FORMA08 - Practical works of training “advanced Use”: Beam DCB 3D elastic

Abstract:

This test $3D$ into quasi-static, enters the frame of the validation of the crack propagation per cohesive model. A beam DCB is charged in tension.

1 Problem of reference

1.1 Geometry

One considers a beam here known as doubles Cantilever three-dimensional whose geometry is presented below. It presents an initial cracking of the quarter its length.

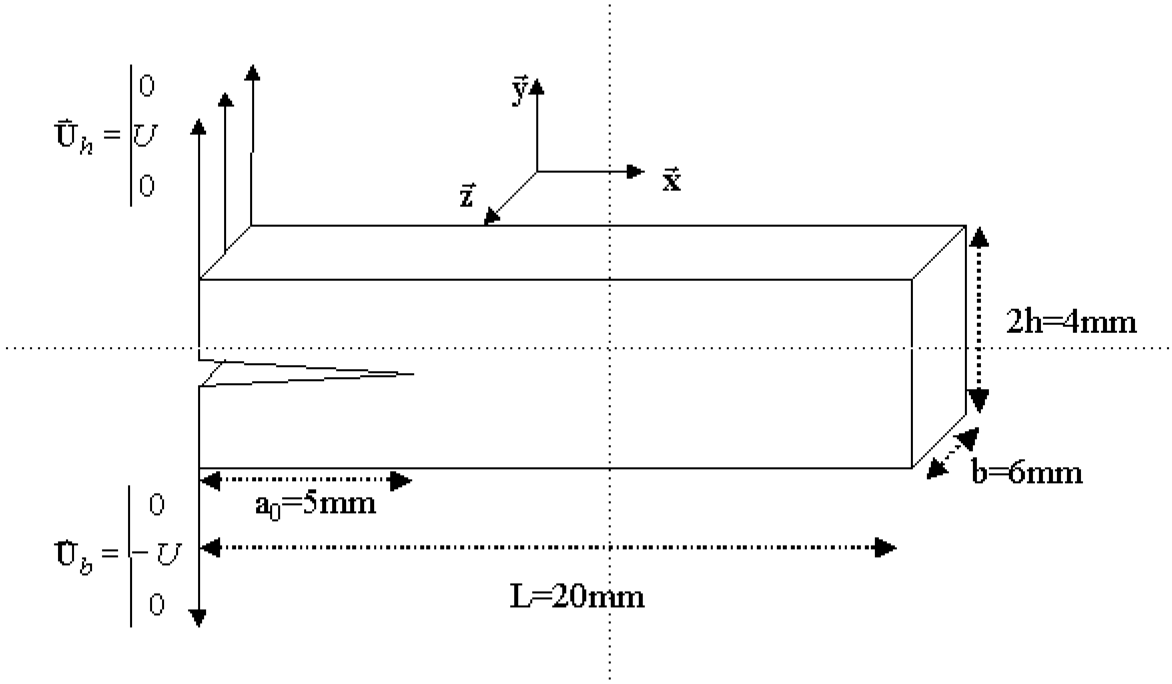

Figure 1.1 : geometry

1.2 Material properties

the material of the beam is supposed to be isotropic linear elastic with the following characteristics:

- Young modulus: $E = 100\text{MPa}$ (yes, it is very very weak!)
- Poisson's ratio: $\nu = 0{,}3$
- Energy of cracking: $G_C = 0{,}9\,MPa.m^{-1}$
- Stress criticizes cracking: $\sigma_c = 4\text{MPa}$

1.3 Boundary conditions and loadings

One imposes a purely vertical displacement of the side sides left of the DCB.
In taking into account symmetry, it is possible to consider only the upper part of the DCB. It is then necessary to add the condition of symmetry like limiting condition of the problem.

2 Reference solution

2.1 Method used for the reference solution

There exists an approximate analytical solution with the cracking of this kind of beam in tension (pure I mode), determined by the method known as of kindness.
This solution is based on the theory of the beams. If one considers the sufficiently slender DCB and the sufficiently small strains, the Bernoulli's equations can be applied. The deflection U is expressed then according to the bending moment M_f by the relation:

$$\frac{\partial^2 U}{\partial^2 x}=\frac{m_f}{EI}=\frac{Fx}{EI}\Rightarrow U(x)=\frac{F}{EI}\left(\frac{x^3}{6}-\frac{a^2}{2}x+\frac{a^3}{3}\right)$$

with a the length of crack, $I=\frac{bh^3}{12}$ quadratic moment of the beam, and F the applied force on the side sides.
Kindness (relationship between the opening and the force necessary to cause it) is thus expressed:

$$C=\frac{U(0)}{F}=\frac{a^3}{3\mathrm{EI}}$$

Rate of energy restitution associated with a crack length a is expressed consequently:

$$G=\frac{F^2}{2\mathrm{b}}\frac{dC}{da}=\frac{9\mathrm{EI}}{2\mathrm{ba}^4}U^2$$

By supposing the stable crack propagation and following the model of Griffith, it is possible to determine the length of crack according to the loading.

$$a=\left(\frac{9\mathrm{EI}}{2\mathrm{bG}_c}\right)^{\frac{1}{4}}U^{\frac{1}{2}}$$

By combining the formulas of the deflection and length of crack, one obtains the statement of the force then:

$$F=\frac{(EI)^{\frac{1}{4}}(2\mathrm{b}G_c)^{\frac{3}{4}}}{(3\mathrm{U})^{\frac{1}{2}}}$$

This relation is approximate because of the assumptions and also owing to the fact that she does not answer by exactly the assumptions of the cohesive zones (but of pure Griffith). One can however regard it as a base of comparison to the experimental results.

2.2 Results of reference

With the numerical values of the statement, one finds analytically the results following:

$$U=10\mathrm{mm}\Rightarrow F=4{,}86\,N$$

$$U=10{,}651\,mm\Rightarrow F=4{,}71\,N$$

3 Modelization a: ELEMENTS 3D_JOINTS

3.1 Unfolding of the TP

the command file corresponding to the requests is provided: file forma08a.comm. However
, it is preferable to use the file to be supplemented. Mesh

3.1.1 In

taking into account the symmetry of the geometry, it is possible to consider only the upper part of beam DCB. For
more speed, a linear mesh including only hexahedrons HEXA 8 is provided (forma08a.med). The useful
groups of this mesh are the following: volume

- of the DCB: mesh groups DCB_1 and DCB _2 volume
- of the cohesive zone: mesh group DCB_J application
- of displacement: mesh group and nodes group DCB_GB (corresponding to the side of $X=0$ the face of loading), containing a nodes group NO7 (useful to recover displacement) application
- of symmetry: mesh group JOINED _B It is

also possible to recreate a free mesh in HEXA8 and PENTA 6 for this study; it will then be necessary to take care of several points: to define

- an additional zone, of low thickness 1 element which will represent the layer of cohesive elements to define
- the mesh groups necessary to the application of displacement to define
- the surface representing lower lips and higher mesh groups cohesive zone. Mechanical computation

3.1.2 To carry out

computation by means of cohesive model CZM_EXP_REG and of elements 3D_JOINT . One will carry out
computation until an imposed displacement of 10 per increment mm of 0.05formule mm

3.1.3 force To recover it

the nodal force, to trace it according to imposed displacement, and to compare with the theoretical results. Visualization

3.1.4 deformed To print

with med format the results of mechanical computation, on the meshes voluminal ones (not to insert meshes of joint) and to visualize the deformed shape. Influence

3.1.5 constitutive law To carry it out

and the same computation same postprocessings by means of cohesive model CZM_LIN_REG . To compare
the Quantities results

3.2 tested and results Identification

Reference	**% tolerance**	**Forces**
nodal for an imposed displacement of 10formule mm	N	Table

3.1: 3.1 for the Modelization A Modelization

4 b: ELEMENTS 3D_INTERFACE _S Déroulement

4.1 of the TP the command file

corresponding to the requests is provided: file formed 08b.comm. However
, it is preferable to use the file to be supplemented. Mesh

4.1.1 In taking into account

the symmetry of the geometry, it is possible to consider only the upper part of beam DCB. For more speed, a linear mesh including only hexahedrons HEXA8 is provided (forma08b.med). The useful groups of this mesh are the following: volume of

- the DCB: mesh groups DCB_1 and DCB _2 volume of
- the cohesive zone: mesh group DCB_J application
- of displacement: mesh group DCB_GB, containing a nodes group NO7 (useful to recover displacement) application
- of symmetry: mesh group JOINT_B Mechanical computation

4.1.2 To carry out

computation by means of cohesive model CZM_OUV_MIX and of elements 3D_INTERFACE . To think

of the convergence criterion to be used… One will carry out

the first computation until a displacement imposed from increment $10\,mm$ of formula. Computation $0{,}05\,mm$

4.1.3 To recover it

the nodal force, to trace it according to imposed displacement, and to compare with the theoretical results. Installation

4.1.4 of control One now

proposes to use the tools for control. To use the control of type elastic prediction with a selection of type residue on the displacement imposed and meshes of interface, and to ask time limits of 4,5 per increment of 0,05. What does one note? Does the curve force displacement is conforms to the analytical curve? Visualization

4.1.5 deformed To print with

med format the results of the mechanical computation controlled, on the meshes voluminal ones (not to insert meshes of joint) and to visualize the deformed shape. Quantities tested

4.2 and results Identification

Reference	% tolerance	Forces nodal
for an imposed displacement of 10,561 formula 4,71 mm	5,0% N	4.1

: Results 4.1 of the Modelization B