ABAQUS XFEM Tutorial: 2D Edge Crack

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 rectangle tool to draw a square from (-2,-2) to (2,2). Click Done.
4. Double click on Materials. Enter name as Aluminum. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 70 GPa and Poisson's ratio as 0.33. Click on Mechanical, then Damage for Traction Separation laws, then Maxps Damage. Enter a value of 500 MPa. From the Suboptions menu click on Damage Evolution. Enter Displacement at Failure as 1. Click Ok. Click Ok.
5. Double click on Sections. Name as Main. Accept default settings by clicking Continue. Select Aluminum as material and click box by Plane stress/strain thickness. Enter 1 as thickness. 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 Number. Select the Domain. Click Done. Enter 41 as Number of elements along the edges. Hit Enter. Click Done.
8. From the top menu select Mesh, then Controls. Select Quad, Structured. 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 Cracked Domain

1. Double click on Parts. Enter name as Crack, Modeling Space is 2D Planar, Type is Deformable, Base Feature is Wire and Approximate Size is 5. Click Continue.
2. Draw a line from (-2,0) to (-1,0). Click Done.
3. Expand Assembly, then double click on Instances. Select Crack. Accept default settings by clicking Ok.
4. Double click on Interactions. Click Cancel. From top menu click Special, then Crack, then Create. Name as EdgeCrack, Type is XFEM. Click Continue. Select the uncracked domain as the Crack Domain. On the menu which appeaars, Specify the Crack Location by clicking on the line signifying the crack. Click Ok.
5. 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 EdgeCrack. Click Ok.

ABAQUS XFEM Tutorial: 2D Edge Crack with Inclusion

Creating the Plate 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 10. Click Continue.
3. Use the rectangle tool to draw a square from (-2,-4) to (2,4). Use the circle tool to draw a circle with radius 1 and center at (0,-2). Click Done.
4. Double click on Materials. Enter name as Mat1. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 1 MPa and Poisson's ratio as 0.33. Click on Mechanical, then Damage for Traction Separation Laws, then Maxps Damage. Enter a value of 1 kPa. Click on the Suboptions box, then Damage Evolution. Type is Energy, Softening is Linear, Degredation is Maximum, Mixed-Mode Behavior is Mode-Independent, Mode Mix Ratio is Energy. Enter a value of 50 for the Fracture Energy. Click Ok. Click Ok.

5. Double click on Sections. Name as Plate. Accept default settings by clicking Continue. Select Mat1 as material and click box by Plane stress/strain thickness. Enter 1 as thickness. Click Ok.

Creating the Inclusion Domain

1. Double click on Parts. Enter name as Inclusion, Modeling Space is 2D Planar, Type is Deformable, Base Feature is Shell and Approximate Size is 10. Click Contine.
2. Use the circle tool to draw a circle with radius 1 and center at (0,-2). Click done.
3. Double click on Materials. Enter name as Mat2. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 10 MPa and Poisson's ratio as 0.33. Click on Mechanical, then Damage for Traction Separation Laws, then Maxps Damage. Enter a value of 10 kPa. Click on the Suboptions box, then Damage Evolution. Type is Energy, Softening is Linear, Degredation is Maximum, Mixed-Mode Behavior is Mode-Independent, Mode Mix Ratio is Energy. Enter a value of 1000 for the Fracture Energy. Click Ok. Click Ok.
4. Double click on Sections. Name as Inclusion. Accept default settings by clicking Continue. Select Mat2 as material and click box by Plane stress/strain thickness. Enter 1 as thickness. Click Ok.

Creating the Total Uncracked Domain

1. Expand Assembly, then double click on Instances. Select both the plate and inclusion. Click Ok.
2. Merge the two parts together using the merge button on the left of the viewport. Name the part Total, Merge is Geometry, Options is Suppress and Intersecting Boundaries is Retain. Click Continue. Select the two parts and click Done.
3. Expand Parts then Total. Double click on Section Assignments. Select the plate section from the viewport. Click Done. In Edit Section Assignment window, pick Plate. Click Ok.
4. Double click on Section Assignments. Select the inclusion section from the viewport. Click Done. In Edit Section Assignment window, pick Inclusion. Click Ok.
5. Double click on Mesh. From the top menu select Seed, then Edge By Number. Select Total. Click Done. Enter 41 as Number of elements along the edges. Hit Enter. Click Done.
6. From the top menu select Mesh, then Controls. Select Total. Select Quad as Element Shape. From the top menu select Mesh, then Part. Click Yes.

Creating the Cracked Domain

1. Double click on Parts. Enter name as Crack, Modeling Space is 2D Planar, Type is Deformable, Base Feature is Wire and Approximate Size is 5. Click Continue.
2. Draw a line from (-2,0) to (-1.5,0). Click Done.
3. Expand Assembly, then double click on Instances. Select Crack. Accept default settings by clicking Ok.
4. Double click on Interactions. Click Cancel. From top menu click Special, then Crack, then Create. Name as EdgeCrack, Type is XFEM. Click Continue. Select the uncracked domain as the Crack Domain. On the menu which appeaars, Specify the Crack Location by clicking on the line signifying the crack. Click Ok.
5. 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 EdgeCrack. Click Ok.

Create the Boundary Conditions and Loads

1. Double click on Steps. Enter Name as Loading. On Incrementation Tab enter Type as Automatic, Maximum Number of Increments at 100000 and Increment Size as 0.01, 1e-005, 0.01. Click Ok.
2. Double click on Loads. Enter name as TopPressure, Category is Mechanical, Type is Pressure. Click Continue. Select the top edge of the domain. Click Done. Enter -700 as Magnitude, other settings are default. Click Ok.
3. Repeat step 2 for the bottom edge of the domain, entering the name as BottomPressure.
4. Double click on BCs. Enter name as FixedBRC, Step is Initial, Category is Mechanical, Types for Selected Step is Displacement/Rotation. Select Mesh. Click on the bottom right corner of the domain. Click Done. Set U1, U2 and UR3 to zero. Click Ok.

5. Repeat step 4 for the top right corner of the domain. Enter name as RollerTRC. Set U1 and UR3 to zero.
6. 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 IncCrack. Click Continue. Accept defualt settings by clicking Ok.
2. Expand Jobs. Right click on IncCrack and click Submit.
3. Right click on IncCrack, click Results to view results.

ABAQUS XFEM Tutorial: 3D Edge Crack

Creating the Uncracked Domain
1. Open ABAQUS/CAE 6.9 or later.
2. Double click on Parts. Enter name as Solid, Modeling Space is 3D, Type is Deformable, Base Feature is Solid and Approximate Size is 5. Click Continue.
3. Use the rectangle tool to draw a square from (-2,-2) to (2,2). Click Done. Enter 4 for the depth. Click Ok.
4. Double click on Materials. Enter name as Aluminum. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 70 GPa and Poisson's ratio as 0.33. Click on Mechanical, then Damage for Traction Separation laws, then Maxps Damage. Enter a value of 500 MPa. From the Suboptions menu click on Damage Evolution. Enter Displacement at Failure as 1. Click Ok. Click Ok.
5. Double click on Sections. Name as Main. Accept default settings by clicking Continue. Select Aluminum as material. Click Ok.
6. Expand Parts then expand Solid. Double click on Section Assignments. Select the domain. Click Done. Accept default settings. Click Ok.
7. Expand Solid. Double click on Mesh. From the top menu select Seed, then Edge By Number. Select the Domain. Click Done. Enter 21 as Number of elements along the edges. Hit Enter. Click Done.
8. From the top menu select Mesh, then Controls. Select Hex, Structured. Click Ok. From the top menu select Mesh, then Part. Click Yes.
9. Expand Assembly. Double click on Instances. Select Solid. Accept default settings by clicking Ok.

Creating the Cracked Domain
1. Double click on Parts. Enter name as Crack, Modeling Space is 3D, Type is Deformable, Base Feature is Shell, Type is Extrusion and Approximate Size is 5. Click Continue.
2. Draw a line from (-2,0) to (-1,0). Click Done. Enter 4 for depth. Click Ok.
3. Expand Assembly, then double click on Instances. Select Crack. Accept default settings by clicking Ok.
4. Double click on Interactions. Click Cancel. From top menu click Special, then Crack, then Create. Name as EdgeCrack, Type is XFEM. Click Continue. Select the uncracked domain as the Crack Domain. On the menu which appears, Specify the Crack Location by clicking on the line signifying the crack. Click Ok.
5. 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 EdgeCrack. Click Ok.

Create the Boundary Conditions and Loads

1. Double click on Steps. Enter Name as Loading. Accept default setting and click Continue. Accept default settings and click Ok.
2. Double click on Loads. Enter name as TopPressure, Category is Mechanical, Type is Pressure. Click Continue. Select the top edge of the domain. Click Done. Enter -1 as Magnitude, other settings are default. Click Ok.
3. Repeat step 2 for the bottom edge of the domain, entering the name as BottomPressure.
4. Double click on BCs. Enter name as FixedBREdge, Step is Initial, Category is Mechanical, Types for Selected Step is Displacement/Rotation. Click on the bottom right edge through the thickness of the domain. Click Done. Set U1, U2 and UR3 to zero. Click Ok.
5. Repeat step 4 for the top right corner of the domain. Enter name as RollerTRC. Set U1 and UR3 to zero.
6. 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 EdgeCrack3D. Click Continue. Accept default settings by clicking Ok.
2. Expand Jobs. Right click on EdgeCrack3D and click Submit.
3. Right click on EdgeCrack3D, click Results to view results.

ABAQUS XFEM Tutorial: 3D Penny Crack

Creating the Uncracked Domain

1. Open ABAQUS/CAE 6.9 or later.
2. Double click on Parts. Enter name as Solid, Modeling Space is 3D, Type is Deformable, Base Feature is Solid and Approximate Size is 5. Click Continue.
3. Use the rectangle tool to draw a square from (-2,-2) to (2,2). Click Done. Enter 4 for the depth. Click Ok.
4. Double click on Materials. Enter name as Aluminum. Click on Mechanical, then Elasticity, then Elastic. Enter Young's modulus as 70 GPa and Poisson's ratio as 0.33. Click on Mechanical, then Damage for Traction Separation laws, then Maxps Damage. Enter a value of 500 MPa. From the Suboptions menu click on Damage Evolution. Enter Displacement at Failure as 1. Click Ok. Click Ok.
5. Double click on Sections. Name as Main. Accept default settings by clicking Continue. Select Aluminum as material. Click Ok.
6. Expand Parts then expand Solid. Double click on Section Assignments. Select the domain. Click Done. Accept default settings. Click Ok.
7. Expand Solid. Double click on Mesh. From the top menu select Seed, then Edge By Number. Select the Domain. Click Done. Enter 20 as Number of elements along the edges. Hit Enter. Click Done.
8. From the top menu select Mesh, then Controls. Select Hex, Structured. 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 Cracked Domain

1. Double click on Parts. Enter name as Crack, Modeling Space is 3D, Type is Deformable, Base Feature is Shell, Type is Planar and Approximate Size is 5. Click Continue.
2. Draw a circle with center at (0,0) and radius of 0.5. Click Done.
3. Expand Assembly, then double click on Instances. Select Crack. Accept default

settings by clicking Ok.
4. Under the menu to the left of the viewport, click Translate Instance. Select the Crack. Click Done. Initial vector is (0,0,0), second vector is (0,0,2). Click Ok.
5. Expand Assembly, then Instances. Right click on Crack, then suppress.
6. Double click Set. Name as Domain. Type is Element. Click Continue. Select all elements. Click Done.
7. Expand Assembly, then Instances. Rick click on Crack, then Resume. Right click on Solid, then Suppress.
8. Double click on Interactions. Click Cancel. From top menu click Special, then Crack, then Create. Name as Penny, Type is XFEM. Click Continue. Select the previously created set as the Crack Domain. On the menu which appears, Specify the Crack Location by clicking on the planar segment corresponding to the crack. Click Ok.
9. Expand Assembly, then Instances. Right click on Solid, then Resume.

Create the Boundary Conditions and Loads

1. Double click on Steps. Enter Name as Loading. Accept default setting and click Continue. Accept default settings and click Ok.
2. Double click on Loads. Enter name as TopPressure, Category is Mechanical, Type is Pressure. Click Continue. Select the top edge of the domain. Click Done. Enter -1 as Magnitude, other settings are default. Click Ok.
3. Repeat step 2 for the bottom edge of the domain, entering the name as BottomPressure.
4. Create three sets of boundary conditions. For an origin at the bottom center of the solid the following points are fixed such that they are not allowed to move in the x-direction: (2,0,4), (-2,0,4), (2,0,0) (-2,0,0), y-direction: (0,2,4), (0,-2,4), (0,2,0), (0,-2,0) and zdirection: (2,2,2), (2,-2,2), (-2,2,2), (-2,-2,0).
6. 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 Penny. Click Continue. Accept default settings by clicking Ok.
2. Expand Jobs. Right click on Penny and click Submit.
3. Right click on Penny, click Results to view results.