FRACTURE MECHANICS IN ANSYS R16

Session – 02
How to Perform Fracture Mechanics Analysis in Workbench
PERFORMING A FRACTURE ANALYSIS

• Fracture analysis requires that you define a crack.

• Since fracture parameter calculation requires knowledge of the mesh characteristics around the crack, the mesh must be generated before solving for fracture parameters.

• Fracture parameter computation is only applicable to static structural analyses.
1. The steps shown below describe setting up the fracture analysis when the location of crack is known. The crack location and its alignment are dictated by the coordinate system selected by the crack object.

2. In ANSYS Workbench, insert a Static Structural analysis in the project schematic.

3. Input geometry.

4. Locate a coordinate system with a graphic pick point, coordinates, or topology. The coordinate system must be located on the surface.

5. Align the axes of the coordinate system of the crack. The specified coordinate system's y-axis must be pointing in the direction normal to the crack surface. For cracks lying on curved surfaces, ensure that the coordinate system's x-axis is pointing normal to the surface of the body at the coordinate system location.
6. Insert a Fracture folder in the Tree Outline.

7. Insert a Crack object under the Fracture folder.

8. Specify the crack object details.

9. Generate the mesh by right-clicking the Fracture folder and selecting Generate All Crack Meshes.

10. Apply loads and boundary conditions.

11. Apply any pressure on crack face if necessary.

12. Ensure the Fracture setting under Solver Controls in the Analysis Settings is turned on.

13. Solve.
FRACTURE ANALYSIS WORKFLOWS


15. Post process the Fracture Result.

16. Export to Excel or copy/paste from the chart if necessary.
LIMITATIONS OF FRACTURE ANALYSIS

1. A Fracture analysis does not support adaptive mesh refinement.

2. Only 3D analyses support cracks (Crack object).

3. You can scope a crack to one body only and the crack cannot span more than one face. The base mesh on that body must be quadratic tetrahedron mesh.

4. The stiffness behavior of the scoped geometry selection of the Crack object must be flexible.

5. The scoped crack front nodal selection of the Pre-Meshed Crack object must exist in geometries with a flexible stiffness behavior definition.

6. Fracture parameter computations based on the VCCT technique are only supported for lower order crack mesh. Hence, VCCT based fracture parameter computations are only supported for Pre-Meshed Crack object.
LIMITATIONS OF FRACTURE ANALYSIS

7. Solution Restarts are not supported with the computation of fracture parameters. Solution Restarts can be used for solving an analysis of cracks without computing the fracture parameters by turning “Off” the “Fracture” setting under Solver Controls.

8. The Crack object only supports semi-elliptical surface cracks.

9. The crack top and bottom face nodes are not connected through any constraint equation. So the nodes of the top face can penetrate the bottom face or vice versa based on the applied loads and constraints. In these scenarios, you may need to create a constraint equation between crack faces during solution using the Commands object.

10. The graphical view of the crack may differ from the generated mesh.
LIMITATIONS OF FRACTURE ANALYSIS

11. Crack object is not supported for Cyclic Symmetry Region and Structural Linear Periodic Symmetry Region objects.

12. Interpolated displacements for the facets in a surface construction object may fail to demonstrate the proper deformation of a crack.
The Meshing application supports fracture meshing capabilities enabling you to insert multiple surface cracks into a mesh.

These fracture meshes can then be used to analyze crack fronts in static structural analyses. Fracture meshing uses a *Fracture* object that can contain multiple *Crack* objects.
OVERVIEW OF FRACTURE MESHING

1. Body to which the crack is scoped.
2. Base mesh, which always consists of quadratic tetrahedron elements. For the generation of fracture mesh, the base mesh inside and surrounding the region of the buffer zone (3) must be a quadratic tetrahedron mesh.
3. Buffer zone, which is always filled with quadratic tetrahedron elements.
4. Interface between the buffer zone (3) and the fracture affected zone (5), illustrated by the green line. Contact pair 1 is defined at the interface between the elements in the buffer zone and the elements in the fracture affected zone.
5. Fracture affected zone, which is filled with quadratic elements (hex and wedge).
6. Crack front, located along the crack shape and illustrated by the red line. The crack shape is semi-elliptical.
7. Discontinuity planes, illustrated by the light blue shading. Consist of two planes at the same location (the crack front plane).
CHARACTERISTICS OF FRACTURE MESHING

• Fracture meshing is a post mesh process in that it occurs in a separate step after the base mesh is generated. If you do not generate a base mesh before you invoke Generate All Crack Meshes, the base mesh is generated first and the crack meshing occurs in a separate step after base meshing is complete.

• For the base mesh inside and surrounding the region of the buffer zone, fracture meshing supports quadratic tetrahedron elements only. Linear elements may exist farther away from the buffer zone on the same body to which the crack is scoped.

• Fracture meshing is a part-based meshing operation. It is supported for all part/body-based tetrahedron mesh methods. It is not supported for assembly meshing algorithms.

• Fracture meshing does not support insertion of a crack that spans multiple bodies and multiple surfaces.
CHARACTERISTICS OF FRACTURE MESHING

• The fracture mesh overrides the settings of the base mesh. You can insert a crack on topology to which a sizing control, match control, or mapped Face Meshing control is already applied. However, fracture meshing does not respect sizing controls. It may also eliminate or disable a match or mapped Face Meshing control. For example and as illustrated below, fracture meshing has overwritten a mapped Face Meshing control.

• Fracture meshing supports crack insertion on planar surfaces and curved surfaces. For curved surfaces, you can insert cracks on convex or concave surfaces. If you try to insert a single crack that extends across a surface that has both types of curves, crack insertion may fail.

• Fracture meshing does not support crack insertion at a corner.

• Fracture meshing does not support mesh refinement.

• Fracture meshing is supported for static structural analyses only.
CHARACTERISTICS OF FRACTURE MESHING

• Crack insertion does not alter the geometry/topology of a model.

• For the Shape Checking control, fracture meshing supports the Standard Mechanical option only.

• Once inserted, the Fracture object cannot be suppressed or deleted, even if it is empty.

• Crack objects can be suppressed, deleted, or duplicated. When a crack definition changes after meshing, only the Fracture object in the Tree Outline is invalidated.

• The crack plane always lies in the X-Z plane of the specified coordinate system.

• The major radius of the crack grows in the Z direction, and the minor radius grows in the positive X direction of the specified coordinate system.
CHARACTERISTICS OF FRACTURE MESHING

• The specified coordinate system should lie on the surface of the body to which the crack is scoped. If it does not, and the Project to Nearest Surface property is set to Yes, the software projects the coordinate system onto the body’s surface and modifies the center of the ellipse, as shown below.
In this case, you will need to define the Hit Point Normal and orient the primary axis. The center of the ellipse is moved in the X direction so the major radius of the crack grows in the Z direction, and the minor radius grows in the positive X direction of the specified coordinate system. The offset projects back to the surface of the body, making the offset close to 0. The illustration below shows the resultant mesh. For more information on creating a coordinate system aligned with a hit point, see the Creating a Coordinate System Based on a Surface Normal section of the help.
CHARACTERISTICS OF FRACTURE MESHING

• As illustrated by the figure below, the mesh generated for the mesh contours (black lines) will not necessarily match the preview of the mesh contours (white lines).
CHARACTERISTICS OF FRACTURE MESHING

• When cracks are inserted in curved surfaces, the projection of the crack template on the surface may be slightly distorted on the template, as shown in the following figure.
CRACK OVERVIEW

• A crack is characterized by its shape, crack front/tip, crack discontinuity plane, crack normal, and crack direction. A crack front in three dimensional analyses represents the line of separation of the discontinuous crack surface. The same is represented by a crack tip in two dimensional analyses. A crack inside ANSYS Mechanical is defined using a Crack object or Pre-Meshed Crack object. These objects can be inserted under the Fracture folder.
CRACK OVERVIEW

• Crack objects, for which you define geometry parameters that define the generated crack mesh, is used to analyze crack front. Internally, the crack mesh generation is performed after the creation of the base mesh. The geometric parameters define the semi-elliptical shape of the crack in three dimensional analyses. The crack definition is complete only after the successful generation of the crack mesh. By default, the crack mesh generation automatically creates a node-based named selection for the crack front under the crack object.
CRACK OVERVIEW

- A Pre-Meshed Crack definition assumes that the crack meshes, representing the discontinuity or flaw in the structure, have already been generated. In other words, the pre-meshed crack does not internally generate the crack mesh using Fracture Meshing, as the Crack object does, but instead assumes that the crack mesh has been generated beforehand. A Pre-Meshed Crack object uses a node-based named selection to analyze crack front; this nodal named selection is required for the computation of fracture parameters. If a geometric edge represents a crack front, you must first convert it to a node-based named selection using the Worksheet criteria before it can be used by the Pre-Meshed Crack object.

- The orientation of the crack plays a vital role in the fracture parameter calculations. The orientation of a Crack object is reflected by its Crack Coordinate System, while the orientation of a Pre-Meshed Crack object is reflected by its Coordinate System property setting. The orientation must be defined such that the y-axis is normal to the crack surface while the x-axis is pointing along the crack extension direction.
DEFINING A CRACK

1. Select the Model object in the Tree Outline.

2. Insert a Fracture object into the Tree by right-clicking on the Model object and selecting Insert > Fracture from the context menu. Alternatively, click the Fracture button on the toolbar.
   
   a) Only one Fracture object is valid per Model.

3. Insert a Crack object into the Tree by right-clicking on the Fracture object and selecting Insert > Crack from the context menu. Alternatively, click the Crack button on the toolbar.

4. A crack definition must always be scoped to a single solid body. Use the Body selection filter to pick a body in the Geometry window, click the Geometry field in the Details View, and then click Apply.
5. To further define the crack, use the following controls. These controls appear in the Details View of the Crack object. As you specify values for the controls, the image in the Geometry window previews the entered data.

6. The following figure shows an example of a crack definition and its corresponding image. The semi-elliptical curve defines the shape of the crack front, as shown by the red line.
DEFINING A CRACK

This figure provides a more detailed illustration of the fracture affected zone defined above. Notice that the values shown in the image below correspond to the Details View settings above.
DEFINING A CRACK

• **Coordinate System**: This property specifies the coordinate system that defines (along with the Align with Face Normal and Project to Nearest Surface properties) the position and orientation of the crack. The Y axis of the specified coordinate system defines the crack plane normal. The coordinate system that you create must be a Cartesian coordinate system (Type property) and its origin cannot lie outside the bounding box of the body scoped to the crack.

• **Align with Face Normal**: This property defines the orientation of the Crack Coordinate System object by aligning the primary axis of the coordinate system specified in the Coordinate System property to the normal of the nearest surface. The default setting is Yes. Setting this property to No excludes the capability of this property.
DEFINING A CRACK

• **Project to Nearest Surface**: This property defines the origin of the Crack Coordinate System by projecting the origin of the coordinate system specified in the Coordinate System property to the nearest surface. The default setting is Yes. Setting this property to No excludes the capability of this property.

  • Note: The meshing process automatically creates the Crack Coordinate System object as a child of the Crack object from the inputs of the Coordinate System, the Align with Face Normal, and the Project to Nearest Face properties.
DEFINING A CRACK

- **Crack Shape**: Read-only and always set to Semi-Elliptical.

- **Major Radius**: Specifies the major radius, which defines the size of the crack shape along the Z axis (that is, the width of the crack). Enter a value greater than 0.

- **Minor Radius**: Specifies the minor radius, which defines the size of the crack shape along the X axis (that is, the depth of the crack). Enter a value greater than 0.
DEFINING A CRACK

• **Fracture Affected Zone**: The fracture affected zone is the region that contains a crack. The Fracture Affected Zone control determines how the fracture affected zone height is defined:
  
  • **Program Controlled**: The software calculates the height, and Fracture Affected Zone Height is read-only. This is the default.
  
  • **Manual**: You enter the height in the Fracture Affected Zone Height field.
DEFINING A CRACK

- **Fracture Affected Zone Height:** This value specifies two things: 1) the height of the Fracture Affected Zone, which is in the Y direction of the crack coordinate system; and 2) the distance in totality by which the Fracture Affected Zone is extended in the positive and negative Z direction of the crack coordinate system from the crack front extremities.

- The shape of a Fracture Affected Zone is rectangular, regardless of the shape of the crack. Although buffer zones may overlap, care should be taken when defining multiple cracks that the zones do not overlap, as shown here, or the crack generation will fail.
DEFINING A CRACK

- **Largest Contour Radius:** Specifies the largest contour radius for the crack shape. Enter a value greater than 0.

- **Circumferential Divisions:** Specifies the number of circumferential divisions for the crack shape. The value you enter must be a multiple of 8, and must be 8 or greater. The default is 8.
  - The Geometry window can display only a maximum of 360 circumferential divisions, but you can specify a higher value and fracture meshing will respect it.

- **Mesh Contours:** Specifies the number of mesh contours for the crack shape. The value you enter must be 1 or greater. The default is 6.
  - The Geometry window can display only a maximum of 100 mesh contours, but you can specify a higher value and fracture meshing will respect it.
DEFINING A CRACK

• **Crack Front Divisions:** Specifies the number of divisions for the crack front. The value you enter must be 3 or greater. The default is 15.
  
  • The Geometry window can display only a maximum of 999 crack front divisions, but you can specify a higher value and fracture meshing will respect it.

• **Solution Contours:** Specifies the number of mesh contours for which you want to compute the fracture result parameters. The value you enter must be less than or equal to the value of Mesh Contours, and cannot be greater than 99. By default, the value is Match Mesh Contours, indicating the number of Solution Contours is equal to the number of Mesh Contours. Entering 0 resets the value to Match Mesh Contours.

• **Suppressed:** Toggles suppression of the Crack object. The default is No.
  
  • The Crack object is suppressed automatically if the scoped body is suppressed.
DEFINING A CRACK

• **Buffer Zone Scale Factors:** Control the size of the buffer zone in the X, Y, and Z directions, relative to the size of the fracture affected zone. For each scaling parameter, use the slider to set a value from 2 to 50. The default is 2. The maximum dimension among the three directions of the fracture affected zone is multiplied by the corresponding scale factors to create a buffer zone:
  
  • X Scale Factor
  
  • Y Scale Factor
  
  • Z Scale Factor

• As illustrated here, buffer zone scale factor annotations are not drawn in the negative X direction. That is, half of the cuboid is drawn to depict the buffer zone scale factors, but the half of the cuboid in the negative X direction is not drawn.
• **Crack Front Nodes**: Identifies the Named Selection that is created automatically for the crack front (NS_crackname_Front). Contains nodes used for postprocessing of results.

• As shown, the crack front is always perpendicular to the X axis.
DEFINING A CRACK

- **Crack Faces Nodes:** Determines whether Named Selections are created automatically for the crack’s top face and bottom face. These faces are both located in the XZ plane and are discontinuous.

- The default is Off. If On, the following additional fields appear:

- **Top Face Nodes:** Identifies the Named Selection that is created automatically for the top face (NS_crackname_TopFace). This face is discontinuity plane 1. Contains nodes used for applying a pressure to the top face.

- **Bottom Face Nodes:** Identifies the Named Selection that is created automatically for the bottom face (NS_crackname_BottomFace). This face is discontinuity plane 2. Contains nodes used for applying a pressure to the bottom face.
DEFINING A CRACK

- **Contact Pairs Nodes**: Determines whether Named Selections are created automatically for the contact and target faces of the contact pair. The default is Off. If On, the additional fields listed below appear. Fracture meshing creates contact pair 1 between the fracture affected zone and the buffer zone.

- **Contact 1 Nodes**: Identifies the Named Selection that is created automatically for contact face 1 (NS_crackname(Contact1). Contains nodes located on the contact face.

- **Target 1 Nodes**: Identifies the Named Selection that is created automatically for target face 1 (NS_crackname_Target1). Contains nodes located on the target face.
DEFINING A CRACK

• Select the Fracture object or Crack object in the Tree Outline, right-click, and select Generate All Crack Meshes.

• When the fracture mesh is generated, the requested Named Selections are inserted into the Tree Outline under the Crack object with which they are associated.

• Click Show Mesh on the Graphics Options toolbar to display the fracture mesh.
A crack simulation example will be demonstrated in Ansys Workbench

Table 2
Experimental threshold data used for model validation

<table>
<thead>
<tr>
<th>Material name</th>
<th>References</th>
<th>Loading case</th>
<th>$K_{I,th}$ (MPa m$^{1/2}$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6061Al</td>
<td>[34]</td>
<td>I + II</td>
<td>3.9</td>
</tr>
<tr>
<td>7075-T6 Aluminum alloy</td>
<td>[35]</td>
<td>I + II</td>
<td>1.6</td>
</tr>
<tr>
<td>316 Stainless steel</td>
<td>[22]</td>
<td>I + II</td>
<td>5.81</td>
</tr>
<tr>
<td>Aluminum alloy</td>
<td>[36]</td>
<td>I + II</td>
<td>2.75</td>
</tr>
<tr>
<td>2017-T3 Aluminum alloy</td>
<td>[35]</td>
<td>I + II</td>
<td>1.6</td>
</tr>
<tr>
<td>Mild steel</td>
<td>[37]</td>
<td>I + III</td>
<td>6.6</td>
</tr>
<tr>
<td>2024Al</td>
<td>[38]</td>
<td>I + II</td>
<td>3.9</td>
</tr>
<tr>
<td>SiCp/2024Al composite</td>
<td>[38]</td>
<td>I + II</td>
<td>4.8</td>
</tr>
</tbody>
</table>
• What’s is the difference between $K_{1C}$ and $K_{th}$?
  
  • $K_{1C}$ is for immediate fracture
  • $K_{th}$ is for fatigue crack growth initiation

• For Aluminum 7075
  • $K_{1C} = 26$ ksi in$^{1/2}$
  • $K_{th} = 2$ ksi in$^{1/2}$
STAY AHEAD DURING CHALLENGING TIMES

• To purchase ANSYS or other software, please contact us: info@ozeninc.com  
  (408) 732-4665

• If you would like us to have a demo for your specific case or for any other question, please contact us at: info@ozeninc.com
STAY AHEAD DURING CHALLENGING TIMES

Please contact us at info@ozeninc.com to avail the following Services:

• Software Sales
• Technical Support
• Training
• Consulting
Thank you for your attention!

info@ozeninc.com
(408) 732-4665
1210 E. Arques Ave.
Sunnyvale, CA 94085