Operator **AFFE_MODELE**

1 Goal

To define the modelled physical phenomenon (mechanical, thermal or acoustic) and the type of finite elements.

This operator allows to affect modelings on whole or part of the grid, which defines:

- degrees of freedom on the nodes (and the equation or the associated conservation equations),
- types of finite elements on the meshes,

The possibilities of the finite elements being able to be selected are described in the booklets [U3].

The types of meshes are described in the document “Description of the file of grid of Code_Aster” [U3.01.00].

This operator also allows to define a distribution of the finite elements in order to parallel elementary calculations and the assemblies.

Product a structure of data of the type model.
2 Syntax

```
Mo [model] = AFFE_MODELE {

  ♦ GRID = my, / [grid]
  ♦ | AFFE = _F {
        ♦ | ALL = 'YES',
        ♦ | GROUP_MA = g_mail, [l_gr_maille]
  
  ♦ | PHENOMENON = 'MECHANICAL',
  ♦ | MODELING = ... (see §3.2.1)

  ♦ | PHENOMENON = 'THERMAL',
  ♦ | MODELING = ... (see §3.2.1)

  ♦ | PHENOMENON = 'ACOUSTIC',
  ♦ | MODELING = ... (see §3.2.1)

  },

  | AFFE_SOUS_STRUC = _F {
        ♦ | ALL = 'YES',
        ♦ | SUPER_MAILLE = l_mail, [l_maille]
  
  ♦ VERI_JACOBIEN = / 'YES' [DEFECT]
  ♦ / 'NOT'

  ♦ GRANDEUR_CARA = _F {
        ♦ LENGTH = will lcara, [R]
        ♦ PRESSURE = will pcara, [R]
        ♦ TEMPERATURE = will t cara, [R]

  ♦ DISTRIBUTION = _F {
        ♦ METHOD =
          ♦ / 'SOUS_DOMAINE' [DEFECT]
          ♦ NB_SOUS_DOMAINE = nb_proc [DEFECT]
          ♦ / nb_sous_dom
          ♦ PARTITIONNEUR = / 'MONGREL'

          ♦ / 'MAIL_CONTIGU'
          ♦ / 'MAIL_DISPERSE'

          ♦ INFORMATION = / 1 [DEFECT]
          ♦ / 2,
        
```

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.
Copyright 2020 EDF R&D - Licensed under the terms of the GNU FDL (http://www.gnu.org/copyleft/fdl.html)
3 Operands

3.1 Operand GRID

- GRID = my
  
  Name of the associated grid on which one affects the elements.

  **Note:**
  
  For axisymmetric modelings, the axis of revolution is the axis \( Y \) grid. All the structure must be with a grid in \( X \geq 0 \).

3.2 Keyword AFFE

- AFFE
  
  Defines the entities of the grid and the types of elements which will be affected for them. For each occurrence, one can introduce a list of modelings. The rule of overload applies between various modelings, from left to right.

  For example:
  
  \[
  \text{AFFE=\_F \{ Tout=' OUI', Phénomène=' MECANIQUE', \}
  \text{ Modélisation= ('AXIS', 'AXIS\_SI'), \}}
  \]

  Various modelings “overload” the ones the others: AXIS\_SI overload AXIS on the meshes where AXIS\_SI exist (mesh QUAD4 and QUAD8).

  **Note:**
  
  The code stops in error \(<F>\) if modelings of the list are not the very same “dimension” (for example \(\text{MODELISATION=} ('3D', 'D\_PLAN')\)). Moreover, for an occurrence of AFFE, the specified meshes whose dimension is that of the dimension of modeling must be all affected. If not the code emits one <A>alarm. This alarm protects the user who uses modelings “with holes”. If for example, it uses only modeling AXIS\_SI on a grid containing only TRIA6.

  The entities of the grid are specified by the operands:

<table>
<thead>
<tr>
<th>Operands</th>
<th>Contents/significance</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALL</td>
<td>Assignment with the totality of the meshes</td>
</tr>
<tr>
<td>GROUP_MA</td>
<td>Assignment with a list of groups of meshes</td>
</tr>
</tbody>
</table>

  The type of element is specified by the operands:

<table>
<thead>
<tr>
<th>Operands</th>
<th>Contents/significance</th>
</tr>
</thead>
<tbody>
<tr>
<td>PHENOMENON</td>
<td>Modeled physical phenomenon (associated conservation equation)</td>
</tr>
<tr>
<td>MODELING</td>
<td>Type of interpolation or discretization</td>
</tr>
</tbody>
</table>

3.2.1 Operands PHENOMENON and MODELING

- PHENOMENON
- MODELING

  Are obligatory for each occurrence of the keyword factor AFFE. This couple of keywords defines in a bijective way the type of affected element in a kind of mesh. Possible modelings are indicated below by listing them by “packages”:

Warning: The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.
ACOUSTICS
ACOUSTICS 2D continuous mediums
  PLAN U3.33.01

ACOUSTICS 3D continuous mediums
  3D U3.33.01

THERMICS
THERMICS 2D hull
  COQUE_AXIS U3.22.01
  COQUE_PLAN U3.22.01

THERMICS 2D continuous mediums
  AXIS_DIAG U3.23.01
  AXIS_FOURIER U3.23.02
  AXIS U3.23.01
  PLAN_DIAG U3.23.01
  PLAN U3.23.01

THERMICS 3D hull
  HULL U3.22.01

THERMICS 3D continuous mediums
  3D_DIAG U3.24.01
  3D U3.24.01

MECHANICS 2D
MECHANICS 2D discrete elements
  2D_DIS_TR
  2D_DIS_T

MECHANICS 2D fluid-structure
  2D_FLUIDE U3.13.03
  2D_FLUI_ABSO U3.13.13
  2D_FLUI_PESA U3.14.02
  2D_FLUI_PESA U3.14.03
  AXIS_FLUIDE U3.13.03
  AXIS_FLUI_STRU U3.13.03
  D_PLAN_ABSO U3.13.12

MECHANICS 2D continuous mediums
  AXIS U3.13.01
  AXIS_FOURIER U3.13.02
  AXIS_SI U3.13.05
  C_PLAN_SI U3.13.05
  C_PLAN U3.13.01
  D_PLAN_SI U3.13.05
  D_PLAN U3.13.01

MECHANICS 2D quasi incompressible
  AXIS_INCO_UP R3.06.08
  D_PLAN_INCO_UP R3.06.08
  AXIS_INCO_UPG U3.13.07 and R3.06.08
  D_PLAN_INCO_UPG U3.13.07 and R3.06.08
  AXIS_INCO_UFO R3.06.08
  D_PLAN_INCO_UPO R3.06.08
MECHANICS 2D **not room**
- D_PLAN_GRAD_VARI
- D_PLAN_GVNO: R5.04.04
- AXIS_GVNO: R5.04.04
- D_PLAN_GRAD_SIGM: R5.03.24

MECHANICS 2D **plates and hulls**
- COQUE_AXIS: U3.12.02

Mechanics 2D **elements joined for the propagation of crack**
- PLAN_JOINT: U3.13.14
- AXIS_JOINT: U3.13.14
- PLAN_JOINT_HYME: R3.06.09
- PLAN_INTERFACE: R3.06.13
- PLAN_INTERFACE_S: R3.06.13
- AXIS_INTERFACE: R3.06.13
- AXIS_INTERFACE_S: R3.06.13

Mechanics 2D **elements with internal discontinuities for the starting and the propagation of crack**
- PLAN_ELDI: U3.13.14
- AXIS_ELDI: U3.13.14

MECHANICS 2D **thermo-hydro-mechanics**
- AXIS_HH2MD
- AXIS_HH2MS
- AXIS_HHMD
- AXIS_HHMS
- AXIS_HHM: U3.13.08
- AXIS_HMD: U3.13.08
- AXIS_HMS
- AXIS_HM
- AXIS_THH2D
- AXIS_THH2S
- AXIS_THH2MD
- AXIS_THH2MS
- AXIS_THHD
- AXIS_THHS
- AXIS_THHMD
- AXIS_THHMS
- AXIS_THMD
- AXIS_THMS
- AXIS_THM: U3.13.08
- AXIS_HHD: R5.04.03
- AXIS_HHS: R5.04.03
- AXIS_HH2D: R5.04.03
- AXIS_HH2S: R5.04.03

- D_PLAN_HH2MD
- D_PLAN_HH2MS
- D_PLAN_HHMD
- D_PLAN_HHMS
- D_PLAN_HHM: U3.13.08
- D_PLAN_HMD
- D_PLAN_HMS
- D_PLAN_HM: U3.13.08
- D_PLAN_HM_P
- D_PLAN_THH2D
- D_PLAN_THH2S
For the grids 2D, allows to inform the groups of meshes or the meshes likely to be crossed by the crack when the contact is defined on the lips of the crack. Are allowed the following types of meshes: QUAD8 and TRIA6 and the meshes of edge of these elements, are them SEG3. If the meshes are linear, they should as a preliminary be transformed into quadratic meshes (with LINE_QUAD of the operator CREA_MAILLAGE).

### MECHANICS 3D

**bars and cables**

2D_BARRE

BAR U3.11.01

CABLE_POULIE U3.11.03

CABLE U3.11.03

CABLE_GAINE R3.08.10

**discrete elements**

DIS_TR U3.11.01

DIS_T U3.11.01

**fluid-structure**

3D_FAISCEAU

3D_FLUIDE U3.14.02

**absorbing border**

3D_ABSO U3.14.09

3D_FLUI_ABSO U3.14.10

**grids of concrete reinforcements**

GRILLE_MEMBRANE

GRILLE_EXCENTRE U3.12.04

**continuous mediums**

3D_SI U3.14.01

3D U3.14.01

**not room**

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.
3D_GRAD_VARI
3D_GVNO

MECHANICS 3D plates, hulls and membranes
COQUE_3D U3.12.03
DHT U3.12.01
DST U3.12.01
Q4G U3.12.01
DKTG U3.12.01
Q4GG U3.12.01
MEMBRANE U3.12.04

MECHANICS 3D beams
FLUI_STRU U3.14.02
LOUSE_FLUI_STRU U3.14.02
POU_D_EM U3.11.07
POU_D_E U3.11.01
POU_D_TGM U3.11.04
POU_D_TG U3.11.04
POU_D_T_GD U3.11.05
POU_D_T U3.11.01

MECHANICS 3D quasi incompressible
3D_INCO_UP U3.14.02
3D_INCO_UPG U3.14.06 and R3.06.08
3D_INCO_UPO R3.06.08

MECHANICS 3D thermo-hydro-mechanics
3D_HHMD
3D_HHM U3.14.07
3D_HMD U3.14.07
3D_HM U3.14.07
3D_THHD
3D_THHMD
3D_THHM U3.14.07
3D_THMD U3.14.07
3D_THVD
3D_THH2MD
3D_THH2M
3D_HH2MD
3D_HH2MS
3D_THH2S
3D_THH2D
3D_HHD R5.04.03
3D_HHS R5.04.03
3D_HS R5.04.03
3D_HH2D R5.04.03
3D_HH2S R5.04.03

MECHANICS 3D hydraulics unsaturated with finished volumes
3D_HH2SUDA

MECHANICS 3D pipes
TUYAU_3M U3.11.06
TUYAU_6M U3.11.06

MECHANICS 3D massive element of hull
SHB U3.12.05
For the grids 3D, allows to inform the groups of meshes or the meshes likely to be crossed by the crack when the contact is defined on the lips of the crack. Are allowed the following types of meshes: HEXA20, PENTA15, TETRA10, and the meshes of edges of these elements, are them QUAD8 and TRIA6. If the meshes are linear, they should as a preliminary be transformed into quadratic meshes (with LINE_QUAD of the operator CREA_MAILLAGE).

Mechanics 3D elements joined for the propagation of crack

- 3D_JOINT
- 3D_JOINT_HYME
- 3D_INTERFACE
- 3D_INTERFACE_S
3.3 **Keyword AFFE_SOUS_STRUC**

<table>
<thead>
<tr>
<th>AFFE_SOUS_STRUC</th>
</tr>
</thead>
</table>

Is usable only for one using model of the static substructures [U1.01.04].

SUPER_MAILLE = _mail

_l_mail_ is the list of the super-meshes which one wants to affect in the model. As for the finite elements, it is not obligatory to affect all the meshes of the grid. It is AFFE_MODELE who confirms which are the substructures which will be used in the model. The difference with the classical finite elements is that on the super-meshes, one does not choose nor MODELING nor PHENOMENON because the macronutrient (built by the operator MACR_ELEM_STAT [U4.62.01]) who will be affected on the super-mesh has his own modeling and his own phenomenon (those which were used to calculate it).

Caution! Your model must contain at least a finite element (keyword AFFE with the §3.2) when you use definite static substructures starting from a physical grid (read by LIRE_MAILLAGE) because it is not possible to have only macronutrients in this case.

/ ALL = ‘YES’

All them (super) meshs are affected.

3.4 **Operand VERI_JACOBIEN**

| VERI_JACOBIEN = ‘YES’/‘NOT’ |

This keyword is used to check that the meshes of the model are not distorted too much. One calculates the jacobien of the geometrical transformation which transforms the element of reference into each real mesh of the model. So on the various points of integration of a mesh, the jacobien changes sign, it is that this mesh is very “badly rotten”. An alarm (CALCULEL_?) is then emitted.

3.5 **Operand GRANDEUR_CARA**

| GRANDEUR_CARA = _F (LENGTH = will lcara,…) |

This keyword is used to define some physical sizes characteristic of with the dealt problem. These sizes are currently used “have-to dimension” certain terms of the estimators of error in “HM”. See [R4.10.05].

3.6 **Keyword DISTRIBUTION**

| DISTRIBUTION = _F ( METHOD = methoof,…) |

This keyword makes it possible to distribute the finite elements of the model for the parallelism of elementary calculations, the assemblies and certain linear solveurs. Cf. [U2.08.06] “Note of use of parallelism”.

It defines how (or not) the meshes/elements for the phases paralleled will be distributed of Code_Aster. The user thus has the possibility of controlling this distribution between the processors.

Parallelism operates:

• on elementary calculations and the assemblies of matrices and vectors (it is what the keyword factor DISTRIBUTION allows to control),

• with the resolution of the linear system if the solvor is paralleled (cf. [U4.50.01]).
Note:
It is possible to modify the mode of distribution during its study. It is enough to use the order MODI_MODELE [U4.41.02].

Note:
It can be practical to continue a parallel calculation with a number of processors different from that used for initial calculation. In particular, one can want to carry out certain postprocessings into sequential. It is recommended to use the order MODI_MODELE to define the distribution to be used in continuation. More precisely, Lorsque initial calculation used it parallelism “by groups of elements” (‘GROUP_ELEM’ or ‘SOUS_DOMAINE’), the order MODI_MODELE is useless. On the other hand, Lorsque initial calculation has used it parallelism “by elements” (‘MAIL_CONTIGU’ or ‘MAIL_DISPERSE’), the order MODI_MODELE is obligatory. If it is forgotten, one is stopped during calculation by one error message.

3.6.1 Keyword METHOD

3.6.1.1 METHOD =/’CENTRALIZES’

Parallelism starts only on the level of the linear solver. Each processor builds and provides to the solver the entirety of the system to be solved. Elementary calculations are not paralleled.

3.6.1.2 METHOD =/’GROUP_ELEM’

CE mode of distribution allows a perfect balancing of load (in term of numbers of calculations elementary) a priori, i.e. each processor will carry out, for a kind of element given, the same number of elementary calculations with near. Obviously that does not prejudice of anything the final balancing of load in particular in non-linear calculations where the cost of an elementary calculation depends on other parameters but the type of element.

In this mode, the elements of the model are gathered by “group” in order to pool certain calculations which makes it possible to gain in effectiveness. The number of elements by group can be selected in the order BEGINNING [U4.11.01].

In addition, it is a question of the only mode able of distributing elementary calculations induced by the late elements, i.e. by the loadings such as the boundary conditions dualized or the continuous contact.

3.6.1.3 METHOD =/’MAIL_DISPERSE’

The distribution takes place on the meshes. They are distributed equitably on the various processors available. The meshes are distributed on the various processors as it is made it when one distributes cards to several players. One also speaks about “cyclic” distribution.

For example, with a model comprising 8 meshes, carried out on 4 processors, one obtains the following distribution:

<table>
<thead>
<tr>
<th>Mode of distribution</th>
<th>Mesh 1</th>
<th>Mesh 2</th>
<th>Mesh 3</th>
<th>Mesh 4</th>
<th>Mesh 5</th>
<th>Mesh 6</th>
<th>Mesh 7</th>
<th>Mesh 8</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAIL_DISPERSE</td>
<td>Proc. 0</td>
<td>Proc. 1</td>
<td>Proc. 2</td>
<td>Proc. 3</td>
<td>Proc. 0</td>
<td>Proc. 1</td>
<td>Proc. 2</td>
<td>Proc. 3</td>
</tr>
</tbody>
</table>

It is seen that with this mode of distribution, a processor will treat meshes regularly spaced in the order of the meshes of the grid. The advantage of this distribution is that “statistically”, each processor will treat as many hexahedrons, of pentahedrons,.., and of triangles.

The workload for elementary calculations in general will be well distributed. On the other hand, the matrix assembled on a processor “will be very dispersed”, contrary to what occurs for the mode ‘MAIL_CONTIGU’.

3.6.1.4 METHOD =/’MAIL_CONTIGU’

The distribution takes place on the meshes. They are divided into packages of contiguous meshes on various processors available.

Warning : The translation process used on this website is a "Machine Translation". It may be imprecise and inaccurate in whole or in part and is provided as a convenience.
Copyright 2020 EDF R&D - Licensed under the terms of the GNU FDL (http://www.gnu.org/copyleft/fdl.html)
For example, with a model comprising 8 meshes, a machine of 4 processors available, the following distribution is obtained:

<table>
<thead>
<tr>
<th>Mode of distribution</th>
<th>Mesh 1</th>
<th>Mesh 2</th>
<th>Mesh 3</th>
<th>Mesh 4</th>
<th>Mesh 5</th>
<th>Mesh 6</th>
<th>Mesh 7</th>
<th>Mesh 8</th>
</tr>
</thead>
<tbody>
<tr>
<td>MAIL_CONTIGU</td>
<td>Proc. 0</td>
<td>Proc. 0</td>
<td>Proc. 1</td>
<td>Proc. 1</td>
<td>Proc. 2</td>
<td>Proc. 2</td>
<td>Proc. 3</td>
<td>Proc. 3</td>
</tr>
</tbody>
</table>

For this mode of distribution, the workload for elementary calculations can be less balanced. For example, a processor can have to treat only “easy” meshes of edge. On the other hand, the matrix assembled on a processor is in general more compact.

### 3.6.1.5 Keyword CHARGE_PROC0_MA

\[
\text{\texttt{\textbf{CHARGE\_PROC0\_MA}}} = \text{\texttt{/100}} \text{ [DEFECT]} / \text{pct}
\]

This keyword is accessible only for the modes from distribution 'MAIL\_DISPERSE' and 'MAIL\_CONTIGU'. Indeed these modes of distribution do not distribute in general equitably the load of calculations because of boundary conditions dualized whose elementary calculations are treated by processor 0.

If one wishes to relieve processor 0 (or on the contrary to overload it), one can use the keyword CHARGE_PROC0_MA. This keyword makes it possible to the user to choose the percentage of load which one wishes to assign to processor 0.

For example, if the user chooses \texttt{CHARGE\_PROC0\_MA} = 80, processor 0 will treat 20% of elements of less than the other processors, is 80% of the load which it should support if the division were equitable between the processors.

### 3.6.1.6 METHOD = /'SOUS\_DOMAINE' [DEFECT]

This distribution of the meshes is based on a decomposition grid under-fields, built by a tool external of partitioning defined by the keyword PARTITIONNEUR.

\[
\text{\texttt{\textbf{PARTITIONNEUR}}} = \text{\texttt{/\texttt{MONGREL}}} \text{ \texttt{[DEFECT]}} / \text{\texttt{SCOTCH TAPE}}
\]

The number of under-fields can be given by the user, via the keyword \texttt{NB\_SOUS\_DOMAINE}.

\[
\text{\texttt{\textbf{NB\_SOUS\_DOMAINE}}} = \text{\texttt{/nbproc}} \text{ \texttt{[DEFECT]}} / \text{\texttt{nb\_sous\_dom}}
\]

By default, the number of under-fields is taken equal to the number of processors implied in calculation (nbproc).

The elements of the model finite elements carried by the meshes of each under-field are then distribute by groups of similar elements (as in the distribution corresponding to the method GROUP_ELEM), in order to balance elementary calculations as well as possible.

The preliminary partitioning of the grid under-fields makes it possible to ensure that all the elements of a group of finite elements belong to only one under-field.
4 Production run

Starting from the keywords PHENOMENON and MODELING, one creates a structure of data specifying the type of element attached to each mesh.

A brief recall of the assignments is systematically printed (INFO=1) in the file message.

For example:

ON          612 MESHS OF THE GRID MY
ONE WITH REQUEST THE ASSIGNMENT OF          612
ONE WITH PU TO AFFECT SOME                    612

<table>
<thead>
<tr>
<th>MODELING</th>
<th>FINITE ELEMENT</th>
<th>TYPE NETS</th>
<th>NUMBER</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>MECA_TETRA4</td>
<td>TETRA4</td>
<td>52</td>
</tr>
<tr>
<td>3D</td>
<td>MECA_PENTA6</td>
<td>PENTA6</td>
<td>16</td>
</tr>
<tr>
<td>...</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3D</td>
<td>MECA_FACE3</td>
<td>TRIA3</td>
<td>60</td>
</tr>
</tbody>
</table>

5 Example

Mo = AFME_MODELE
    ( GRID = my,
      AFFE = ( _F ( GROUP_MA = gma,
                  PHENOMENON = 'MECHANICAL',
                  MODELING = '3D' ),
    ) )

For a modeling of the phenomenon 'MECHANICAL', one affects on the group of meshes gma elements 3D isoparametric.