_MA_ESCL='AB',
                        GROUP_MA_MAIT='M1M2M3M4'),);
```

Connection of pipe models

- **Defined in the command AFFE_CHAR_MECA / LIAISON_ELEM**
  - COQ_TUYAU
  - 3D_TUYAU

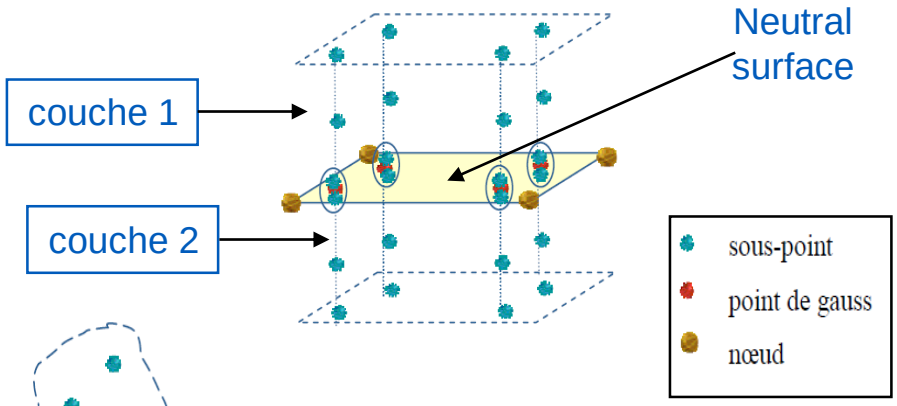


```
char=AFFE_CHAR_MECA(
          MODELE=model,
          LIAISON_ELEM=_F(
          OPTION='COQ_TUYAU',
          GROUP_MA_1='CERCL2',
          GROUP_NO_2='NOPOU1',
          CARA_ELEM=CAREL1,
          AXE_POUTRE=(COS30,0.5,0.0,),
                    ),);
```

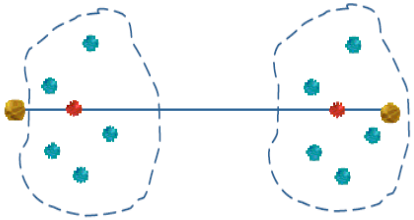
Post-processing with structural elements

Notion of sub-points

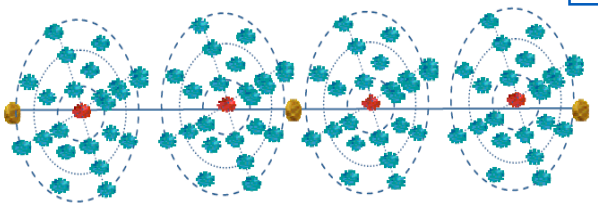
- **Sub-points are integration points in the thickness of the structural element**



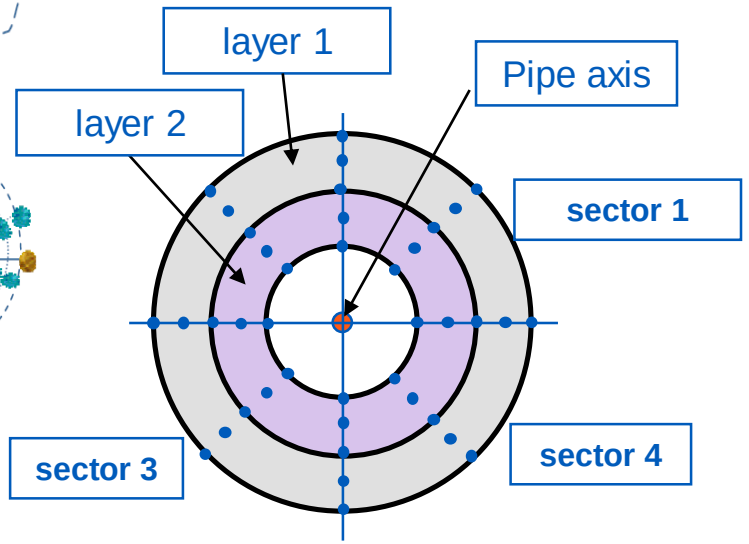
- **Plates and shells :**
  - 3 sub-points per layer
  - Example : QUAD4, 4 Gauss points
  - NBSP=3 x NCOU



- **Multi-fibers beams :**
  - 1 sub-point per layer
  - Example : SEG2, 2 Gauss points
  - NBSP=NBFIBRE



- **Pipes :**
  - 3 sub-points per layer and angular sector
  - Example : SEG3, 3 Gauss points
  - NBSP=(2 x NCOU+1)(2 x NSEC+1)



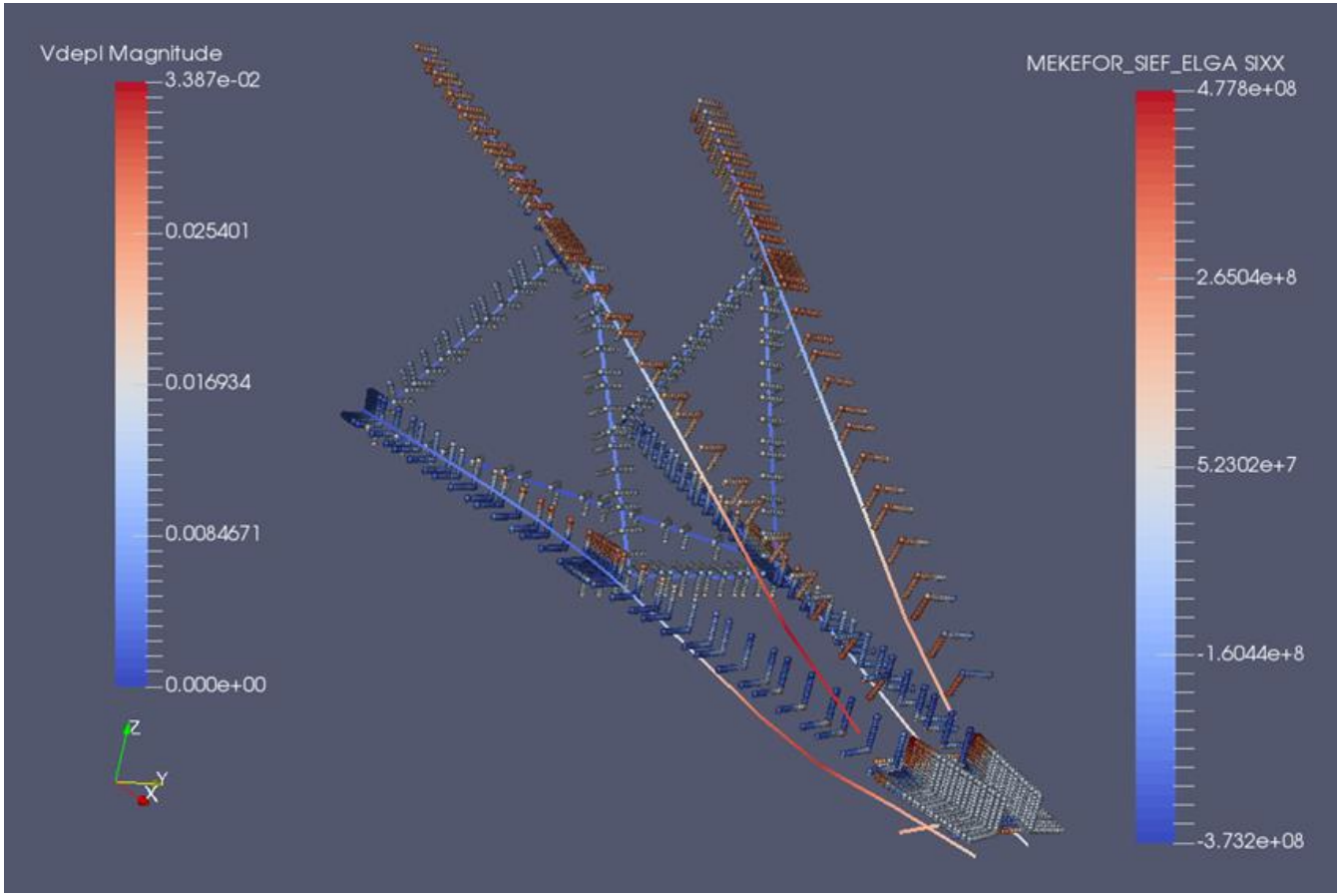
Post-processing

- **Post-processing is done with the usual operators (`CALC_CHAMP / IMPR_RESU`) but**
  - The fields are expressed in the user reference
  - The `*_ELGA` fields are defined on all sub-points
  - The `*_ELNO` fields are calculated from the `*_ELGA` fields (not needed for beams and pipes elements)
  - The `*_NOEU` fields are defined in a single layer, in a single point from `*_ELNO` fields (not needed for beams and pipes elements)
- **Change of reference with the `MODI_REPERE` command**

  (not needed for beams and pipes elements)

Visualization

- **Visualize the fields with sub-points**

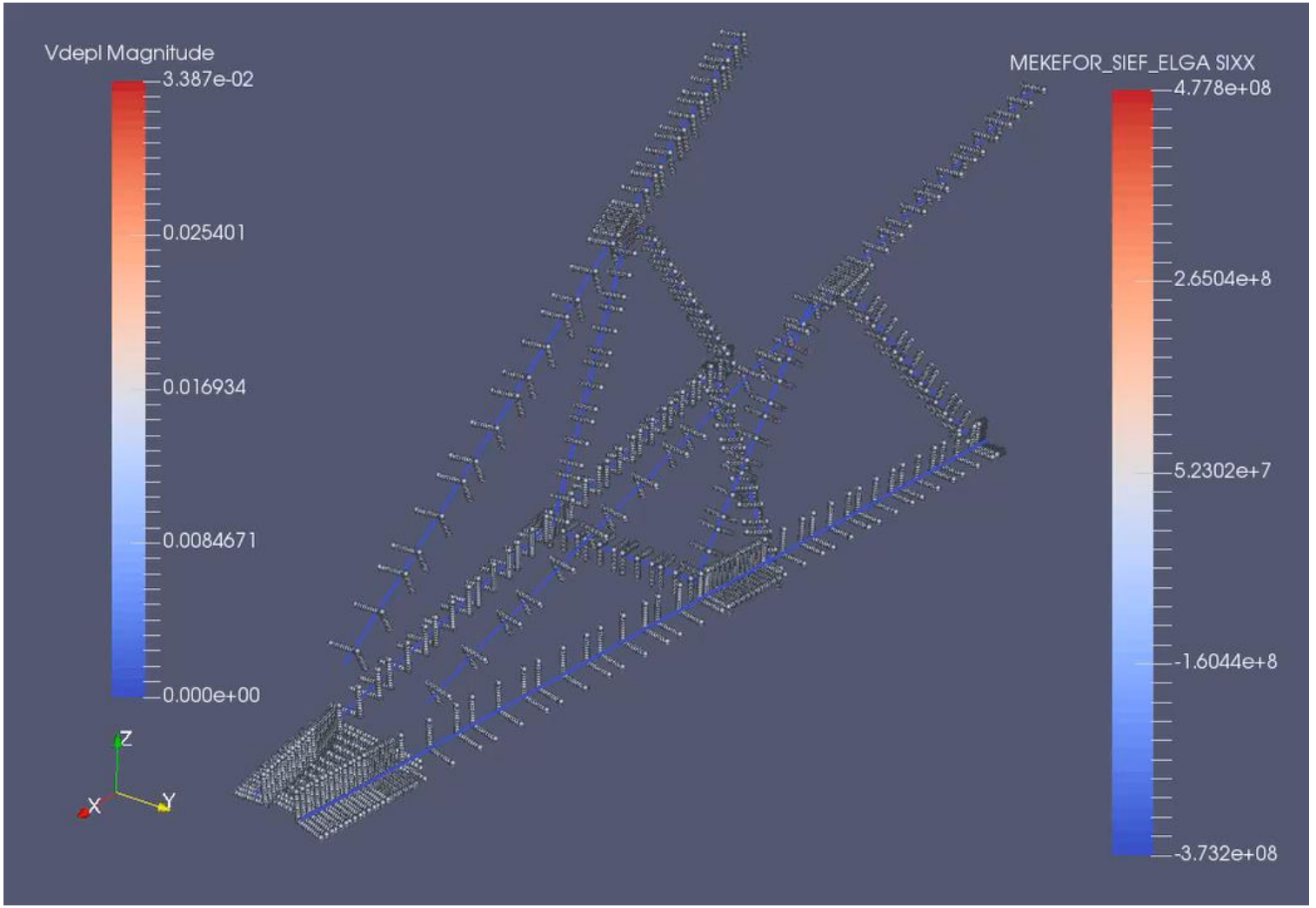



```
# Post-treatment
IMPR_RESU(FORMAT='MED', UNITE=41,
    RESU=_F(RESULTAT=RESU, CARA_ELEM=CAEL, NOM_CHAM =("SIEF_ELGA","EPSI_ELGA"),),
)

IMPR_RESU(FORMAT='MED', UNITE=42,
    RESU=_F(RESULTAT=RESU, CARA_ELEM=CAEL, NOM_CHAM ="VARI_ELGA", IMPR_NOM_VARI="NON" ),)
```

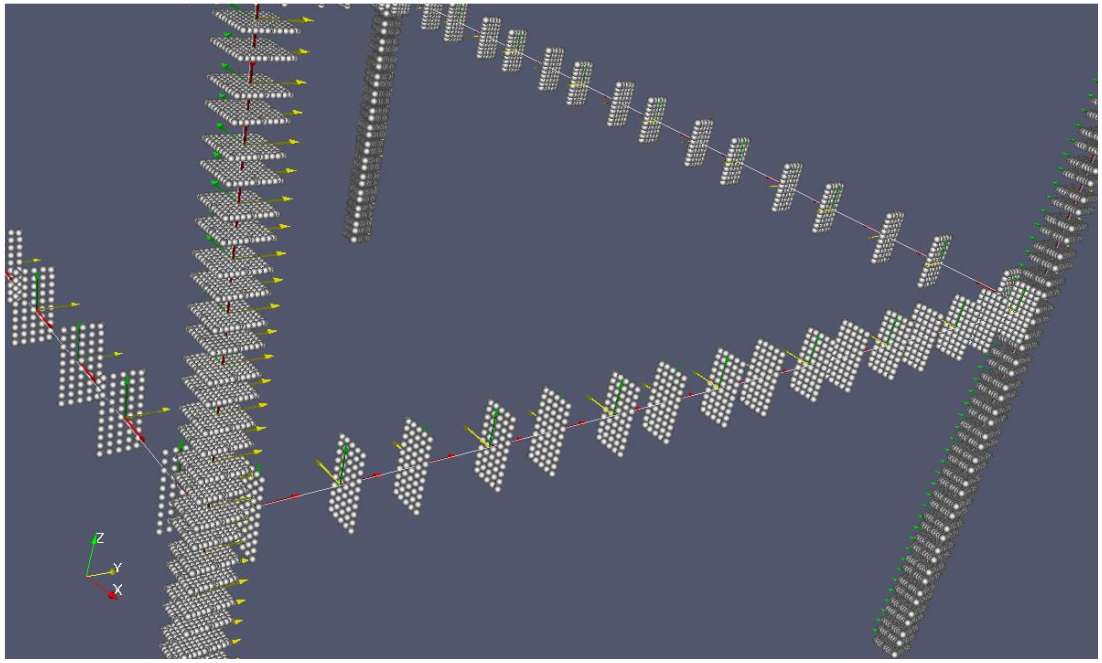
Visualization

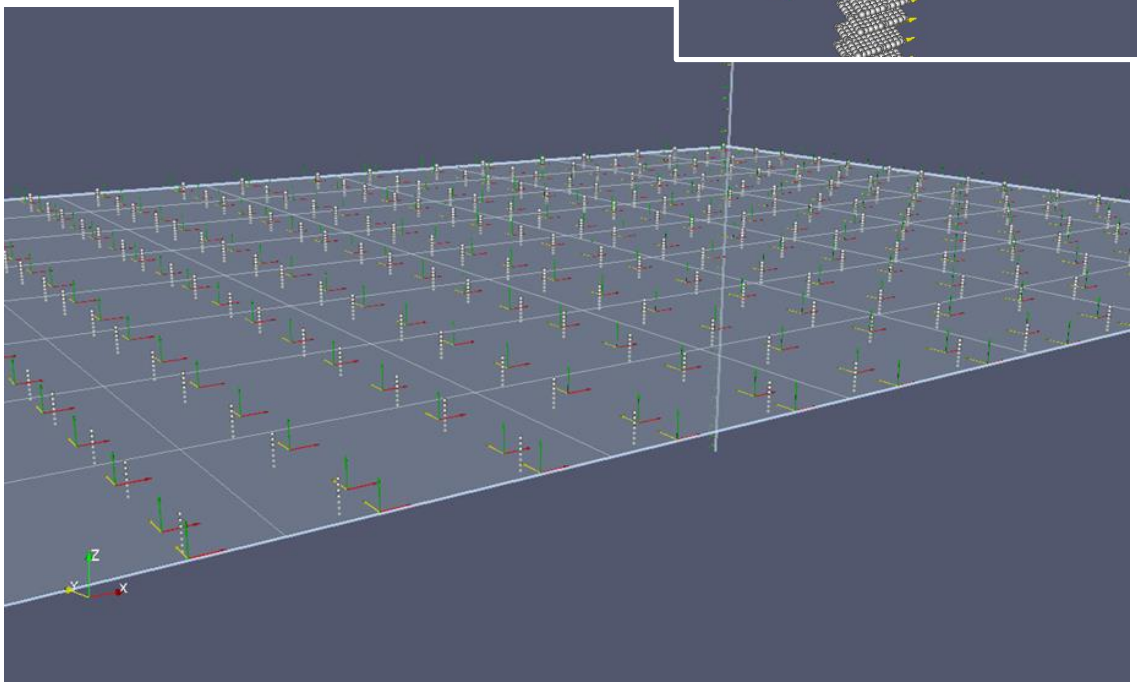
- **Time animation of the fields with sub-points**



Visualization

- **Check multi-fibers and multi-layers position and their local axis.**





End of presentation

Is something missing or unclear in this document?

Or feeling happy to have read such a clear tutorial?

Please, we welcome any feedbacks about Code_Aster training materials.

Do not hesitate to share with us your comments on the Code_Aster forum dedicated thread.