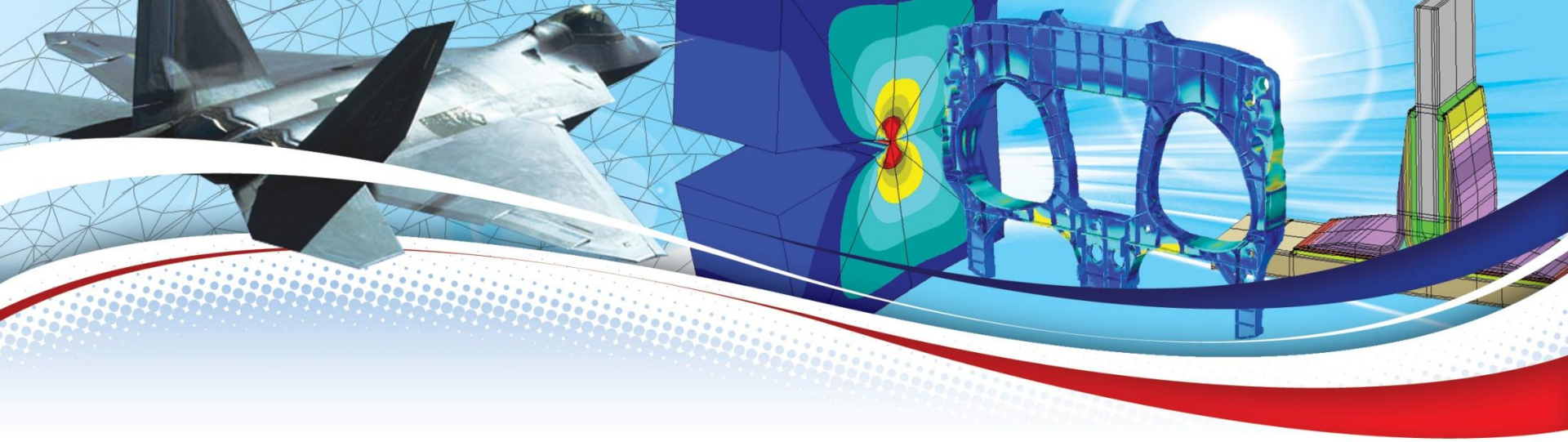


StressCheck Hints and Tips: **Fracture Mechanics**

2012 Afgrow User's Conference
September 11-12, 2012

Brent Lancaster
ESRD, Inc.

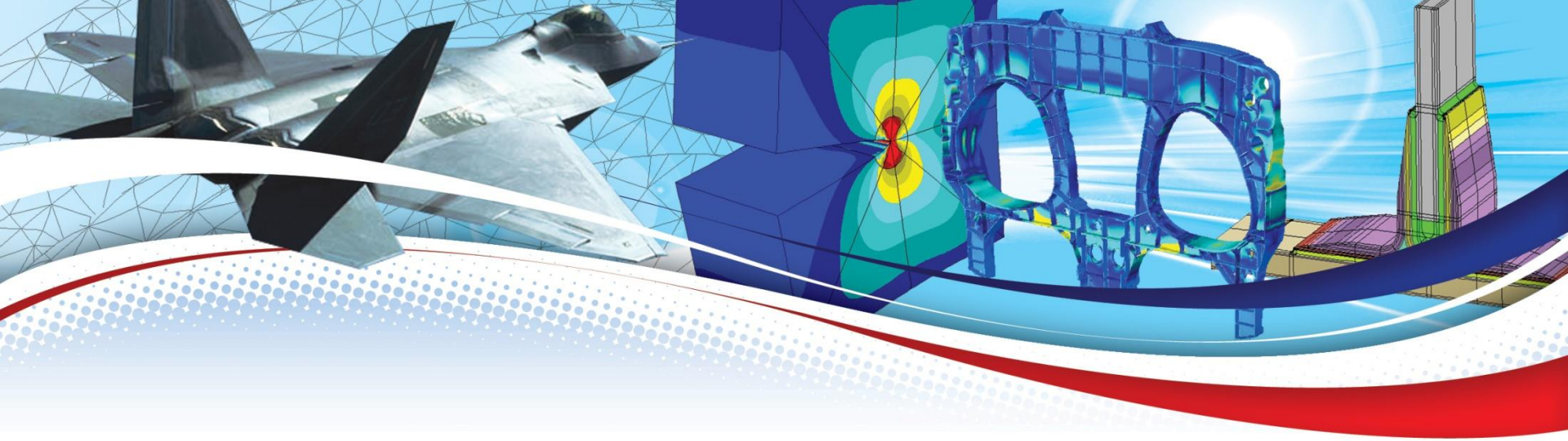
esrd

StressCheck®

Overview

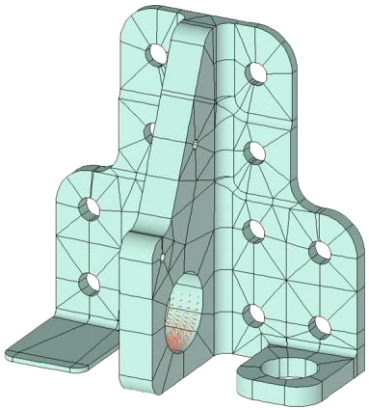
STRESSCHECK

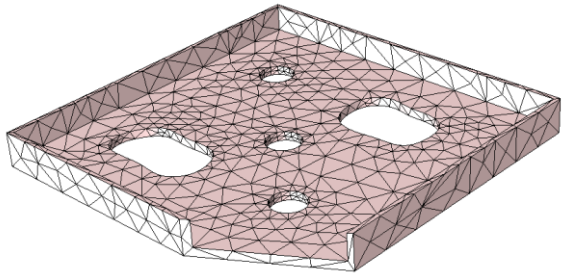
- In the next hour, ESRD will cover the following best practices:
  - General meshing guidelines for singularities
  - Crack automeshing in 2D/3D
  - Extraction of SIF in 2D/3D

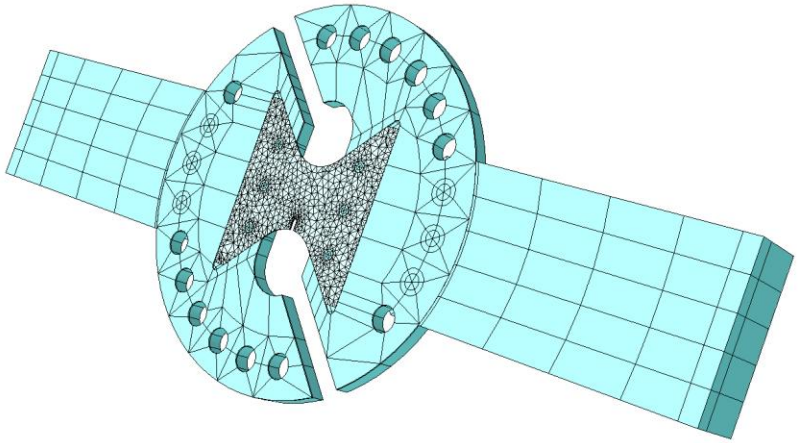


StressCheck Hints & Tips

Meshing Overview & Best Practices







esrd

StressCheck®

Topics

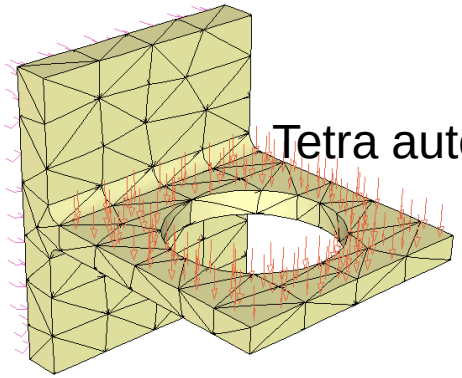
STRESSCHECK

- Meshing in StressCheck
  - Element Library
  - Element Associativity
- General Meshing Guidelines
  - Vertex Angles
  - Category A Problem Mesh Strategy
    - Solution Analytic Everywhere in Domain
      - No discontinuities in geometry or boundary conditions
    - Examples of Meshes for Category A
  - Category B Problem Mesh Strategy
    - Solution Analytic Everywhere Except in Finite Number of Locations
      - Discontinuities in geometry or boundary conditions exist
        - Re-entrant corners and/or point loads
    - Examples of Meshes for Category B

StressCheck Meshing Overview: Element Library

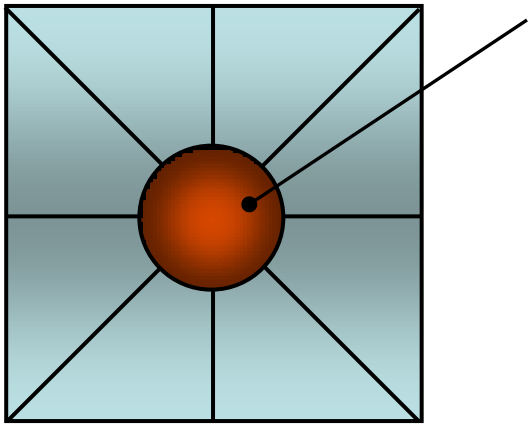
STRESSCHECK

- Simple element library based only on topology:
  - Quadrilaterals, Triangles
  - Beams, Fasteners, Links
  - Contact Regions
  - Hexas, Pentas, Tetras

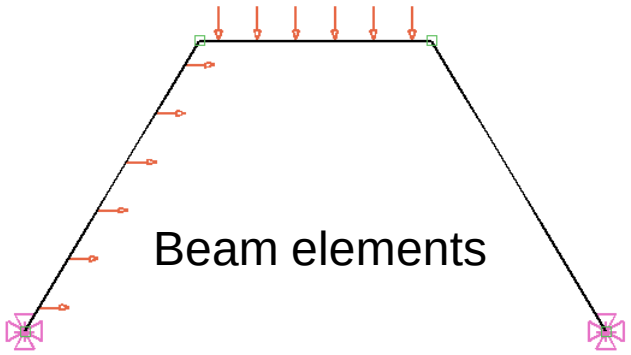


Tetra automesh

Triangles

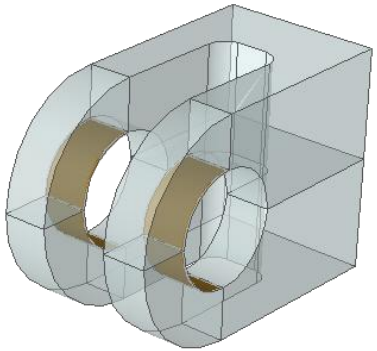


Fastener



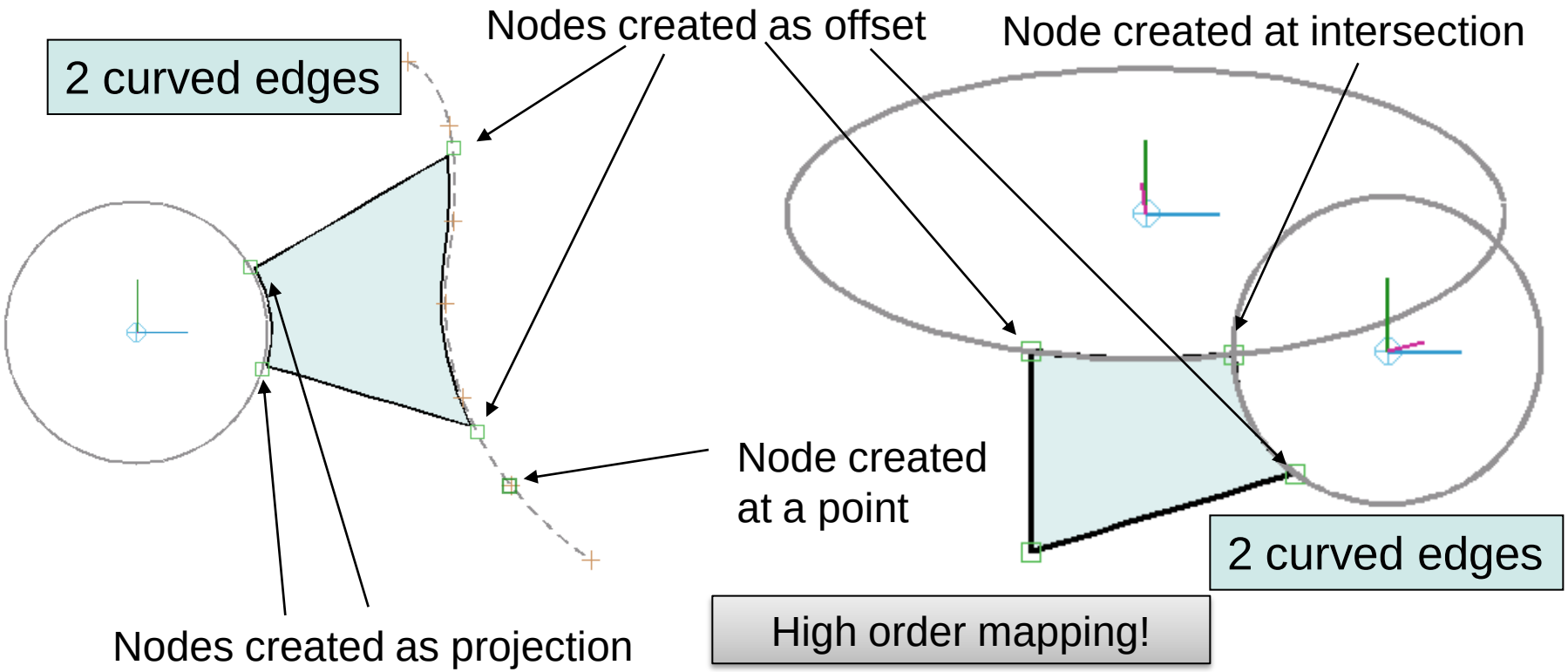
Beam elements

Contact Regions



StressCheck Meshing Overview: Element Associativity

- Elements are geometrically associative.
- Associativity determined by node creation.



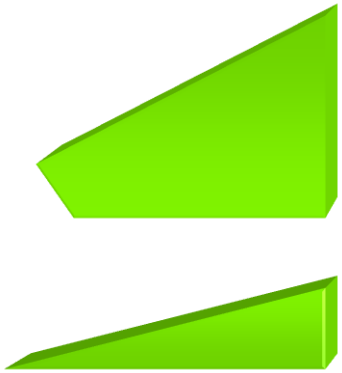
General Meshing Guidelines: Vertex Angles

STRESSCHECK

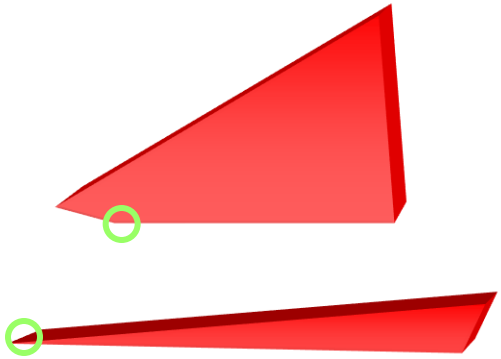
- Triangles, Pentahedral and Tetrahedral elements
  - Vertex angles as close to 60 degrees as possible.
- Quadrilateral and Hexahedral elements
  - Vertex angles as close to 90 degrees as possible.
- Practical limits: 10° < Vertex Angles < 150°

Ideal

Acceptable



Not recommended



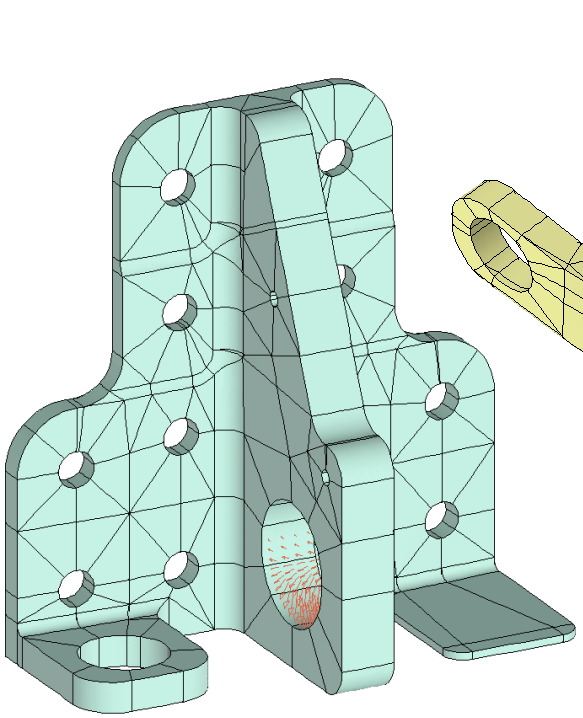
General Meshing Guidelines: Category A Problem Mesh Strategy

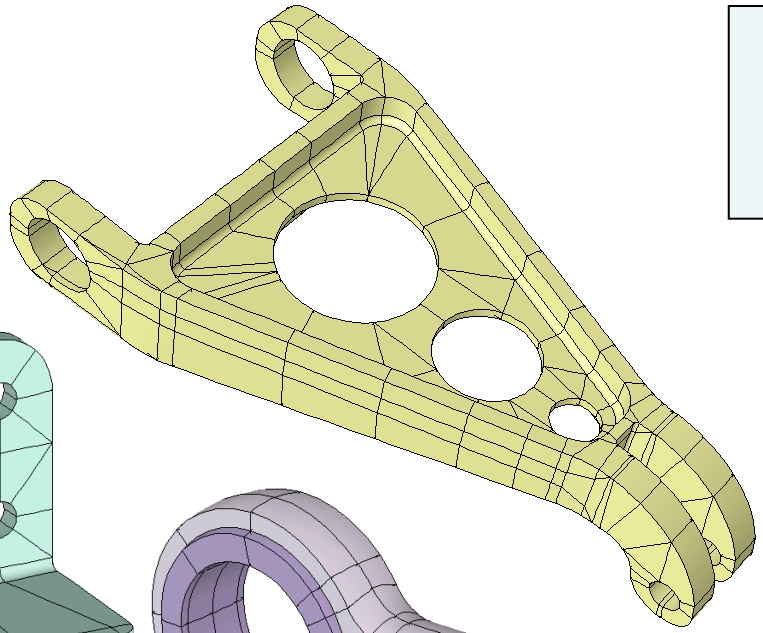
- For problems in Category A, *uniform meshes are optimal.*
  - Describe boundaries with smoothly connected fillets.
  - The number of elements is controlled by the geometric complexity of the part.
  - Place nodes at boundary points where the curvature changes abruptly.
  - Place nodes along boundaries to create curved elements using options: offset, intersection, mid-offset, projection.
  - If the solution is not acceptable for the highest p-level, increase mesh density in region of interest and repeat p-extension.
    - Use h-refinement tools if applicable (manual meshing only).

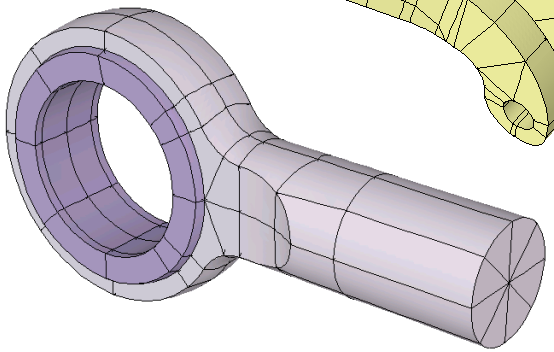
General Meshing Guidelines: Category A Problem Mesh Examples

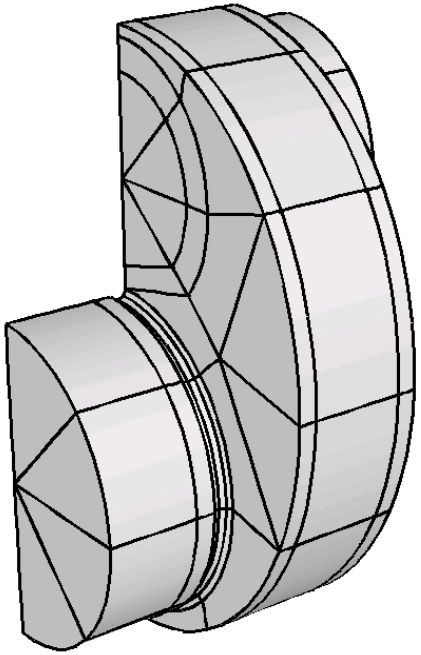
STRESSCHECK

- Hand-meshes are shown for concept
- Note that fillets are used for transitions





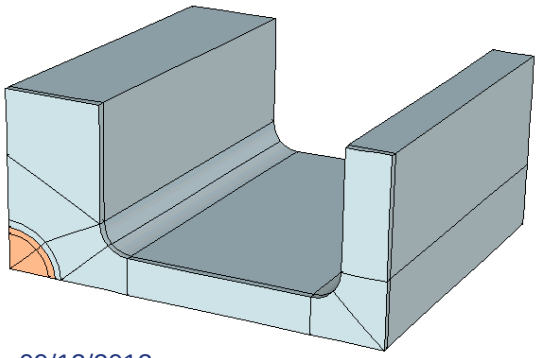


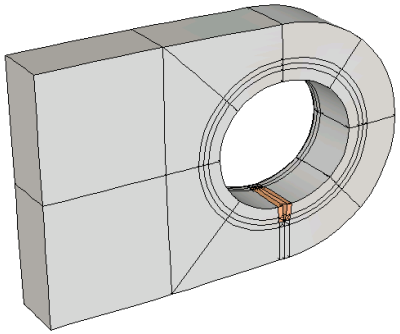


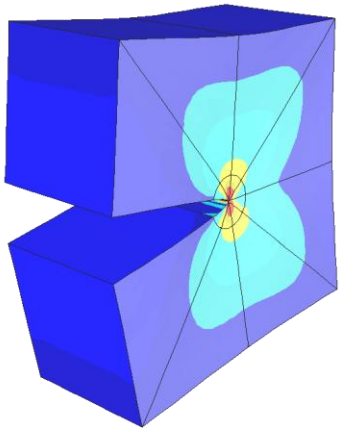
General Meshing Guidelines: Category B Problem Mesh Strategy

STRESSCHECK

- For problems in Category B, *geometrically graded meshes are optimal.*
  - The diameter of elements should decrease in geometric progression toward singular points with a common factor of 0.15.
  - One or two layers of geometrically graded elements around singular points is typically sufficient.
  - Elsewhere the size of elements is controlled by the geometric complexity of the part.

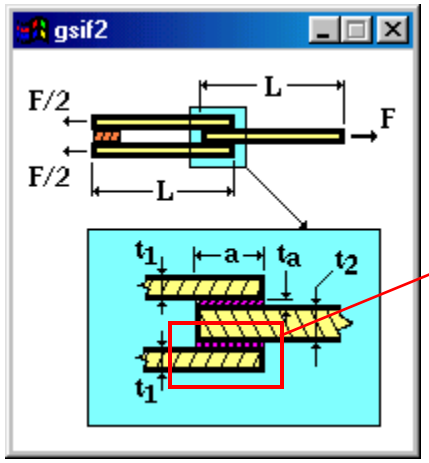


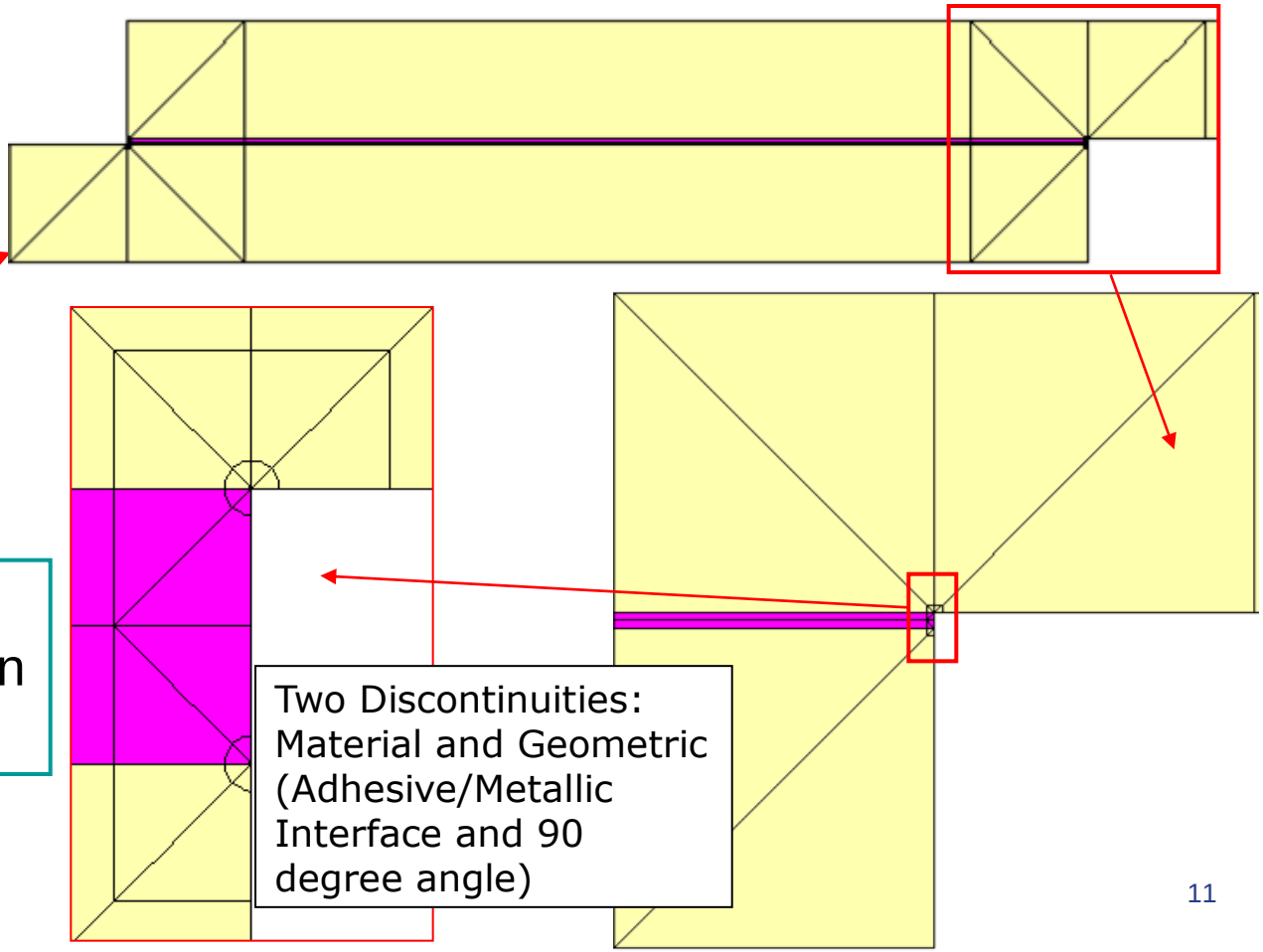




General Meshing Guidelines: 2D Category B Problem Mesh Example

- Example of 2D-meshing with refinement



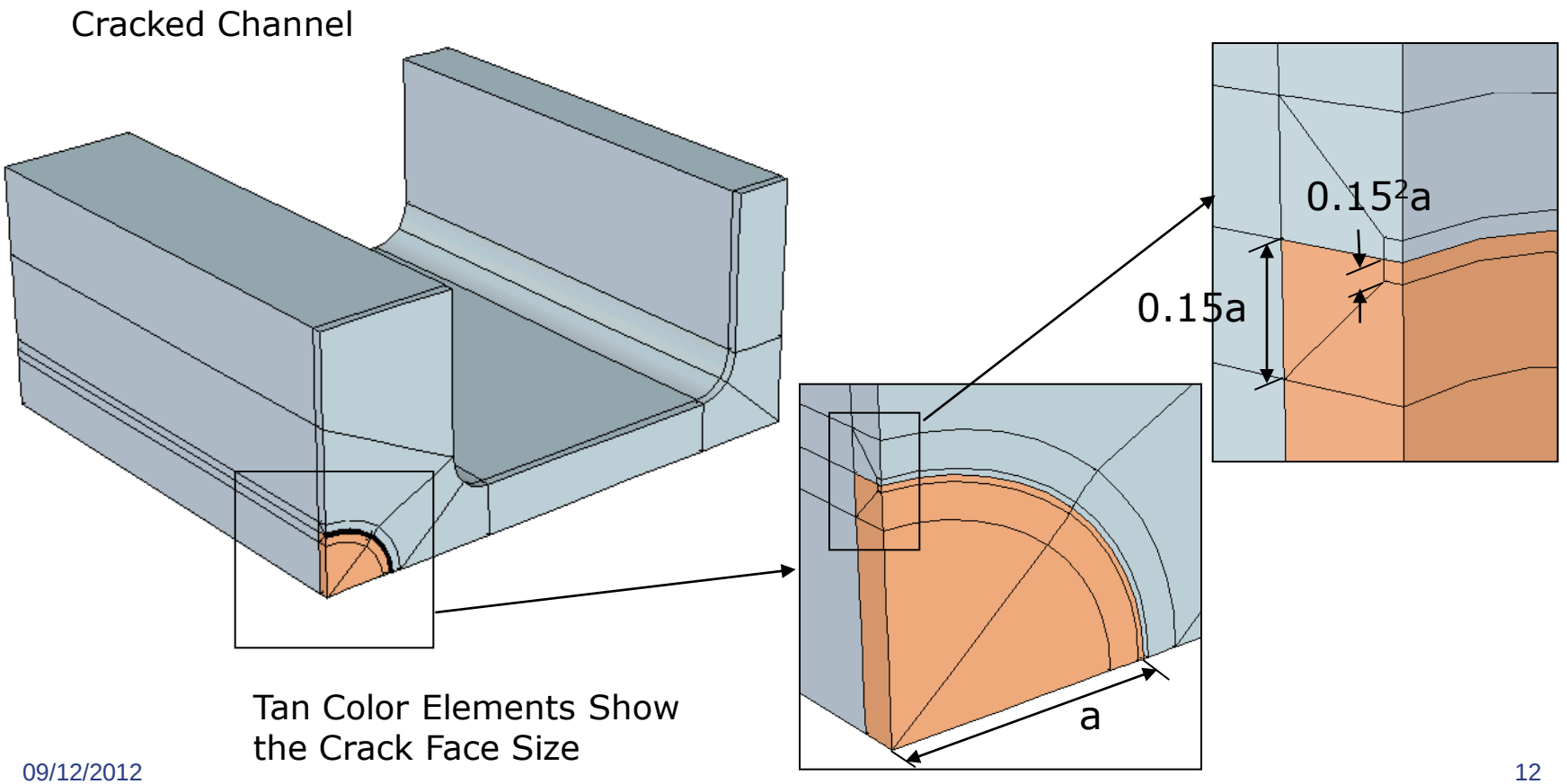


Lap-joint specimen
Geometric progression
Common factor 0.15

Two Discontinuities: Material and Geometric (Adhesive/Metallic Interface and 90 degree angle)

General Meshing Guidelines: 3D Category B Problem Mesh Example

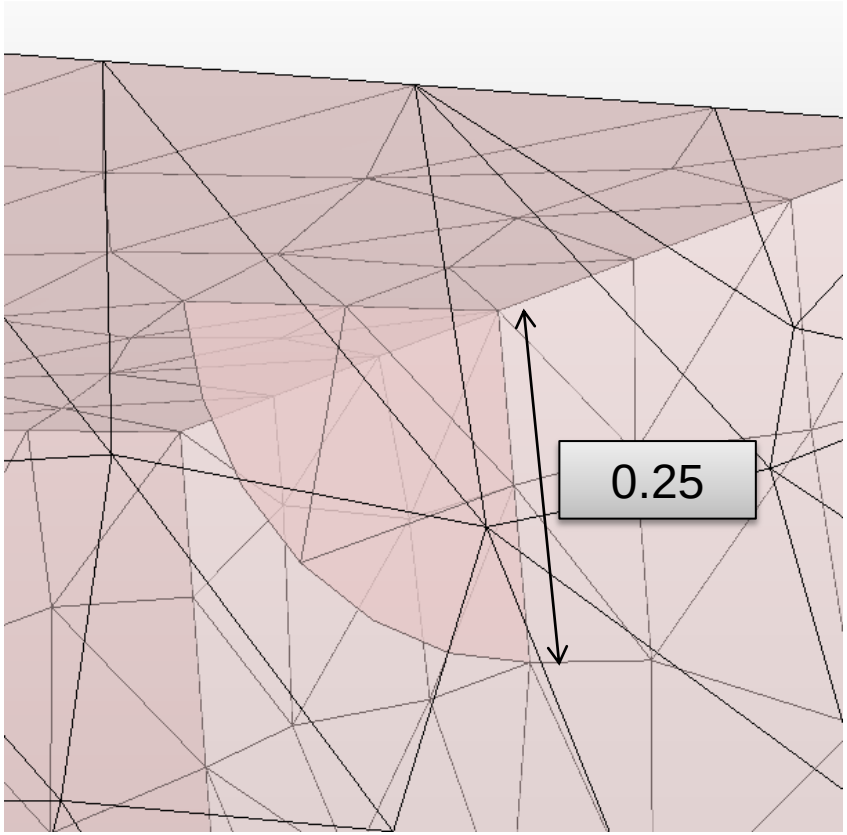
- Example of 3D-meshing with refinement:



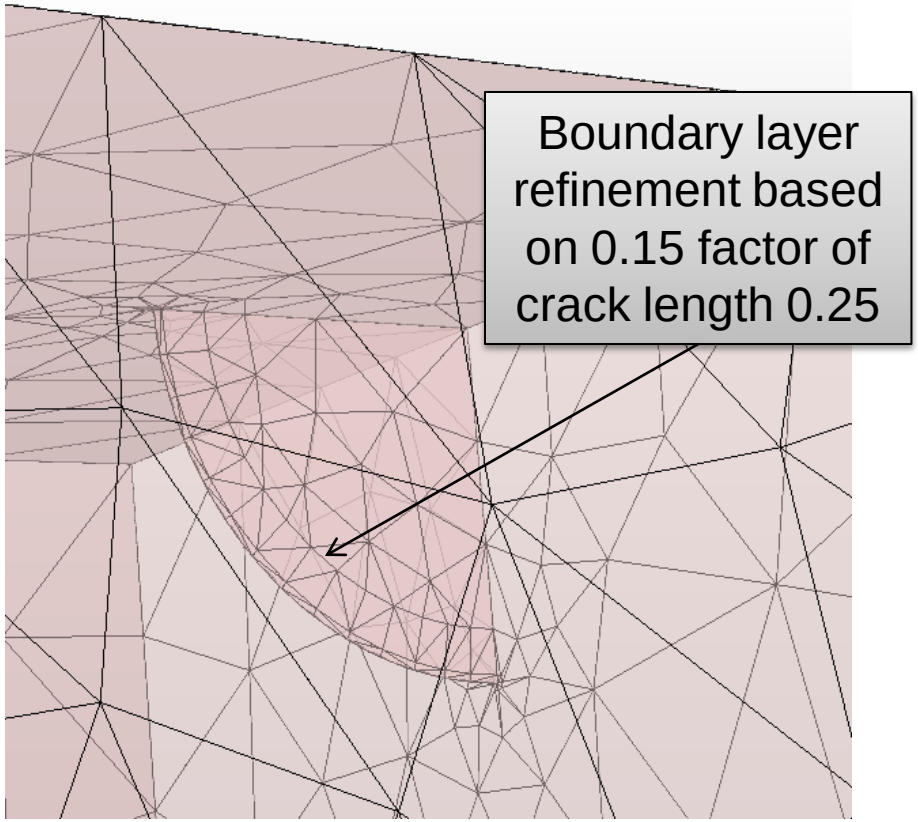
General Meshing Guidelines: 3D Category B Problem Mesh Example

STRESSCHECK

Corner Crack
Poor Automesh

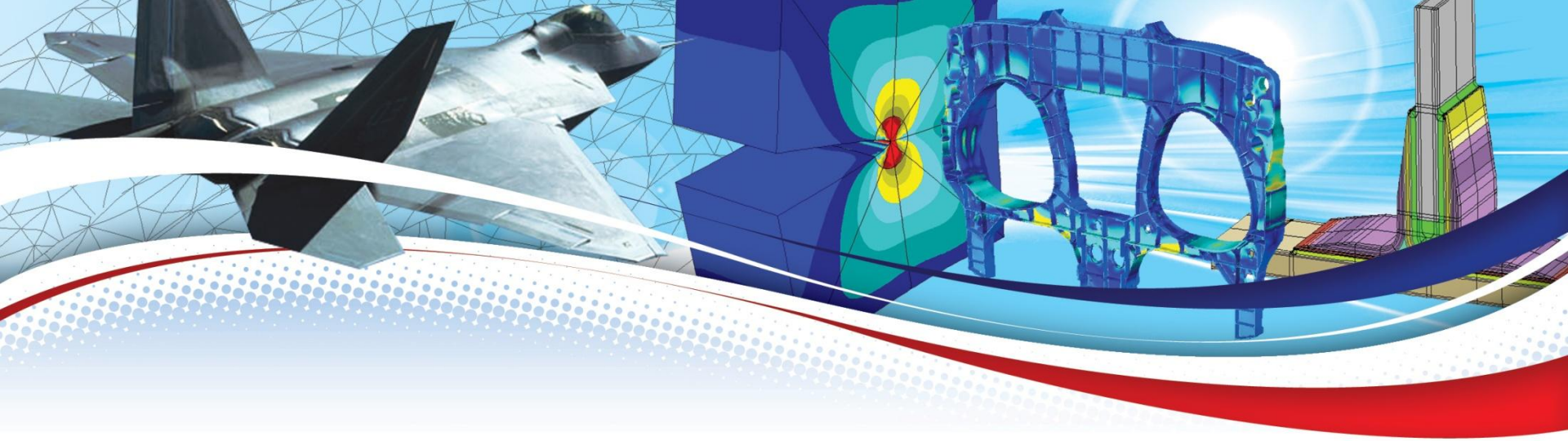


Corner Crack
Ideal Automesh

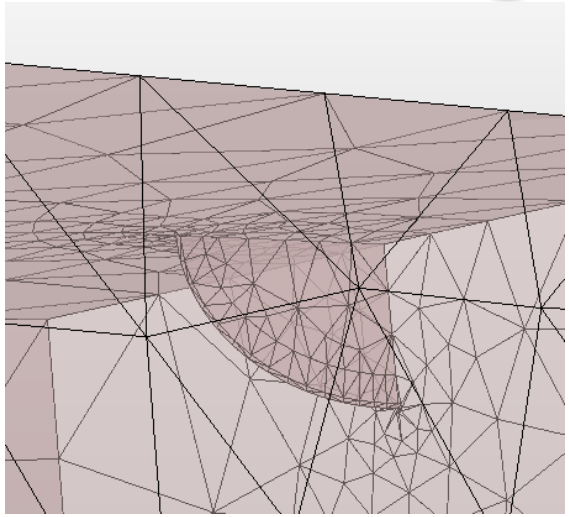


Summary and Conclusions

- The control of discretization, or approximation, errors is simple when p-extensions are used with properly designed meshes.
  - Determine a priori if the mesh for the problem at hand should be designed for Category A or Category B.
- It is important to produce accurate representations of the domain and boundary conditions so that these potential sources of error do not affect the accuracy of the data of interest.
  - Improper boundary conditions can lead to problems for which the exact solution does not exist or is trivially zero.



StressCheck Hints and Tips:
Crack Automeshing in 2D and 3D



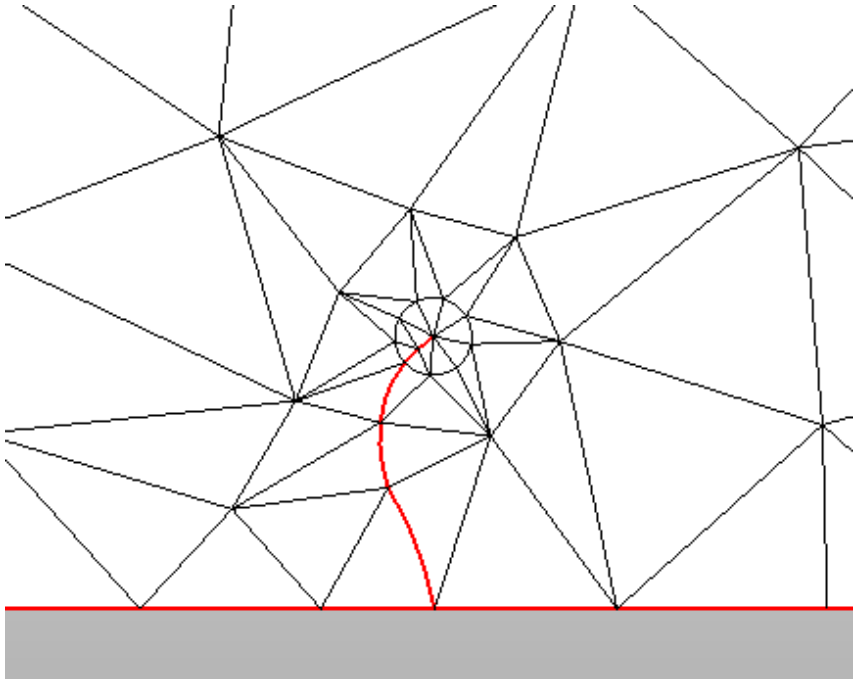
esrd

StressCheck®

Automeshing Cracks in 2D

STRESSCHECK

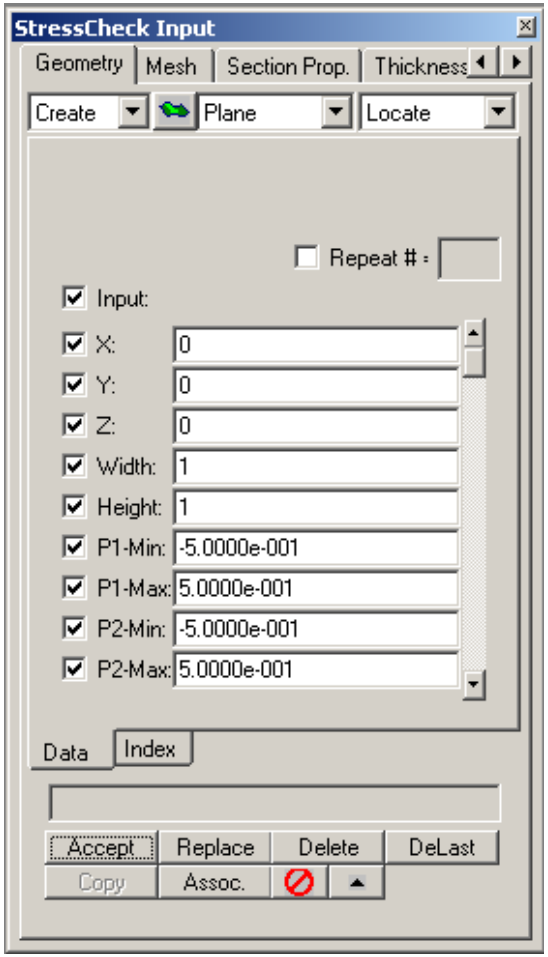
- Basic Process:
  - Produce a valid sheet body
  - Generate curvature representing crack shape
  - Imprint curvature representing crack in sheet body
  - Specify resulting curvature as a “Crack Edge” in automesher



Automeshing Cracks in 2D

STRESSCHECK

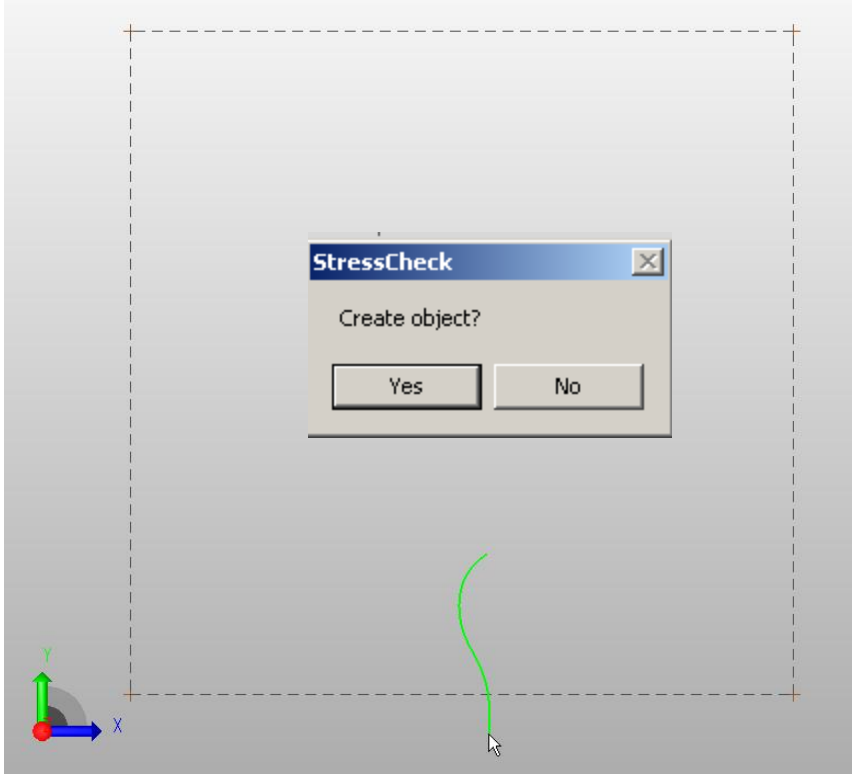
- Sheet Body:
  - Change reference to “Planar”
  - Go to the “Geometry” tab in “Model Input”
  - Use “Create->Plane->Locate”
  - Enter values as shown in image on right
  - Click “Accept”



Automeshing Cracks in 2D

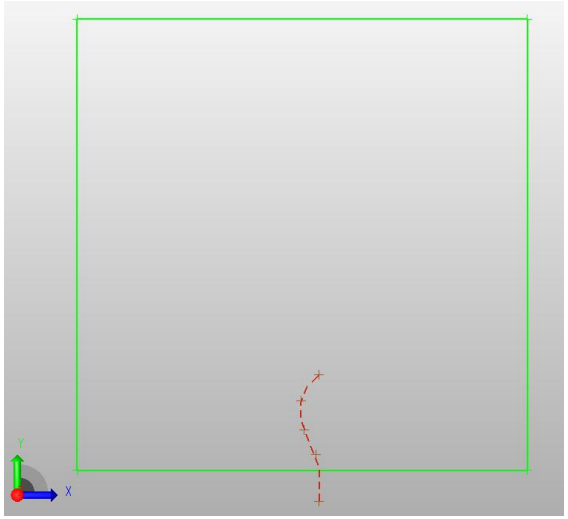
STRESSCHECK

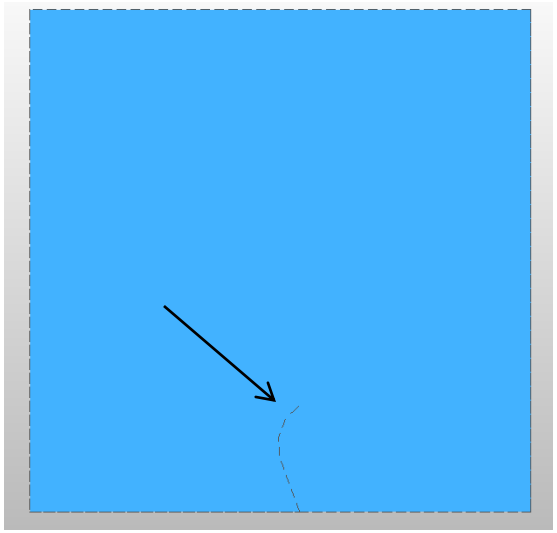
- Crack Shape:
  - Change to “Curve” selector
  - Use “Create->Spline->Locate”
  - De-activate inputs
  - Left click within the plane domain several times until general curvature results
  - Double-left click outside domain (see image on right) and click “Yes” to create the spline



Automeshing Cracks in 2D

- Imprint Curvature on Plane:
  - Change to “Surface” selector
  - Use “Create->Body-Imprint->Curve Normal”
  - Select the Plane
  - Select the spline
  - Press Accept

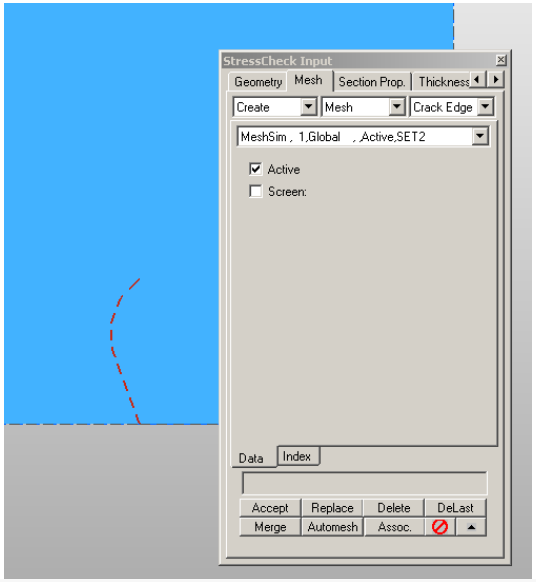


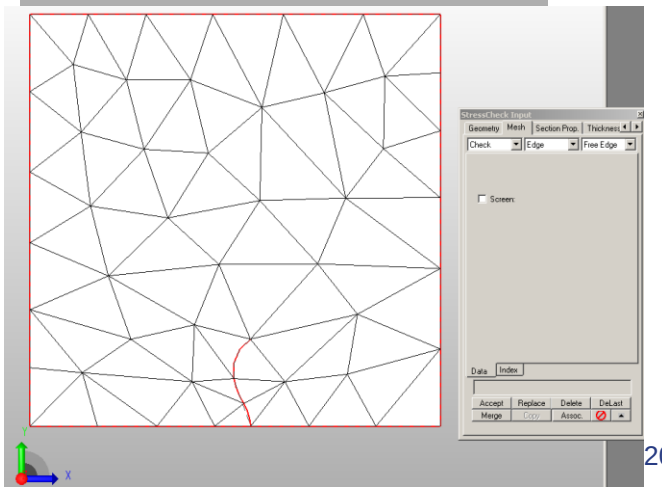


Automeshing Cracks in 2D

STRESSCHECK

- Specify curvature as “Crack Edge”:
  - Click on the “Mesh” tab
  - Use “Create->Mesh->Auto”
  - Select sheet body
  - Click “Accept”
  - Use “Create->Mesh->Crack Edge”
  - Select the imprinted curve
  - Click “Accept”
  - Click “Automesh”
  - Use “Check->Edge->Free Edge”
    - Crack highlights red

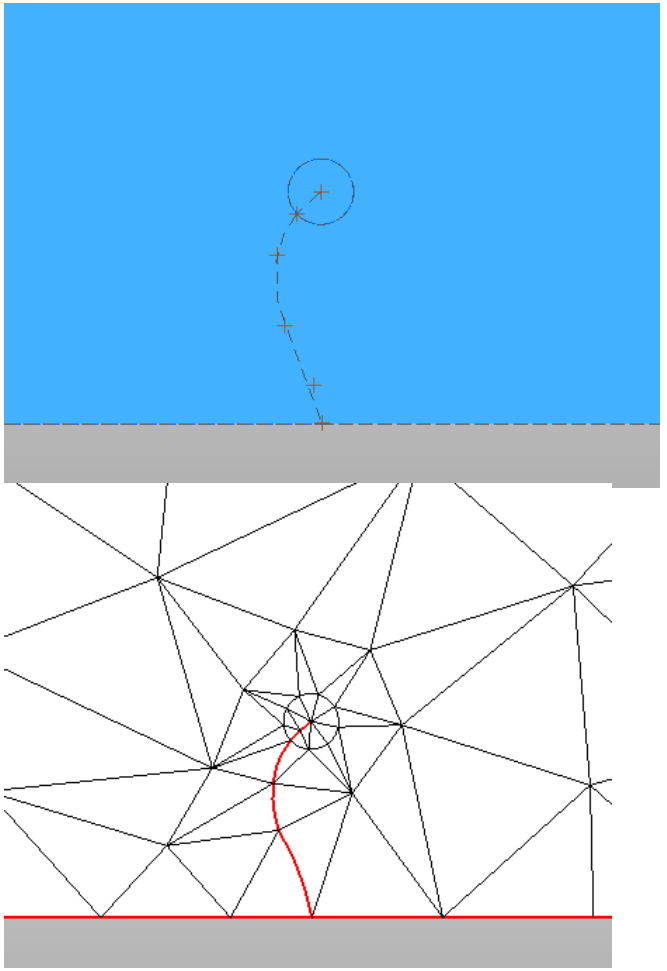




Automeshing Cracks in 2D

STRESSCHECK

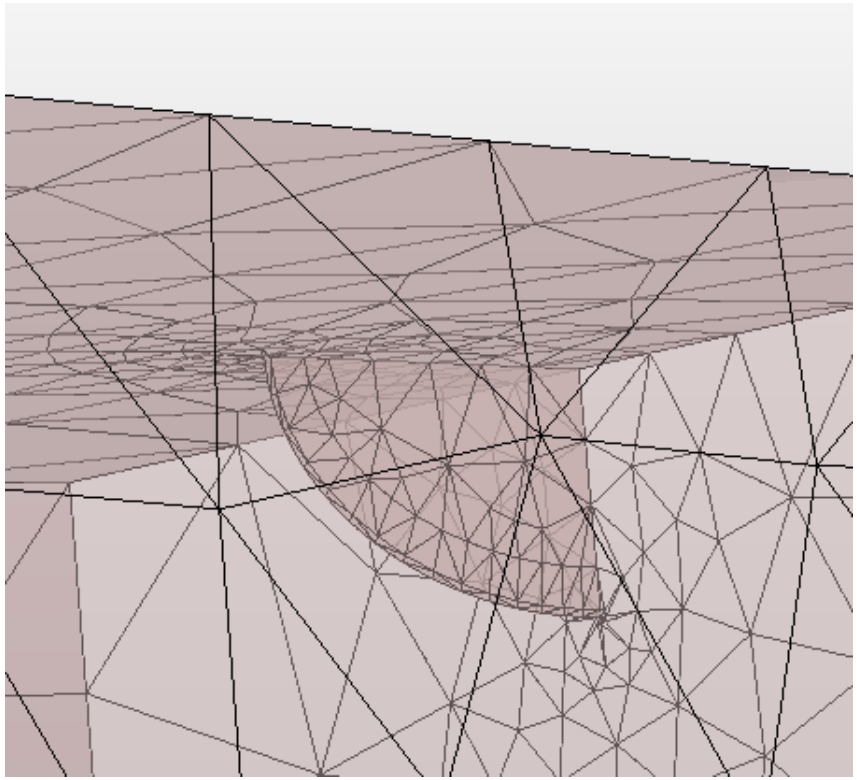
- Refinement at Crack Tip:
  - Delete all mesh records
  - Need to create a boundary layer with a circle
    - Go to the “Geometry” tab
    - Use “Check->Point->Distance”
      - Check crack length
    - Use “Select->Point->Locate”
    - Click on the “Index” tab
    - Locate “Offset” point and click on list item
    - Click “Blank”
      - Don’t want to allow picking
    - Toggle to “Curve” selector
    - Use “Create->Circle->Point”
    - Activate radius, type 0.15*crack length
    - Click on crack tip point
  - Now, imprint the circle and re-mesh, making sure to select each segment of the crack curvature



Automeshing Cracks in 3D

STRESSCHECK

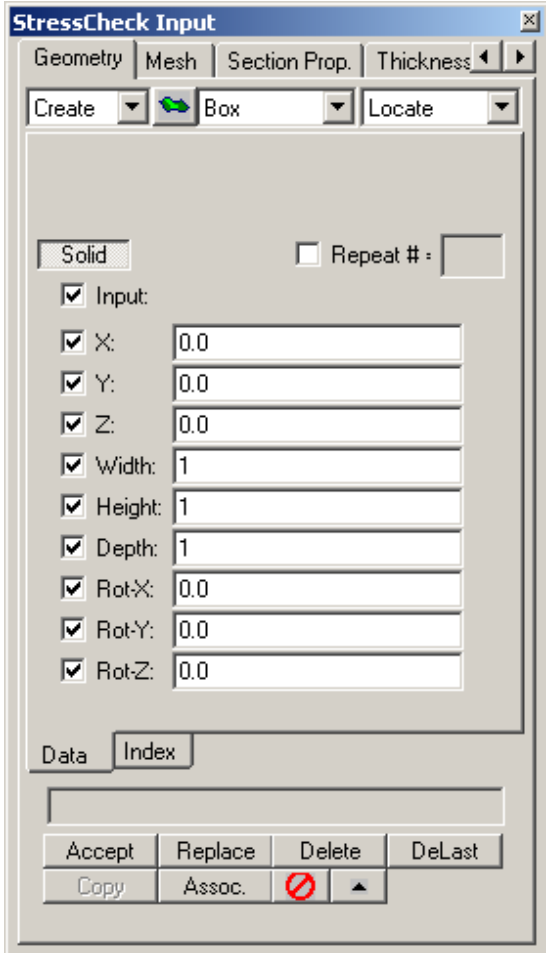
- Basic Process:
  - Produce a valid solid
  - Generate a surface representing the crack
  - Embed the surface in the solid using Boolean-Union
  - Specify the now embedded surface as a “Crack Face” in the automesher
  - If available, use Boundary Layer automesher
    - Otherwise, use surfaces as boundary layers



Automeshing Cracks in 3D

STRESSCHECK

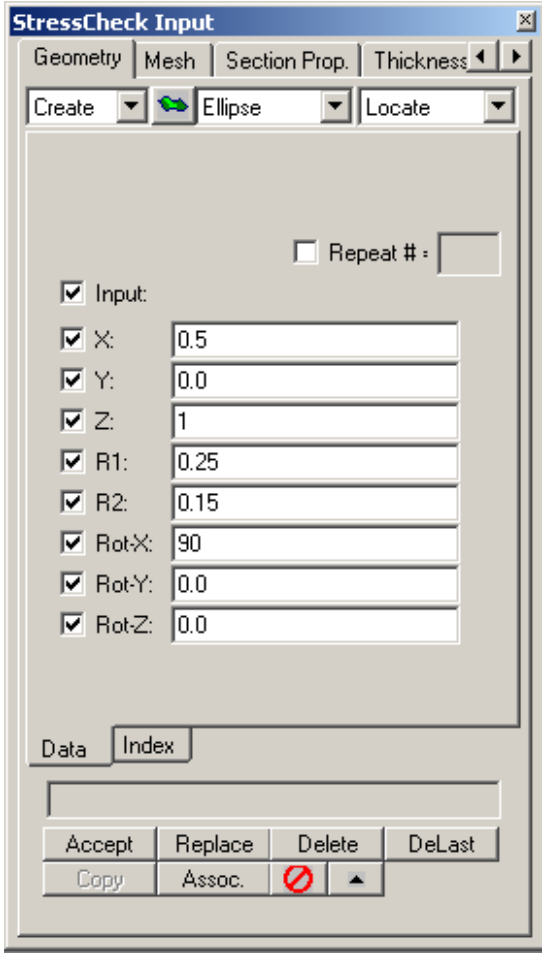
- Solid Body:
  - Set reference to “3D”
  - Go to the “Geometry” tab in “Model Input”
  - Use “Create->Box->Locate”
  - Enter values as shown in image on right
  - Click “Accept”

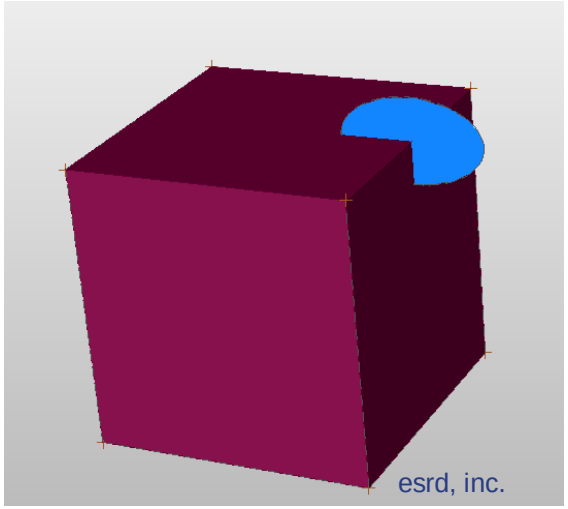


Automeshing Cracks in 3D

STRESSCHECK

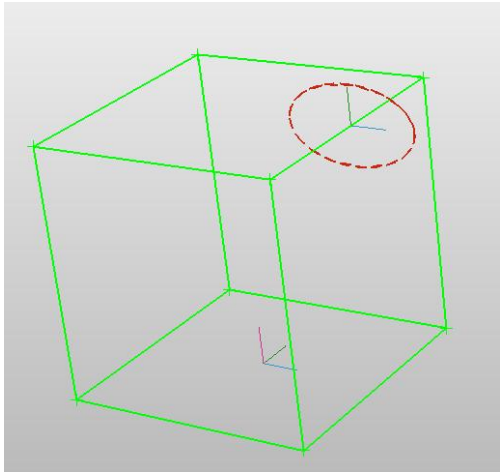
- Crack surface:
  - Use “Create->Ellipse->Locate”
  - Enter values as shown in image on right
  - Click “Accept”

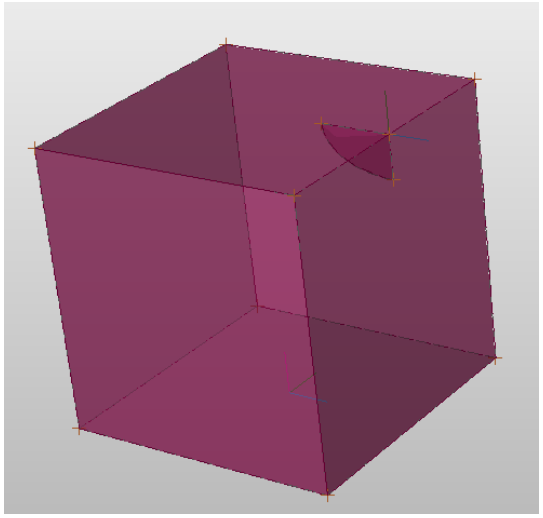




Automeshing Cracks in 3D

- Embed crack:
  - Use “Create->Body->Bool-Union”
  - Select the solid
  - Select the surface
  - Click “Accept”

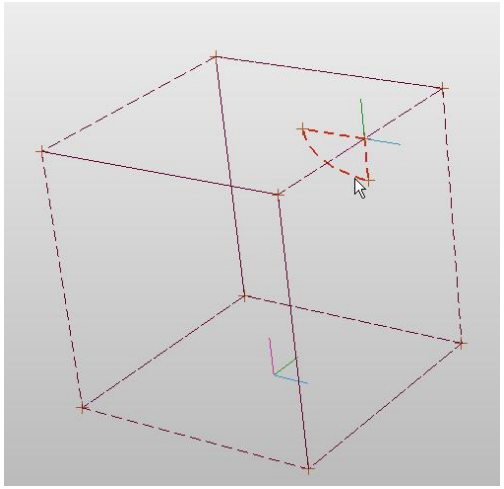


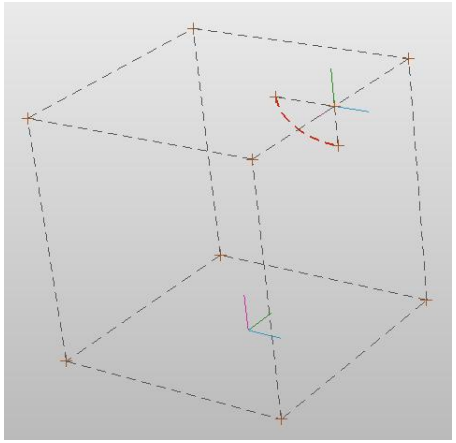


Automeshing Cracks in 3D

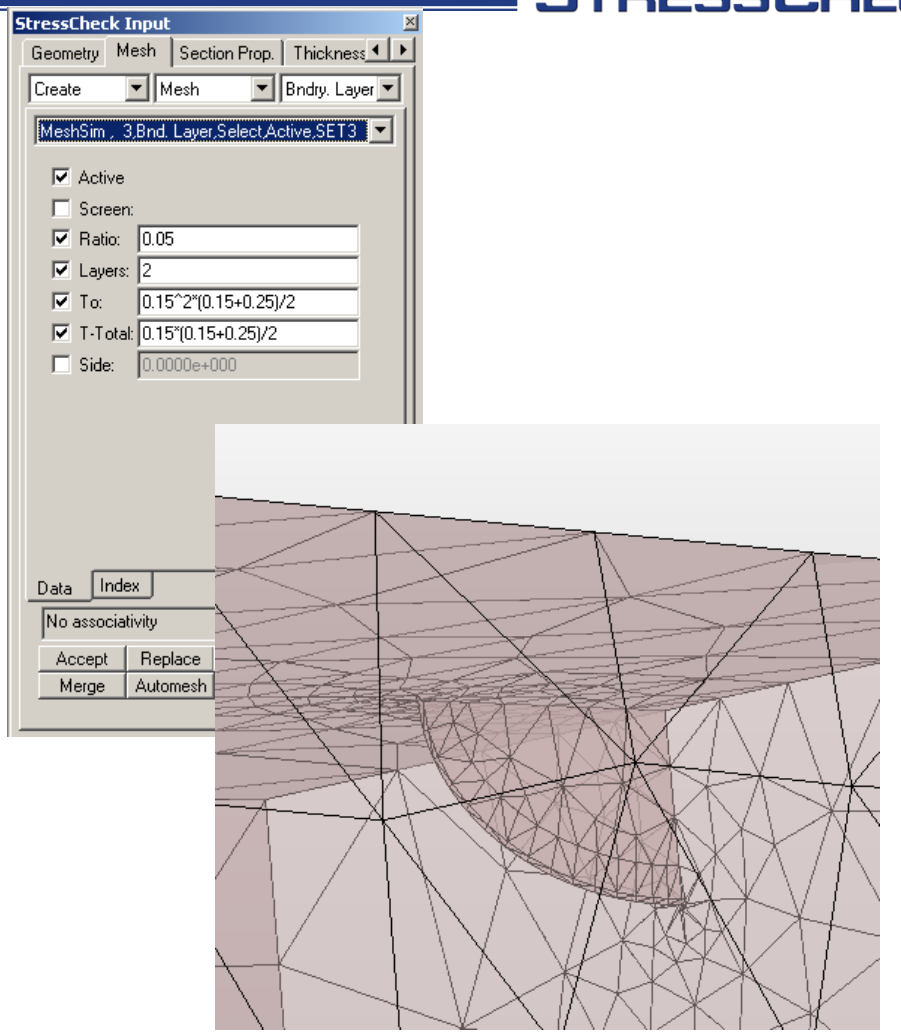
STRESSCHECK

- Automesh crack with “Crack Face”:
  - Click on the “Mesh” tab
  - Use “Create->Mesh->Auto”
  - Click on the solid
  - Click “Accept”
  - Use “Create->Mesh->Crack Face”
  - Turn off curve display
  - Turn on wireframe display
  - Select boundary of embedded surface (top right)
  - Click “Accept”
  - Use “Create->Mesh->Bndry. Layer”
  - Turn on curve display
  - Turn off surface display
  - Select ellipse (bottom right)

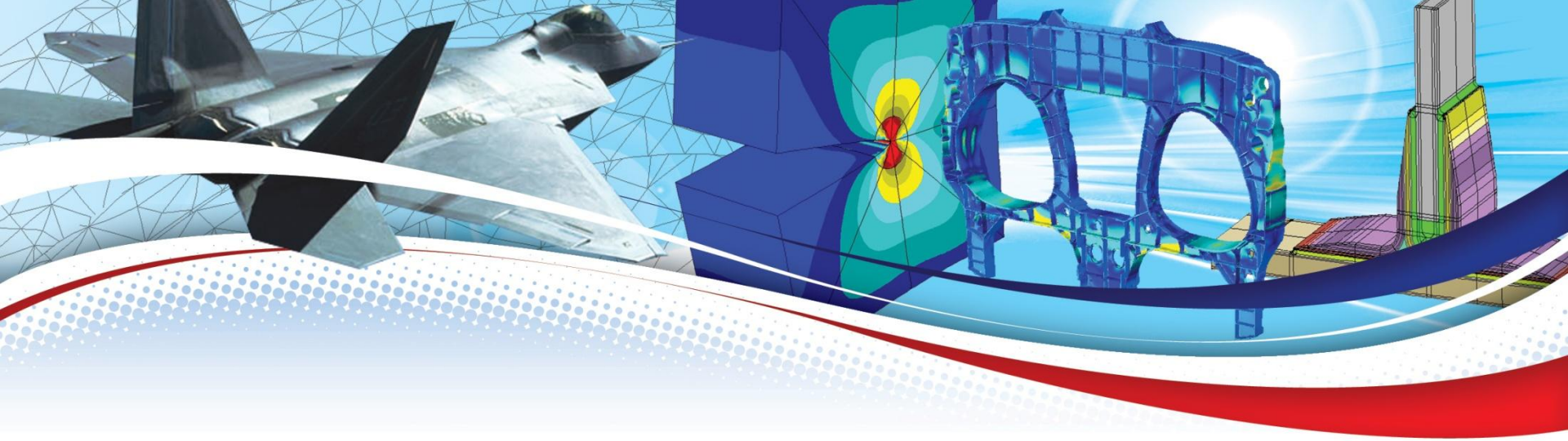




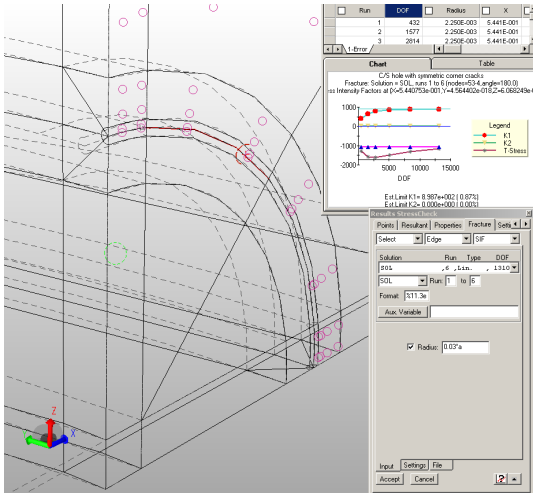
Automeshing Cracks in 3D



- Automesh crack with "Crack Face" (cont.):
  - Enter values as shown in image on right
  - Click "Accept"
  - Click "Automesh"



StressCheck Hints and Tips:
Fracture Mechanics Extractions



esrd

StressCheck®

Selection of Contour Integral Radius for SIF Extractions

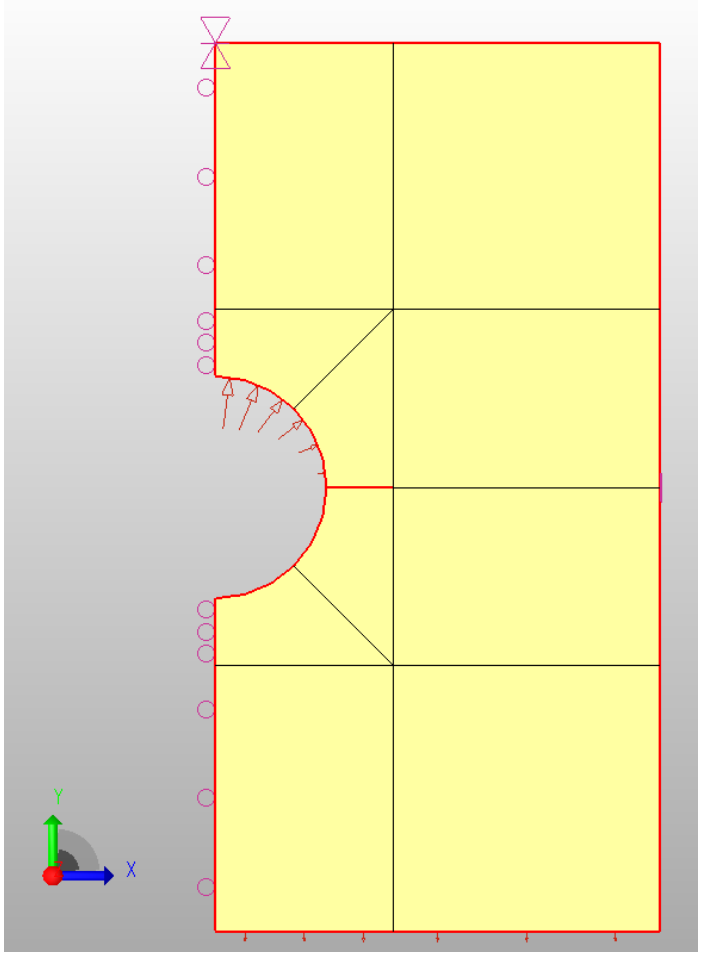
STRESSCHECK

- Planar reference:
  - Computation of SIF by CIM is theoretically path independent, so the selection of integration radius is in general arbitrary
    - A practical size for the integration radius must be chosen such that it lies within domain
    - This independence can be proven by extracting SIF for several integration radii
  - Additionally, it is not always necessary to include layers of refinement to obtain low error in the SIF

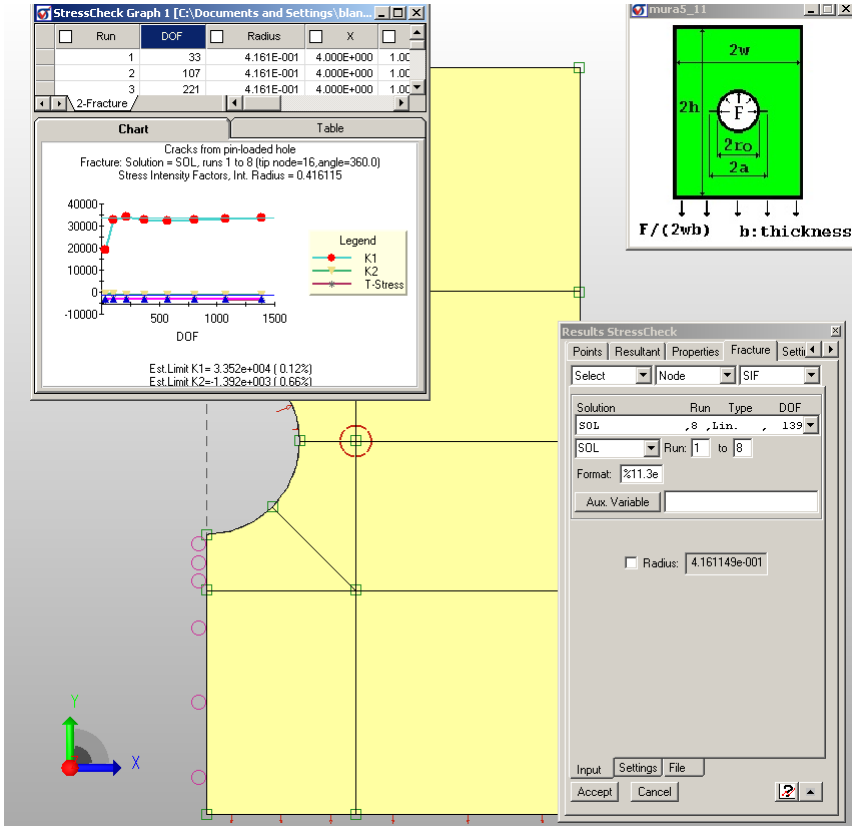
Example Planar Problem

STRESSCHECK

- “mura5_11” from “2D-Fracture” Handbook
- No refinement at the crack tip
- Solve Linear p=1 to 8
- Go to the “View Results” interface
- Click on the “Fracture” tab



Example Planar Problem



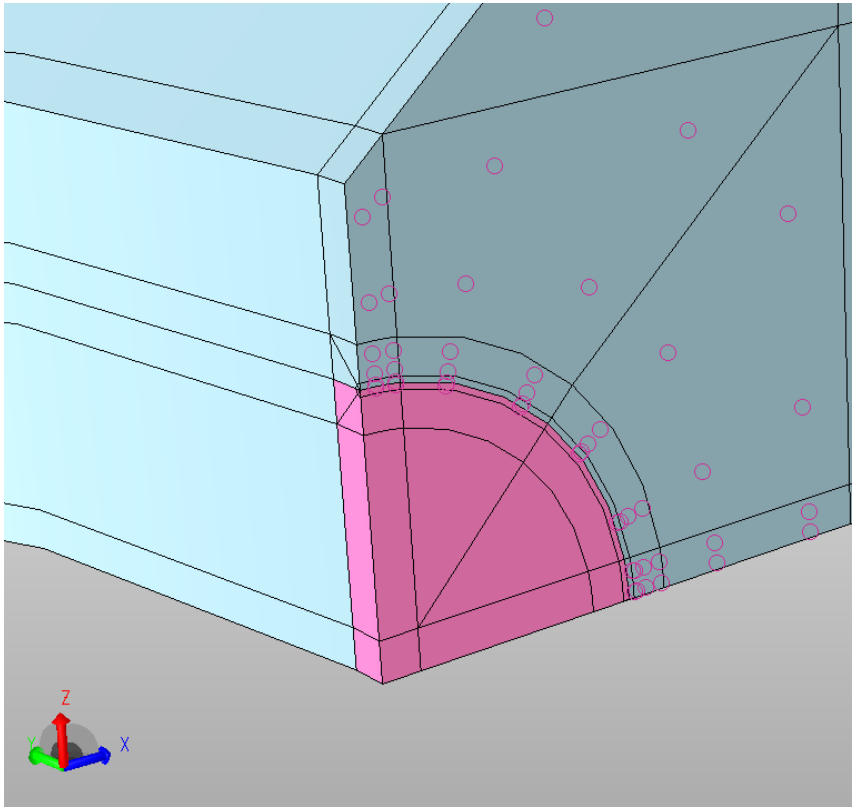
- Turn on node display
- Turn on curve display
- Use “Select->Node->SIF”
- Use Runs 1 to 8
- Left click and drag outward from the crack tip node
  - Radius will be calculated by motion
- Release left button
  - Integration circle will be drawn
  - SIF will be computed automatically
- Try several other radii and compare to SIF
  - Values should remain stable, within 1%

Selection of Contour Integral Radius for SIF Extractions

STRESSCHECK

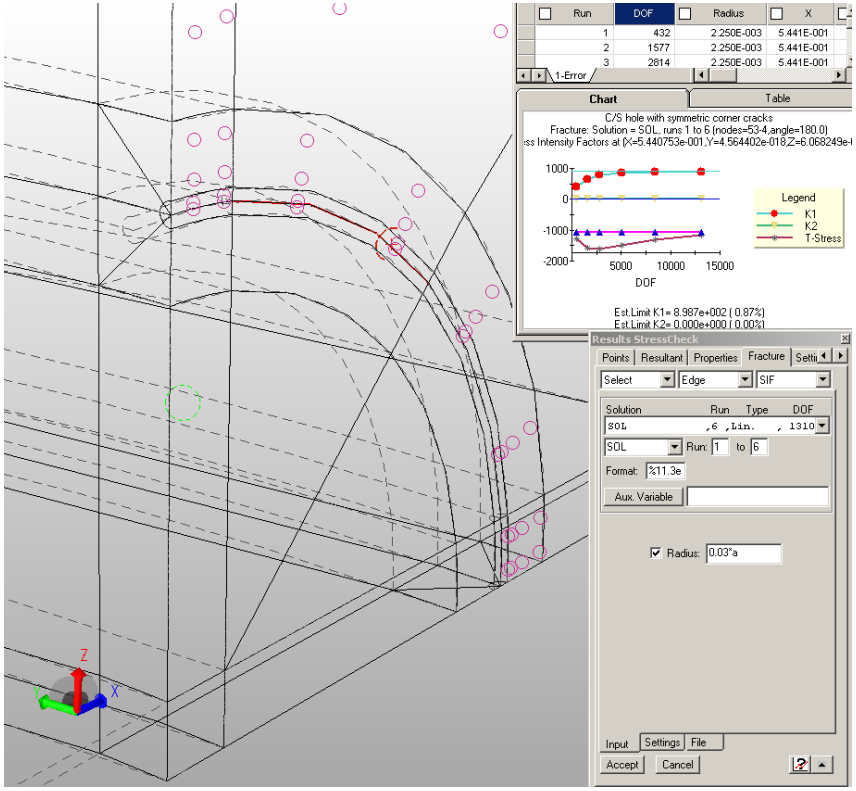
- 3D reference:
  - Computation of SIF by CIM is NOT path independent, so care must be taken in the mesh design and selecting the integration radius
    - Most particularly for curved cracks
  - **Best**: geometrically graded layers of refinement around crack tip using a factor of **0.15** and based on the characteristic length (i.e. crack size)
    - **Two is usually recommended**
    - The reason for using layers of refinement graded in geometric progression towards the singular point is to distribute the error evenly on the elements in the vicinity of the crack tip
  - **Best**: selection of integration radius should be such that it is just outside of the innermost layer of refinement
    - Using an integration radius as small as possible will decrease the influence of adjacent points when computing the SIFs
    - A best practice is to extrapolate integration radius to zero

Example 3D Problem



- "SurfaceCrackCSHole.sci" in "3D-Fracture" Handbook
- Note two layers of geometric refinement around crack (pink)
- Solve p=1 to 6
- Go to the "View Results" interface
- Click on the "Fracture" tab

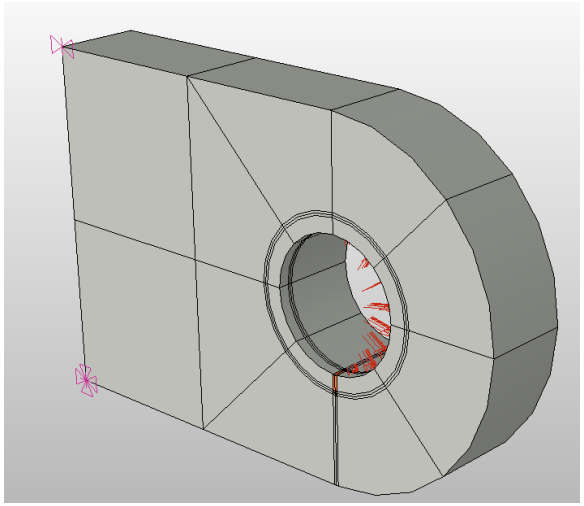
Example 3D Problem

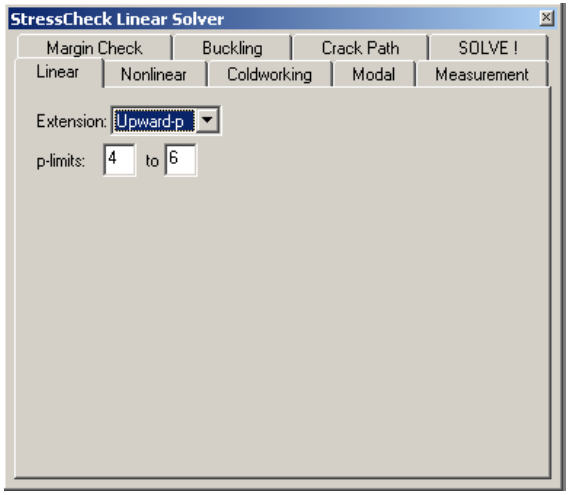


- Use “Select->Node->SIF”
- Use Runs 1 to 6
- In the “Radius” field, enter “0.03*a”
  - Since first layer is “0.0225*a”, want to enter radius just outside first layer
- Turn on curve display
- Turn on wireframe mode
- Select an element edge on the crack front
  - SIF should be automatically extracted
- Change integration radius to “0.025*a”, and click “Accept”
  - Note the change in value
  - Can use several more values to extrapolate to R->0

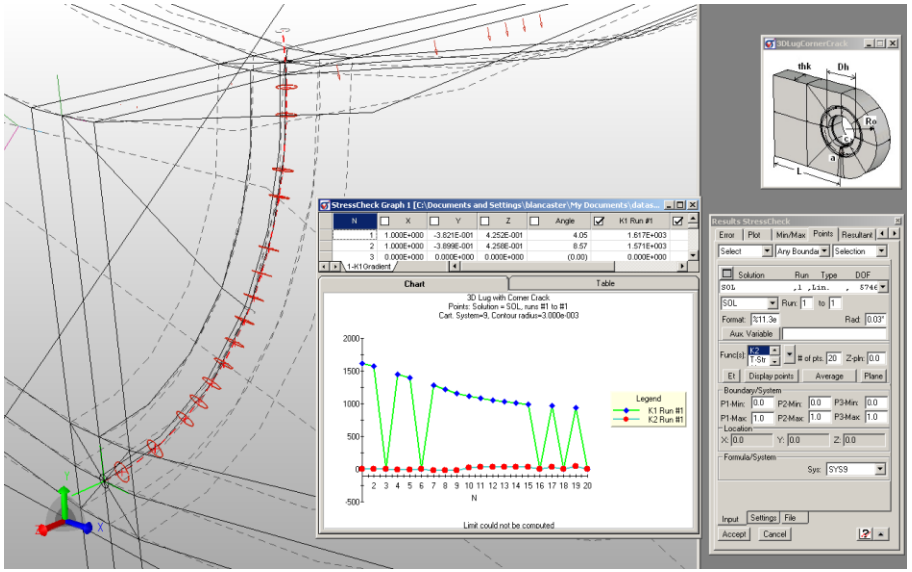
Computing SIF distributions on crack boundary

- Load the handbook problem “3DLugCornerCrack.sci ” from Fracture
- Go to the “Compute Solution” interface
- Click on the “Linear” tab
- Enter p-limits “4” to “6”
- Click on the “SOLVE!” tab
- Click “Solve”





Computing SIF distributions on crack boundary

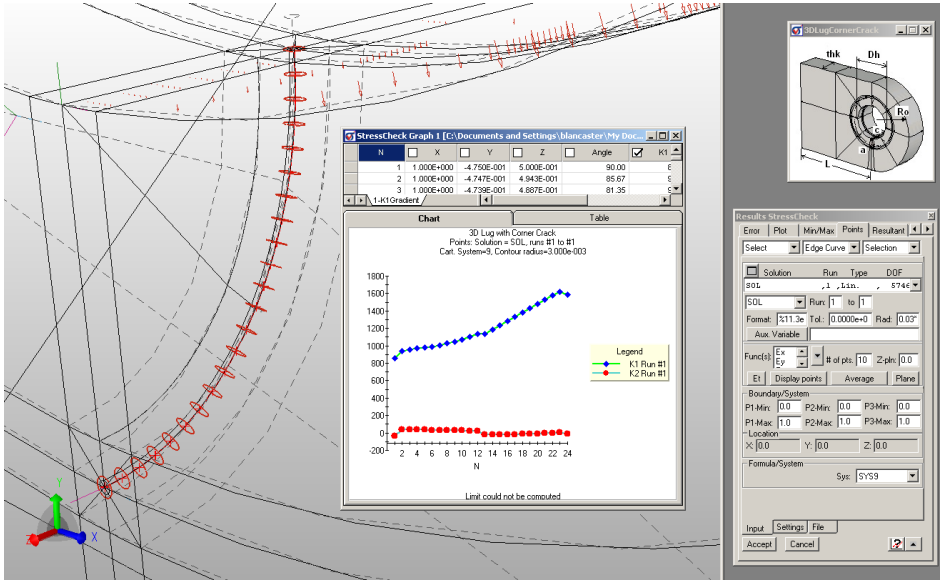


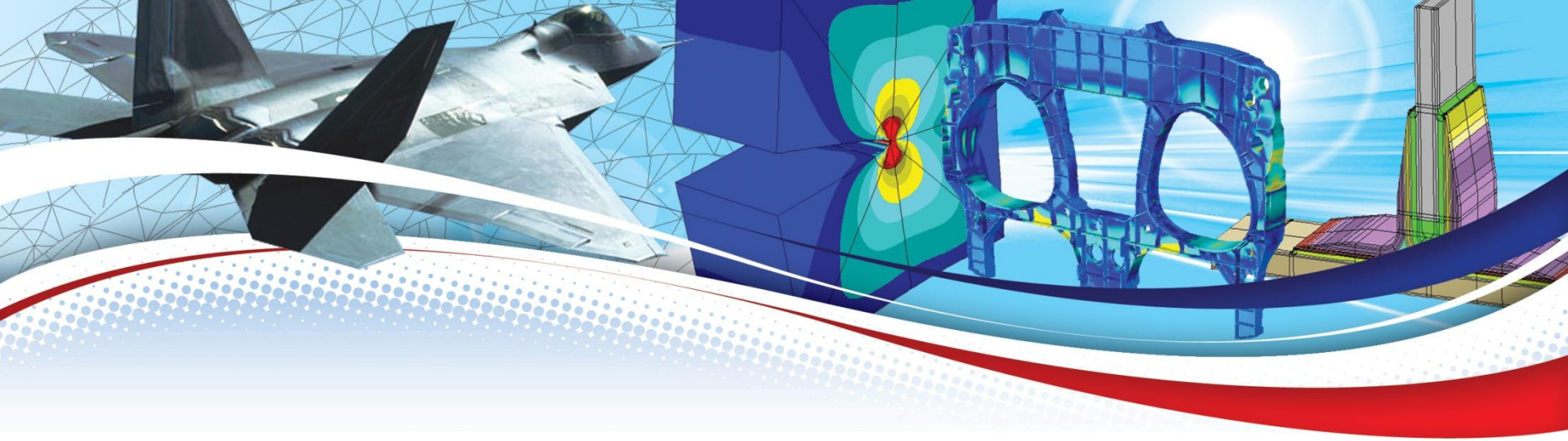
- Go to the “View Results” interface
- Click on the “Points” tab
- Use “Select->Any Boundary”
- Use Runs 1 to 3
- For “Func(s):”, select “K1” and “K2”
  - Hold CTRL to select both
  - “Rad:” will appear for input of contour integral radius
- For “Rad:” enter “0.03*a”
- For “# of pts.”, enter “20”
- Change “Sys:” to “SYS9”
  - Can extract the angle
- Select the curve representing the crack front
- Click “Accept”
- **Note that not all locations were computed**
  - **This is common for curved cracks**

Computing SIF distributions on crack boundary

STRESSCHECK

- Why does this happen?
  - Inverse mapping failure
  - Most common for automeshed tetrahedra
- What is the solution?
  - Use the element edges instead
  - Change to “Select->Edge Curve”
  - Select an element edge on the crack front
  - Reduce “# of pts.” to “10”
  - Click “Accept”





Thanks!

support@esrd.com