FORMA05 - Practical works of training “advanced Use”: Plate fissured in tension

Summarized:

This test $2D$ strain plane, into quasi-static, enters the frame of the validation of postprocessings in linear elastic fracture mechanics. The fissured plate is put in tension.

1 Problem of reference

1.1 Geometry

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

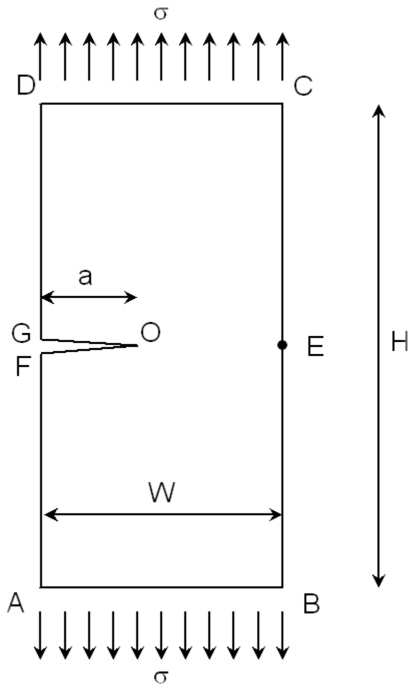

Figure 1.1-1: diagram of the fissured plate

1.2 Material properties

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

- Young modulus $E=210\ 000\,MPa$
- Poisson's ratio $\nu=0,3$

1.3 Boundary conditions and loadings

the plate is in tension ($\sigma=10\,MPa$).

2 Reference solution

2.1 Method used for the reference solution

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

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

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

2.3 Bibliographical references

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

3 Modelization a: FEM 2d

3.1 Déroulement of the TP

3.1.1 Géométrie and mesh with Salome-Meca

In taking into 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 modulus of Salome-Meca by means of the functionality `New Enity/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 means of the functionality `New Enity/Explode.`

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

3.1.2 Computation of the field of displacement

Besides the tensile force imposed on the segment CD (to be imposed with `FORCE_CONTOUR`), it is necessary to take into account the condition of symmetry, by blocking the following displacement Y of the segment OE .

To prevent rigid body motions, one will block the side displacement of the node E (or O). To visualize the fields of displacement and stresses obtained.

(to save time, you can make use of the file `forma05a.comm) Computation`

3.1.3 of K and G To define

the crack tip in DEFI `_FOND_FISS from` the mesh groups of the bottom and the lips. To calculate the factors of intensity of the stresses and rate of energy restitution with POST `_K1_K2_K3 and` CALC_G `.` Parameters of operators (R_INF`,` R_SUP `,` ABSC_CURV_MAXI …) will be to define according to the mesh used. One is reminded that

commands POST `_K1_K2_K3 and` CALC_G `produce` data structures of the type Counts. It is necessary to add command IMPR_TABLE to display the computation results. To compare

the solution obtained with the reference solution. To put

elements of Barsoum in crack tip (operator MODI `_MAILLAGE/MODI_MAILLE/NOEUD_QUART)` and to look at how result is modified. Studies

3.1.4 of influence On

- CALC_G `:` to check the independence of result with the choice of integration contours of the field theta; On
- POST `_K1_K2_K3:` to study the influence of parameter ABSC_CURV_MAXI `;` One
- can also modify the refinement of the mesh (Local *`sizes with`* NETGEN 1D-2D). Quantities

3.2 tested and Tests results

3.2.1 on G Identification

Reference	%	tolerance of
G CALC_G , option CALC_G 192	192	0.3%
G CALC_G , option CALC_K_G 192	192	0.3%
G_{Irwin} CALC_G , option CALC_K_G 192	192	0.3%
G POST _K1_K2_K 192	192	0.6%

3.2.2 on KI Identification

Reference	%	tolerance of
K_I CALC_G , option CALC_K_G 6,65	106 0,3%	0.3%
K_I POST _K1_K2_K 6,65	106 0,4%	0.4%

4 This

TP make it possible to highlight:

- Interest of a quadratic mesh compared to a linear mesh;
- The invariance of the results to the choices of contours;
- Improvement of the accuracy of the results resulting from POST _K1_K2_K thanks to the use of elements of Barsoum.