

## To introduce a new modeling into AFFE\_MODELE

---

### Summary:

This document describes what it is necessary to do to introduce a new modeling into the operator AFFE\_MODELE of Code\_Aster .

In a few words, it is necessary:

- To add modeling to the catalogue of the order AFFE\_MODELE,
- To introduce a small block of text into the catalogue of the phenomena and modelings,
- To write one or more catalogues of elements.
- To write the routines of elementary calculations specific to the elements of this new modeling.

The object of this document is mainly to present the catalogue `Commons/phenomenons_modelisations.py` and the general structure of a catalogue of `Elements/xxx.py`.

The rest of the actions to be made is described in the document [D5.02.05] "To introduce a new elementary calculation".

## Contents

---

<a href="#">1 Introduction.....</a>	<a href="#">3</a>
<a href="#">2 Addition of modeling to the catalogue of order.....</a>	<a href="#">3</a>
<a href="#">3 Modification of the catalogue of the phenomena and modelings.....</a>	<a href="#">3</a>
<a href="#">3.1 Presentation of the catalogue phenomenons_modelisations.py.....</a>	<a href="#">3</a>
<a href="#">3.1.1 Arguments: modeling, dim and code :.....</a>	<a href="#">4</a>
<a href="#">3.1.2 Argument attrs.....</a>	<a href="#">4</a>
<a href="#">3.1.3 Argument elements.....</a>	<a href="#">5</a>
<a href="#">3.2 Introduction of a new modeling into the catalogue phenomenons_modelisations.py.....</a>	<a href="#">5</a>
<a href="#">4 Creation of the catalogues of elements.....</a>	<a href="#">7</a>
<a href="#">4.1 Summary presentation of a catalogue of elements.....</a>	<a href="#">7</a>

## 1 Introduction

---

The choice of modeling is carried out through the operator `AFFE_MODELE` of *Code\_Aster*. For example, the user will write in his command file:

```
KID = AFFE_MODELE (GRID = E-MAIL,  
                  AFFE=_F (ALL = 'YES', MODELING = 'AXIS_JOINT_HMS',  
                           PHENOMENON = 'MECHANICAL'))
```

In the objective to be able to propose to the user of other modelings, one will describe in this document a methodology to introduce a new modeling into *Code\_Aster*.

To introduce a modeling into *Code\_Aster* require to put the following questions:

- In which phenomenon I will add my modeling?
- Which are geometrical and topological dimensions finite elements?
- Which are the meshes concerned with this modeling?
- Which are the "principal" elements and the elements of edge?
- Which are the attributes which one can define?
- Which are realizable calculations with this modeling?

We will answer these questions in this document.

Other relative questions with the finite elements are treated in other documents:

- [D5.02.01] How to introduce a new size or new components (CMP) in an existing size?
- [D5.02.02] How to introduce a new type of mesh (`type_maille`) or a new element of reference (`ELREFE`) ?
- [D5.02.05] How to introduce a new elementary calculation?

## 2 Addition of modeling to the catalogue of order

---

- Catalogue to modify: `code_aster/Catastrophes/Commands/affe_modele.py`

New modeling is to be added to the whole of modelings usable by the order `AFFE_MODELE`. It is enough to add it to the list of possible (field `into`).

## 3 Modification of the catalogue of the phenomena and modelings

---

- Catalogue to modify: `Commons/phenomenons_modelisations.py`

### 3.1 Presentation of the catalogue `phenomenons_modelisations.py`

This catalogue breaks up into several parts: a part dedicated to each phenomenon: mechanics, thermics, acoustics,...

For each phenomenon, there exists a block corresponding to each modeling. For example, for modeling `'AXIS_SI'phenomenon'MECHANICS'`, we have:

```
phen.add ('AXIS_SI', Modeling (dim= (2.2), code=' AXS',
```

*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 2021 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

```
attrs= (  
  (AT.AXIS, 'YES'),  
  (AT.NBSIGM, '4'),  
  (AT.TYPMOD, 'AXIS'),  
) ,  
elements= (  
  (MT.QUAD8      , EL.MEAXQS8),  
  (MT.SEG3       , EL.MEAXSE3),  
) )
```

We will present the arguments of this block:

- modeling
- dim
- code
- attrs
- elements

### 3.1.1 Arguments: modeling, dim and code :

- The argument `modeling` provides the name of modeling. In the example above, it is modeling `'AXIS_SI'`.
- The argument `dim` (2 entreties) respectively provides topological dimension and the geometrical dimension of modeling:
  - Geometrical dimension corresponds to dimension of ambient space (3 or 2),
  - topological dimension corresponds corresponding to “principal” elements of modeling.

In this example, topological dimension is identical to geometrical dimension, but it is not always the case.

For example, a modeling `'DKT'` (thin hulls) collects elements whose meshes support are of dimension 2 (triangles, quadrangles), however the nodes of the grid are expressed in the reference mark 3D (according to `X`, `Y`, `Z`). One thus has for this modeling: `dim= (2 . 3)`

- The argument `codes` provides as its name indicates it, a code. It is about a chain of 3 characters making it possible to identify modeling. For modeling `'AXIS_SI'`, the selected code is `'AXS'`. This “code” is inevitably different for all modelings. It is one alias (on 3 characters exactly) of the name of modeling.

### 3.1.2 Argument `attrs`

The argument `attrs` allows to define the attributes carried by all the elements of modeling.

Example: `attrs= (`  
 `(AT.AXIS, 'YES'), ...`

The list `attrs` is made of couples (attribute, value)

For what are used they?

They make it possible to provide information in source FORTRAN and to consider treatments according to this information. In the example above, the attribute `AT.AXIS=OUI` can be questioned in an elementary routine of calculation in order to modify the weight of integration of the points of Gauss.

The attributes are also used to gather elements which are joint: for example, elements of “beam”, “hull”,...

These regroupings are used in the blocks “ CondCalcul ” catalogues of options.

The attributes all are defined in the catalogue `Commons/attributes.py`

It is very important to document these attributes well so that their direction is the least ambiguous possible.

### **Note:**

The various attributes defined in this catalogue are assigned to the whole of `type_element` modeling. If it is wanted that an attribute is associated only with one `type_element`, it is then necessary to define this attribute in the catalogue of `Type_element`.

Caution: When a finite element is used by several modelings (it is often the case of the elements of edge), there can be ambiguity on the value of the attributes of this finite element. The element will inherit ALL the attributes defined in the whole of modelings which use it. If the same attribute is defined several times with different values, one affects the value to him “ ### ”. It is always the case of the attribute `MODELI`.

How to recover the value of an attribute in source FORTRAN?

Routines `lteatt.F90` and `teattr.F90` give access the attributes of one `type_element`.

## 3.1.3 Argument elements

Example:

```
elements= (  
    (MT.QUAD8      , EL.MEAXQS8) ,  
    (MT.SEG3      , EL.MEAXSE3) ,  
)
```

This argument is used to define all the elements of modeling (and of its edge). It is a list of couples (`type_maille, type_element`).

In our example, the couple: `(MT.QUAD8 , EL.MEAXQS8) ,`

mean that one allots for this modeling, the element of the type `MEAXQS8` with the quadrangular meshes with 8 nodes of the type `QUAD8`.

## 3.2 Introduction of a new modeling into the catalogue `phenomenons_modelisations.py`

First of all, it is necessary to be placed in the part corresponding to the phenomenon of your modeling (`MECHANICS` , `THERMIQUEU` or `ACOUSTIQUE`). Then place with the writing of the block corresponding to your modeling.

You must start with:

- to choose a name for your modeling (with more the 16 characters),
- to allot a code to your modeling (3 characters exactly),
- to determine “dimensions” of your modeling.

You must then ask you the question of the attributes carried by the elements of the new modeling. It is important to traverse the list of all the existing attributes to know if your elements are concerned with these attributes. For example, it would be an error not to declare `AT.AXIS=' OUI '` for an axisymmetric element.

The following stage consists in choosing:

- types of meshes which you wish to associate with your modeling. You can consult the catalogue `Commons/mesh_types.py` to become acquainted with the types of meshes present in *Code\_Aster* as well as their elements of reference ,
- the name of the type of the finite element which you wish to associate with each type of mesh. You must determine a name of with more the 16 characters which is sufficiently explicit to know the type of its mesh to the reading of its name.

We have just answered some questions concerning new modeling.

To go further, it is necessary to write it (or them) catalogues (S) describing the new finite elements of modeling, as well as routines FORTRAN `te00ij.f` carrying out their elementary calculations.

In the following paragraph, we describe (rather briefly) the catalogue of `type_element`.

For further details on this catalogue and the writing of associated routines FORTRAN, one will refer to the document [D5.02.05] "To introduce a new elementary calculation"

## 4 Creation of the catalogues of elements

- The catalogues of elements are localised in ... /catalo/cataelem/Elements
- A catalogue makes it possible to describe a family of "close" elements (for example various elements of the same modeling). The catalogue is called after the first element which there figure.

### 4.1 Summary presentation of a catalogue of elements

We will present the broad outlines. For more information, the reader is invited to consult documentation D5.02.05 (" *to introduce an elementary calculation* ").

We will present an extract of the catalogue `ther_hexa20.py` . We will propose the option of calculation `FLUX_ELGA` who allows to calculate the heat flux at the points of Gauss of the element starting from the field of temperature.

```
...  
  
MVECTTR = ArrayOfComponents (phys=PHY.VTEM_R, locatedComponents= (DDL_THER  
  
MMATTTR = ArrayOfComponents (phys=PHY.MTEM_R, locatedComponents= (DDL_THER  
DDL_THER))  
  
#-----  
class THER_HEX20 (Element):  
    meshType = MT.HEXA20  
    elrefe = (  
        ElrefeLoc (MT.H20, gauss = ('RIGI=FPG27', ..., ), ...),  
    )  
    calculations = (  
        ...  
  
        OP.FLUX_ELGA (te=62,  
            para_in= ((SP.PCAMASS, CCAMASS), (SP.PGEOMER, NGEOMER),  
                    (SP.PMATERC, LC.CMATERC), (SP.PTEMPER, DDL_THER),  
                    (SP.PTEMPSR, CTEMPSR), (OP.FLUX_ELGA.PVARCPR, LC.ZVARC  
            )),  
            para_out= ((OP.FLUX_ELGA.PFLUXPG, EFLUXPG),),  
        ),  
        ...  
    )
```

#### Text 1: Extracted the catalogue `ther_hexa20.py`

- One starts by describing the local modes specific to the elements described in this file ( `MVECTTR`, `MMATTTR`, ... )
- All the elements described in this file, are classes python which inherit the class `Element` . `THER_HEX20` being the first element of the file, it is him who gives the name of the file. Other elements of the file ( `THER_PENTA15`, `THER_TETRA10`, ...) are obtained by heritage of `THER_HEX20` .
- First lines:

```
class THER_HEX20 (Element):  
    meshType = MT.HEXA20  
    elrefe = (  
        ElrefeLoc (MT.H20, gauss = ('RIGI=FPG27', ..., ), ...),  
    )
```

Allow to declare the type of mesh associated with the element ( MT.HEXA20 ) and its (or its) elements of reference ( elrefe = (...) )

- Then the list comes from all the elementary calculations carried out by the element (argument calculations). Elements of the tuple calculations are elementary calculations.  
For example for FLUX\_ELGA :

```
OP.FLUX_ELGA (te=62,  
    para_in= ((SP.PCAMASS, CCAMASS), (SP.PGEOMER, NGEOMER),  
    ...  
    ),  
    para_out= ((OP.FLUX_ELGA.PFLUXPG, EFLUXPG), ),  
),
```

The number of the routine is indicated te00ij (here te0062. F90 ) associated with elementary calculation, as well as the description of and the exit inlet limits of elementary calculation. The description of a field is made by associating (in a couple) the parameter of the option and the local mode which is associated for him.