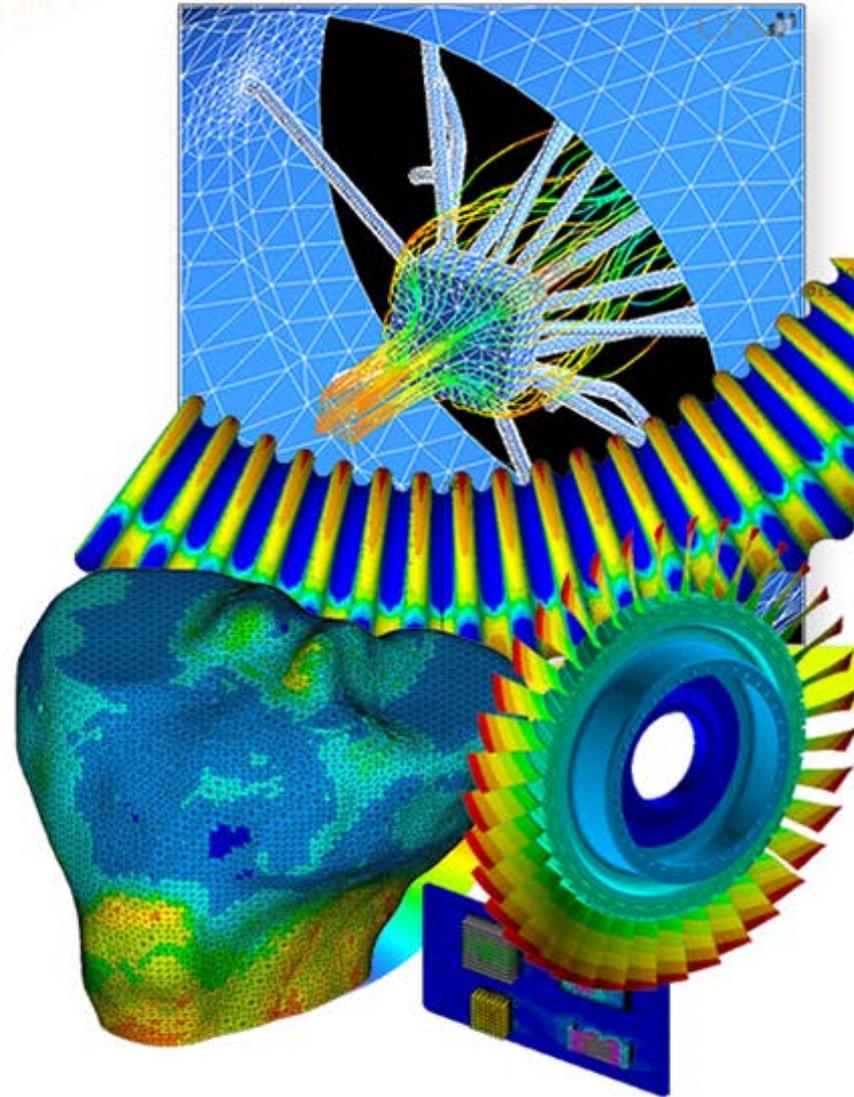
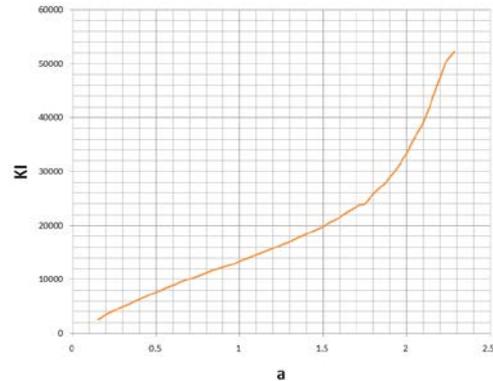


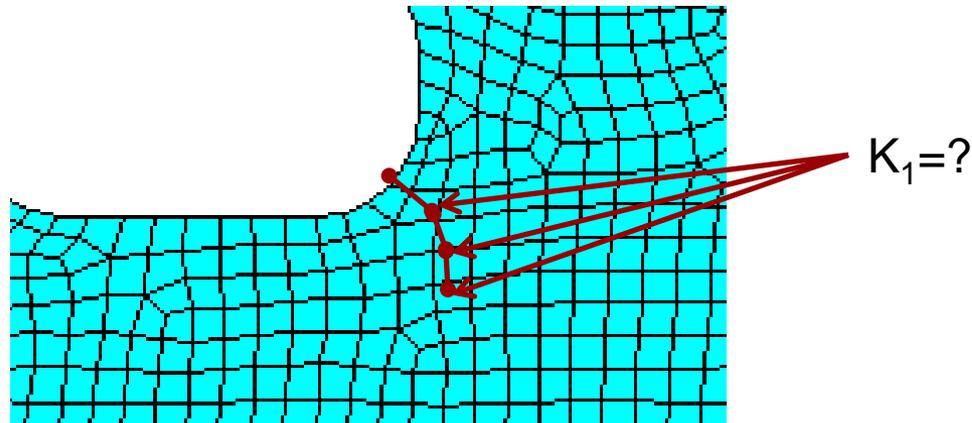
Automated Methodology For Modeling Crack Extension In Finite Element Models



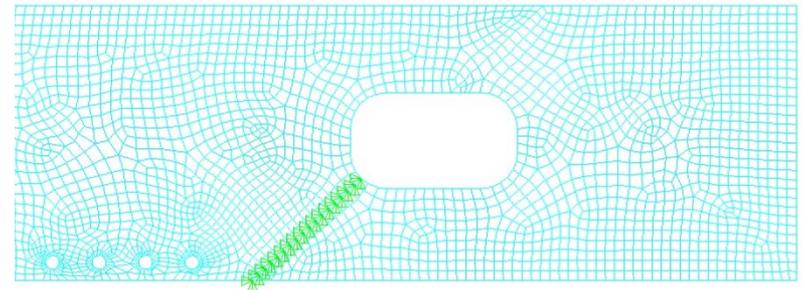
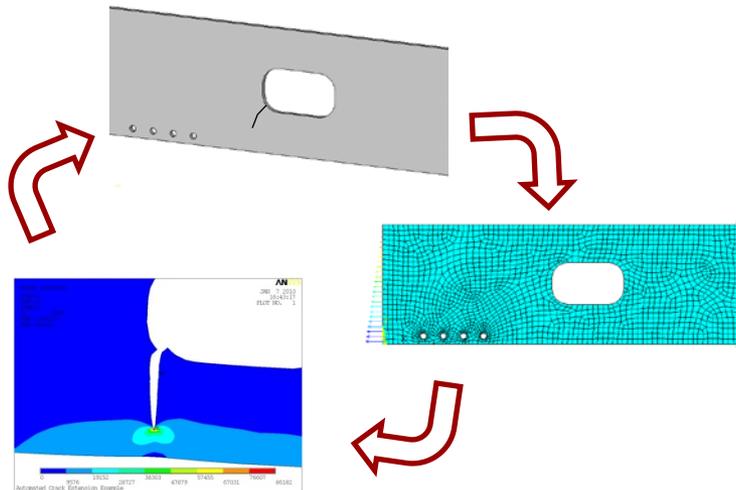
- As we have seen, one of the required inputs to a crack growth code like nCode is the stress intensity (K_I) at the crack tip as a function of the crack length (a).



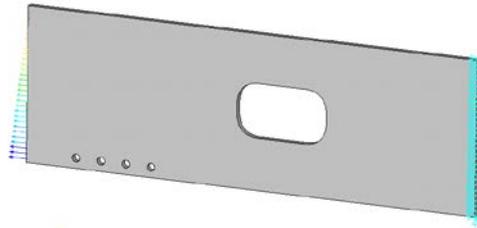
- To determine this, cracks of varying length must be analyzed in the finite element code.



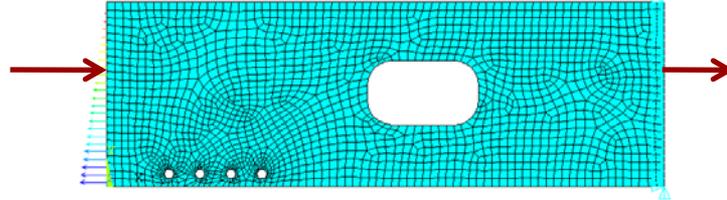
- There are 2 general methods of including a varying crack length in a finite element model.
 - Build/modify the CAD geometry to include the updated crack length. Re-mesh and re-apply loads for each increment of crack length.
 - Model the full crack length in the model. Initially nodes are coupled along the crack boundary, and the couples are deleted to extend the crack length.



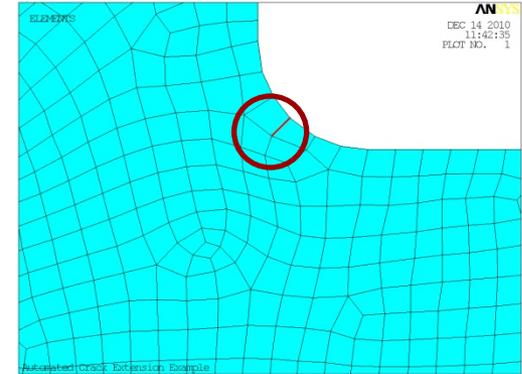
- Each of these existing methods has limitations.
 - Editing the CAD geometry and re-meshing and re-applying loads is a tedious manually intensive process.
 - Using couples requires pre-determination of the crack path.
- CAE Associates has developed an Automated Crack Extension (ACE) procedure for 2D finite element models by which an initial crack can be added and subsequently extended by increments automatically.
 - The procedure moves/morphs the nodes so that they lie along the predicted crack extension direction.
 - A duplicate copy of the nodes along the crack face are defined.
 - Elements on one side of the crack face are redefined to use new nodes.
 - Makes use of a combination of ANSYS[®] macros and Fortran routines in a custom compiled version of ANSYS[®].



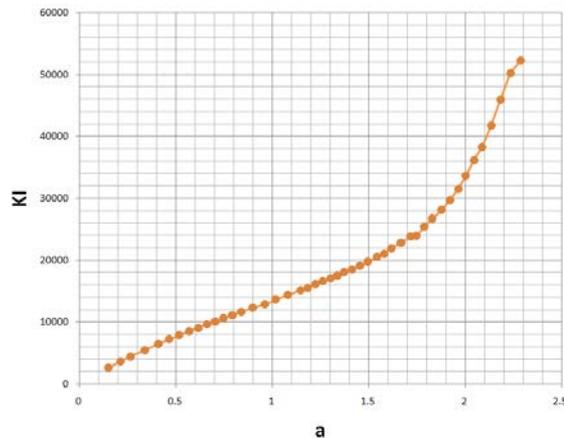
1. CAD geometry



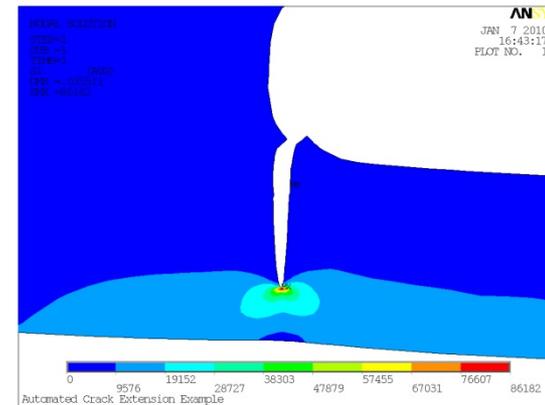
2. Baseline FE mesh



3. Automated initial crack insertion

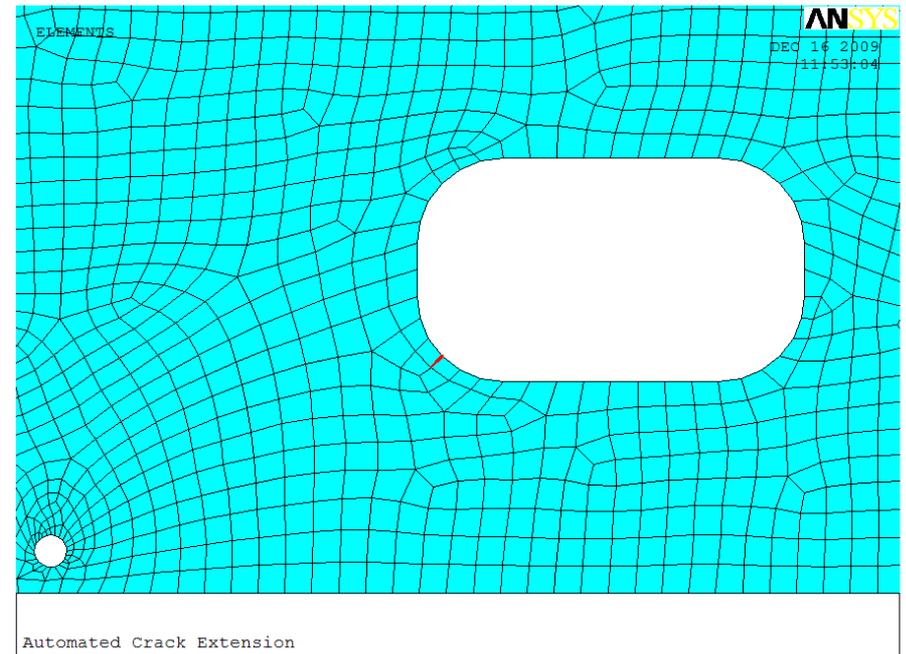
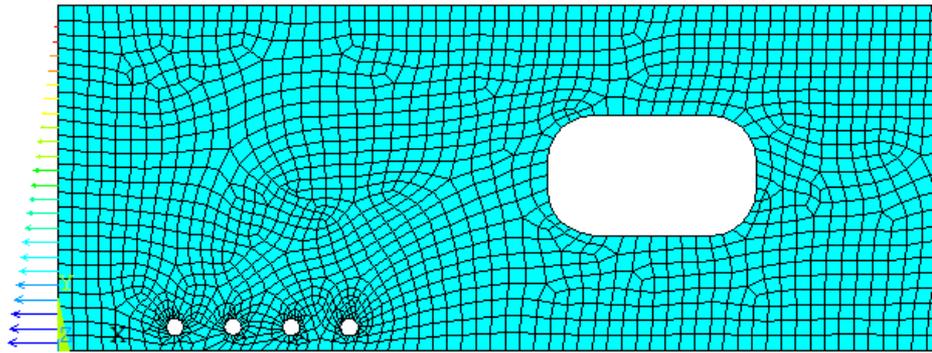


5. K_I vs. crack length



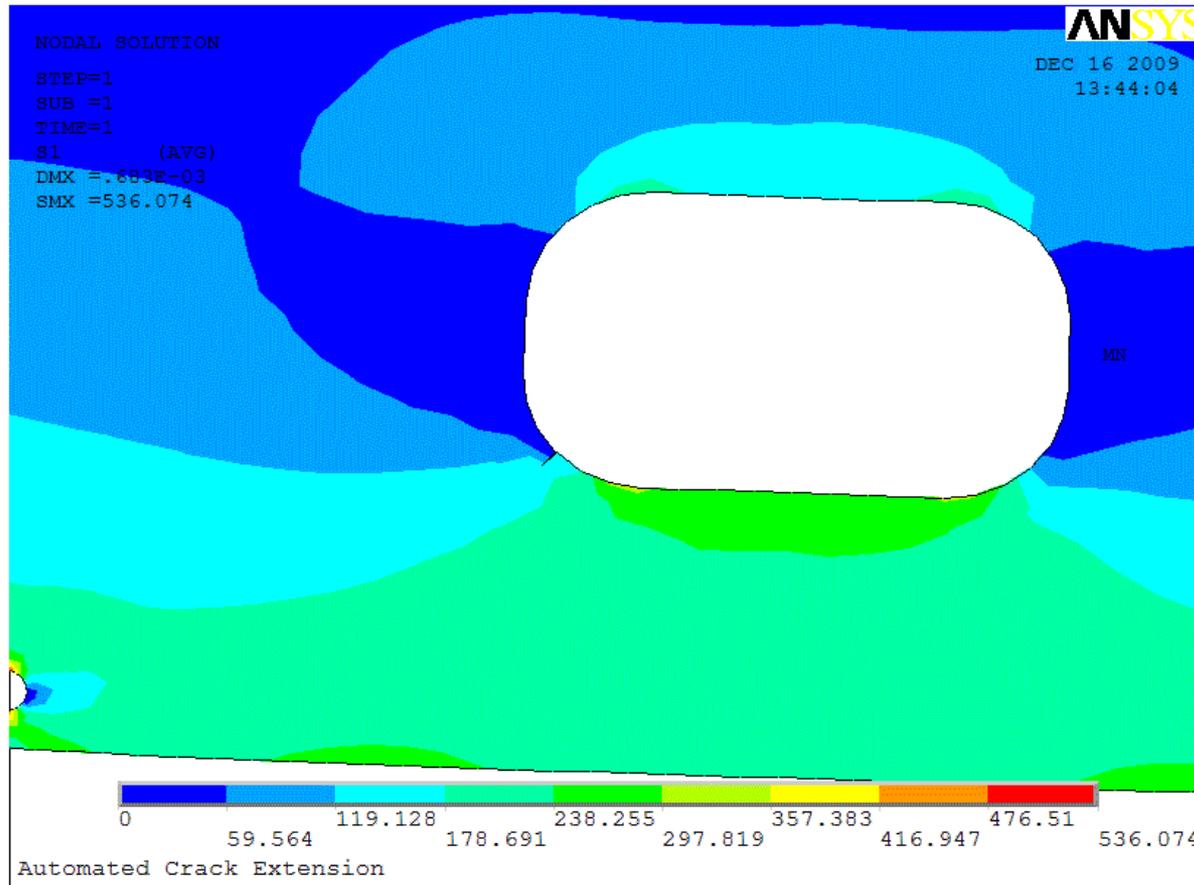
4. Automated analysis of crack extension

- Example mesh morphing to follow predicted crack path.
 - Crack direction based on principal stress just ahead of crack tip.
 - Quad elements split into triangles if necessary.



Automated Crack Extension Example

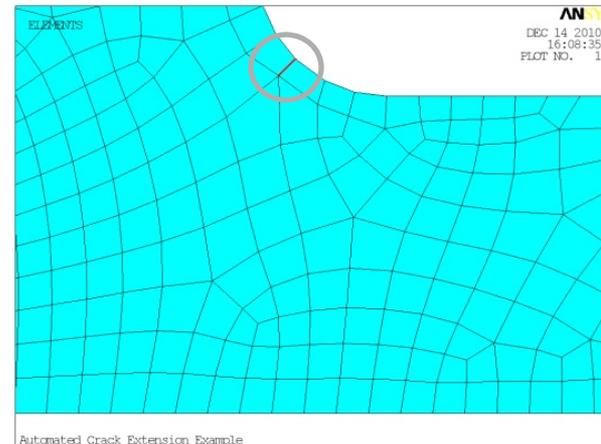
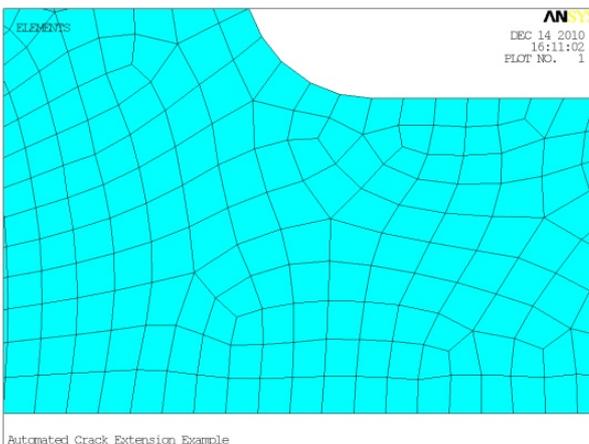
- First principal stress near crack



- ACE utilizes the ANSYS[®] general purpose finite element code to perform a series of stress analyses of a cracked structure. The ACE process consists of the following four steps:
 - Definition of the un-cracked structure and loading.
 - Initial crack definition.
 - Structural analysis & K_I calculation.
 - Crack extension & mesh morphing along the new crack front.

- Definition of the un-cracked structure and loading.
 - The model of structure is created in the normal manner using either a CAD file or an existing FE model. At this step, the structure has no crack and the FE mesh need not anticipate the location or path of the crack.
 - All loads, boundary conditions, material properties, solution settings are defined and the model is ready to solve.

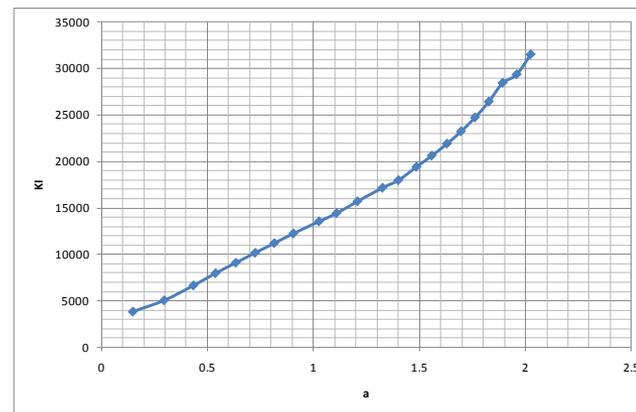
- Initial Crack Definition
 - Specify starting node and ending coordinates for initial crack.
 - ACE inserts the crack in the un-cracked FE mesh. A straight line is assumed between the start and end points of the crack.
 - The element connectivity is modified and new nodes are added on one side of the crack so that elements no longer share nodes on the crack faces. The coordinate locations of the crack face nodes are morphed so they lie directly along the initial crack.



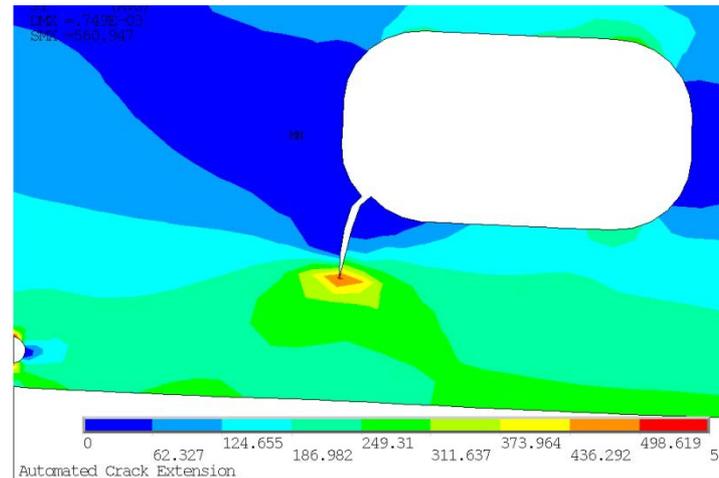
- The node location modification and element redefinition is performed using the user programmable feature (UPF) capabilities of ANSYS®. A user routine was developed to perform these modifications quickly, without requiring a full re-mesh of the geometry.
- Element numbers of the original mesh are maintained.
- New nodes are generated on the crack front
- Applied loads are automatically copied to new nodes.

- Structural analysis and K_I calculation
 - Structural analysis of the cracked body is performed to obtain the K_I versus crack length. This analysis includes the crack and will capture load redistributions due to its presence.
 - The ANSYS® CINT command is used to calculate the J-integral at the crack tip. The K_I is calculated from the J-integral and the element material modulus. The K_I and present crack length is saved to a text file for post processing.

```
0.15 3871.634
0.297 5078.741
0.435 6681.871
0.539 7997.716
0.635 9126.548
0.726 10190.8
0.816 11226.899
0.906 12264.405
1.027 13569.005
1.109 14434.93
1.208 15721.88
1.326 17178.539
1.401 18000.297
1.486 19429.474
1.558 20630.428
1.63 21917.352
1.696 23242.566
1.762 24737.903
1.827 26430.389
1.892 28448.729
1.958 29357.11
2.024 31506.865
```

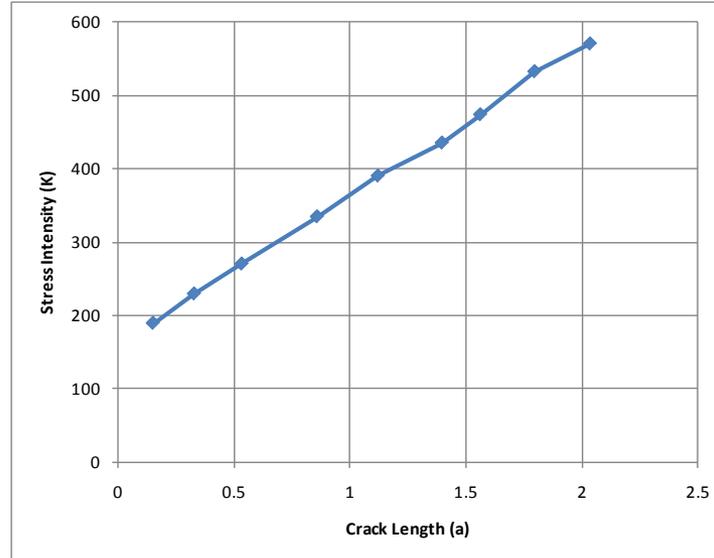


- Crack Extension and mesh morphing along the new crack front.
 - The cracked-body stress field is evaluated and the crack is advanced by a prescribed amount in a direction perpendicular to the cracked body maximum principal stress.



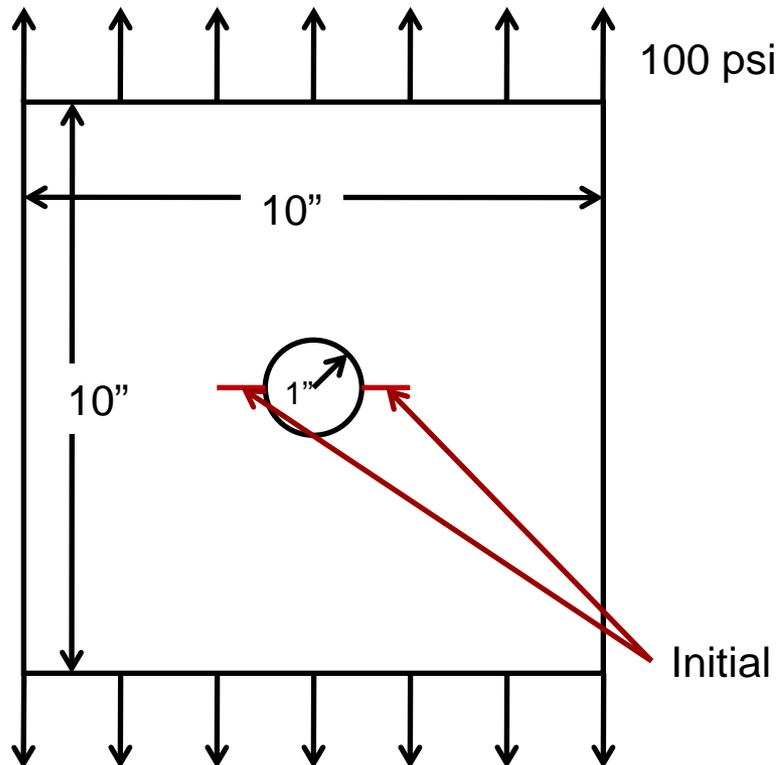
- As with the initial crack, the elements are separated along the crack boundary by generating duplicate nodes and redefining the element connectivity on one side of the crack. The nodes are moved so that they lie exactly on the new crack face.

- The routine checks the new element shapes. If an element will become too distorted by moving its nodes to the crack face, that element is automatically split into 2 triangles.
- The process is repeated during each increment of crack extension until the final defined crack length is reached.
- During each increment of crack advance, the K_I and crack length are appended to the text file. This file of crack length vs. K_I can then be used to calculate the number of cycles it takes to grow a crack using a crack growth code.



- User specifies:
 - Starting node number and end point coordinates for initial crack.
 - Final length of crack or number of increments (1 increment =1 element edge).
 - Radius for smoothing.
 - Number of contours for CINT command.

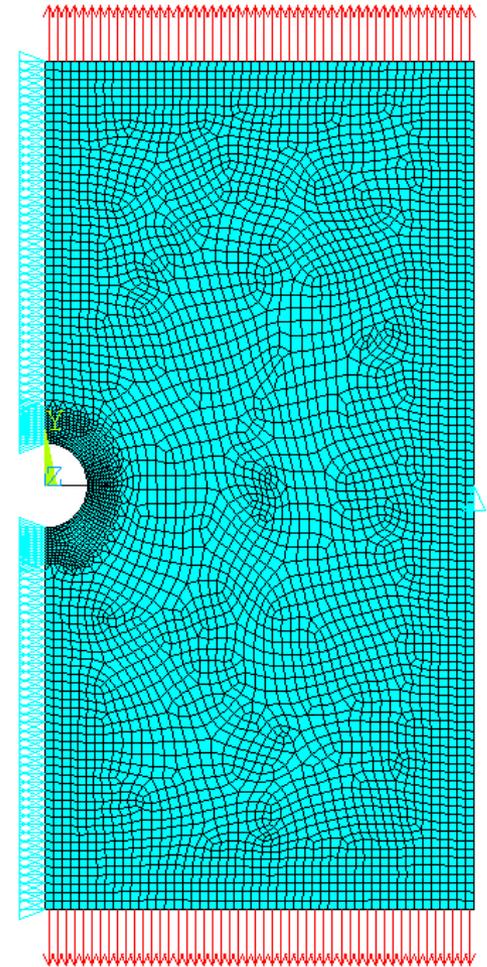
- The automated procedure was verified using a model of a plate with a hole under uniaxial tension, which has a known theoretical solution for K_I vs. a .



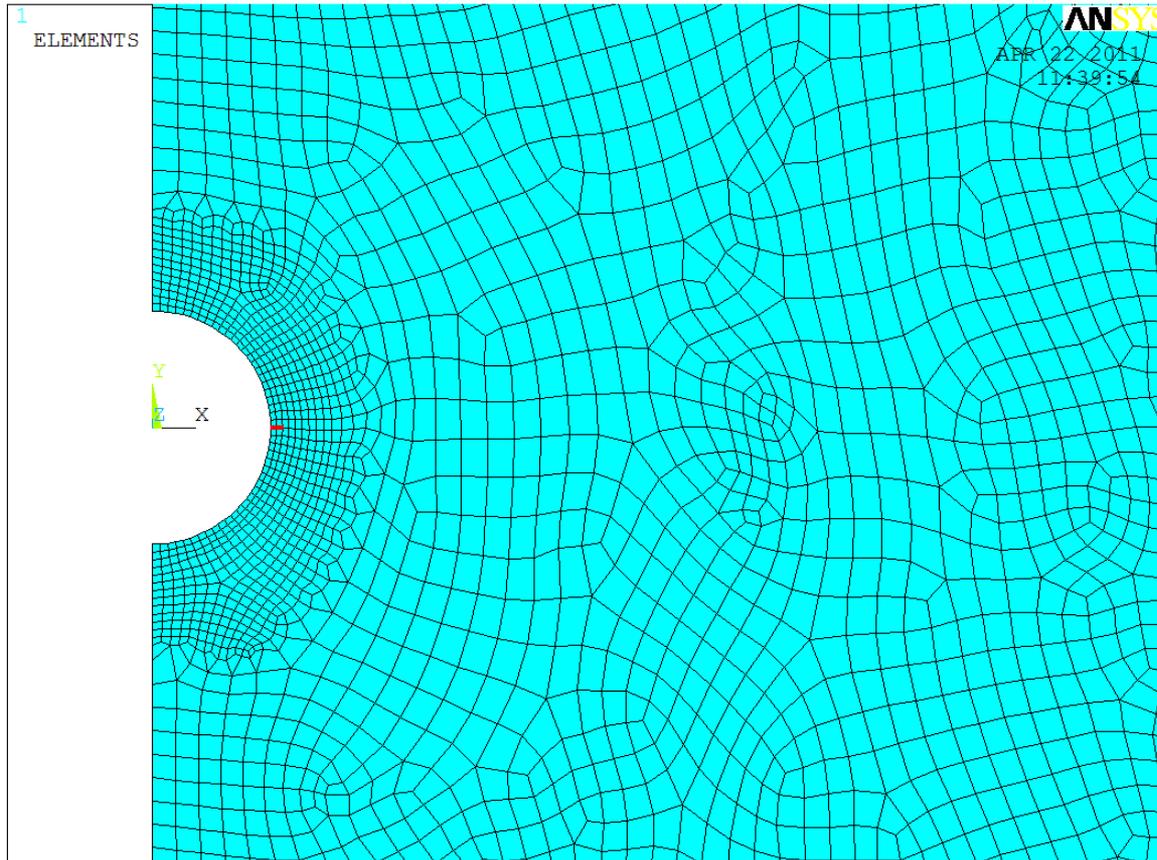
Young's Modulus = 30×10^6
Poisson's Ratio = 0.3

Initial crack locations (0.1" length)

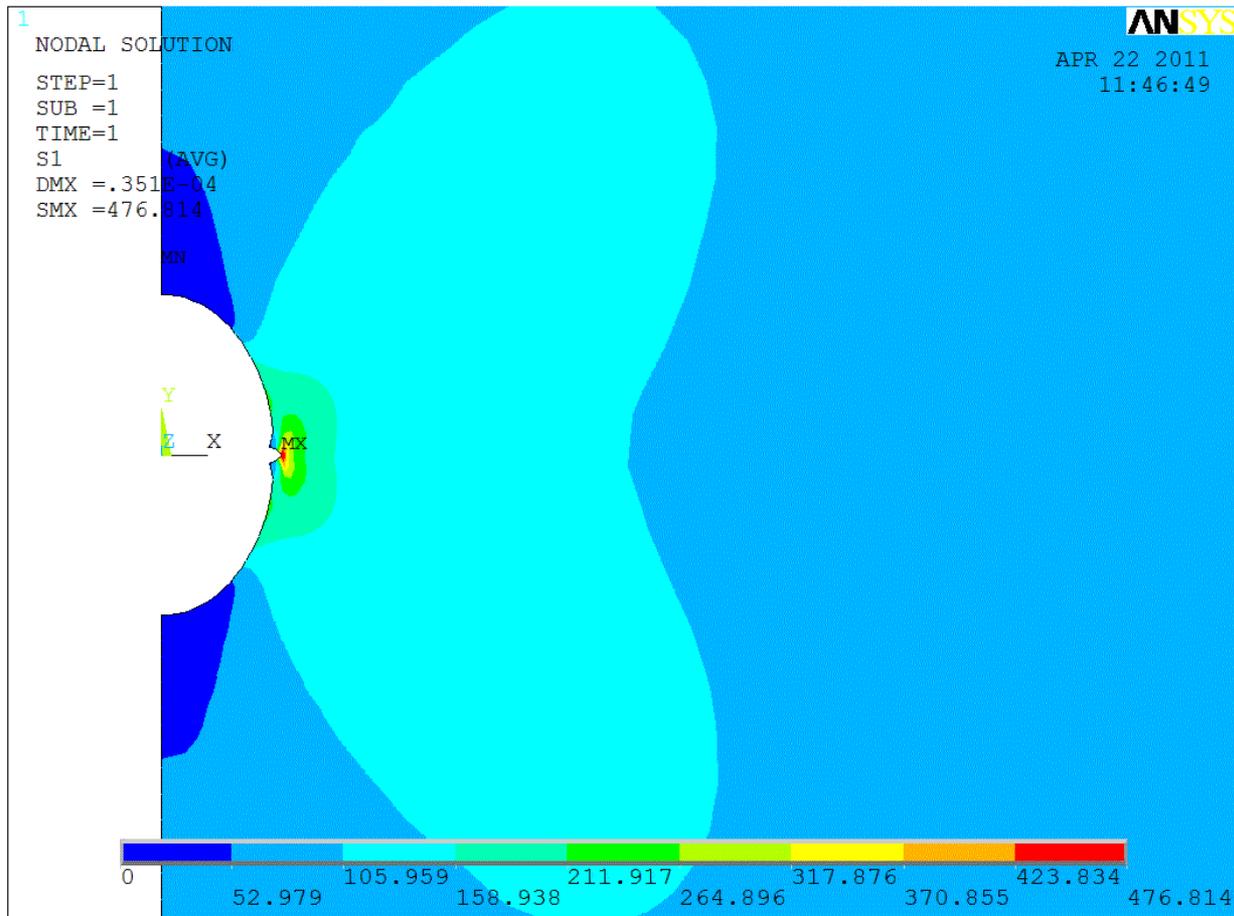
- A half symmetry model was built in ANSYS®. The loading was applied and mesh was generated.
- A refined mesh around the hole region was used. No special meshing was used to define crack path.



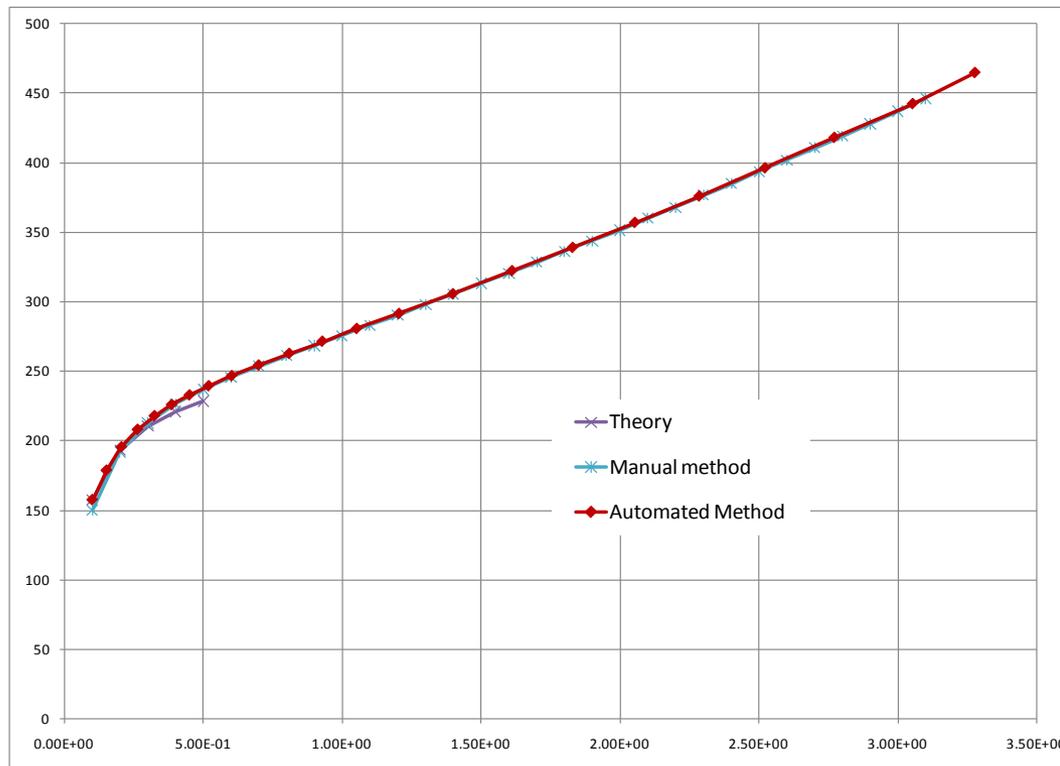
- Crack extension in verification model.



- First principal stress in verification model.



- The K_I vs. a data calculated from this analysis was compared to a theoretical value as well as values from a model generated and updated using manual modifications to the geometry and mesh.



- The use of the ACE method provides an automated technique for inserting and extending a crack in an existing FE mesh.
- This procedure makes significant improvements in time and accuracy over existing techniques for calculating crack growth.

- The ACE method has been developed for 2D finite element models with a single crack.
- Future enhancements to the ACE method would include extending the capability to 3D, where the crack boundary would be a surface, and the capability to have multiple cracks growing simultaneously.

- Additional enhancements would involve automatically passing the K_I vs. a data into a fracture life code such as nCode DesignLife(TM).