

FORMA07 - Practical works of the formation “advanced Use”: circular crack in infinite medium

Summary:

This test 3D into quasi-static, enters within the framework of the validation of postprocessings in linear elastic breaking process. The structure comprising a circular crack is put in traction.

1 Problem of reference

1.1 Geometry

One is interested in a cube subjected to a tractive effort comprising a horizontal circular crack of ray $a = 2\text{ m}$. A cube on side will be considered $10a$.

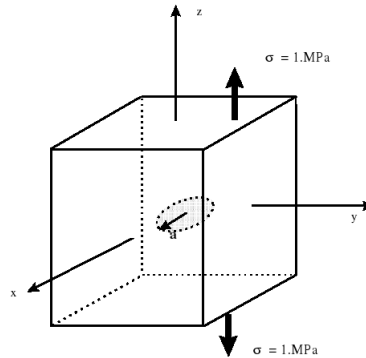


Figure 1.1-a: geometry

1.2 Material properties

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

- Young modulus $E = 200\,000\text{ Mpa}$,
- Coefficient of Fish $\nu = 0,3$.

1.3 Boundary conditions and loadings

The structure is in traction ($\sigma = 1\text{ MPa}$).

2 Reference solution

2.1 Method used for the reference solution

For a circular crack of ray a in an infinite medium, subjected to a uniform traction σ according to the normal with the plan of the lips, the rate of refund of energy room $G(s)$ is independent of the curvilinear X-coordinate along the face of crack s and expresses itself in the following way [1]:

$$G(s) = \frac{(1-\nu^2)}{\pi E} 4\sigma^2 a$$

The stress intensity factor $K_I(s)$ is given by the formula of Irwin:

$$G(s) = \frac{(1-\nu^2)}{E} K_I^2, \text{ that is to say } K_I(s) = \frac{2\sigma\sqrt{a}}{\sqrt{\pi}}$$

2.2 Results of reference

With the digital values of the statement, one finds: $K_I = 1,5957\text{ MPa} \cdot \sqrt{m}$ and $G = 11,59\text{ J.m}^{-2}$.

2.3 Bibliographical references

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

3 Modeling a: FEM 3D

3.1 Unfolding of the TP

3.1.1 Grid

The quadratic grid of the structure is provided to format MED: `forma07a.mmed`. By taking into account certain symmetries of the problem, only a quarter of the structure is represented. One could only represent $1/8^{\text{ème}}$ structure. The grid was generated with software GIBI, and of the tori are defined around the bottom of crack:

- ray of the smallest torus: $0,12\text{ m}$,
- ray of the largest torus: $0,53\text{ m}$.

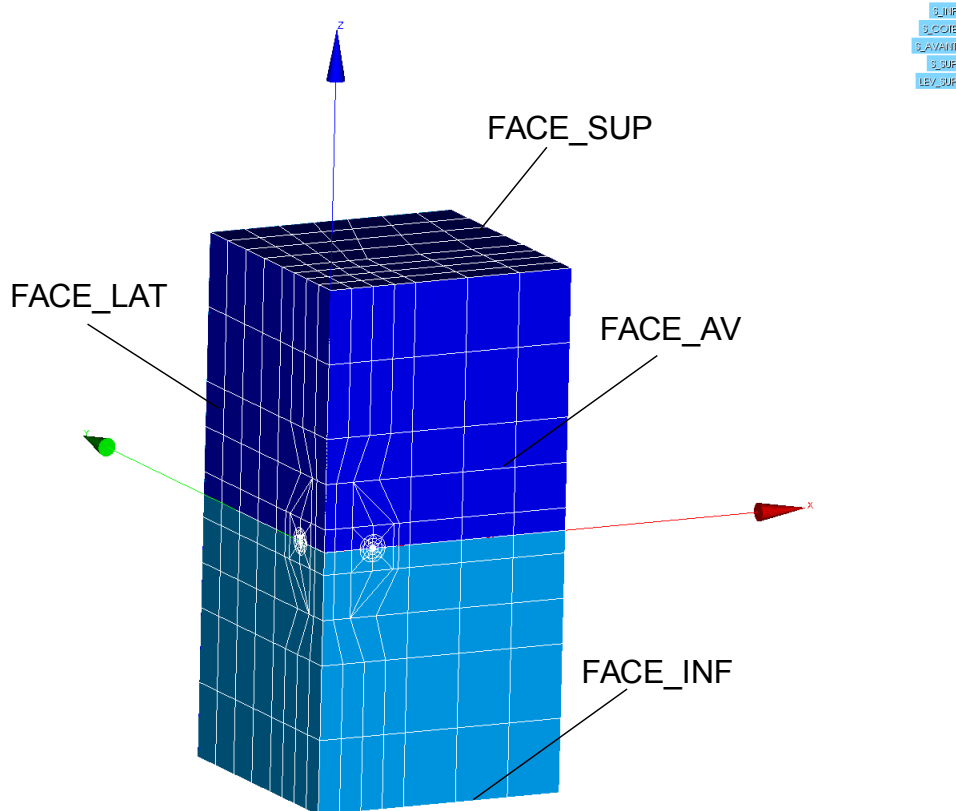


Figure 3.1.1-a: Grid

The nodes mediums of the edges of the elements touching the bottom of crack are moved with the quarter of these edges.

3.1.2 Creation of the command file without postprocessing of the rupture

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

Read has mesh (LIRE_MALLAGE). Select the grid `forma07a.mmed` and format MED

Modify has mesh (MODI_MALLAGE). Choose the grid read previously and select it `reuse`. Select the action `ORIE_PEAU_3D` to reorientate the normals with faces towards the outside of the grid (groups of elements `FACE_AV`, `FACE_LAT`, `FACE_SUP`, `FACE_INF`, `LEV_INF` and `LEV_SUP`).

Assign finite element (`AFFE_MODELE`). Choose the mechanical phenomenon and the modeling of the continuous mediums 3D (`3D`)

Definition and assignment of material: **Define has material (DEFI_MATERIAU)** and **Assign has material (AFFE_MATERIAU)**

Definition of the limiting conditions and loadings: **Assign mechanical load (AFFE_CHAR_MECA)** :

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>)

<ul style="list-style-type: none"> • Symmetry as regards symmetry 'FACE_LAT' (Enforce DOF); • Symmetry as regards symmetry 'FACE_AV' (Enforce DOF); • Blocking of the rigid modes (Enforce CF on GROUP_NO 'OF'); • Application of traction on 'FACE_SUP' and 'FACE_INF' (PRES_REP)
Resolution of the elastic problem: Static mechanical analysis (MECA_STATIQUE) ;
For visualization with Paravis: <ul style="list-style-type: none"> • Calculation of the stress field extrapolated with the nodes (CALC_CHAMP, option ' CONSTRAINT ' with the field 'SIGM_NOEU ') • Calculation of the equivalent stress field (CALC_CHAMP, option ' CRITERIA ' with the field 'SIEQ_NOEU') <p>For that, one will enrich the concept resulting from MECA_STATIQUE in taking again the same name of concept.</p>
Impression of the results to format MED: Results output (IMPR_RESU).
To visualize the fields of displacement and constraints obtained in Paravis

3.1.3 Postprocessing for the rupture

In order to separate calculation and postprocessing, you can add a new stage (New internship) to your case of study (Box).
Definition of the bottom of crack (DEFI_FOND_FISS). To define the bottom of crack in DEFI_FOND_FISS to leave D U group of meshes bottom LFF and lips LEV_INF and V_SUP.
Calculation D U rate of refund of energy with CALC_G (OPTION=' CALC_K_G'). To supplement information at once THETA : <ul style="list-style-type: none"> • bottom of crack FOND_FISS • rays of the crown of the field theta (R_INF , R_SUP), to define according to the grid used. To print the values of G (IMPR_TABLE).
Calculate of K and G with POST_K1_K2_K3 : <ul style="list-style-type: none"> • to inform the bottom of crack • to inform the parameter ABSC_CURV_MAXI • to print the results in a table (IMPR_TABLE)
To trace the values of G and K1 exits of CALC_G and of POST_K1_K2_K3 according to the curvilinear X-coordinate of the face of crack (column 'ABSC_CURV') in a spreadsheet.

3.1.4 To go further: Study of the influence of smoothing 3D

In CALC_G, to inform the keyword SMOOTHING with the keywords LISSAGE_G=' LAGRANGE' and LISSAGE_THETA=' LAGRANGE'. To compare the values with those obtained previously. For recall, when SMOOTHING is not informed, the value by default is SMOOTHING_G=' LEGENDRE' and LISSAGE_TEHTA=' LEGENDRE'.

In the second time, to add the keyword NB_POINT_FOND= 5 with the smoothing of Lagrange then to observe the values of G or K1 along the face. To also compare the computing times of CALC_G with and without NB_POINT_FOND of well informed.

3.2 Sizes tested and results

Identification	Reference	% tolerance
K_I of POST_K1_K2_K3	1.5957 10 ⁶	0.8%

4 Modeling b: X-FEM 3D

The problem considered is the same one as that studied in part II. However, contrary to the preceding part where the crack was with a grid, one considers in this part a healthy grid. The crack is thus not with a grid. For that, method X-FEM is used.

4.1 Unfolding of the TP

4.1.1 Refinement of the maillage

The objective of this first stage is of to use LOBSTER to obtain a refined grid starting from a relatively coarse healthy grid .

You can Créer coarse grid with Salomé (¼ of the structure is modelled) or to use that provided:

`forma07b.mmed`

In order to carry out the refinement of the grid, it is necessary to use a loop python. The command file of this part is thus not éditable graphically with AsterStudy. One will thus use the text mode of AsterStudy for the first stage of calculation.

Launch Salomé-Meca then choose AsterStudy. Go in the preferences (File → Preferences → AsterStudy) and strip external editor. Create a new stage for your calculation (New Internship) which you can re-elect (Refinement) if you wish it. Make a left click on this new stage and click on Edict. A window will appear on the left and you will be able to publish your command file.

Initially, it will be necessary to calculate the iteration count necessary to obtain the refined grid.

For that, the library maths should first of all be imported: importation maths

Inform following information:

- Size of mesh in the zone of insertion of the crack h_0
- Size of mesh targets h_c ($a/20$.)

Calculation amongst iterations to reach h_c :

- $n = (\ln(h_0) + \ln(h_c)) / \ln(2)$
- $nb_raff = E(N)$ (whole part of N : $\text{int}(N)$)
- $nb_iter = nb_raff + 1$

Calculation of the final size of the elements close to the face of crack:

- $h = h_0 / 2^{nb_raff}$

Initialize the whole of the vectors which will be used in the loop of refinement:

- $MY = [Nun] * (nb_iter + 1)$ (Grid)
- $CHERR = [Nun] * nb_iter$ (Error)
- $FISS = [Nun] * nb_iter$ (Crack)

To see the grid with format MED (**LIRE_MAILLAGE**): $MY[0] = \text{LIRE_MAILLAGE} (\text{UNITE}=20, \text{FORMAT}='MED')$;

To choose the grid `forma07b.mmed` that it will be necessary to add in the mitre **Dated Files** under unit 20.

Carry out the loop of refinement:

for i_raff in arranges (nb_raff):

- **Definition of the crack**

```
FISS[i_raff]=DEFI_FISS_XFEM (MAILLAGE=MA [i_raff],  
                             TYPE_DISCONTINUE='FISSURE',  
                             DEFI_FISS=_F (FORM_FISS='ELLIPSE',  
                             DEMI_GRAND_AXE=a,  
                             DEMI_PETIT_AXE=a,  
                             CENTRE=(0,0,0),  
                             VECT_X=(1., 0., 0.),  
                             VECT_Y=(0., 1., 0.,),),);
```

- **Calculation of the criterion of refinement**

```
CHERR [i_raff] =RAFF_XFEM (FISSURE=FISS[i_raff], TYPE='ZONE', RAYON=r_raff) with  
r_raff=5*h
```

- **Definition of the type of the grid n+1 --> to say that MAILLAGE_NP1 is standard 'a grid'**
MY [i_raff+1] =CO ('MA_%d' % (i_raff+1))
- **Adaptation of the grid**
MACR_ADAP_MAIL (ADAPTATION = 'REFINEMENT',
CHAM_GD = CHERR [i_raff],
CRIT_RAFF_ABS = 0.5,
DIAM_MIN = hc,
MAILLAGE_N = MY [i_raff],
MAILLAGE_NP1 = MY [i_raff+1],);

To visualize the new grid, print it with format MED:

IMPR_RESU (FORMAT=' MED', UNITE=80, RESU=_F (MAILLAGE=MA [nb_raff]));

To choose the grid forma07b_raff.mmed that it will be necessary to add in the mitre **Dated Files** under unit 80.

If you wish to pass this stage, you can also directly use the provided refined grid:

forma07b.41

In the provided grids (visualize them in Salomé), the names of the groups of meshes are the following:

- group of nodes: D,
- groups of faces: FACE_AV, FACE_LAT, FACE_SUP, FACE_INF,
- group of volumes: CUBIC.

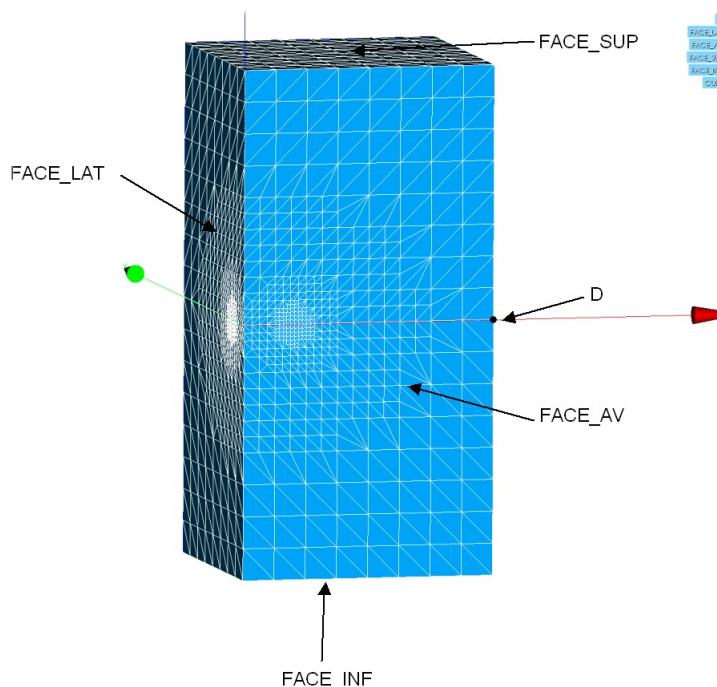


Figure 4.1.1-a: Groups defined on the provided grid

4.1.2 Creation of the command file without postprocessing of the rupture

a) Reading of the refined healthy grid and definition of the model not enriched

LaunchZ Salomé-Meca, then choose the AsterStudy module. Add the following stages to your case.

Read has mesh (LIRE_MAILLAGE). Select the grid refined that you created and format MED

Modify has mesh (MODI_MAILLAGE). Choose the grid read previously and select the reuse one. Select action ORIE_PEAU_3D to reorientate the normals with faces towards the outside of the grid (groups of elements FACE_AV, FACE_LAT, FACE_SUP, FACE_INF)

Assign finite element (AFFE_MODELE). Choose the mechanical phenomenon and the modeling of

the continuous mediums 3D

b) Definition of the crack and elements X-FEM

In the mitre **Fracture and Tiredness** : Definition of the crack **DEFI_FISS_XFEM**. Use the catalogue of the cracks preferably (**FORM_FISS='ELLIPSE'**)

In the mitre **Fracture and Tiredness** : Modification of the model to take into account elements X-FEM (**MODI_MODELE_XFEM**)

c) Definition of material, the conditions and resolution of the mechanical problem

Definition and assignment of material: **Define has material (DEFI_MATERIAU)** and **Assign has material (AFFE_MATERIAU)**

Definition of the limiting conditions and loadings on the enriched model. **Assign mechanical load (AFFE_CHAR_MECA)** :

- Symmetry as regards symmetry **'FACE_LAT'** (**Enforce DOF**);
- Symmetry as regards symmetry **'FACE_AV'** (**Enforce DOF**);
- Blocking of the rigid modes (**Enforce DOF** knownR 'Of)
- Application of E traction on **'FACE_SUP'** and **'FACE_INF'** (**PRES_REP**)

Resolution of the elastic problem: **Static mechanical analysis (MECA_STATIQUE)** with the enriched model

d) Postprocessing of displacements and the constraints with X-FEM and visualization with Paravis

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

In the mitre **Fracture and Tiredness** : Creation of a grid of visualization (**POST_MAIL_XFEM**)

In the mitre **Finite Element** : Creation of a model for visualization (**AFFE_MODELE**) on the grid created for visualization

In the mitre **Fracture and Tiredness** : Creation of a field of results on the grid of visualization X-FEM (**POST_CHAM_XFEM**)

For visualization with Paravis:

- Calculation of the stress field extrapolated with the nodes (**CALC_CHAMP**, option **'CONSTRAINT'** with the field **'SIGM_NOEU'**)
- Calculation of the equivalent stress field (**CALC_CHAMP**, option **'CRITERIA'** with the field **'SIEQ_NOEU'**)

For that, one will enrich the concept resulting from **POST_CHAM_XFEM** in taking again the same name of concept.

Impression of the results to format MED: **Results output (IMPR_RESU)**.

4.1.3 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 and factors of intensity of the constraints with `CALC_G` (`OPTION=' CALC_K_G'`) and the smoothing of Legendre of degree 5 .

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.

To print the values of the table G (`IMPR_TABLE` in the mitre `Output`).

b) 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 Lcrack has
- to inform the parameter `ABSC_CURV_MAXI`
- to print the results in a table.

Compare the results with the analytical solution. Compare times between 2 orders.

To test smoothing `LAGRANGE`. What does one observe T?

In order to improve the results, to test the uniform distribution of n points along the bottom of crack (`CALC_G/THETA/NB_POINT_FOND`) with various values of n .

Other tracks: refinement/enrichment on several layers.

4.2 Sizes tested and results

Identification	Reference	% tolerance
K_I of <code>POST_K1_K2_K3</code>	1.5957 10 ⁶	1%