

Implementation of the approach “great slips with X-FEM”

Summary:

This document describes implementation the data-processing of the approach “great slips with X-FEM”. It is a question of enriching modeling by the structures fissured by the X-FEM by giving him the possibility of taking into account the contact on the level of the interfaces, under great displacements, but still under the assumption of the small deformations. The scientific formulation of this approach is presented in the handbook [R5.03.53] and his first reading highly is recommended. In this document one presents mainly the data-processing routines which contribute to the digital implementation of this approach (news routines, old routines which was modified or just old routines which was not modified but which contributes in a general way to modeling X-FEM). One also gives the trees of call of these routines, a very important tool when one launches out in the development of Code_Aster.

The digital implementation of this approach in Code_Aster relates to the cases 2D and 3D.

1 Introduction

This document describes the digital implementation in Code_Aster of the approach "Great slips with X-FEM". One presents in particular the routines FORTRAN which build the operators of Code_Aster necessary to the construction of model X-FEM, as well as the tree structure of these operators.

The document is compiled for the people who will continue this work (trainees, doctorands, or post-doctorands) and to make maintenance of it. This reading is to be supplemented by the comments which are inserted a little everywhere in routines FORTRAN (news or just modified).

After this introduction, the six chapters which follow respect the order of the operators to be defined in a command file of Code_Aster in order to carry out an analysis by using the approach "Great slips with X-FEM". In each chapter corresponding to an operator, one also presents his tree of call where only the routines relating to this approach are illustrated.

One encloses with the presentation of the developmental perspectives to make this new approach most general possible.

Important remarks:

- 1) The digital implementation of this approach relates to the cases 2D (linear and quadratic) and 3D (linear). The treatment of the elements of funds of crack and multi-fissured is possible. The terms of friction were implemented in all the cases except for the multi-fissured elements.
- 2) This approach was the object of an evolution of Code_Aster controlled by the cards (10708, 10751.10752 and 10753,11957,11958,12608,10556,13178,13179 and 15559). It is operational in the code since restitution 9.1.14 of version STA9.1 and was put up to date since in version STA10.0.

2 Introduction of the cracks

The introduction of cracks X-FEM into a model of Code_Aster is done using the operator `DEFI_FISS_XFEM` [U4.82.08]. This operator did not undergo modifications/additions following the implementation of the approach *great slips with X-FEM*. However, it is important to give here short descriptions for its routines which build in fact the structure of data `FISS_XFEM` [D4.10.02]. The developer will be able to be thus found there if ever it wants to modify or enrich these structures of data (SD). Throughout this document, the reader will meet name "routine hat". It is about a routine which is only used to launch arithmetic operations by calling other routines.

Below, one presents an example of introduction of a crack, by analytical functions level-sets, for an analysis X-FEM. The catalogue of the order (`defi_fiss_xfem.capy`) rather frequently evolve, therefore one recommends his consultation before the drafting of the command file.

```
LSN=FORMULE (NOM_PARA= ('X', 'Y'), VALE=' Y-2.5');  
LST=FORMULE (NOM_PARA= ('X', 'Y'), VALE='-X-10. ');  
  
FISS1=DEFI_FISS_XFEM (MODELE=MODELEIN,  
                     DEFI_FISS=_F (FONC_LT=LST, FONC_LN=LSN, ),  
                     INFO=1, );
```

The description of routines FORTRAN is given below and the tree of call is illustrated on Figure 1.

OP0041 - routine of launching of the operator which carries out mainly the reading of the data introduced into the command file and the launching of the various routines which follow.

XLORIE – routine for the reading of data concerning the bottom of crack. Parameters introduced by the user under the keywords 'RAYON_ENRI' and 'NB_COUCHES' are read here in order to fill the SD `FISS//'.CARAFOND'` with the ray of enrichment around the head of crack and the number of layers of elements to be enriched.

XLENRI – the reading of the list of potentially enriched meshes, list provided by the user realizes, and creates the SD: `FISS//'.GROUP_MA_ENRI'` and `FISS//'.GROUP_NO_ENRI'`.

XINILS – routine hat to launch the calculation of the level-sets. If the level-sets are introduced by analytical functions, calculation is done directly in this routine and one fills the SD: `FISS//'.LNNO'` and `FISS//'.LTNO'`

XINLSJ – routine for the recovery of the levels set junction. It is called if the user defined the operand `JUNCTION` in `DEFI_FISS_XFEM`. In this case one recovers locally the levels set normals and tangents of the cracks on which `FISS` connects in one `cham_no`. These levels set are then used in the routine `XSTANO` for the calculation of the statute of enrichment of the nodes. To store the total information in order to feed the operator `MODI_MODELE_XFEM`, the fields here are created `FISS//'.JONFISS'` and `FISS//'.JONCOEF'` who contain the list of the cracks on which `FISS` connects and associated coefficients.

XLS3D – routine for the calculation of the level-sets if the crack is introduced by surface meshes (case 3D).

XLS2D - routine for the calculation of the level-sets if the crack is modelled by linear meshes (case 2D).

XAJULS – routine for the readjustment of the level-sets. The values of the LS are modified if they are too close to zero to avoid the errors of digital integration due to the bad conditioning of the matrix of rigidity.

XGRALS – routine for the launching of the calculation of the gradients of the level-sets. One leaves with the SD: `FISS//'.GRLNNO'` and `FISS//'.GRLTNO'`.

TE0024 – the elementary calculation of the gradients of the level-sets (option realizes 'GRAD_NEUT_R').

XENRCH – launching of the calculation of the enrichment and the points of the bottom of crack.

XMAFIS – routine to find the meshes where the level-set normal changes sign. One leaves with the list the fissured meshes.

XSTANO – the statute (enrichment) determines nodes (HEAVISIDE, ACE TIP or HEAVISIDE and ACE TIP). The SD is filled FISS//`.STNO' .

XSTAMA – calculation of the statute of the meshes and possible actualization of the statutes of the nodes (if enrichment by layer).

XSTAM1 – calculation of the statute of the meshes according to the statute of the nodes.

XPTFON – research points of the funds of crack.

XORIFF – orientation of the point of the funds of crack.

XLMAIL – creation and filling of the SD: FISS//`.MAILFISS .INDIC', FISS//`.MAILFISS .HEAV', FISS//`.MAILFISS .CTIP', FISS//`.MAILFISS .HECT', FISS//`.FONDFISS', FISS//`.FONDMULT' .

XBASLO - creation of the SD FISS//`.BASLOC' who contains the local base at the points of the associated bottom of crack.

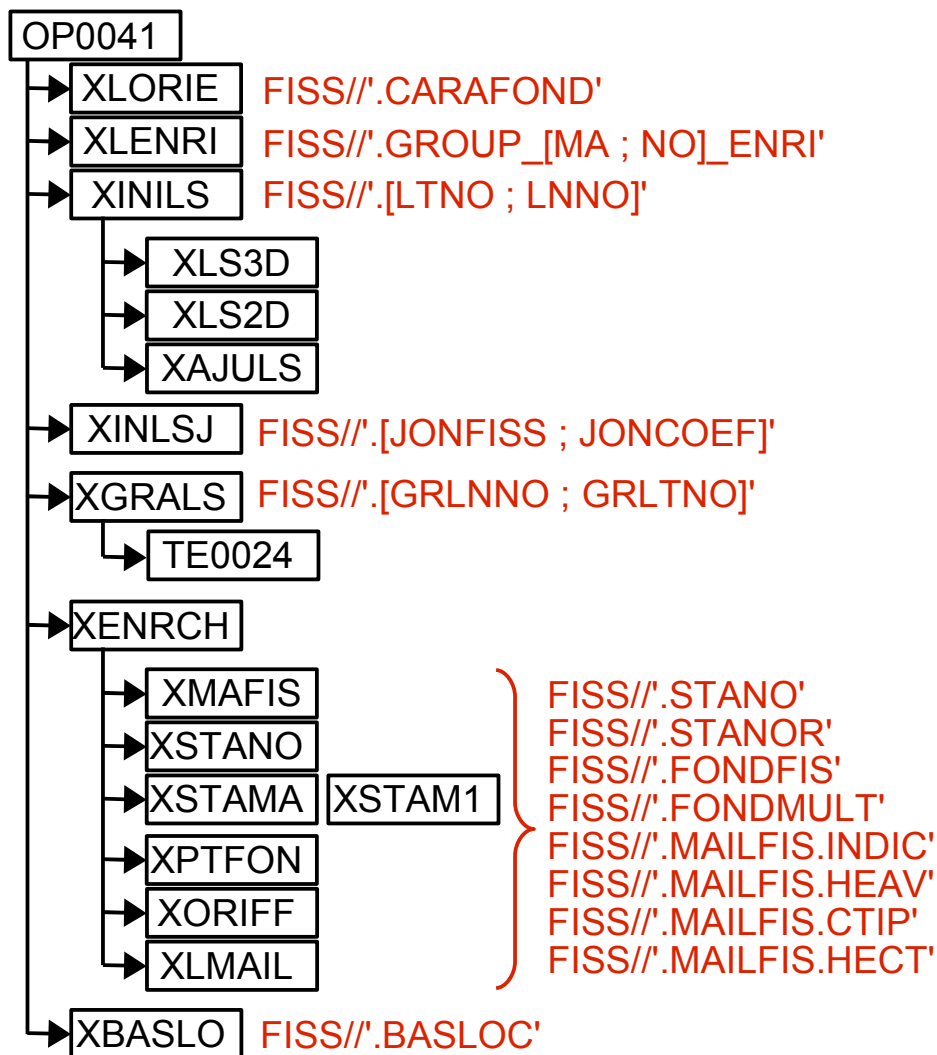


Figure 1. The tree of call of the operator DEFI_FISS_XFEM

3 Enrichment of the model

It is the operator `MODI_MODELE_XFEM` [U4.41.11], developed especially for the analysis by X-FEM [R7.02.12] with the goal to modify certain classical finite elements in order to transform them into finite elements nouveaux riches. The purpose of this operator is in particular to prepare the model with a calculation X-FEM into which the user would have introduced several cracks. The cracks can possibly cut out the same mesh to see connecting one on the other. One calls on routine `XCONNO` who carries out the concatenation of SD X-FEM defined by crack, the principle being of having SD defined on the model, for the whole of the cracks. These concatenées SD are used then for cutting of the elements under elements (`XTOPOI`) and to determine the facets of contact (`XTOPOC`).

For the approach *great slips with X-FEM*, the modifications made here relate to the topology of the facets of contact. More exactly, contents of the object `MODEL`.TOPOFAC.PI'` (see [D4.10.02]) was changed: it does not store any more the real coordinates of the points of intersection, but the coordinates of reference in the element relative. One adds at the same time two new objects: `MODEL`.TOPOFAC.GE'` and `MODEL`.TOPOFAC.GM'` who will store the real coordinates of the points of intersection for the facets of contact, respectively *slave* and *Master*. One thus duplicated the facets of contact compared to the implementation HP. By convention one indicates like facet slave that formed by points of intersection attached to the side of the negative LSN (LSN<0 with the nodes nouveaux riches), while the facet Master is formed by the points of intersection attached to the side of the positive LSN (LSN>0 with the nodes nouveaux riches). It is observed that the user has the possibility of playing with the statute of the facets by changing the sign of level-set LSN in the command file. If the crack is not introduced by an analytical function but by surface meshes, the change of the statute main slave/is possible by the change of order of the nodes which compose these meshes. This last operation can prove to be expensive and thus the introduction of a new keyword is possible to allow the user the easy choice on the positive sides and negative, directly in the command file. For more details on calculation of the level-sets for the two methods of introduction of a crack, to see chapter 2.2 of the reference material of the X-FEM [R7.02.12].

Several SD which are mentioned here are attached to `MODEL` and presented in detail in the handbook [D4.06.02] "Structures of Data ligrel and model".

Here an example of call of operator X-FEM in the command file:

```
MODELEK=MODI_MODELE_XFEM (MODELE_IN=MODELEIN,  
                           FISSURE= (FISS1, FISS2),  
                           CONTACT=' P1P1 ',  
                           INFO=1, );
```

The description of routines FORTRAN for the operator `MODI_MODELE_XFEM` is given below and the tree of call is presented on Figure 2.

`OP0113` – is the basic routine of the operator. Three SD here are created: `MODEL`.XFEM_CONT'` (information on the existence or not contact stores), `MODEL`.NFIS'` (the number of the cracks in the model stores) and `MODEL`.FISS'` (stores the names of the cracks).

`XTYELE`– routine which determines the type of element X-FEM, according to the type of enrichment and the type of contact.

`XTYHEA` – routine which counts the number of ddl Heavisides credits in the mesh, in the case of Multi-Heaviside.

`XMOLIG` – routine for the modification of `LIGREL` X-FEM, according to the type of enrichment.

`XCODEC` – routine hat for the launching of the calculation of cutting under-tetrahedrons, the facets of contact and checking of the criteria of conditioning.

XCONNO – routine which carries out the concatenation of the fields nodes to pass to multi-cracking. The new fields are attached to the model, they are of standard field elno.

XFISNO – routine which creates the sd `MODELE'//.FISSNO'` and `MODELE'//.HEAVNO'`. It is fields ELNO useful in the case of the elements which see several cracks. The first SD turns over the number of crack considering locally by the element associated with the number with ddl Heaviside with the node. The second carries out the operation reverses and turns over the number of ddl Heaviside to be associated with the local number of crack to the node.

XTOPOI – launching of the calculation of cutting of the elements fissured under-tetrahedrons.

TE0514 – elementary calculation of the partitions of the fissured elements.

One leaves with the SD:

`MODEL'TOPOSE.PIN'`, `'TOPOSE.CNS MODELS'`, `'TOPOSE.HEA MODELS'`, `MODEL'TOPOSE.LON'` and `MODEL'TOPOSE.CRI'`. When the mesh sees several cracks, it is cut out sequentially by each one cracks. For this purpose, various operations from it TE are encapsulated in a loop on the cracks.

XDECOU – routine called since TE0514 to find the points of intersection enters the edges and the plan of crack. Let us note that it is the routine XDECQU who is called in the quadratic case.

XDECOV - routine called since TE0514 to cut out a tetrahedron under-tetrahedrons. Let us note that it is the routine XDECQV who is called in the quadratic case.

XTOPOC – launching of calculation for the under-cutting of the surface of contact and the creation of the SD `MODEL'.TOPOFAC.PI'`, `MODEL'.TOPOFAC.AI'`, `MODEL'.TOPOFAC.CF'`, `MODEL'.TOPOFAC.LO'`, `MODEL'.TOPOFAC.BA'` and `MODEL'.TOPOFAC.OE'` Let us note that the SD `MODEL'.TOPOFAC.PI'` contains the coordinates of the points of intersection in the element relative: it is very useful to find the value of the functions of form of the points of intersection. The SD `MODEL'.TOPOFAC.OE'` analogue contains the coordinates of the points of intersection in the total reference mark. It is useful to feed it TE0519 during the geometrical reactualization. In the case of multi-fissured meshes, all these SD are duplicated as many times as the element sees cracks. The SD `MODEL'.TOPOFAC.HE'` is then added to store, by facet of contact, the value of the Heaviside functions of each crack on both sides of the facet.

TE0510 – elementary calculation for the topology of the facets of contact. Let us note that the facets are also useful in the case of the elements without contact, in order to impose efforts of pressure on the level of the lips of the crack. In the case of the elements containing a junction between crack, certain facets should be redécoupées on both sides of the junction. This work was not implemented and the facets are not taken into account.

XCFACE – routine called by TE0510 to find the points of intersection enters the edges and the plan of the crack and to determine the cutting of the facets. Let us note that it is the routine XCFAQ2 who is called in the quadratic case.

XCFACF – routine to find the points of intersection enters the bottom of crack and the faces for the elements in bottom of crack.

XCFACJ – routine to find the points of intersection enters the junction of cracks and the faces for the elements multi-cracks.

XAIN2 – routine which modifies the SD `MODELE:/'TOPOFAC.AI'` in the case of multi-fissured meshes. It should be made sure that the facets connected to the cut edge have well the same values of Heaviside function. In the contrary case the edge is eliminated to avoid any conflict LBB.

XSTAN2 – routine which modifies the SD `MODELE://'.STNO'` in order to cancel them degrees of freedom HEAVISIDE for reasons of conditioning of matrix and to avoid worthless pivots in the matrix of rigidity.

XCRVOL – routine called by XSTAN2 helping for the calculation of the voluminal criterion in link with the conditioning of elements X-FEM.

XORIFE – routine to direct the subelements of skin of elements X-FEM (in 2D the meshes of skin are segments of edge).

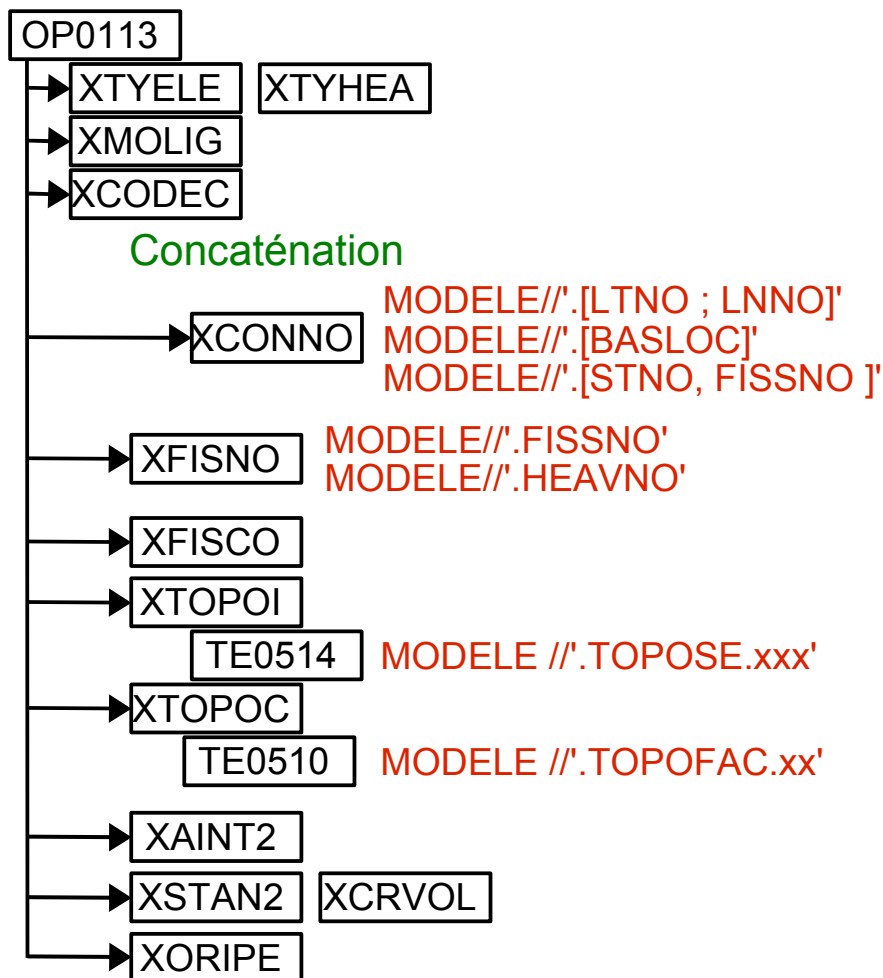


Figure 2. The tree of call of the operator `MODI_MODELE_XFEM`

4 Assignment of the contact

At the time of the drafting of this document, the assignment of the contact on the lips of cracks X-FEM is done via the operator `DEFI_CONTACT`, under the keyword `ZONE`.

Most important for implementation X-FEM is the fact that the SD dedicated to the contact, for any analysis with Code_Aster, were unified starting from version 9.0.22 and that it is here that they are enriched for the approach *great slips with X-FEM*.

During restitution 9.1.6 this operator was enriched with the routines which relate to algorithm LBB: `XLAGSP`, `XLAGSC`, `XSELLA`, `XRELL1`, etc (see [R7.02.12] for more details on algorithm LBB), routines which were before in the operator `MODI_MODELE_XFEM`.

Figure 3 presents the tree of current call (by taking account also of the overload “great slips”). Indeed, it is here that the first routine dedicated to this new approach appears – `XMACON`. the SD then is created `TANK '.CONTACT.MAESCL'` containing a certain number of relative information with the meshes of contact X-FEM (see [D4.06.14] for the description of this SD). One treats the contact by the method continues and one follows the same approach exactly as that of the routine `MASON` for the classical case, except that in the SD mentioned above one has less of occupied positions. Always in this routine, `XMACON`, the object is dimensioned `TANK '.CONTACT.TABFIN'` who will be filled later, during the pairing carried out by the routine `XAPPAR` (see the operator `STAT_NON_LINE`). Lastly, two new routines, `XNEUVI` and `XBARVI`, allow the creation of the SD `FISSI//'.CNCTE'`. Each one of these SD stores useful information, on the groups of vital edges connected between them, with the improvement of the integration of the contributions of contact (see [R5.03.53]). More details on all the SD relating to the contact with X-FEM are also provided in the description of routines FORTRAN (see `CALICO` below).

Below one gives an example of introduction of the contact into the command file for an analysis X-FEM with the approach great slips. Attention, the catalogue of the order (`defi_contact.capy`) was enriched to allow the definition of the maximum number of iterations on the geometrical loop (`ITER_GEOM_MAXI`) as well as the level of tolerance for the projection of the points of contact on surface Master (`TOLE_PROJ_EXT`), which has the same function as in the classical case.

```
CTXFEM = DEFI_CONTACT (MODELE=MODELEK,  
                      FORMULATION=' XFEM',  
                      FROTTEMENT=' COULOMB',  
                      ITER_FROT_MAXI=10,  
                      ITER_CONT_MAXI=10,  
                      ITER_GEOM_MAXI=10,  
                      ZONE= (_F (INTEGRATION=' NOEUD',  
                                COULOMB=0.5,  
                                COEF_REGU_FROT=100,  
                                COEF_REGU_CONT=100.,  
                                TOLE_PROJ_EXT =0.0,  
                                CONTACT_INIT=' OUI',  
                                FISS_MAIT = FISS1,  
                                ALGO_LAGR=' VERSION2',),  
                              _F (INTEGRATION=' NOEUD',  
                                COULOMB=0.5,  
                                COEF_REGU_FROT=100,  
                                COEF_REGU_CONT=100.,  
                                TOLE_PROJ_EXT =0.0,  
                                CONTACT_INIT=' OUI',  
                                FISS_MAIT = FISS2,  
                                ALGO_LAGR=' VERSION2',),),),)
```

Description of routines FORTRAN:

`OP0030` – the principal routine of the operator `DEFI_CONTACT`. One calls from here `CALICO`.

`CALICO` – the routine which launches the creation and the filling of the SD related to the contact for the method continues. One particularly recommends the reading of the comments to the beginning of this

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>)

routine because one finds here the description detailed of all the objects (official documentation [D4.06.14] not being put yet up to date).

SYMECO - routine for the determination amongst zones of contact for symmetrical pairing and the filling of the associated SD `TANK \.CONTACT.SYMECO'`. This SD will be common for all the formulations (continuous, discrete, X-FEM).

CAMISOLE – routine hat to recover the characteristics of contact introduced by the user. the SD here are created `TANK \.CONTACT.FORMCO'` and `TANK \.CONTACT.METHCO'`, the last also being filled here.

CAZOCO – routine of reading of the characteristics of contact for each zone. One fills here `TANK \.CONTACT.FORMCO'`.

CMFXSD – routine hat for the launching of the creation of the SD of definition of the contact for the three formulations (continues, discrete, X-FEM).

CARACX – creation of the SD of definition of the contact for formulation X-FEM.

CAZOXC – routine specialized for formulation X-FEM (in the same way exists `CAZOCD` for the discrete formulation and `CAZOCC` for the formulation continues). Here are filled, for case X-FEM, SD `TANK \.CONTACT.ECPDON'`, `TANK \.CONTACT.CARACF'` and, `TANK \.CONTACT.TOLECO'` having the same structure as in the classical case, like, `TANK \.CONTACT.MODELX'` who stores the name of model X-FEM used by the contact. One adds also the information of existence of the contact slide in `TANK \.CONTACT.METHCO'`.

LIMACO – routine hat for the reading of the meshes of contact and the creation of the SD which relate to them. One is interested here only in the calls of the routines for the formulation continues in case X-FEM.

XMACON – new routine (similar to `MASON` for the classical case) for the reading of the meshes of contact X-FEM and the creation of the SD `TANK \.CONTACT.MAESCL'` and `TANK \.CONTACT.TABFIN'`. First is filled here while second is just dimensioned.

XMELIN - new routine called by `XMACON` (similar to `MMELIN` method continues) which turns over the number of points of integration according to the type of element of contact and the diagram of integration of contact.

LIMACX – routine for the reading of the cracks in contact. Here and two new specific objects for the X-FEM filled are created: `TANK \.CONTACT.XFIMAI'` (contains, for each zone of contact, the name of the SD `FISS_XFEM`) and `TANK \.CONTACT.NDIMCO'` (the dimension of the problem).

XCONTA – routine of preparation of the relative data to the contact, relating to the linear relations specifically to force to satisfy condition LBB. Two SD here are created: `TANK \.CONTACT.XNRELL'` (stores the names of the four objects necessary for the linear relations created by `XRELL1` or `XRELL2`) and `TANK \.CONTACT.XNBASC'` (the name of the nodal field for the base covariante stores).

XDEFKO – routine hat for the launching of the choice of the space of Lagrangian of contact.

XLAGSP – preparation of the objects of work for the implementation of algorithm LBB. One fills the base covariante (only for the points of contact corresponding to the completely cut elements) SD `FISSi//\BASC'`.

XLAGSC – routine of effective launching for the selection of the nodes and the writing of the relations of equality and linear for algorithm LBB.

XSELLA – selection of the nodes carrying *ddl* of contact which will be implied in the relations for the LBB.

XRELL1 – creation of the lists of relations to in the case of impose for the respect of condition LBB application of the first algorithm: `FISSI//'.LISEQ'`, `FISSI//'.LISRL'` and `FISSI//'.LISCO'`.

XRELL2 – fact the same thing as XRELL1 but in the case of the application of the second algorithm. Let us recall that for the formulation with the nodes tops, it is preferable to use this second algorithm.

XNEUVI – called by XRELL2, creates for each crack the SD `FISSI//'.CONNECTANT'` and `FISSI//'.CONNECTES'`. The first SD stores the nodes having a score strictly larger than 1 (the score corresponds to the number of vital edges connected to the node at the end of the algorithm), each one of these nodes defining a group of vital edges then, where the number of vital edges is equal to the score of the 'vital' node. The second SD stores the nodes of each group which, paired with their vital centre, form the couples of nodes defining the edges of the group. These 2 SD are temporary. They are used then destroyed in XBARVI.

XLAG2C – routine allowing to write for each crack SD `FISSI//'.LISEQ_LAGR'` corresponding to SD `FISSI//'.CONTACT.LISEQ'`. This SD store multi-fissured meshes in the case of, the couples of numbers of Lagrangian which it is necessary to associate at the time of the imposition of the relations of equalities.

XBARVI – called by XCONTA, it is here that one informs the vital edges and the groups of vital edges. One fills the 5th component of the SD `MODEL '.TOPOFAC.AI'`, which gives the vital information or not intersected edge, while making use of the list of the nodes forming the vital edges. And one creates for each crack the SD with 4 components `FISSI//'.CNCTE'`, which informs about the groups of vital edges. It stores for each edge belonging to a group, its number of group, its number of edge in the group, its number of mesh, its number of local edge in the mesh. To do that the SD are used `FISSI//'.CONNECTANT'` and `FISSI//'.CONNECTES'`, that one destroys at the end of the routine.

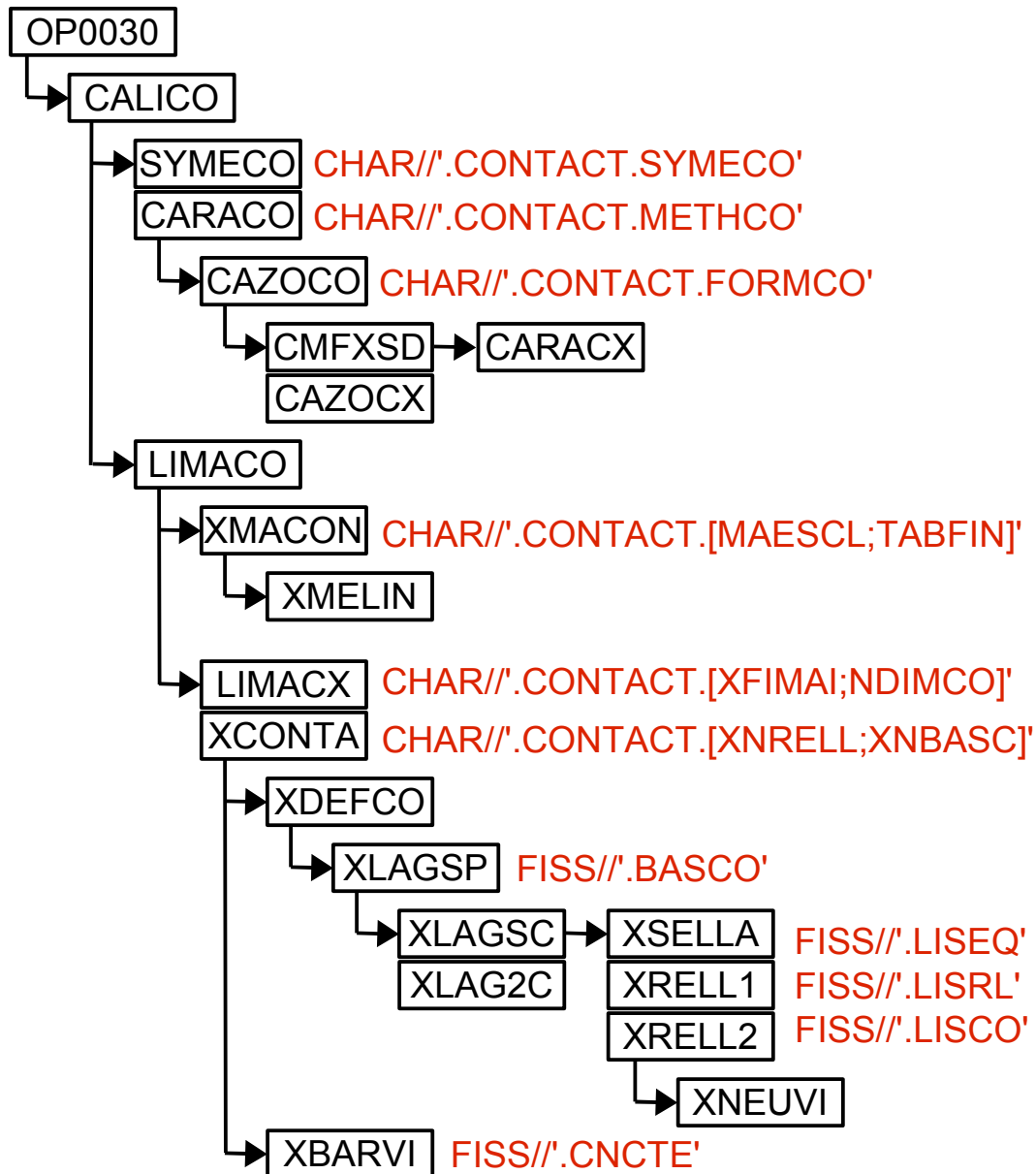


Figure 3. The tree of call of the operator DEFI_CONTACT for the keyword FORMULATION=' XFEM'

5 Assignment of connection X-FEM

It is the treatment of the keyword here 'LIAISON_XFEM' in 'AFFE_CHAR_MECA', specific for the X-FEM, and no change intervened for the passage towards the contact in great slips. One briefly presents routines FORTRAN by specifying that the principal goal is to apply the relations between degrees of freedom of contact to satisfy condition LBB. The treatment is also used to eliminate them degrees of freedom of contact in excess in the case of the old formulation.

Here is an example of call in the command file:

```
CHXFEM=AFFE_CHAR_MECA (MODELE=MODELEK,  
CONTACT_XFEM=CTXFEM,  
LIAISON_XFEM=' OUI',);
```

The description of routines FORTRAN and their tree of call (Figure 4) are given below.

CAXFEM – the routine of launching for the operations of cancellation of degrees of freedom of contact in excess.

XDELCO - routine to remove them degrees of freedom of contact in excess. Only for the old formulation of the contact, where the Lagrangian one are stored with the edges.

XRELCO – routine to apply the relations between the degrees of freedom of contact in order to satisfy condition LBB. It is pointed out that these relations are created now in the operator `DEFI_CONTACT` (`FORMULATION=' XFEM'`). Concerning the respect of condition LBB for the great slips, it is that the algorithms developed and implemented for the case HP [8] function well in this case, which is logical taking into account owing to the fact that the unknown factors contact are stored only on the part slave of the crack.

AFLRCH – the routine which assigns the list of relations to the object of the type charges. In other words, this routine makes effective the application of the list of relations created previously.

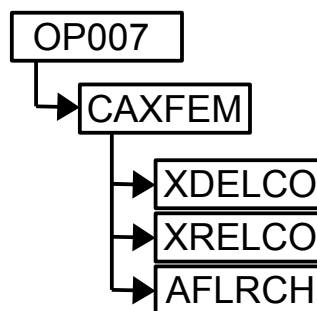


Figure 4. The tree of call of the operator `AFFE_CHAR_MECA`, keyword `LIAISON_XFEM`

6 Calculation of the contributions of contact

For a problem of contact treated with the method continues, the resolution is done with 5 loops imbricated in the operator `STAT_NON_LINE` [U4.51.03]:

- buckle on the steps of load,
- buckle geometrical,
- buckle on the thresholds of friction,
- buckle active constraints,
- iterations of Newton.

To understand operation general of the operator `STAT_NON_LINE`, the reading of the document [D9.05.01] is recommended highly. Figure 5 represents the tree of call of this operator for the specific case of a problem of contact X-FEM. One can thus find on this figure the sites of the loops mentioned above.

Figure 5 also makes it possible to locate where the major data-processing developments carried out for the approach are *great slips with X-FEM*, namely:

- the tree of call of the geometrical reactualization (presented Figure 6),
- the tree of call of the calculation of the contributions of contact-friction (presented Figure 7),
- the tree of call of the reactualization of the statutes of contact (presented Figure 8),

Here, below, an example of call to the operator `STAT_NON_LINE` for an analysis X-FEM:

```
UTOT1=STAT_NON_LINE (  MODELE=MODELEK,  
                        CHAM_MATER=CHAMPMAT,  
                        EXCIT= (  _F (CHARGE=CTXFEM, ),  
                                _F (CHARGE=CHXFEM, ),  
                                _F (CHARGE=CH1, FONC_MULT=VAR1) ),  
                        COMPORTEMENT=_F (RELATION=' ELAS ',  
                                           GROUP_MA=' SURF ', ),  
                        INCREMENT=_F (LIST_INST=L_INST, ),  
                        CONVERGENCE=_F (ITER_GLOB_MAXI=8,  
                                         RESI_GLOB_MAXI=1E-06, ),  
                        SOLVEUR=_F (METHODE=' MUMPS ',  
                                     PCENT_PIVOT=500, ),  
                        ARCHIVAGE=_F (CHAM_EXCLU=' VARI_ELGA ', ),  
                        INFO=1, );
```

The description of routines FORTRAN:

`OP0070` - only one change here: the call to the routine of zero setting of Lagrangian of contact. The routine `XMISZL` fact, for the case X-FEM, the same thing as the old routine `MISAZL` in the standard case of the contact with the method continues.

`NMCTGE` – routine hat which launches the calculation of the geometry of the facets of contact (`XREACG`) and pairing (`XAPPAR`).

`XREACG` – new routine which reactualizes the position of the points of intersection forming the facets of contact. One uses `SD MODEL '.TOPOFAC.OE'`, containing the initial position of the points of intersection, to calculate them `SD MODEL '.TOPOFAC.GE'`, and `MODEL '.TOPOFAC.GM'`, while calling `XGECFI`. These 2 last `SD` correspond respectively to the reactualized coordinates on the sides slaves and Masters.

`XGECFI` – the calculation of the new geometry of the facets of contact (slave and Master) launches. One passes by an elementary calculation (`TE0519` with the option "GEOM_FAC").

`TE0519` – the new geometry of the points of intersection calculates forming the facets of contact (slave and Master). One returns with the initial geometry of the facets, stored in `MODEL '.TOPOFAC.OE'`.

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>)

One leaves with the reactualized geometries slave and Master in MODEL ` .TOPOFAC .GE ' and MODEL ` .TOPOFAC .GM ' . The reactualization is carried out with each beginning of geometrical loop. It is used then to pair the points of contact main slaves to the meshes.

XAPPAR – new routine for pairing in case X-FEM. It is a complex routine, which calls in its turn of other routines (see the tree of call). The principle is to buckle on the zone of contact, then on the meshes slaves, then on the facets of contact of the mesh slave, and finally on the points of contact pertaining to the facet of contact. For each point of contact one fills TANK ` .CONTACT .TABFIN ' with information on the point of contact slave and the mesh Master found, in order to create pairing. To retain: there must be as many late meshes of points of contact.

MMGAUS – calculation coordinates of reference of the point of integration in the facet of contact.

XCOPCO – new routine called by XAPPAR to calculate the real Coordinates of a Point of Contact starting from the knowledge of its coordinates of reference (given with the old routine MMGAUS) and of the coordinates of the points of intersection of the facet of contact of the mesh slave.

XMREPT – new routine for the Research of the Point of intersection on the main side, nearest to the point of current contact. It provides a certain number of information on this point of intersection: its number local, number local of edge (or the node if the crack passes by a node) which contains it, and the total number of the mesh which contains it. This information will nourish the research of the project on the lip Master. Let us note that this stage makes it possible to restrict the number of facets on which the point slave will be project in XMREMA, in order to gain in performances. In certain cases (even in classical finite elements) this selection too restrictive and is led to false results. This approach would have to be re-examined.

XMREMA – new routine To search the Mesh Master nearest knowing the main point of intersection nearest to the point to contact and to make projection then. One leaves from here with essential information on the project and the mesh Master (coordinated reference, total number of the mesh Master, the tangent base at the point of projection, information on projection out of surface Master, etc). In 2D, the single facet of contact associated with the mesh Master is always a SEG2. In 3D, the associated facet Master is always a TRI3, but as there are several facets by mesh, it is also important to determine the number of facet. In the case multi-crack, one determines also the local number of the crack corresponding to the zone of contact in the main element. This number at summer previously given in XMACON and stored in SD TANK ` .CONTACT .MAESCL '.

MMPROJ – old routine of the method of contact FEM, which carries out the projection of a point of contact on a given mesh. For method X-FEM, this mesh "fictitious" and is defined by the reactualized coordinates of the points of intersection defining the facet of main contact.

XMCOOR – new routine which calculates the coordinates of reference of a point of contact in the mesh relative (slave or Master) knowing the coordinates of reference of this point in the facet of contact, and the coordinates of reference of the points of the facet of contact in the mesh relationship.

XMRLST – new routine which calculates starting from the level set tangent, the square root of the distance to the bottom of crack for the points of integrations slaves and Masters. That is useful when the element is enriched by the functions Tip Ace.

XPIVIT– new routine which determines if the point of integration is vital or not. It is looked at if the point of contact is on an edge, and if it is the case, one gives him the statute of the edge in question (statute stored in TOPOFAC .AI). One leaves from here with information "not vital" or "not nonvital", as well as the number of group and number of edge of this group if the point of contact is on an edge belonging to a group of connected edges.

NMCTCL – routine hat for the preparation of the data in order to launch the calculation of the contributions of contact. Modifications were operated in order to launch from here the creation of

LIGREL (Standard List of Groups of Elements in the same way) containing the hybrid elements of contact (routine XMLIGR) and of the cards of contact (routine XMCART).

XMLIGR – new routine for the creation of LIGREL for the new late elements of contact. It is here that the elements of contact are created by association between a mesh slave and a mesh Master. One then arranges them in GREL then in one LIGREL. The routine is similar to the old routine MMLIGR. Late meshes 2D of the type QU8QU4 and TR6TR3 are relative to the old formulation (P1-P2). They will be reabsorbed later on. For the new formulation, one lays out of the late meshes 2D QU4QU4, TR3TR3, QU4TR3 and TR3QU4 in order to treat the contact into linear (P1-P1) as well as meshes QU8QU8 and TR6TR6 for the quadratic one (P2-P1). In 3D only meshes HE8HE8, PE6PE6 and TE4TE4 are available and the treatment of the contact is not possible that into linear (P1-P1). Finally to treat the various types of elements x-fem, each late mesh described above (only for P1-P1) is associated with several late elements (specificity of X-FEM). Thus one lays out besides the element $H-H$ elements $H-HCT$, $HCT-H$, $HCT-HCT$ and finally CT (which is all alone because one treats it in small slips for the moment) to treat the contact great slip at a peak of crack. In the case of multi-cracking, it is also possible to pair elements not having the same number of degrees of Heaviside freedom. Thus it is also possible to treat the elements $H-H_2$, $H-H_3$, $H-H_4$, H_2-H , H_2-H_2 , H_2-H_3 , H_2-H_4 , H_3-H , H_3-H_2 , H_3-H_3 , H_3-H_4 , H_4-H , H_4-H_2 , H_4-H_3 , H_4-H_4 . The number of the types of multi-Heaviside elements being high (because they should also be generated for each type of mesh), the libraries of the late multi-Heaviside elements were generated using a script python.

XMELEL – provides information on the late element of contact. This routine functions for all the elements stated in XMLIGR. Considering the significant number of elements, this routine turns over several information on the late element (standard of meshes slave and Master, type of element slave and Master, type of modeling) in order to rebuild the type of dynamic element of manner in XMLIGR.

XMCART – routine to create the cards of contact (fields which contain information on the elements of contact). Cards are used but it would be necessary to reprogram all that for rather using CHAM_ELEM as in the classical case (routine MMCHML). In the classical case one creates one field. In X-FEM one creates several cards because one must provide to the elementary calculation of information on the topology of elements X-FEM. Moreover, some of this information must be provided at the same time for the mesh slave but also for the mesh Master. The map is created RESOCO//'.XFPO', which is equivalent to CHAM_ELEM RESOCO//'.CHML' in the classical case; the map RESOCO//'.XFST' who stores the statutes of enrichment of the nodes of the meshes slave and Master; the map RESOCO//'.XFPI' containing the positions of the points of intersection slaves in the coordinates of reference; the map RESOCO//'.XFAI' containing information on the intersected edges slaves; the map RESOCO//'.XFCE' containing information on the connectivity of the facets of contact slaves; if at least one of the elements Master or slave multi-is fissured, the 2 additional cards are added: RESOCO//'.XFHF' who contains the value of the Heaviside functions corresponding to enrichments on the sides Master and slave and RESOCO//'.XFPL' who contains the place of Lagrangian associated with the nodes in the matrix.

NMELCM – routine to launch the calculation of the elementary matrices of contact friction. It is common to the approaches of contact continuous (FEM, X-FEM HP, X-FEM GG).

NMELCV – routine to launch the calculation of the second members of contact friction. It is common to the approaches of contact continuous (FEM, X-FEM HP, X-FEM GG).

TE0366 – new routine of elementary calculation for the contributions of contact rubbing to the matrix of rigidity, in the approach *great slips with X-FEM*. A large number of routines (presented thereafter) are called here. The majority are also called in TE0367 and TE0363.

TE0367 - new routine of elementary calculation for the contributions of contact rubbing to the second member, in the approach *great slips with X-FEM*.

XMELET – new routine to recover information on the element of current contact. One returns with the name of the late element. One leaves with the names the meshes slave, Master; the number of the facet of contact; numbers of ddls, slaves and Masters, of the facet of contact; the full number of ddls; information on the type of formulation (nodes medium or nodes tops).

XMPINT – new routine for the calculation of the real coordinates of the points of intersection constituting the facet of contact.

XTFORM – calculates the functions of forms and its derivative of the point of integration using the parametric coordinates in the element relative. It is called by the TE0366 and TE0367, the derivative of the functions of form are then used to calculate the jacobien of the point of contact in XMMJAC.

XLACTI – calculates the active statute or not the Lagrangian ones of contact. A node is active if it belongs to a cut edge or if it is directly cut by the crack.

XMOFFC – distributes the functions of forms of the nodes or the Lagrangian ones are to be eliminated on the active nodes.

XMMJAC – new routine used to calculate the jacobien of the point of contact.

MMNORM – old routine to calculate the outgoing normal with the project of the point of contact. Attention, in 3D one does not call MMNORM, because the facets are directed in the other direction for the method of contact X-FEM compared to the method of contact FEM.

XTCALN – calculates the normal vector.

PROVEC – old routine which is used to make the vector product of the tangent vectors to obtain the normal vector in 3D.

NORMEV – old routine which standardizes a vector. It is used to standardize the normal vector in 3D, but it is also called to project the vector friction on the ball unit in the slipping case of friction.

XOULA – routine suitable for X-FEM, which is used for connectivity of the ddl as contact. It is not used any more for the formulation for the nodes tops.

XPLMA2 – new routine which provides the place of one *ddl* of contact in the elementary matrix of rigidity, corresponding to late elements X-FEM. The corresponding routine in HP is XPLMAT.

XTLAGM – Lagrangian contact and of friction for the point of contact calculates.

XTLAGC – Lagrangian contact for the point of contact calculates

XTLAGF – Lagrangian friction for the point of contact calculates

XTDEPM – calculate the increment of displacement since the step of previous time of the point of contact slave and his project on the facet Master.

XMMJEU – new routine called by TE0366 and TE0367 to calculate the game enters the point of contact and its project. During iterations of Newton, one has access with total displacements of the step of previous time (idepl) and with the increments of displacements since the step of previous time until Newton in progress (idepm). The position is thus calculated by making initial geometry + total displacement of the step of previous time + increment until Newton in progress.

XMMJEC – new routine called by TE0363 to calculate the game enters the point of contact and its project. During the reactualization of the statutes of contact, one has access to total displacements (idepl): the position is thus calculated by making initial geometry + total displacement.

TTPRSM – old routine which calculates g_{τ} , $\|g_{\tau}\|$ and $[P_{\tau}]$.

XMMAA1 – new routine for the calculation of the terms A_u , A and A^T (case with contact) of the matrix of rigidity due to the contact. The expressions of the terms of contact-friction in the matrix of rigidity and the second member are given in documentation [R5.03.53].

XMMAA0 – new routine for the calculation of the terms due to the modeling of the contact in the matrix of rigidity (case without contact).

XMMAB0 - new routine for the calculation of the terms due to the modeling of the contact in the matrix of rigidity (case of the contact without friction).

XMVEC1 - new routine for the calculation of the terms of the second member (case with contact).

XMVEC0 - new routine for the calculation of the terms of the second member (case without contact).

XMMAB2 - new routine for the calculation of the terms Bu , B , BT and F in the matrix of rigidity due to friction (case of the contact slipping friction).

MKKVEC – old routine which applies a transformation to a vector. This routine is called by XMMAB2, it is useful to calculate the matrices of friction dependent on $K(\mathbf{g}_\tau)$.

XMMAB1 - new routine for the calculation of the terms Bu , B , BT in the matrix of rigidity due to friction (case of the contact adherent friction).

XMMAB0 - new routine for the calculation of the term of friction F in the matrix of rigidity due to friction (case of the contact without friction or case without contact).

XMVEF1 - new routine for the calculation of the terms of friction $L^{1\text{frott}}$ and $L^{3\text{frott}}$ of the second member (case of the rubbing contact).

XMVEF0 - new routine for the calculation of the term of friction $L^{3\text{frott}}$ of the second member (case without contact and case without friction).

XTEDD2 – new routine which eliminates the degrees of freedom Heaviside and Crack Tip in excess for the contributions from contact in great slips X-FEM. It eliminates also the dds from contact in excess.

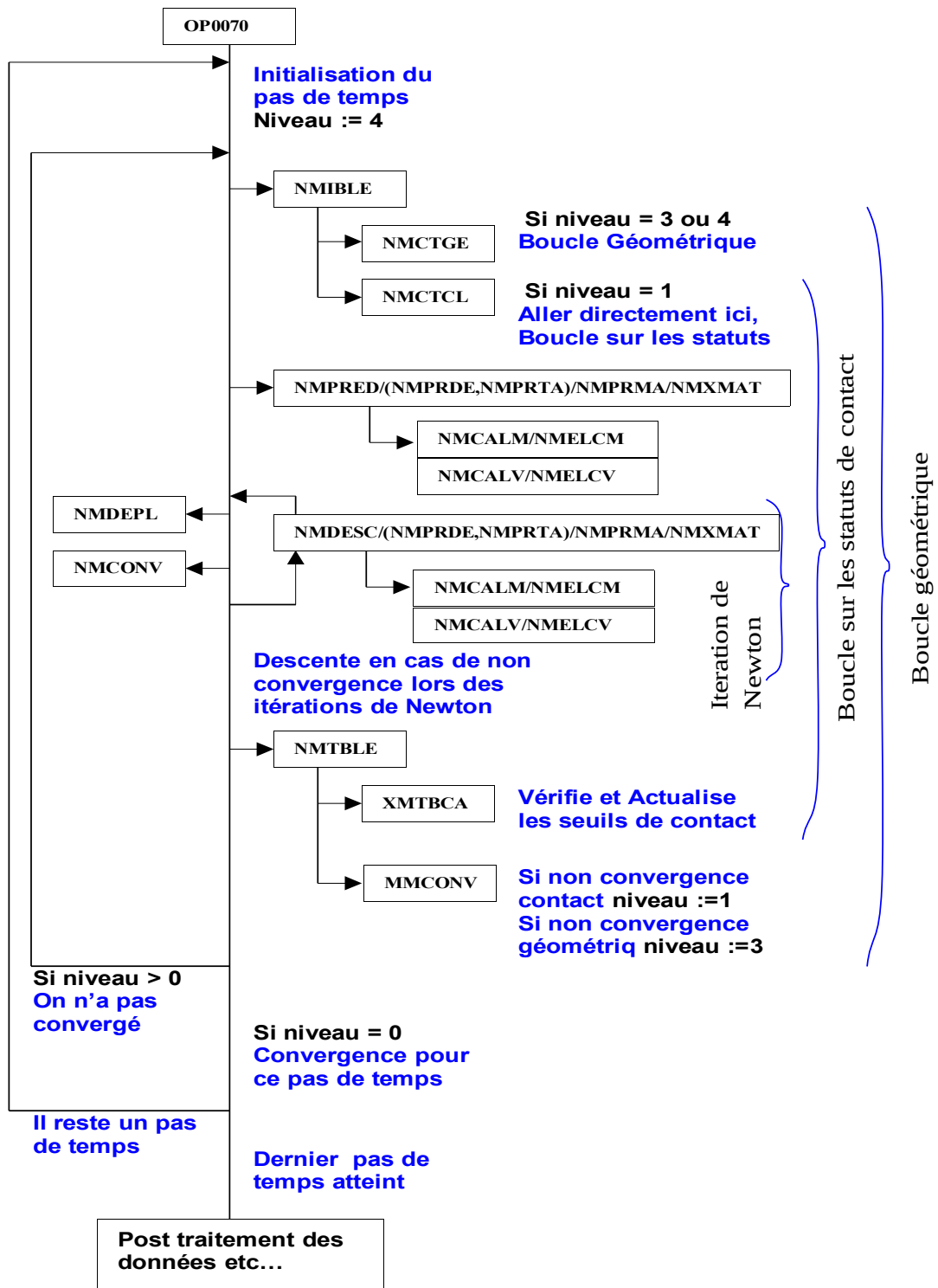


Figure 5. The tree of call of the operator `STAT_NON_LINE` for the treatment of a problem of contact in great slips with XFEM.

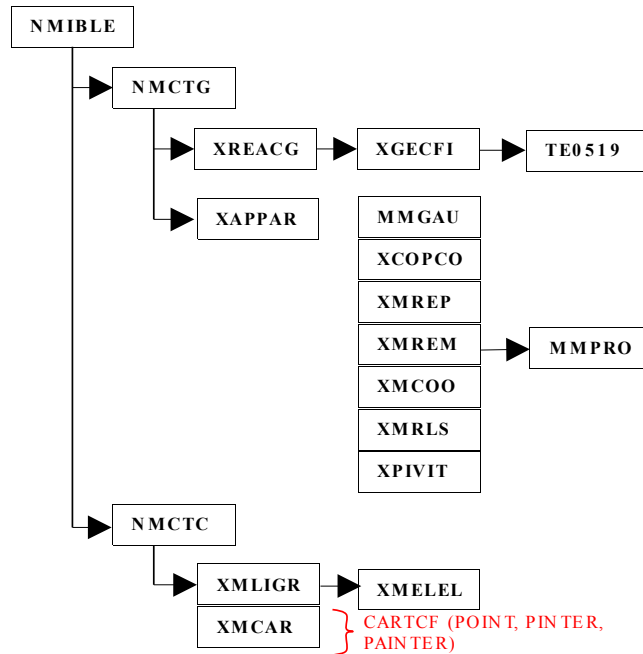


Figure 6. The tree of call at the time of réappariement geometrical for a problem of contact in great slip with X-FEM.

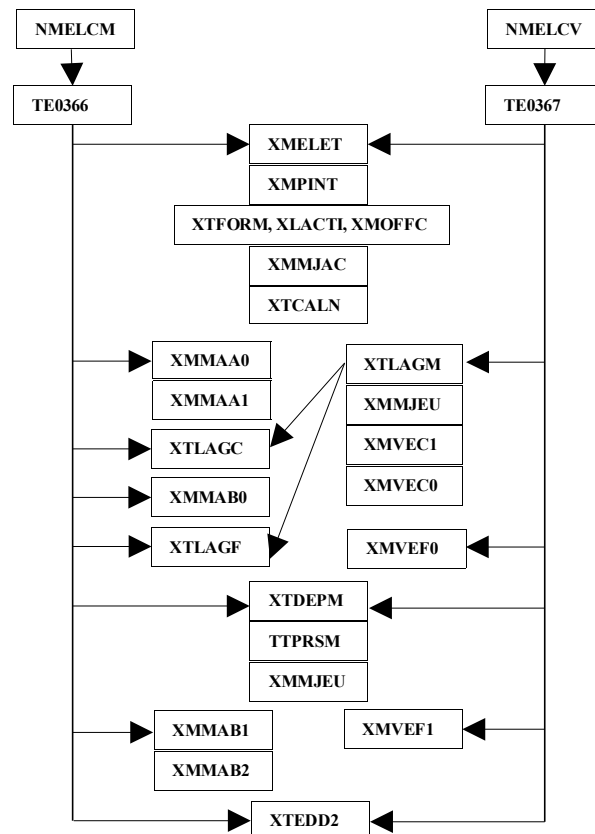


Figure 7. The tree of call for the calculation of the contributions of contact in the matrix and the second member for a problem of contact in great slip with X-FEM.

7 Reactualization of the statutes of contact and the thresholds of friction

The routines called during the reactualization of the statutes of contact and the thresholds of friction (see the tree of call in Figure 8) are described below.

NMTBLE – routine of launching for the loop of active constraints and that on the thresholds of friction (cousin of **NMIBLE**). One changed the call to the routine here **XMMBCA** (valid for the case HP) by the call to the new routine **XMTBCA** (for the great slips). For the thresholds of friction, one does nothing, because this loop is not active any more in great slips X-FEM.

XMTBCA – new routine which launches calculation to check convergence on the loop of active constraints and to if need be correct the statute of the points of contact. The checking is done by the call of **TE0363** for each point of integration of contact. For each point of integration, one stores the value of change of status. This value is worth 1 if there is change of status for this point, it is worth 0 if not. Then, there is summation of all these values, if the sum is strictly positive, one thus has a code of negative return not of convergence. It is here that one adds also the checking on the statute of the points of integration of contact which belong to a group of vital edges.

TE0363 – new routine of elementary calculation associated with **XMTBCA** to check convergence on the loop of active constraints. Attention, this routine does not have correspondence for the classical case (the checking is done then in the routine **MMALGO**).

MMCONV – old routine which ensures the management of convergence for the method continues. In the case HP one had always imposed geometrical convergence (because there was no geometrical reactualization). One thus modified the routine so that the checking is made in the case *great slips with X-FEM*.

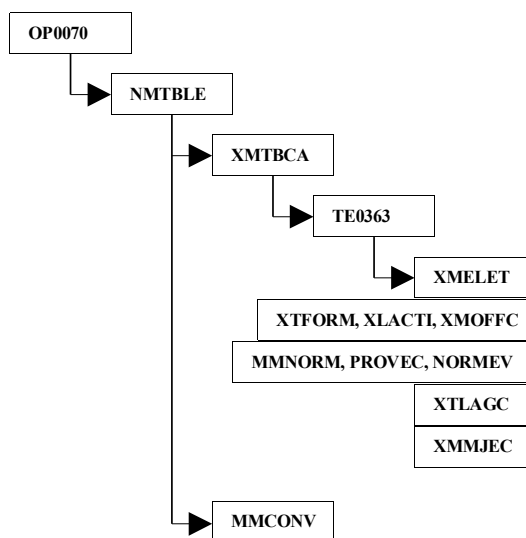


Figure 8. The tree of call of the operator **STAT_NON_LINE** for the loop on the thresholds of friction and that of the active constraints

8 Developmental perspectives

In conclusion, digital implementation of the approach *great slips with X-FEM* was validated under the conditions stated at the beginning of this document. The approach is thus usable with friction in Code_Aster in 2D (forced plane and plane deformations for the types of elements TRI3, QUAD4, TRI6 and QUAD8) and in 3D (for elements HEXA8, PENTA6 and TETRA4). The approach was also generalized at the bottom of crack.

Approach *large slip with X-FEM* is also validated for the elements multi-fissured (junctions in particular) without friction in 2D and 3D. There remain however efforts to make to guarantee the robustness of the method in this case.