

FORMA06 - Practical works of training “advanced Use”: plate multi-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 plate multi-is fissured. The cracks are represented by the method X-FEM.

1 Problem of reference

One studies the behavior of a plate multi-fissured in tension. In order not to net cracks, the method X-FEM is used.

One considers an infinite plate in tension, comprising 2 cracks length $2a$ (see Figure 1-1).

A Pair of Eccentric Parallel Cracks (Isida [1973])

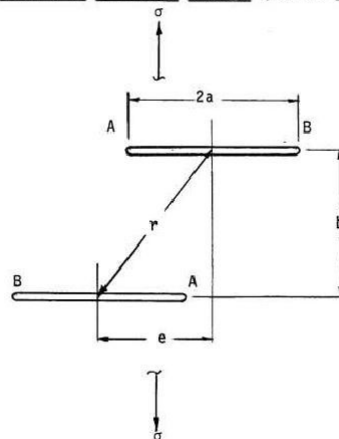


Figure 1-1: geometry of the problem

One proposes to check the abacuses provided by the “Handbook of stress-intensity factors” of $G. S. Sih$. The factor of intensity of the stresses K_I at the point A is given by the following formula:

$$K_I^A = F\left(\frac{2a}{r}, \frac{e}{b}\right) \sigma \sqrt{\pi a} \text{ where } F\left(\frac{2a}{r}, \frac{e}{b}\right) \text{ is given by the graphics of 2.1 2.1-1.}$$

2 Modelization A

2.1 Unfolding of the TP

Under Salomé-Méca, carry out the geometry.

One will be able to consider a plate centered in the beginning, of finished size: $2m$ of with dimensions.

Carry out the mesh. It is pointed out that the cracks are not with a grid, one will be able to thus use a sufficiently fine structured mesh of quadrangles everywhere (algorithm 1D = Wire discretization + algorithm 2D = Quadrangle).

The command file for only one horizontal crack length $2a=0,3m$ is given: `forma06a.comm`, in which one applies a tractive effort to the sides “M_haut” and “M_bas”, the rigid modes are blocked by embedding the node in bottom on the left “N_A” and blocking the following displacement of Y the node in bottom on the right “N_B”. It however remains to supplement postprocessing. Realize

to card-index it commands for the following case: and

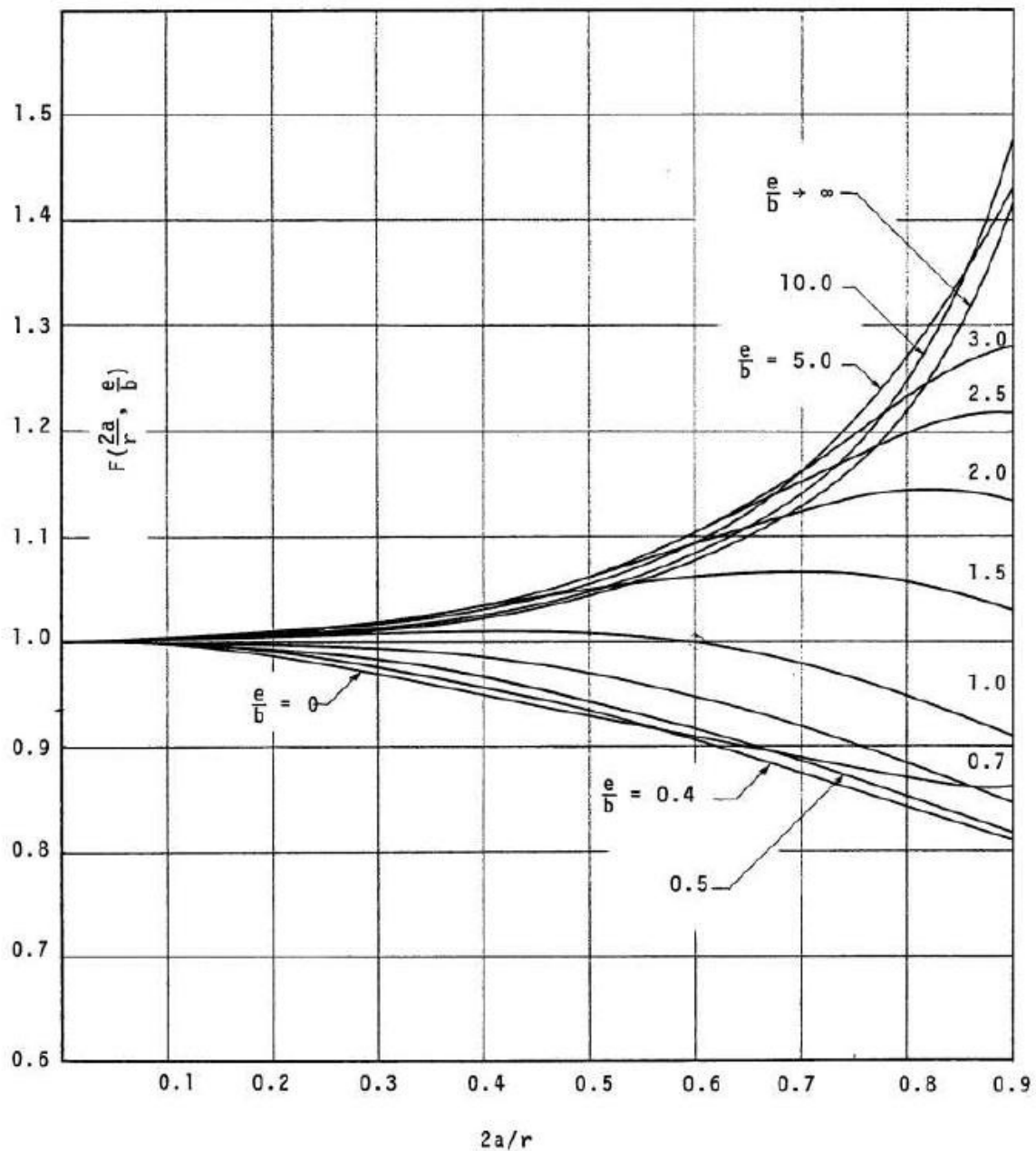
$$a=0,15 \text{ (that is to say } b=0,4 \text{) } 2a/b=0,75 \text{ One is reminded that } e=0$$

each call `DEFI_FISS_XFEM` produces a crack. For 2 cracks, this command should be called twice. Also, commands `POST_K1_K2_K3` and `CALC_G` produce data structure of the type Counts. Yew is necessary to add command `IMPR_TABLE` to display the computation results. Compare

with the solution of Handbook.

To go further, one will be able: to prolong

- the abacuses for (for example $2a/r > 0.9$), $2a/r = 1$ to study
- the smoothness of the mesh, to make
- a parametric study for (to think $e = [0 ; 2b]$ of using python), to study
- other configurations (inclined cracks, addition of other cracks...). Figure



2.1 2.1-1