
Note of use of the models of cohesive zones

Summary

This documentation is intended to provide the necessary information to user wishing to model the starting and the propagation of crack with the cohesive models (CZM).

In a first part one recalls the broad outlines of this kind of models, their scopes of application, advantages and limitations. The second part is intended to provide specific recommendations for the realization of the grid. Advices of use in various fields of the breaking process are detailed in the third part. Lastly, one finishes this documentation for an example of command file commented on by referring to the advices lavished in the preceding parts.

Contents

1	Introduction.....	3
2	Synthesis of the cohesive models.....	4
2.1	Tally theoretical.....	4
2.2	Model finite elements.....	4
2.3	Cohesive laws.....	5
2.4	Definition of the local reference mark to the crack.....	6
3	The Councils on the grid.....	8
3.1	General information.....	8
3.2	Creation of the grid.....	8
3.2.1	Tools Code_Aster.....	9
3.2.2	The Councils GMSH.....	10
3.2.3	The Councils Salomé.....	11
3.3	Modeling of the cohesive zone.....	11
3.4	Definition of one prefissures cohesive.....	12
3.5	Multi-cracking.....	12
4	The Councils of use.....	13
4.1	General information.....	13
4.1.1	Mechanical parameters.....	13
4.1.2	Digital parameters.....	13
4.1.3	Loadings and boundary conditions.....	14
4.1.4	Convergence criteria.....	14
4.1.5	Linear Solvor.....	14
4.2	Calculations in brittle fracture.....	15
4.2.1	Linear material.....	15
4.2.2	Nonlinear material.....	15
4.2.3	Digital tools.....	15
4.3	Calculations in ductile rupture.....	17
4.3.1	Digital tools.....	17
4.4	Fatigue analyses.....	18
4.5	Calculations in dynamics.....	19
4.6	Postprocessing of cracking and order POST_CZM_FISS.....	19
5	Implementation of a calculation CZM.....	21
5.1	List of the cases tests with the cohesive models.....	21
5.2	Example with accompanying notes, beam DCB in fatigue.....	22
6	Conclusion.....	26
7	Bibliography.....	27

1 Introduction

The ambition of the cohesive models is to represent the evolution of surfaces of damage and/or rupture of a solid in 2D or 3D, initially healthy or already partially fissured, subjected to a range of requests very broad (mixed mode, cyclic loadings, thermics...) in quasi-static or dynamic mode for a material having or not nonlinearities in its behavior (elasticity, plasticity, viscosity,...). This ambition however is limited, for the moment, with an evolution of surfaces whose direction is known *a priori*.

This approach, largely developed in the literature these last years, is gravitational to describe the phenomena of cracking. It is Intermédiaire between the criteria of starting (rate of refund of energy or criteria in constraint) and mechanics continues damage. When the geometrical localization of the potential cracks is known, she provides a sure and robust framework to predict the kinetics of propagation.

The objective of this documentation is to provide a user guide to the users. On the one hand with the research units with an aim of maintaining these models and of making them evolve; but also to facilitate their use by engineering. In a first part one briefly recalls the broad outlines of the models available in *Code_Aster*, their respective advantages and limits as well as the references necessary to a finer comprehension of their formulation. The precautions to be taken at the time of the realization of the grid are exposed daNS the second part. Third is dedicated to the advices of use in the various fields of the breaking process. Lastly, one finishes this documentation for an example of use commented on.

2 Synthesis of the cohesive models

2.1 Tally theoretical

The cohesive models presented here are based on the variational approach of the rupture suggested by Frankfurt and Marigo [bib2]. Based on a principle of minimization of energy, it makes it possible to represent the starting and the propagation of cracks, without assumptions a priori on their space-time evolutions and whatever the loading applied. This theory applies that the configuration of balance of a structure is that which minimizes, with each increment of loading, its total energy, summons potential energy and energy of surface. The latter corresponds to energy dissipated to create one or more cracks.

The digital models suggested here are limited in search of state of balance (or satisfactory with the equations of the movement in dynamics) of a structure for which one knows the potential way of cracking as a preliminary.

2.2 Model finite elements

There exists in Code_Aster three type of finite elements intended to model cracking with a cohesive law. The two first presented below are classically used in literature (see [bib5] or [bib6]), third are innovator and make it possible to combine the advantages of the two precedents even if, for the moment, it remains in the course of validation.

The element of joint (EJ) classic, laid out along the potential cracks, makes it possible to discretize the jump of displacement with an order in conformity with the other elements of the grid. On the other hand, it requires a regularization of the cohesive law by penalization of the conditions of contact and adherence, apparently source of instabilities of the total answer of the structure. Moreover, it leads to space oscillations of the cohesive forces which can prove to be crippling, because of their collocation at the points of Gauss of the elements.

Modelings: PLAN_JOINT, AXIS_JOINT, 3D_JOINT
Documentations: [R3.06.09] and [U3.13.14]

The element with discontinuity (ED) in cluse discretizes at the same time the jump of displacement (constant by piece) and the field of surrounding voluminal displacement (QI). It fills its objective, namely the taking into account without regularization of the conditions of adherence. On the other hand, as opposed to what one hoped, the geometrical position of the crack is not left free, except deviating from an energy formulation. Moreover, its limitation with an elastic behavior into quasi-static and the insufficient order of discretization return it limited in practice.

Modelings: PLAN_ELDI, AXIS_ELDI
Documentations: [R7.02.14] and [U3.13.14]

More recently, the development of a third family finite elements made it possible to reconcile the advantages of the two preceding ones. Also laid out along the potential cracks, **the mixed element of interface (EI)** provides a discretization of the field of displacement ($P2$), cohesive forces (PI) and of the jump of displacement (PI - discontinuous or $P2$ - discontinuous). So that:

- it authorizes it treatment conditions of adherence without any regularization, from where more regular structural answers;
- it respects condition LBB which guarantees a satisfactory convergence with the refinement of grid, with the same rate as for the elements of volume (quadratic convergence);
- it is compatible with any law of voluminal behavior.
- on the other hand it is compatible only with one quadratic grid

For the elements of interface, one distinguishes a standard modeling *_INTERFACE and an under-integrated modeling *_INTERFACE_S. In the first, the jump of displacement is modelled by a field $P2$ - discontinuous, while in the second it is modelled by a field PI - discontinuous. Each one of

these modelings has advantages and disadvantages. Standard modeling limits the risk of appearance of worthless pivots in the tangent matrix, but do not check condition LBB strictly. To cure it, one advises to choose a parameter of penalization of Lagrangian increased not too large (this choice brings a little "flexibility" in modeling). Under-integrated modeling checks condition LBB best, but more often presents of the worthless pivots in the tangent matrix. These pivots appear in particular when the interface is on a symmetry plane, or between a volume and an element of structure (MEMBRANE, GRID or BAR).

The first applications confirm the potential of this element which will have made a success of the synthesis of the two preceding families. Its application in dynamics to treat the brutal propagations is envisaged in the short run.

Modelings: PLAN_INTERFACE, AXIS_INTERFACE, 3D_INTERFACE, PLAN_INTERFACE_S, AXIS_INTERFACE_S, 3D_INTERFACE_S
Documentations: [R3.06.13] and [U3.13.14]

Model finite element	<i>EJ</i>	<i>ED</i>	<i>EI</i>
Mechanical problem	Minimum of energy	Minimum of energy	Minimum of energy under constraints
Modeling	2D plan, axis and 3D	2D plan, axis	2D plan, axis and 3D
Dimension finite elements	Surface	Voluminal	Surface
Type of elements	QUAD4, HEXA8, PENTA6	QUAD4	QUAD8, HEXA20, PENTA15
Form of energy	Regularized	Not regularized	Not regularized
Surrounding material	Linear or Non Linéaire	Linear	Linear or Non Linéaire
Interpolation	P1 displacement, P1 jump	P1 displacement, P0 jump	Displacement P2, Forced P1, P1 jump discontinuous or discontinuous P2
Influence of the digital parameters	Great influence of the digital parameter	Digital Pas de parameter	Weak influence of the digital parameter
Type of calculations	Statics and dynamic	Statics	Statics and dynamic (?)

Table 1: synthesis of the advantages and limits of the models finite element quoted previously.

In table 1 one finds the advantages of the models *EJ* and *ED* in the model *EI* without having the limits of them: not regularized law, or not linear behavior linear material, jump *P1* and digital parameter which does not influence the results of calculation while it is the case with that of *EJ*.

2.3 Cohesive laws

The cohesive laws are a relation between the vector forced and the jump of displacement in each point of gauss. All these laws are non-linear and must be used with the operator non-linear STAT/DYNA_NON_LINE (consequently they are not adapted for MECA_STATIQUE). For the preceding models the cohesive laws are baptized:

EJ : CZM_LIN_REG, CZM_EXP_REG
ED : CZM_EXP
EI : CZM_OUV_MIX, CZM_TAC_MIX, CZM_EXP_MIX CZM_FAT_MIX, CZM_TRA_MIX, CZM_LAB_MIX

One gives an example in mode of opening (figure 2.3), relation between the normal component of the vector forced σ_n and the normal jump δ_n . The variable interns threshold κ allows to manage the

irreversibility of cracking. The mechanical parameters of the law are the critical stress of starting σ_c and the rate of critical refund of energy G_c . The report of these two parameters defines a critical jump $\delta_c = 2G_c / \sigma_c$ with beyond which the constraint is cancelled.

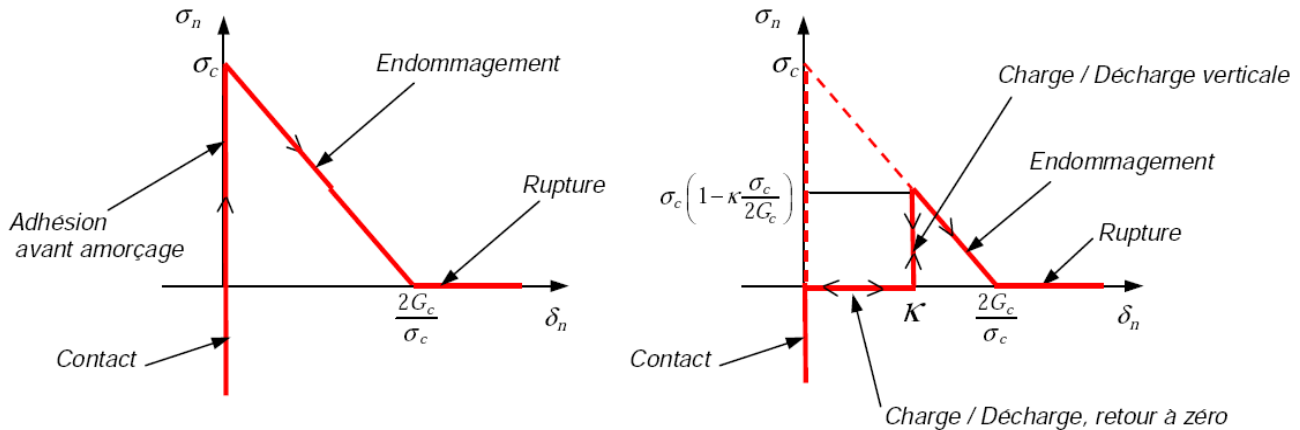


Figure 2.3 : Normal component of the vector forced according to the normal jump for the law CZM_OUV_MIX (threshold κ no one on the left and positive on the right) .

Principal differences between the laws CZM* relate on the phase of starting and the form of the law post-peak. Laws *REG take into account the contact part/adhesion in a penalized way while the others carry out these phases in a perfect way (infinite slope in the beginning, to see figure 2.3). Let us note that the regularization of the cohesive laws affects the results of calculation (see the part 13). In addition, laws *EXP* (except CZM_EXP_MIX whose law was truncated voluntarily) have an exponential behavior decreasing which tends asymptotically towards zero with the opening of the lips. Formally, those do not make it possible to define a point of crack, one will prefer the other laws with linear decrease to them which explicitly provide the position of the bottom of crack (critical jump of rupture, to see figure 2.3).

Generally, at the time of the phase of propagation of the crack, the form of the cohesive law (linear or exponential) has little influence on cracking, on the other hand it can play a role on the phase of starting. For more details on these laws to see [R7.02.11] .

2.4 Definition of the local reference mark to the crack

For the integration of the cohesive law which utilizes a jump of displacement, function of the nodal displacements expressed in the total reference mark, it is necessary to know a local reference mark $(n, t1, t2)$ with the element. For the EI and the EJ two different methods were adopted, each one having its advantages but also its limitations.

For EI : the user provides the necessary information to define the local reference mark (in 2D, a swing angle is enough, in 3D it is necessary to know the three nautical angles). It is relatively simple for a rectilinear or plane crack. That becomes much more complex when it is not any more the case. On the other hand this method makes it possible to impose a single local reference mark on all the elements of a crack planes given and thus authorizes a coherent visualization of the jumps of displacement in mode II and III . We will see that it is not the case for EJ and ED .

For a group of mesh of interface FISS in a given plan, the user informs in the following way the characteristics of the EI (angles nautical (a, b, c) in degrees):

```
ORIEN_FI=AFPE_CARA_ELEM (
    MODELE=MO,
    MASSIF= (
        _F (GROUP_MA=' FISS', ANGL_REP= (has, B, c)),
```

));

then CARA_ELEM = ORIEN_FI in STAT_NON_LINE.

In 2D it is easy to know the angle between the normal of the local reference mark n and the vector X total reference mark. This angle is the first nautical angle, the two others are indicated to zero. In 3D it is a little more difficult. To help the user one provides some examples of nautical angles with the definition of the local reference mark $(n, t1, t2)$ in the total reference mark (X, Y, Z) .

$$\begin{aligned}(0, 0, 0) : (n, t1, t2) &= (X, Y, Z) \\ (90, 0, 0) : (n, t1, t2) &= (Y, -X, Z) \\ (-90, 0, -90) : (n, t1, t2) &= (-Y, -Z, X) \\ (0, -90, -90) : (n, t1, t2) &= (Z, X, Y)\end{aligned}$$

For EJ and them ED : The calculation of the local reference mark is carried out (into hard) starting from the geometrical coordinates of the element. No information is given by the user, the taking into account of or not plane nonrectilinear cracks is thus immediate. On the other hand, for a plane crack, the elements have all the same normal but tangent vectors $(t1, t2)$ different from one element to another. That of course does not affect any the validity of calculations, on the other hand that prevents a clean visualization of the components of the jump in mode *II* or *III*.

3 The Councils on the grid

After some general information on different the type of finite elements, one provides advices on the realization of the grid like on his characteristics in order to correctly modelling the cohesive crack in various configurations.

3.1 General information

Generally, the cohesive models make it possible to represent a crack on a way Γ given *a priori*. These models give an account of the evolution but not of the direction of the propagation. It is thus necessary to carry out a grid compatible with the potential way of crack. That consists in having the cohesive elements along this way.

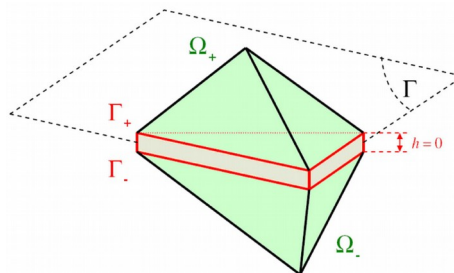


Figure 3.1-a : Diagram 3D of an element *EI* or *EJ* inserted between two tetrahedrons.

For the surface models (*EJ* and *EI*), in 2D the lips of the crack correspond to large the side of the cohesive elements (degenerated quadrangles), in 3D the lips are defined by the triangular faces (for the degenerated pentahedrons) or rectangular (for the degenerated hexahedrons). For the element with internal discontinuity *ED*, the crack passes by the center of the element and is parallel to the two opposite sides. Models *EJ* and *ED* have a linear interpolation of displacements, they are compatible with a linear voluminal grid *Q1* or *P1*. *EI* with a quadratic interpolation of displacements and is compatible with quadratic voluminal meshes *Q2* or *P2*.

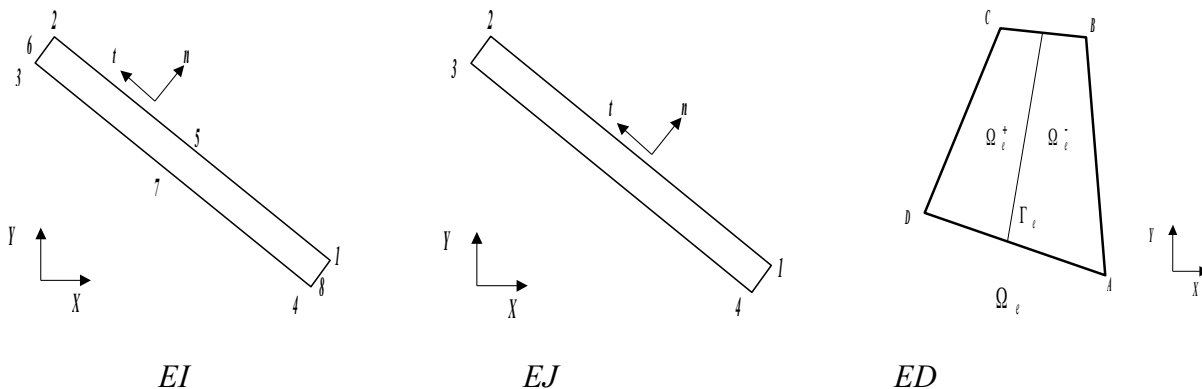


Figure 3.1-b : Diagram 2D of the cohesive elements

3.2 Creation of the grid

One gives in this part the tools available in *Code_Aster* to guarantee the good realization of a grid with a cohesive crack then one provides advices for the maillieurs GMSH and Salomé.

3.2.1 Tools Code_Aster

Meshes of the cohesive elements EI and EJ are surface, their orientation is defined by a local reference mark, it is thus necessary that their local connectivity is defined *a priori*. It is also the case of ED who are voluminal since they are crossed by a surface discontinuity.

Two techniques are available in *Code_Aster* to carry out this spot. The first, once the grid carried out, consists in checking that they have a local classification *ad hoc* (orientation). The second consists in carrying out the grid of the cohesive elements starting from two groups of nodes in glance (automatic creation).

Remarks :

- *The thickness of the elements EJ and EI does not have influence on the solution. The latter can however modify dimensions of the structure, the user must carry out the grid consequently. When that is possible one advises to carry out meshes worthless thickness.*
- *For postprocessing in 2D a small thickness can however facilitate the visualization of the variables at the points of gauss (this problem does not arise in 3D because one visualizes a surface).*
- *On the other hand, them ED have necessarily a nonworthless thickness. One recommends to choose a size of the same order of magnitude as that of the surrounding noncohesive elements.*

Orientation

The keyword `ORIE_FISSURE` (order `MODI_MAILLAGE`) allows to make sure of the good local classification of the meshes and to renumber correctly if necessary. For more details on the use of this procedure to see [U4.23.04]. One advises systematically to use this order which functions in 2D and in 3D (provided that the thickness of the meshes is nonworthless) and whatever the type of cohesive elements (linear or quadratic). One gives an example of use for the cohesive layer 'JOINT' :

```
MA=MODI_MAILLAGE (reuse =MA,  
                 MAILLAGE=MA, ORIE_FISSURE=_F (GROUP_MA=' JOINT'),)
```

Automatic creation

Another possibility: `CREA_FISS L` allows creation of cohesive meshes starting from two groups of nodes has (in glance or confused two to two). Some care are necessary for the use of this order, it is thus advised to read the documentation of `CREA_MAILLAGE` [U4.23.02]. This functionality is available in 2D (the equivalent 3D is possible with a python Salomé, to see 11 below), here is an example starting from the groups of nodes `GN1` and `GN2`.

```
MY = CREA_MAILLAGE (  
  GRID = MA_INI,  
  CREA_FISS = (  
    _F (NOM=' JOINT', GROUP_NO_1=' GN1', GROUP_NO_2=' GN2', PREF_MAILLE='  
MJ'),  
  )  
)
```

Remarks :

- *Once the order `CREA_FISS R` éa smoothed it is not necessary to pass the shredder `ORIE_FISSURE` if a linear grid is carried out .*
- *If one wishes to use this procedure with the elements of interface (thus quadratic grid), it is initially necessary to carry out the preceding procedure on a linear grid, then to pass into quadratic (`LINE_QUAD/TOUT=' OUI'`) and finally to pass by again the shredder `ORIE_FISSURE` on the cohesive layer because the order `LINE_QUAD` perhaps modified local numeration.*

Lastly, tools are under development to facilitate the creation of cohesive meshes. In particular in Salomé version 5 (see 11).

3.2.2 The Councils GMSH

With free maillor GMSH, it is rather easy to build a layer of cohesive finite elements intended to be elements of joint or interface whatever the nature of the voluminal grid. It is enough to use the order "Extrude" and to take some precautions.

In 2D : a line is extruded N of an edge of the field, a thickness ep (small but nonworthless) in the direction *there*, and it is specified that one wishes only one layer of elements with the keyword "Layers", finally one adds "Recombine" so that the maillor creates quadrangles (triangles by default).

```
ep=0.001
NbCouche = 1;
JOINT [] =Extrude Line {N, {0, - ep, 0}}
{
  Layers {NbCouche, 1}; Recombine;
}; ;
```

Then one can define one "Physical Line" new line creates or one "Physical Surfaces" surface creates in the following way:

```
Physical Line (a)      = {JOINT [0]};
Physical Surfaces (b) = {JOINED [1]};
```

The first can be used for example with the application of boundary conditions as symmetry, the second corresponds to the layer of cohesive elements. If one wishes to include the crack inside a field, one proceeds like previously, then the new line is extruded `JOINT [0]` to create the second part of volume.

In 3D : one proceeds in the same way with extrusion of a surface border of the field, it "Recombine" allows to create hexahedrons if the trace of the grid on extruded surface comprises quadrangles and pentahedrons if the latter comprises triangles.

To finish, the "physical" create in the .geo are used in the command file .comm in a usual way:

```
Physical Line (I)      corresponds to 'GMi'      if the grid is with the format .msh
Physical Surfaces (J) corresponds to 'GMj'      if the grid is with the format .msh
Physical Line (K)     corresponds to 'G_1D_k'   if the grid is with the format .med
Physical Volume (L)   corresponds to 'G_3D_l'   if the grid is with the format .med
```

Caution :

There exists a bug in GMSH 2.3 for extrusion 3D with the algorithm of grid 'Tetgen+Delaunay'. This algorithm is that proposed by default, it is thus necessary to modify it in small the Option/mesh/General and to choose 3D algorithm='Netgen'. This bug is indexed on the Web site of GMSH and should be modified in the future versions.

Remarks :

- The meshes created by GMSH are topologically correct but the local classification of the nodes is not necessarily well carried out. In Code_Aster the latter must be defined in a specific way in order to distinguish the orientation from the element cohesive. After the reading of the grid to the format .msh or .med it is thus necessary to pass by one `MODI_MALLAGE` with the order `ORIE_FISSURE` who carries out this operation of renumerotation of the cohesive elements in an automatic way (see part 9).*
- Let us note that extrusions evoked above are also easily realizable in an interactive way using the mouse. Once the automatically generated .geo, it is necessary however to add it "Recombine" to create quadrangles in 2D or hexahedron/pentahedral in 3D.*

3.2.3 The Councils Salomé

In Salomé, it is possible to extrude a layer of surface elements of an edge of the field. That makes it possible to create cohesive meshes which one will change connectivity so necessary using the order `ORIE_FISSURE` (see part 9).

Work is in hand to carry out the grid of cohesive elements between two volumes or along a surface layer in a volume. The order python `DoubleNodeGroups`¹ allows to duplicate the nodes of a group of surface meshes or a group of nodes given (see manual reference of module MESH). The weaving of the cohesive elements between these two groups of nodes can then be carried out in 2D in the command file using the keyword `CREA_FISS` (see part 9) or starting from a script python² in 3D.

Notice :

Whatever the technique adopted to net the potential way of crack it is necessary that this one crosses the structure on both sides. Indeed, there do not exist cohesive finite elements intended to model a point of crack (triangle for example) which cannot be propagated. That would not have besides really a direction since opposite with the logic of the modeling of a potential way.

3.3 Modeling of the cohesive zone

A crack CZM is defined by three zones, the first corresponds to a free crack of constraint, the second, called cohesive zone, where the forces of cohesion are nonworthless and finally a healthy zone (or zones potential cracking) where the constraint of starting was not reached (see figure 3.3).

The cohesive zone makes it possible to carry out a progressive transition from healthy material a true crack. The size of the latter is related to the characteristic length G_c/σ_c cohesive model, it adapts to remove the singularity of the constraints at a peak of crack. The lower this length is, the more the size of the cohesive zone will be reduced. With the limit, if this one tends towards zero, the cohesive zone disappears and one finds oneself in the case of the classical breaking process with the theoretical possibility to have infinite constraints at a peak of crack. Formally, that amounts taking a stress the rupture σ_c infinite.

One stresses here the importance of the mechanical point of view than digital to so much simulate this cohesive zone correctly. Generally and whatever the type of calculation, one advises to net this zone sufficiently so that it is made up from at least four or five cohesive finite elements. That makes it possible to finely represent the evolution of the crack on a mechanical level but also makes it possible to avoid problems of digital convergence involved in a too coarse discretization of this zone with strong gradient of constraint.

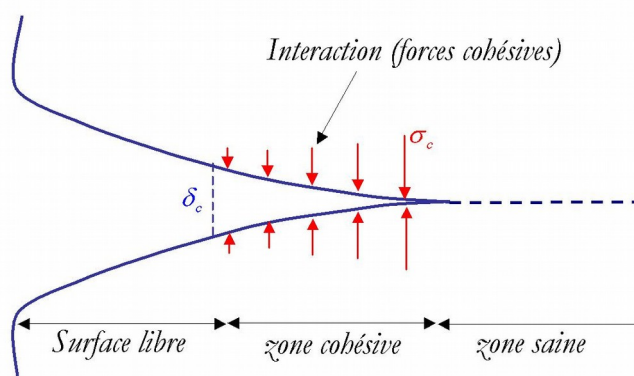


Figure 3.3 : Diagram of the cohesive crack.

¹ Available only in version 5 of Salomé

² A model in the course of validation is available in the author of this documentation

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>)

In the literature [bib7] there exists an estimate of size cohesive zone S_{zc} for a crack in an elastic infinite material:

$$S_{zc} \simeq \frac{E}{1-\nu^2} \frac{G_c}{\sigma_c^2}$$

with E Young modulus, ν coefficient of Fish. This value is given as an indication to estimate the size of cohesive zone, however the latter can vary with various parameters of the mechanical problem.

3.4 Definition of one prefissures cohesive

In most case, one has the cohesive elements in a plan of cracking or the prolongation of an existing crack. The latter is in general represented by a free surface. However, in certain cases, it can be useful to have the cohesive meshes on an initial crack. These last can for example be used as elements of contact (small deformations) or be used to define initial plan or semi-elliptic faces of cracking for example.

To define an initial crack, it is necessary to build a grid with two zones of cohesive elements contiguous, one for prefissures and the other the potential crack. The idea then consists in imposing the variable interns of prefissures of such kind that those correspond to a free zone of constraint, and not to modify those of the potential crack (i.e to impose them on 0). With this intention, orders are used `CREA_CHAMP` to build a field of variable interns at the point of gauss. This last is then used as initial state as starter in `STAT_NON_LINE` with the order `ETAT_INIT`. Let us note that it is necessary to define the whole of the internal variables of the structure, including those of the voluminal part. In the case of an elastic material this operation is commonplace, into nonlinear it is not possible to easily know them except imposing them worthless what is not inevitably realistic.

Notice :

It is not possible to define an initial crack by imposing the parameters materials σ_c and G_c worthless.

3.5 Multi-cracking

It is possible to model several cracks with all the types of cohesive elements provided that the grid is carried out consequently. With the element ED one can for example have the layers of elements in order to take into account a network of parallel cracks (see example [bib5]). For element EJ and EI the test of the perforated plate (ssnp133 [V6.03.133]) makes it possible to model the simultaneous projection of two cracks.

However, with this kind of approach one disadvises having the cohesive elements everywhere (i.e on each stops grid). In addition to the difficulty to complete such a grid, many works in literature show that the results of the way of cracking are strongly dependent on the grid. Moreover, for the EJ, the regularization of the cohesive law can also contribute to the variability of the results and play a harmful part on the quality of the solution. Finally that asks the noncommonplace question of the conditions to respect with the intersection of two cracks. No particular treatment is carried out for the models presented here.

4 The Councils of use

The cohesive models cover a rather broad spectrum of use in the field of the breaking process. One begins the advices of use on general information concerning the use of the parameters and on the loadings, valid whatever the type of simulation. One continues by giving specific advices for calculations in brittle fracture (or not linear material), the fatigue analyses and calculations into dynamic. Thereafter one stresses the importance to be pressed on digital tools to improve convergence of calculations. Finally one finishes on advices for the postprocessing of the results.

4.1 General information

4.1.1 Mechanical parameters

The mechanical parameters of the law are the critical stress of starting σ_c and the rate of critical refund of energy G_c . On figure 2.3 on the left, the constraint σ_c corresponds to the peak of the law, the density G_c is connected to the surface under the triangle. The user defines a material and values (nonworthless) of these parameters under the keyword `RUPT_FRAG`. In the case of a thermal loading it is possible to make depend the latter on the temperature. One uses in this case the keyword `RUPT_FRAG_FO`, to see example in the SDNS105a test.

The report of these two parameters defines a critical jump $\delta_c = 2G_c / \sigma_c$ with beyond which the constraint is cancelled for the laws `CZM_LIN_REG`, `CZM_OUV_MIX`, `CZM_TAC_MIX`. For the two laws `CZM_EXP_REG` and `CZM_EXP` the constraint tend asymptotically towards zero. For the law `CZM_EXP_MIX`, a critical jump is defined in $\delta_c = 3.2G_c / \sigma_c$. This jump corresponds to an error on the rate of refund of critical energy of 1 %. For the law of tiredness `CZM_FAT_MIX`, the value of the jump beyond which the constraint remains worthless depends on the loading. For the ductile law `CZM_TRA_MIX` the parameters of form provided by the user influence this critical jump. For more details to see documentation [R7.02.11].

Notice :

When one imposes conditions of symmetry on the lips of the crack it can be necessary to divide the value of G_c by two, to see part 14 .

4.1.2 Digital parameters

Elements with internal discontinuity : There is no digital parameter to inform for this model.

Element of joint : This model has two digital parameters intended to penalize adherence `PENA_ADHERENCE` and the contact `PENA_CONTACT`. One replace the infinitely rigid parts (contact and adherence) by elastic contributions strongly rigid (see Doc. [R7.02.11]). By default `PENA_CONTACT` 1 is worth and led to a penalization identical to that of adherence, it is the value which one recommends. The parameter `PENA_ADHERENCE` corresponds to a percentage internal length of the cohesive law, it must be nonnull and lower than 1. More the value is low plus the slope of penalization is important.

The computation results are strongly dependent on the value of this parameter. Generally, one observes:

- Oscillations (snap back local) in the total answer of the structure related to the successive opening of the joints. But also of the oscillations of the profile of constraint along the crack all the more marked since the rigidity of the joint is important.
- An important influence of the parameter on the quality of the solution in constraint. For values of `PENA_ADHERENCE` rather weak the error is reasonable, for higher values that to lead little to very important variations.
- The convergence of the algorithm of total Newton is all the more difficult as the penalization is important.

One thus notes antagonistic effects of the parameter of penalization which imply to make a compromise between convergence of the calculation and precision of the solution. It is thus difficult to recommend a general rule providing the good value of this parameter. One thus advises before realizing a digital simulation to carry out a test of sensitivity to this parameter. In all the cases, the initial rigidity of the elements of joint must be more important than the rigidity of the close voluminal elements.

Element of interface : This model has a digital parameter `PENA_LAGR` who corresponds to the penalization of Lagrangian increased (see [R3.06.13]). The computation results do not depend on the value of this parameter. One advises to take value by default (100). This value must be strictly higher than 1 in order to ensure the unicité of the solution during the integration of the cohesive law. In addition laws `CZM_OUV_MIX` and `CZM_TRA_MIX` intended for the opening in mode I only have a baptized parameter `RIGI_GLIS` who allows to impose a nonworthless rigidity in slip and to thus avoid the problems of null pivot during the rupture of the elements. One advises to use the value by default of 10. The solution is not affected by this parameter provided that the modes of slip are not activated.

4.1.3 Loadings and boundary conditions

All mechanical types of loadings and thermics can be *a priori* applied to a structure comprising of the cohesive elements: imposed displacements, surface efforts, voluminal forces, fields of temperature resulting from a preliminary thermal calculation.

To carry out a saving of time calculation, taking into account symmetries of the problem, it is often useful to simulate a half-structure. Conditions of symmetry on the lips of the cohesive crack (i.e them faces of the cohesive elements), leading to an opening in pure mode I, are possible of two way different:

- The first consists in blocking normal displacements of the faces of the cohesive elements which are not connected to volume. These last coincide then with the axis of symmetry of the structure. In this case, a precaution is to be taken on the value of the density of energy of surface criticizes entered by the user. Since only one lip of crack is modelled, it is necessary, for a value material given of G_c , to inform $G_c = G_c/2$ in the command file. In other words, that costs twice less energy to open a half-crack.
- The second consists in making entirely open the crack. On the lips of the crack one imposes symmetrical displacements with respect to an axis placing between those (order `AFFE_CHAR_MECA/LIAISON_GROUP`). This kind of modeling makes it possible to take into account the crack as a whole and does not require to divide it G_c by two. On the other hand that poses a problem on the energy assessment since one models a half-structure (thus a voluminal energy twice less) with a complete crack. This method is thus to use with much precaution, one councilE rather to adopt the preceding one.

4.1.4 Convergence criteria

Convergence criteria of the algorithm of Newton `RESI_REFE_RELA` allows to build a nodal force of reference, which is used to estimate, nullity (approximate) residue (see [R5.03.01]). This one makes it possible to improve convergence if the residues are heterogeneous, i.e. when different degrees of freedom of nature are treated. It is the case of the mixed model EI for which one recommends the use of this criterion.

With the cohesive models, the force is defined starting from a constraint and of its dual displacement. To use this criterion a constraint and a displacement of reference are thus informed: `SIGM_REFE` and `DEPL_REFE`. To choose these values, one proposes those resulting from the cohesive law: the constraint of starting σ_c for the first, and the characteristic length G_c/σ_c for the second. With the models EJ and ED it is not possible (and *a priori* not useful) to use this kind of residue.

4.1.5 Linear Solvor

Method of resolution of the total linear system `MUMPS` is essential for the mixed elements having degrees of freedom of the Lagrange type. It is of which necessary to use it with the EI. Its interest the

main thing lies in its capacity to swivel lines and/or columns of the matrix during factorization in case of small pivot. Other models itments finished EJ and ED do not have this restriction, they can be used with the method by default (multifrontale).

In addition, the tangent matrix of the laws cohesive is symmetrical, the use of the keyword `SYME='YES'` intended to symmetrize the latter is not necessary.

4.2 Calculations in brittle fracture

Possible models : EJ, EI, **Advised model** : EI

The cohesive models are adapted to model brittle fracture in a linear material (all) or not linear (EJ and EI only). One gives here some specific advices to this kind of simulations. Digital tools make it possible to guarantee a good robustness of the algorithms of resolution, one also provides advices on their use.

4.2.1 Linear material

For a linear elastic material, there are no specific precautions of implementation others only the preceding general information (part 13) and that on the grid (left 8). For the robustness of calculations, the use of the piloting of the loading is paramount. One devotes below a part to the advices of use of this method.

4.2.2 Nonlinear material

Cohesive models EJ and EI can interact with a surrounding nonlinear material *a priori* whatever it is. Possible simulations remain in the field of brittle fracture in the presence of plasticity, this one being confined in the vicinity of the point of crack. No specific digital processing is necessary for this kind of calculation, if they are not the usual precautions on the smoothness of the grid (see part 8) in order to collect the cohesive zone but also the plastic band along the lips of the crack. The convergence of the algorithm of Newton-Raphson is slowed down (more iterations of Newton to step of loading given) and time calculation is increased but that remains all made reasonable and does not block the robustness of the cohesive models in this scope of application.

From a mechanical point of view, the plasticization of material as well as cracking are two phenomena which enter in interaction. Thus, a special attention will have to be brought in the choice of the report of the constraint of starting of the cohesive law and the elastic limit of material.

According to the values of these parameters, one could be is in the case of figure where cracking intervenes without development of plasticity, that is to say in the contrary case where the constraint in material reaches the plastic threshold and evolves without exceeding the constraint of starting, or finally the case where plasticity and cracking evolve jointly. In this last case, one observes a plastic zone which is initiated at a peak of crack and which evolves with the propagation of this one.

In addition, some calculations with viscoplastic materials were carried out on a purely illustrative basis, the weak experience feedback in this field does not enable us to provide specific advices.

4.2.3 Digital tools

Linear research

It is possible to use linear research with the cohesive models. That can punctually improve convergence of calculations. This method however is not best adapted to cross instabilities related to the brutal propagation of crack into quasi-static. In this case, case the piloting of the loading will be preferred to him.

Piloting of the loading

The piloting of the loading, available for all the cohesive models, makes it possible to ensure the convergence of the algorithm of Newton-Raphson into quasi-static in spite of an instability in the total answer of the structure (brutal propagation of crack for example). This method is valid for the lenitive

laws which can lead to this kind of answers and in particular for the cohesive laws. The idea consists in authorizing the relaxation of the loading to restore a part potential energy and to advance cracking gradually by controlling the damage of the cohesive zones locally.

The user provides the direction of the loading (displacement or surface effort) from one `AFFE_CHAR_MECA`, this one is multiplied by the scalar parameter of piloting η , new unknown factor of the problem which gives its intensity. This last parameter replaces the multiplicative function of the loading `FONC_MULT`. At the time of the resolution of the mechanical problem, one adds an additional equation (control of piloting) which closes the system. For more details on this technique, the reader can refer to general documentation [R5.03.80] and documentations specific to piloting for the cohesive models [R7.02.11] and [R7.02.14].

This digital tool is particularly important to ensure the convergence of the algorithm. It is often necessary, including when the total answer does not admit a back return. Indeed, in certain configurations, of the local back returns, related to a brutal opening at the elementary level, can intervene. All the cohesive laws can be controlled by elastic prediction (`PRED_ELAS`), the report of the step of time Δt on the coefficient `COEF_MULT` allows to manage the evolution of cracking. On the level of law from behavior that amounts controlling the exit of the threshold. More this report is more high one wishes to thus pass instability quickly and more η will be high.

In the order `STAT_NON_LINE`, one chooses to control the load `TRACTION` (classically well informed) in the following way:

```
EXCIT      = _F (LOAD = TRACTION, TYPE_CHARGE = `FIXE_PILO`),
```

One adds then the data suitable for piloting:

```
PILOTING   = _F (
    SELECTION = `RESIDUE`,
    TYPE = `PRED_ELAS`,
    GROUP_MA = `JOINED`,
    COEF_MULT = c_mult_pilo,
),
```

Remarks and advices:

- One advises to define the direction of loading controlled by an unit vector. Thus, the parameter of piloting η will correspond exactly to its intensity.
- One advises with the user to start by choosing UN report Δt on `COEF_MULT` equal to 0.05. This way, one seeks to damage of 5% the point of Gauss more requested.
- If the algorithm does not converge in spite of piloting, one advises to reduce the preceding report: under automatic or manual step division and/or increase in `COEF_MULT`. The weaker the report will be, the slower cracking will be and conversely.
- It is sometimes necessary to break the symmetry of the problem to prevent that piloting oscillates between two values of η if those provide an identical state of cracking. For that, one advises to carry out the first nonlinear calculation with a very weak step of loading not controlled. One remains thus in *Domaine* linear of the answer and calculation converges quickly.
- Another manner of avoiding the oscillations of the value of piloting consists in informing one `ETA_PILO_R_MIN` equal to zero in order to authorize only positive values of the parameter of piloting.
- It is necessary to finely control (weak exit of the criterion) the first steps of loading controlled in order to correctly collect the value peak of the *répontotal*.
- If it piloting is used in a case where one could free oneself some, the solution is not modified. The only difference is that one directly does not control the intensity of the load.
- It is not not possible to control thermal loadings.

4.3 Calculations in ductile rupture

Possible models : El

The law `CZM_TRA_MIX` was introduced in version 10.4 to seek to model the ductile rupture. The use of this one must be circumscribed with a framework of research upstream. Indeed, it is in the course of validation and will certainly be brought to evolve.

The law for the ductile rupture takes as a starting point the laws classically suggested in the literature. Besides the parameters of critical stress and density of energy critical, one adds two parameters related to the form of the law (`COEF_EXTR` and `COEF_PLAS`). The latter is selected trapezoidal. Calculations of sensitivity to the parameters as well as tests on test-tubes CT (`SSNP151`) and AE (`SSNA120`) were realized. The first conclusions show that this model makes it possible to correctly reproduce the total answer and the kinetics of propagation observed with a model of the type Gurson (Rousselier) on a CT 2D (`SSNP151a`). It is noted however that the retiming of the parameters of the law to approach the fine calculation of the local approach remains difficult. On the other hand the saving of time calculation and robustness is significant. When one passes on a CT 3D (`SSNP151b`) the results are less conclusive starting from a certain level of loading. It seems that is partly due to the bad taking into account of the voluminal effects in the cohesive model and in particular the effect of containment. A work is in hand to try to take into account the influence of the triaxiality. Others limiting are also to underline: as for the other cohesive models, the way of crack must be known *a priori* . Cracking can be made only in mode I (it is however possible to introduce the modes of slip if that proves to be judicious). Lastly, the large openings of cracks are not taken into account, one can however use great deformations in surrounding material.

No specific digital processing is necessary for this kind of calculation, if they are not the usual precautions on the smoothness of the grid (see part 8) in order to collect the cohesive zone but also the plastic band along the lips of the crack. The convergence of the algorithm of Newton Raphson is slowed down (more iterations of Newton to step of loading given) and time calculation is increased but that remains all made reasonable and does not block the robustness of the cohesive models in this scope of application.

From a mechanical point of view, the plasticization of material as well as cracking are two phenomena which enter in interaction. Thus, a special attention will have to be brought in the choice of the report of the constraint of starting of the cohesive law and the elastic limit of material. According to the values of these parameters, one could be in the case of figure where cracking intervenes without development of plasticity, that is to say in the contrary case where the constraint in material reaches the plastic threshold and evolves without exceeding the constraint of starting, or finally the case where plasticity and cracking evolve jointly. In this last case, one observes a plastic zone which is initiated at a peak of crack and which evolves with the propagation of this one.

4.3.1 Digital tools

Linear research

It is possible to use linear research. That can punctually improve convergence of calculations.

Piloting of the loading

It is considered that the cracks are propagated in a progressive way in ductile mode of rupture. The piloting of the loading is thus not developed for this kind of law.

4.4 Fatigue analyses

Possible model : *EI*

The law available for to treat phenomenon of tiredness in mode *I* is `CZM_FAT_MIX` (see Doc. [R7.02.11]) usable with elements of interface. One can carry out calculations in 2D or 3D whatever the material and for all types of mechanical loadings (of amplitude constant or variable), or thermal loadings. One will be able to also refer to tests SSNP118JKL and SSNP139B and the part 22.

To give an account of tiredness with this law, it is necessary to traverse all the cycles of the loading. It is understood easily that it is not possible to simulate a large number of cycles. Nevertheless, the first results highlight a certain number of phenomena observed in fatigue (see report [bib3]). For an elastic material:

- One manages to reproduce laws of evolution of the Paris type which one identifies the coefficients *a posteriori*,
- One notes the weak influence of an overload on the projection,
- L is observed'absence D' effect of sequence with variable amplitude,
- One simulates realistic evolutions of face of crack 3D.

In the presence of plasticity several effects observed in experiments are reproduced:

- ON highlights the effect of overload which leads to the deceleration of the propagation due to the compression induced by the increase in the plastic zone,
- LE deceleration is all the more marked as the overload is important,
- increase in the report of load contribute to reduce the delay to report of overload and amplitude of load fixed.

In addition work on the jump of cycles are considered. It will make it possible, by extrapolation, to simulate a large number of cycles.

The Councils on the discretization of the loading:

- *At the time of the discretization of the cyclic loading some precautions are to be taken to optimize time calculation. risk is to carry out with each cycle an automatic recutting of the step which can become very expensive in CPU. The experience feedback enables us to advise with the user to carry out approximately 4 increments of the loading by cycle when the material is elastic and to double this value when this one with a nonlinear behavior. This rule is not necessarily valid in all the situations and will have to be adapted on a case-by-case basis to optimize the time spent in each cycle.*

The Council on the definition of the cyclic function:

*For the construction of the multiplicative function of the loading one can use `DEFI_FONCTION` or of the python in the file of order. One proposes below an example of function in tooth of saw to variable amplitude. The odd moments are the ends of assembled in load (amplitude max) and the even moments are the ends of discharge (amplitude min). This way, one finishes it *i* ème cycle at the moment $2i$. In example *Ci* below the loading varies*

*Of `val_inf` with `val_sup` during cycles 1 with `nb_cy_1`,
Of `val_inf` with `val_surch` during cycles `nb_cy_1+1` with `nb_cy_2`,
Of `val_inf` with `val_sup` during cycles `nb_cy_2+1` and `nb_cy_3`*

Example of cyclic function *F* of variable amplitude:

```
F = []
for I in xrange (nb_cy_1):
    f.extend ((2*i, val_inf, 2*i+1, val_sup))

for I in arranges (nb_cy_1, nb_cy_2):
    f.extend ((2*i, val_inf, 2*i+1, val_surch))

for I in arranges (nb_cy_2, nb_cy_3):
    f.extend ((2*i, val_inf, 2*i+1, val_sup))
```

The definition of the concept function in *Code_Aster* is then carried out classically:

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>)

```
FCT_FAT = DEFI_FONCTION (  
    NOM_PARA = 'INST',  
    PROL_GAUCHE=' LINEAIRE',  
    VALE      = F,  
)
```

The Council on the data storage

For to avoid a storage of the too important bases one advises to keep only the values of the fields at the top of each cycle using the keyword `FILING` of `STAT_NON_LINE`. With this intention, it is of course necessary to carry out a discretization of the loading which passes by the tops.

4.5 Calculations in dynamics

Possible models : EJ , EI (in the course of validation), **Advised model** : EJ

The fact of taking in COMpte the dynamic aspects introduces an additional complexity into modeling. In addition to the precautions of use into quasi-static (refinement of grid sufficient to discretize the cohesive, relatively small zone for the characteristics of steels (about 1 mm)), the dynamic mode imposes additional rules. One proposes the few following practical advices resting on the experience feedback. One will be able to also refer to the report [bib1].

- The diagram of integration first of all: diagram of Newmark known as “of average acceleration ($d=1/2$, $a=1/4$), unconditionally stable and nondissipative in linear medium, seems best adapted (not of digital dissipation detected in the presence of cohesive zones). Very few tests were tested with other diagrams (explicit, Wilson...) but led to problems of convergence. A thesis is in hand (D. Doyen) to optimize diagrams in the presence of shocks.
- As regards the temporal discretization, all depends on the phenomena which one seeks to simulate. To finely determine the reflections of the waves on the cohesive zone (involving a disturbance of the kinematics of the crack), it is necessary (always for the usual characteristics of steels) to consider steps of times lower or equal to $10^{-8} s$. If a coarser follow-up of kinematics is sufficient and that the important phenomenon is starting and the stop, then of the steps about $10^{-7} \leq \Delta t \leq 10^{-5} s$ are enough. With the top, problems of convergence appear ($\Delta t \approx 10^{-6} s$ seem to be a good compromise).
- A very detailed attention must be related to the passage of a stable phase to an unstable phase (more precisely a phase where the cohesive zone extends in a progressive way, then where it is propagated in a brutal way). In this case, it is necessary to adapt the steps of time to collect the moment when the cohesive zone accelerates, under penalty of gumming “the dynamic effect completely”. The use of the piloting of the loading in quasi static, associated with the observation of the snap back of the total answer can provide information over this critical moment.
- The small parameter `PENA_ADHERENCE` representing stiffness joint before decoherence (parameter of regularization) must be taken lower than 10^{-4} . Indeed, below this value the results are not sensitive any more to this parameter.

Generally, the experience feedback is limited in this scope of application. The use of these models in dynamics remains, for the moment, intended for specialists.

4.6 Postprocessing of cracking and order `POST_CZM_FISS`

Visualization of cracking for EJ and EI

To visualize a cohesive crack, several solutions are possible:

- The variable interns at the point of gauss (`VARI_ELGA`) $V3$ is one **indicator of cracking** who allows to know if the point of gauss is healthy ($V3=0$), damaged ($V3=1$) or broken ($V3=2$). See the example figure 4.5.
- The variable $V4$ vary continuously of 0 (healthy) to 1 (broken); it corresponds to **percentage of energy of dissipated surface** and can be interpreted like an indicator of damage. See the example figure 4.5.

- One can also to visualize the values of **jump of displacement** in the three modes of rupture I, II and III respectively starting from the internal variables $V7$, $V8$ and $V9$.
- One **indicator of dissipation** $V2$ allows to know if the point of gauss dissipates energy or is in discharge (the values taken by this internal variable depend on the cohesive law, to see Doc. [R7.02.11]).
- Finally one can visualize them components cohesive force in the three modes of rupture I , II and III respectively starting from the vector forced at the point of gauss $SIEF_ELGA$: $SIGN$, $SITX$ and $SITY$. The sign of $SIGN$ allows to know if the point is in contact or not.

Attention, for EJ visualization tangential jumps in mode II and III (respectively $V8$ and $V9$) does not have direction taking into account the fact that the tangential vectors of the local reference mark are independent from one element to another (see on this subject the part 6).

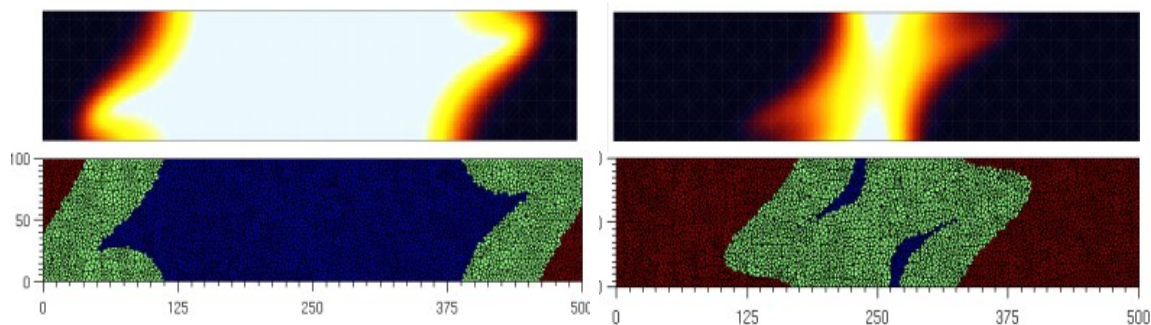


Figure 4.5 : Evolution of the rupture in mixed mode in a beam 3D, visualization of the plan of cracking. Internal variable $V4$ in top (1 in black, 0 in white) and $V3$ in bottom (0 in blue, 1 in green, 2 in red).

Visualization of cracking for the ED

For ED the internal variables are not the same ED one, the percentage of dissipated energy is stored in $V5$ and the indicator of dissipation in the variable $V4$. One also stores the constraints normal and tangential in $V6$ and $V7$ as well as the jumps of normal and tangential displacement in $V1$ and $V2$.

POST_CZM_FISS : calculation length of crack cohesive in 2D

This order makes it possible to know, at every moment, the size of the cohesive crack as well as the size of the cohesive zone starting from the punctual coordinates $POINT_ORIG$ and of a directing vector $VECT_TANG$ given by the user. This last must also inform the non-linear computation result $evol_noli$ as well as the group of cohesive meshes $grma$ on which it wishes to carry out postprocessing. Syntax is the following one:

```
table_sdaster = POST_CZM_FISS (RESULT = evol_noli,
                              GROUP_MA = grma,
                              POINT_ORIG = (R, R),
                              VECT_TANG = (R, R))
```

This order is usable for modelings $PLAN_JOINT$, $PLAN_INTERFACE$, $AXIS_JOINT$ and $AXIS_INTERFACE$ and cohesive laws CZM_OUV_MIX , CZM_TAC_MIX , CZM_LIN_REG , CZM_EXP_REG . The lengths of cracks are evaluated starting from the geometrical position of the points of Gauss and of the variable $V3$ of the cohesive laws interns which makes it possible to know the state of the point (healthy, damaged, broken). One will refer to U4 documentation of $POST_CZM_FISS$ for more information. An example of use is proposed in the tests $ssnp139a$ and $ssnp139c$.

Calculation length of crack in 3D

It is possible to extend the macro one above for the 3D but that is not envisaged in the short run.

5 Implementation of a calculation CZM

The cohesive models presented here were the object of a certain number of calculations in various mechanical problems, generally on structures targets (test-tubes). Let us quote some examples:

- Starting and propagation of crack in a beam rubber band or elastoplastic in brittle fracture,
- propagation of a crack under cyclic loading in an elastoplastic beam to model tiredness as well as the effect of overload.
- Propagation of a plane crack 3D in brittle fracture or tiredness, followed face of cracking.
- Modeling of propagation dynamics of crack in a bimatérial.
- Cracking and propagation brutal of two cracks through a perforated plate (quasi-static calculations with piloting).
- Wrenching of one reinforcement rigid.

In this part, one present various tests being pressed on the cohesive elements in *Code_Aster* then one comments on one of them.

5.1 List of the cases tests with the cohesive models

ELEMENTARY TEST, A COHESIVE ELEMENT AND AN ELASTIC ELEMENT

```
ssnp118a: EJ 2D PLAN,      LAW CZM_EXP_REG
ssnp118b: EJ 3D (HEXA),   LAW CZM_EXP_REG
ssnp118c: EJ 3D (PENTA),  LAW CZM_EXP_REG
ssnp118d: EJ 2D PLAN,     LAW CZM_LIN_REG
ssnp118e: EJ 3D (HEXA),   LAW CZM_LIN_REG
ssnp118f: EJ 3D (PENTA),  LAW CZM_LIN_REG
ssnp118g: EI 2D PLAN,     LAWS CZM_OUV_MIX AND CZM_TAC_MIX
ssnp118h: EI 3D (HEXA),   LAWS CZM_OUV_MIX AND CZM_TAC_MIX
ssnp118i: EI 3D (PENTA),  LAWS CZM_OUV_MIX AND CZM_TAC_MIX
ssnp118j: EI 2D PLAN,     LAW CZM_FAT_MIX
ssnp118k: EI 3D (HEXA),   LAW CZM_FAT_MIX
ssnp118l: EI 3D (PENTA),  LAW CZM_FAT_MIX
ssnp118m: EI 2D PLAN,     LAW CZM_TRA_MIX
ssnp118n: EI 3D (HEXA),   LAW CZM_TRA_MIX
ssnp118o: EI 3D (PENTA),  LAW CZM_TRA_MIX
ssnp118q: EI 2D PLAN,     LAW CZM_EXP_MIX
ssnp118r: EI 3D (HEXA),   LAW CZM_EXP_MIX
ssnp118s: EI 3D (PENTA),  LAW CZM_EXP_MIX
```

WRENCHING OF A RIGID REINFORCEMENT

```
ssna115a: ED 2D AXIS, LAW CZM_EXP
ssna115b: EJ 2D AXIS, LAW CZM_LIN_REG
ssna115c: EI 2D AXIS, LAW CZM_TAC_MIX
```

AXISYMMETRIC TEST-TUBE ENTAILLEE (AE) ELASTOPLASTIC

```
ssna120a: EI 2D AXIS, LAW CZM_TRA_MIX
```

CRACKING OF A PLATE, COMPARISON WITH ANALYTICAL SOLUTION

```
ssnp128a: ED 2D PLAN, LAW CZM_EXP
```

BRUTAL CRACKING OF A PLATE TROUEE WITH PILOTING OF THE LOADING

```
ssnp133a: EJ 2D PLAN, LAW CZM_EXP_REG
ssnp133b: ED 2D PLAN, LAW CZM_EXP
ssnp133c: EI 2D PLAN, LAW CZM_OUV_MIX
```

BEAM Compact Tension (CT) 2D ELASTOPLASTIC

```
ssnp151a: EI 2D PLAN, LAW CZM_TRA_MIX
```

BEAM Compact Tension (CT) ELASTOPLASTIC 3D

ssnp151b: EI 3D (HEXA), LAW CZM_TRA_MIX

BEAM Doubles Cantilever Beam (DCB) 2D ELASTOPLASTIC, MONOTONOUS AND TIRES

ssnp139a: EI 2D PLAN, LAW CZM_OUV_MIX

ssnp139b: EI 2D PLAN, LAW CZM_FAT_MIX (example with accompanying notes below)

BEAM Doubles Cantilever Beam (DCB) ELASTIC 3D

ssnv199a: EJ 3D (HEXA), LAW CZM_EXP_REG

ssnv199b: EJ 3D (PENTA), LAW CZM_EXP_REG

ssnv199c: EJ 3D (HEXA), LAW CZM_LIN_REG

ssnv199d: EJ 3D (PENTA), LAW CZM_LIN_REG

ssnv199e: EI 3D (HEXA), LAW CZM_OUV_MIX

ssnv199f: EI 3D (PENTA), LAW CZM_OUV_MIX

DYNAMIC PROPAGATION 2D AND DEPENDENCE IN TEMPERATURE OF the COHESIVE LAW

sdns105a.comm: EJ 2D PLAN, LAW CZM_EXP_REG

BEAM ENTAILLEE IN INFLECTION 3 POINTS

ssnp166a.comm: EI 2D PLAN, LAW CZM_EXP_MIX

5.2 Example with accompanying notes, beam DCB in fatigue

The test here is presented SSNP139, modeling B, corresponding to the simulation of the propagation of crack by tiredness in a beam Cantilever Beam (DCB) Doubles two-dimensional. One presents the diagram of the mechanical problem on figure 5.2 then the command file commented on by referring to the various advices lavished above. One underlines in red the passages specifically dedicated to the cohesive elements.

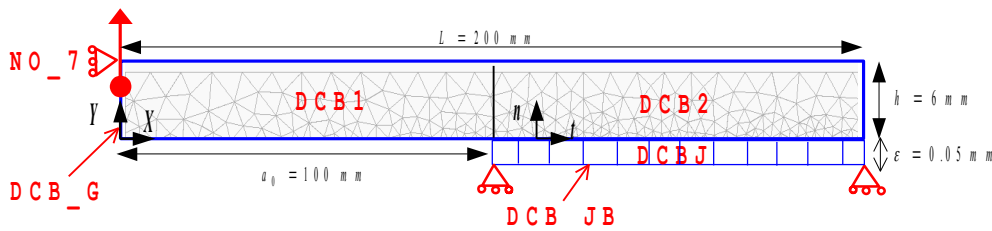


Figure 5.2 : Geometry of the DCB and boundary conditions.

```
#####
# PLASTIC BEAM DCB 2D WITH ELEMENTS of INTERFACE IN FATIGUE: LAW CZM_FAT_MIX
#####
```

```
BEGINNING (CODE=_F (NAME = 'SSNP139B'))
```

```
# LAWS OF BEHAVIOR OF the DCB AND the COHESIVE CRACK (see 5)
```

```
loi_vol=' VMIS_ISOT_LINE'
```

```
loi_czm=' CZM_FAT_MIX'
```

```
# MANY CYCLES MAXIMUM
```

```
nb_cy_max= 101
```

```
# FINAL MOMENT (ODD AND LOWER A 2*nb_cy_max-1)
```

```
tfin=5
```

```
# NUMBERS OF STEP (see 18)
```

```
nbpas = 5*tfin
```

```
# ONE FILES the TOPS (see 18)
```

```
nbarch = (tfin-1)/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 2021 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

VALUES OF THE FUNCTION OF LOADING

```
delta_U = 0.4
val_inf = 0.0
val_sup = val_inf + delta_U
val_surch = 1.2
```

NUM OF CYCLE FOR VARIABLE CHANGE AMPLITUDE

```
# to make a single overload at the moment 2*x+1 put: nb_cy_1 = X, nb_cy_2 = x+1, nb_cy_3=... ,
# Ci below, overload at the moment 23.
nb_cy_1 = 11
nb_cy_2 = 12
nb_cy_3 = nb_cy_max
```

PARAMETERS MECHANICAL OF LAW CZM (values tests, to see 13)

```
tenac = 0.259/2 # one models only one half fissures (division by two of Gc material to see 13)
sigc = 35
cd. = 2*tenac/sigc
```

PARAMETERS DIGITAL OF the ELEMENTS OF INTERFACE (see 13)

```
ri_gli=10
pen_lag=100
```

PARAMETERS OF the ELASTOPLASTIC LAW (values tests)

```
Young = 200000
pea = 0.3
splas = 30
nut = 3577
```

```
PRE_GMSH ()
MA0 = LIRE_MAILLAGE ()
```

PASSAGE OF the GRID INTO QUADRATIC (see 8)

```
MY = CREA_MAILLAGE (GRID = MA0, LINE_QUAD = _F (TOUT=' OUI'))
```

DEFINITION OF THE GROUPS

```
MY = DEFI_GROUP (
  reuse = MY,
  GRID = MY,
  CREA_GROUP_MA = (
```

```
# LINES
  _F (NAME = 'DCB_G', GROUP_MA = 'GM981'), # PLACE OF THE LOADING IMPOSES
  _F (NAME = 'DCB_JB', GROUP_MA = 'GM984'), # FACES FOR CONDITION OF SYMMETRY
```

```
# SURFACES
  _F (NAME = 'DCB1', GROUP_MA = 'GM988'), # LEFT VOLUMINAL LEFT THE BEAM
  _F (NAME = 'DCB2', GROUP_MA = 'GM989'), # LEFT VOLUMINAL RIGHT-HAND SIDE THE BEAM
  _F (NAME = 'DCBJ', GROUP_MA = 'GM990'), # ELEMENTS OF INTERFACE
),
```

```
  CREA_GROUP_NO = (
    _F (NAME = 'DCB_G', GROUP_MA = 'GM981'),
    _F (NAME = 'NO_7', GROUP_MA = 'GM954'), # NODE BLOCKS IN DIRECTION X
  )
)
```

ORIENTATION OF THE ELEMENTS OF INTERFACE (SEE 9)

```
MA=MODI_MAILLAGE (reuse=MA,
  MAILLAGE=MA,
  ORIE_FISSURE = _F (GROUP_MA= ' DCBJ '),
)
```

ASSIGNMENT OF MODELINGS

```
MO=AFFE_MODELE (
  MAILLAGE=MA,
  AFFE= (
    _F (
      GROUP_MA = ('DCB1', 'DCB2', 'DCB_G'),
      PHENOMENON = 'MECHANICAL',
      MODELING = 'D_PLAN'
    ),
    _F (
```

```
GROUP_MA = 'DCBJ',  
PHENOMENON = 'MECHANICAL',  
MODELING = 'PLAN_INTERFACE' # MODELING PLAN FOR THE ELEMENTS OF INTERFACE  
)  
)
```

DEFINITION OF the POSITION OF the CRACK IN the TOTAL REFERENCE MARK (see 6)

```
ORIEN_FI=AFPE_CARA_ELEM (MODELE=MO,  
MASSIF= (_F (GROUP_MA='DCBJ',  
ANGL_REP= (90, 0, 0))); # IN 2D ONLY the FIRST ANGLE
```

DEFINITION AND ASSIGNMENT OF MATERIAL

HERE ONLY ONE MATERIAL, ONE CAN ALSO CREATE TWO DIFFERENT MATERIALS ONE FOR VOLUME AND THE OTHER
FOR THE COHESIVE CRACK

```
MATE=DEFI_MATERIAU (  
ELAS=_F (  
E = Young,  
NAKED = pea),  
ECRO_LINE=_F (  
D_SIGM_EPSI=ecrou,  
SY=splas),  
RUPT_FRAG = _F (  
GC = tenac, # KEYWORD FOR THE CZM IN BRITTLE FRACTURE  
SIGM_C = sigc, # PARAMETER MATERIAL (see 13)  
PENNA_LAGR = pen_lag, # PARAMETER MATERIAL (see 13)  
RIGI_GLIS = ri_gli, # PARAMETER DIGITAL (see 13)  
)
```

```
CM=AFPE_MATERIAU (  
MAILLAGE=MA,  
AFPE=_F (  
GROUP_MA = ('DCB1', 'DCB2', 'DCBJ'),  
MATER = SUBDUE),  
)
```

LOADINGS

```
SYMMETRY = AFPE_CHAR_MECA (  
MODEL = MO,  
FACE_IMPO = (_F (GROUP_MA='DCB_JB', DY = 0)) # CONDITION OF SYMMETRY ON THE LOWER LIP  
)
```

```
TRACTION = AFPE_CHAR_MECA (  
MODEL = MO,  
DDL_IMPO = (_F (GROUP_NO = 'NO_7' , DX = 0)), # BLOCKING OF THE RIGID MODE  
FACE_IMPO= (_F (GROUP_MA = 'DCB_G' , DY = 1), # DIRECTION OF THE LOADING  
)
```

DEFINITION OF THE LIST OF MOMENT

```
deb= (val_inf/delta_U) - 0.0000001 # VALUE OF THE BEGINNING OF THE MOMENTS ALLOWING TO REACH A LEVEL  
# OF LOADING val_inf AT MOMENT 0 GRADUALLY
```

```
L_INST=DEFI_LISTE_REEL (BEGINNING = deb.,  
INTERVALLE= (  
_F (JUSQU_A = 0, NUMBER = 10),  
_F (JUSQU_A = tfin , NUMBER = nbpas),  
)
```

FILING OF MOMENT 0 AND THE MOMENTS ODD (TOPS OF EACH CYCLE)

```
INS_ARCH=DEFI_LISTE_REEL (BEGINNING = deb.,  
INTERVALLE= (  
_F (JUSQU_A = 0, NUMBER = 1),  
_F (JUSQU_A = 1, NUMBER = 1),  
_F (JUSQU_A = tfin , NUMBER = nbarch),  
)
```

DEFINITION OF THE CYCLIC EVOLUTION OF THE LOADING

a: FUNCTION CONSTANT AMPLITUDE WITH OVERLOAD (see 18)

```
has = []
```

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>)


```
for I in xrange (nb_cy_1):
    a.extend ((2*i, val_inf, 2*i+1, val_sup))

for I in arranges (nb_cy_1, nb_cy_2):
    a.extend ((2*i, val_inf, 2*i+1, val_surch))

for I in arranges (nb_cy_2, nb_cy_3):
    a.extend ((2*i, val_inf, 2*i+1, val_sup))

FCT_FAT = DEFI_FONCTION (
    NOM_PARA = 'INST',
    PROL_GAUCHE=' LINEAIRE',
    VALE      = has,
)

DEFLIST =DEFI_LISTE_INST (DEFI_LIST=_F (LIST_INST =L_INST),
    ECHEC= F (
        SUBD_METHODE=' UNIFORME',
        SUBD_PAS     = 3,
        SUBD_PAS_MINI = 1.E-10,
        SUBD_NIVEAU=10,
    ),)

# CALCULATION NONLINEAR

U=STAT_NON_LINE (
    MODEL     = MO,
    CHAM_MATER = CM,
    CARA_ELEM = ORIEN_FI,      # ASSIGNMENT OF THE CHARACTERISTICS OF ORIENTATION TO THE ELEMENTS OF
INTERFACE
    EXCIT     = (
        _F (LOAD = SYMMETRY),
        _F (LOAD = TRACTION, FONC_MULT = FCT_FAT),
    ),
    BEHAVIOR  = (
        _F (RELATION = loi_vol,  GROUP_MA = ('DCB1', 'DCB2')),
        _F (RELATION = loi_czm,  GROUP_MA = 'DCBJ'),
    ),
    INCREMENT = _F (LIST_INST = DEFLIST,
        INST_FIN = tfin,
    ),
    NEWTON    = _F (MATRIX = 'TANGENT' , REAC_ITER=1),
    CONVERGENCE = _F (
        RESI_REFE_RELA = 1.E-6,
        SIGM_REFE      = sigc,      # FORCED OF REFERENCE (see 14)
        DEPL_REFE      = cd.,       # DISPLACEMENT OF REFERENCE (see 14)
        ITER_GLOB_MAXI = 8,
    ),
    SOLVEUR   = _F (
        METHOD = 'MUMPS', PCENT_PIVOT = 100 ), # SOLVEUR MUMPS NECESSARY FOR the EI (see 14 )
    FILING   = _F (LIST_INST = INS_ARCH
    ))

# POST_TRAITEMENT: IMPRESSIONS OF the INDICATOR Of V3 DAMAGE AND the PERCENTAGE Of
ENERGY DISSIPEE V4 (see 19 )

IMPR_RESU (FORMAT=' RESULTAT', RESU= (_F (RESULTAT=U,
    NOM_CHAM=' VARI_ELGA',
    INST=tfin,
    NOM_CMP= ('V3', 'V4'))))

END (FORMAT_HDF=' OUI')
```

6 Conclusion

Generally, the use of the elements of interface (EI) is recommended when several choices are possible. Indeed, this approach joins together qualities of both others and fulfills best the requirements of robustness, reliability and performance desired for models intended in the long term for engineering. In dynamics, model EI is in the course of validation, one will prefer the model of joint to him. This last can answer a certain number of waitings provided that one uses it with prudence. Lastly, the element with internal discontinuity has the advantage of not regularizing energy, on the other hand it cumulates the disadvantages of the other approaches. One advises to use it only in quite particular cases, for example for multi-cracking in a given direction *a priori* (see [bib5]).

7 Bibliography

- [1] Debruyne G. and Laverne J. Propagation of crack in dynamics, digital simulation using elements CZM, *CR-AMA-09.192*, 2009.
- [2] Frankfurt G., Marigo J. - J. (1998) Revisiting brittle fracture ace year energy minimization problem. *J. Mech. Phys. Solids* **46** (8), 1319-1342.
- [3] Laverne J. A cohesive model for tiredness, effect of overload and propagation 3D, *CR-AMA-09.038* , 2009.
- [4] Laverne J., ductile Rupture with the cohesive models, *CR-AMA-09.074*, 2009.
- [5] Laverne J. Formulation energy of the rupture by cohesive models of force, *Doctorate, University Paris 13, November 2004*.
- [6] Lorentz E., Modeling and digital simulation of the damage of the structures. *Enabling To direct Research, University Paris 6, July 2008*.
- [7] Rice J.R. The mechanics of earth quake rupture, *Proceding of international School of Physics pp. 555-649 (1990)*.