

---

## FORMA05 - Practical works of the formation “advanced Use”: Plate fissured in traction

---

### Summary:

This test 2D (plane deformations) models into quasi-static a fissured plate put in traction. It makes it possible to validate Lbe postprocessings in linear elastic breaking process `CALC_G` and `POST_K1_K2_K3`. It comprises three modelings.

## 1 Problem of reference

### 1.1 Geometry

One considers a rectangular plate height  $H = 2\text{ m}$ , of width  $W = 1\text{ m}$ , in plane deformation, with an emerging horizontal crack of depth  $a = 0,1\text{ m}$ .

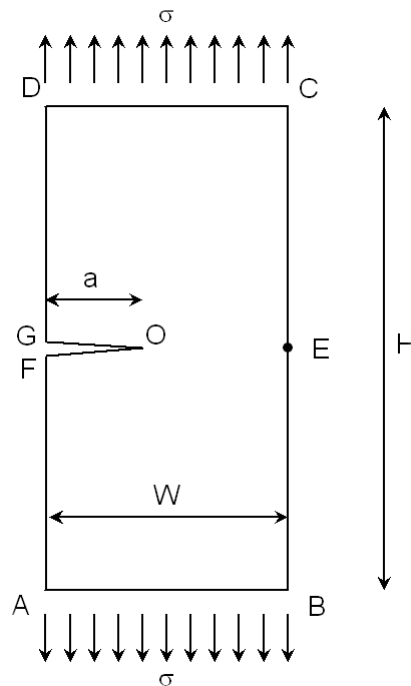


Figure 1.1-1: diagram of the fissured plate

### 1.2 Properties DU material

One considers a linear elastic isotropic material homogeneous whose characteristics are the following ones:

- Young modulus  $E = 210\,000\text{ MPa}$
- Coefficient of Fish  $\nu = 0,3$

### 1.3 Boundary conditions and loadings

The plate is in traction ( $\sigma = 10\text{ MPa}$ ).

## 2 Reference solution

---

### 2.1 Method used for the reference solution

The reference solution [1] is expressed in the following way:

$$K_I = \sigma \sqrt{\pi a} F\left(\frac{a}{W}\right)$$

$$\text{with } F\left(\frac{a}{W}\right) = 1,122 - 0,231\left(\frac{a}{W}\right) + 10,55\left(\frac{a}{W}\right)^2 - 21,71\left(\frac{a}{W}\right)^3 + 30,382\left(\frac{a}{W}\right)^4$$

The precision of this empirical formula is of 0,5 for  $\frac{a}{W} \leq 0,6$ .

One can also calculate  $G$  thanks to the formula of Irwin:

$$G(s) = \frac{(1 - \nu^2)}{E} K_I^2$$

### 2.2 Results of reference

With the digital values of the statement, one finds:  $K_I = 6,65 \text{ MPa} \cdot \sqrt{\text{m}}$  and  $G = 192 \text{ J.m}^{-2}$ .

### 2.3 Bibliographical references

- H. Tada, P. Paris, G. Irwin, The stress analysis of aces handbook, 3<sup>rd</sup> edition, 2000

## 3 Modeling a: FEM 2D

### 3.1 Unfolding of the TP

#### 3.1.1 Geometry and grid with Salomé-Meca

By taking of account the symmetry of the geometry defined on Figure 1, only the higher half of the model will be represented. This geometry can be built with Geometry module of Salomé-Meca by using the functionality `New Entiy/BASIC/2D Sketch` then `Build/Face`. One will take care to define the geometrical groups `O`, `E`, `CD`, `GO` and `OE` on the face thus created by using the functionality `New Entiy/Explode`.

With the module Mesh de Salome-Meca, by using Netgen 1D-2D: to choose `NETGEN 2D Parameters` then `Max. Size = 0,1 m`, `Min. Size = 0,005 m`, option `Fine Very` and coachman the box `Second Order` to obtain a quadratic grid directly. This algorithm also makes it possible to define a size of the meshes locally (mitre `Room sizes`). One will be able for example to specify elements of `0,005 m` near the bottom of crack. Do not forget by importing the groups of meshes and nodes according to the geometrical groups `O`, `E`, `CD`, `GO` and `OE`.

#### 3.1.2 Creation of the command file without postprocessing of the rupture

Reading of the grid ( <code>LIRE_MAILLAGE</code> ) with format MED;
Definition of the finite elements used ( <code>AFFE_MODELE</code> ) One opens documentation by a click-right on the name of the order and one finds modeling corresponding to a modeling in plane constraints for the elements of continuous mediums 2D ( <code>D_PLAN</code> )
Reorientation of the normals to the elements: one will use <code>MODI_MAILLAGE/ORIE_PEAU_2D</code> in the same way to direct all the elements, with a normal turned towards outside (to use the group <code>`CD`</code> );
Definition and assignment of material ( <code>DEFI_MATERIAU</code> and <code>AFFE_MATERIAU</code> );
Definition of the limiting conditions and loadings ( <code>AFFE_CHAR_MECA</code> ): <ul style="list-style-type: none"><li>• Symmetry on half of the plate <code>`OE`</code> (<code>DDL_IMPO</code>);</li><li>• Blocking of the rigid modes (<code>DDL_IMPO</code>);</li><li>• Application of traction on <code>`CD`</code> (<code>FORCE_CONTOUR</code>)</li></ul>
Resolution of the elastic problem ( <code>MECA_STATIQUE</code> );
For visualization with Paravis: <ul style="list-style-type: none"><li>• Calculation of the stress field extrapolated with the nodes (<code>CALC_CHAMP</code>, option <code>`CONSTRAINT`</code> with the field <code>`SIGM_NOEU`</code>)</li><li>• Calculation of the equivalent stress field (<code>CALC_CHAMP</code>, option <code>`CRITERIA`</code> with the field <code>`SIEQ_NOEU`</code>)</li></ul> For that, one will enrich the concept resulting from <code>MECA_STATIQUE</code> in taking again the same name of concept.
Impression of the results to format MED ( <code>IMPR_RESU</code> ).

#### 3.1.3 Calculation of the field of displacement

Launching of calculation with ASTK.

To visualize the fields of displacement and constraints obtained.

## 3.1.4 Postprocessing for the rupture

### a) Definition of the bottom of crack

To define the bottom of crack in `DEFI_FOND_FISS` starting from the groups of meshes of the bottom and lips. The model being symmetrical, it is necessary to specify `SYME=' OUI '`.

To specify the bottom of crack (`FOND_FISS`) and the upper lip (`LEVRE_SUP`).

### b) Calculation of G with `CALC_G`

To calculate the rate of refund of energy G with `CALC_G` (`OPTION=' CALC_G' [ value by default]` ).

To use the result of `MECA_STATIQUE` (`RESULT`).

To supplement information at once `THETA` :

- bottom of crack
- rays of the crown of the field theta (`R_INF`, `R_SUP`), to define according to the grid used.

Imprimer values of G (`IMPR_TABLE`).

### c) Calculation of K with `CALC_G`

To also calculate the factors of intensity of the constraints (K1, K2) (`OPTION=' CALC_K_G'` ).

Idem that previously.

To print the results in a new table (`IMPR_TABLE`).

### d) Calculation of K and G with `POST_K1_K2_K3`

To calculate K and G with `POST_K1_K2_K3` :

- to use the result of `MECA_STATIQUE` (`RESULT`)
- to inform the bottom of crack
- to inform the parameter `ABSC_CURV_MAXI`
- to print the results in a table (`IMPR_TABLE`)

To compare the solutions obtained with the reference solution.

## 3.1.5 To go further: Use of the elements of BARSOU

After the `LIRE_MALLAGE`, to modify the grid by transforming the elements into point of crack in elements of Barsoum (operator `MODI_MALLAGE/MODI_MAILLE/NOEUD_QUART` ) and to look at how the result is modified.

## 3.1.6 To go further: études of influence of the sizes of crowns

- On `CALC_G` : to check the independence of the result to the choice of the crowns of integration of the field theta;
- On `POST_K1_K2_K3` : to study the influence of the parameter `ABSC_CURV_MAXI`.

## 3.2 Sizes tested and results

### 3.2.1 Tests on G

Identification	Reference	% tolerance
$G$ of CALC_G, option CALC_G	192	0,3%
$G$ of CALC_G, option CALC_K_G	192	0,3%
$G_{Irwin}$ of CALC_G, option CALC_K_G	192	0,3%
$G$ of POST_K1_K2_K	192	0,6%

### 3.2.2 Tests on KI

Identification	Reference	% tolerance
$K_I$ of CALC_G, option CALC_K_G	$6.65 \cdot 10^6$	0,3%
$K_I$ of POST_K1_K2_K	$6.65 \cdot 10^6$	0,4%

## 4 Modeling B : FEM 2D without symmetry

### 4.1 Unfolding of the TP

#### 4.1.1 Recovery of grid with Salomé-Meca

For this TP, you can recover the grid forma05b.mmed directly. The groups are décrits on the Figure 4.1.1-1 :

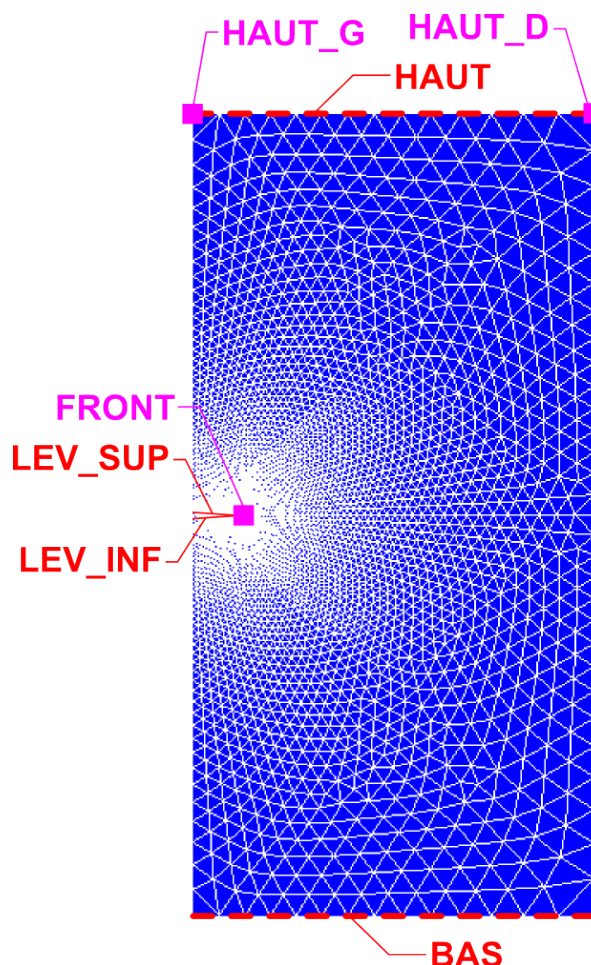


Figure 4.1.1-1: Grid forma05b with its groups

For recall, one seeks to apply a pressure equal to 10MPa to the line TOP and the line of BOTTOM. To avoid the movements of rigid body, it is necessary to block displacement according to X and Y on the group HAUT\_G and displacement according to Y on group HAUT\_D.

The group of node containing the face of the crack is FACE and the upper lips and lower are respectively defined by the groups of meshes LEV\_SUP and LEV\_INF.

#### 4.1.2 Creation of classical case in AsterStudy

To launch Salome\_Meca, then choose the AsterStudy module. Add the following stages to your case:

**Read has mesh (LIRE\_MAILLAGE).** Select the grid forma05b.mmed and format MED

**Modify has mesh (MODI\_MAILLAGE).** Choose the grid read previously and select it reuse. Select the action ORIE\_PEAU\_2D to reorientate the normals with the edges towards the outside of the grid (groups of elements HIGH, LOW, LEV\_SUP, LEV\_INF). [This stage obligatory here, but is not strongly advised in a general way]

<p><b>CREA_MALLAGE.</b> Select the preceding grid and give a new name. Choose the action <code>LINE_QUAD</code> to transform the linear elements into quadratic elements. [Choose the option <code>PREF_NOEUD=Z</code>. This option allows to change the prefix by default of the name of the nodes. This option is not obligatory in this case, but can become essential for large grids]</p>
<p>Assign finite element (<code>AFFE_MODELE</code>). Choose the mechanical phenomenon and the modeling of the continuous mediums 2D in deformation planes (<code>D_PLAN</code>)</p>
<p>Definition and assignment of material: <b>Define has material (<code>DEFI_MATERIAU</code>)</b> and <b>Assign has material (<code>AFFE_MATERIAU</code>)</b></p>
<p>Definition of the limiting conditions and loadings: <b>Assign mechanical load (<code>AFFE_CHAR_MECA</code>)</b> :</p> <ul style="list-style-type: none"> <li>• Blocking of the rigid modes (<b>Enforce DOF</b> on <code>HAUT_G</code> and <code>HAUT_D</code>);</li> <li>• Application of pressure on 'HIGH' and 'LOW) (<b>PRES_REP</b>)</li> </ul>
<p>Resolution of the elastic problem: <b>Static mechanical analysis (<code>MECA_STATIQUE</code>)</b>;</p>
<p>For visualization with Paravis:</p> <ul style="list-style-type: none"> <li>• Calculation of the stress field extrapolated with nodes (<code>CALC_CHAMP</code>, option 'FORCED' with the field 'SIGM_NOEU')</li> <li>• Calculation of the equivalent stress field (<code>CALC_CHAMP</code>, option 'CRITERIA' with the field 'SIEQ_NOEU')</li> </ul> <p>For that, one will enrich the concept resulting from <code>MECA_STATIQUE</code> in taking again the same name of concept.</p>
<p>Impression of the results to format MED: Results output (<code>IMPR_RESU</code>).</p>

You can cancel the case to check the good performance of your setting in data.

### 4.1.3 Addition of the stages of post treatment in breaking process

<p>In order to separate calculation and postprocessing, you can add a new stage (New internship) to your case of study (Box).</p>
<p><b>Definition of the bottom of crack ( <code>DEFI_FOND_FISS</code> ).</b> To define the bottom of crack in <code>DEFI_FOND_FISS</code> to leave D U group of node bottom <code>FACE</code> and <code>LEVRE_INF</code> and <code>LEVRE_SUP</code>.</p>
<p><b>Calculation D U rate of refund of energy with <code>CALC_G</code> (OPTION=' <code>CALC_G</code>' [ value by default] )</b></p> <p>To supplement information at once <code>THETA</code> :</p> <ul style="list-style-type: none"> <li>• bottom of crack <code>FOND_FISS</code></li> <li>• rays of the crown of the field <code>theta</code> ( <code>R_INF</code> , <code>R_SUP</code> ), to define according to the grid used.</li> </ul>

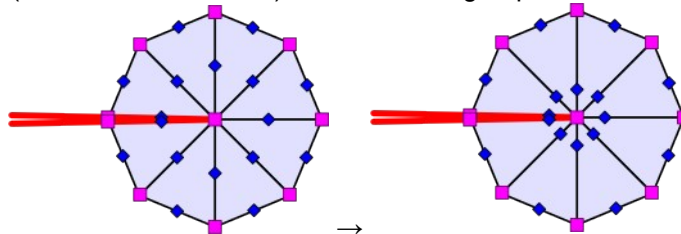


To print the values of G ( IMPR_TABLE ).
<b>Calculation of S factors of intensity of the constraints with CALC_G (OPTION=' CALC_K_G' ) .</b> Idem that previously. To print the results in a new table ( IMPR_TABLE ).
Calculate of K and G with POST_K1_K2_K3 : <ul style="list-style-type: none"><li>• to inform the bottom of crack</li><li>• to inform the parameter ABSC_CURV_MAXI</li><li>• to print the results in a table ( IMPR_TABLE )</li></ul>
To compare the solutions obtained with the reference solution.

## 4.1.4 To go further: Use of the elements of BARSOU

For that, add the following order afterwards CREA\_MAILLAGE :

**Modify has mesh (MODI\_MAILLAGE).** Choose the quadratic grid and it reuse. Action MODI\_MAILLE with the option NOEUD\_QUART to move the nodes mediums of the elements connected to the forefront of the crack to the quarter of the distance (elements of Barsoum). For recall the group of node of the bottom of crack is FACE.



## 4.1.5 To go further: Studies of influence of the sizes of crowns

- In CALC\_G : to check the independence of the result to the choice of the crowns of integration of the field theta;
- In POST\_K1\_K2\_K3 : to study the influence of the parameter ABSC\_CURV\_MAXI.

## 5 Modeling C : XFEM 2D without symmetry

### 5.1 Unfolding of the TP

#### 5.1.1 Recovery of grid with Salomé-Meca

For this TP, you can recover the grid forma05 directlyC.mmed. The provided grid is quadratic. The groups are describe on the Figure 5.1.1-1 :

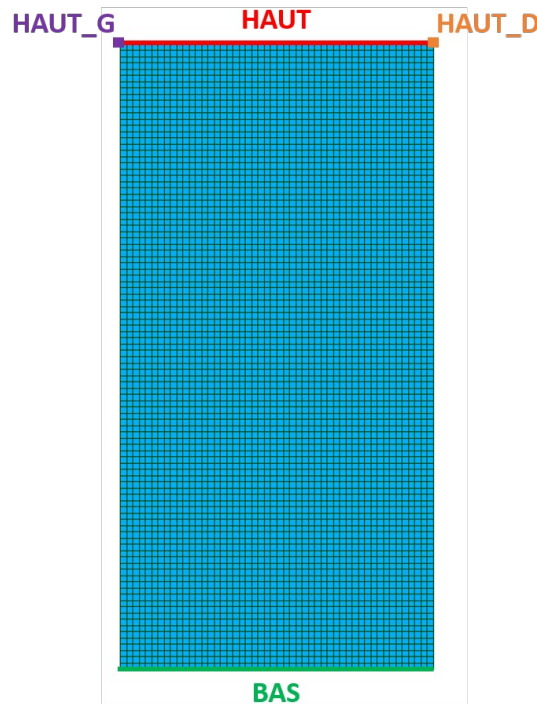


Figure 5.1.1-1: Grid forma05C with its groups

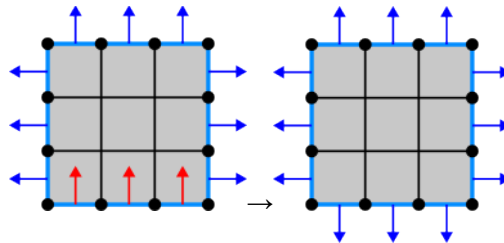
The limiting conditions and the loadings are the same ones as those of modeling B.

#### 5.1.2 Creation of classical case in AsterStudy

To launch Salome\_Meca, then choose the AsterStudy module. Add the following stages to your case:

**Read has mesh (LIRE\_MALLAGE).** Select the grid forma05C.mmed and format MED

**Modify has mesh (MODI\_MALLAGE).** Choose the grid read previously and select the reuse one. Select action ORIE\_PEAU\_2D to reorientate the normals with the edges towards the outside of the grid (groups of elements HIGH, LOW). [This stage obligatory here, but is not strongly advised in a general way]



**Assign finite element (AFFE\_MODELE).** Choose the mechanical phenomenon and the modeling of the continuous mediums 2D in deformation planes (D\_PLAN)

Definition and assignment of material: **Define has material (DEFI\_MATERIAU)** and **Assign has material (AFFE\_MATERIAU)**

In the mitre <b>Fracture and Tiredness</b> : Definition of the crack <b>DEFI_FISS_XFEM</b> with $a=0,1m$ . Use the catalogue of the cracks preferably ( <code>FORM_FISS='DEMI-DROITE'</code> )
In the mitre <b>Fracture and Tiredness</b> : Modification of the model to take into account elements X-FEM ( <b>MODI_MODELE_XFEM</b> )
Definition of the limiting conditions and loadings: <b>Assign mechanical load (AFFE_CHAR_MECA)</b> : <ul style="list-style-type: none"><li>• Blocking of the rigid modes (<b>Enforce DOF</b> on <code>HAUT_G</code> and <code>HAUT_D</code>)</li><li>• Application of pressure on <code>'HIGH'</code> and <code>'LOW'</code> (<b>PRES_REP</b>)</li></ul>
Resolution of the elastic problem: <b>Static mechanical analysis (MECA_STATIQUE)</b>

In order to separate calculation and postprocessing, you can add a new stage (New internship) to your case of study (Box).

In the mitre <b>Fracture and Tiredness</b> : Creation of a grid of visualization ( <b>POST_MAIL_XFEM</b> )
In the mitre <b>Finite Element</b> : Creation of a model for visualization ( <b>AFFE_MODELE</b> ) on the grid created for visualization
In the mitre <b>Fracture and Tiredness</b> : Creation of a field of results on the grid of visualization X-FEM ( <b>POST_CHAM_XFEM</b> )
For visualization with Paravis: <ul style="list-style-type: none"><li>• Calculation of the stress field extrapolated with the nodes (<code>CALC_CHAMP</code>, option <code>'CONSTRAINT'</code> with the field <code>'SIGM_NOEU'</code>)</li><li>• Calculation of the equivalent stress field (<code>CALC_CHAMP</code>, option <code>'CRITERIA'</code> with the field <code>'SIEQ_NOEU'</code>)</li></ul> For that, one will enrich the concept resulting from <b>POST_CHAM_XFEM</b> in taking again the same name of concept.
Impression of the results to format MED: Results output ( <b>IMPR_RESU</b> ).

### 5.1.3 Addition of the stages of post treatment in breaking process

For the postprocessing of the rupture you can also add a new stage (New internship) to your case of study in progress.

For the whole of the following stages, it will be necessary to go in the mitre **Fracture and Tiredness**.

#### a) Calculation of G with **CALC\_G**

To calculate the rate of refund of energy G with **CALC\_G** (`OPTION='CALC_G'`).

To use the result of static calculation not to confuse with the result created for visualization with **POST\_CHAM\_XFEM** (`RESULT`).

To supplement information at once **THETA** :

- bottom of crack **CRACK**
- rays of the crown of the field theta (`R_INF`, `R_SUP`), to define according to the grid used (the size of the elements is of 0,02m).

To print the values of the table G (**IMPR\_TABLE** in the mitre **Output**).

#### b) Calculation of K with **CALC\_G**

To also calculate the factors of intensity of the constraints (K1, K2) (`OPTION='CALC_K_G'`).

Idem that previously.

To print the results in a new table.

#### c) Calculation of K and G with **POST\_K1\_K2\_K3**

To calculate K and G with **POST\_K1\_K2\_K3** :

*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 2019 EDF R&D - Licensed under the terms of the GNU FDL (<http://www.gnu.org/copyleft/fdl.html>)

- to use the result of MECA\_STATIQUE (RESULT)
- to inform Lcrack has
- to inform the parameter ABSC\_CURV\_MAXI
- to print the results in a table.

To compare the solutions obtained with the reference solution.

## 5.1.4 To go further: Studies of influence of the sizes of crowns

- On CALC\_G : to check the independence of the result to the choice of the crowns of integration of the field theta;
- On POST\_K1\_K2\_K3 : to study the influence of the parameter ABSC\_CURV\_MAXI .

## 6 Synthesis

---

This TP makes it possible to highlight:

- Two methods of calculating of G and K available in Code\_Aster
- Improvement of the precision of the results with L " use of the elements of Barsoum (FEM)
- The possibility of simulating a crack without netting it (X-FEM)
- The invariance of the results to the choices of the crowns;