

ABAQUS XFEM Tutorial: 2D Crack Initiation

Written By: Matthew Jon Pais, University of Florida (2010)

Website: www.matthewpais.com

Email: mpais@ufl.edu, matthewjpais@gmail.com

Creating the Uncracked Domain

1. Open ABAQUS/CAE 6.9 or later.
2. Double click on Parts. Enter name as Plate, Modeling Space is 2D Planar, Type is Deformable, Base Feature is Shell and Approximate Size is 5. Click Continue.
3. Use the line tool to draw lines from (0,0) to (0.4,0), (0.6,0) to (1,0), (1,0) to (1,0.5), (1,0.5) to (0,0.5), and (0,0.5) to (0,0). Use the arc tool to draw an arc from (0.4,0) to (0.6,0) with radius 0.1. Click Done.
4. Double click on Materials. Enter name as Generic. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 10 MPa and Poisson's ratio as 0.3. Click on Mechanical, then Damage for Traction Separation laws, then Maxps Damage. Enter a value of 2. From the Suboptions menu click on Damage Evolution. Enter Displacement at Failure as 1E-9. Click Ok. Click Ok.
5. Double click on Sections. Name as Main. Accept default settings by clicking Continue. Select Generic as material. Click Ok.
6. Expand Parts then expand Plate. Double click on Section Assignments. Select the domain. Click Done. Accept default settings. Click Ok.
7. Expand Plate. Double click on Mesh. From the top menu select Seed, then Edge By Size. Select all the straight edges. Click Done. Enter size of 0.05. Hit Enter. From the top menu select Seed, then Edge By Number. Select the hole, enter 15. Hit Enter. Click Done.
8. From the top menu select Mesh, then Controls. Select Quad, Free, Advancing Front. Click Ok. From the top menu select Mesh, then Part. Click Yes.
9. Expand Assembly. Double click on Instances. Select Plate. Accept default settings by clicking Ok.

Creating the Possibly Cracked Domain

1. Double click on Interactions. Click Cancel. From top menu click Special, then Crack, then Create. Name as Crack, Type is XFEM. Click Continue. Select the uncracked domain as the Crack Domain. Click Ok.
2. Double click on Interactions. Enter name as Growth. Select Initial Step and Types for Selected Step as XFEM Crack Growth. Click Continue. XFEM Crack should have Crack. Click Ok.

Create the Boundary Conditions and Loads

1. Double click on Steps. Enter Name as Loading. Accept default setting and click Continue. On the Incrementation tab Type is Automatic, Maximum number of increments is 100000. Initial is 0.001, Minimum is 1E-20, Maximum is 0.01. Click Ok.
2. Double click on Steps. Click Cancel. From the top menu select Other, then General Solution Controls, then Manager. For Loading select Edit, then accept the warning by clicking Continue. On the Time Incrementation tab, select the first More Option, then change I_A to 100. Click Ok.
3. Double click on Loads. Enter name as RightEdge, Category is Mechanical, Type is Pressure. Click Continue. Select the right edge of the domain. Click Done. Enter -1 as Magnitude, other settings are default. Click Ok.
4. Repeat step 2 for the left edge of the domain, entering the name as LeftEdge.
5. Double click on BCs. Enter name as FixedTRC, Step is Initial, Category is Mechanical, Types for Selected Step is Displacement/Rotation. Click on the top right corner of the domain. Click Done. Set U1, U2 and UR3 to zero. Click Ok.
6. Repeat step 4 for the top left corner of the domain. Enter name as RollerTLC. Set U2 and UR3 to zero.
7. Expand Field Output Requests, double click on F-Output-1. Expand the Failure/Fracture options and check the box next to PHILSM, Level set value phi. Click Ok. This will allow you to view the level set function defining the crack.

Solving the System of Equations

1. Double click on Jobs. Enter name as crackInitiation. Click Continue. Accept default settings by clicking Ok.
2. Expand Jobs. Right click on crackInitiation and click Submit.
3. Right click on crackInitiation, click Results to view results.