

FORMA07 - Practical works of training “advanced Use”: fissure circular in Summarized infinite

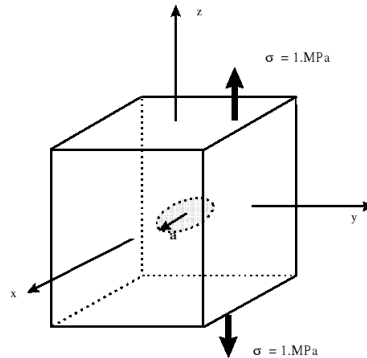
medium:

This test 3D into quasi-static, enters the frame of the validation of postprocessings in linear elastic fracture mechanics. The structure comprising a circular crack is put in tension.

1 Problem of reference

1.1 Geometry

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



Appear 1.1-a: geometry

1.2 Material properties

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

- Poisson's ratio, Young $E = 200\,000\text{ MPa}$ modulus
- $\nu = 0,3$.

1.3 Boundary conditions and loadings

the structure is in tension ($\sigma = 1\text{ MPa}$).

2 Reference solution

2.1 Method used for the reference solution

For a circular crack of radius a in an infinite medium, subjected to a uniform tension σ according to the norm with the plane of the lips, local rate of energy restitution $G(s)$ is independent of the curvilinear abscisse along the crack front s and is expressed 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 numerical values of the statement, one finds: $K_I = 1,5957\text{ MPa} \cdot \sqrt{\text{m}}$ 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 Modelization a: FEM 3D

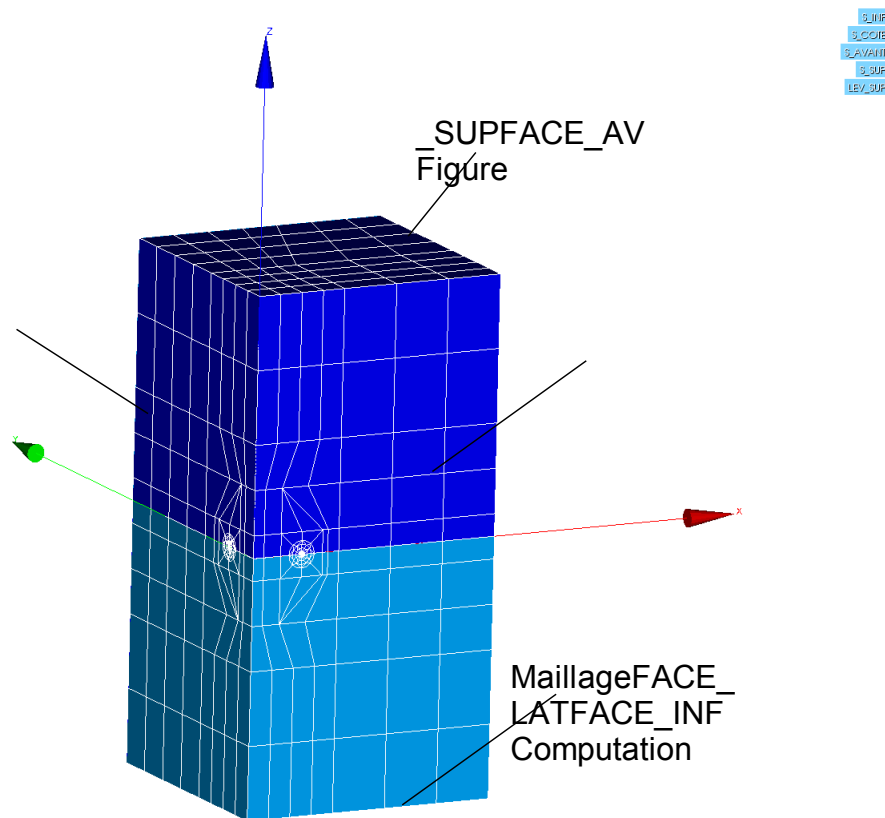
3.1 Unfolding of the TP

3.1.1 Maillage

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

- of the smallest torus: , 0,12 m radius
- of the largest torus: . 0,53 m

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



3.1 3.1.1-a

3.1.2 of the field of displacement

the command file with the model (3D), the material (elastic linear), the loadings (tension) and the conditions of symmetry on the two side sides is provided: file formed 07a .com. To visualize

the fields of displacement and stresses obtained. Computation

3.1.3 of K and G In

the provided file, the crack tip is already defined with `DEFI_FOND_FISS` from the mesh groups of the bottom and the lips. To calculate local rate of energy restitution (lissage by default) and total with `CALC_G` and to compare with the reference solution. For

the computation of the factors of intensity of the stresses along the crack tip, to use POST
_K1_K2_K3 and to compare the solution obtained with the reference solution. Studies

3.1.4 influence Studied

the influence of the options of lissage in CALC_G (options CALC_G) : degree of the polynomial of
LEGENDRE , lissage LAGRANGE and its alternatives... One

will compare the results got by plotting the curves. (

to save time, you can die-comment the last instructions of the file formed 07a .comm) Quantities

3.2 tested and results Identification

Reference	%	tolerance formulates
K_I POST _K1_K2_K3 3D	1,5957 106	0.4%

4 Modelization b: X-FEM

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 sane mesh. The crack is thus not with a grid. For that, the method X-FEM is used. Unfolding

4.1 of the TP Maillage

4.1.1 From

a relatively coarse sane mesh, to use HOMARD to obtain a refined mesh. To create the coarse mesh with Salomé or to use that provided: formed

07b.mmed a procedure

of refinement to be supplemented is provided. She calls on Homard in a loop python (file not éditable with Efficas). Note that to use it you must define crack (see following paragraph): 07b.comm

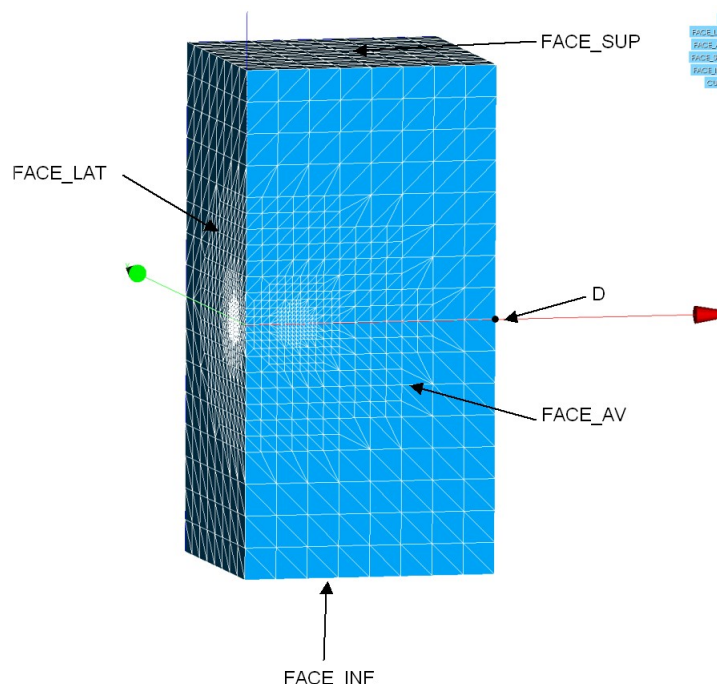
formed If you

wish to pass this stage, you can also directly use the provided refined mesh: forma07 B

.41 In

the provided meshes (visualize them in Salomé), the names of the mesh groups are the following: nodes group

- : D, groups
- of sides: FACE_AV, FACE_LAT, FACE_SUP, FACE_INF, group of
- volumes: CUBIC. Figure 4.1.1-a :



4.1.1-a defined on the mesh provided Definition

4.1.2 of crack and the elements X-FEM the crack

is defined by the command DEFI_FISS_XFEM . Use the crack catalog preferably. This command requires a mesh and a model. The elements

X-FEM are then defined via a new model, by the command `MODI_MODELE_XFEM` . Computation of

4.1.3 the field of displacement the command file

with the model (3D), the material (elastic linear), the loadings (tension) and the conditions of symmetry on the two side sides is provided, it misses only the definition of crack (a base is created to facilitate postprocessing -> to use `astk`): `forma07 b.com 1 To visualize`

the fields of displacement and stresses obtained. Computation of K

4.1.4 and G In poursuite

, calculate and along G K the crack tip with commands `CALC_G` and `POST_K1_K2_K3`. Compare the results with the analytical solution. Compare times between the 2 commands. To test

lissage LAGRANGE. Qu "does one observe T? So D" to improve

the results, to test the uniform distribution of points along n crack tip (`CALC_G/THETA/NB_POINT_FOND`) with various values of formula. Other n

: refinement/enrichment on several Quantities layers tested

4.2 and results Identification

Reference	% tolerance	formulates
K_I $K2_K$ 1,5957 106	0,4%	0.4%