



3D Finite Element Software for Cracks

Version 3.2.034

User's Manual



1965 57th Court North, Suite 100

Boulder, CO 80301, USA

Office: +1-303 415-1475

FAX: +1-303 415-1847

<http://www.questintegrity.com/software-products/feacrack>

E-mail: support@questintegrity.com

Table of Contents

Overview	6
Crack Geometry Library.....	6
WARP3D Finite Element Program	6
System Requirements	7
Installation	7
Licensing	8
Getting Started.....	8
Navigation	9
Creating, Opening and Saving Files	10
Defining the Geometry	11
Configuration.....	11
Crack Location and Dimensions	12
Entering Dimensions	13
Navigate Menu	15
Select Menu.....	15
Custom Through-Wall Crack	15
Shift and Rotate the Crack.....	20
Pull-Down Menus.....	23
File Menu	23
Edit Menu	24
View Menu	24
Tools Menu.....	25
Program Options Control Panel.....	26
Help/About and Log File Location.....	26
Analysis Parameter Controls	26
ANSYS Command Syntax	30
ANSYS Contour Integral and CINT option	31
Abaqus Contour Integral Syntax Options.....	31
Load Step Functions	32
File Translator	35
Translator GUI	35
Command Line Program	36
Command Line Parameters	38
File of Input Names	38
Mesh File Format	38
Using the Log Files	41
FEA Options.....	41
Model Options Control Panel.....	44
Run From the Command Line.....	45
Command Line Option Labels	46
Command Line Options	47
Run with multi-threading	50
Weld and Heat Affected Zone (HAZ) Options	51
Boundary Conditions.....	53
Flat Plate 3-Point Bending.....	53
Analytic Rigid Surface	54
Load Steps	57
Load Increment Values.....	57

Internal Pressure Input.....	58
Crack Face Pressure	59
Tabular Crack Face Pressure	59
Omit Crack Face Pressure	62
Omit Pressure Thrust Load.....	62
Other Loads and Constraints	62
User Defined Load History	64
Characteristic Stress	65
Properly Constrained Models:.....	65
ANSYS Submodel Node File.....	67
ANSYS Submodel Analysis Global File.....	70
ANSYS Submodel Analysis Global Results Step	70
Tabular Temperature: Apply A Custom Thermal Gradient	70
Tabular Stress: Apply A Custom Stress Gradient	74
Material Properties	77
Material: Abaqus Stress-Strain Hardening Option.....	79
Material: User-Defined Abaqus UMAT.....	80
Material: Omit standard material syntax	83
Building and Viewing Models.....	83
Running Finite Element Analyses	85
Echo FEA Status File	86
Binary Packets File.....	87
Interaction Integral Results.....	87
Viewing Results	87
J-Integral Path Dependence Index	92
Elastic-Plastic Analysis	92
Special Considerations	92
J-Based FAD	93
Inner Plastic Zone Option.....	96
J-integral Contour Plotting Tool.....	101
Crack Mesh Controls.....	103
Number of Rings in Crack Tube	103
Refinement along the Crack Front	103
Crack Block Type.....	106
Crack Tube Diameter	106
Shallow Crack Cut-Off.....	107
Advanced Mesh Controls	108
General Mesh Controls.....	109
Element Type	109
Mesh Transition Option.....	109
General Mesh Refinement.....	111
Macro Text for ANSYS Solve	114
Macro Text for Warp3D and Abaqus	115
Subset Mesh	116
WARP3D Files For Analysis On A Remote Computer	121
Cylinder Subset Mesh Example	124
Steps for the subset mesh analysis:	125
<i>Abaqus</i> Subset Mesh	127
Cell Crack Mesh.....	131
Cell Crack Mesh Introduction	131
PRO Mesh Parameters.....	136

Advanced Refinement Form.....	138
Advanced Transitions Form	138
Advanced Crack Face Form	141
Advanced Right Side Form	143
Additional PRO Mesh Parameters.....	146
Vertical Mesh Lines Option	146
Inner Crack Tube Length Option.....	149
Inner Mesh Box Multiplier Option.....	149
Node Release Crack Growth	152
Compare J-integral Results.....	156
Cell Crack Mesh J-integral Results	158
Focused Crack Mesh J-integral Results	159
Compare Crack Mesh J-integral Results	161
Complex Crack Shape	162
User-Defined Geometry	165
Building A User-Defined Crack Mesh.....	165
Importing the Definition Mesh.....	166
Definition Mesh Corner Nodes Comment Data Block.....	167
Updating the Definition Mesh Data.....	168
Automatic Combining of Crack Mesh and Master Mesh.....	170
Step-By-Step Tasks To Combine Meshes	173
Fatigue Analysis Module.....	177
Fatigue: Preliminary Steps	177
Fatigue: Cyclic Loading	178
Fatigue Properties.....	181
Fatigue: Running the Analysis	182
Fatigue: K Smoothing Options.....	185
Fatigue: Life Prediction Methodology	187
Fatigue Crack Growth Equations	188
Fatigue Equations References	191
Fatigue Results	192
Fatigue Validation: Semi-Circular Crack.....	192
Fatigue Validation: Approximate Closed-Form Solution	192
Fatigue Validation: Paris Equation.....	194
Fatigue Validation: Comparison Example	195
Initial Stress Interpolation	199
Stress Interpolation During Fatigue.....	201
DLL for Fatigue with a Custom Crack.....	202
Cylinder Reverse Transformation Example	203
<i>Abaqus</i> Part and Assembly Introduction	204
Part and Assembly Syntax Summary	206
Radial Mesh Lines Option for PRO mesh.....	207
Mid Side Node Position	209
Gauss Integration Rule for Warp3D.....	212
Radial Point Results	212
Normalized Results	218
IDW Interpolation	221
A2 Crack Front Constraint	222
A2 Calculator	223
Q calculator	225
Results and Reference Solution for Q-stress	229

Alpha-H calculator	231
ASTM J Calculator: C(T) and SE(B)	234
Obtaining Load Line Result Values	235
Obtaining Crack Length Result Values	238
Compare J results	240
SE(B) Specimen Geometry	241
SE(B) Zero Size Load and Ram Pins	242
SE(B) Ram Load Pin force option.....	246
SE(B) PRO Mesh Options.....	246
SE(B) Side Groove	248
SE(B) Post Processing.....	250
C(T) Specimen	251
C(T) Specimen Side Groove	254
Modified Boundary Layer Model.....	257
BLM Constraints: K and T-stress.....	260
BLM Cell Mesh.....	262
BLM Cell Mesh Activation.....	264
BLM KT data file	265
Create a BLM cell mesh.....	266
BLM cell mesh log file information.....	267
Friction Stir Plug Weld Model	267
Sub-files Option	276
Warp3D Utility Commands.....	277
Warp3D Function Type “d” For Through-Thickness Cracks.....	277
Warp3D automatic element blocking.....	278
View A Femap Neutral File	279
Crack Face View Option	280
Crack face area option.....	283
Shift Node Coordinates	285
References	285
Example Files	285
Appendix:	286
Example Fortran Source Code.....	286
TABLE OF FIGURES	293

Overview

FEACrack[™] is a 3D finite element crack analysis program with automatic mesh generation. Both elastic K and elastic-plastic J analysis options are available, and a fatigue module provides the ability to perform crack growth analyses. The mesh generator can create *Abaqus*, *ANSYS*, and *WARP3D* input files as well as *PATRAN* and *FEMAP* neutral files.

You don't need to be a finite element analysis (FEA) specialist to use *FEACrack*. An intuitive Windows® interface guides you step by step through the process of building the model, running the analysis, and viewing the results.

Crack Geometry Library

Building a 3D finite element model that contains a crack is significantly more complicated than generating a mesh for an un-cracked component. *FEACrack* contains a pre-defined library of structural geometries and crack shapes. Simply enter the relevant dimensions, and the finite element mesh is created automatically. Unlike conventional 3D software mesh programs; there is no need to build a solid model of the structure. The user-defined geometry option provides the ability to insert a crack into an arbitrarily-shaped 6-sided solid.

The Version 3.0 release of *FEACrack* contains a greatly expanded geometry library relative to earlier versions. The currently available cases include:

- Surface, corner, through-wall, and buried cracks in plates.
- Surface, corner, and through-wall cracks in plates with holes.
- Surface, through-wall, and buried cracks in cylindrical shells.
- Surface and through-wall cracks in spherical shells
- Surface and through-wall crack in piping elbows.
- Weld toe and weld center surface cracks in L- and T-fillet welds.
- Weld toe cracks in lap welds.
- Weld toe surface cracks at pressure vessel nozzles and piping tees.
- Surface cracks at ring grooves in flanges and nozzles
- Corner cracks and circumferential surface cracks in spherical heads with a hole.
- Surface cracks in solid round bars.
- Single-V seam welds in the flat plate, cylinder, sphere, and elbow geometries.
- Surface cracks in arbitrary 6-sided solids using the user-defined geometry.
- Custom surface and through-wall crack shapes defined by a tabular input.

Other geometries and new features are in development.

WARP3D Finite Element Program

WARP3D is a research finite element code developed at the University of Illinois. It was developed specifically for 3D fracture mechanics analysis, and includes built-in J integral post-processing algorithms. This program has been extensively tested and benchmarked, and is

currently being used by a number of universities as well as NASA and Oak Ridge National Laboratory. More information and downloads are available at <http://www.warp3d.net>

By including *WARP3D* with *FEACrack*, we provide you with a stand-alone package for crack analysis. *FEACrack* also works seamlessly with the *Abaqus* and *ANSYS* commercial finite element programs. If you prefer a different finite element program, *FEACrack* can generate *PATRAN* and *FEMAP* neutral files, which can then be used to create input files for virtually any prominent finite element program.

System Requirements

FEACrack will obviously perform better on a computer with a fast processor, but the most important consideration is available memory. 3D crack meshes can contain 50,000 or more nodes, which require significant memory during the finite element analysis. *WARP3D* uses a sparse storage scheme, which results in an efficient use of memory, but the storage requirements are still substantial. *Abaqus* and *ANSYS* have similar large memory requirements.

If your PC has insufficient memory to run large problems, one option is to use *FEACrack* as a pre- and post-processor and run the finite element analysis on a UNIX workstation or mainframe computer. *WARP3D*, *Abaqus* and *ANSYS* are available in a wide range of software platforms, and the input file and results file syntax is platform independent. Although the *FEACrack* mesh generator runs only on the Microsoft Windows® platform, its system requirements are much more modest than for the finite element solver. For information on obtaining *WARP3D* for non-PC platforms, contact technical support at Quest Integrity Group USA, LLC; please refer to the telephone number and e-mail address on the title page.

If you plan to run 3D finite element analyses on your PC with *WARP3D*, the following minimum system specifications should enable you to run models of up to approximately 30,000 nodes:

- Windows 2000/XP/Vista/7 recommended.
- (Windows 95, 98, ME, or NT will run the software but are not recommended.)
- 1 GHz or faster processor recommended
- 512 MB RAM, 2 GB RAM recommended
- 500 MB free hard disk space, 2 GB recommended

When storage requirements exceed the available RAM, the *WARP3D* matrix solver can work out of core: the solver reads and writes data to and from temporary files on the hard disk. This is considerably faster than using virtual memory. Even with the out-of-core solver; however, the execution speed slows considerably when storage requirements exceed the available RAM.

For best results, your monitor resolution should be set to 1024 x 1280 pixels.

Installation

We recommend that you exit all Windows programs before installation.

Insert the CD into the drive. If “autorun” is enabled on your system, a setup menu will appear. Choose “Install *FEACrack*” to begin installation. If the setup menu does not appear, browse to

the CD and run “FEACrack_setup.exe”. Follow the prompts in the setup program to complete the installation.

By default, *FEACrack* will install to your system drive under the “Program Files\Quest Integrity Group\FEACrack” folder. You can specify a different location if you wish.

NOTE: if you are replacing *FEACrack* Version 2.x or earlier, do NOT install into the existing directory. We recommend that you uninstall the previous version after you install *FEACrack* version 3.0.

You may need to restart your computer in order to complete the installation process. The install program will tell you if this is necessary.

Once the program has been installed, copy the “client.lic” license file into the installation directory that you choose in step 3.

Licensing

FEACrack is offered as a single-seat and multi-user floating network license. If you have purchased the network licensing option, additional steps will be required to install and configure the network license server. Please refer to the instructions that were provided at the time of purchase or contact technical support (refer to the telephone number and e-mail address on the title page).

The *FEACrack* license agreement makes it easy to take advantage of future enhancements. The initial purchase price entitles you to 12 months of updates and technical support. After the first year, you can continue to receive updates and technical support for a nominal maintenance fee.

Getting Started

FEACrack is launched like any other Windows program. Simply select the *FEACrack* shortcut from the Start → All Programs → Quest Integrity Group menu. Optionally, a shortcut may have also been added to your desktop. This can be used to launch *FEACrack*.

When the program is first opened, the Home window appears, as shown below. On the home form, the unit system is selected from the “Units” pull-down menu. The crack mesh type is selected from the “Analysis Type” menu; choose between the standard focused crack mesh or a cell-type crack mesh. A brief name can be entered in the “Analysis Name” text box, and descriptive notes can be entered in the “Notes” text box. Both of these text boxes can be left blank. The right-arrow labeled “Continue” proceeds to the next form to begin entering the model data.

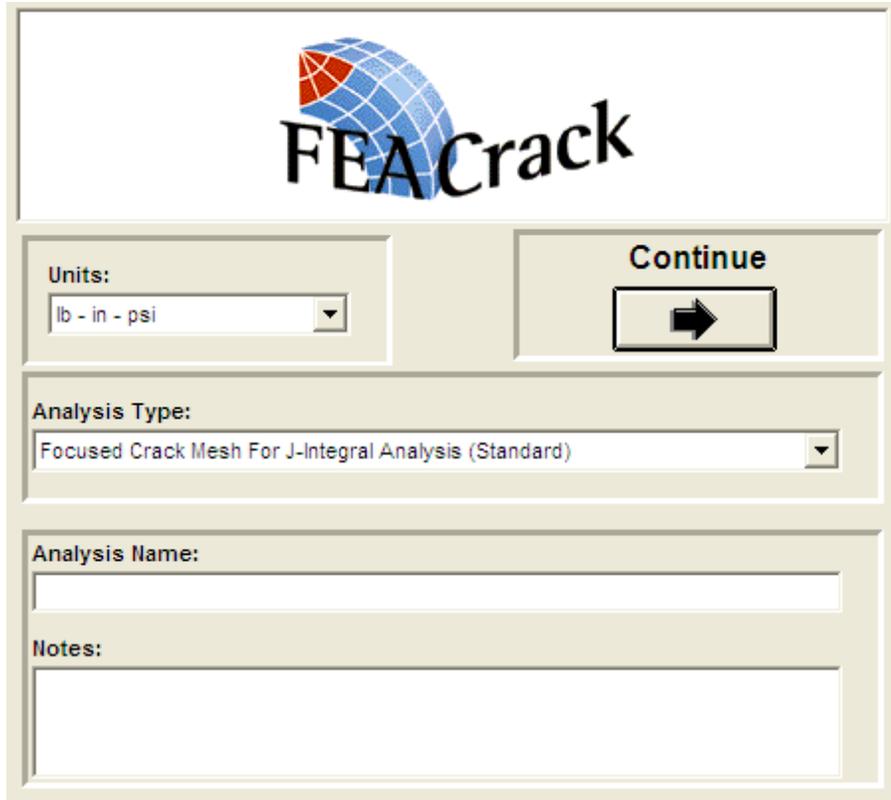
The image shows the FEACrack home form. At the top, there is a logo consisting of a blue and red mesh sphere next to the text "FEACrack". Below the logo, there are several input fields. The first is a "Units:" dropdown menu with "lb - in - psi" selected. To the right of this is a "Continue" button with a right-pointing arrow. Below the units field is an "Analysis Type:" dropdown menu with "Focused Crack Mesh For J-Integral Analysis (Standard)" selected. Below the analysis type field is an "Analysis Name:" text input field. At the bottom is a "Notes:" text area.

Figure 1 FEACrack home form; select the units and analysis type

Navigation

Forward and reverse arrows are provided on each input window to take the user to the next and previous step, respectively. The tool bar along the top of the main window also includes forward and reverse arrows that have the same functionality as the arrows on the input windows.

A navigation bar is provided along the left side of the main window, as illustrated below. This bar contains several icons. Clicking one of these icons will perform an operation (for example, build the mesh) or take the user to the desired place in the program (for example, open the geometry form or open the material properties form). In some cases, a sub-navigation bar will appear next to the main bar. For example, when “Geometry” is selected, as it is below, a sub-navigation bar with several buttons appears to the right of the main bar.

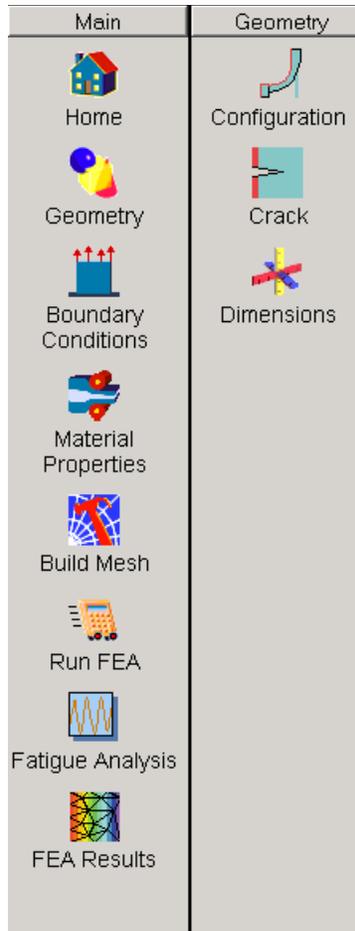


Figure 2 navigation bar

Creating, Opening and Saving Files

FEACrack interacts with three types of files:

User Input Model Data Files. These are small text files that contain the basic model data that the user inputs such as geometry, dimensions, boundary conditions, and material properties. These files have an *.elt* extension. Results of a fatigue analysis, if applicable, are also stored in the **.elt* file. Version 1.x input data files, which have a *.fec* extension, can be imported by selecting “File → Open”.

Finite Element Input Files. These files are created by the mesh generator and include all of the element and nodal data, as well as boundary conditions and material properties. Input files can be created in *Abaqus*, *ANSYS*, and *WARP3D* format. In addition, mesh data can be saved in *PATRAN* or *FEMAP* neutral file format. To specify the type(s) of files to be created, select “Program Options/Analysis Options” from the “Tools” menu. Select the file type and output using the check box options on that form.

Finite Element Results Files. When a finite element analysis is performed, a results file is created by the finite element analysis program. *FEACrack* can then read these files so that the user can

view the results as color maps on the mesh picture and x-y plots of J and K results. Currently, only *Abaqus*, *ANSYS* and *WARP3D* results file formats are supported.

Note that when the user selects “Save” or “Save As” from the File menu, only the *.*elt* files are saved. These files are opened by selecting “File → Open”. The corresponding mesh file will be open if one has been created. The finite element results files are opened by clicking on “FEA Results” in the main navigation bar.

The disk and folder icons in the tool bar along the top of the main window have the same functionality as the “File → Save” and “File → Open” commands, respectively.

Defining the Geometry

Click on the forward arrow in the “Home” window or the “Geometry” icon in the navigation bar to display a series of input windows for specifying geometry and dimensions of the structure of interest.

Configuration

At the “Configuration” window, select the overall type of component (for example, flat plate, fillet weld, nozzle on shell, etc.).

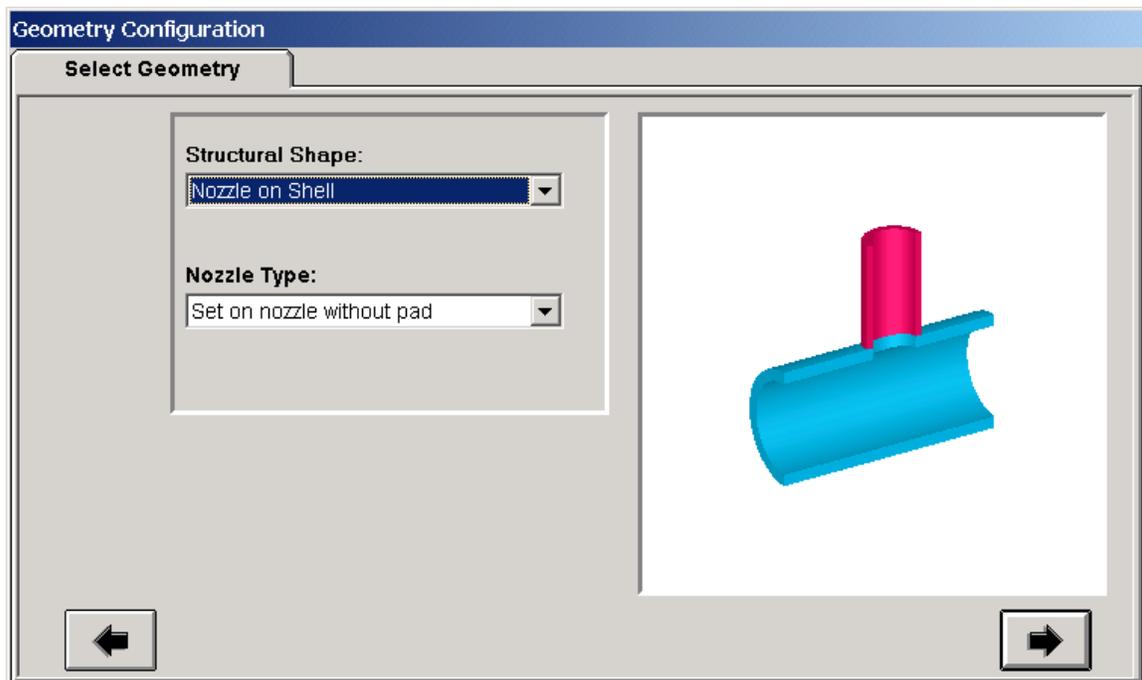


Figure 3 Geometry form, select the geometry model

Crack Location and Dimensions

For each geometry in the library, there are one or more pre-defined locations and orientations for the crack. For the nozzle/shell example shown below, the only location currently supported is at the toe of the nozzle weld. The crack may either be oriented as shown below, where there is a symmetry plane at $x = 0$, or it can be centered about the 12 o'clock position of the shell, such that there is a symmetry plane at $z = 0$.

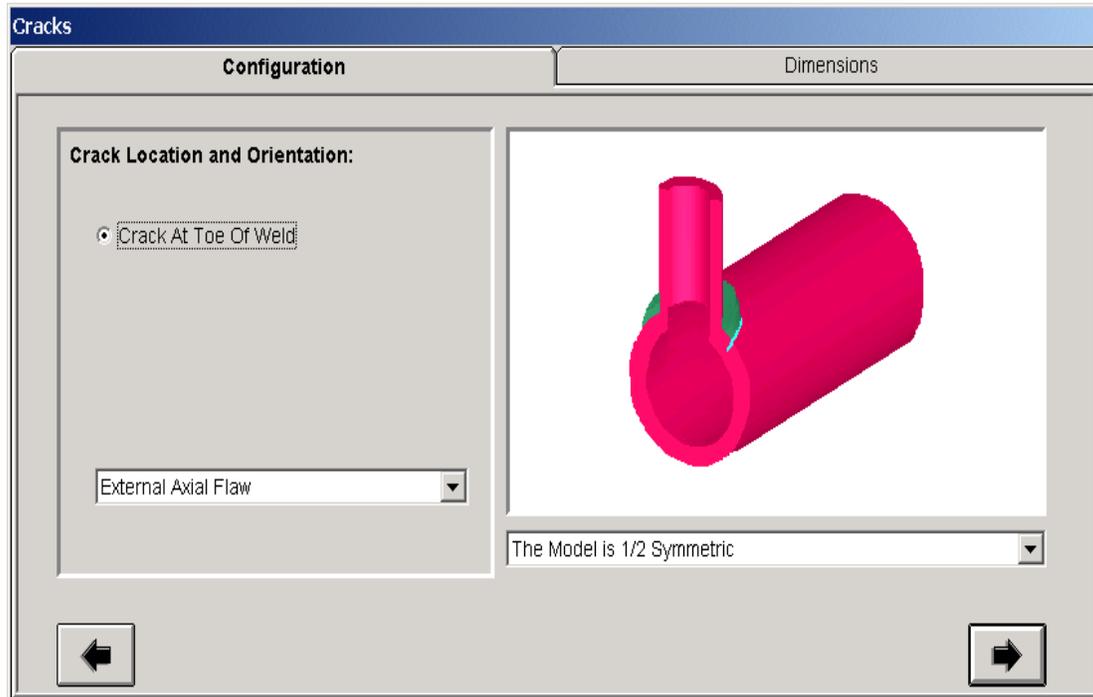


Figure 4 Configuration form to select the crack location and model symmetry

In some cases, more than one symmetry option is available. For example, it is possible to create a full-symmetric model of a surface crack in a flat plate. In this particular instance, the crack orientation and location in the plate is arbitrary. Beveled cracks (i.e., not perpendicular to the free surface) are available in several other configurations.

The available crack shapes varies with component geometry. When entering crack dimensions, the user may either define a single crack size, or he/she may specify a series of crack dimensions as illustrated below.

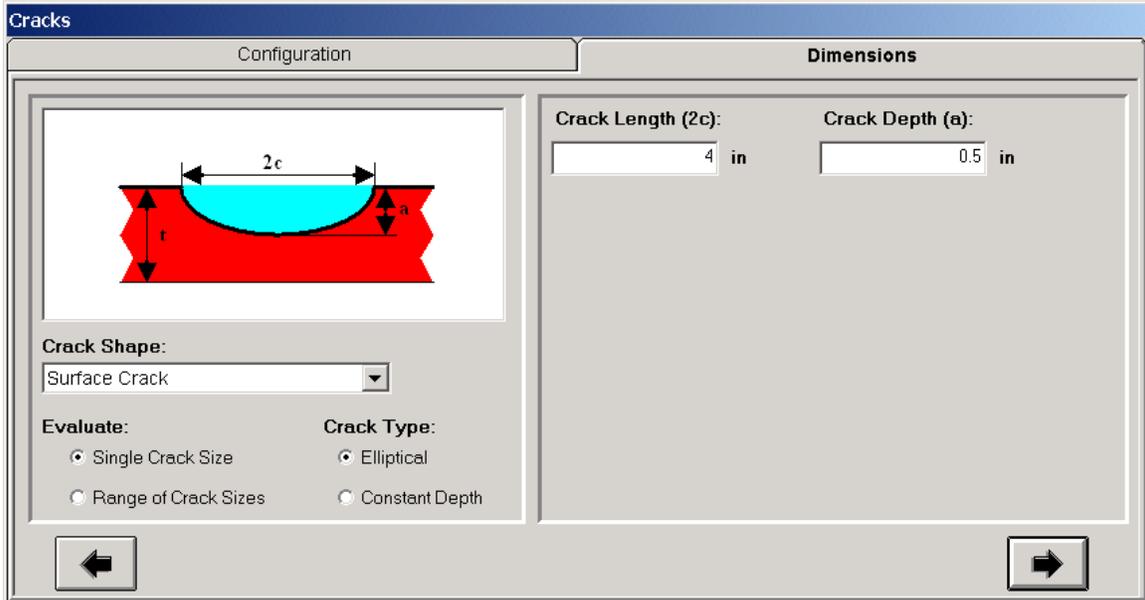


Figure 5 enter the crack dimensions

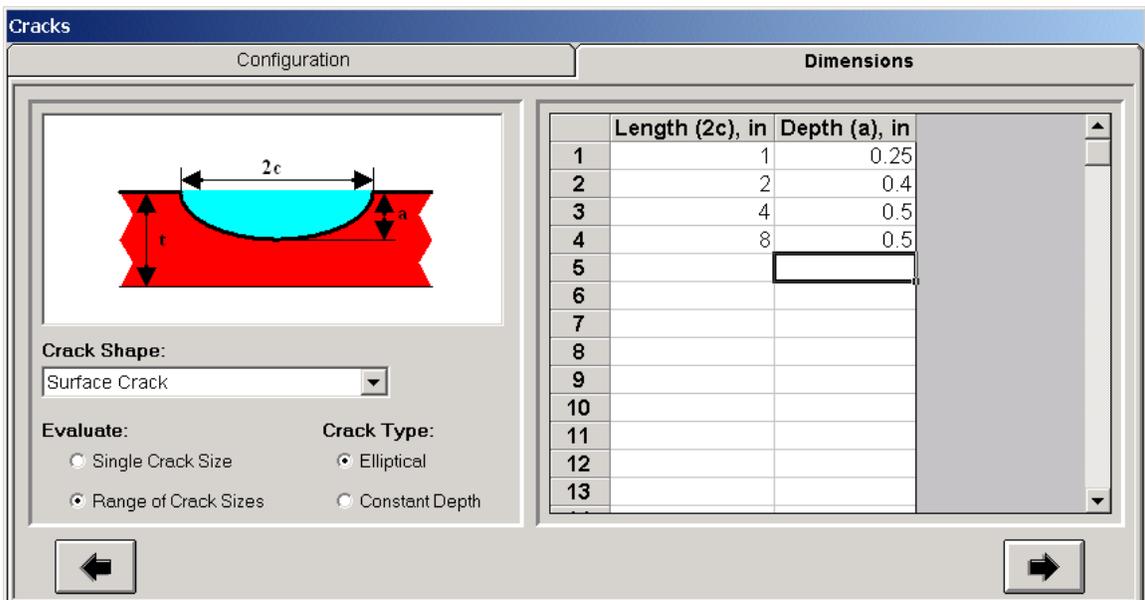


Figure 6 enter a list of crack sizes to generate multiple cracks

Entering Dimensions

Click on the “Dimensions” icon or the forward arrow in the crack dimensions window. Enter the appropriate component dimensions as prompted. Note that the required dimensions include many weld profiles.

In the nozzle/shell example shown below, there are dimension inputs for both the positive and negative x portions of the shell, which correspond to the right and left side of the model,

respectively. Depending on the crack location, there may be a symmetry plane at $x = 0$ (defined at the center of the nozzle); in which case dimensions for the negative x side will be ignored.

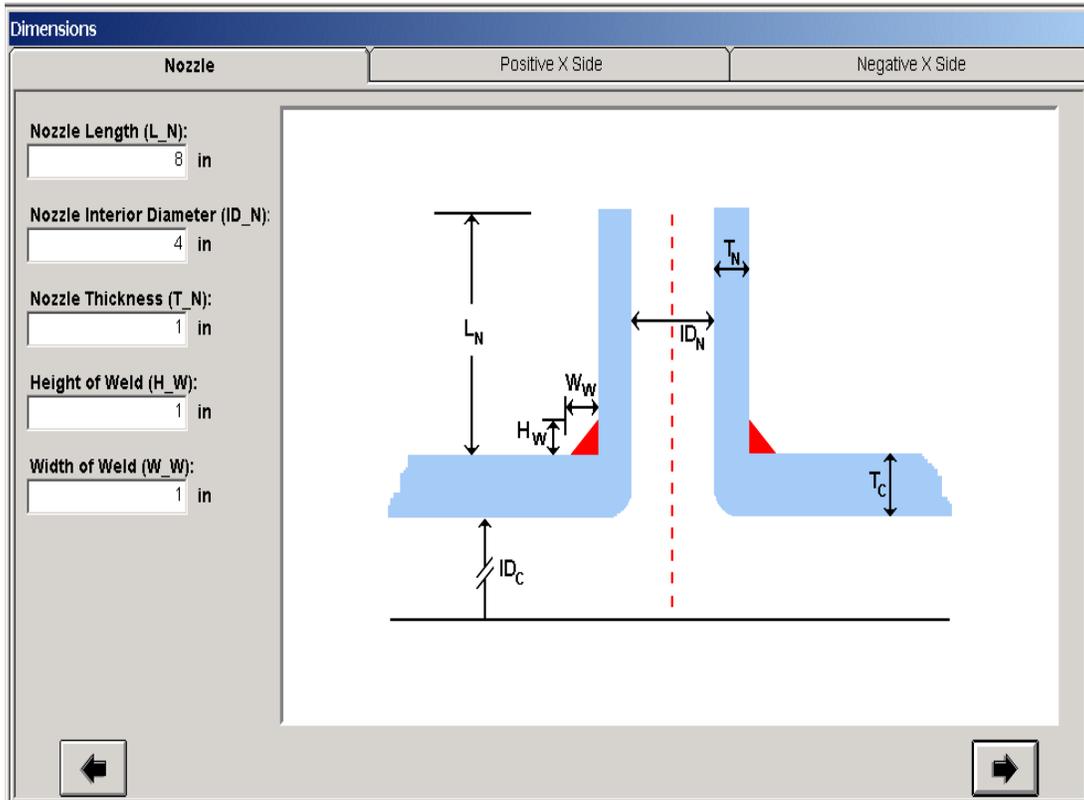


Figure 7 enter the model dimensions

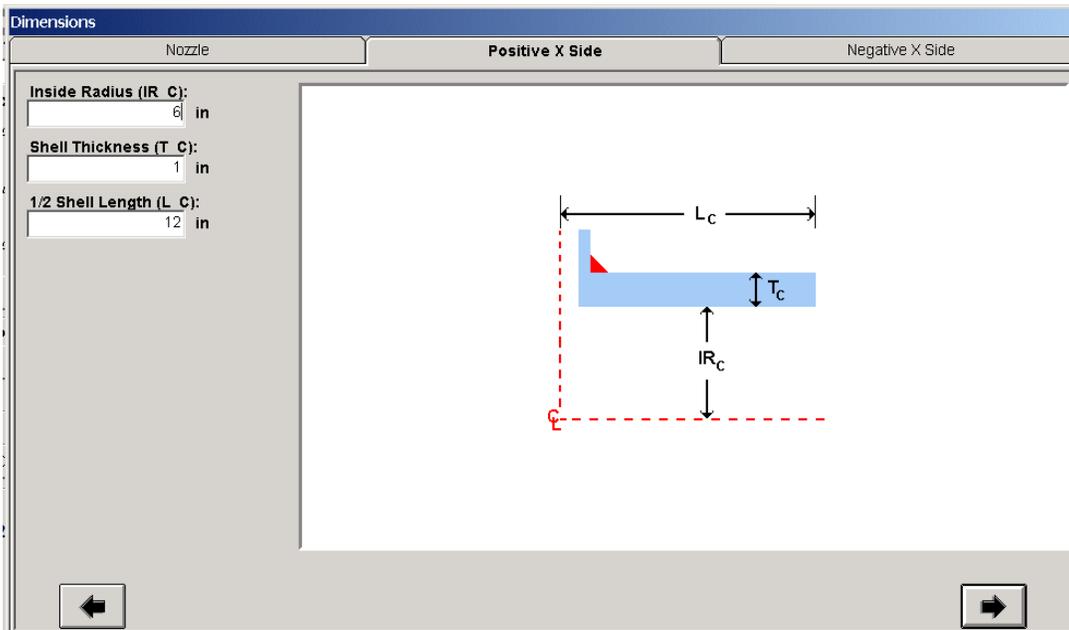


Figure 8 enter additional model dimensions

Navigate Menu

The navigate menu has the same functionality as the navigation bar, described above.

Select Menu

The Select menu includes commands to Select Nodes and Select Elements. The mouse is dragged over the model to select the desired nodes or elements. Either a rectangular or circular selection area can be specified. Once a group of nodes (or elements) is selected, the node ID numbers, coordinates, and results can be displayed in both graphical and tabular form.

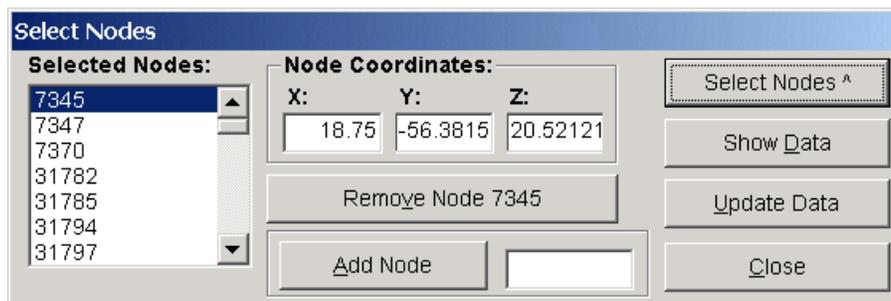


Figure 9 select nodes to get nodal result values in the data table

Custom Through-Wall Crack

A general through-thickness crack front profile is available in *FEACrack* by using the “User Defined crack length table”, shown in the figure below. Pairs of depth and length values at several points along the crack front are given for the desired through crack shape. For example, in the ASTM E 1820 "Standard Test Method for Measurement of Fracture Toughness", nine equally spaced points through the thickness are used to measure the crack length values. Currently an odd number of points are needed for the mesh transformation. Any number of odd points can be used to define the crack front profile, and the points can be arbitrarily spaced. The general through crack profile is available for both the focused and PRO crack meshes.

The first depth position is expected to be at $a_1 = 0.0$ at the top of the through-thickness crack, and the last depth position is expected to be equal to the thickness at the bottom of the crack, $a_N = T$. In the figure below there are $N=9$ values in the table, and the depth position increases from 0 (at the top of the crack) in the first row to 1 (the thickness) in the last row. For a thumbnail shape crack the length position values are symmetric about the middle data row. For example, in the figure below row 3 and row 7 have the same length position value.

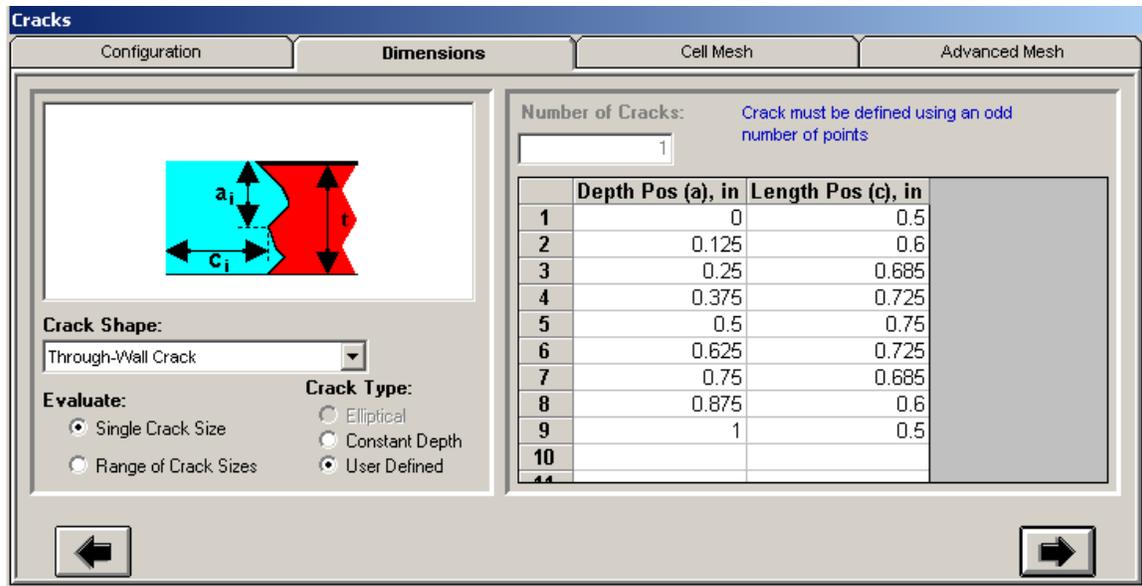


Figure 10 General through-thickness crack profile table

Several examples of the general through-thickness crack front profile are shown in the figures below. A thumbnail shape through crack is just one example of the general crack front profile.

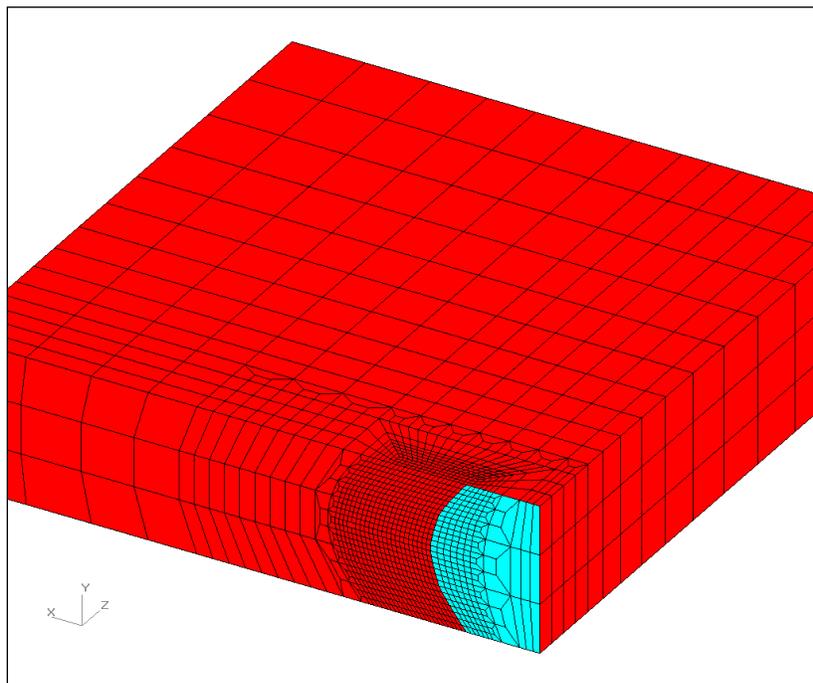


Figure 11 A custom thumbnail shape through crack

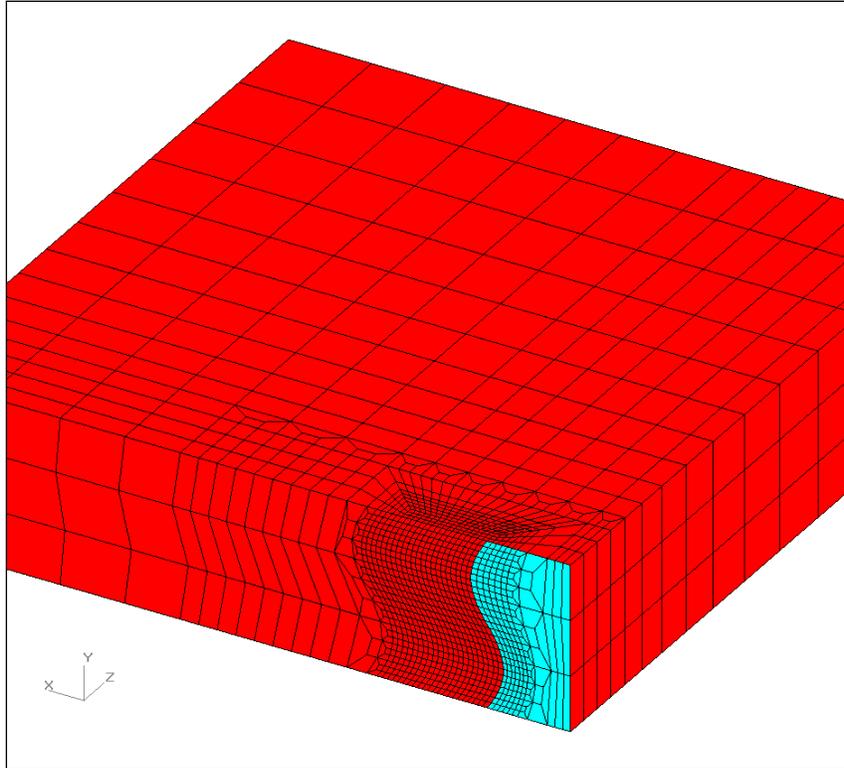


Figure 12 An S-shape through crack based on the fu 1

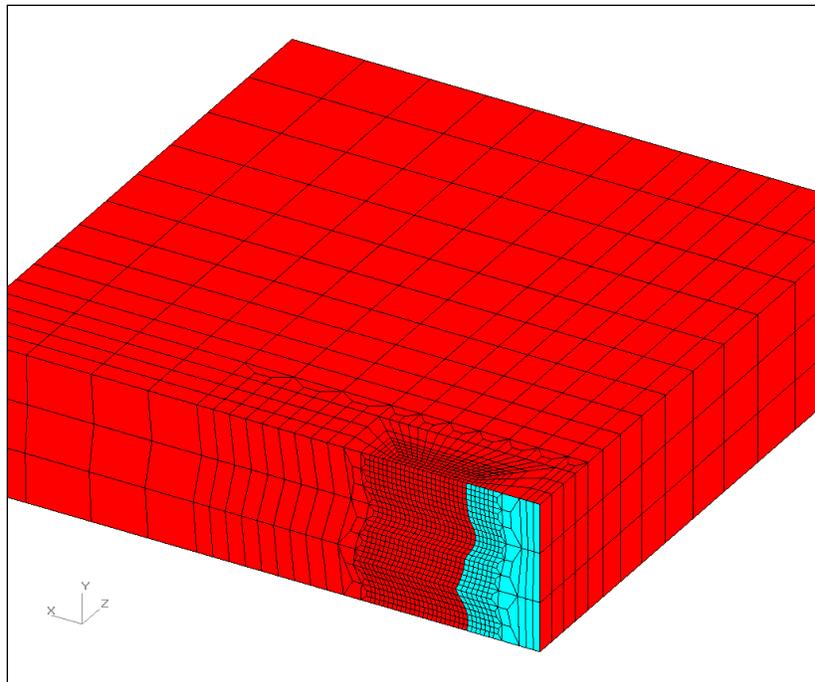


Figure 13 A jagged shape front through crack. 1

A flat plate geometry is used as an example for the user defined crack length table. Using an increasing thumbnail depth for the through crack, J-integral results are compared in the plot

below for the cell meshes using the remote domain and focused crack meshes using the crack front contours. The crack front position and depth values were computed in an *EXCEL* spreadsheet and copied to the table in *FEACrack*. The thumbnail through crack is in a flat plate geometry with an imposed membrane displacement in the Z-direction to give a uniform membrane tension stress. The analysis is elastic with a single step. The J-integral results from the cell and focused crack meshes generally compare very well along the crack front. As the thumbnail becomes deeper (more curved), the J-integral value increases at the top and bottom free surfaces compared to the through-thickness J values. The cell crack meshes have a very abrupt drop in the J value at the top and bottom crack front node; the J value is even negative for the deeper thumbnail cases. When the J value at the top and bottom crack node drops sharply, the contour path dependence is very high; the other crack nodes have low contour path dependence. The focused crack meshes do not have the abrupt drop in the J value at the top and bottom crack nodes.

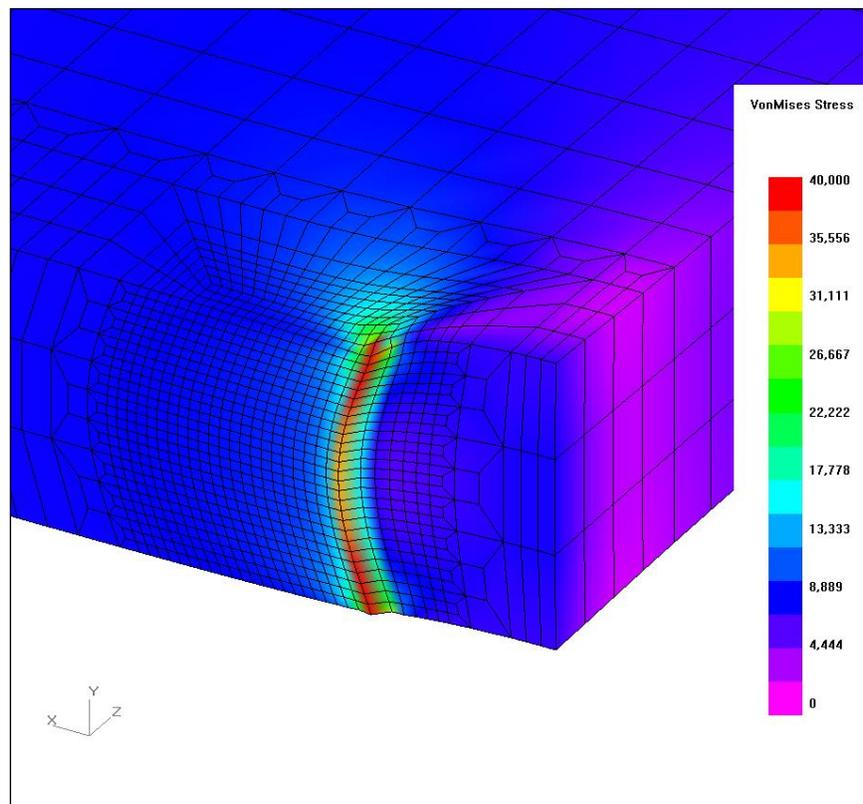


Figure 14 Von Mises stress results for a cell through crack mesh with a thumbnail shape.

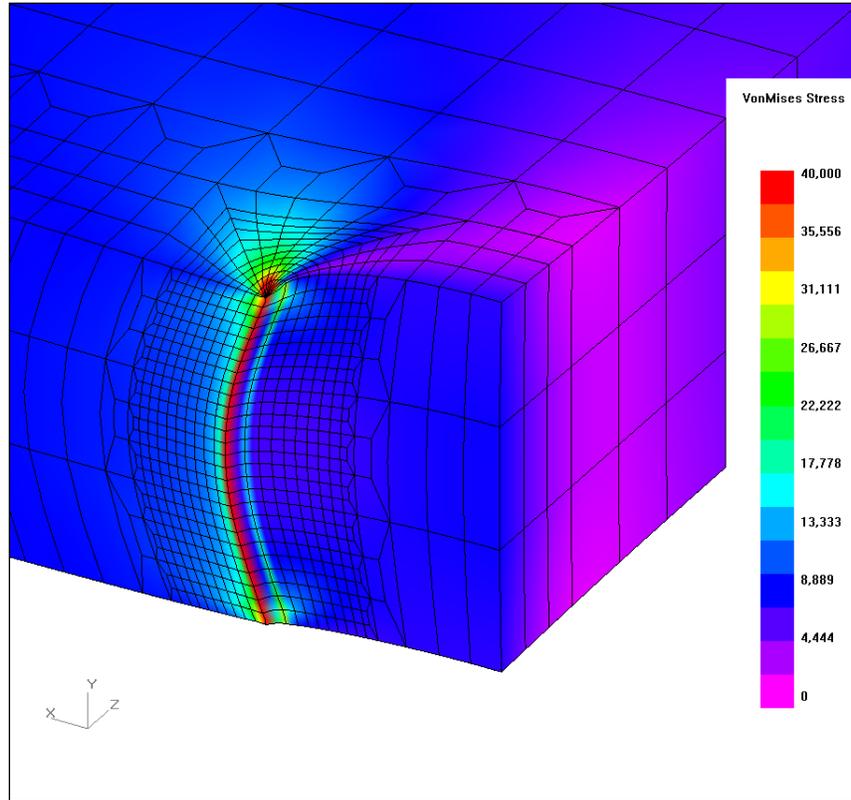


Figure 15 Von Mises stress results for a focused through crack mesh with a thumbnail shape.

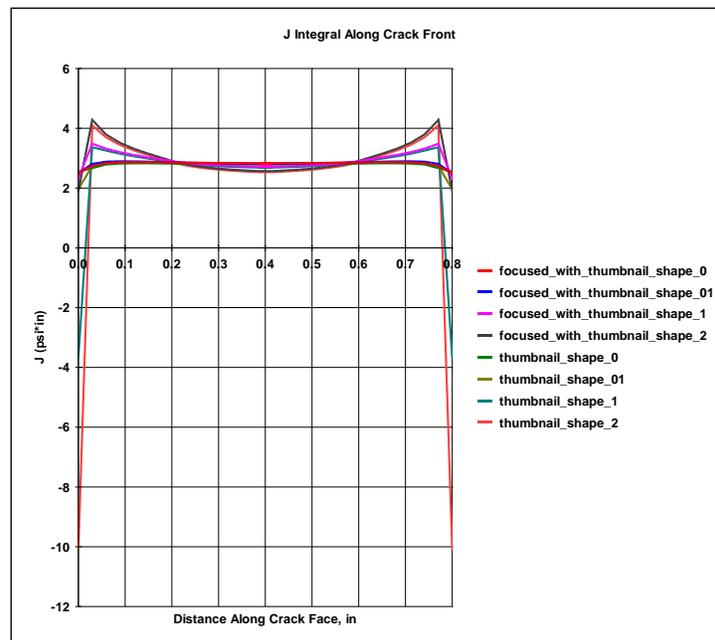


Figure 16 J-integral results for the thumbnail shape through crack. The focused crack meshes use the crack front contours; the cell crack meshes use the remote domain.

Shift and Rotate the Crack

To shift the crack location from the center of the geometry, select the User Controlled Location on the pull-down menu on the “Geometry/Cracks/Configuration form”, see the figure below. Enter the distance values to shift the crack location relative to the front left corner. For a user-defined geometry, the front left reference corner is set in the “Dimensions form” where the crack location is chosen using the definition mesh corner nodes (see below for more information about the user-defined geometry and definition mesh). By default, the “Center Of The User-Defined Shape” locates the crack in the center of the mesh region. Depending on the model symmetry, one or both crack shift directions may be available. For a quarter symmetric model the crack cannot be shifted as it is located on the symmetry planes.

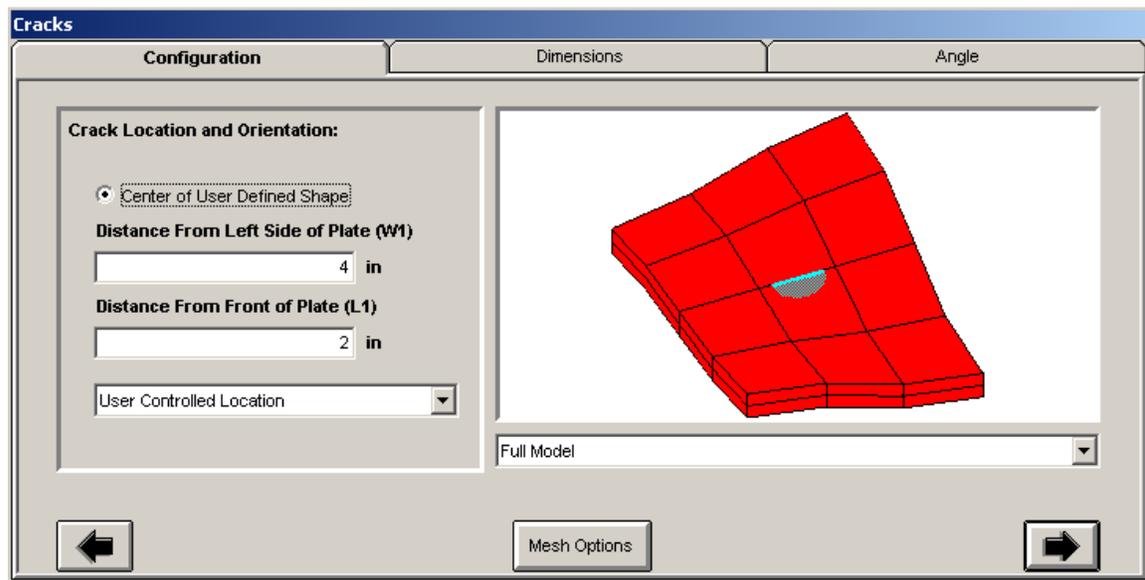


Figure 17 enter the crack shift distances relative to the model

The orientation relative to the mesh edges can be set in the Angle tab, shown in the figure below. Enter a positive or negative Theta Y value to rotate the crack orientation in the mesh surface. Theta_Y can usually be varied from -50° to $+50^\circ$ depending on the geometry, see the figure below. The crack bevel angle changes the crack plane relative to the mesh surface normal. Set Theta_B to zero for the crack plane to be perpendicular to the mesh surface. Theta_B is positive about the crack length direction (the x-axis for a flat plate).

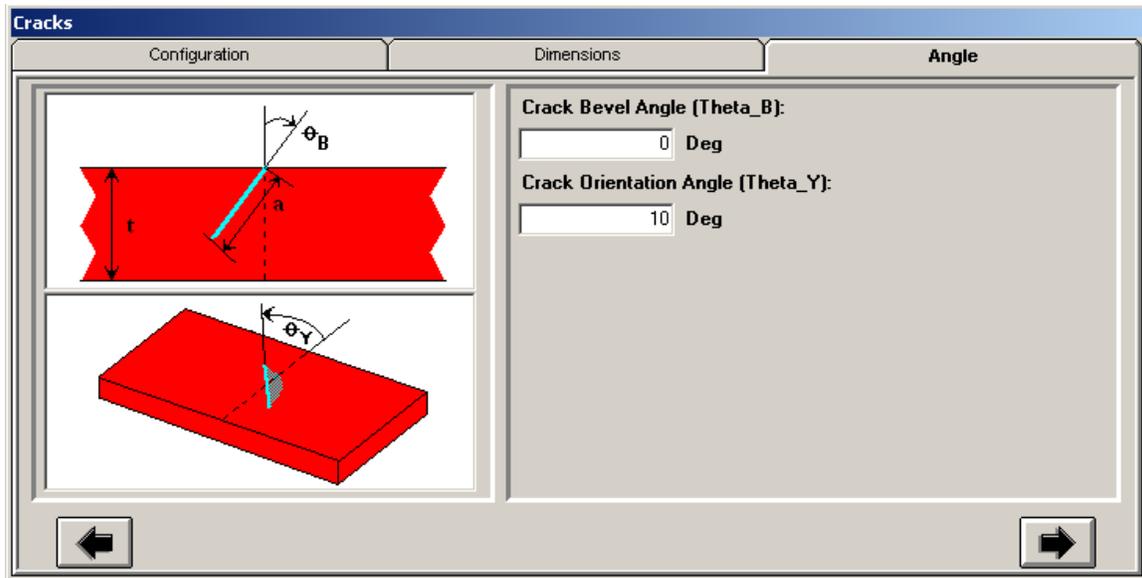


Figure 18 enter the crack bevel and orientation angles to rotate the crack relative to the model

For a user-defined geometry, the crack is located with the Crack Location pull-down menu to select the set of definition mesh corner nodes for the crack opening surface. Then orient the crack relative to the mesh edges using the Crack Orientation pull-down menu. For example, the crack orientation may be oriented parallel or transverse to the length dimension of the definition mesh. The selected crack location and orientation gives the reference position if the crack is also shifted or rotated as described above. The reference front left corner of the mesh is the corner 1 node "C1" in the corner nodes list for the definition mesh at the bottom of the "Dim and Pic form". In the figure below the reference corner is node 3851 in the definition mesh.

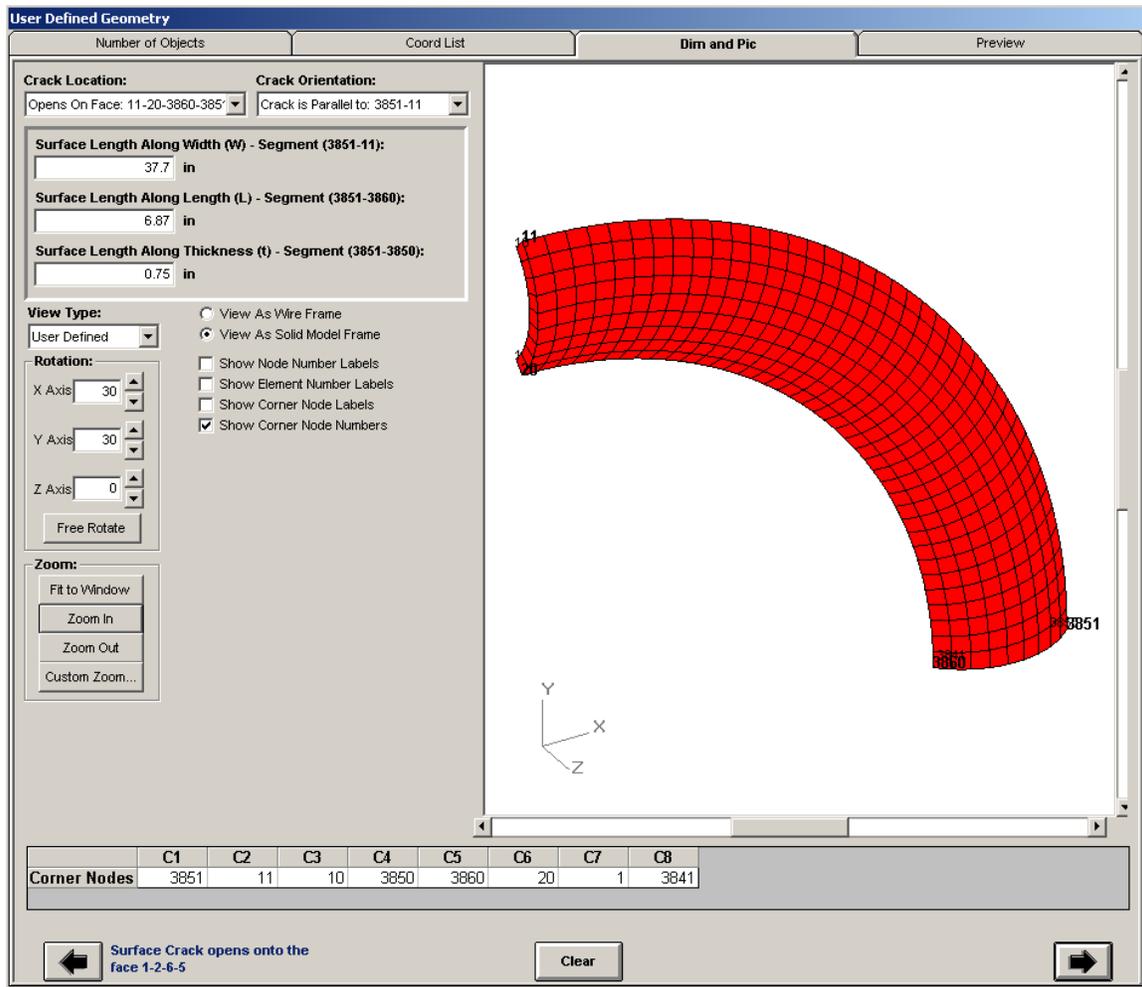


Figure 19 crack location relative to corner node “C1” in a user-defined geometry

For this example, the resulting crack mesh is shown in the figure below. Note that the crack shift distances are relatively small, which locates the crack near the bottom back corner of the definition mesh, close to corner 1.

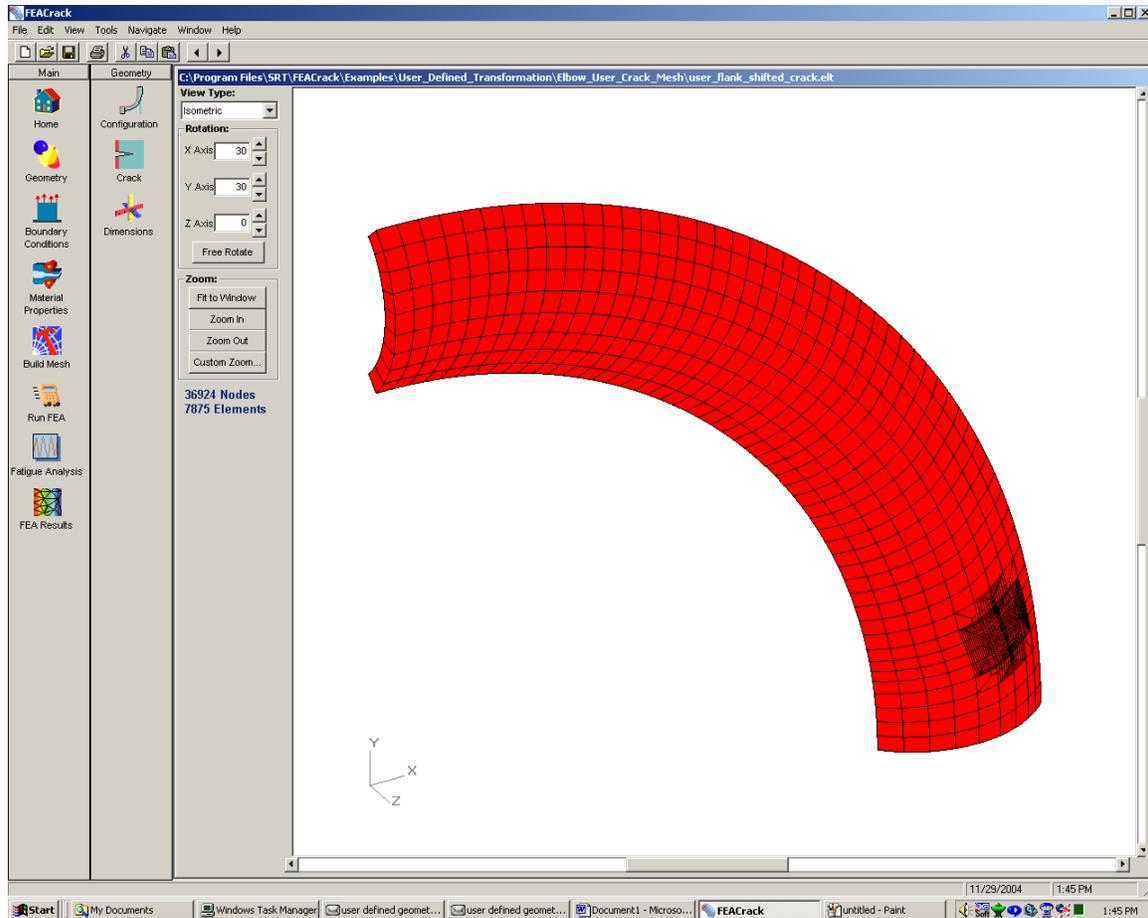


Figure 20 the crack has been shifted and rotated relative to the center of the model

Pull-Down Menus

File Menu

The New command will clear any existing input data. The Open command will load an existing model file. Model files have an *.elt* extension. To load a finite element results file, click on the FEA Results icon on the navigation bar.

Standard Windows® printing commands, such as Print, Print Preview, and Page Setup are listed under the File menu.

Edit Menu

Standard Windows® editing commands, including Cut, Copy, and Paste are given under the Edit menu. These commands apply to text in the selected input field or spreadsheet cell(s). The Copy Down and Copy Right commands function the same as in spreadsheet programs such as *EXCEL*, and are active when a range of cells is selected.

View Menu

Commands under the View menu can be used to show or hide the Windows® tool bar along the top of the main window, as well as the status bar along the bottom. Other commands under the View menu pertain to the graphical display of finite element results, as described below.

Select “Move Scale Bar” to change the location of the color scale bar. The scale bar changes to a blank rectangle, and it can be dragged around the screen with the mouse. When the scale bar is in the desired location, select the View Scale Bar command again to set the new location for the scale bar.

Select the Change Scale command to modify the color scale minimum and maximum values. Alternatively, double click on the scale bar.

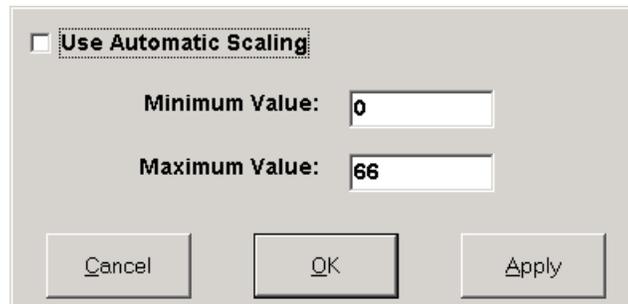


Figure 21 Change the color scale bar value range

The Show Drawing Control opens the Display Controls form as shown below. This form remains on top of the display until it is closed. Use this form to change the various display values such as the displacement scale (exaggerate the deformed shape), the result type (stresses or strains), the result step, the zoom and scroll factors (shift the mesh location in the graphics display). The other view controls are also available while the Display Controls is open. Use the Capture View button to get all the current display settings of the mesh. This is useful when comparing several different meshes by then using the Apply button to update the display settings after loading or selecting another mesh result.

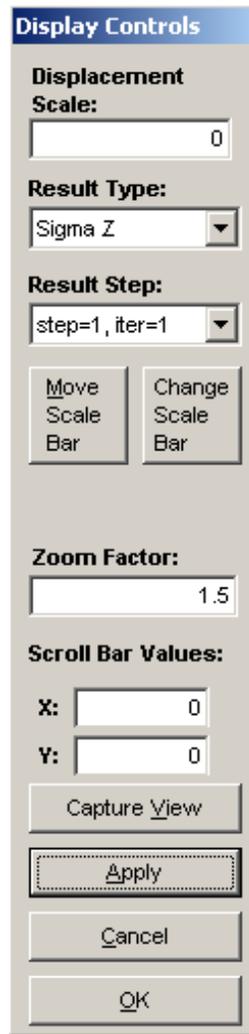


Figure 22 Update the display controls

The Cut Image command enables the user to view a portion of the model. The default cutting planes are on the Cartesian coordinate axes. Alternatively a user-defined cutting plane may be defined by dragging the mouse over the 3D image. In this case, however, the cutting plane may go through one or more elements, and the cross section may have an unusual appearance.

Tools Menu

The Tools menu contains Program Options and Model Options control panels, both of which are described below.

Program Options Control Panel

This control panel, shown below, has three tabs. The Analysis Options tab provides user control on the finite element input and results files. The FEA Options tab is used to set program specific options. The Graphics tab can be used to adjust the appearance of the 3D mesh images.

Help/About and Log File Location

The Help/About form contains basic information about FEACrack, including the host ID number needed to create a license file, and the location of the log files. Most of the log files are located in the System “TEMP” directory, shown near the bottom of the About form, see the figure below. The crack mesh generator log file is written to the same directory as the model file. Or from the Windows CMD command line, type “set” to list the system environment variables, including “TEMP” to get the directory path to the log files.



Figure 23 Help/About form shows program data including the host ID number and log file location

Analysis Parameter Controls

Optional analysis parameter controls are available in the Home form, the Tools/Program Options forms, and Tools/Model Options forms.

On the Home form, shown in the figure below, the analysis units and analysis type (standard focused mesh or cell type mesh) are chosen. The three text boxes: Analysis Name, Model Name, and Notes may have optional text entered, or be left blank. The Analysis Name and Notes text are written as a comment at the top of the FEA input file generated by FEACrack. The Analysis Name is intended to be a short one sentence description, and the Notes text can have up to 512 characters to describe details of the model.

The Model Label is used by Warp3D for the batch messages file name. A name up to 8 characters without spaces is recommended. The Warp3D batch messages option is in the Tools/Program Options/Warp3D form, which is described below.

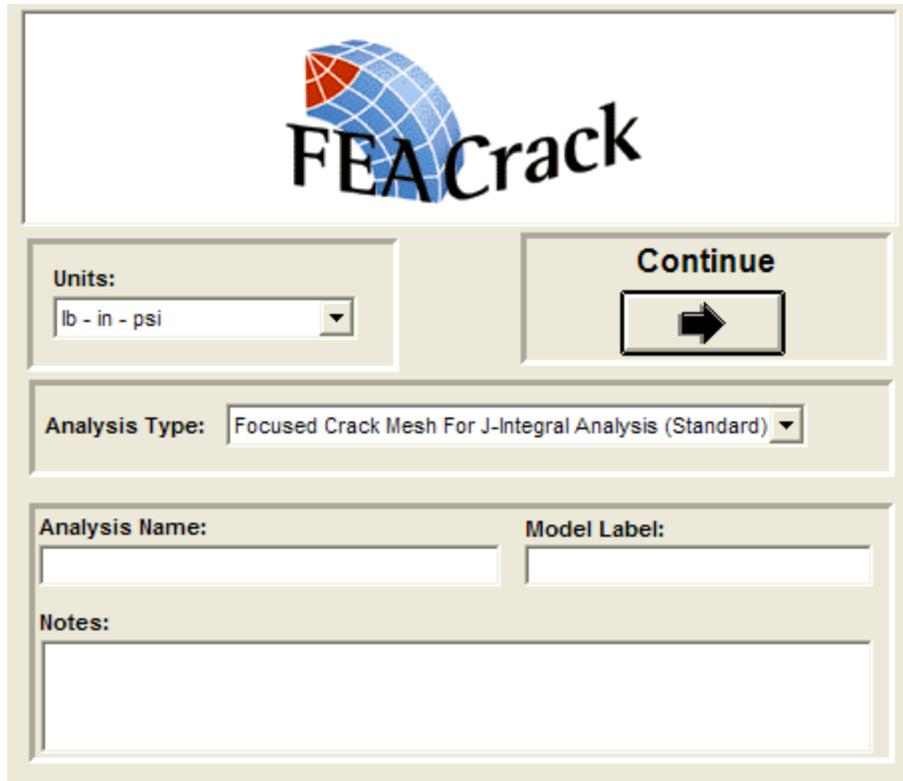


Figure 24 On the Home form enter optional text for the analysis name, model label, and notes

In the Tools/Program Options/Analysis form, shown in the figure below, the FEA input file type and output types are chosen. The restart file option can be selected so that the syntax to write the restart file is written at the end of each analysis step in the FEA input file. The restart file can be quite large, so only select the restart option if the restart file will be needed. More information about using a restart file can be found in the particular FEA program documentation.

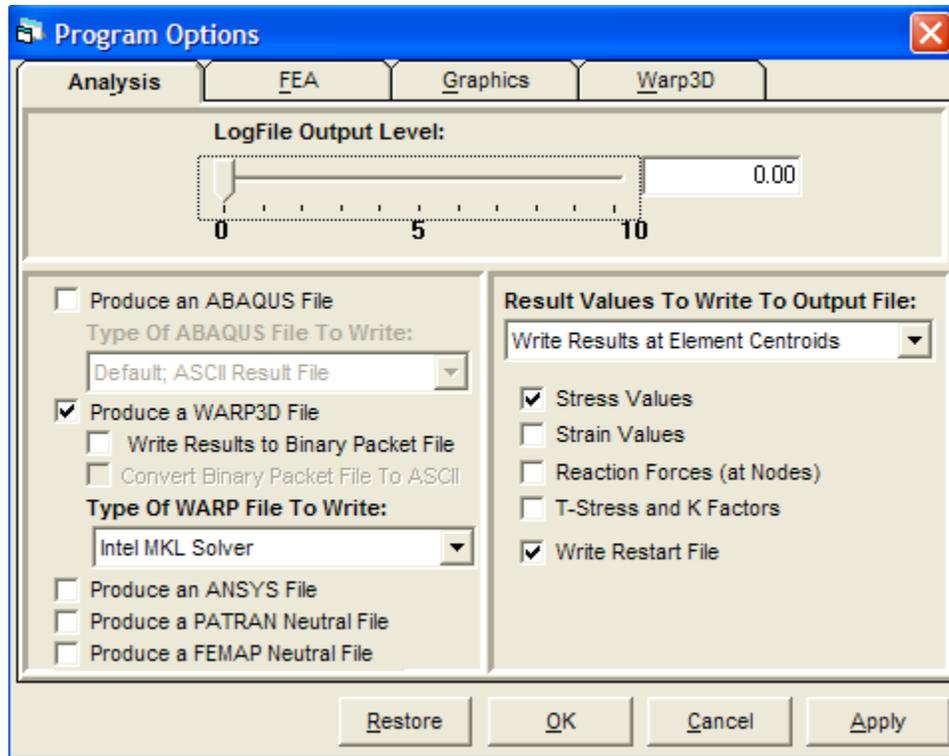


Figure 25 General analysis controls for FEA type and result choices

The Warp3D FEA program has several analysis parameters available. The Warp3D specific analysis parameter controls are in the Tools/Program Options/Warp3D form, shown in the figure below. The recommended default values are shown in the figure, and the default values can be recovered by clicking the “Restore” button at the bottom of the form.

Four convergence tests are available in Warp3D. Any combination of the four convergence tests can be used in Warp3D analysis. All the selected convergence tests must be met for solution convergence. The tolerance values are given as percent values; for example 0.05% is a reasonably strict tolerance for solution convergence. The tolerance values are relative values and may depend on the particular model and unit system used. The “Norm Displacement” test uses the Euclidean norm of the incremental displacement vector. The “Norm Residual Force” test uses the Euclidean norm of the residual force vector. The “Maximum Displacement” test uses the maximum absolute value in the incremental displacement vector. The “Maximum Residual Force” test uses the maximum absolute value in the residual force vector. The Norm Residual Force test with a tolerance of 0.05 has been used successfully in many analyses.

The Batch Messages check box activates writing the batch messages file during the Warp3D analysis. The batch messages file is useful for multi-step analyses to monitor the solution progress. The Model Label text on the home form can be used to set the batch message file name, or if left blank the default “vessel” label will be used. The batch messages file name is <model label>.batch_messages; for example “plate.batch_messages”. The batch messages file can be viewed with a text editor like Notepad.

More information about each analysis parameter can be found in the Warp3D manual, which can be found in the install directory.

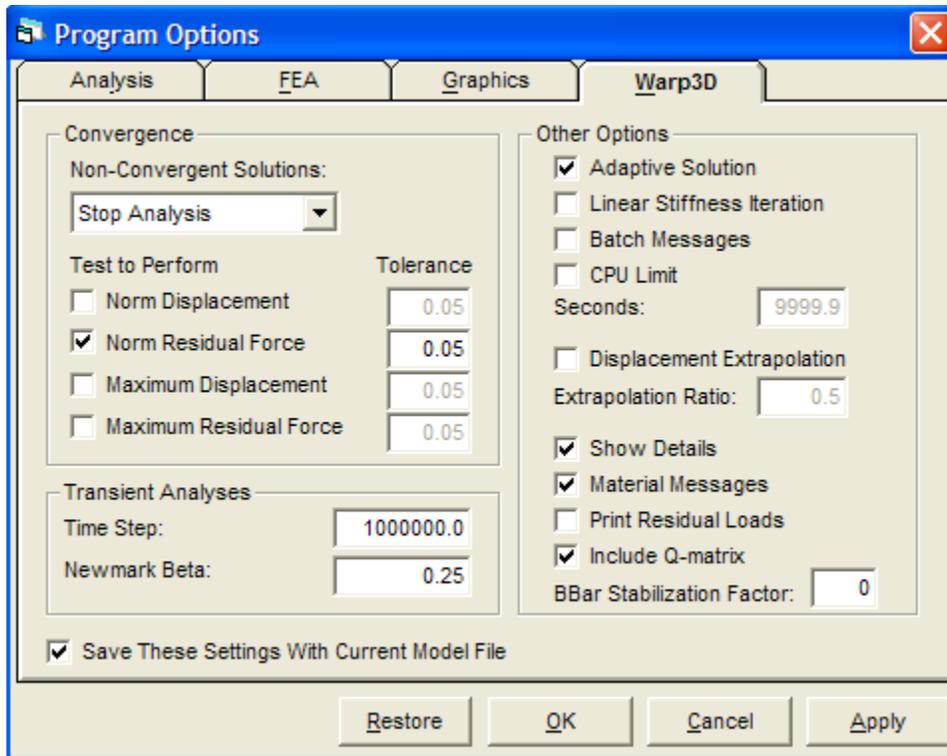


Figure 26 Analysis parameter controls for Warp3D

In the Tools/Model Options/Numbering form, shown in the figure below, the crack mesh can be renumbered and the iteration values can be set. Several FEA programs allow the minimum and maximum number of iterations to be set. Depending on the particular FEA program, the given number of minimum and maximum iterations will be used in the input file. The recommended default number of iterations can be obtained by clicking the “Restore” button at the bottom of the form. The Restore button also resets the parameter values on the other Model Options forms.

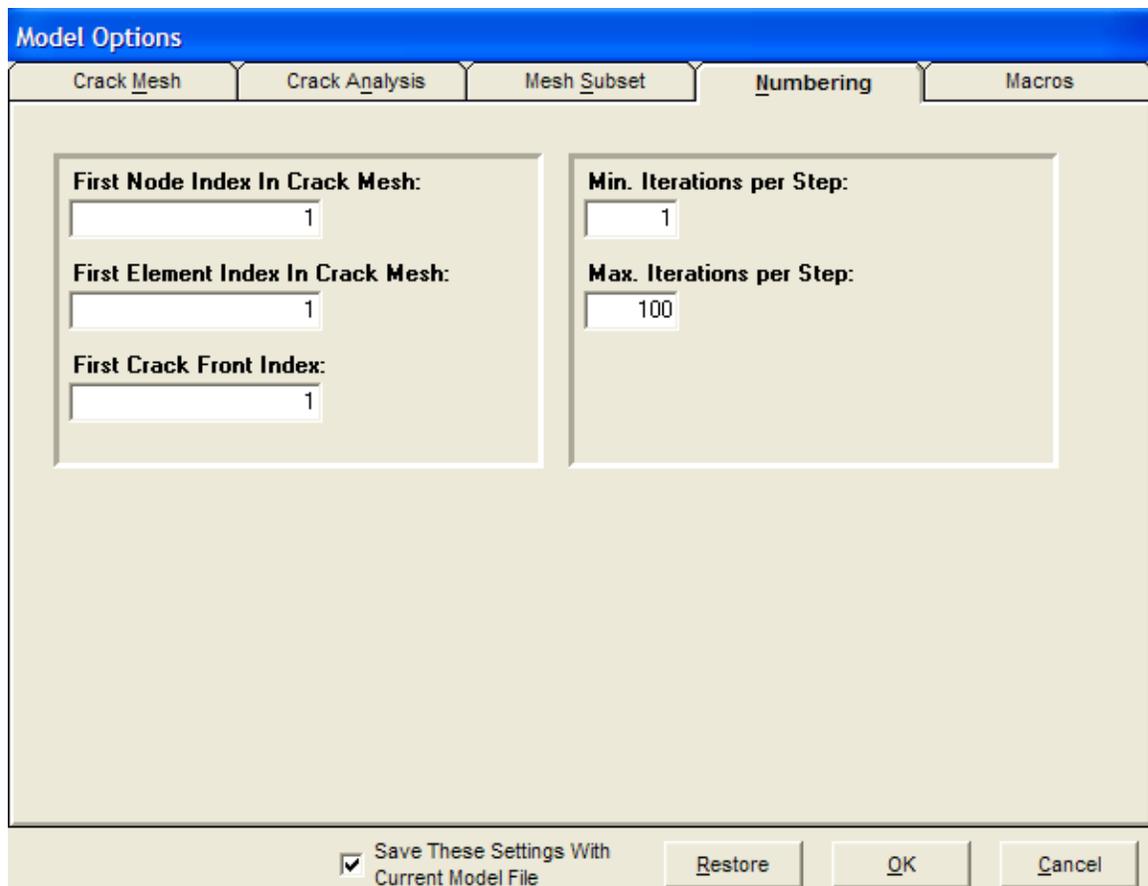


Figure 27 Enter minimum and maximum iteration values used in an analysis step

ANSYS Command Syntax

In item “1” in the figure below, select the path to the ANSYS Mechanical ADPL executable. For example, ANSYS version 19 uses the “ANSYS190.exe” executable. In item “2” below, also update the product syntax for the same version of ANSYS.

In item “3” below, use the check box to include the “SOLVE” command in the ANSYS input file; or uncheck that option if the input file will be loaded into the ANSYS GUI to avoid the solver being started automatically.

In the Tools/Program Options/FEA form, check the “Use ANSYS specific file...” option box to use the “-i” and “-o” ANSYS specific command line syntax to run the ANSYS input file from the Run FEA form. That syntax will be used in the FEA_Driver.bat file, which contains all the ANSYS specific syntax to run an ANSYS analysis. If the option box is unchecked, the default file redirect syntax using “<” and “>” will be used in the FEA_Driver.bat file instead. Refer to item labeled “4” in the figure below.

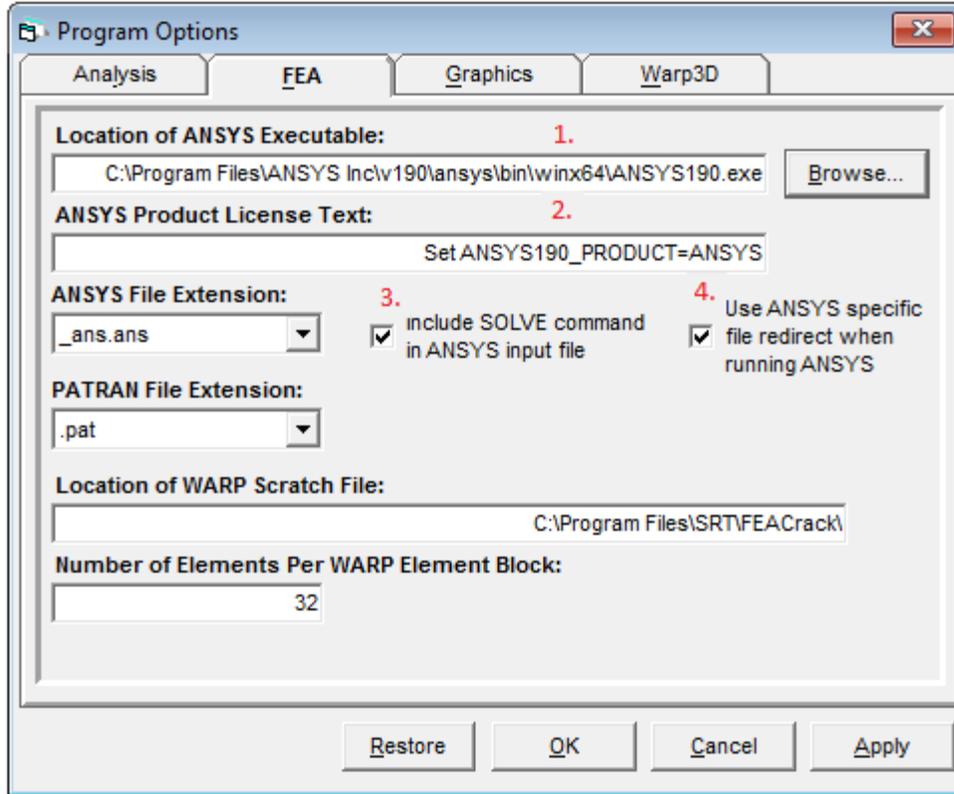


Figure 28 Check the option by item “4” to use the “-i” and “-o” options on the ANSYS command line

ANSYS Contour Integral and CINT option

The ANSYS contour integral CINT command can be included in the input file for ANSYS to compute the J-integral along the crack front. The *GET command is used to extract the J-integral values from the ANSYS database. The *VWRITE command is used to write the J-integral results to the <model>_cint.txt file, which is read by the FEACrack post processor. By default the CINT syntax is omitted from the ANSYS input file. To include the CINT syntax add this key word text in the Home/Notes form (not case sensitive):

*use ansys write cint

Older versions of ANSYS did not have the CINT command, so it can be deactivated and omitted from the ANSYS input file (should be omitted by default) by using this key word text in the Home/Notes form (not case sensitive):

*use ansys omit cint

When the CINT command is omitted, the J-integral will be calculated by the post-processing module using the ANSYS stress and strain Gauss point results.

Abaqus Contour Integral Syntax Options

Two node sets are used to support the syntax for the *CONTOUR INTEGRAL command in the Abaqus input file. At each crack front location the first node set includes all the nodes around the crack tip, which may be a single node for elastic analysis, or several nodes for a plasticity analysis. The second node set includes a single node to define the first crack tip node.

The *CONTOUR INTEGRAL command needs to be declared in just the first analysis step data to active calculation of the J-integral and other crack front values, since the crack front nodes and data do not change during the analysis. This may also help speed up the Abaqus preprocessor by only needing to process the command once. Crack front results are output for all analysis steps.

The node set syntax and single instance of the command is used by default, just build the mesh.

To revert to the older syntax used in previous versions of Abaqus, two key word options are available for use in the Home/Notes form. The key word option to control the node sets used to define the crack front is given by:

*use abaqus omit contour tip node

The key word option to control if the command is included in the first step only or included in every step is given by:

*use abaqus contour all steps

These options can be used individually or combined in the Home/Notes form, separated by a semi-colon “;” as for other “*use” key word options.

An example of the command from an Abaqus input file shows the two node sets for each crack front position, and the addition of the “crack tip nodes” key word:

```
*CONTOUR INTEGRAL, CONTOURS=6, NORMAL, OUTPUT=BOTH, SYMM, CRACK TIP
NODES, FREQUENCY=100, TYPE=J
  0.00000000, 0.00000000, -1.00000000
J1, J1_tip
J2, J2_tip
J3, J3_tip
...
```

Load Step Functions

In the Boundary Conditions/General form one of five functions can be chosen to fill the load step data in the Constraints forms. The load step function is selected in the “Increment Calc. Mode” menu at the bottom of the form, shown in the figure below. The functions are labeled: Default, Uniform, Linear, Logarithmic, and Fibonacci. Each function computes the load step increment value that is multiplied by the given constraint value to fill the load step data. The intent of the functions is to provide a decreasing or uniform increment size that is appropriate for an elastic-plastic analysis; the later load steps will have a smaller increment change in the boundary condition value to help the solution convergence.

The Uniform function defines a uniform load increment for each load step. The Linear function computes a linearly decreasing increment value from the given Final Increment value, which is entered in the text box just below the menu. A final increment of 0.01 is recommended. The Logarithmic function uses the Log10 function to compute the decreasing increment size. The Fibonacci function uses the Fibonacci series to compute a decreasing increment size.

An example with 25 load steps compares the increment size and load history computed by each function. In the first plot shown below, the increment size for each step is plotted. Notice that

the Uniform function has a constant increment size, and the Linear function has a linear decrease in the increment size. The Default, Logarithmic, and Fibonacci functions start with larger increments and decrease to smaller increments by the last step. The second plot shown below compares the load history using the computed increment sizes. The uniform increment gives a load history with a constant slope. The other functions increase the loading more quickly to start, and then gradually approach the maximum value.

The load history can also be given as a custom series by checking the “user defined series” option box for any boundary condition choice.

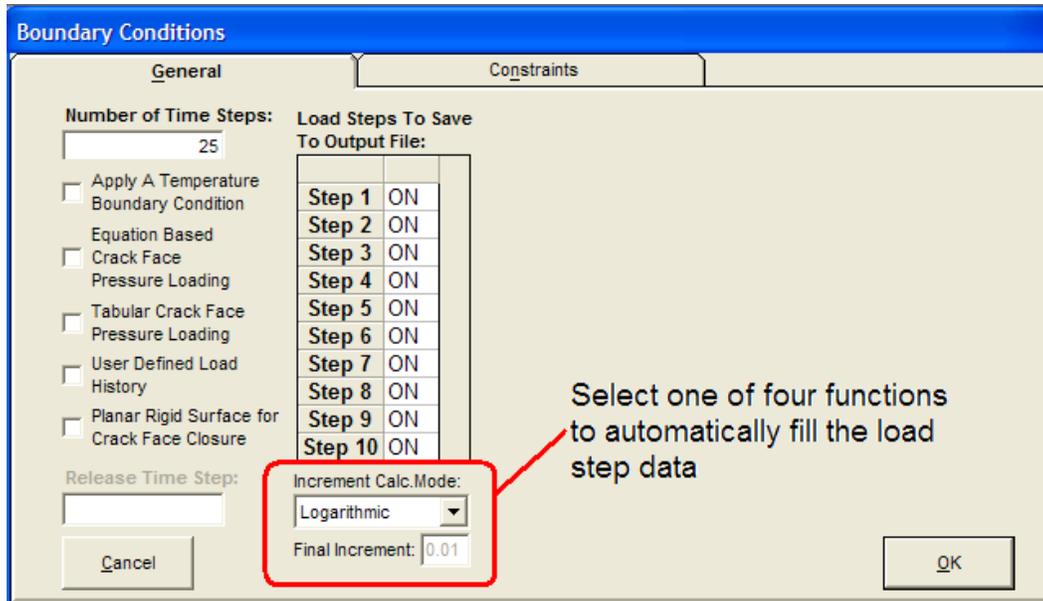


Figure 29 Select one of four functions to automatically fill the load step data in the Constraints forms

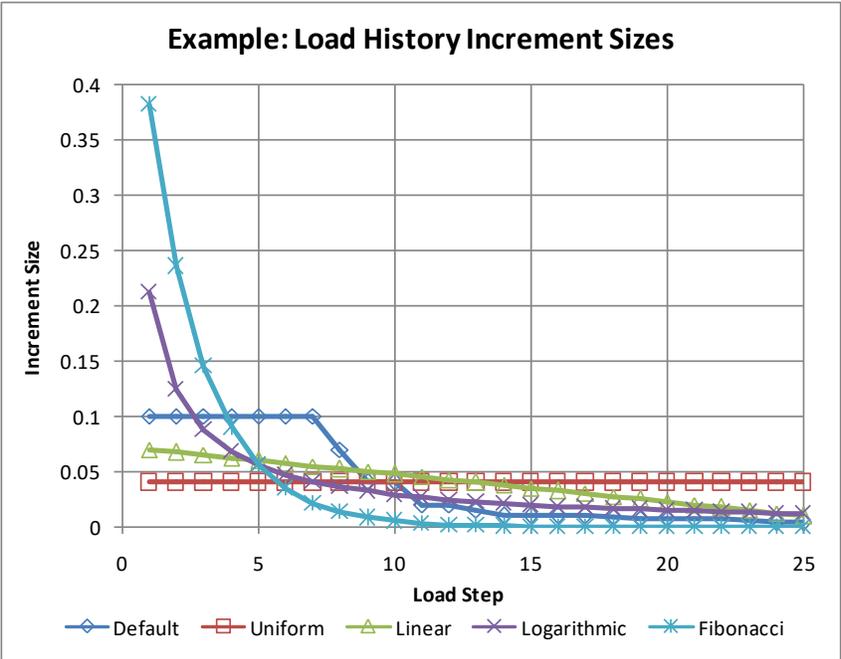


Figure 30 Load history increment sizes computed by each function choice; example using 25 steps

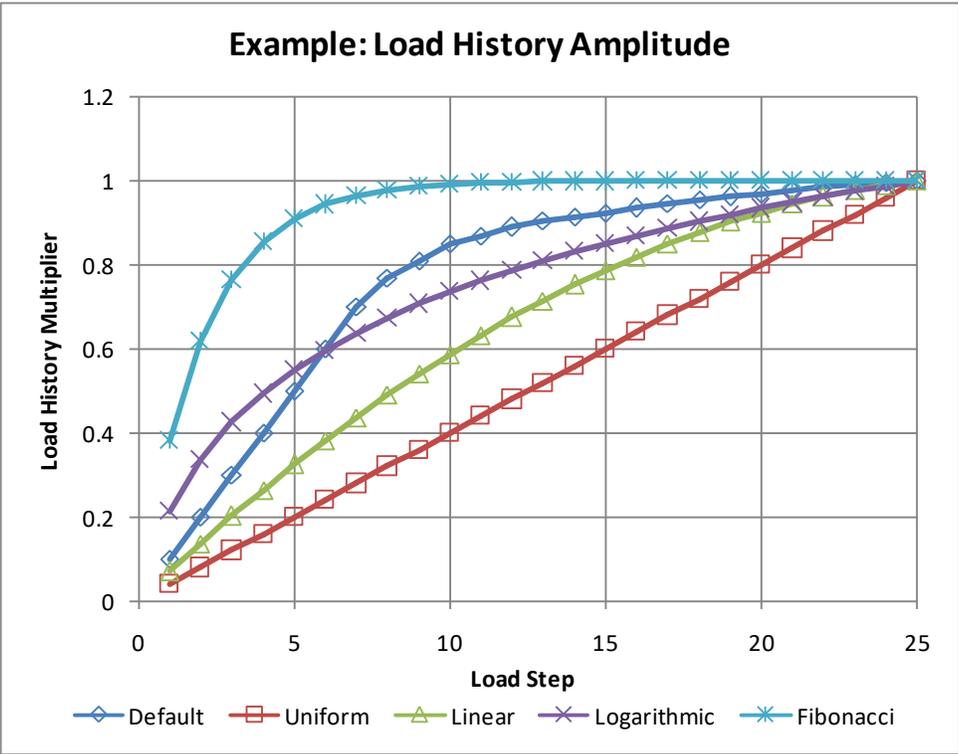


Figure 31 Load history for each function choice; example using 25 steps

File Translator

The Abaqus to Warp3D file translator program is available in the FEACrack/Tools menu by selecting “Abaqus to Warp3D Translator”, or can be run from the command line in a DOS console window. The file translator reads the mesh data from an existing Abaqus input file and then writes a new Warp3D input file. The file translator is intended for use with crack mesh models initially created by FEACrack and possibly combined with larger meshes. The file translator does not support all the Abaqus commands, data, or element types available in Abaqus.

Features available for the file translator currently include:

- Supports the classic Abaqus input file format (“flattened” format)
- Retains the crack mesh data block at the top of the input file generated by FEACrack; needed for crack result post-processing in FEACrack; this also maintains the subset mesh option to reduce the result file size
- Element types: 8 and 20 node brick elements
- Boundary conditions: element face pressure, nodal forces, node constraints and imposed displacements
- Load history amplitude for a multiple step analysis
- Crack front definition and conversion for the J-integral domain calculations in Warp3D; supports both initially sharp crack fronts and initially blunt crack fronts (key hole or finite radius)
- Tied contact surface definitions used to connect a user-defined geometry crack mesh to a larger master mesh

When the mesh subset option is used in the initial crack mesh, the binary packets file will be activated in the translated Warp3D input file. When there are multiple crack fronts in the mesh the comment data block information is used to identify the brick element type for each crack front to give correct domain integral data in the translated Warp3D input file. A blunt crack tip is detected from the “key hole radius” value being greater than zero in the comment data block.

Translator GUI

A graphical user interface (GUI) is available in the FEACrack/Tools menu; see the figure below. Select the existing Abaqus input file name using the file browser or by typing the full file path and file name in the top text box. Then select or enter the file name for the new Warp3D input file that will be created in the second text box. The default file name option creates the new Warp3D file name automatically by appending “_to_wrp.inp” to the given Abaqus file name; the default Warp3D file is located in the same directory as the selected Abaqus input file. Click the “Begin File Translation” button to translate the mesh from the Abaqus file to a new Warp3D input file. When the file translator begins, a console window will open, and then automatically closes when the file translation is finished. A summary of the file translation is written to the text window at the bottom of the form, which lists the file names, mesh size, and any warning or error messages. Use the right-side scroll bar to see the summary text. A successful file translation summary ends with the text: “Program finished”. The file translator summary is also written to the “AbaqusToWarp3D_summary.txt” file, in the same directory as the selected Abaqus input file.

To get more information about warning or error messages reported during the file translation; set the LogFile Output Level to 1 in the Tools/Program Options/Analysis form. Three log files may be written during the file translation: debug_ReadMeshSize.log, debug_ReadInput.log, and

debug_WriteInput.log. The log files contain a listing of all the warning and error messages; only the last message is reported to the GUI.

Since there are many unsupported Abaqus commands in the file translator, all the unsupported commands are listed in the debug_ReadInput.log file. Look for the “INFO:” text followed by more information in the log file. A warning message in the summary also gives a short list of the unsupported commands that were found in the Abaqus input file.

Since there are not corresponding solution parameters in Abaqus to the available solution options in Warp3D, default values are used. In particular, the default windows solver is selected (Intel MKL). A Warp3D analysis can also be run directly from the console window command line; refer to the Warp3D manual for more information.

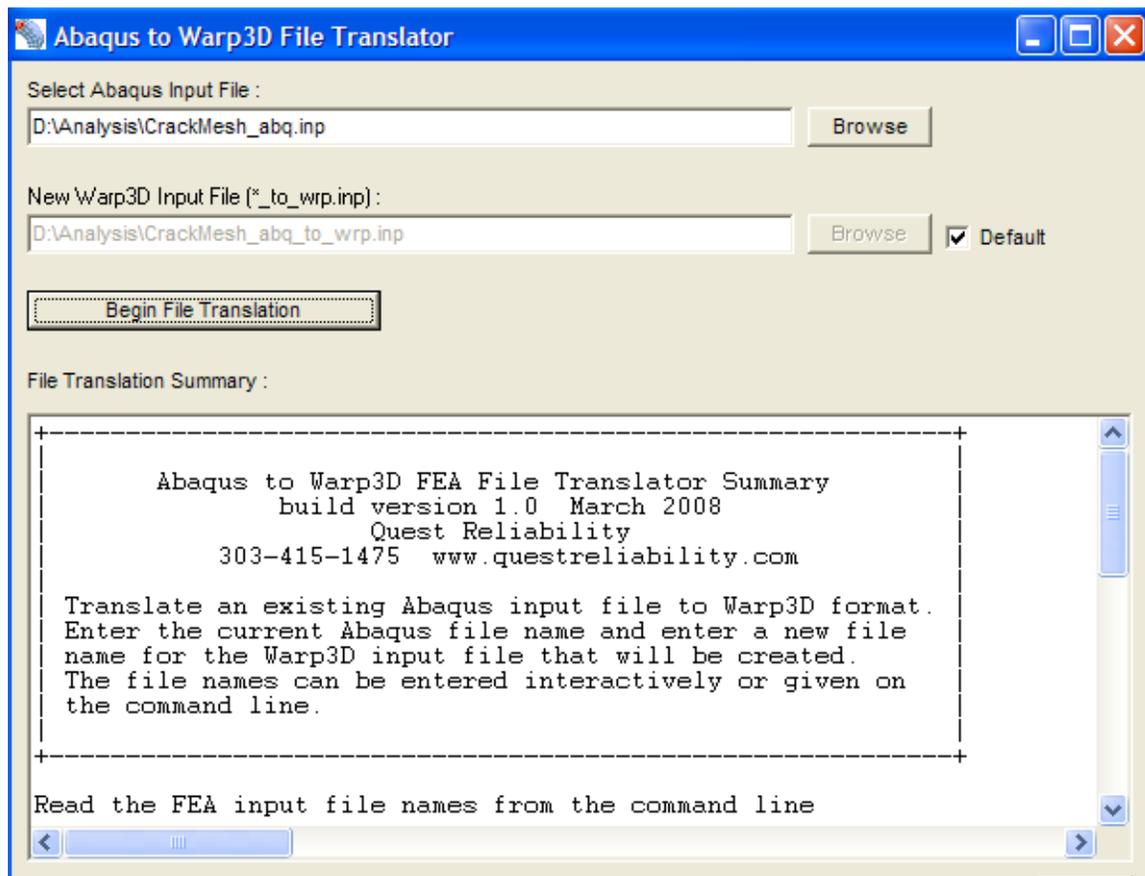


Figure 32 File Translator GUI in the FEACrack/Tools menu

Command Line Program

In addition to the GUI form, the file translator can also be run directly in interactive mode, with file name parameters given on the command line, or by a file containing a list of the Abaqus and Warp3D input file names. The “filetranslate.exe” program is located in the FEACrack installation directory.

To run the file translator in interactive mode, open a “Command Prompt” console window and change directories to the FEACrack install directory and enter “filetranslate.exe” to start the

program, or double click on the executable file icon in the install folder to start the program, which opens a console window. The first input prompts for the full file path and file name for the Abaqus input file, or enter a blank name to exit the program. The second input prompts for the new Warp3D file name, or enter just “d” to use the default Warp3D file name. The default name is shown for reference in the line just above the file name input line. The default Warp3D file name appends the text “_to_wrp.inp” to the given Abaqus input file name. The figure below shows an example of the interactive mode.



```
C:\Program Files (x86)\Quest Reliability\FEACrack>filetranslate.exe
-----
                Abaqus to Warp3D FEA File Translator

                version 1.0
                February 2008

                Quest Reliability
                303-415-1475
                www.questreliability.com

: Translate an existing Abaqus input file to Warp3D format.
: Enter the current Abaqus file name and enter a new file
: name for the Warp3D input file that will be created.
: The file names can be entered interactively or given on
: the command line.
-----

Enter the Abaqus input file name (blank to exit):
>>D:\Example\UnitCube_abq.inp

Enter the new Warp3D file name (blank to exit):
Recommend a file name with extension: "_wrp.inp"
or enter "d" to get the default Warp3D file name.
Default name: D:\Example\UnitCube_abq_to_wrp.inp
>>d

Translating from Abaqus file:
D:\Example\UnitCube_abq.inp
to new Warp3D file:
D:\Example\UnitCube_abq_to_wrp.inp

Read the input file:
D:\Example\UnitCube_abq.inp

Input file mesh data:
number of nodes ..... =    4546
number of elements ..... =     952
mesh format type ..... =   Abaqus
number of node sets ..... =     25
number of load steps ..... =      1
number of contour integrals ... =      1

Write the new input file:
D:\Example\UnitCube_abq_to_wrp.inp

Program finished.
```

Figure 33 Example of running the file translator interactively from the console window command line

Command Line Parameters

The file translator can also be run by including the file names as parameters on the command line. The format of the syntax is:

```
>filetranslate.exe [Abaqus file name] [Warp3D file name or "d" for default file name]  
debug=[n]
```

For example, the filetranslate.exe is run from the FEACrack install directory, the full file path to the Abaqus input file is given, followed by "d" for the default Warp3D file name. The syntax using the same files as in the interactive example above is:

```
C:\Program Files\Quest Integrity Group\FEACrack>filetranslate.exe  
D:\Example\UnitCube_abq.inp d
```

If there is a space in the file path, enclose the full file path and file name in quotes. The third parameter is optional and is used to activate the debugging log files during the file translation. The debug parameter should usually be omitted. The syntax is "debug=[n]" (no spaces between values), where [n] is the log file output level from 1 (minimum debugging output) to 10 (maximum debugging output). The debugging log files are written to the Abaqus input file directory and begin with "debug_".

File of Input Names

The same command line control of the file translation can also be achieved by listing the file names in a text file and using the system redirect characters "<" and ">" to run the file translator and direct input from the file list and to an output file. The syntax is:

```
C:\Program Files\Quest Integrity Group\FEACrack>filetranslate.exe < name_list_file.txt >  
summary_file.txt
```

Where the "name_list_file.txt" file contains the Abaqus and Warp3D file names (or "d" for the default Warp3D file name); the summary report is directed to the "summary_file.txt" file. In some cases it may be easier to copy and paste the selected file names into a text file than to type them in the interactive mode of the file controller.

Mesh File Format

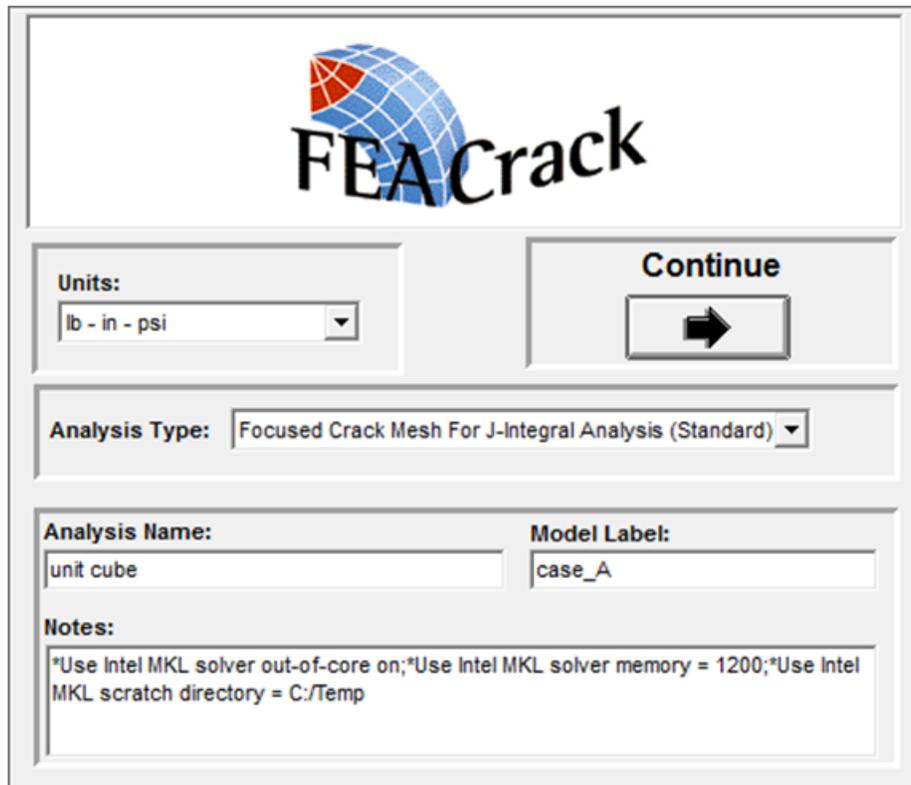
The finite element mesh can be written in a variety of formats, depending on the finite element program to be used. For finite element programs not specifically supported by *FEACrack*, the mesh can be written in *FEMAP* or *PATRAN* neutral format, which facilitates easy translation into a variety of other finite element formats. However, *FEACrack* does not support post-processing of results files from other finite element programs. It is important to note that at least one of the available file formats must be checked in order for a mesh file to be created. If none of the boxes are checked, a mesh will be generated and displayed on the screen, but no input file will be created. Please contact technical support to inquire about the input file format for another FEA program.

There are a variety of options for outputting finite element results. For example, results calculated at nodes, Gauss points, or element centroids can be written to the file. The size of the

results file can be greatly affected by the results that the user chooses to report. Output at the element centroid gives the smallest result file, while output at the nodes or element integration points gives a larger result file. This choice will affect the stress and strain color map in the results graphics window. Element centroid results give a single color per element, integration points give a smaller single color region within the elements (at each integration point in the element), and nodal results give a continuously varying color map on the mesh.

For *ANSYS* be sure to select stress and strain results at the element Gauss (integration) points as those results are required by the post-processor to compute the J-integral for an *ANSYS* mesh.

For analyses with *WARP3D* the Intel MKL sparse matrix solver is recommended, which is selected in the Tools/ProgramOptions/Analysis form. The Intel MKL solver can work out-of-core by using temporary files on the hard drive when there is insufficient computer memory (RAM) for the analysis. The out-of-core (OOC) work files have file names containing “_ooc_” such as: *pardiso_ooc.cfg*, and *warp3d_ooc_solver.**. The default solver memory size is 8000 MB (8 GB), and can be adjusted using key word text in the FEACrack/Home/Notes form. Since the out-of-core option is off by default; key word text in the Home/Notes form is required to activate the out-of-core option. The figure below shows an example of the key word text.



The screenshot shows the FEACrack software interface. At the top is the FEACrack logo. Below it is a 'Continue' button with a right-pointing arrow. The 'Units' dropdown menu is set to 'lb - in - psi'. The 'Analysis Type' dropdown menu is set to 'Focused Crack Mesh For J-Integral Analysis (Standard)'. The 'Analysis Name' field contains 'unit cube' and the 'Model Label' field contains 'case_A'. The 'Notes' field contains the following text: '*Use Intel MKL solver out-of-core on; *Use Intel MKL solver memory = 1200; *Use Intel MKL scratch directory = C:/Temp'.

Figure 34 Enter key word text in the Notes field to adjust the Intel MKL out-of-core options

The three key word options begin with “*Use” and include:

- *Use Intel MKL solver out-of-core <on/off>
- *Use Intel MKL solver memory = <value>
- *Use Intel MKL scratch directory = <directory path>

Where “on” is selected to activate the out-of-core option (currently redundant with the default), or “off” is selected to deactivate the out-of-core option. The solver memory <value> is entered as an integer for the memory size in mega-bytes. A typical 32-bit computer may have 2000 MB of RAM (2 GB), so a memory size of 1000 to 1500 MB is recommended. A typical 64-bit computer may have 4 to 8 GB of RAM, which is often sufficient to run Warp3D in-core.

The temporary OOC work files are written to the same directory as the input file by default. To set a different directory for the work files, enter the <directory path> for the scratch directory.

WARP3D is available for 32-bit and 64-bit computers. For more details refer to the Warp3D manual (PDF file). The 64-bit Warp3D executables are installed by default. The subfolder “Warp3D_files_32” contains 32-bit Warp3D executables.

Abaqus also uses a scratch file for out-of-core solving, but the location is set through Abaqus environmental variables. Consult the Abaqus user manual for more information. The ANSYS work files are usually located in the same directory as the input file; only the **_ans.out** and **_ans.rst** files are needed for post processing; other ANSYS files may be deleted. The **_ans.out** file is created by the command line syntax when running ANSYS; the command line file redirect sends the model information to that file, which is later used by the *FEACrack* post-processor.

For Abaqus there are several choices for the type of syntax used in the input data file. The pull-down menu below the Abaqus check box allows choice of an ASCII or binary results **.fil** file, or to use “part and assembly” syntax if the crack mesh will be used as a part within Abaqus /CAE.

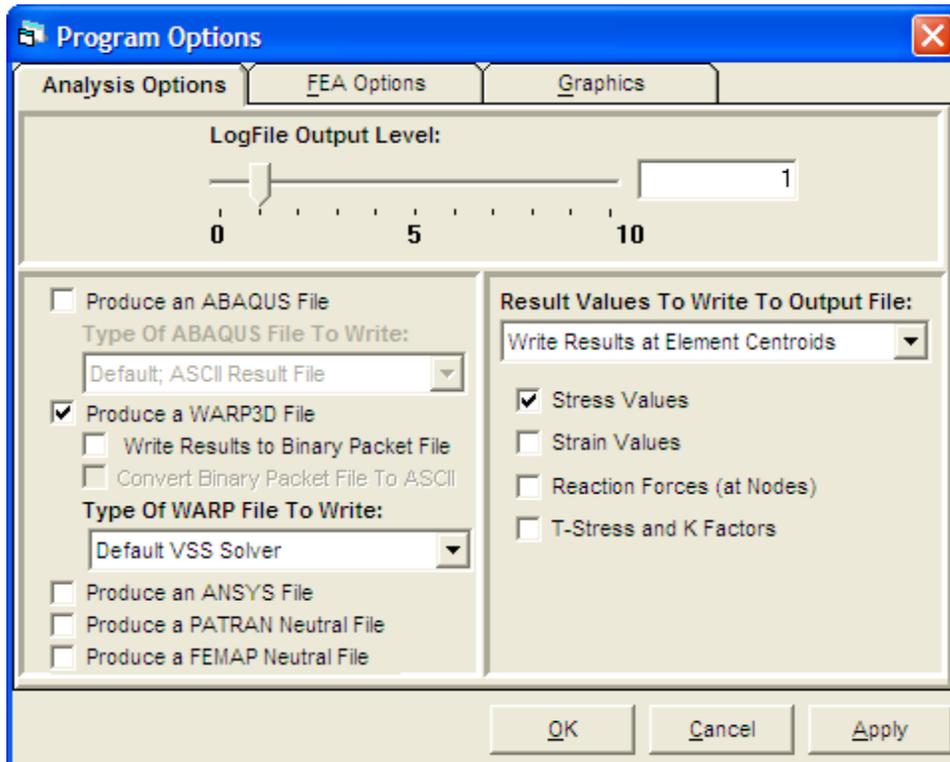


Figure 35 Tools/Program Options form to set FEA choices

Using the Log Files

The Analysis Options tab includes a slide bar for controlling output to one or more log files during mesh generation and post-processing. Any log files created will have a “.log” file extension that can be viewed with any text editor.

The crack mesh generator log file is written to the same folder as the *.elt model file, and uses the same model name. Other log files are written to the system “TEMP” directory, which can vary for different versions of the operating system. This directory is listed in the Help/About form. Or from a Windows CMD command line, type “set” to list the environment variables; then look for the “TEMP” environment variable giving the directory path to that folder.

Setting the “LogFile Output Level” to zero turns off any output to the log file; no log file will be created. Setting the “LogFile Output Level” to 1 will write any warning or error messages to the log file; any error or warning messages that appear in *FEACrack* can then be obtained from the log file. Only the last warning or error message is reported to the screen in *FEACrack*, but all messages will be listed in the log file. The warning and error messages may help diagnose a mesh generation or post-processing problem. Higher values for the “LogFile Output Level” will write more diagnostic information to the log file; the user does not usually need this information, and the higher log file output levels will take longer to generate the mesh. If necessary, we may ask you to create a log file in order to help diagnose a problem.

FEA Options

Select FEA program specific options on the FEA Options form. For *ANSYS* select the directory path to the *ansys.exe* executable, and set the product license text used in the batch file to run the analysis. The typical product license text is shown below for *ANSYS/Mechanical*. A different keyword text is needed for *ANSYS/Multi-physics*. *ANSYS* permits several choices of input file name extension (.txt or .ans). Select a *PATRAN* file extension.

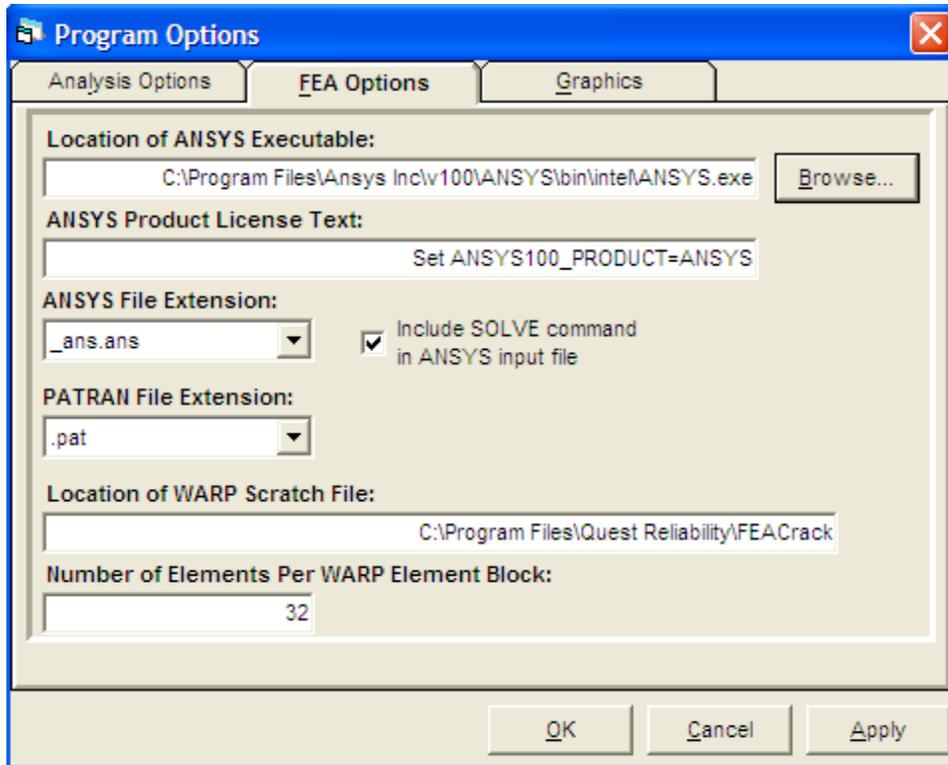


Figure 36 Set file extensions and program locations

Depending on the computer architecture, selecting the number of elements in the element block can optimize the *WARP3D* analysis; *WARP3D* uses this size element block for efficient processing of the mesh data; refer to the *WARP3D* manual for more information. Typical values for the element block size are 16, 32, 64, 128 elements per block (powers of 2); most often 32 or 64 elements per block works well. To optimize the element block size for *WARP3D*, build the same mesh with several different block sizes, run the analyses and compare the run times. Choose the block size that gives the shortest run time.

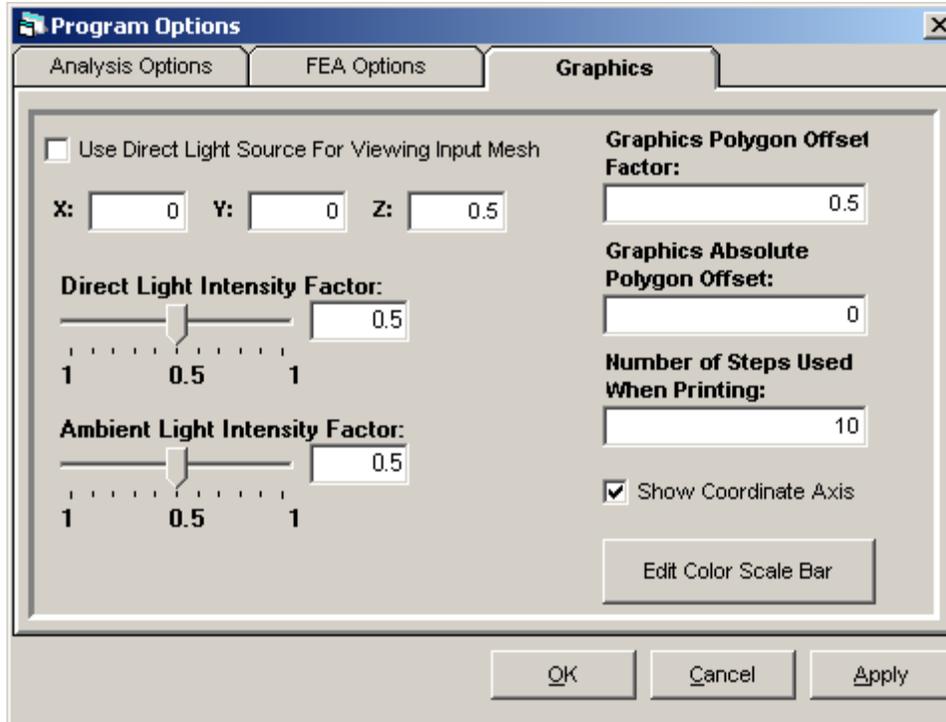


Figure 37 General graphics controls

On the Graphics tab, there are a number of controls for fine-tuning the appearance of the 3D mesh graphics on your computer.

When the mesh is viewed without a false color plot (no specific result and/or time step selected), both a direct and an ambient light source can be used. The relative intensities of these two light sources are controlled with the two slider bars named: *Direct Light Intensity Factor* and *Ambient Light Intensity Factor*. In addition, the location of the direct light source can be adjusted by changing its XYZ location.

The two polygon offset factors shift the element boundary lines slightly so that they stand out better when the full 3D shape is drawn. The *Graphics Polygon Offset Factor* shifts the lines based on each element's individual angle, while The *Graphics Absolute Polygon Offset* shifts the lines a constant number of pixels. Because the amount that a line is shifted is specific to each graphics card, these controls are made available to the user for fine-tuning the graphics image. If the lines are too faint (becoming dashed lines), make the values bigger. If the lines are starting to show through (strange corner effects), make the values smaller; for most graphics cards, the default values of 0.5 (Offset Factor) and 0 (Absolute Offset) should give the best results.

Since most printers have a much higher resolution than most monitors, the printed graphical image can require a large amount of memory. In order to allow printing on machines with smaller amounts of physical memory, the image must be printed in several steps. The exact number of steps that are required is a user-controlled option from this window; the default is 10 printing steps.

In addition the visibility of the XYZ coordinate axis and the values and scale of the color scale bar for results can be controlled from the Graphics tab.

Model Options Control Panel

The Crack Mesh Params tab provides the user with a high degree of control over the mesh. Refer to Page 72 for detailed instructions on using the mesh controls.

The Crack Analysis Params tab contains a number of settings, such as the maximum number of nodes and elements in the mesh. The user can also specify linear or nonlinear kinematic options for the finite element analysis.

When J- integral values are computed in the concentric rings of elements in the focused “crack tube” around the crack front by the FEA program or the *FEACrack* post-processing for *ANSYS* results, path dependence can occur. *FEACrack* discards the value for the first contour and averages the other contour values, and reports the averaged J-integral values in the Crack Result form plots and data table. If significant path dependence is detected in one of the values along the crack front, the user has the option of discarding this value and estimating J at this location either from a linear or polynomial extrapolation. Normally, path dependence occurs in J-integral values computed near the crack tip at the free surface. Usually 4 or 5 rings of elements in the crack tube are sufficient to compute the J-integral.

To obtain all the J-integral contour values check the “create extracted crack results file” option box in the lower right corner of the FEA Results/File Selection form. A text file with a *_res.out* file extension is written containing a variety of detailed crack result values. The extracted results file can be viewed with a text editor like Notepad or Wordpad.

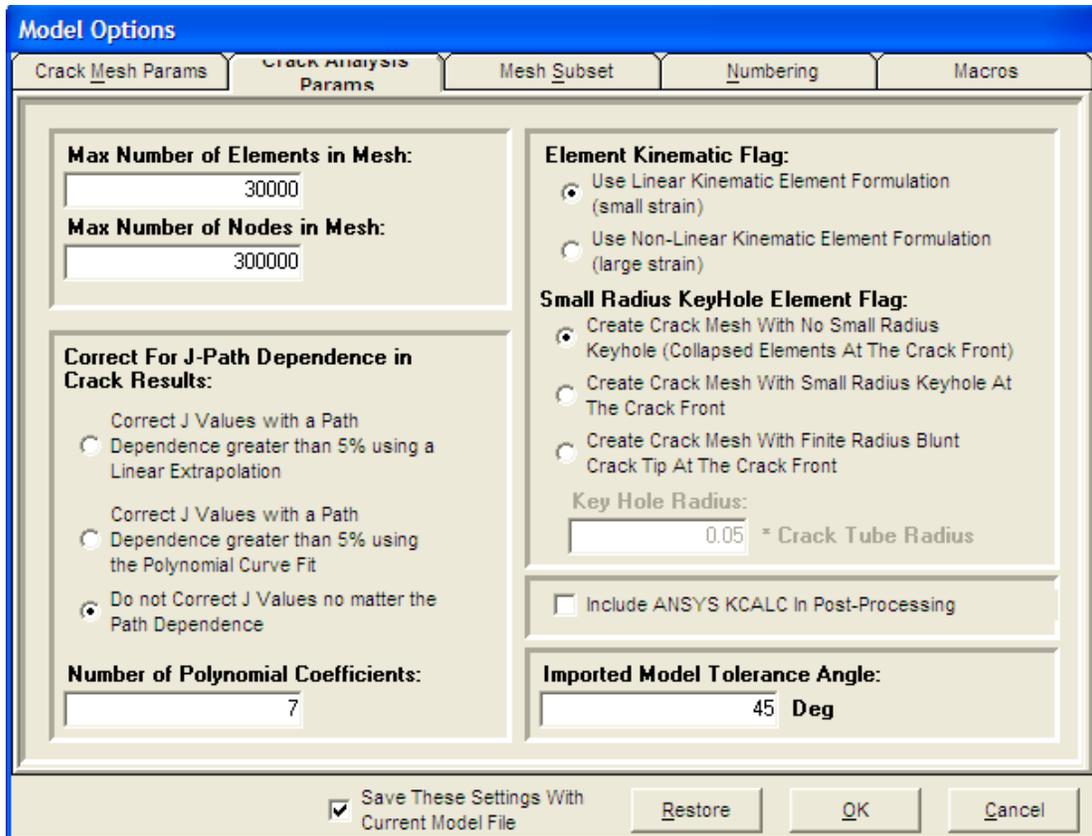


Figure 38 Crack Analysis Params to set maximum mesh size and other options

The available maximum number of nodes and elements in the generated mesh depends on the available computer memory. A typical PC with 2 GB of physical RAM can usually handle a mesh of up to 100,000 nodes. The element kinematic flag allows the user to set the nonlinear geometry flag in the FEA input file, and to include the keyhole as a blunt crack tip feature.

For plasticity analysis and for fillet meshes, ANSYS crack meshes may need to use the small radius keyhole to avoid element errors; a keyhole radius of 0.01 works well.

The imported model tolerance angle is for the user-defined geometry definition mesh. Ideally, the definition mesh has corners close to 90 degrees at the mesh edges (the angle between element faces at a mesh edge). Adjust the import angle as necessary to import the definition mesh and get the correct number of corners and edges. An alternative to the tolerance angle is the “definition mesh corner nodes” comment data block (see below for more information).

Run From the Command Line

FEACrack can be run from the command line, typically to automate analysis of many cases. An existing model file (*.elt) is required, but some model parameters can be modified on the command line. The syntax to run *FEACrack* from the command line is:

```
C:\> "full path...\FEACrack.exe" "full path...\filename.elt"
/option_label
```

For example:

```
C:\> "C:\Program Files\Quest Integrity
Group\FEACrack\FEACrack.exe" "C:\Project Files\pipe_model.elt"
/RunFatigue
```

Or,

```
C:\> "C:\Program Files\Quest Integrity
Group\FEACrack\FEACrack.exe" "C:\Data\case1.elt" /BuildMesh
/NoQuit
```

The full path to the *FEACrack* executable should be defined in quotes if there are any spaces in the file path. The full path to the location of the .elt model file should also be defined in quotes if there are any spaces in that file path. The command line option labels begin with / (slash character), and are not case sensitive. More than one option can be used on the command line. The .elt model file must already have been created using *FEACrack* in interactive mode so that all the data is correctly defined and ready to build a mesh or run the fatigue analysis.

Multiple command lines can be put into a batch file (*.bat) to run several *.elt files sequentially. Put one command per line in the batch file for a list of .elt files to run.

Command Line Option Labels

The command line analysis related option labels include (and described in detail below):

- /BuildMesh
- /RunFatigue
- /RestartFatigue
- /NoSave
- /NoQuit
- /RunFEA
- /ParamFile and file name
- /ExtractedResults
- /Skip3DResultImage
- /PostProcess

The command line model file related option labels include (and described in detail below)::

- /ModelWidth
- /ModelRadius
- /ModelThickness
- /ModelLength
- /LoadSigmaZmax
- /CrackLength
- /CrackDepth

Command Line Options

- /BuildMesh**
- The Build Mesh Flag opens the specified .elt model file given in the command line in FEACrack to build the mesh.
- Usage:
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh
- /RunFatigue**
- The Run Fatigue flag will start a fatigue analysis of the specified .elt model file given in the command line and save the results when the analysis is finished. If any previous results have run, this flag will clear those and start over. This flag can be used with any other command line flags. The mesh does not need to be regenerated since that is part of a fatigue analysis.
- Usage:
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /runfatigue
- /RestartFatigue**
- The Restart Fatigue flag will continue a previously started fatigue analysis for the specified .elt file on the command line. As opposed to the runfatigue flag, this will not start a clean fatigue analysis but continue from where a previous one started. This flag can be used with any other command line flag.
- Usage:
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /restartfatigue
- /NoSave**
- The No Save flag specifies that the .elt file opened from the command line will not be resaved when the command is done running. Any changes to the model parameters will not be saved when FEACrack is closed. This is flag can be used in combination with any of the other command line flags.
- Usage:
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh /nosave
- /NoQuit**
- The No Quit flag specifies to leave FEACrack open after executing the commands from the command line. This flag can also be used in combination with any other flag.
- Usage:
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh /noquit
- /RunFEA**
- The RunFEA flag will run an FEA analysis from an input file that was generated by the .elt model file specified on the command line. This flag must be used along with the /buildmesh flag and appear after that flag. Only one FEA analysis can be performed per command line execution. An alternative to run multiple FEA input files is the “Run

FEA” button. Open the “Run FEA” file control wizard form, then select the input files and run in batch mode from within FEACrack. This approach requires that the crack meshes have already been generated.

Usage:

```
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh /runfea
```

/ParamFile

The ParamFile flag will activate the parameter text file option. Use this option with the parameter text file name on the command line. This flag should appear before other post processing options on the command line.

Parameter file name

The parameter file name is used with the ParamFile flag to give the full path and file name of the parameter text file. If there is a space in the file path or in the file name put the file name within quotes on the command line. The parameter text file contains options to control post processing options, such as the radial result interpolation. This flag should appear after the /ParamFile flag and before other post processing options on the command line.

Usage:

```
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /ParamFile “parameter file name”
```

/ExtractedResults

The Extracted results flag will create the extracted crack results file (*_res.out) when post processing an output file from the .elt model file specified on the command line. This flag activates the check box option on the FEA Results form. This flag must be used along with the /postprocess flag and appear before the /postprocess flag on the command line.

Usage:

```
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh /runfea /extractedresults /postprocess
```

/Skip3DResultImage

The /Skip3DResultImage flag will omit the “show 3D results image” check box option when post processing an output file. This flag deactivates the check box option on the FEA Results form. This flag must be used along with the /postprocess flag and appear before the /postprocess flag on the command line. By skipping the 3D results image, some time is saved by skipping one of the post processing tasks that may not be needed when running a start-to-finish analysis from the command line.

Usage:

```
c:\> “full path...\feackrack.exe” “full path...\filename.elt” /buildmesh /runfea /extractedresults /Skip3DResultImage /postprocess
```

/PostProcess

The Post Process flag will load the results for the specified .elt file on the command line. This flag should be used with both the /buildmesh and /runfea flags and appear last on the command line.

Usage:

```
c:\> "full path...\feacrack.exe" "full path...\filename.elt" /buildmesh  
/runfea /postprocess
```

Model File Modification Flags

/ModelWidth

The Model Width flag will alter the model width dimension of the .elt model file specified on the command line. This flag will only alter geometry models where a width applies, and the value specified on the command line will enter directly into the model regardless of the symmetry of the model. For example a flat plate with $\frac{1}{4}$ symmetry selected has the input for the plate width as $\frac{1}{2}$ width while a full symmetry or $\frac{1}{2}$ symmetry (back half) as the full width for the plate. This flag can be used in combination with any of the other flags. If used in combination with the /buildmesh flag, the /modelwidth flag must come first.

Usage:

```
c:\> "full path...\feacrack.exe" "full path...\filename.elt" /modelwidth  
value /buildmesh
```

/ModelRadius

The Model Radius flag works similar to the /modelwidth flag. Only geometry models that have a radius dimension will be changed. For geometries that there might be more than one radius, the radius that is changed is the dimension that applies to the main geometry. The /modelradius flag must come before the /buildmesh flag.

Usage:

```
c:\> "full path...\feacrack.exe" "full path...\filename.elt" /modelradius  
value /buildmesh
```

/ModelThickness

The Model Thickness flag will alter the thickness dimension of a valid geometry that is specified on the command line. This flag can be used in combination with any other command line flag. If used with the /buildmesh flag, it must come first on the command line.

Usage:

```
c:\> "full path...\feacrack.exe" "full path...\filename.elt"  
/modelthickness value /buildmesh
```

/ModelLength

The Model length flag will alter the length dimension of a valid geometry that is specified on the command line. This flag can be used in combination with any other command line flag. If used with the /buildmesh flag, it must come first on the command line.

Usage:

```
c:\> "full path...\feacrack.exe" "full path...\filename.elt" /modellength  
value /buildmesh
```

/LoadSigmaZmax

The Load Sigma Z Max flag will edit the custom stress value in the Z direction for a boundary condition on the far face of the geometry

specified on the command line. Only geometries that have a far face constraint will be edited by this flag. This flag can be used in combination with any other command line flag. If used with the /buildmesh flag, it must come first on the command line.

Usage:

```
c:\> "full path...\feackrack.exe" "full path...\filename.elt"  
/loadsigmazmax value /buildmesh
```

/CrackLength

The Crack Length flag will edit the crack length parameter, *c* or *2c*, of the crack for the specified geometry on the command line. As with the flags that edit the geometry dimensions, this flag will edit the value as it is specified on the command line regardless of what crack shape is selected. This flag will also only edit values for cracks where the crack length dimension is applicable. This flag can be used in combination with any other command line flag. If used with the /buildmesh flag, it must come first on the command line.

Usage:

```
c:\> "full path...\feackrack.exe" "full path...\filename.elt" /cracklength  
value /buildmesh
```

/CrackDepth

The Crack Depth flag works exactly the same as the cracklength flag except that it edits the crack depth dimension, *a*, for the crack of the model that is specified on the command line. This flag can be used in combination with any other command line flag. If used with the /buildmesh flag, it must come first on the command line.

Usage:

```
c:\> "full path...\feackrack.exe" "full path...\filename.elt" /crackdepth  
value /buildmesh
```

Run with multi-threading

Multi-threading, i.e. parallel programming, is being added using OpenMP in the crack mesh generator to speed up the mesh generation run time by taking advantage of computers with multiple processors. By default OpenMP is deactivated. To activate multi-threading add key word syntax in the Home/Notes form:

```
*use openmp num threads = n
```

For example, to use 4 threads in the crack mesh generator, use the syntax:

```
*use openmp num threads = 4
```

Where “n” is the number of threads to use. Typically the number of threads is less than or equal to the number of physical cores (CPUs) on the computer, or the number of logical threads for an Intel chip with hyper-threading (multiple logical threads per core). To use the default number of threads available on the computer set the number of threads to 0, for example:

*use openmp num threads = 0

To confirm that OpenMP has been activated, set the debug log file level ≥ 2 , rebuild the mesh, then look in the log file for text: INFO: the "use openmp num threads" flag has been activated; also look for variable: omp_threads_flag = 1.

Weld and Heat Affected Zone (HAZ) Options

The crack can be located in the weld material by selecting a weld shape from the "Weld menu" on the "Geometry Configuration form" as shown below. The heat affected zone (HAZ) can also be included in the mesh by checking the Include "Heat Affected Zone option box" below the weld menu. The weld, HAZ, and base metal can each have unique material properties; set the material values for each region in the "Materials form".

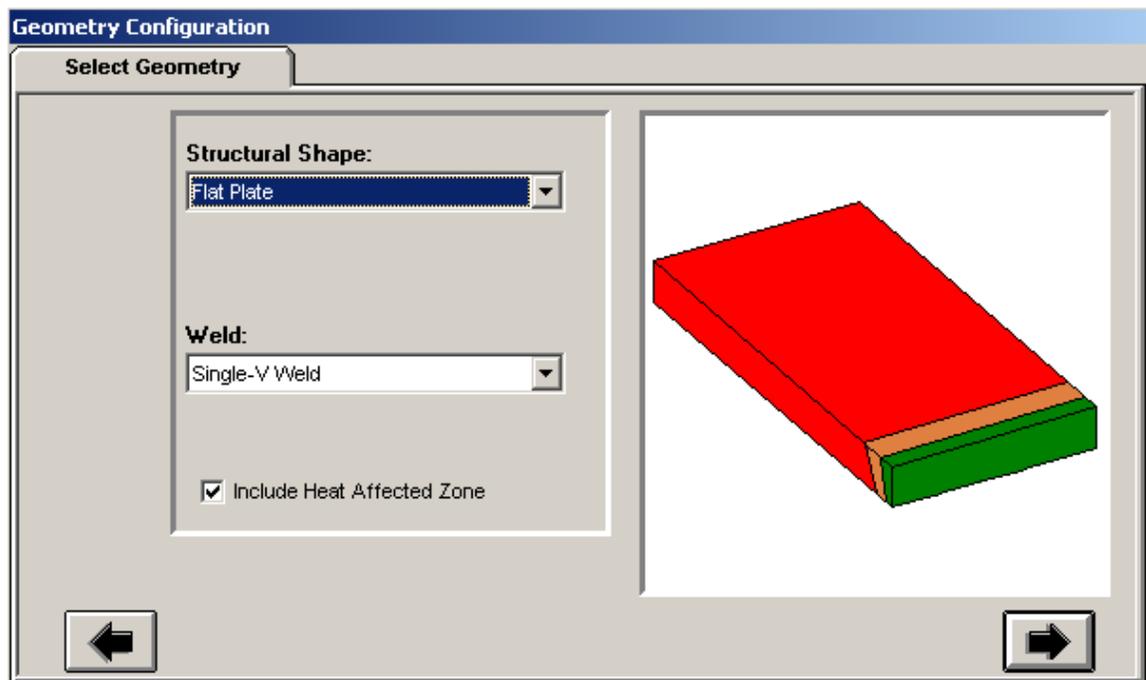


Figure 39 activate the weld heat affected zone

When the standard crack mesh blocks are used, the number of elements along the weld and HAZ depth is fixed for the given mesh block. When the PRO mesh is used, (cell or focused PRO crack mesh) the number of elements in the weld and HAZ can be specified in the Advanced Mesh/Weld tab as shown below. The PRO mesh weld and HAZ default is four elements in each region.

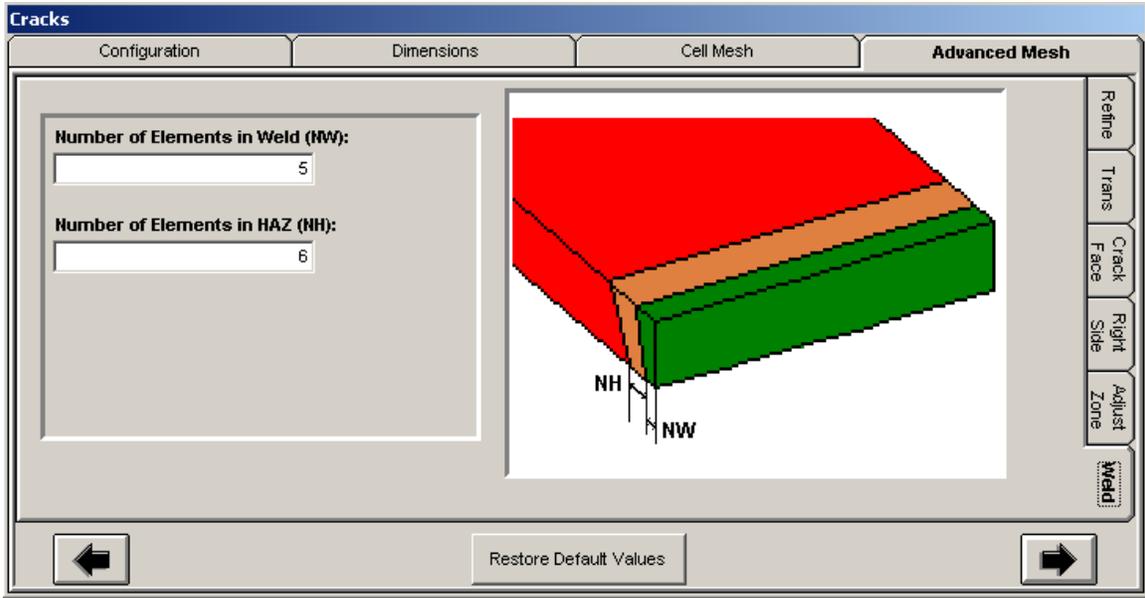


Figure 40 An example of a PRO focused crack mesh with the weld and HAZ material is shown below.

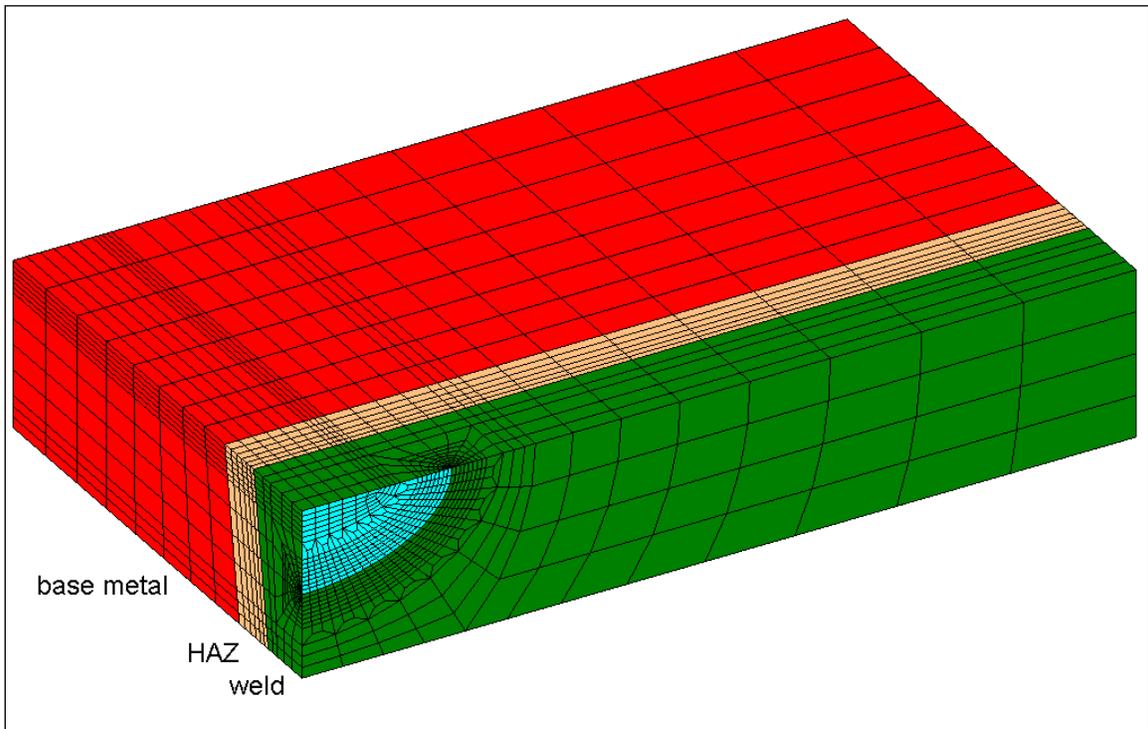


Figure 41 example of a crack in the center of the weld (green) with HAZ (tan) and base material (red)

Boundary Conditions

Click on the final forward arrow on the “Crack Dimensions” window or on the “Boundary Conditions icon” in the main navigation bar.

Flat Plate 3-Point Bending

The flat plate geometry can have 3-point bending boundary conditions applied to model an SC(B) test specimen. The boundary conditions can be applied as cylindrical load pins, which are modeled by analytic rigid surfaces, or applied directly to the mesh nodes. The load pin on the right side of the plate is located at the crack plane. The load pins on the left side of the plate are located at the back left corner, and if the full mesh is chosen another load pin is located at the front left corner of the plate. The analytic rigid cylindrical surface is available for *Abaqus* and *Warp3D* with some small differences.

To apply the 3-point bending boundary conditions, activate the load pin locations in the Load Pin Options menu on the top and bottom or left and right sides of the plate, then use the “Load Pin” menus at those locations, shown in the figures below. The load pin menu choices include custom displacements, custom forces, and the analytic rigid cylinder surface.

On the left and right sides of the plate, the custom displacement is in the global x-axis direction, and can have a non-zero value for an imposed displacement to move the load pin. For example, on the right side of the plate, the center load pin location would have a negative x-axis displacement to bend the plate to the left. Or the left side load pins would have a positive x-axis displacement to bend the plate to the right. The custom force is also in the global x-axis direction. The custom displacement and custom force choices distribute the constraint or force along the line of nodes, across the mesh thickness y-axis direction, at the particular load pin location. On the left side of the plate the constraints or forces are applied at the left back and left front mesh edge; the “Left Side Load Pins” menu applies the same boundary condition to one or both load pins on the left side of the mesh depending on the mesh symmetry.

The load pins on the top and bottom surface are oriented along the global x-axis (load pin cylinder axes in the x-axis direction). The cylinder geometry of the load pins is modeled using analytic rigid surfaces in *Abaqus*. These analytic rigid surfaces contact the top and bottom mesh surfaces of the plate model. Currently, only *Abaqus* supports an imposed displacement applied to an analytic rigid surface, so the *Abaqus* FEA choice must first be selected in the Tools/Program Options/Analysis form.

In the Load Pin menus, select the Analytic Rigid Surface, then enter the data for the load pin displacement U_y , the load pin radius, and the load pin location. Enter data for both load pin locations. A load pin U_y displacement of zero will fix the load pin in place. A positive U_y value should be used for the bottom surface load pin to move the load pin upward and bend the plate about the global x-axis. The load pin location is measured from the crack plane in the +/- z-direction. For example, the top surface load pin locations are given at the pin distance in front (+z) and behind (-z) the crack plane. See the figures below.

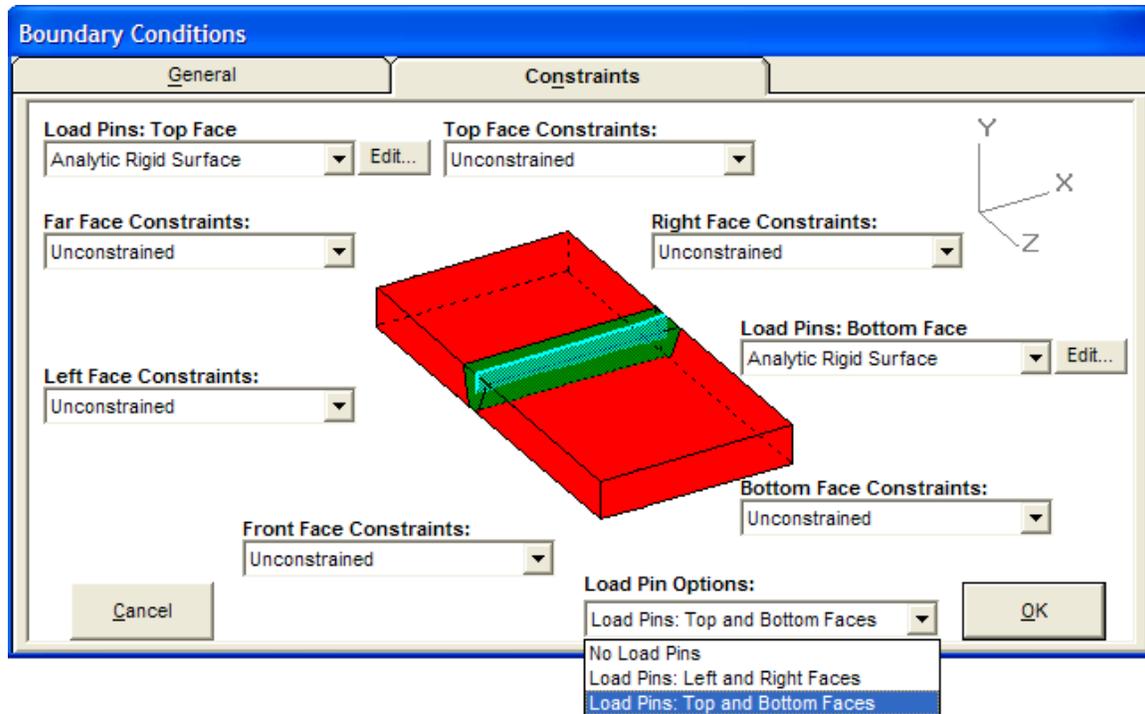


Figure 42 In the Load Pins Options menu select the load pins location or hide the load pin menus

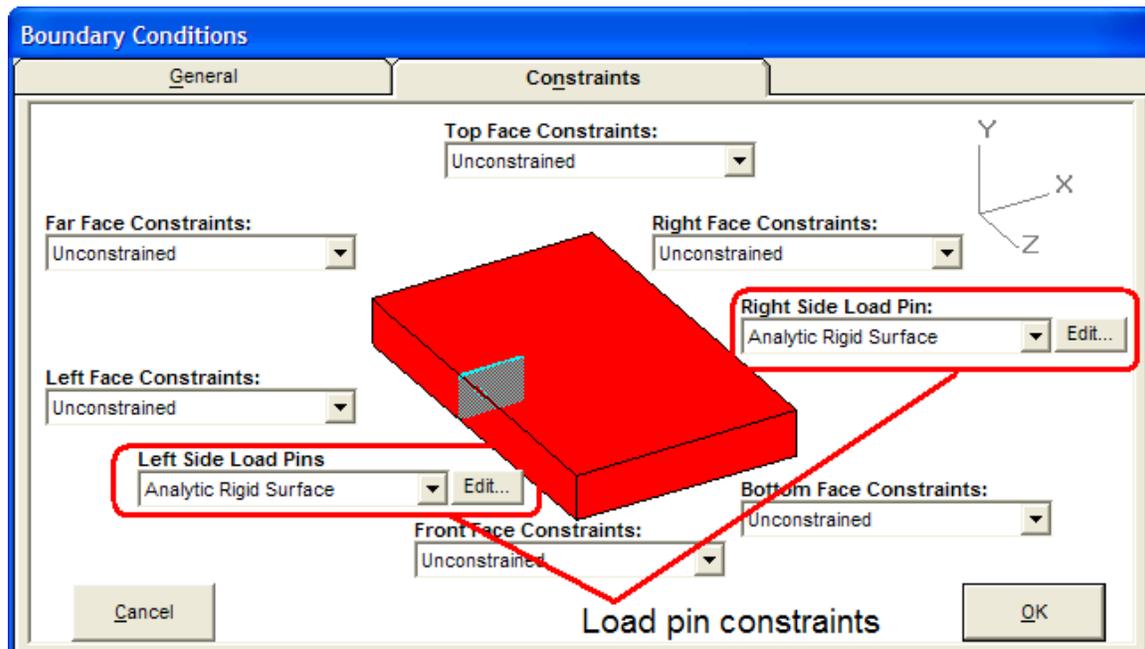


Figure 43 Load pin boundary condition menus on the right and left sides of the plate geometry

Analytic Rigid Surface

The load pin can be modeled as a rigid cylinder surface that imposes constraint on the flat plate by contact to the mesh. The load pin radius is entered to define the cylinder surface. The radius

can be different for the right and left side load pins. The right side load pin is located at the crack plane. For the left side load pins, enter the distance from the crack plane to the load pin to locate the center of the rigid cylinder at the back left and front left sides of the plate model.

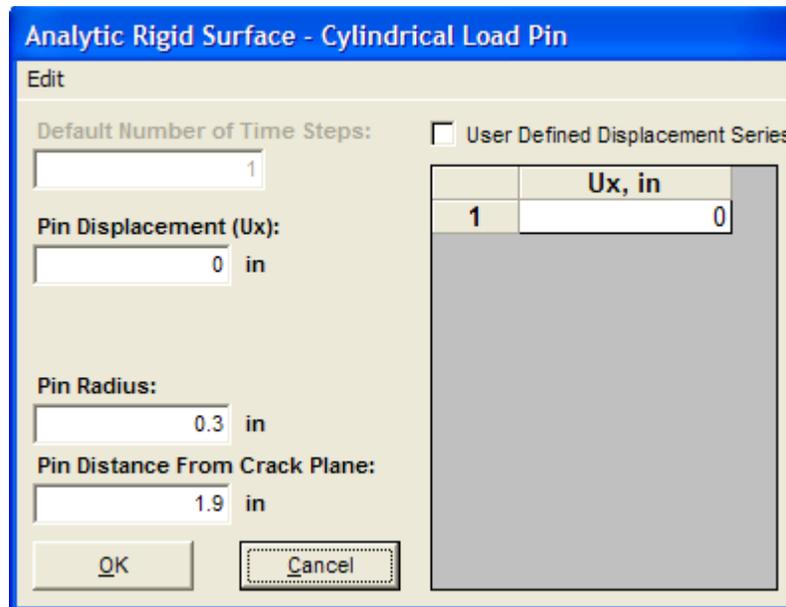


Figure 44 Load pin analytic rigid cylindrical surface data

For an *Abaqus* input file, the rigid cylinder surface has a reference node with 6 degrees-of-freedom (DOF). All the DOF are constrained to constrain the motion of the rigid cylinder load pin. The rigid surface reference node x-axis DOF can have an imposed displacement value applied to move the load pin and cause the plate to bend (Pin Displacement U_x value shown in the figure above). The load pin U_x value has the same effect as the custom displacements choice described above. General contact in *Abaqus* is used for the contact between the rigid surface and the corresponding mesh surface.

For a *Warp3D* input file the rigid cylinder surface does not have a reference node, so it cannot have a displacement value applied to the rigid surface, which means the rigid surface is stationary and fully constrained. For a *Warp3D* input file it is recommended to use the rigid cylinder surface load pin only on one side of the flat plate. For example, select the rigid surface on the right side of the plate, and then impose custom displacements or custom forces on the left side of the plate for 3-point bending.

The cylinder analytic rigid surface is not available for an *ANSYS* input file. Use the custom displacements and custom forces instead to apply boundary conditions at the load pin locations.

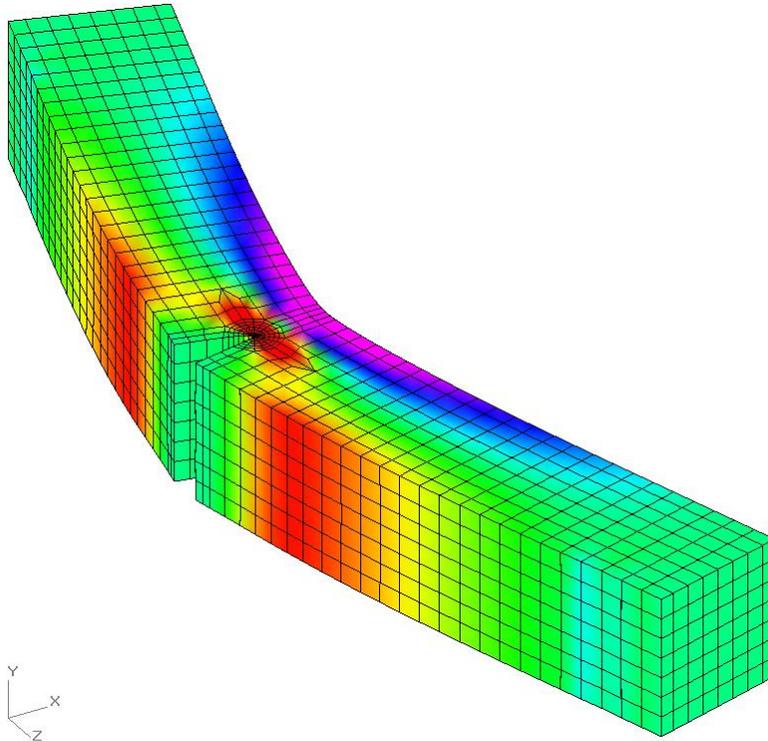


Figure 45 Edge crack plate model results with 3-point bending using the load pin constraints

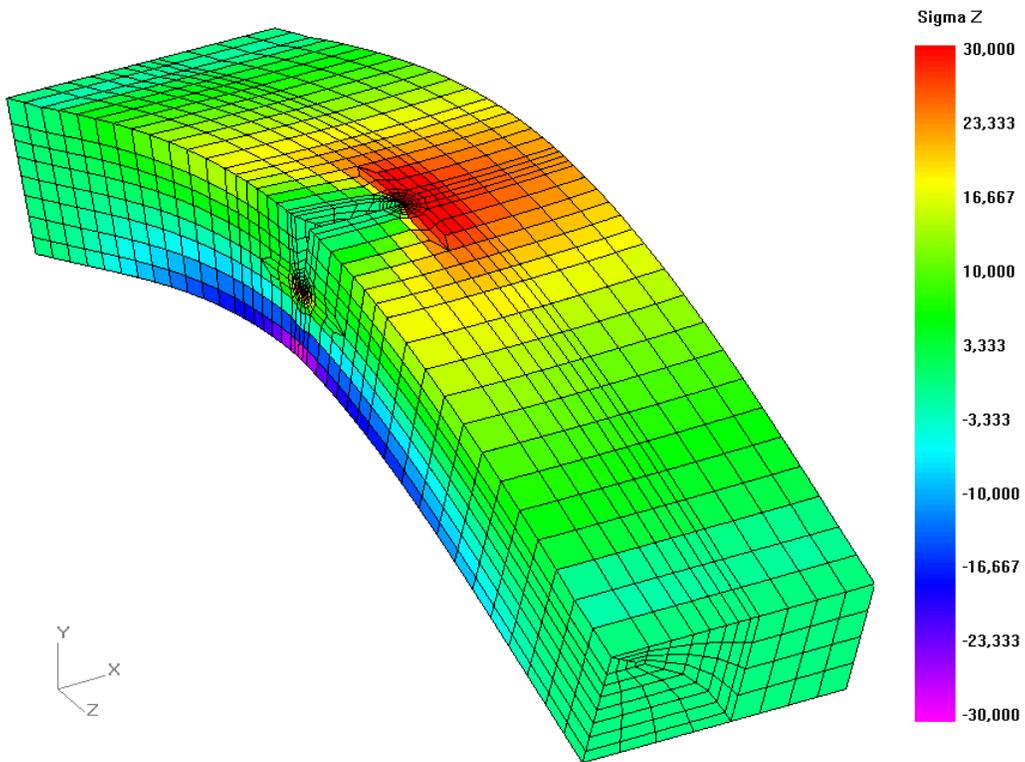


Figure 46 Corner crack plate with 3-point bending using load pins on the top and bottom surfaces

Load Steps

Specify the number of load steps for the analysis. The appropriate number of load steps depends on whether the analysis is elastic or elastic plastic. This choice is made in the “Materials input”.

For an elastic analysis, one load step is sufficient unless you are interested in multiple load cases (e.g., pressure only, pressure + nozzle loads, etc.).

If you are running an elastic-plastic analysis, multiple load steps are normally required for convergence of the solution. The number of required load steps for a plasticity analysis depends on the final load level. More load steps are recommended if the analysis goes well into the plastic range, where large plastic strains are attained. More load steps are also recommended if FAD and reference stress are desired in the results post processing.

The maximum values of applied loads must be chosen carefully. If applied loads are too high, an elastic-plastic analysis may not converge. If you encounter convergence difficulties with a particular model, the magnitude of the applied loads is the likely culprit. If the reason for the convergence difficulties is not immediately apparent, try running an elastic analysis on the model and look for locations of high von Mises stress when viewing the results. If an elastic analysis produces von Mises stresses well in excess of yield, one or more of the applied loads are probably too high.

The screenshot shows the 'Boundary Conditions' dialog box with the 'General' tab selected. The 'Number of Time Steps' is set to 10. The 'Load Steps To Save To Output File' table is as follows:

Step	Save
Step 1	ON
Step 2	ON
Step 3	ON
Step 4	ON
Step 5	ON
Step 6	ON
Step 7	ON
Step 8	ON
Step 9	ON
Step 10	ON

Figure 47 set the number of time steps and select steps with output

Load Increment Values

The “Iteration/Increment Values table” will only appear on the “General Boundary Conditions form” if *Abaqus* or *ANSYS* have been selected in the “Program Options form” (Tools menu); the

table does not appear for *WARP3D* since the sub step increments are automatically controlled. When using *Abaqus* to run the analysis, the user can set the initial load step increment sizes for each analysis step; refer to the *STATIC command in the *Abaqus* Keywords Manual. When using *ANSYS* to run the analysis, the user can set the number of iterations, or sub steps, for each analysis step; refer to the NSUBST command in the *ANSYS* manual. The default increment or iteration values are obtained by leaving the “User Defined check box” unchecked above each column of the table.

Internal Pressure Input

Shell-type geometries, such as spheres, cylinders, elbows, and nozzles, include an internal pressure boundary condition. For the default input, specify the maximum pressure, and the table of pressure versus load step will be filled in automatically with monotonically increasing values. For the user-defined input, the values and sequence of the pressure input are arbitrary. Note that the pressure need not increase monotonically with successive load steps. Pressure may increase and then decrease in one or more cycles. For plasticity analyses, however, make sure that you include sufficient intermediate load steps in a given cycle.

The pressure thrust (i.e. the equivalent axial load due to pressure) is automatically applied to ends of the model that are not otherwise constrained in the axial direction; see the option to omit the automatic pressure thrust load. See below for additional information on applying appropriate constraints to a model.

For internal surface cracks and through-wall cracks, the pressure load is automatically applied to the crack face(s); see the option to omit the automatic crack face pressure.

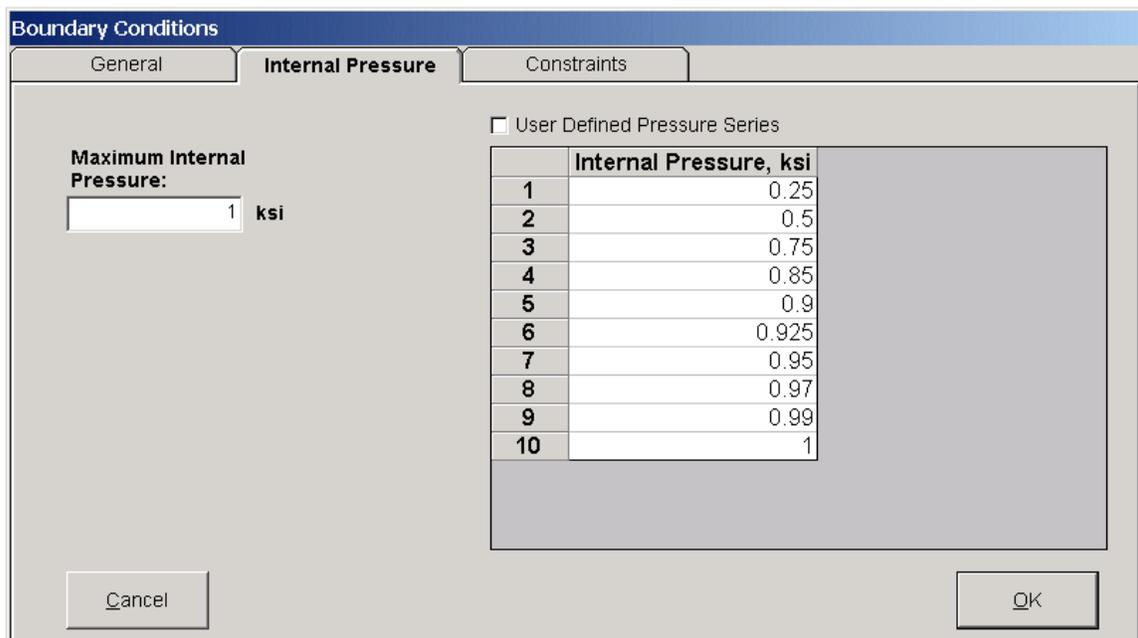


Figure 48 use the default load step values or enter load step values

Crack Face Pressure

For elastic analyses with a single load step, the user may apply uniform or non-uniform pressure to the crack face. Such loading is customarily applied when generating stress intensity solutions for handbooks or compendia. A stress distribution from an un-cracked analysis can also be applied as a crack face pressure, thereby taking advantage of elastic superposition. Uniform, linear, quadratic, and cubic distributions are available with the “Equation Based Crack Face Pressure Loading option”. A polynomial distribution may be applied by superimposing one or more power-law terms. The “Tabular Crack Face Pressure Loading option” allows the user to input an arbitrary pressure distribution.

The screenshot shows the 'Boundary Conditions' dialog box with the 'Crack Face Pressure' tab selected. Under 'Crack Face Traction', the 'Uniform' and 'Quadratic' options are checked. The 'Quadratic' option is selected, with the equation $\sigma(x) = \sigma_2 \left(\frac{x}{a}\right)^2$ displayed. The coefficient σ_2 is set to 0.25 ksi. A diagram on the right shows a crack with a stress distribution curve $\sigma(x) = \sigma_n \left(\frac{x}{a}\right)^n$. The 'General' tab is also visible, and the 'Cancel' and 'OK' buttons are at the bottom.

Figure 49 enter crack face pressure polynomial values

Tabular Crack Face Pressure

In addition to the crack face pressure polynomial, crack face pressure can be defined by using a 2-D table. The tabular crack face pressure allows the user to define an arbitrarily varying crack face pressure along the crack length and crack depth directions.

To select the tabular crack face pressure, check the “Tabular Crack Face Pressure Loading” option on the “General” tab of the “Boundary Conditions” form. Then on the “Crack Face Pressure” tab enter the transposed crack face pressure table. In the transposed table the table columns are along the crack depth direction (from 0 to a) and the table rows are along the crack length direction (from $-c$ to c or from 0 to c depending on the mesh symmetry). Using a transposed table allows for many rows of crack face pressure values along the crack length direction; the spreadsheet table can accommodate many more rows than columns; the spreadsheet table is limited to 256 columns but can have 16000 rows.

For example, a cylinder with an external axial crack has crack face traction defined with the tabular crack face pressure. It may be easier to begin with the initial crack pressure table with the table columns along the crack length direction and the table rows along the crack depth direction to the cylinder axial crack case. This initial crack pressure table would be aligned with the global x and y coordinates, with x along the crack length direction (initial table columns), and y in the crack depth direction (initial table rows). In the table below, an example crack face pressure table is given; the initial table has 4 rows and 5 columns.

	c=0.0	c=0.7	c=2.0	c=6.0	c=10.0
a=0.0	100	95	80	70	60
a=0.5	85	92	75	65	55
a=0.6	30	25	20	15	10
a=1.0	12	11	9	7	3

Figure 50 Example of the initial crack face pressure table

The column index values are along the crack length direction beginning at the center of the crack. At $c = 0.0, 0.7, 2.0, 6.0, 10.0$; the mesh will use symmetry and only include the right half of the crack. The row index values are along the crack depth direction beginning at $a = 0.0, 0.5, 0.6, 1.0$. The column and row index values define the locations of the crack face pressure magnitude values given in the table. The pressure applied to each element within the crack face will be obtained from the given tabular pressure values by 2-D interpolation.

When using *WARP3D*, the pressure at the center of an element face is used as the load on that element. The element face pressure will vary across the crack to follow the given pressure table values. When using *Abaqus*, the DLOAD user subroutine is automatically created by the mesh generator and included in the input file. The user subroutine provides the crack face pressure for each integration point on an element face allowing the pressure to vary across an element face in the analysis.

Note that the crack face pressure table can extend beyond the crack length or crack depth. If multiple crack sizes are given (multiple crack meshes), the crack pressure table should be defined with index values past the largest crack length and largest crack depth. If the same distribution across the crack for each crack size is desired, the crack face pressure table must be updated to match each crack size. For example, the desired crack face pressure varies from zero to the same maximum value along the crack length (or crack depth). In that case the index values are updated from $-c$ to c (or from 0 to a) for each unique crack size with the same maximum pressure values in the table.

The transposed crack face pressure table is given in Table 2. The transposed table has 5 rows and 4 columns. The transposed table columns are along the crack depth direction (from 0 to a) and the transposed table rows are along the crack length direction (from $-c$ to c or 0 to c). The values from the transposed table are entered in the “Crack Face Pressure” spreadsheet in *FEACrack*. Note that a crack face pressure table can be copied and pasted from a spreadsheet program (like *EXCEL*) in to *FEACrack*. The transposed crack pressure table corresponds to a transposed crack orientation as shown in the figure below.

	a=0.0	a=0.5	a=0.6	a=1.0
c=0.0	100	85	30	12
c=0.7	95	92	25	11
c=2.0	80	75	20	9
c=6.0	70	65	15	7
c=10.0	60	55	10	3

Figure 51 Transposed crack face pressure table

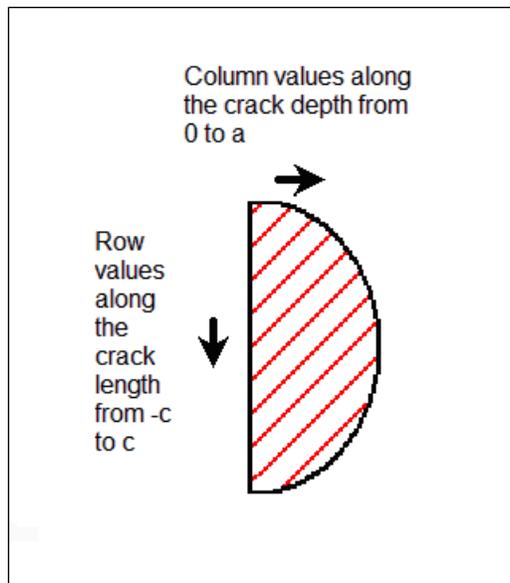


Figure 52 Transposed Crack Orientation

An example is available to view the tabular crack face pressure load values. Open the “Crack_Tabular_Ex.elt” model into *FEACrack* (available in the example file download, contact us for the download link). Then view the tabular crack face pressure values in the boundary conditions form. The crack face pressure table that is entered in *FEACrack* does not have the “a=” or “c=” labels used for clarity in the above tables, just the index values. Enter the values in the table below into *FEACrack* for the tabular crack face pressure. The crack face pressure table data is also available from the “Crack_Face_Pressure_Table_Example.xls” *EXCEL* spreadsheet file.

	0.0000	0.5000	0.6000	1.0000
0.0000	100.00	85.00	30.00	12.00
0.7000	95.00	92.00	25.00	11.00
2.0000	80.00	75.00	20.00	9.00
6.0000	70.00	65.00	15.00	7.00
10.0000	60.00	55.00	10.00	3.00

Figure 53 Transposed crack face pressure table to enter in FEACrack

To validate the tabular crack face pressure, results can be compared from an analysis using the polynomial crack face pressure. For example, crack dimensions of $a = 0.7$ and $c = 1.2$ with linear crack face pressure, set the maximum pressure value equal to the σ_1 value in the polynomial crack

face pressure form. The table below shows the transposed crack face pressure table that gives a linear crack face pressure where the maximum pressure occurs at the full crack depth.

	a=0.0	a=0.7
c=0.0	0	1000
c=1.2	0	1000

Figure 54 example of the pressure table for linear crack face pressure

The table below gives the pressure table that can be entered in *FEACrack* (index labels removed).

	0.0	0.7
0.0	0	1000
1.2	0	1000

Figure 55 example of the transposed tabular crack face pressure to enter in *FEACrack*

Omit Crack Face Pressure

For pressure containing geometries, such as the cylinder, sphere, elbow, spherical head and perforated cylinder, the pressure load is also applied to the crack face depending on the crack location. An internal crack will have the internal pressure automatically applied to the crack face, and an external crack will have the external pressure automatically applied to the crack face.

To deactivate the default and not automatically apply pressure to the crack face, use this key word in the Home/Notes form:

*use omit auto crack pressure

Omit Pressure Thrust Load

For pressure containing geometries, such as the cylinder, elbow, and nozzles, the pressure load also contributes to the axial pressure thrust load. To deactivate the default and not automatically include the pressure thrust load, use this key word in the Home/Notes form:

*use omit auto axial pressure

Other Loads and Constraints

The Constraints tab on the boundary conditions window provides options for applying various loads and constraints on the model.

Certain loads and constraints are automatically applied by *FEACrack*. For example, pressure thrust (axial membrane stress due to pressure) is automatically applied to a nozzle or open cylindrical shell that is not constrained in the direction of the stress. See the option to omit the automatic pressure thrust load. When the user specifies a symmetry plane, the mesh generator automatically applies the appropriate displacement constraint. For simple geometries such as cylinders and flat plates, all necessary constraints for normal loading conditions are applied automatically. In a ¼ -symmetric flat plate, for example, symmetry conditions on two faces constrain rigid-body motion in the x-z horizontal plane, and a constraint at a single node restricts

rigid-body motion in the vertical (y) direction. For more complicated shapes, however, more care is required on the part of the user to ensure that a model is properly constrained, as discussed below.

Loads and moments can be applied to a nozzle or to a shell, as long as the shell is not closed on the end where the loads are applied. Note that an axial force applied to the end of a shell structure is added to the pressure thrust, which is automatically applied by the software. Therefore, the *pressure thrust should not be included in user-specified forces*.

Input of user-specified forces moments, and displacements works in much the same way as the pressure input. If you select the default input, you will be prompted for the maximum values, and the intermediate values at each load step will be filled in automatically. Alternatively, you can manually input the applied loads and moments. As was the case with pressure input, the applied loads need not increase monotonically throughout the analysis. Moreover, the applied loads need not increase or decrease in phase with one another. For example, you can first apply internal pressure and then apply loads and moments. If you are running an elastic-plastic analysis, however, make sure that each step size is sufficiently small to ensure convergence.

Note that the model must be properly constrained in order to obtain a valid solution. A certain amount of knowledge and expertise on the part of the user is required, at least for more complicated shapes and load cases. Examples of proper and improper boundary conditions are illustrated on the next two pages.

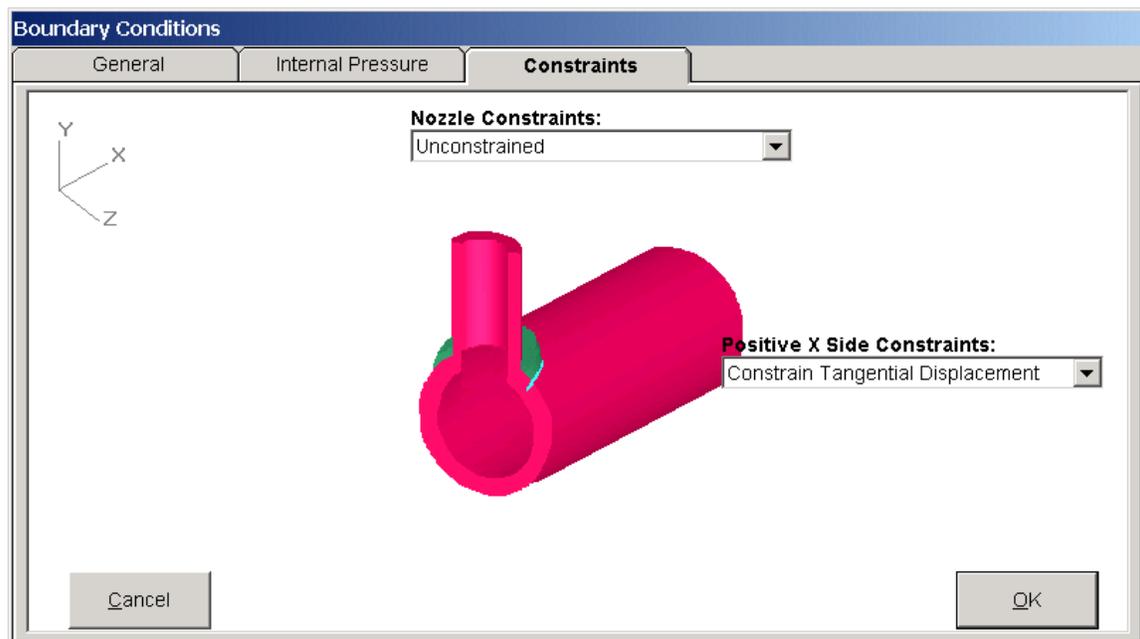


Figure 56 enter constraint values at available locations

When applying constraints, forces, or stresses, the values are given in the global coordinate system. For example, when applying an axial stress to a cylinder geometry, the tensile (positive) direction is in the x-axis direction, but for a compressive (negative) stress a negative stress value must be entered to have the stress applied along the negative x-axis direction. For a flat plate geometry, a tensile stress is in the negative z-axis direction on the back side of the plate. Any rotation, moment, or bending stress values are positive right-hand-rule about each global axis as

shown in the figure below. For example, a positive bending stress value about the z-axis applied to the right end of a cylinder will cause compression on the top of the cylinder and tension on the bottom of the cylinder. A negative bending stress value about the z-axis will cause tension on the top of the cylinder and compression on the bottom of the cylinder. Using the deformed shape with the results is helpful to confirm that the applied boundary condition values cause the intended direction of displacement and deformation of the mesh.

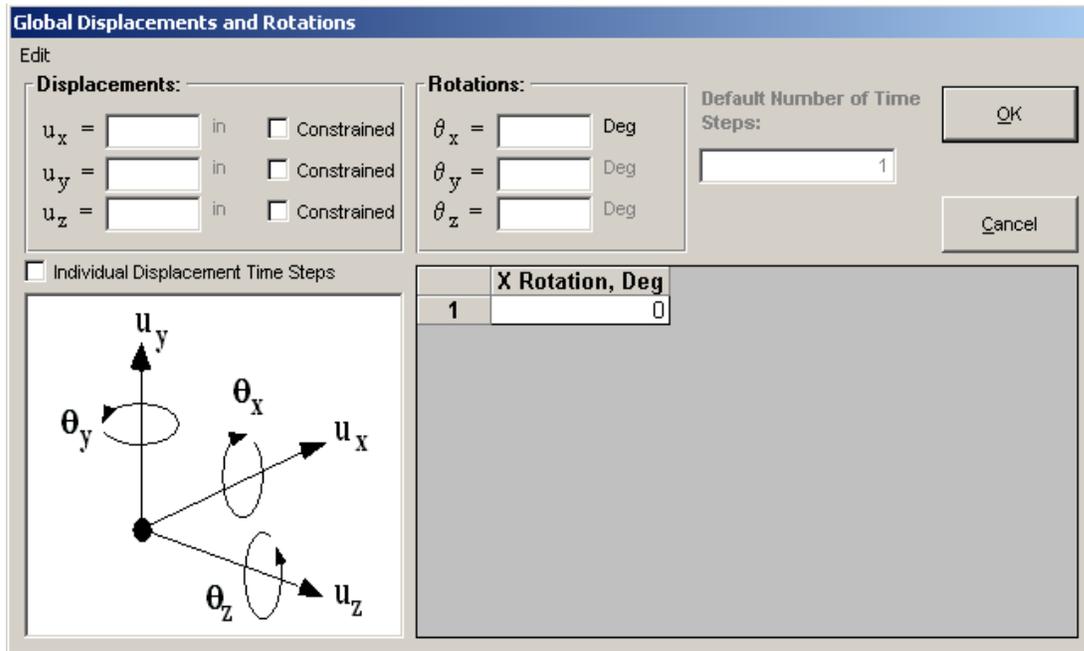


Figure 57 enter displacement constraints; enter a value for a non-zero constraint

User Defined Load History

During post processing of elastic-plastic analysis results, a load history is needed to compute the FAD and Reference Stress tables. For geometries in the standard geometry library, the load history is automatically obtained from the given boundary conditions, and the user defined load history is not needed. For a user-defined geometry, a load history may not have been defined for the crack mesh, particularly if the user defined crack mesh has been added to a larger model. Refer to the later section on creating user defined geometry crack meshes. To aid the elastic-plastic post processing of a user defined geometry, one or more user defined load histories can be entered in the UD Load tab. One load history is sufficient for post processing, additional load histories will allow result plots to be made versus other values. Enter a text label for each load history (up to 10 characters) at the top of each column. Enter the load history values in the table; these must be positive monotonic increasing values. If more than one load history is given, the values should all follow the same amplitude history. The values increase at the same rate, see the figure below. The user defined load history values will be included in the load amplitude table within the J-node Data comment block in the input file. Including the comment data block in the combined crack mesh input file makes the user defined load history available for the elastic-plastic post processing.

Boundary Conditions

General Constraints **UD Load**

Number Of User Defined Loads: (max of 6)

	MyPressure	RemoteLoad	user_label
1	12.5	2500	0.25
2	25	5000	0.5
3	37.5	7500	0.75
4	42.5	8500	0.85
5	45	9000	0.9
6	46.25	9250	0.925
7	47.5	9500	0.95
8	48.5	9700	0.97
9	49.5	9900	0.99
10	50	10000	1

 Only The First 10 Characters Of Any Load Label Can Be Written To The FEA Input File

Figure 58 user defined load history values, used in post-processing plot x-axis

Characteristic Stress

If the “characteristic stress” value is given in the comment data block, the post processing will compute the non-dimensional stress intensity “G” values from the stress intensity results. A characteristic load may be automatically selected from the given loads for a single load. For multiple loads the default characteristic stress will be zero.

To set a characteristic stress value for post processing, in the Boundary Conditions form activate the “User Defined Load History” check box. In the UD Load tab use the specific label “charstress” for one of the user defined load histories. If a default characteristic stress is not found the given charstress value will be used. The user defined loads are not applied to the mesh, just used for post processing.

Properly Constrained Models:

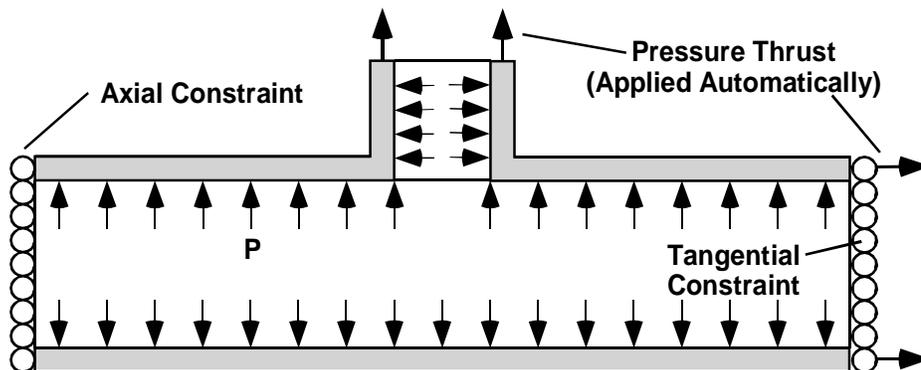


Figure 59 constraint example

Loading from internal pressure only.

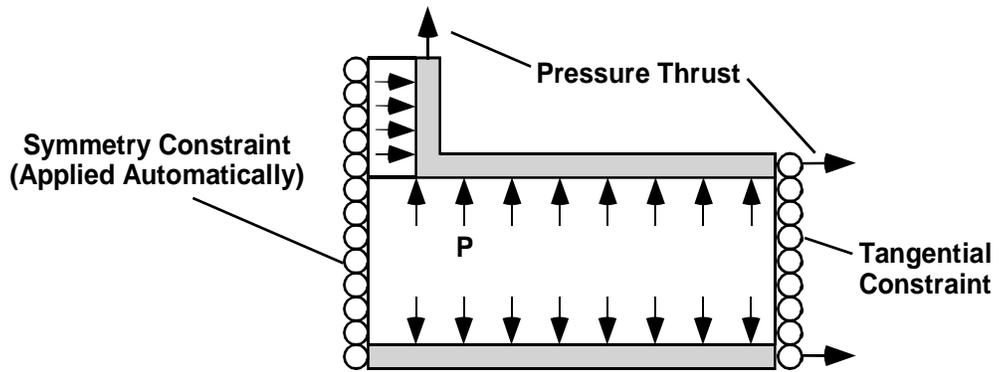


Figure 60 constraint example

Same as above, but with a symmetry plane.

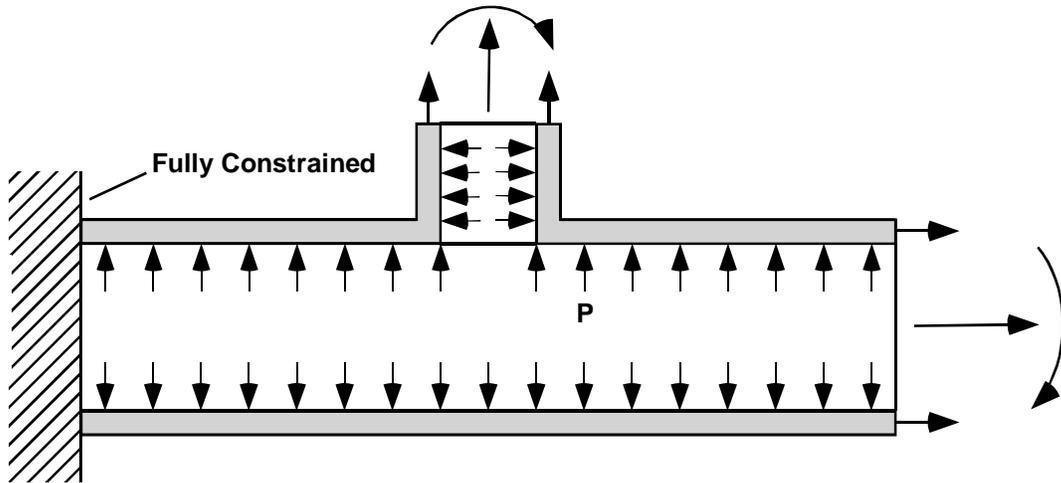


Figure 61 constraint example

Applied loads on nozzle and shell.

Improperly Constrained Models:

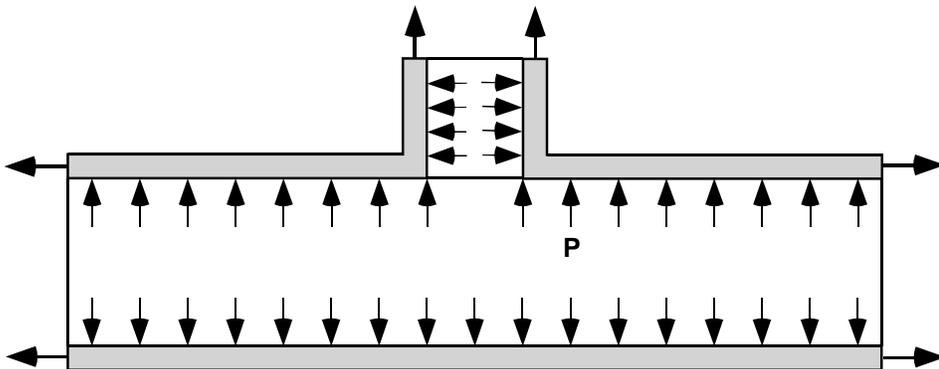


Figure 62 constraint example

Underconstrained. Although the applied forces balance, there must be at least one constraint to prevent rigid body translation.

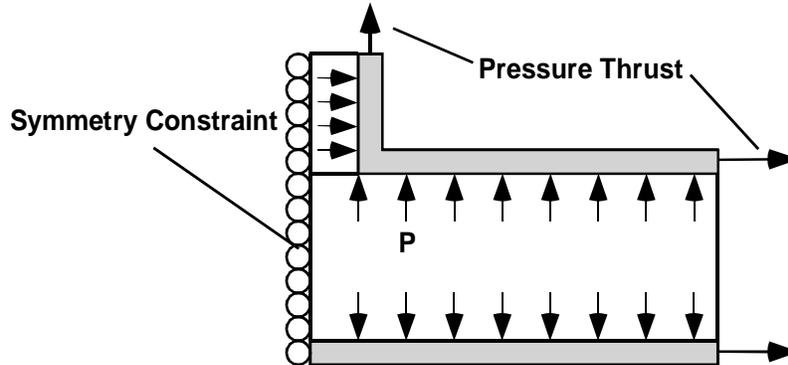


Figure 63 constraint example

Underconstrained. Same as above with symmetry condition.

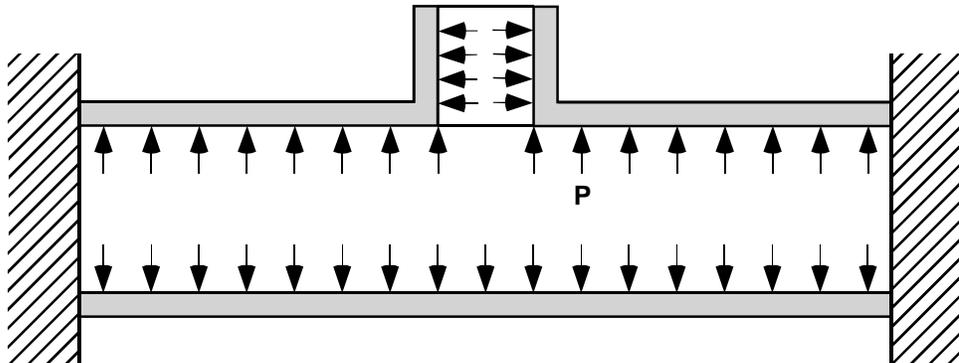


Figure 64 constraint example

Overconstrained. Fixing both ends of the shell prevents axial pressure thrust from being applied.

ANSYS Submodel Node File

When the “Submodel Boundary” condition choice is selected in the Boundary Condition/Constraints form for one or more mesh surfaces in a user-defined geometry, see the picture below, the submodel boundary nodes can be written to an ANSYS node file to aid in the submodel analysis. A node set is written in the ANSYS input file that contains the submodel boundary nodes on the selected mesh surfaces. The submodel method is also known as the cut-boundary displacement method. For more information refer to the ANSYS help documentation in “Mechanical APDL”, “Advanced Analysis Techniques Guide”, Chapter 9 Submodeling.

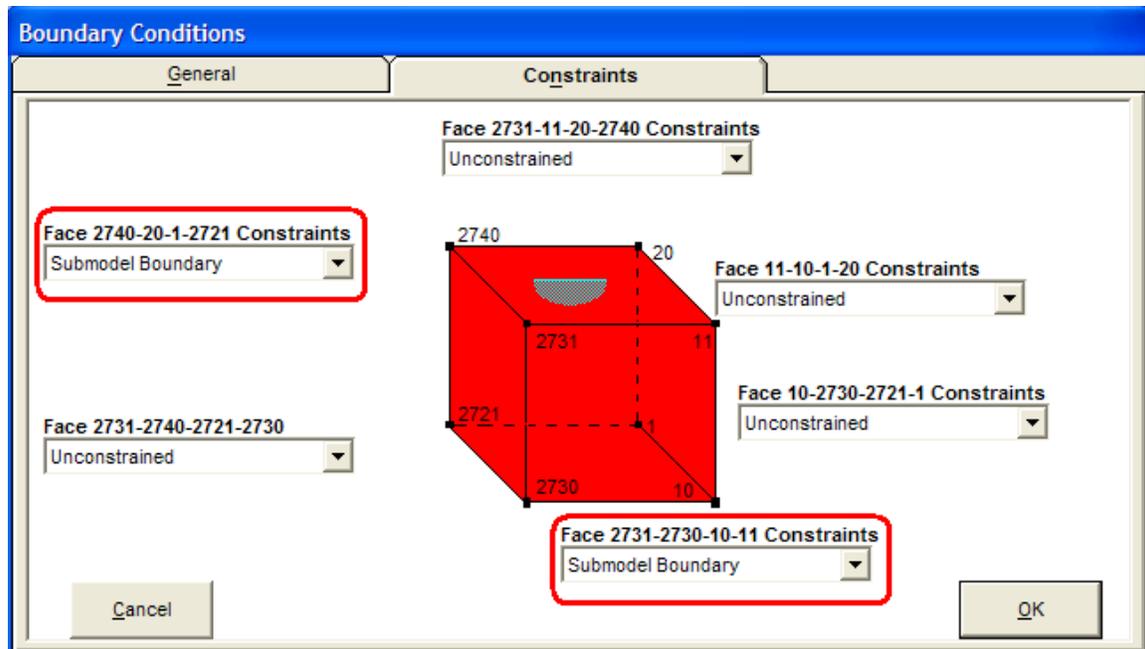


Figure 65 Select the “Submodel Boundary” for one or more mesh surfaces

To activate the ANSYS node file, include this text in the Home/Notes form, and see the picture below:

*use ansys node file

The ANSYS node file name is based on the given model name: <model name>_ans.node
 The node file can be viewed or edited with a text editor, such as Notepad.

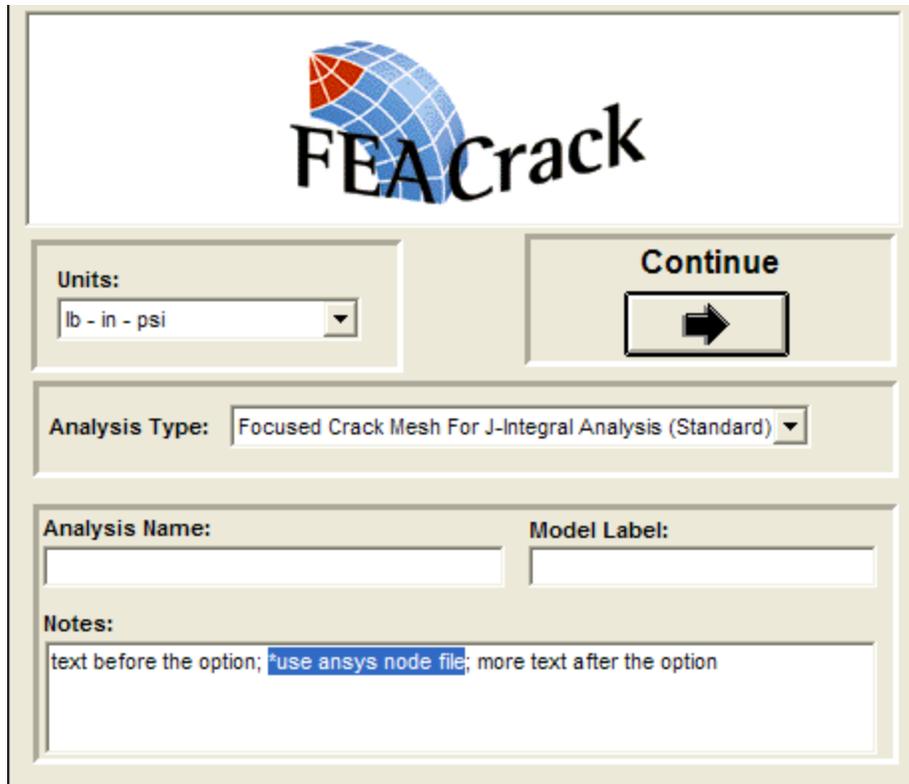


Figure 66 Include key text: “*use ansys node file” in the Notes to activate the ANSYS node file

The submodel boundary nodes are written to the ANSYS input file as a single node set. The node set is given a component label at the end of the node list; the syntax and format are given as:

```
!
!   SUBMODEL NODE SET
!   number of nodes = 204
!   submodel nodes also written to file: D:\Analysis\model_name_ans.node
!
NSEL, S, NODE,,    261
NSEL, A, NODE,,    262
NSEL, A, NODE,,    263
NSEL, A, NODE,,    264
...
NSEL, A, NODE,,    7398
NSEL, A, NODE,,    7399
CM, submodel_nodes, NODE
```

Note that when the node file is active, the node file name is listed in the comments before the node set in the input file. The ANSYS node file is written using the same format as specified for the ANSYS NWRITE command. On each line there is an 8-digit integer for the node ID number, followed by 20-digit floating point numbers for the node coordinates. As an example:

```
261 14.59556007385 14.59556007385 3.358757019043
262 14.97056007385 14.97056007385 2.828427076340
263 14.21055777657 14.97066220867 3.358757019043
264 14.57567073489 15.35529851185 2.828427076340
```

ANSYS Submodel Analysis Global File

When the “Submodel Boundary” condition choice is selected in the Boundary Condition/Constraints form for one or more mesh surfaces in a user-defined geometry, the ANSYS global model file name can be set using a key word option so that the ANSYS submodel analysis can be run from within FEACrack. The crack mesh is the submodel and should be located within the appropriate region of the global model volume using the definition mesh.

Define the ANSYS global model file name, without a file extension, using the keyword:

```
*use global file = <filename>
```

Where <filename> is the ANSYS global model file name (no file extension), which is included in the ANSYS input file submodel syntax. Both the filename.db database file and filename.rst results file should exist before running the ANSYS submodel analysis. Use this global file key word with the previously described “*use ansys node file” key word option as the *.node file is needed for the submodel node displacement constraints during the submodel analysis..

ANSYS Submodel Analysis Global Results Step

When the global model results file has more than one results step, the specific result step that the submodel should use can be specified with the keyword:

```
*use global step = <step>
```

Where <step> is the integer value to select the result step number. For example, if the submodel should use result step 3 from the global results to get the interpolated boundary node displacements the syntax: *use global step = 3

Tabular Temperature: Apply A Custom Thermal Gradient

A tabular temperature data file can be used to apply a custom through thickness temperature gradient to the flat plate and cylinder geometry in FEACrack. The tabular temperature option is activated by key word text in the Home/Notes form. The overall temperature boundary condition is activated in the Boundary Conditions form. The temperature value in the Boundary Conditions/Temperature tab is used as a multiplier to adjust the amplitude of the tabular temperature values applied to each node in the mesh.

The tabular temperature option is activated by key word text in the FEACrack Home/Notes form, shown below. The key word text is:

```
*use tabular temperature file = <file name>
```

Where <file name> is the data file name containing the tabular temperature data. The data file is expected to be located in the same directory as the *.elt model file. This option can be used with other key word options by using the semi-colon “;” separator between the key word text.



FEACrack

Units: lb - in - psi

Continue

Analysis Type: Focused Crack Mesh For J-Integral Analysis (Standard)

Analysis Name:

Model Label:

Notes: *use tabular temperature file = temperature table.txt

Figure 67 Key word text in the Home/Notes form to activate the tabular temperature option

The format of the tabular temperature data file includes a title at the top of the file, the number of values in the table, column labels, and two columns of position and temperature values. An example of a tabular temperature data file is shown below.

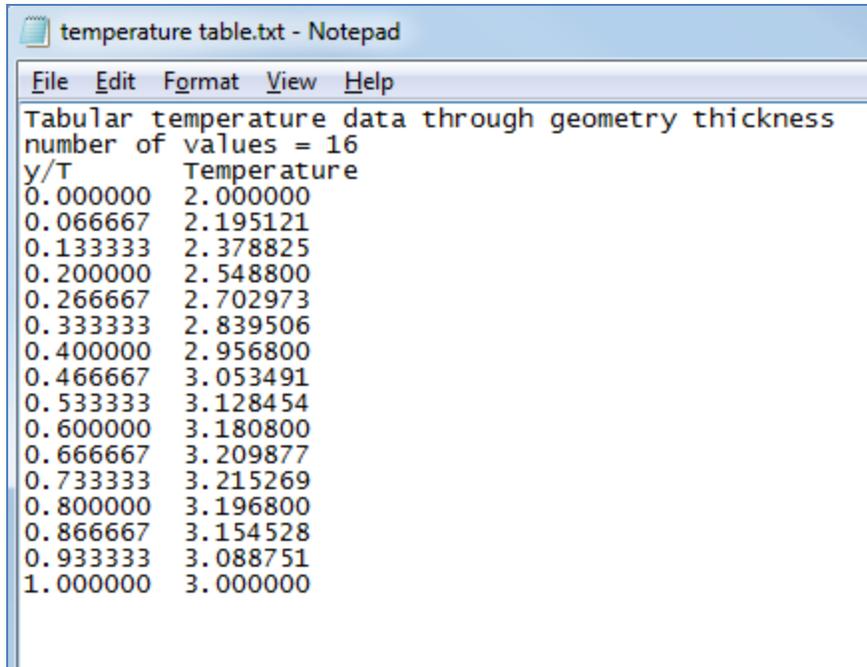


Figure 68 Example of a tabular temperature data file

The title line must begin with “Tabular temperature data” to identify the contents of the file for the crack mesh generator. The syntax for the number of values in the table is:

number of values = <number>

Where <number> is the integer number of rows of data in the table.

There is a label for each table column. The first column label text is expected to be “y/T” to identify the position ratio values, and the second column label text is expected to be “Temperature” to identify the temperature values.

The position ratio gives the non-dimensional location of each temperature value through the thickness of the mesh. The y/T ratio is expected to have values from 0.0 to 1.0. The temperature values are used as a multiplier for the temperature values given in the model file data.

To activate the temperature boundary conditions in the crack mesh, check the “Apply a temperature boundary condition” option box in the Boundary Conditions/General form, shown below.

Use the table of temperature values per analysis step in the Boundary Conditions/Temperature form as a multiplier to adjust the magnitude of the tabular temperature profile, shown below.. The number from the form is multiplied with the temperature value at each node interpolated from the tabular temperature data to get the nodal temperature value.

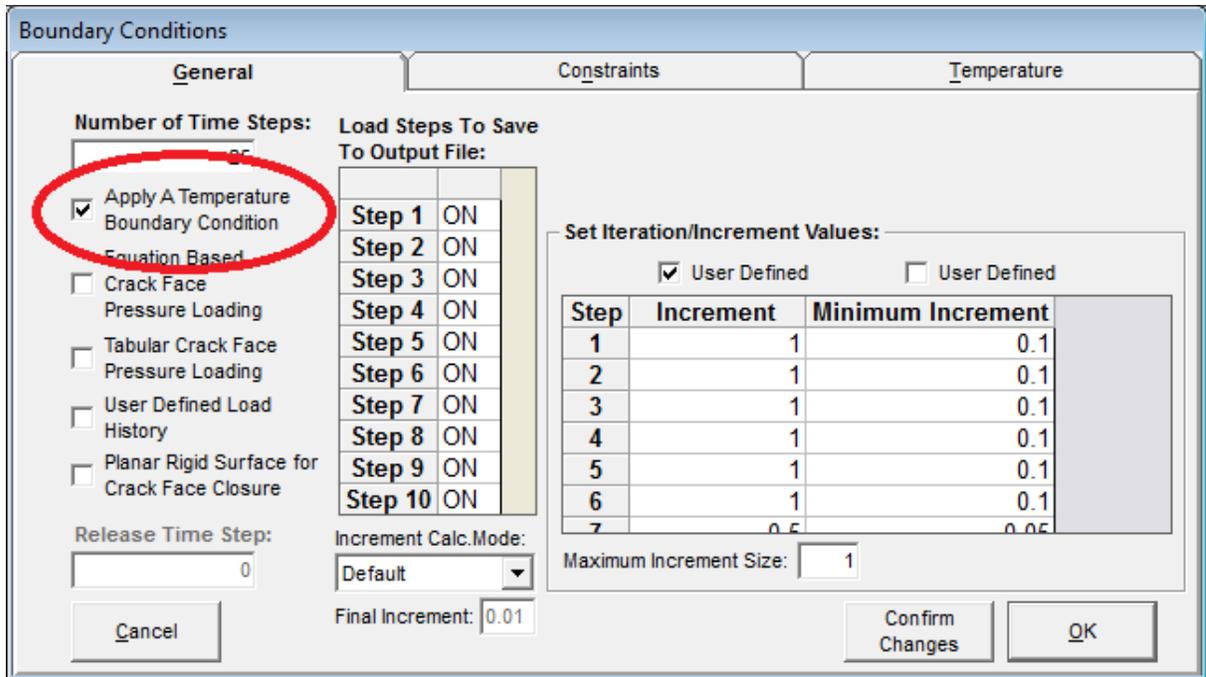


Figure 69 Check the temperature option to activate the temperature boundary condition

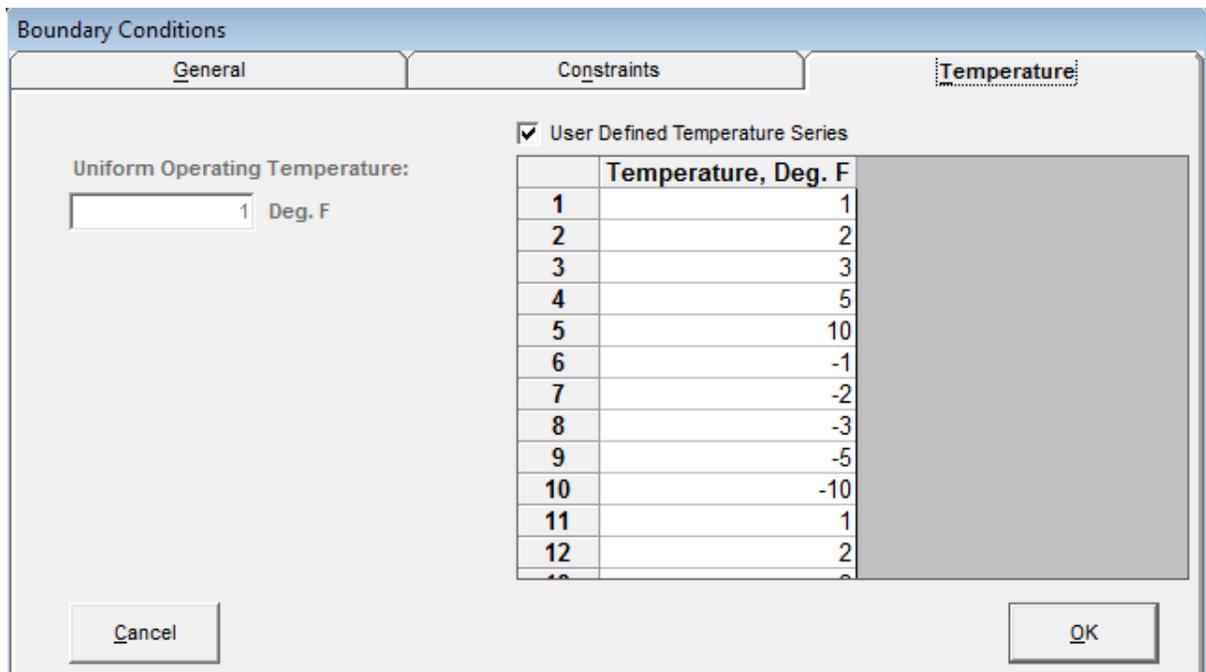


Figure 70 Use the temperature values as a multiplier for each analysis step

The temperature value at each node is written as a boundary condition in the input file.

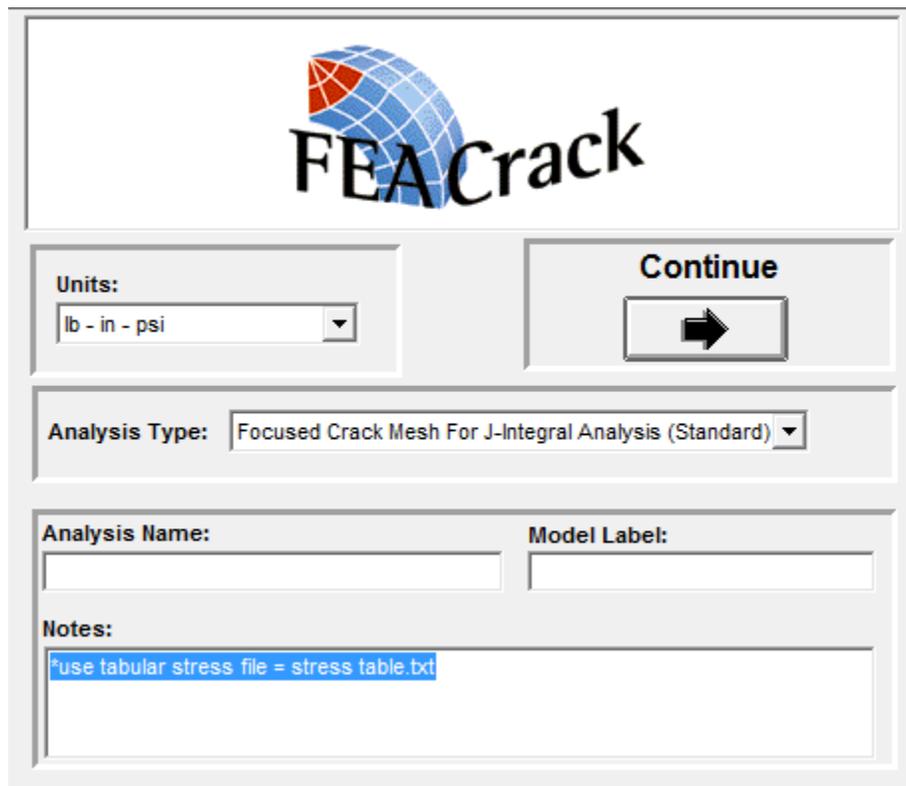
Tabular Stress: Apply A Custom Stress Gradient

A tabular stress data file can be used to apply a custom through thickness stress gradient to back face of the flat plate geometry. The tabular stress option is activated by key word text in the Home/Notes form. The overall stress boundary condition is activated in the Boundary Conditions/Constraints form. The stress value in the Boundary Conditions/Constraints tab is used as a multiplier to adjust the amplitude of the tabular stress values applied to each node on the back face of the plate geometry mesh. The equivalent nodal forces are computed from the given tabular stress distribution, and the nodal forces are used to apply the mesh surface traction loading.

The tabular stress option is activated by key word text in the FEACrack Home/Notes form, shown below. The key word text is:

`*use tabular stress file = <file name>`

Where <file name> is the data file name containing the tabular stress data. The data file is expected to be located in the same directory as the *.elt model file. This option can be used with other key word options by using the semi-colon “;” separator between the key word text.



The image shows a screenshot of the FEACrack software interface. At the top, there is a logo for FEACrack featuring a blue globe with a red and orange crack pattern. Below the logo, there are several input fields and buttons. On the left, there is a 'Units:' dropdown menu with 'lb - in - psi' selected. To the right of this is a 'Continue' button with a right-pointing arrow. Below these, there is an 'Analysis Type:' dropdown menu with 'Focused Crack Mesh For J-Integral Analysis (Standard)' selected. Further down, there are two empty text input fields labeled 'Analysis Name:' and 'Model Label:'. At the bottom, there is a 'Notes:' section with a text area containing the key word text: `*use tabular stress file = stress table.txt`.

Figure 71 Key word text in the Home/Notes form to activate the tabular stress option

The format of the tabular stress data file includes a title at the top of the file, the number of values in the table, column labels, and two columns of position and stress values. An example of a tabular stress data file is shown below.

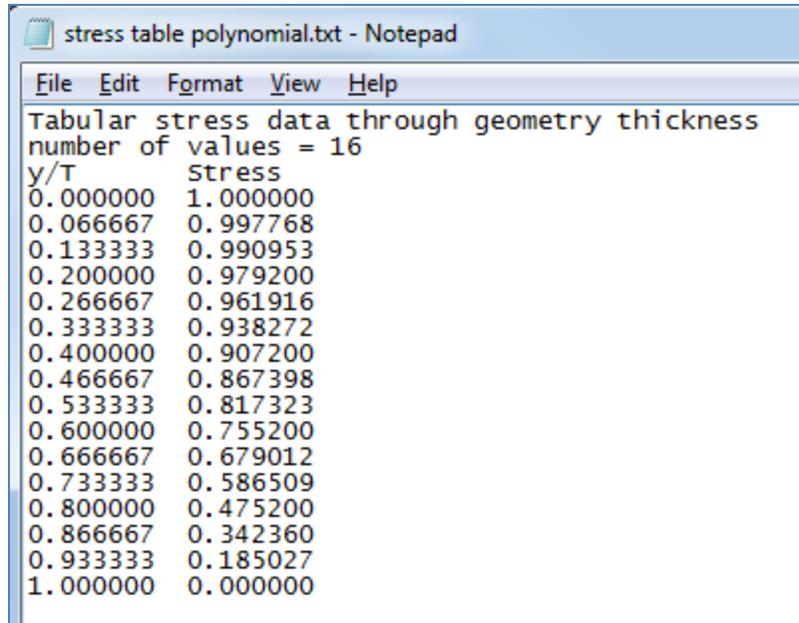


Figure 72 Example of a tabular stress data file

The title line must begin with “Tabular stress data” to identify the contents of the file for the crack mesh generator. The syntax for the number of values in the table is:

number of values = <number>

Where <number> is the integer number of rows of data in the table.

There is a label for each table column. The first column label text is expected to be “y/T” to identify the position ratio values, and the second column label text is expected to be “Stress” to identify the stress values.

The position ratio gives the non-dimensional location of each stress value through the thickness of the mesh. The y/T ratio is expected to have values from 0.0 to 1.0. The stress values are a multiplier of the membrane stress given in the model file data.

To activate the stress boundary conditions in the crack mesh, select “Custom Stress” in the Boundary Conditions/Constraints “Far Face Constraints” menu for the flat plate geometry. Check the “Apply a temperature boundary condition” option box in the Boundary Conditions/General form, shown below.

Use the table of membrane stress values per analysis step in the Boundary Conditions/Constraints/Custom Stresses form as a multiplier to adjust the magnitude of the tabular stress profile, shown below.

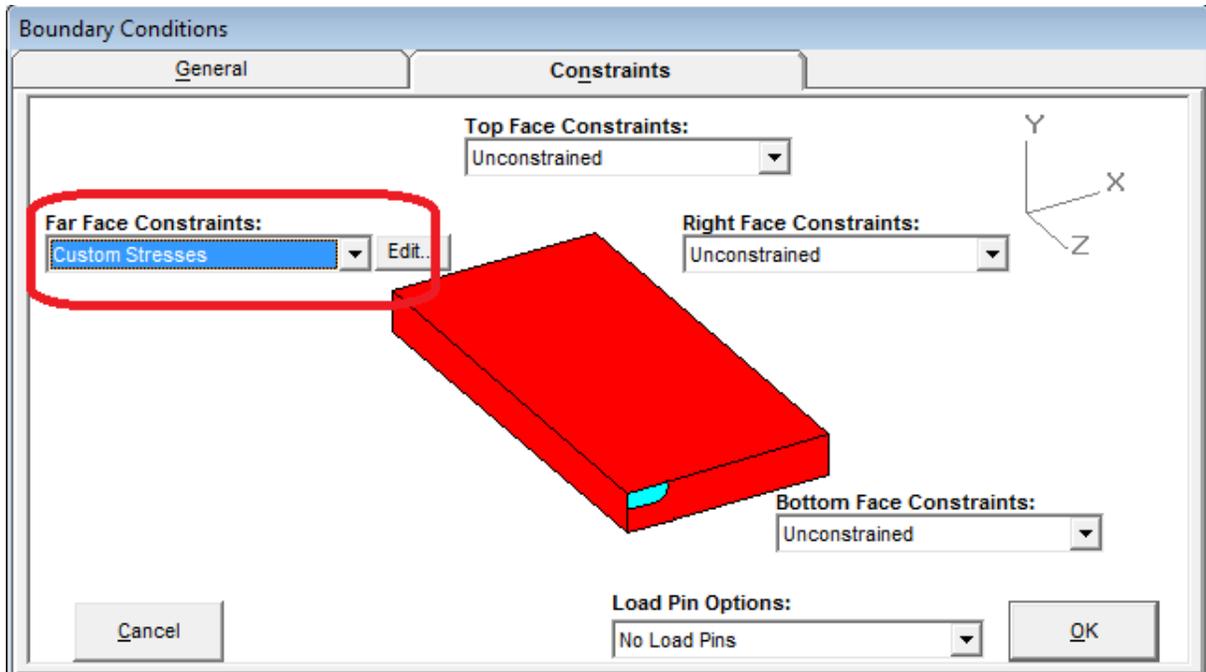


Figure 73 Select Custom Stresses to activate the stress boundary condition

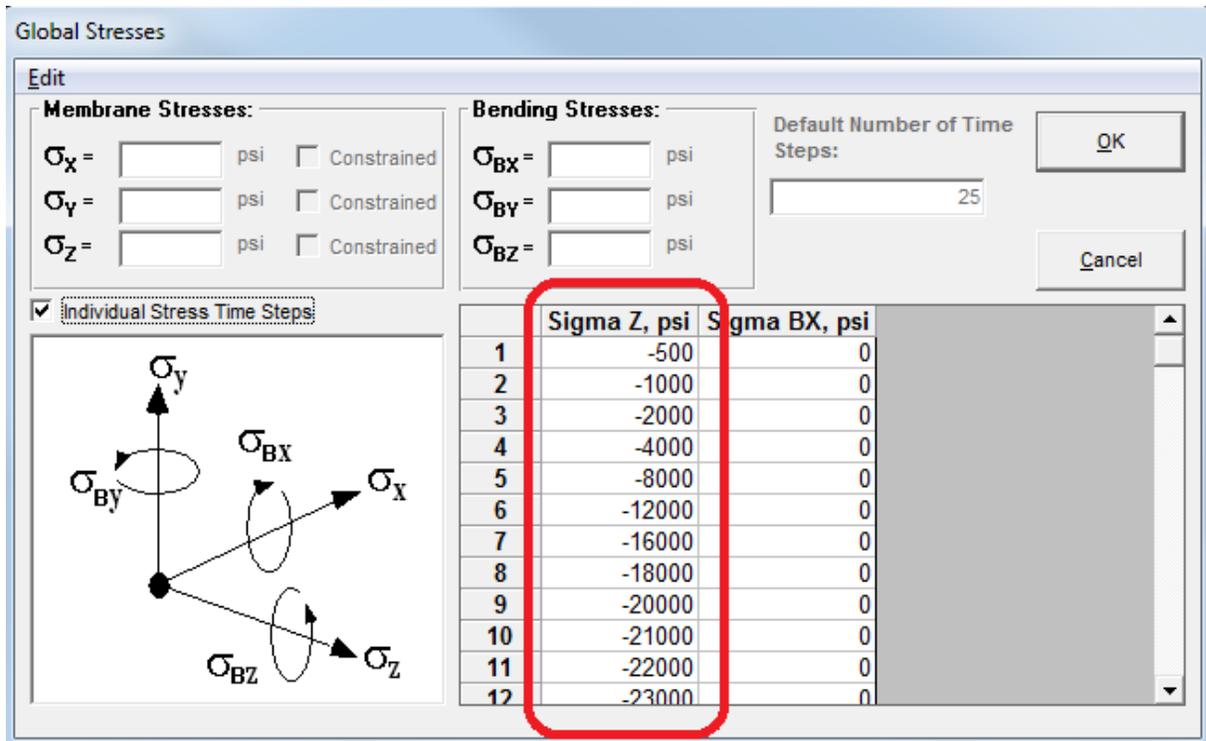


Figure 74 Use the membrane stress values as a multiplier for each analysis step

The membrane stress value from the form is multiplied with the tabular value at each y-coordinate location interpolated from the tabular stress data. The y-coordinate locations are obtained from the element face Gauss integration point locations. The total stress value is used to compute the equivalent nodal forces through the thickness on the back face of the plate mesh.

Material Properties

Three material models are currently available on the Material Properties, Material Model Type form:

- Elastic material behavior
- Deformation plasticity
- Incremental plasticity

For plasticity analyses, a stress-strain curve must be input. For incremental plasticity, the following three options are available:

- Linear plus power law
- Ramberg-Osgood equation
- Tabular stress-strain input

For the deformation plasticity option, *Warp3D* uses the linear plus power law stress-strain behavior, and *Abaqus* uses the Ramberg-Osgood power-law. Deformation plasticity in *Warp3D* supports keywords including: deformation, mises, and cyclic. To activate these options, first select the *Warp3D* choice in the Tools/Program Options/Analysis form; see the figure below. Then on the Material Properties form select the Deformation Plasticity option, shown in the figure below. When the *Warp3D* “cyclic” key word is selected, a second tab is activated to enter additional material properties, shown in the figure below. Refer to the *Warp3D* manual, section 3.10 for more information. *ANSYS* only supports incremental plasticity.

Certain structural components, such as those that include welds, provide for separate material groups, where different properties can be assigned. For example, nozzle structures consist of three material groups: shell, nozzle, and weld material groups. These material types are automatically assigned to the appropriate locations in the model. The different material groups are color-coded when the mesh is displayed.

For each material type, you can specify different elastic and elastic-plastic material coefficients (if applicable). If the analysis is elastic-plastic but you wish for one material to remain elastic, simply set its yield strength to a sufficiently high value, or select the “Assume Elastic Behavior” option on the material form for a specific material group.

Note that the input window for each material type contains a drop-down list of materials. The materials database contains elastic constants for each material, as well as typical yield and tensile strengths. The tensile properties from the database are used to estimate Ramberg-Osgood coefficients for the material of interest. Alternatively the user can input his/her own tensile properties and allow the software to compute Ramberg-Osgood coefficients.

Plate Material Properties

Material Properties

Plate Material: A36 Plt; LA, HHA, 3% NACl
 A36 Plt; LA, HHA, 3% NACl

Material Model Option: Mises

Use Defaults
 User Specified
 Auto Calc Plasticity

Assume Elastic Behavior
 Assume Elastic-Plastic Behavior

Young's Modulus: 30000000 psi
 Poisson's Ratio: 0.3
 Yield Strength: 44000 psi
 Tensile Strength: 78000 psi

Type of Plasticity Data:
 Linear Plus Power Law
 Ramberg-Osgood Model
 Stress-Strain Curve

Stress Free Reference Temperature: 0 Deg. F
 Coefficient of Thermal Expansion: 0 1/(Deg F)

$\frac{\epsilon}{\epsilon_0} = \frac{\sigma}{\sigma_0}$ Elastic Range

$\frac{\epsilon}{\epsilon_0} = \left(\frac{\sigma}{\sigma_0}\right)^n$ Fully Plastic Range

Reference Stress: 44000 psi
 Hardening Exponent (n): 5.983402
 Fitting Constant (alpha): 1.363636

Cancel OK

Figure 75 material data form for Warp3D to select the Mises material option

Plate Material Properties

Cyclic Parameters

H_bar (kinematic hardening parameter):

rho (mass density):

Gamma (kinematic hardening pressure):

Q_bar (isotropic hardening modulus):

b (isotropic hardening modulus):

ns_fact (subincrementation factor):

sig_tol (subincrementation tolerance):

Cancel OK

Figure 76 Warp3D cyclic material parameters

Material: Abaqus Stress-Strain Hardening Option

Incremental plasticity in *Abaqus* supports keywords including: isotropic, kinematic, and combined. To activate these options, first select the *Abaqus* choice in the Tools/Program Options/Analysis form. Then on the Material Properties form select the Incremental Plasticity option and the stress-strain curve option, shown in the figure below.

Then on the shell material properties form, select the keyword in the Hardening Model Option menu, shown in the figure below. The corresponding keyword, “Isotropic”, “Kinematic”, or “Combined”, will be written to the *Abaqus* input file.

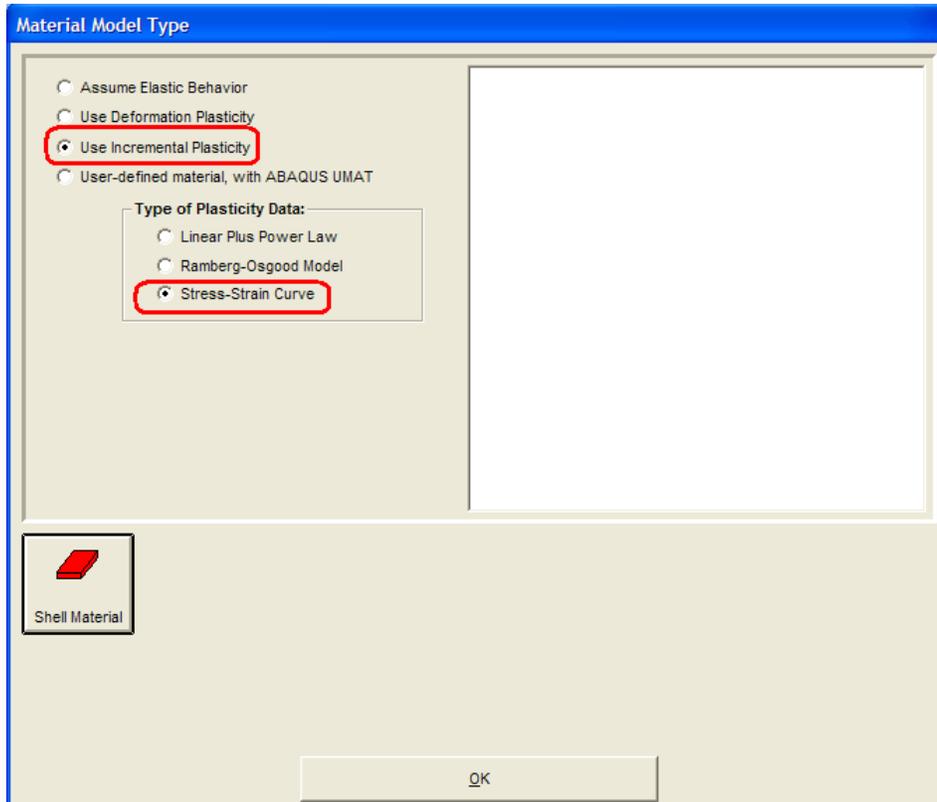


Figure 77 Select incremental plasticity and the stress-strain curve for Abaqus hardening options

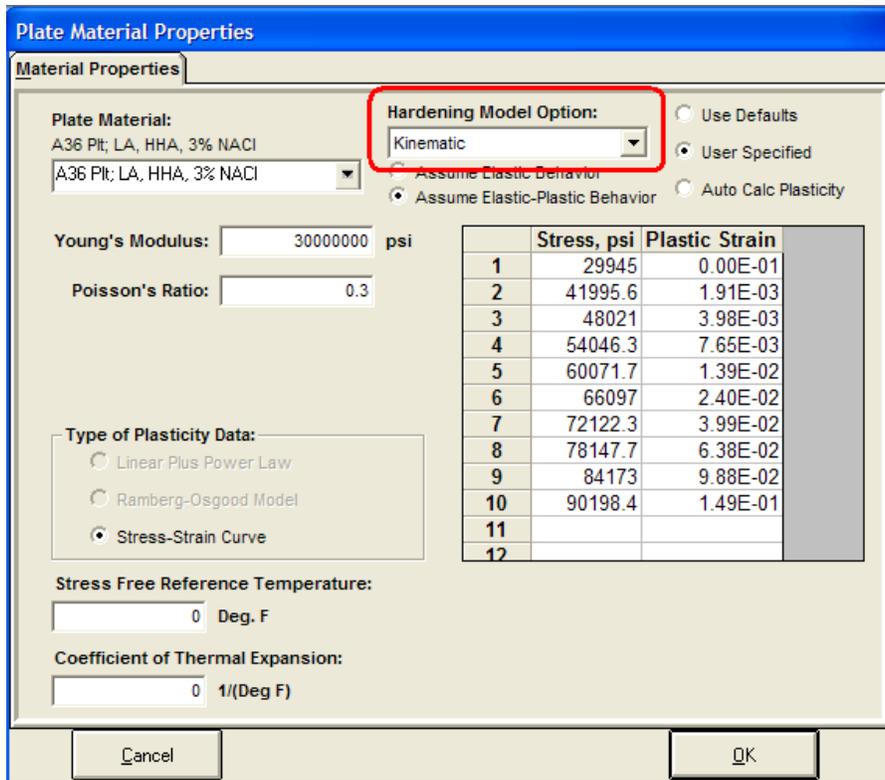


Figure 78 Select the hardening key word for the Abaqus input file

Material: User-Defined Abaqus UMAT

In addition to the elastic and elastic-plastic material choices, a user-defined material can be selected when generating crack meshes for *Abaqus*. On the Material Properties form, select the “User-defined material, with *Abaqus* UMAT” option; see the figure below. An *Abaqus* UMAT user subroutine must also be provided to run the *Abaqus* analysis; refer to the “*Abaqus* User Subroutines Reference Manual”, section 1.1.31 for more information.

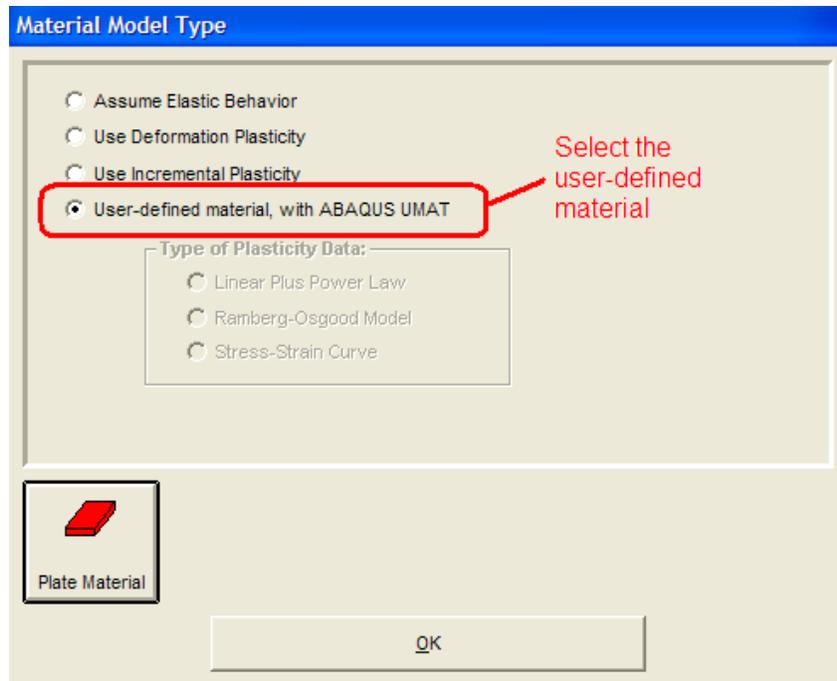


Figure 79 Select the user-defined material option on the Material Properties form

On the material data form, enter the number of dependent variables, the number of material constants, and the material constant values; see the figure below. The number of dependent variables should be a positive number corresponding to the particular *Abaqus* UMAT routine. Enter the number of material constants needed for the UMAT routine (expecting a positive number of constants), and enter the corresponding material constant values in the data spreadsheet. The number of rows in the data table is set by the number of material constants. An optional text label can be entered in the Name column to help identify each constant. The floating point material constants are entered in the Value column. Refer to the “*Abaqus* Keywords Reference Manual” and the *DEPVAR and *USER MATERIAL commands for more details.

Plate Material Properties

Material Properties

User Specified

Number Of Dependent Variables (DEPVAR):

Number Of Material Constants

	Name	Value
1	a	1.5
2	b	0
3	c	0.1
4	d	0.2

Stress Free Reference Temperature: Deg. F

Coefficient of Thermal Expansion: 1/(Deg F)

Figure 80 Enter the number of dependent variables and the material constants

The usual elastic or elastic-plastic data in the *Abaqus* input file is replaced with the user-defined material commands: *DEPVAR for the number of dependent variables, and *USER MATERIAL for the material constant values. When running the *Abaqus* analysis a UMAT Fortran subroutine must be used, which is provided by the user (see the *Abaqus* user manual for more information to link a user subroutine to an analysis).

An excerpt from an *Abaqus* input file shows an example of the user-defined material syntax generated by *FEACrack* from the values given in the Material Properties form. Note the absence of the usual *ELASTIC or *PLASTIC material commands.

```

**
**----- MATERIAL DATA -----
**
*SOLID SECTION, ELSET=PROP1, MATERIAL=MAT1
1.0
*MATERIAL, NAME=MAT1
**
** USER-DEFINED MATERIAL
** INFO: An Abaqus UMAT routine is needed for this analysis
**      number of user constants =      4
**      number of dependent variables =  92
**
*USER MATERIAL, CONSTANTS=4
1.0000E+00, 0.0000E+00, 0.0000E+00, 0.0000E+00
*DEPVAR

```

```
*EXPANSION, ZERO=0.00000000E+00  
0.00000000E+00, 0.00000000E+00
```

Material: Omit standard material syntax

The material data syntax written to the input file can be omitted (deactivated) by using this key word option in the Home/Notes form:

*use material data omit

The material data syntax should be provided in the input file using the master mesh file. For an Abaqus input file the macro text can also be used to add material data in the model data section. The combination of omitting the standard material data syntax and adding the desired syntax allows full customization of the material data used in the analysis.

Building and Viewing Models

Once the geometry, boundary conditions, and material properties have been specified, click on the Build Mesh button to run the mesh generator. You will be prompted at this point to save the *.elt file. Generating the finite element mesh is a highly computationally intensive task that typically takes several minutes. Of course, the time required to build a given model is a function of the size of the model and the speed of your computer.

The files that are created by the mesh generator depend on the settings in the Program Options control panel, which is accessed from the Tools menu. Check the box(es) corresponding to the file format(s) you wish to create. This selection must be made *before* you click the Build Mesh button. If none of the file formats are selected, you will be able to view the mesh, but no files will be written to disk.¹ All files will be written to the same directory as the current *.elt file. Keep this in mind when setting up your directory structure for multiple models.

Once the model has been built, a 3D rendering of the finite element mesh will appear on the screen, as shown below. Note that the number of elements and nodes is given below the zoom controls.

¹ If you have a license that limits the number of input files you may create, you should first build the mesh with all boxes unchecked. Once you are satisfied with the mesh, you can re-build the mesh and create the input file.

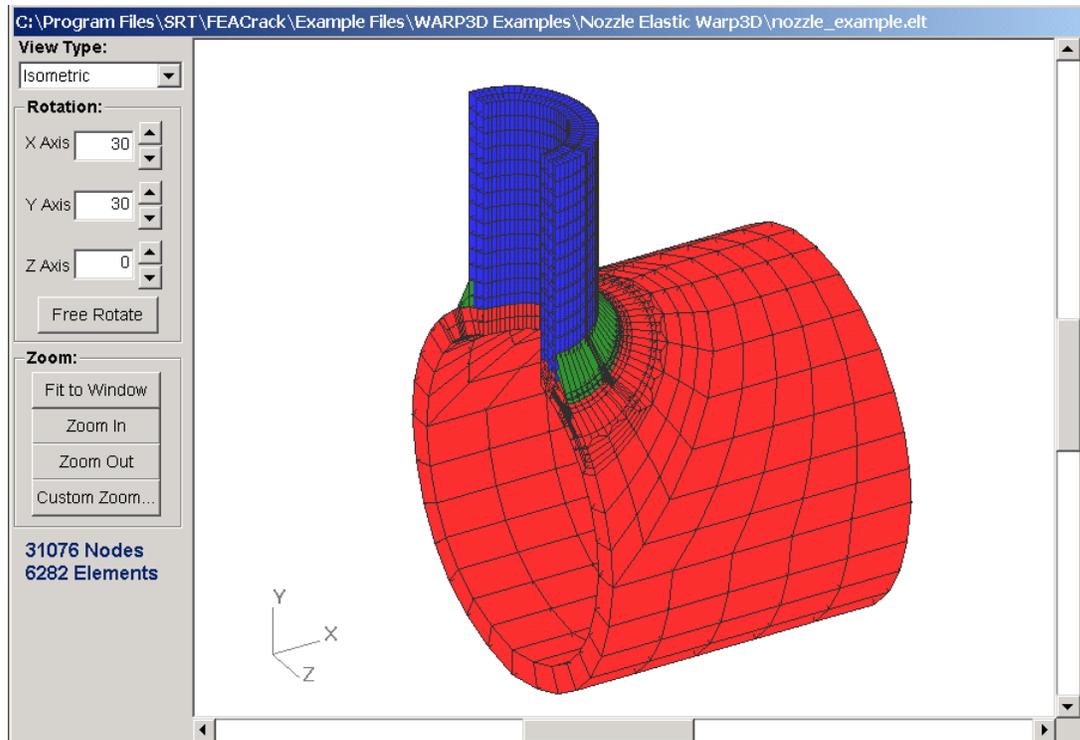


Figure 81 mesh view

You can change the view with zoom and rotate controls. The Zoom In button doubles the size of the mesh on the screen, and Zoom Out decreases the size by a factor of two. To zoom in on a specific area, use the following procedure:

1. Click on the Custom Zoom button. The mouse cursor will change to a +.
2. Move the cursor to the upper left-hand corner of the region of interest.
3. Drag a rectangle over the area of interest while holding down the left mouse button. Release the mouse button. An enlarged view of the region of interest will then be displayed.

Several default view angles are available in the View Type drop-down list. To obtain a custom view angle, click on the up and down arrows in the Rotation controls to change the viewing angle of the model. Alternatively, you can type in the desired rotation angles for the x, y, and z axes. A third rotation method involves clicking the Free Rotate button and moving the mouse over the model while holding down the left mouse button. When completed, click on the Stop Rotate button. The free rotate functionality can also be accessed by dragging the mouse over the model while holding down both the left mouse button and the R key. This free rotate method requires some practice to achieve the desired results.

The scroll bars on the bottom and right-hand side of the viewing window can be used to shift the relative position of the model on the screen. Alternatively, you can drag the model with the mouse by holding down the left mouse button and the Ctrl key.

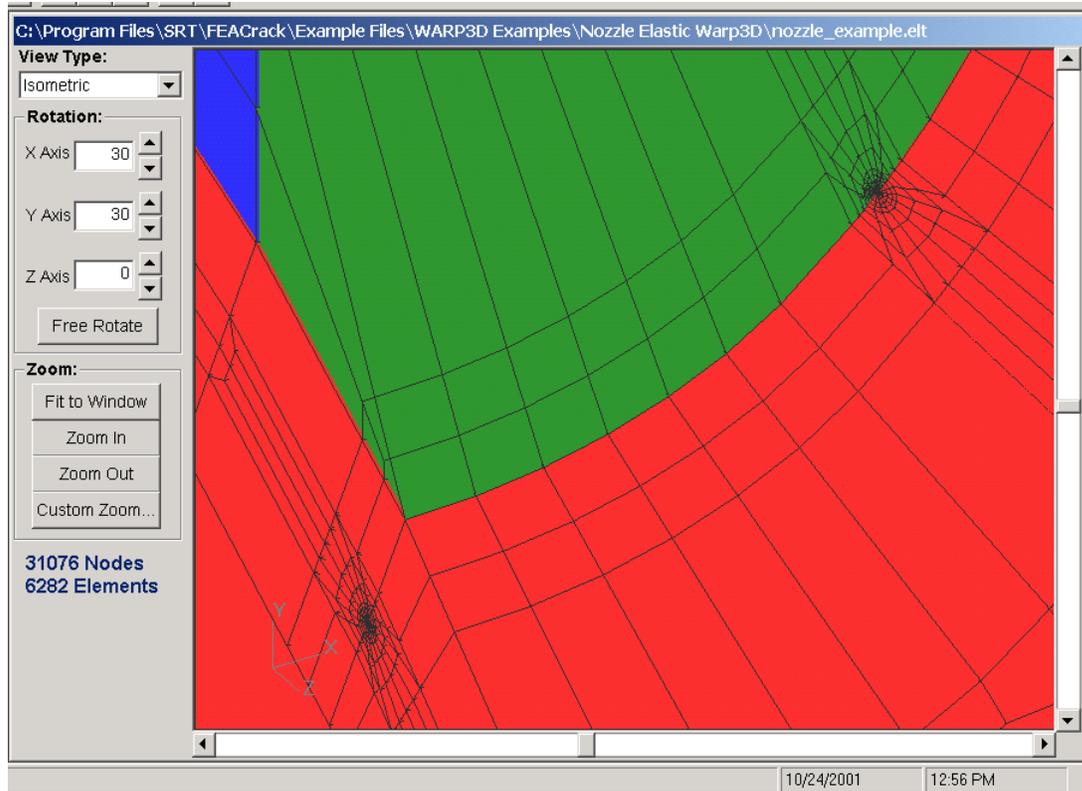


Figure 82 mesh view close up

Running Finite Element Analyses

FEACrack can run multiple finite element analyses (FEA) sequentially in batch mode. You can run many cases over-night, or a few cases during your lunch hour or during a meeting.

Note: If you have limited RAM, we recommend that you exit *FEACrack* after generating models, and re-launch the application immediately prior to running analyses. This will give *FEACrack* a smaller memory footprint, thus freeing up more memory for *WARP3D*, *Abaqus*, or *ANSYS*.

Click on the Run FEA icon to view the analysis control panel shown below. Find the folder(s) that contain the desired input file(s). Click on an input file to select it, and then click on the right arrow button to add the input file to the Run list (right side column of the form). To remove an input file from the Run list, select it and click on the left arrow button. When the Run list has all the input files listed, click on the Run Input Files button to begin running the analyses.

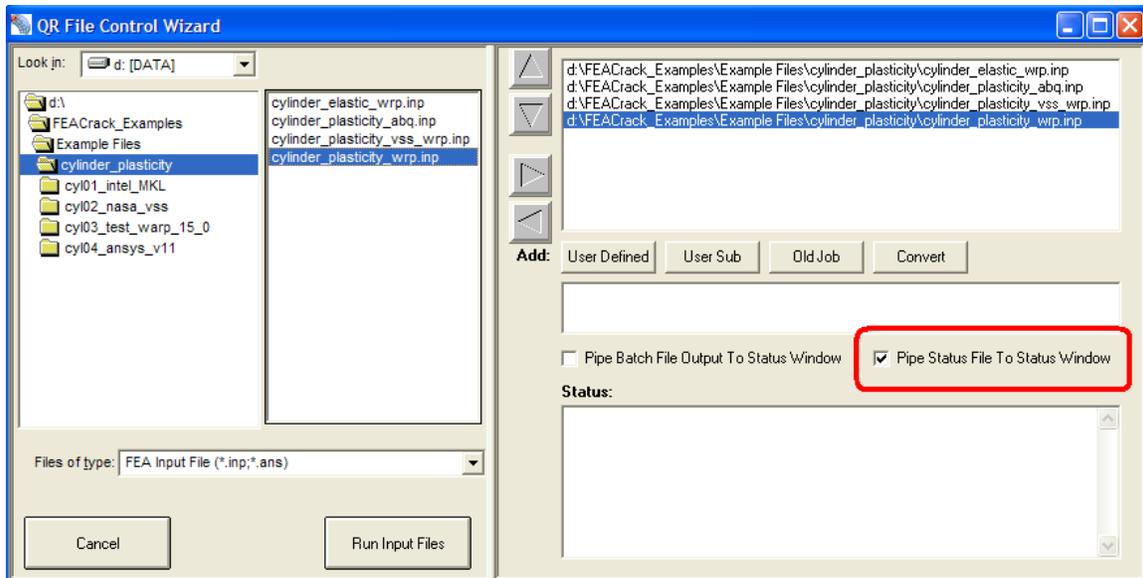


Figure 83 Run FEA batch control form; select input files and add to the queue; select the option to pipe the FEA solution status file to the bottom window

Echo FEA Status File

During an analysis, FEA programs write a solution status file to give information about the current analysis step and iteration increment. For Abaqus it is the *.sta file, for ANSYS it is the *.mntr file, and for Warp3D it is the *.batch_messages file.

For a Warp3D analysis the option to write the batch messages file is in the Tools/Program Options/Warp3D form. The Model Label text on the Home form is used as the batch messages file name; if blank the default “vessel” label is used.

In the File Control Wizard form, the “Pipe Status File to Status Window” option controls the echo of the FEA status file to the Status window at the bottom of that form, which allows the analysis progress to be easily monitored. The option is circled in red in the figure above.

Binary Packets File

WARP3D results can be written to a binary packets file. Activate output to the binary packets file on the Tools/Program Options/Analysis Options form. Check the Produce a *WARP3D* file box, and then check the Write Results to Binary Packet File box. When the *WARP3D* input file is created the binary packets file syntax will be included. The standard *WARP3D* output file has a *_wrp.out file extension, and the binary packets file has a *_wrp.bpf file extension based on the input file name. When selecting result files for post processing there is a *_wrp.bpf filter choice in the “Files of Type menu” on the “FEA Results/File Selection form”; since the binary packets file does not contain any mesh data or crack data, the mesh and crack post processing data are read from the input file. Both the input file and binary packets file must be in the same folder for post processing. A new check box is available at the bottom right on the File Selection form to request the binary packet results be written to a text (ASCII) file.

Future development will allow the binary packets file to be used to save results for a subset of the mesh for faster post processing of crack results from large meshes.

Interaction Integral Results

The interaction integral results: T-stress, K factors, J estimate from K factors, is available by checking the T-stress and K factors box on the “Tools/Program Options/Analysis Options form”. Then when the crack mesh is generated, the syntax to compute the interaction integrals is included in the *WARP3D* or *Abaqus* input file (not available for *ANSYS*). When reading the results file the T-stress, K factors, and J estimate results are automatically read and place in the Crack Results in addition to the J-integral and stress intensity K. The analysis must use elastic material and have one analysis step. *WARP3D* reports two sets of interaction integral results: one for a plane strain material assumption, and the other for a plane stress material assumption. Both sets of interaction integral results from *WARP3D* are given in the Crack Results; the two sets give similar values along the crack front. *Abaqus* reports just one set of interaction integral results, which are given in the Crack Results.

The elastic T-stress is described in section 3.6.1 of Fracture Mechanics [T. L. Anderson, 3rd ed., p.137-140]. The K factors can be compared to the K from J and K from displacement results. The J estimate from the K factors can be compared to the J-integral results.

Viewing Results

Click on the Results icon to show the post-processing window, which is shown below. Select results files from the hard disk in the same way you selected input files for analysis. Note that *Abaqus* results files have a .fil extension, *ANSYS* results have a .rst extension, while *WARP3D* results have a .out extension. Click on the Refresh Results button when you have selected the desired files.

Note: If a results file is incomplete or contains errors, an error message will be displayed and results will not be shown. The example folders included results files for the several models. However, if you ran the example input files, you will have overwritten the corresponding files. If the example models did not run correctly on your computer and

consequently you are unable to read in results, complete and correct versions of the example results files may be copied from the CD or from our web site. Alternatively, you can read in the *Abaqus* results files if you overwrote the *WARP3D* versions, and vice versa.

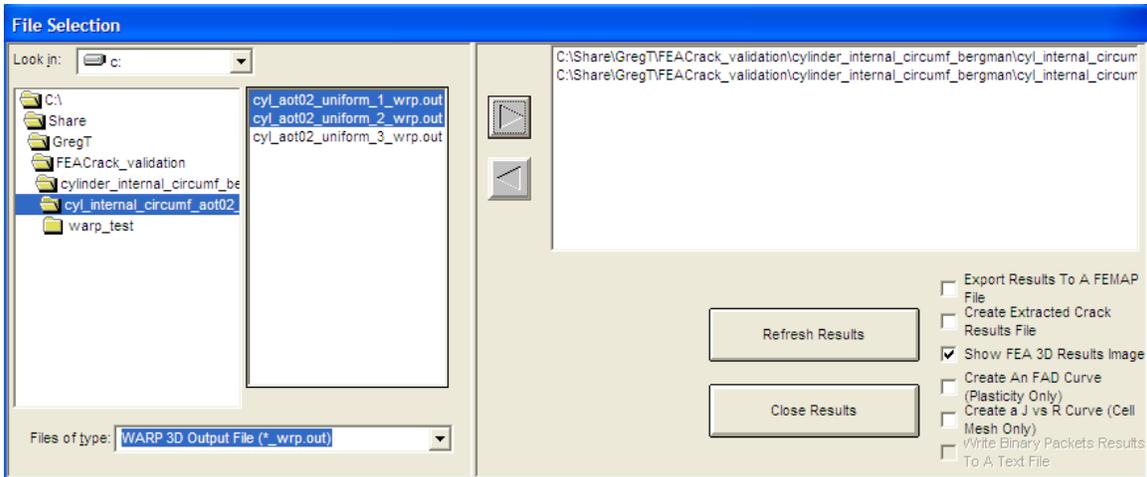


Figure 84 FEA Results form, select result files for post-processing

The mesh, along with stress, strain, displacement, and crack parameter (J- integral and/or stress intensity) results will be loaded into memory. This may take several minutes depending on the speed of your computer.

The viewing controls (e.g., Zoom and Rotate) are explained above in the section on building and viewing models. Additional controls are visible when results are loaded into memory, as described below.

The Displacement Scale field allows a user-defined exaggerated displacement view. For example, typing “200” in this input field will result in a deformed view that is exaggerated by a factor of 200. Note that when the program detects that a number has been typed into this field, it will refresh the view. There is a slight delay before the redraw to ensure that the user has finished typing. Thus it is recommended that you type quickly when entering a number with two or more digits. If, for example, you type “200” very slowly, the mesh will be redrawn three times, with displacement scales of 2, 20, and 200.

The Result Type drop-down list shows the stress, strain, and displacement components that were extracted from the results file. When a result file is first loaded into memory, the drop down is defaulted to *No Results*, and only the mesh is shown.

For analyses with multiple load steps, the user must select the Result Step from the drop-down list. This selection will govern both the contour results and the deformed shape. Clicking on the Animate Results button will cause the results to cycle through the various load steps. Click on the same button to stop the animation. Note that the mesh will redraw with each time step, so the animation can be relatively slow except on a very fast computer.

Select Change Scale from the View menu to alter the color scale.

Click on the Crack Results icon to display a plot of the J-integral along the crack front, as illustrated below. You can also view the stress intensity factor along the crack front. Numerical values are listed in a spreadsheet-like grid and can be copied and pasted to other programs such as *EXCEL*.

For the special case where the loading is a single crack face traction (uniform, linear, quadratic, or cubic power law), a non-dimensional K solution is also displayed. The following K expression is assumed:

$$K_I = \sigma \sqrt{\frac{\pi a}{Q}} G \quad (\text{K 1.1})$$

Where σ is the applied stress, Q is a flaw shape parameter, and G is a non-dimensional geometry factor.

FEACrack plots the results as a function of crack front positioning. For elliptical crack slopes, the location on the crack front is defined by the angle ϕ , which is defined below.

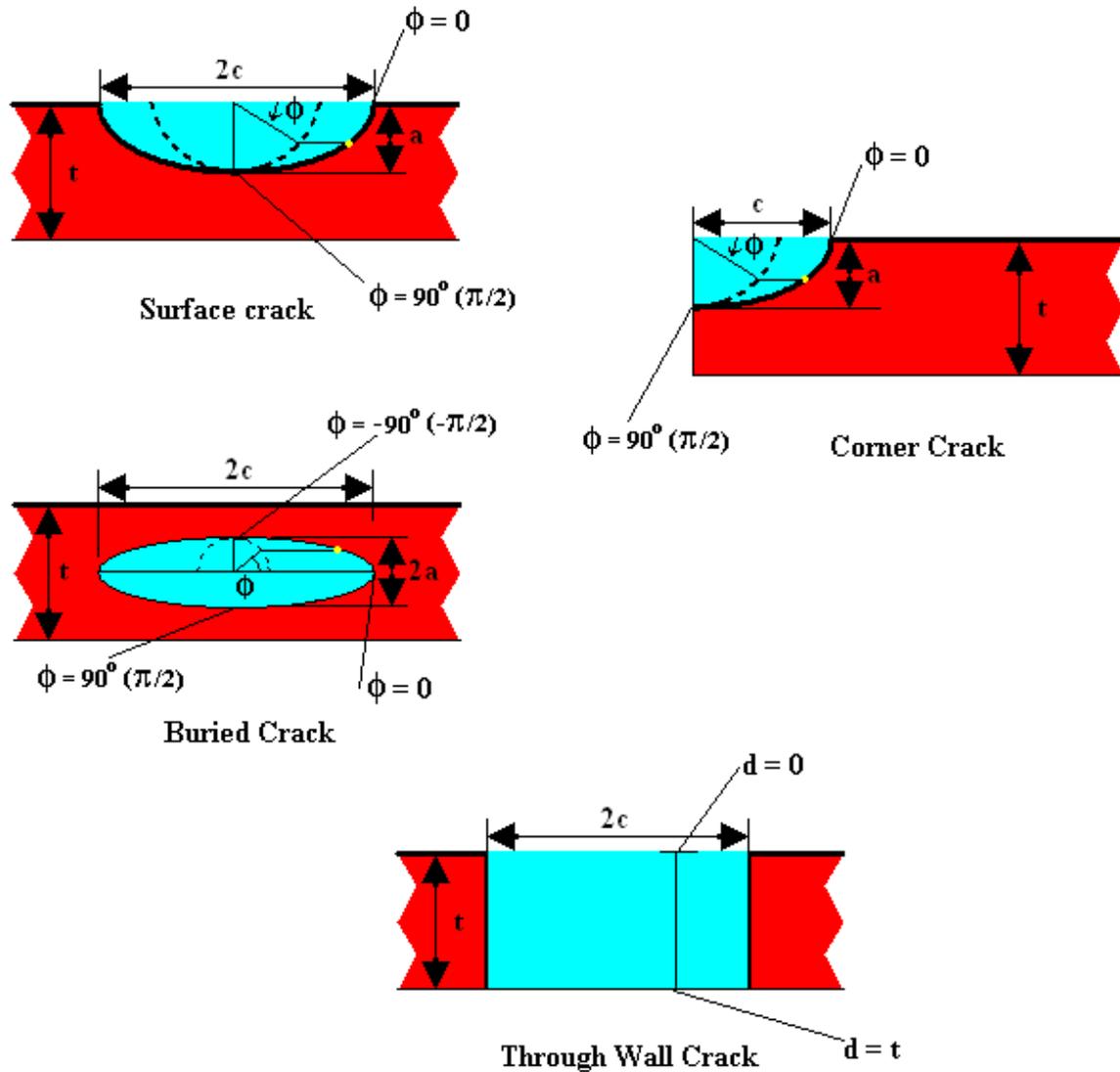


Figure 85 Crack front position by ϕ angle or d

As part of the post-processing algorithm, G is fit to a polynomial:

$$G = A_0 + A_1 \left(\frac{2\phi}{\pi} \right) + A_2 \left(\frac{2\phi}{\pi} \right)^2 + A_3 \left(\frac{2\phi}{\pi} \right)^3 + A_4 \left(\frac{2\phi}{\pi} \right)^4 + A_5 \left(\frac{2\phi}{\pi} \right)^5 + A_6 \left(\frac{2\phi}{\pi} \right)^6 \quad (\text{K 1.2})$$

Where ϕ is the parametric angle along the crack front; the polynomial fit to G can be plotted (see below), and the fitting coefficients A_i are tabulated. The order of the polynomial can be set in the Model Options control panel under the Tools menu.

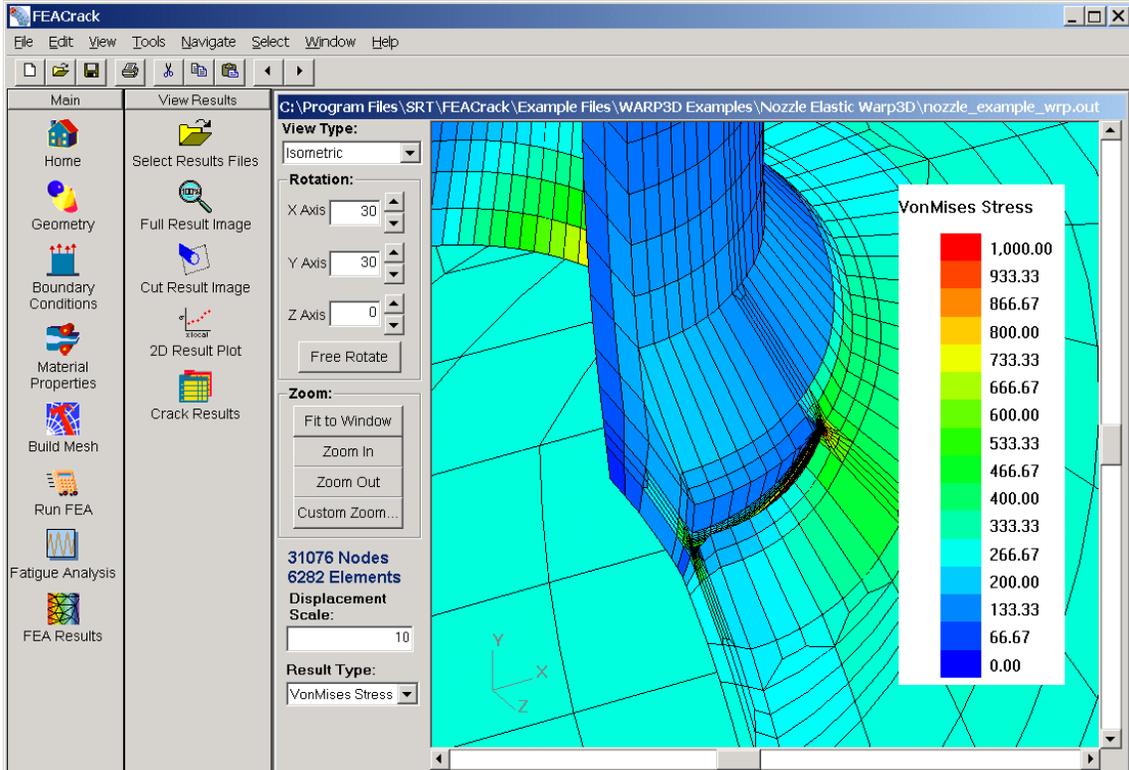


Figure 86 FEA results, color map of stress results on the mesh picture

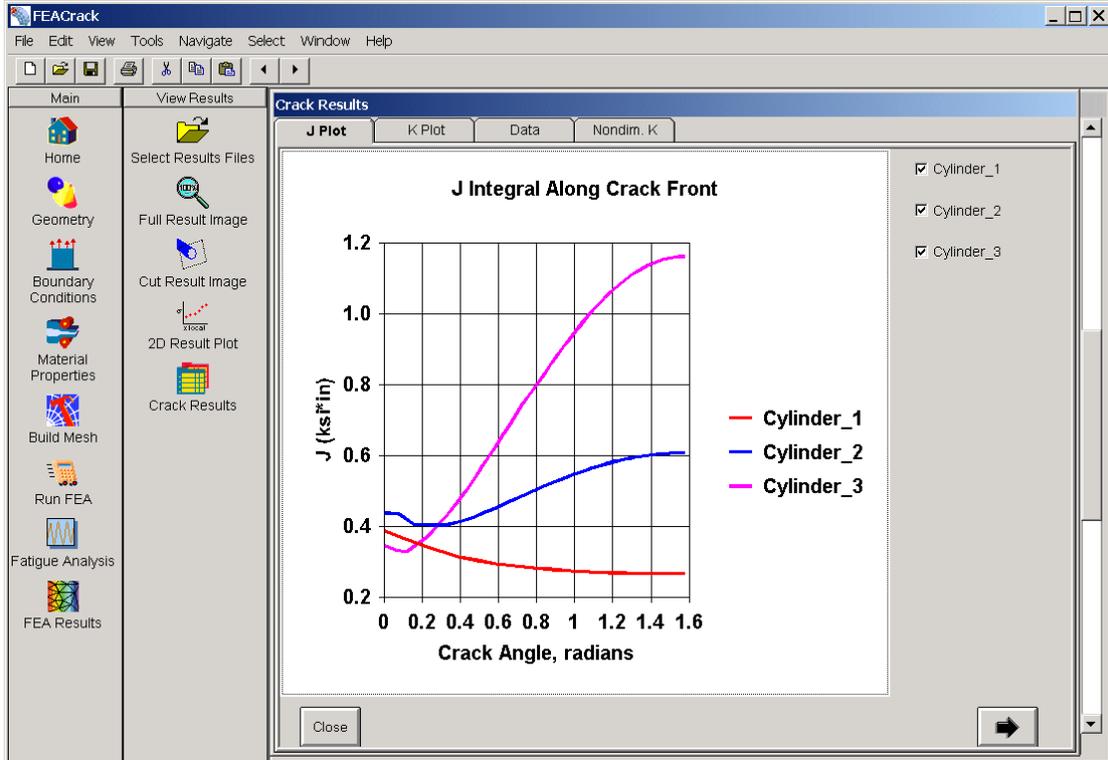


Figure 87 Crack results, plot of J-integral results along the crack front

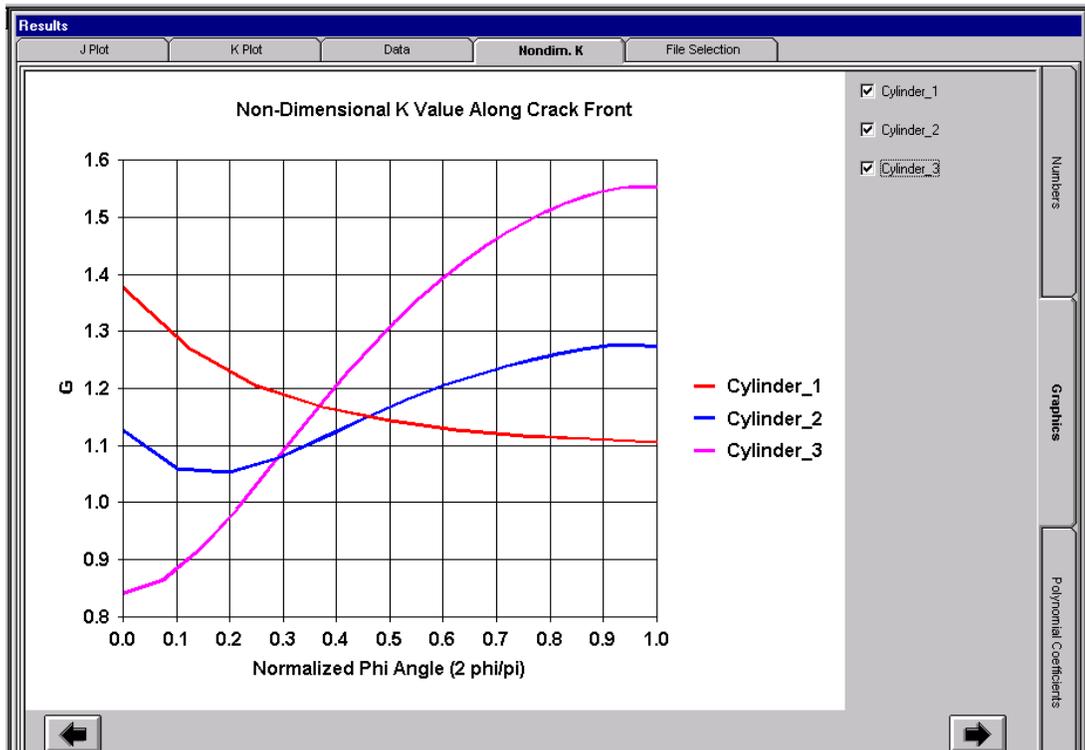


Figure 88 Crack results, plot of the non-dimensional G value

J-Integral Path Dependence Index

The J-integral path dependence index, d_p , is the ratio of the maximum minus minimum J-integral contour values, divided by the average J value; shown in the equation below. The first contour is omitted; the second contour through the total number of contours is used. For each crack front nodes, the post-processor finds the maximum and minimum J values, and computes the average J value. A path dependence index below 0.05 usually indicates a good J-integral result, since the J values are not varying much across the contours. If the path dependence index value is high for several crack front nodes, additional crack front refinement may be needed.

$$\text{J path dependence index, } d_p = \frac{J_{max} - J_{min}}{2J_{avg}}$$

Equation 89 J path dependence index

Elastic-Plastic Analysis

Special Considerations

Since elastic problems scale linearly, the absolute value of applied load or displacement is not important. In elastic-plastic problems, however, the applied loads or displacements must be chosen carefully. If the maximum load is too high, such that stresses well in excess of yield are

induced, the finite element analysis will not converge. If the load is too low, plastic deformation will be negligible.

There are a number of considerations when selecting load step size. Smaller load steps make convergence easier, but numerous load steps increase solution time and increase the size of the output file. The effect of load step size on convergence is more important in incremental plasticity than it is in deformation theory. The desired number and location of output points also may dictate the prescribed load steps.

Results from elastic-plastic analyses are read into memory in the same way as the elastic analyses. In this case, however, a plot of J versus applied stress is shown.

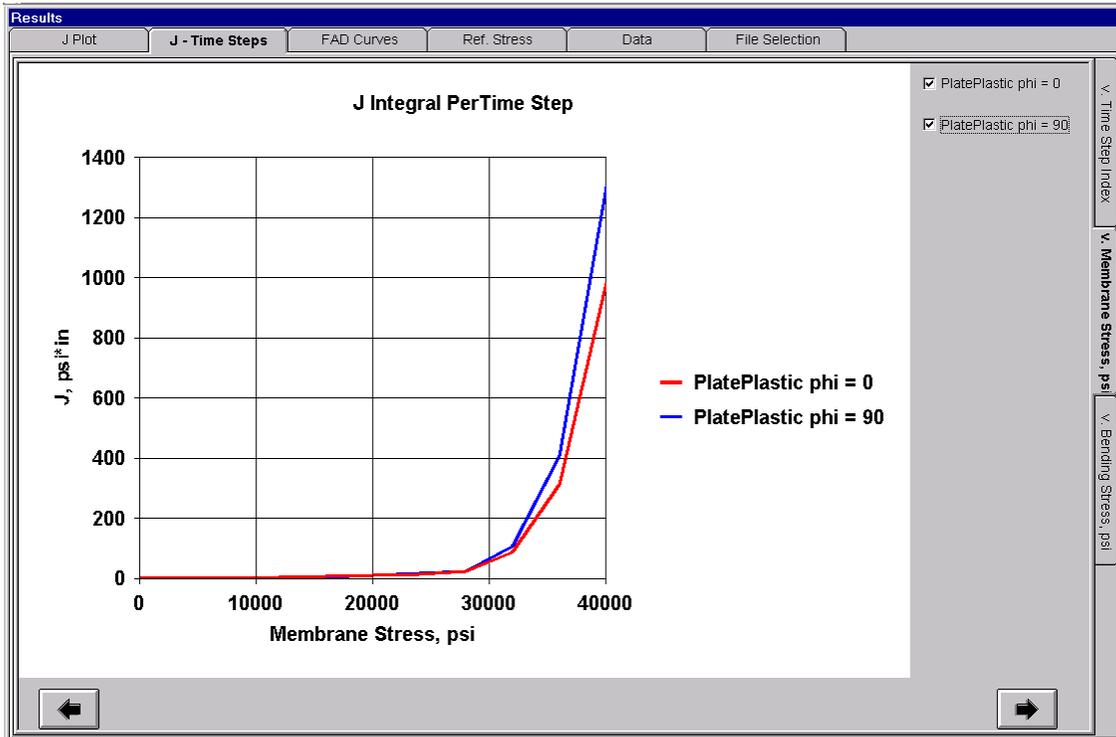


Figure 90 Crack results, elastic-plastic multi-step J results

J-Based FAD

Elastic-plastic J-integral results can be plotted in terms of a failure assessment diagram (FAD). *FEACrack* performs this task automatically using a procedure in Appendix B of API 579. This approach is outlined briefly below.

The vertical ordinate (y axis) of the FAD is given by

$$K_r = \sqrt{\frac{J_{elastic}}{J}} \tag{J 1.1}$$

FEACrack uses an extrapolation procedure to infer the elastic component of J . The above ratio is plotted against the load ratio, defined as:

$$L_r = \frac{\sigma_{ref}}{\sigma_{YS}} \quad (J 1.2)$$

Where σ_{ref} is the reference stress; in conventional FAD approaches, the reference stress is based on net-section yield or limit load solutions. This definition of σ_{ref} leads to a significant geometry dependence in the FAD. In Appendix B of API 579, a self-consistent definition of reference stress is given, which removes virtually all of the geometry dependence in the FAD curve. In API 579, the point at which $L_r = 1$ is defined by the ratio of total J to elastic J :

$$\left. \frac{J}{J_{elastic}} \right|_{L_r=1} = 1 + \frac{0.002E}{\sigma_{YS}} + \frac{1}{2} \left(1 + \frac{0.002E}{\sigma_{YS}} \right)^{-1} \quad (J 1.3)$$

The above expression assumes that σ_{YS} is the 0.2% offset yield strength. The reference stress is linearly related to the nominally applied stress through a geometry factor, F :

$$\sigma_{ref} = F \sigma_{nominal} \quad (J 1.4)$$

Where F is inferred from the nominal stress at $L_r = 1$:

$$F = \frac{\sigma_{YS}}{\left. \sigma_{nominal} \right|_{L_r=1}} \quad (J 1.5)$$

The FAD curves and computed reference stress for the flat plate example are shown below.

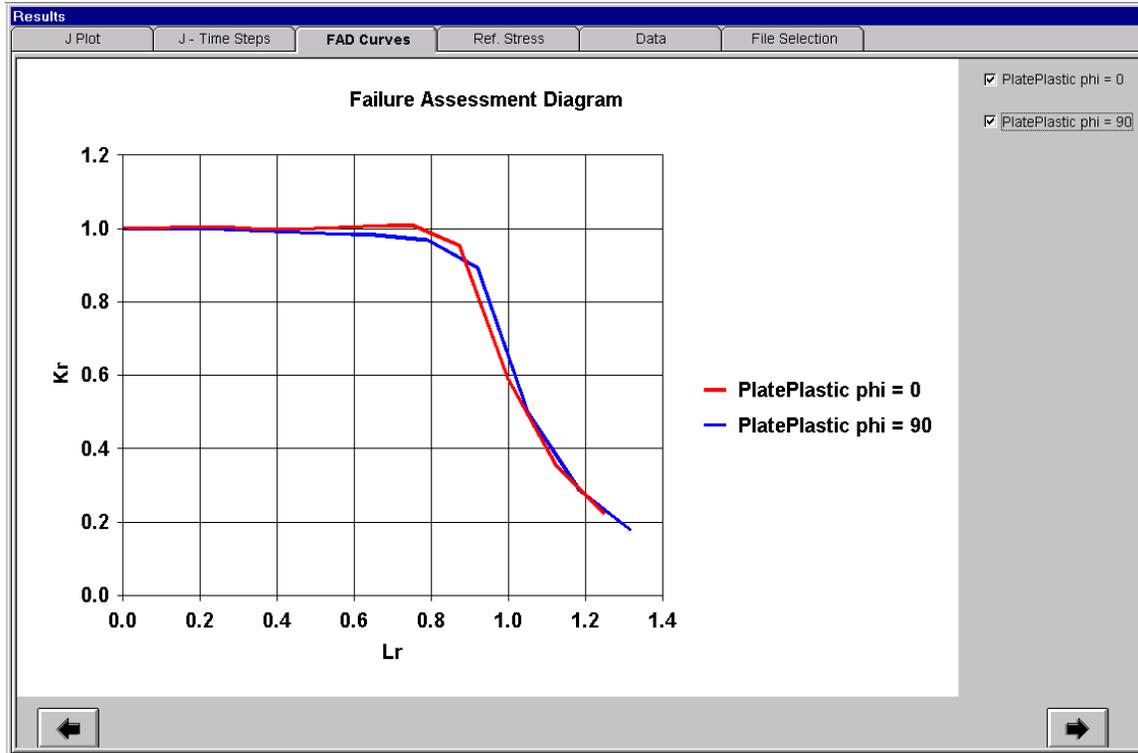


Figure 91 Crack results, FAD plot computed from elastic-plastic J-integral results

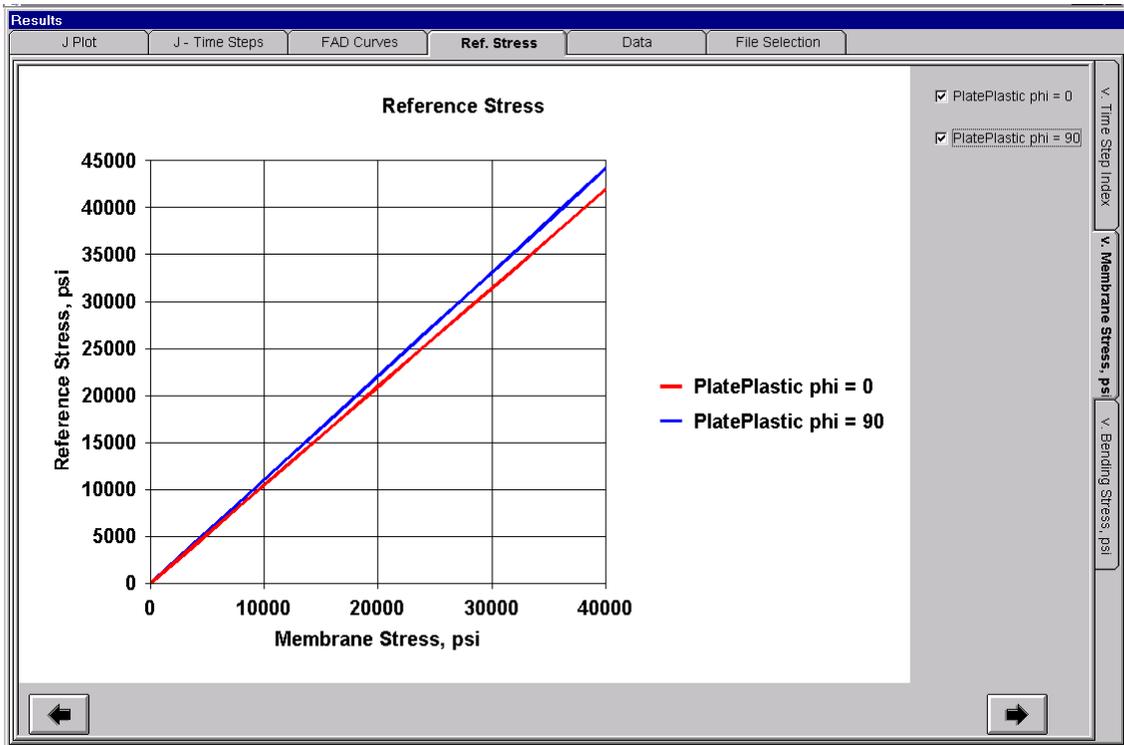


Figure 92 Crack results, reference stress computed from elastic-plastic J-integral results

For the linear + power-law stress strain curve that *WARP3D* uses in plasticity problems, an iterative procedure is used to compute the 0.2% offset yield strength. For the Ramberg-Osgood expression, the 0.2% offset yield strength can be inferred directly from the following expression:

$$\sigma_{YS} = \sigma_o \left(\frac{0.002}{\alpha \varepsilon_o} \right)^{1/n} \quad (\text{J 1.6})$$

For a tabular (piece-wise linear) stress-strain curve, the stress at 0.2% plastic strain is obtained by interpolation within the stress-strain table.

When elastic plastic results are read into memory, *FEACrack* check the “Create an FAD Curve” option box in the FEA Results form so that the post-processor will automatically attempt to compute the reference stress and FAD curve. A warning message is returned if *FEACrack* is unable to generate the FAD curve. There are a number of possible reasons why such a calculation might be unsuccessful. If there are insufficient data in the plastic range, i.e. if the analysis is stopped before L_r reaches unity, *FEACrack* cannot compute the reference stress. At the other extreme, if there are insufficient data in the elastic range, *FEACrack* will have difficulty estimating the elastic J . Some trial and error may be required to specify load steps properly so as to output sufficient data to compute the FAD curve.

Inner Plastic Zone Option

The inner plastic zone option allows for an increased number of elements near the crack front. The crack tube (concentric elements that surround the crack front) is separated into two zones: the inner plastic zone, and the outer zone. The number of elements and the element size bias can be controlled in each zone. The inner plastic zone option is available only for the PRO mesh block, which allows for a greater level of crack mesh refinement.

To activate the inner plastic zone, check the option box in the Geometry/Configuration form, as shown in the figure below.

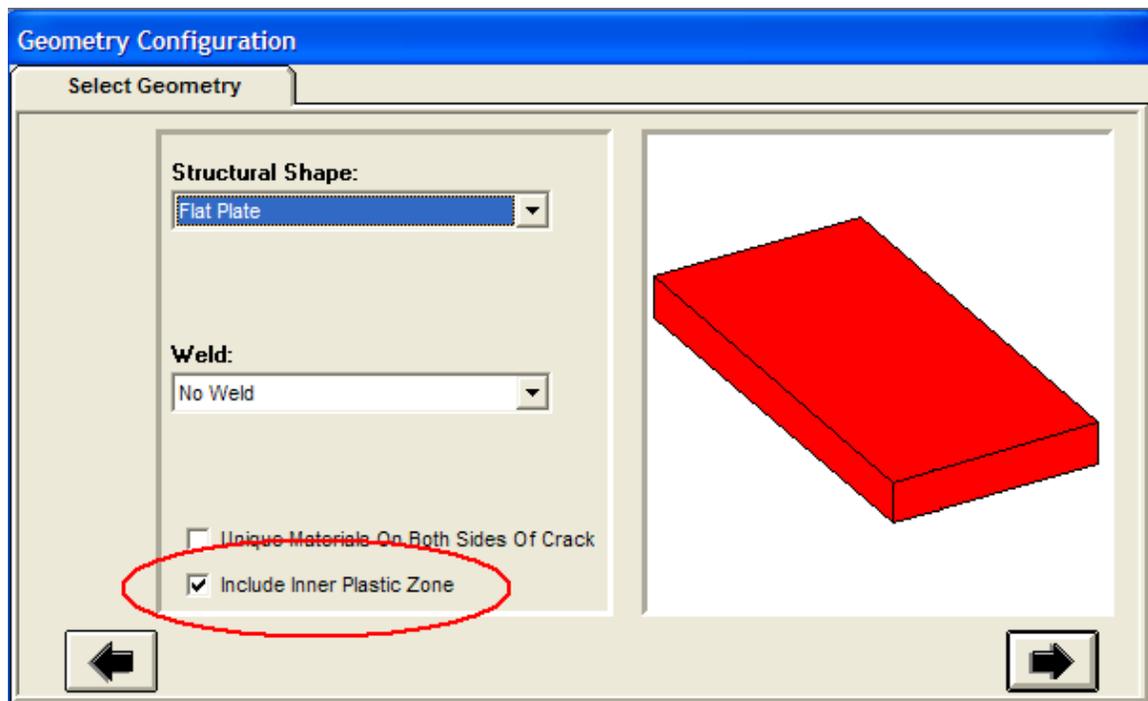


Figure 93 Check the option box to activate the inner plastic zone mesh option.

Next, enter the number of elements in the inner plastic zone and the outer zone, and enter the element size bias factors in the Geometry/Crack/Advanced Mesh/Refine form, shown in the figure below. The radial size bias value controls the element size ratio in the inner plastic zone from the crack front across the inner plastic zone. A bias value of 1.0 gives uniform size elements; or a bias value greater than 1.0 gives smaller elements at the crack front to larger elements on the outside of the zone. The number of elements in the inner plastic zone should be at least 1, but several elements are recommended. These are the elements closest to the crack front. The outer zone extends from the inner plastic zone across the remainder of the crack tube. At least 1 element is needed in the outer zone, but several elements are recommended. An outer zone element size bias of 1.0 gives uniform size elements; or a value greater than 1.0 gives smaller elements near the inner plastic zone to a larger element size on the outside of the crack tube.

Also enter the number of J-integral contours for the crack results on this form. Typically, the number of J contours is the sum of the number of element in the inner zone plus the number of elements in the outer zone, but can be less. The value entered in this form takes precedence over the number of element rings in the Tools/Model Options form.

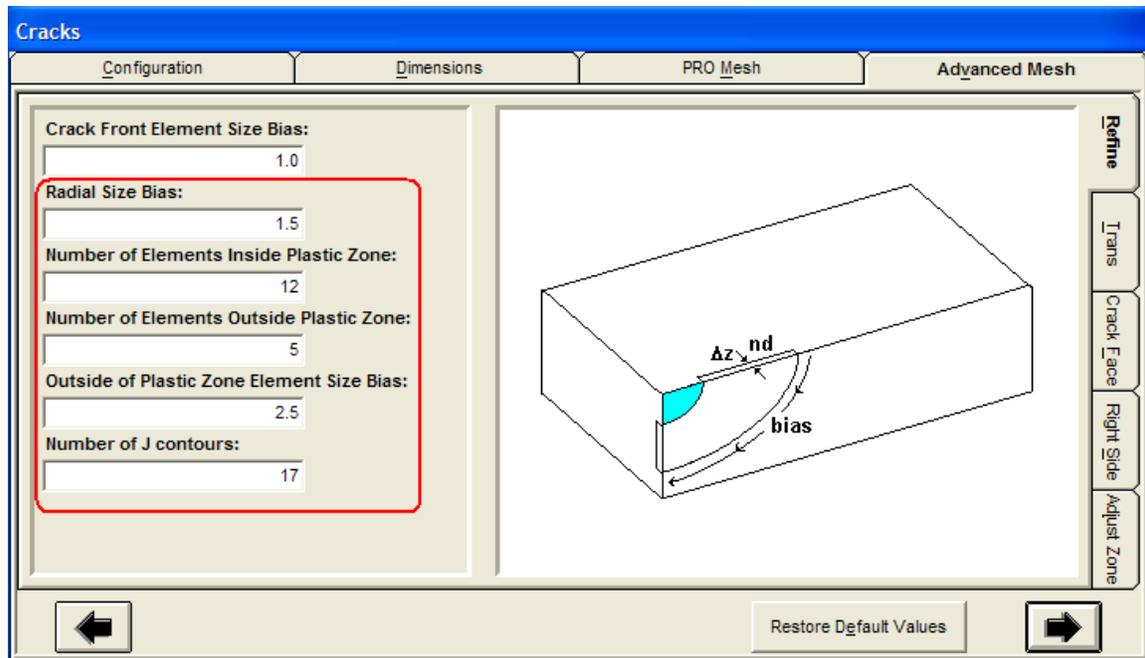


Figure 94 Enter the number of elements and the element size bias for the inner plastic zone. Enter the number of J-integral contours.

Enter the dimensions for the crack tube radius and the inner plastic zone radius in the Geometry/Dimensions form, shown in the figure below. The crack tube radius value entered in this form replaces the “User Defined Crack Tube Diameter” multiplier in the Tools/Model Options/Crack Mesh Params form; the value from this form is used to generate the mesh. The inner plastic zone radius must be smaller than the crack tube radius so that the inner plastic zone fits within the crack tube.

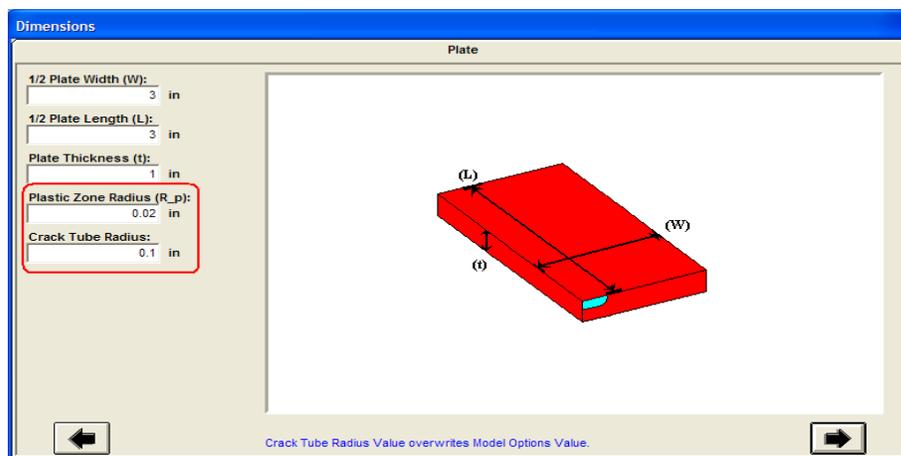


Figure 95 Enter the crack tube radius and inner plastic zone radius.

Examples of the inner plastic zone are shown below.

Figure 96 Circumferential surface crack with the inner plastic zone mesh option

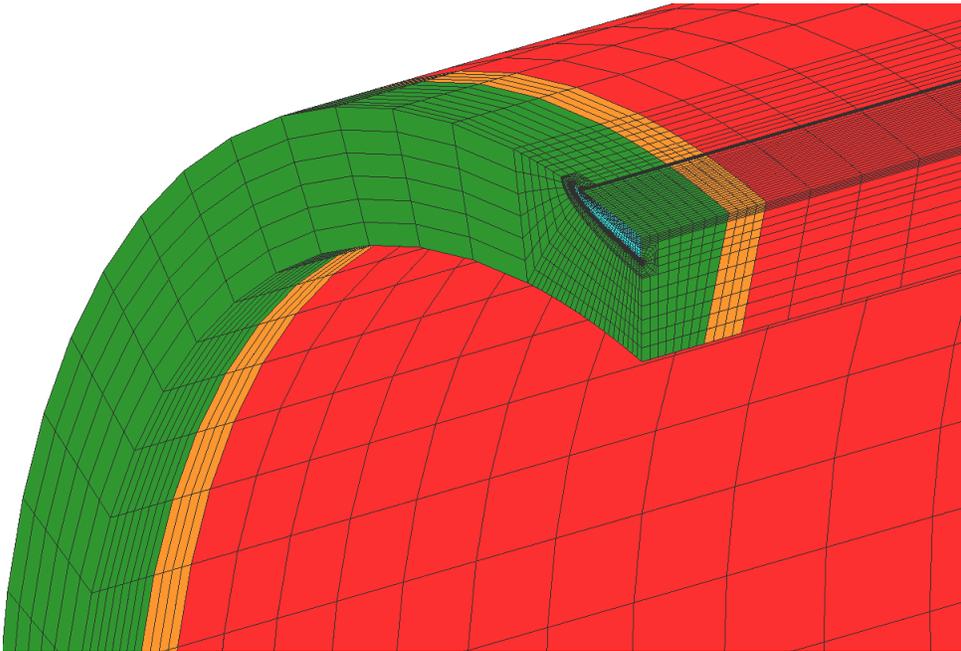
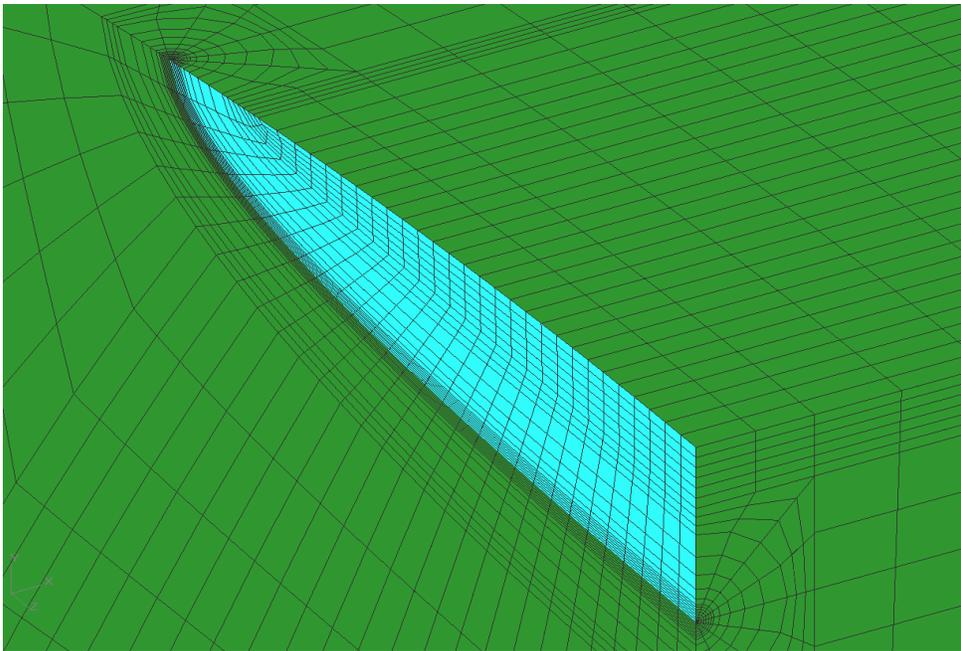


Figure 97 Close up of the crack front showing the inner plastic zone



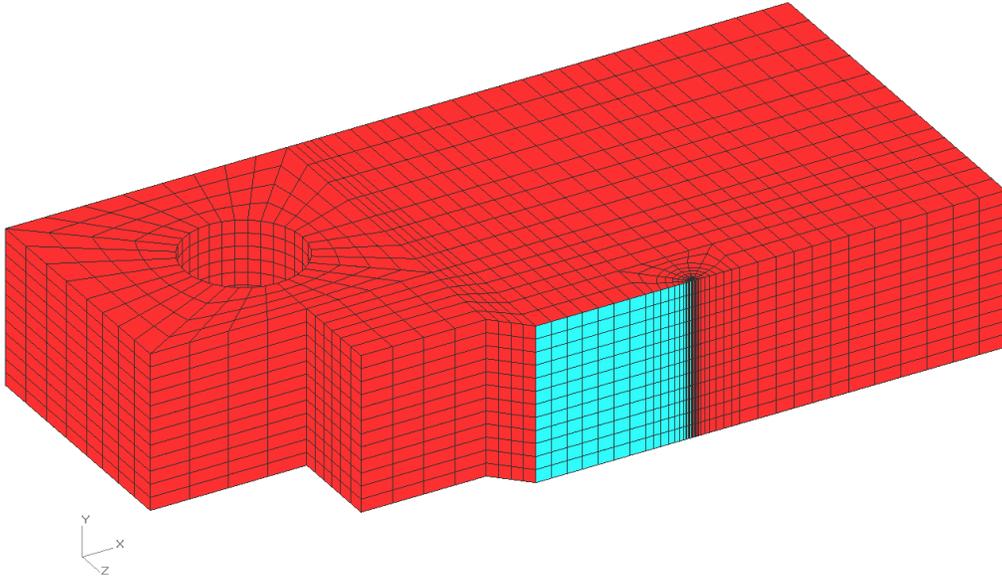


Figure 98 C(T) specimen mesh with the inner plastic zone

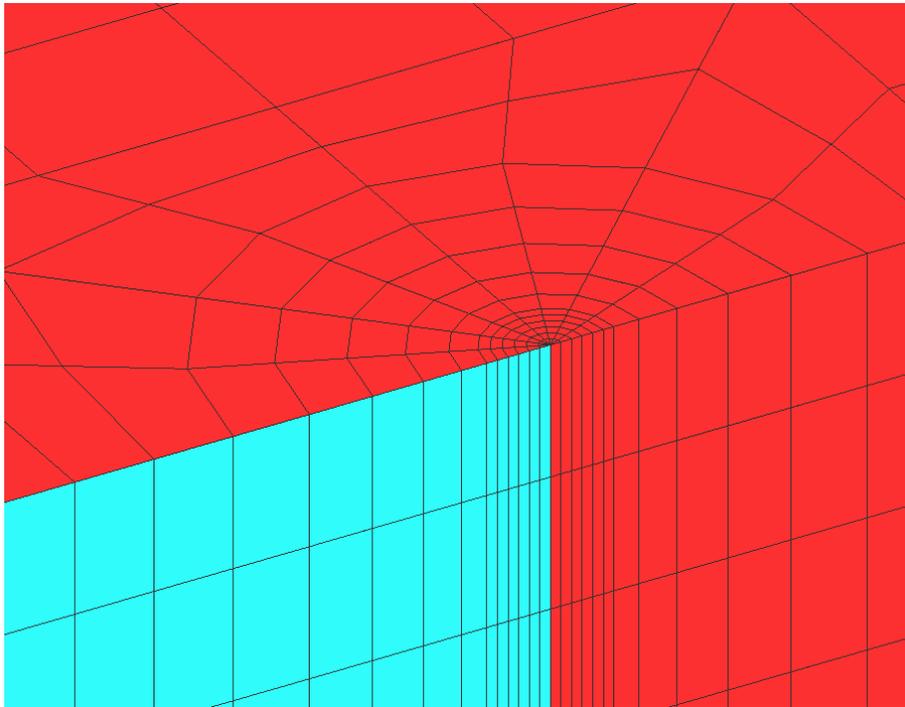


Figure 99 Close up of the crack front and inner plastic zone

J-integral Contour Plotting Tool

The crack front J-integral results can be examined in more detail using the Tools/J Contour Plotting calculator form. First, load a result file as usual, and then open the J Contour Plotting form. By plotting the J-integral at selected crack front nodes, the J-integral contour path dependence can be observed. Using the J versus contour index plot, select the starting and ending contour numbers at the top of the form to compute the average J along the crack front in the “J vs. phi angle” tab. Note that the data points are shown on the J versus contour curves for the selected contours.

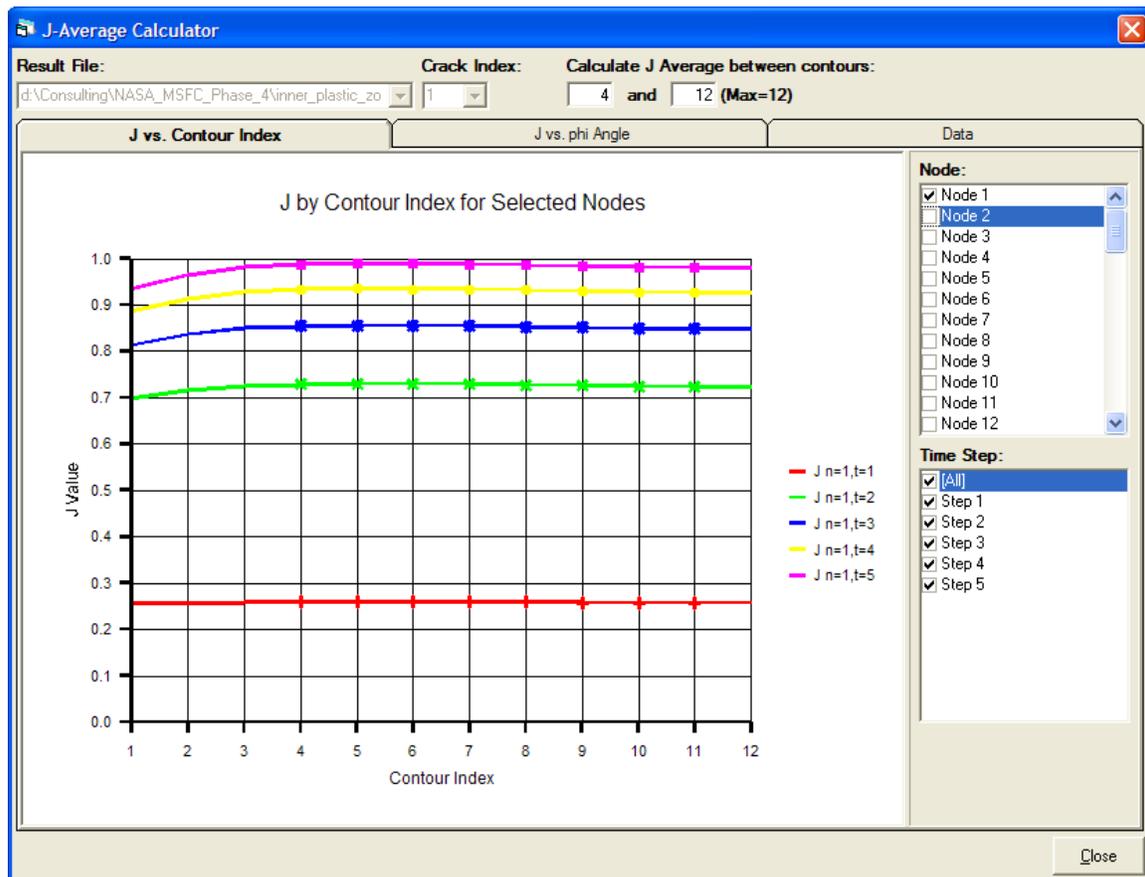


Figure 100 J Contour Plotting tool; plot the J-integral versus the contour index to examine the path dependence

In the “J vs. phi angle” tab, select the “J Average” plot or a particular contour. A specific contour can also be obtained by setting the starting and ending contour index number to the same value at the top of the form. The starting and ending contours must be from 1 to the available maximum number of contours in the crack mesh. The “phi angle” value on the plot’s horizontal axis is the crack front phi angle value defined from the crack tip to the deepest point of the crack.

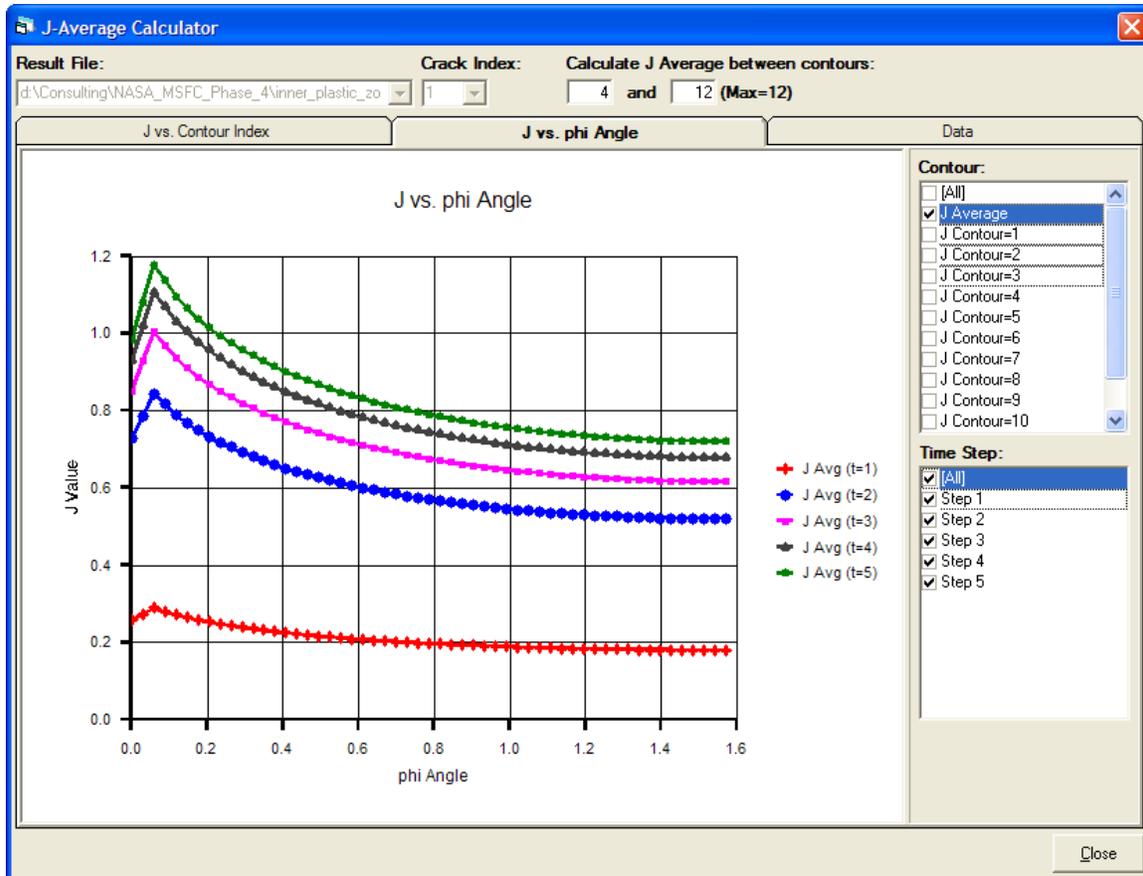


Figure 101 The average J along the crack front is computed using the starting and ending contour values

There are two side tabs on the Data tab form. The selected node contour J values are listed in the “J by Contour Index” side tab, and the crack front J average and J per contour values are listed in the “J and phi by Node” side tab. These values can be selected and copied to the clipboard to be pasted into another program for additional calculations.

Node Index	phi angle	J Average	Contour 1	Contour 2	Contour 3	Contour 4	Contour 5	Contour 6	Contour 7
1	0.2582995062	0.254715304	0.2553567826	0.257698658	0.2588711527	0.2591882397	0.259198545	0.25907345	
2	0.029118	0.2738211594	0.2749498296	0.2717900665	0.2731987212	0.2740700628	0.2744253436	0.2744549902	0.27437705
3	0.058236	0.2893428127	0.2951843531	0.2882033503	0.2886988043	0.289268973	0.2896624474	0.2897114353	0.28968064
4	0.087354	0.280434119	0.2863339653	0.2799053504	0.2801527372	0.2803962332	0.2805868107	0.2805904631	0.28056091
5	0.116472	0.2715254254	0.2774835775	0.2716073505	0.2716066701	0.2715234934	0.271511174	0.2714694908	0.27144118
6	0.14559	0.264932706	0.27031235	0.2650021196	0.2650154746	0.2649622111	0.2649714287	0.2649504196	0.26493630
7	0.174707	0.2584331157	0.2631411224	0.2583968866	0.2584242791	0.2584009287	0.2584316833	0.2584313483	0.25843142
8	0.203825	0.2531544748	0.2574718319	0.2530958125	0.2531313653	0.2531159295	0.2531541626	0.2531576326	0.25315992
9	0.232943	0.2478758338	0.2518025414	0.2477947383	0.2478384514	0.2478309903	0.247876642	0.2478939169	0.24789842
10	0.262061	0.2434225702	0.247026792	0.2433345946	0.2433817732	0.2433766299	0.2434226042	0.2434302002	0.24343493
11	0.291179	0.2389693065	0.2422510427	0.2388744509	0.238925095	0.2389223294	0.2389685663	0.2389764835	0.23898143
12	0.320297	0.2351654898	0.2381915229	0.2360696736	0.2351205942	0.2351181666	0.2351640547	0.2351717702	0.2351766
13	0.349415	0.231361673	0.2341320031	0.2312648964	0.2313160935	0.2313140038	0.231359543	0.2313670568	0.23137189
14	0.378533	0.2280521721	0.2306221098	0.227956006	0.2280067841	0.2280046428	0.2280495234	0.228056365	0.22806163
15	0.407651	0.2247426711	0.2271122166	0.2246471156	0.2246974748	0.2246952819	0.2247395039	0.2247466162	0.22475136
16	0.436769	0.2218503094	0.2240511335	0.2217558672	0.2218055596	0.2218031898	0.2218468277	0.2218537911	0.22185849
17	0.465887	0.2189979477	0.2209900504	0.2188646188	0.2189136445	0.2189110977	0.2189541515	0.218990966	0.21899562
18	0.495004	0.2164081352	0.2182956691	0.2163159666	0.2163643261	0.2163616092	0.2164041464	0.2164109519	0.21641547
19	0.524122	0.2138583227	0.2156012879	0.2137673144	0.2138150077	0.2138121207	0.2138541413	0.2138607377	0.21386532
20	0.55324	0.2115979185	0.213215756	0.2115080327	0.2115550961	0.211552059	0.2115936355	0.2116001477	0.21161046
21	0.582358	0.2093375144	0.2108302242	0.2092487511	0.2092951846	0.2092919973	0.209331297	0.2093395576	0.20934405
22	0.611476	0.2073252649	0.208708436	0.2072375501	0.2072833987	0.2072800862	0.2073208362	0.2073271912	0.20733164
23	0.640594	0.2053130155	0.206587463	0.2052263492	0.2052716128	0.2052681751	0.2053085427	0.2053148249	0.20531923
24	0.669712	0.2035165127	0.2046953105	0.2034308226	0.2034755364	0.2034719984	0.2035120309	0.203518249	0.20352261
25	0.69883	0.2017200099	0.202803158	0.2016352961	0.20167946	0.2016758217	0.201715519	0.2017216731	0.20172599
26	0.727948	0.2001132408	0.2011121434	0.2000294224	0.2000730775	0.2000693652	0.2001087623	0.2001149569	0.20011913
27	0.757066	0.1985064717	0.1994211289	0.1984235487	0.1984669551	0.1984629086	0.1985020057	0.1985080407	0.19851228
28	0.786184	0.1970867533	0.1979094203	0.1969866717	0.197029322	0.1970254721	0.197064306	0.1970702877	0.19707448
29	0.815301	0.1956310348	0.196397718	0.1955497947	0.195591949	0.1955880356	0.195626063	0.1956325348	0.19563669
30	0.844418	0.194242993	0.195049242	0.194235332	0.194280319	0.194276451	0.1943149909	0.1943213609	0.19432519

Crack Mesh Controls

Number of Rings in Crack Tube

This control specifies the number of concentric element rings in the focused crack mesh (the “crack tube”). *Abaqus* and *WARP3D* will calculate the J-integral at each ring. The *FEACrack* post-processing module uses *ANSYS* results to compute the J-integral at each ring of elements.

Refinement along the Crack Front

There are five choices for the crack front mesh refinement. The Normal refinement choice uses the basic crack mesh pattern; no additional refinement is added. The 3x Refinement Along Full Crack Front choice increases the number of elements along the entire crack front by three times. Likewise, the 9x Refinement Along Full Crack Front choice increases the number of elements along the entire crack front by nine times by using two levels of 3 to 1 refinement. The 3x and 9x refinement along the crack front are recommended for short crack fronts, where the total number of nodes in the mesh will not increase to an unreasonably large number. The 3x Refinement at Free Surface and the 9x Refinement at Free Surface options increase the mesh refinement along the crack front at the mesh free surfaces. For example, a surface crack modeled with symmetry will have a mesh free surface at $\phi = 0$. The symmetry plane at $\phi = \pi/2$ is not a free surface. The mesh refinement will be added along the crack front near $\phi = 0$. If the full-symmetry surface crack is modeled, then mesh refinement will be added along the crack front at $\phi = 0$ and $\phi = \pi$. For a through-thickness crack the free surfaces are at the top and bottom of the plate. In most cases, confining the mesh refinement to the free surfaces in a through-crack does result in much savings in the number of nodes in the mesh.

Examples of each of the 5 crack front refinement options are shown below.

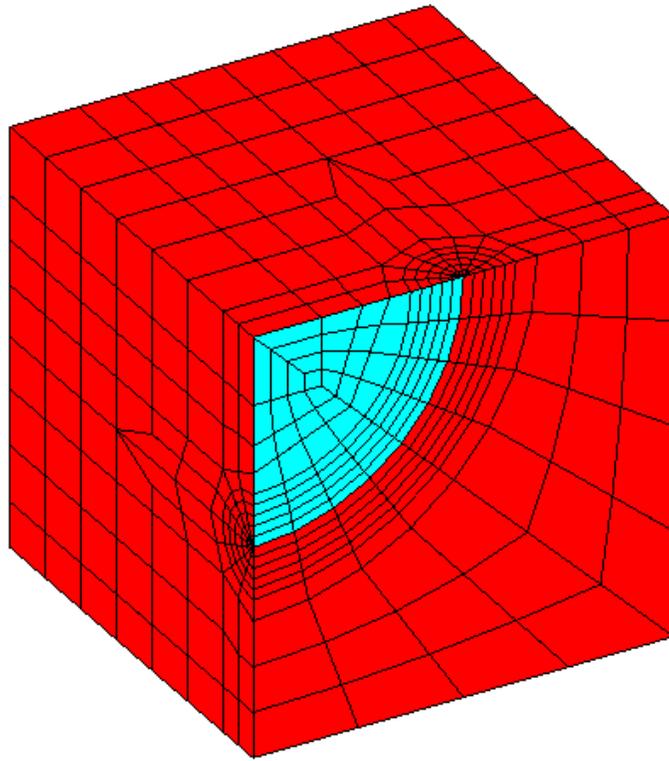


Figure 103 Normal crack front refinement.

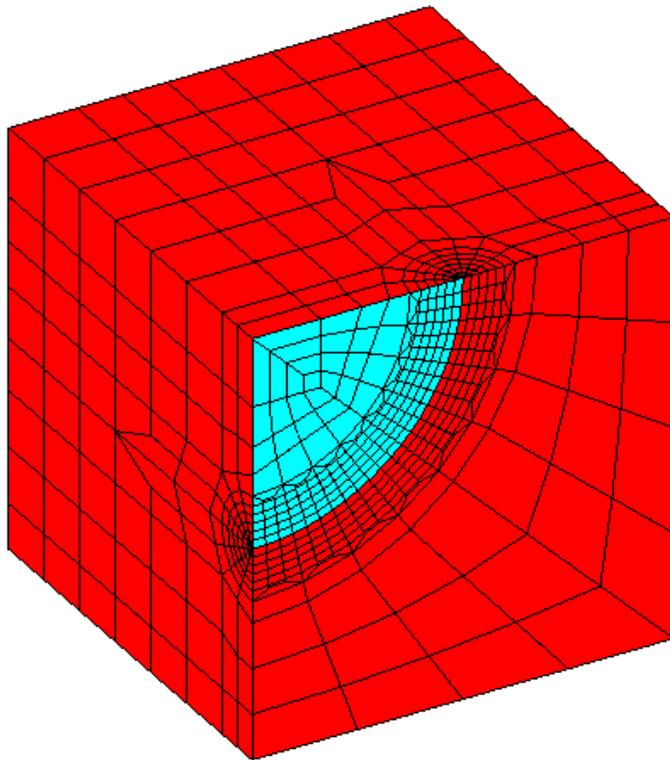


Figure 104 3x refinement along the entire crack front.

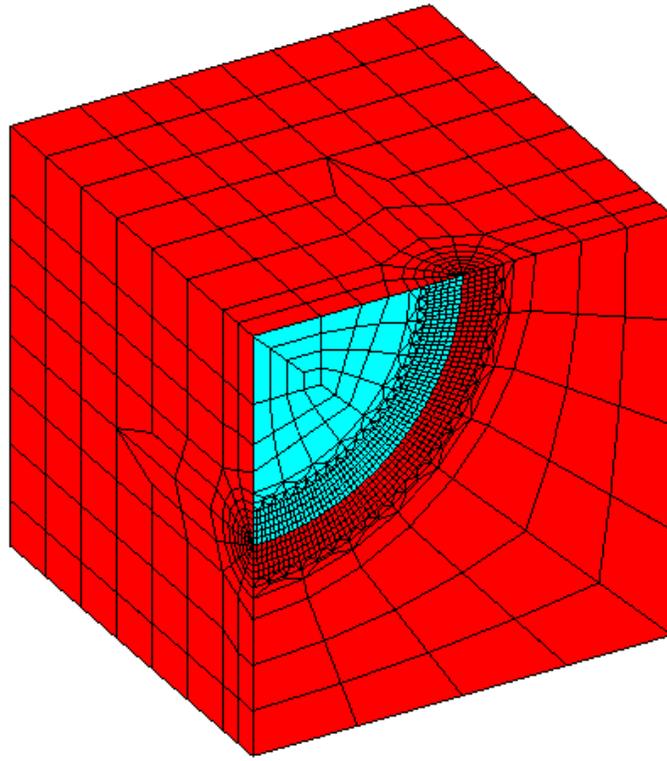


Figure 105 9x refinement along the entire crack front.

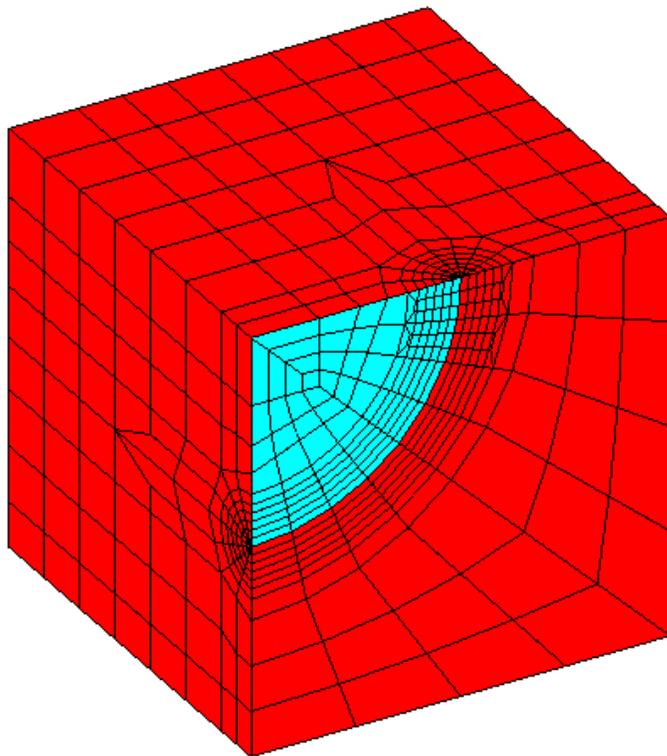


Figure 106 3x refinement near the free surface.

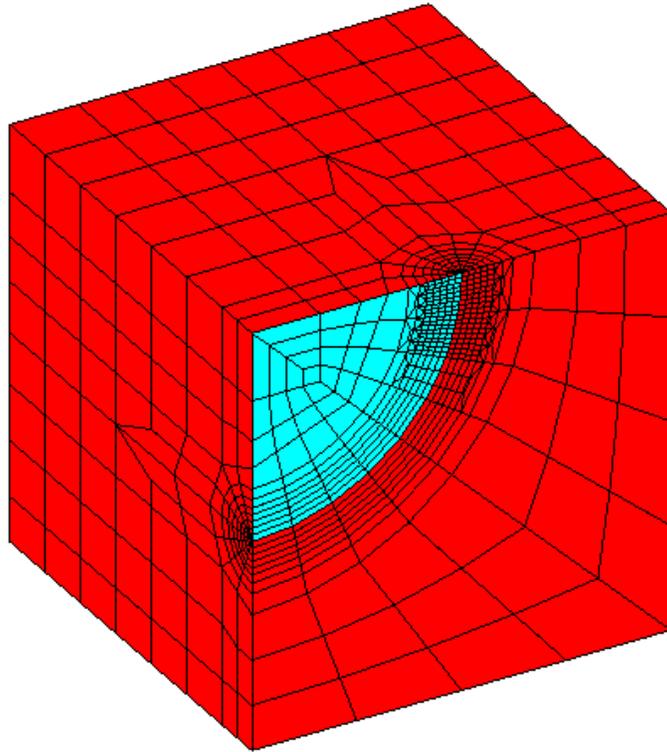


Figure 107 9x refinement near the free surface. Note the 3x transition zone.

Crack Block Type

Three types of crack mesh primitives are currently available. The mesh generator will use the default crack mesh block type for the geometry of interest when the Auto Mesh Block Type is selected. The Refined Mesh Block has more mesh refinement on the crack plane. The Coarse Mesh Block actually has more elements along the crack front, but generally has a more coarse mesh pattern on the crack plane. The coarse mesh block is typically used in geometries that have the crack at the toe of a fillet like the T-fillet, L-fillet, and nozzles. The Focused Cell Mesh has a focused crack mesh (collapsed elements at the crack front) within the cell crack mesh, which gives more mesh refinement and mesh pattern controls to the user; the cell crack mesh is discussed in more detail below.

Crack Tube Diameter

The diameter of the focused mesh region (the “crack tube”) is normally automatically chosen by the mesh generator, but can be specified by the user. Check the option box to activate the control and the Crack Tube Diameter multiplier value. This multiplier value gives the desired crack tube size such that all the concentric elements in the crack tube will be inside this region. If the user-specified crack tube size is too large for the crack size or geometry it will be automatically

reduced (a warning message will indicate the modified crack tube size). Typical values for the crack tube diameter multiplier range from 0.01 to 0.1.

There are a variety of situations where overriding the default crack tube diameter is appropriate. For example, a small crack tube may be necessary when the crack tip is close to a free surface.

Shallow Crack Cut-Off

The quarter-circular crack primitive for surface, corner, and buried cracks has optimal element aspect ratios for $c/a = 1$ and $a/t = 0.5$. For longer cracks, where $c/a \gg 1$, additional crack blocks with straight crack fronts are added to the mesh to prevent severe distortion of the quarter-circular primitive when meshing a long elliptical crack. For shallow cracks, where $a/t \ll 0.5$, additional mesh is added below the crack block to minimize distortion of the elements near the crack front. The shallow crack pattern also increases the mesh refinement below the crack. The default cut-off for the shallow crack pattern is $a/t = 0.15$. Crack depths less than this cut-off value will have the shallow crack pattern and depths greater than the cut-off will be achieved by transforming the crack block. Examples of the standard and shallow crack pattern for $a/t = 0.25$ are shown below.

In general, better answers are achieved for $a/t < 0.5$ when the shallow crack pattern is used. The model size will increase, however. Note that when the shallow crack pattern is used by the mesh generator, the surrounding mesh will *not* transition to a single element through the thickness, irrespective of the setting in the general mesh controls.

Use the mesh stretch or refinement index in the y direction (Y_{mult} or N_y , respectively) to set the number of elements below the crack block.

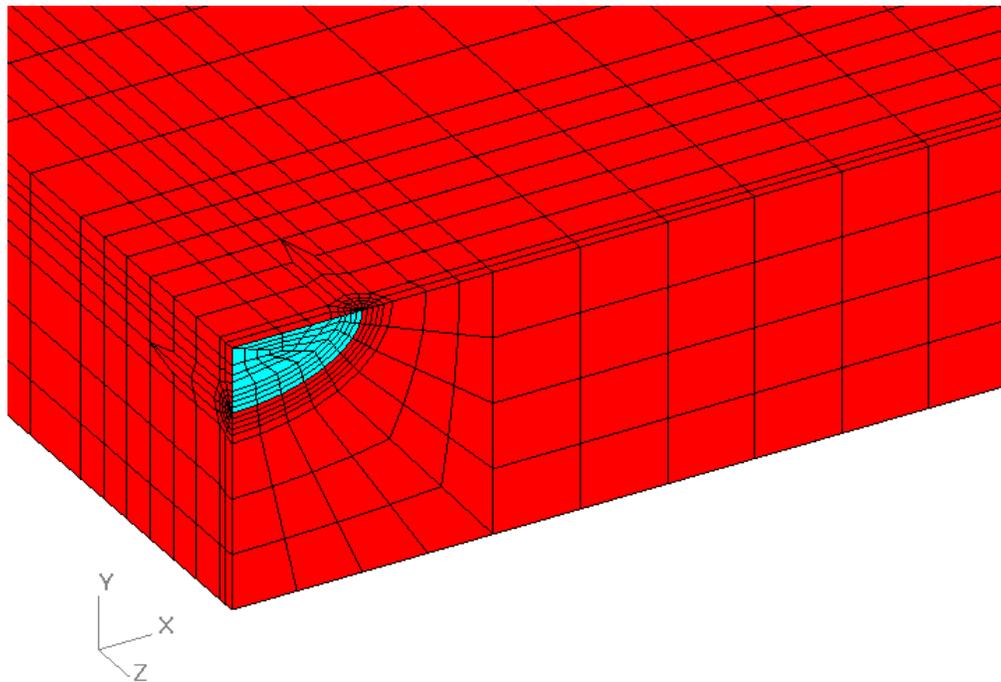


Figure 108 Standard pattern for $a/t = 0.25$. The shallow crack pattern is not used in this case.

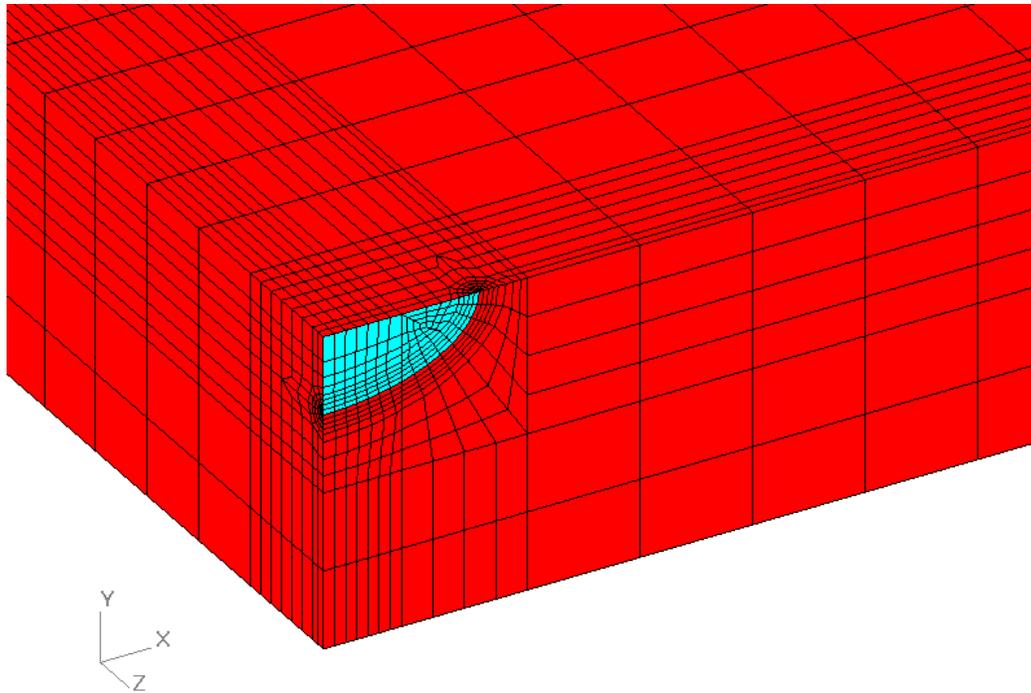


Figure 109 Shallow crack pattern for $a/t = 0.25$.

Advanced Mesh Controls

The user has an extensive level of control over the meshes created in *FEACrack*. Select Model Options from the Tools menu to display various mesh controls, as illustrated below. Note that there are normally two goals in designing the mesh: minimizing the model size and maximizing the accuracy of the solution. These are often conflicting goals, since the best solution tends to come from the most refined mesh. For a given problem, it may be necessary to experiment with the mesh controls to obtain a good mesh that is not too large to run in a reasonable time frame.

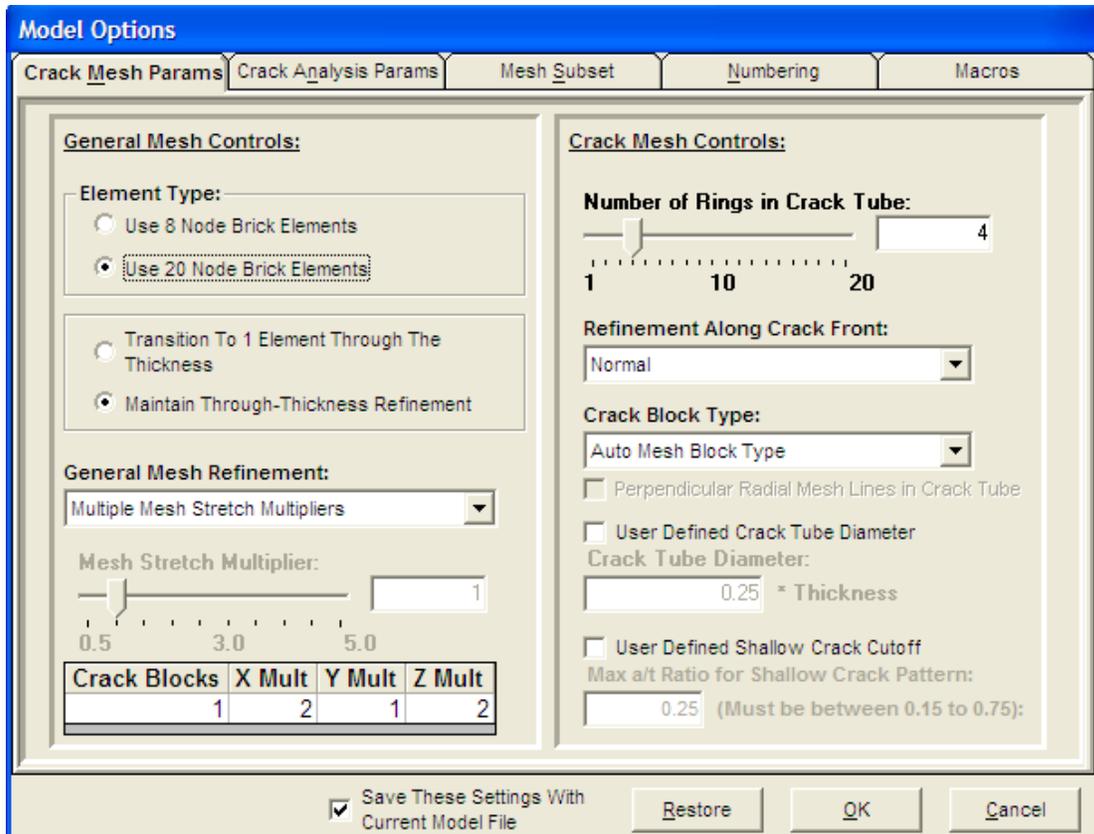


Figure 110 General mesh controls

General Mesh Controls

Element Type

The recommended element type is the 20 node brick, but the 8 node brick can be selected if desired. However, it is strongly recommended that the 20 node brick be selected if the mesh transitions to one element though the thickness (see below).

Mesh Transition Option

In order to reduce the number elements in the model, the mesh can transition to one element through the thickness. Alternatively the mesh pattern from the crack region can be extruded through the model. Examples of both mesh patterns are shown below.

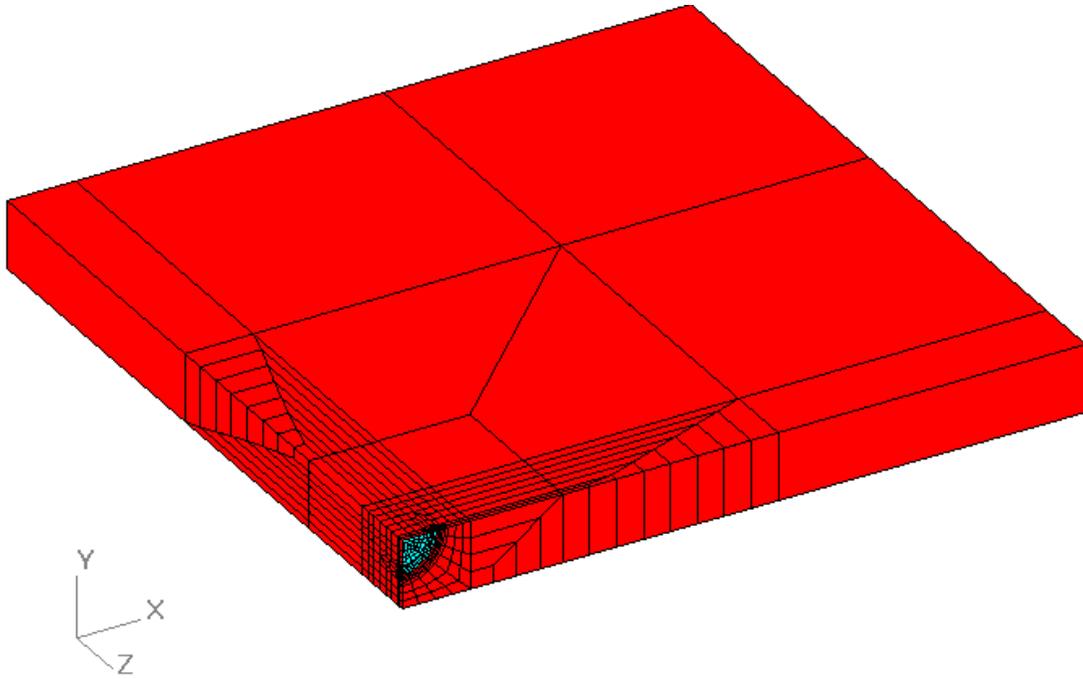


Figure 111 Transitioning to(1) element through the thickness.

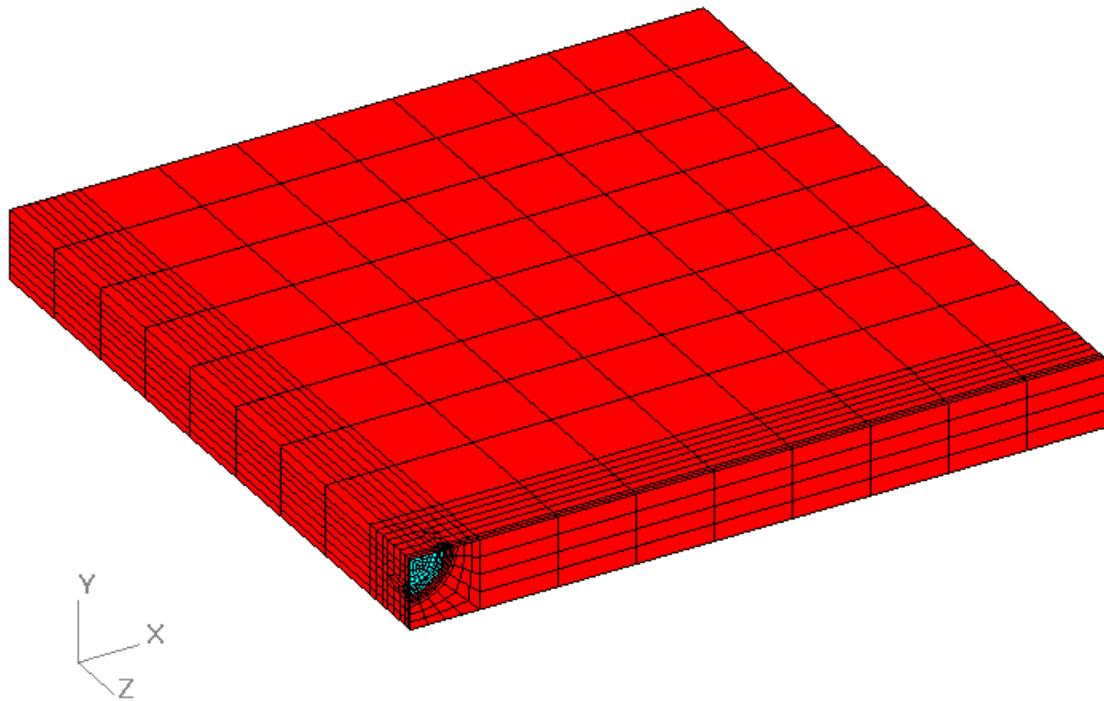


Figure 112 Maintaining through-thickness refinement by extruding the mesh pattern away from the crack.

General Mesh Refinement

This drop-down list controls the mesh pattern outside of the crack. There are two alternative approaches: mesh stretch parameters or refinement indices.

The mesh stretch parameter is inversely related to the level of refinement. That is, “stretching” the mesh pattern results in fewer elements in a given solid. There are mesh stretch parameters for each of the three dimensions (x, y, & z) of the plate primitive, as well as a stretch parameter for the crack block. All four stretch parameters can be set to the same value, or each can be defined independently. The examples below illustrate the effect of varying the mesh stretch parameters.

Refinement indices are proportional to the level of refinement. The values N_x , N_y , and N_z are related to the number of subdivisions in the x, y, and z directions, respectively. When using the fatigue crack growth module, it is recommended that the refinement indices be used rather than the mesh stretch parameters. The latter are scaled to the crack size, so the mesh pattern outside of the crack will change dramatically as the crack grows. With refinement indices, the mesh pattern outside of the crack region will remain relatively constant with crack growth.

An important note about coordinate systems: The general refinement parameters relate to a local coordinate system defined from the crack plane. The crack is always on the local x-y plane, and z is the local normal to the crack plane. For the special case of a flat plate, the local coordinate system is the same as the global system. For a circumferential crack in a cylindrical shell, the local y direction is through the shell thickness, and the local x & z directions are reversed from the global x & z.

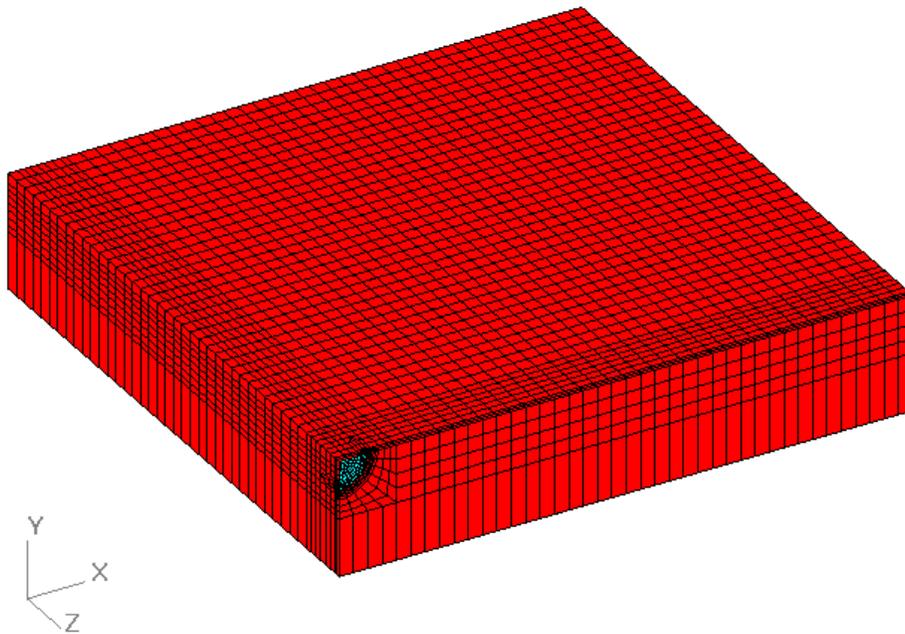


Figure 113 All mesh stretch parameter set to 1.0

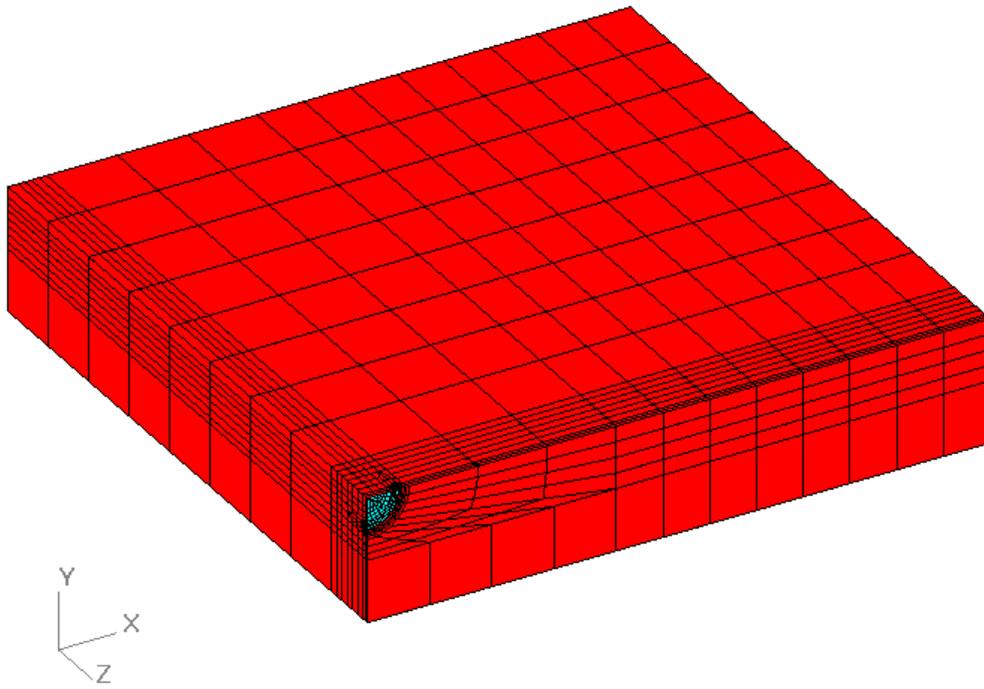


Figure 114 All mesh stretch parameters set to 4.0.

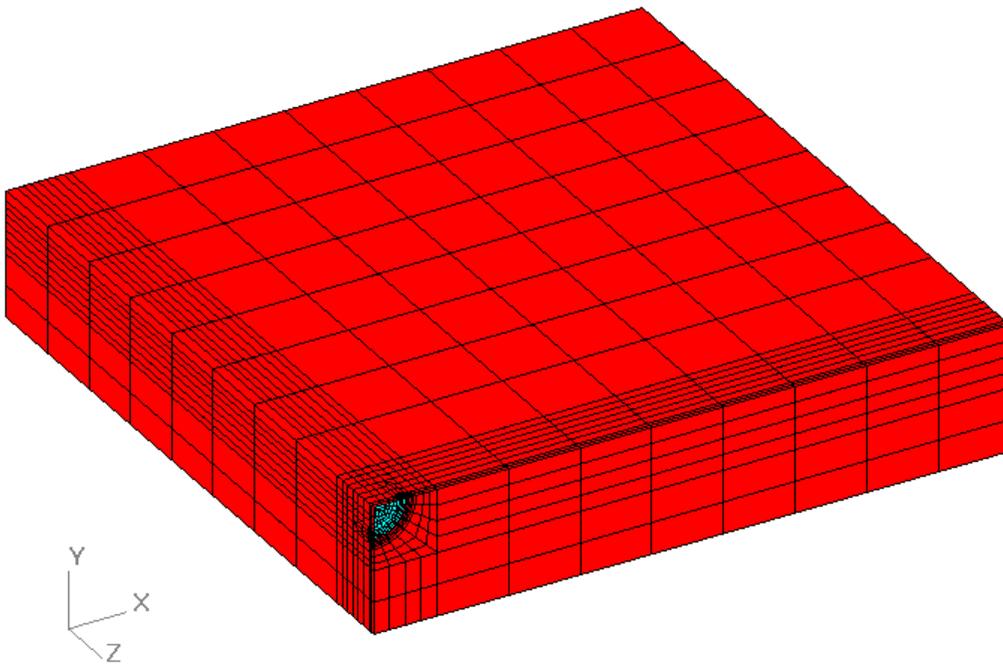


Figure 115 Variable mesh stretch multipliers. Crack Block = 1.0, X Multiplier = 4.0, Y Multiplier = 0.5, Z Multiplier = 4.0

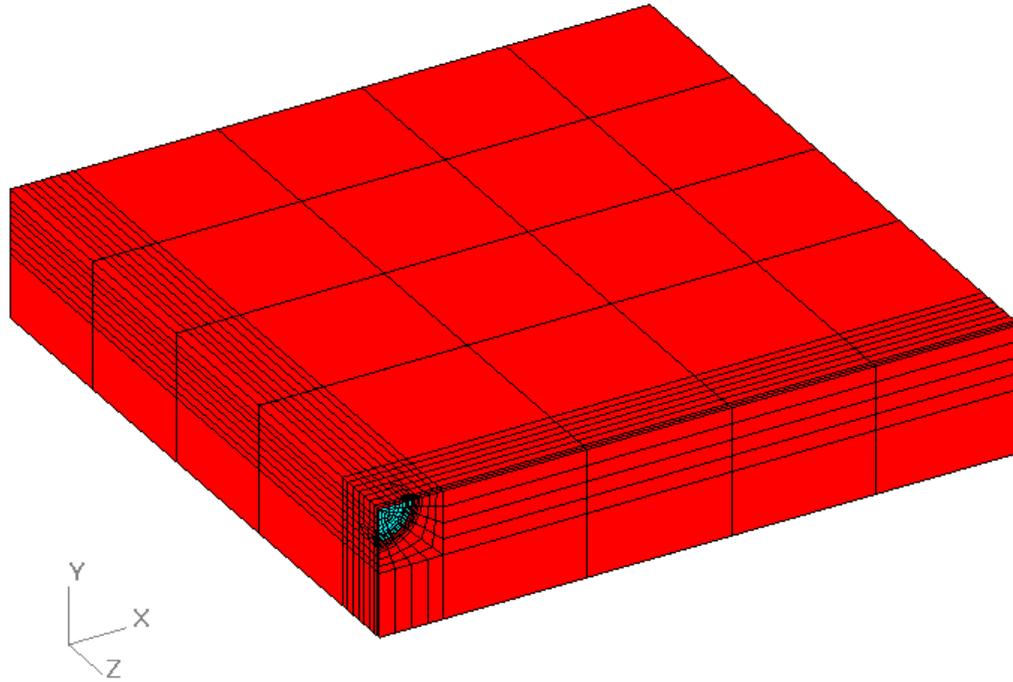
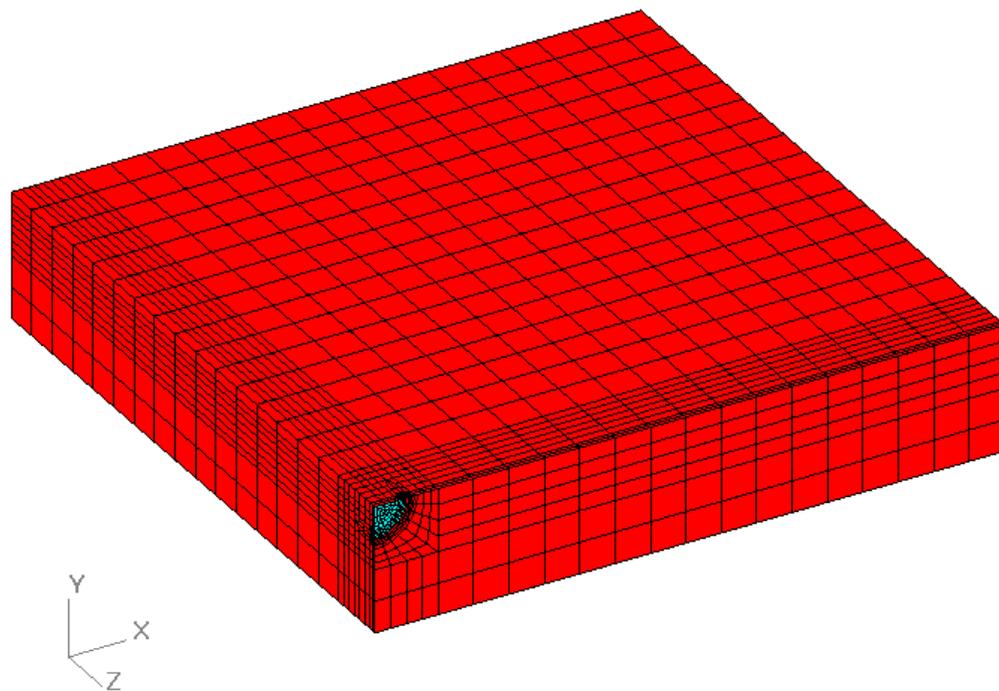


Figure 116 Refinement indices. $N_x = N_y = N_z = 1.0$



Macro Text for ANSYS Solve

If additional boundary conditions or solution data is needed in an ANSYS crack mesh input file, the desired text can be entered in the Tools/Model Options/Macros text form. Activate the macro text with the check box “Replace SOLVE command in ANSYS Input File With Macro”. A full listing of the macro text is shown in the figure below. ANSYS also supports an input command to read data from other text files, so just the /INP command can be entered in the macro text.

The given macro text will replace the usual SOLVE command in the ANSYS format input file created by the crack mesh generator. Be sure to include the SOLVE command in the macro text so that ANSYS will run the analysis. If the SOLVE command is omitted from the macro text it will not be present in the ANSYS input file, and the solution will not occur.

The SOLVE command can also be omitted from the ANSYS format input file by using the check box in the Tools/Program Options/FEA Options form.

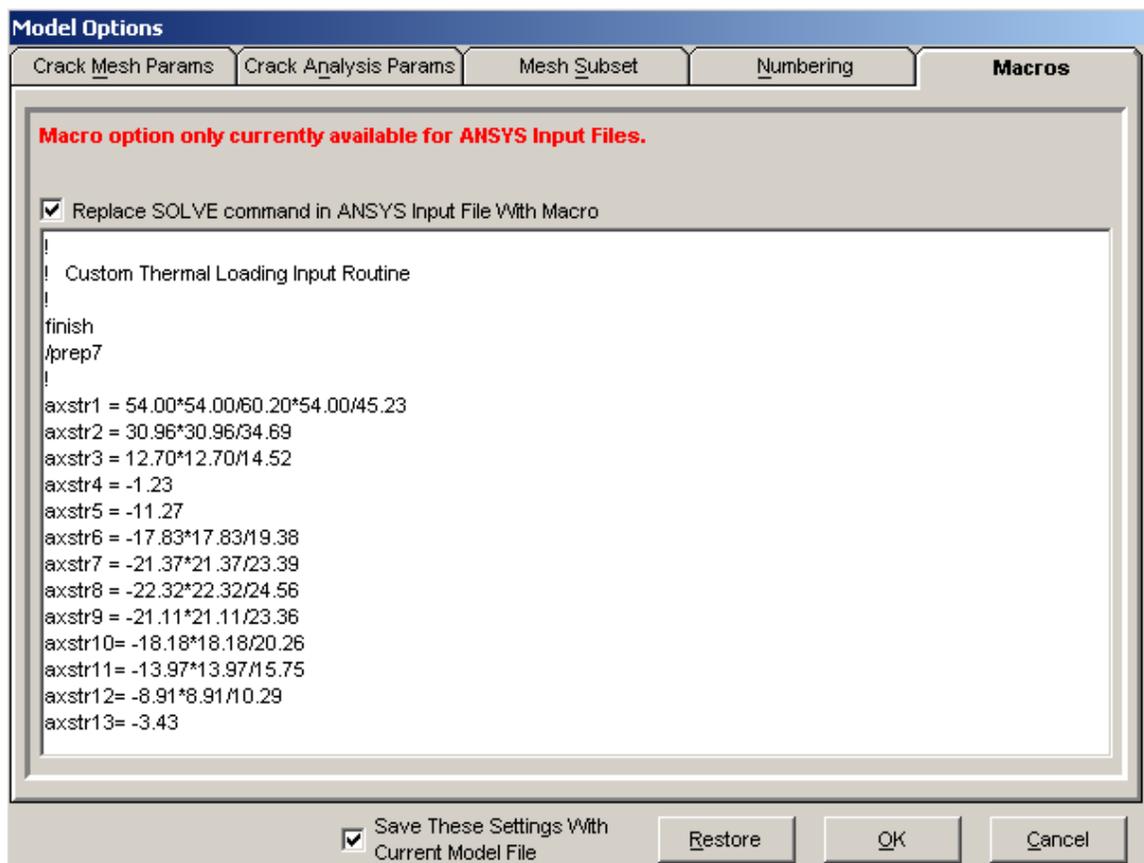


Figure 118 Custom macro text to replace the ANSYS SOLVE command in the input file.

Macro Text for Warp3D and Abaqus

If additional data is needed for a crack mesh to run as a stand-alone analysis, or with the master mesh file combining option, activate the check box and add the extra Warp3D or Abaqus data in the Tools/Model Options/Macros form. The macro text can be added to several locations within the input file by using the appropriate key word for each section of data. Enter the key word as a comment line since it is repeated in the input file, and enter the key word twice to indicate the start and finish of that section of macro text data. The data and syntax entered in the macro needs to be specific to the selected FEA solver: Warp3D or Abaqus.

The extra macro text data is useful in a user-defined geometry model where more constraints or loads are needed. Add single node constraints or forces, or add analytic rigid surfaces to the model. The FEA solver sub-file or read file can also be used where the macro text adds the syntax to read data from another file. The macro text option can be used with the master mesh combining option.

The macro text key words available for each FEA solver are listed in the input file and mesh log file when the macro text check box is selected. Look for “macro text information” in the input file. The location of each macro text section can be identified by including each key word in the macro text (use each key word twice), build the mesh, and look in the input file for the “macro text” label. Then add the additional data within the appropriate section of the macro text.

The available macro text key words for the Abaqus input file are:

- add abaqus model data
- add abaqus constraints
- add abaqus node forces
- add abaqus element loads

When adding reference nodes for analytic rigid surfaces, renumber the crack mesh to start at a higher initial node ID number like 3 or 4, and add the new reference nodes within the model data as node ID 1, 2, 3, etc.

An example of the Abaqus macro key words to enter in the Macros form; “***” is the Abaqus comment:

```
***
** add abaqus model data
   ... add syntax here ...
** add abaqus model data
**
** add abaqus constraints
   .. add constraint syntax here ...
** add abaqus constraints
**
```

The available macro text key words for the Warp3D input file are:

- add warp3d model data first
- add warp3d constraints
- add warp3d multipoint
- add warp3d node forces

- add warp3d element loads
- add warp3d model data second
- add warp3d compute displacements
- add warp3d step <n> (where step <n> is the step index: 1, 2, 3, ...)

The “add warp3d model data first” is located after the element blocking and is useful to adding named lists to select nodes in the crack mesh. The named list option in Warp3D can be used to find single nodes or find sets of nodes and assign them to a label (a node set). The list label can be used to add constraints and loads in the other macro text sections. The “add warp3d model data second” is located after the loading data and before the analysis step data and is a good location for adding analytic rigid surfaces to the model.

An example of the Warp3D macro text key words to enter in the Macros form; “c” is the Warp3D comment:

```
c
c add warp3d model data first
    ... add syntax here ...
c add warp3d model data first
c
c add warp3d constraints
    ... add syntax here ...
c add warp3d constraints
c
```

Subset Mesh

To speed up the crack results post processing; especially for results computed on remote computers, a subset of the crack mesh can be selected in *FEACrack* when generating the FEA input file. The mesh subset gives a smaller result file to make the transfer of result files from a remote computer back to a PC easier. The subset mesh results can then be used for crack result post processing using *FEACrack* on a PC. The mesh subset option is available when results are output to a *WARP3D* binary packets file or *Abaqus* result file. Select the *WARP3D* packets file or *Abaqus* file output option in the Tools/Program Options/Analysis Options form. The size of the subset mesh region is chosen in the Tools/Model Options/Mesh Subset form.

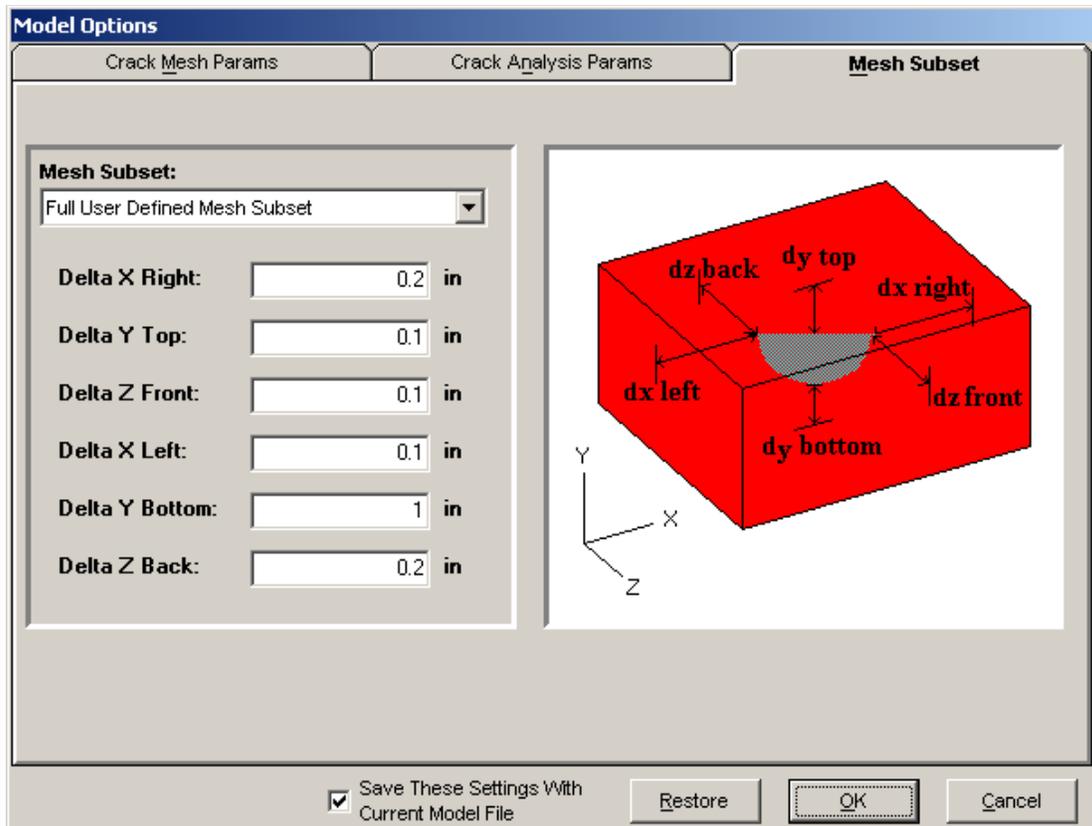


Figure 119 Mesh subset sizes given from the ends of the crack.

The subset mesh includes the region around the crack front, including the crack front nodes, and crack face elements. Increasing the size of the subset region will include more of the surrounding mesh. There are several options in the Mesh Subset menu to select the subset region size: the entire mesh, a default size, entering a single size value, or entering 6 custom size values.

Selecting the entire mesh for the subset is intended for use with the user-defined geometry when the crack mesh is added to a larger model. All of the crack mesh will be in the subset and none of the larger model. The default subset size uses the crack dimensions to set the uniform region size; this option is useful for small cracks in large models. A single value can be entered as the uniform size of the subset region; this option allows the easiest custom subset region size. To fully customize the subset region size, enter all 6 values to define the subset region around the crack. The subset region is defined as a rectangular box in the global Cartesian coordinates. The subset box is defined by adding the given delta x, y, and z values to the maximum and minimum crack node coordinates. For example, a shallow surface crack may not include all of the mesh through the thickness using a default or small single value. Enter a large Delta Y Bottom value to include all of the mesh through the thickness below the crack. Or for a cylinder with an internal surface crack, use a large Delta Y Top to select the mesh through the thickness above the internal crack location.

Since the subset mesh is obtained from the completed crack mesh, select the subset region size based on the geometry dimensions, crack location, and crack orientation. The crack mesh is renumbered so that the subset mesh is listed first in the mesh data. The subset mesh flag and mesh sizes are written to the Mesh Size Data comment block at the top of the input file. Since the

subset mesh is numbered consecutively starting and node and element ID 1, a minimum amount of RAM is used during post processing. The subset mesh option does not affect the standard *WARP3D* output file, the mesh results are deactivated in the *_wrp.out file when the packet file is used.

When loading the results, select the *WARP3D* Packets file type (*.bpf and *.tpf file extension for the binary and text format packets file) or *Abaqus* *.fil file in the FEA Results/File Selection form. The subset mesh results are contained within the result file. The *FEACrack* post processing will display the subset mesh for the stress and strain color map and all crack results. Data outside the subset mesh is ignored by the post processor.

A small flat plate surface crack mesh (cube shape, quarter symmetric mesh) is shown in the figures below to demonstrate the different subset mesh sizes. In order, the figures show all of the mesh, the default subset, the subset using a single size value (0.1 in), and the custom subset using 6 size values (0.2, 0.1, 0.1, 0.1, 1.0, 0.2 in). The custom delta y bottom value, 1.0 in, is large enough to include the mesh in the remaining thickness below the surface crack.

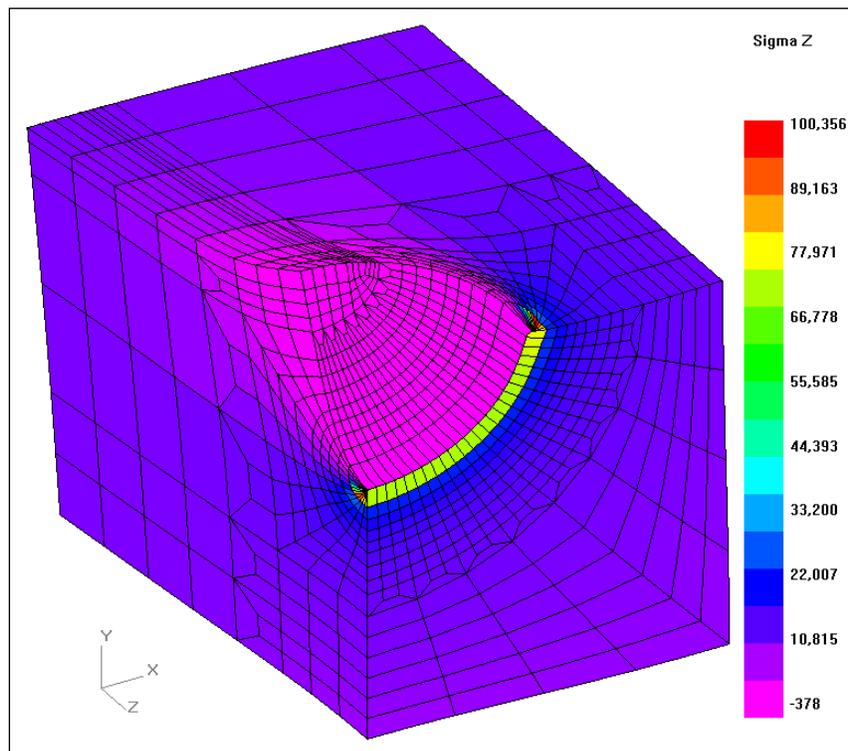


Figure 120 All of the surface crack mesh.

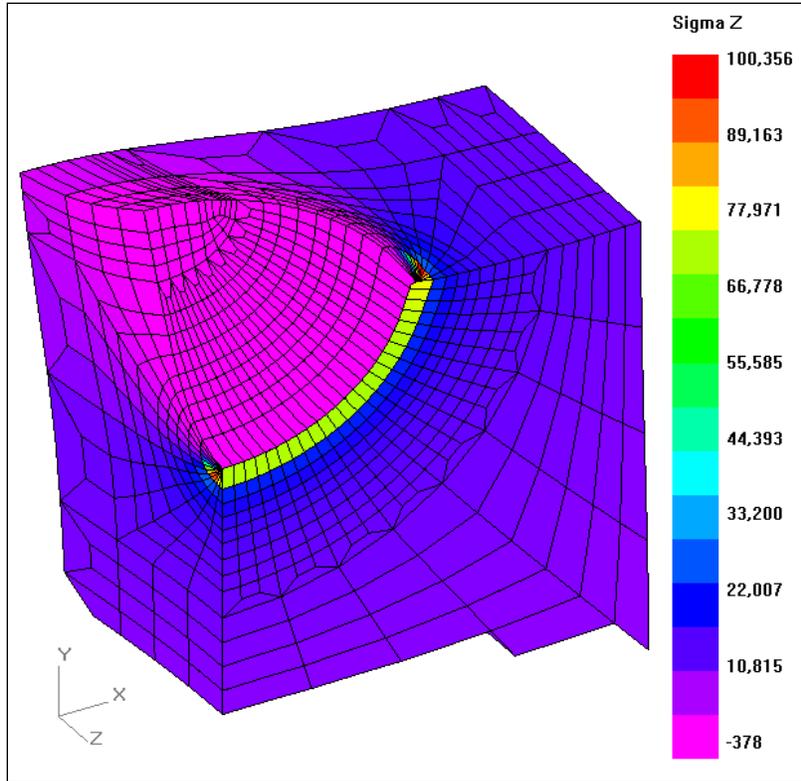


Figure 121 Default subset mesh, subset uniform region size based on the crack size.

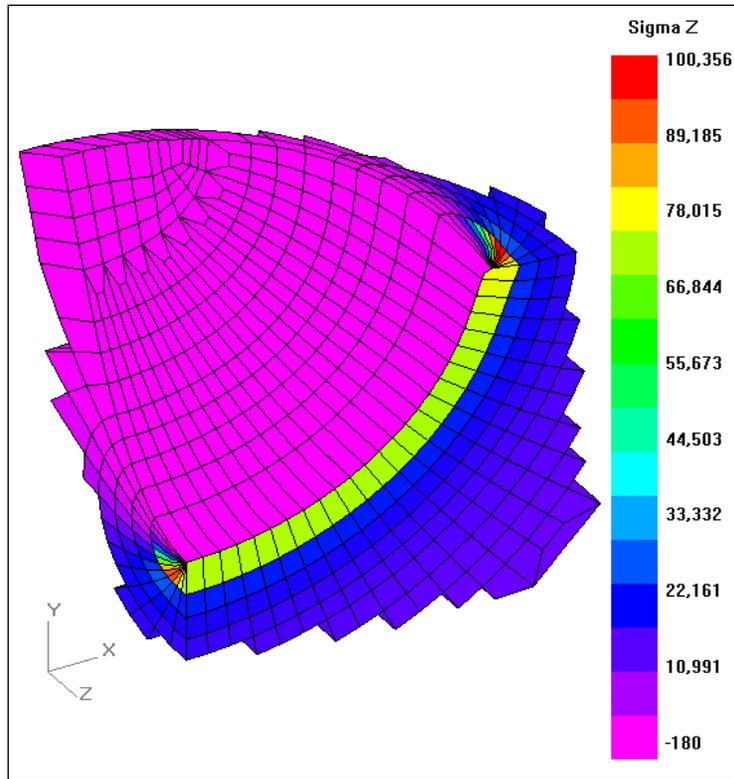


Figure 122 Uniform subset region defined by a single value.

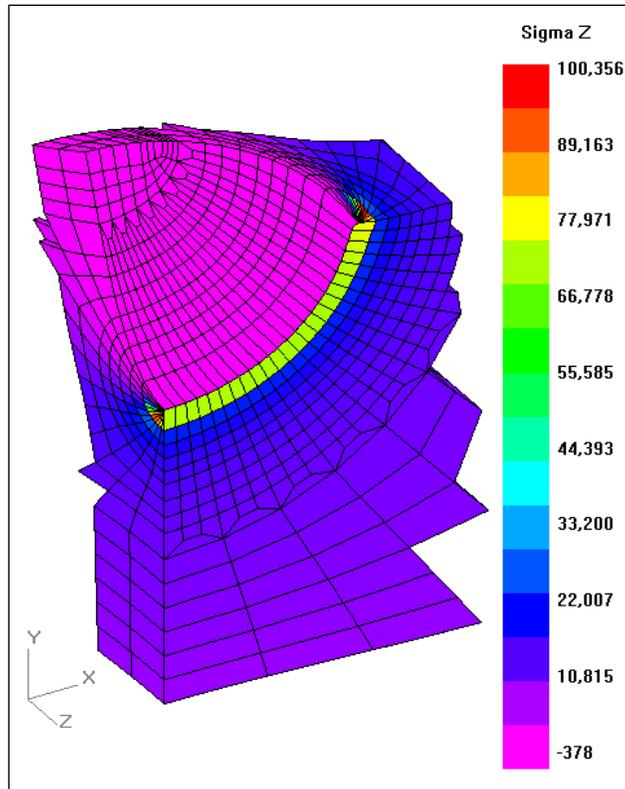


Figure 123 Custom subset mesh size defined by 6 values; large Delta Y to include the mesh through the thickness.

WARP3D Files For Analysis On A Remote Computer

The crack mesh analysis can be run on a remote computer by transferring the input file generated on a PC to the remote computer (a faster *UNIX* work station for example). To make the transfer of results back to the PC easier, just the *WARP3D* packet file needs to be returned to the PC for post processing. A detailed example is given below.

When running an analysis on a *UNIX* computer, it may be necessary to convert the text format of the input file (line termination with line feed only versus line feed plus carriage return in the text file). After transferring the input file from the PC to the *UNIX* computer, use the “to_ UNIX” command to convert the Windows/DOS format input text file from the PC to a *UNIX* format text file, then run the analysis using the *UNIX* format input file.

When the binary packets file format is incompatible between the remote computer and the PC (*UNIX* and Windows binary files are incompatible), the binary packets file must first be converted to a text format packets file on the remote computer, using the packet reader program “packet_reader_QR.exe”. This command line program is available on request; please contact technical support (refer to the telephone number and e-mail address on the title page). The packet reader program is based on and similar to the default packet reader program provided with *WARP3D*. It reads the *WARP3D* binary packets file (*.bpf) and writes the results to a text packet file (*.tpf) in the format supported by the *FEACrack* post processor. Before returning the text packets file to the PC, convert the text file format from *UNIX* to Windows/DOS format using the

“to_dos” UNIX command. Then the text packets file can be returned to the PC for post processing. *FEACrack* will read results from both the *WARP3D* binary packets (Windows compatible format) and text packets files.

When reading a packet results file in *FEACrack*, the result file must have the same base name as the input file, and be in the same directory as the input file. The input file contains several comment data blocks at the top of the file that contain crack data required for post processing. For example, if the input file name is “cylinder_mesh_wrp.inp”, the binary packets file name must be “cylinder_mesh_wrp.bpf”, or the text packets file name must be “cylinder_mesh_wrp.tpf”.

Run the `packet_reader_QR.exe` on the UNIX computer from the command line; it may be convenient to put a copy of the `packet_reader` program in the same directory as the packets file. Screen shots from running the `packet_reader` program on a UNIX computer are shown below.

- First, enter the binary packets file name.
- Second, select the output packets by number from the list to be included when the text packets file is written. As a recommended short cut enter “0” to include all available packets when writing the text packets file.
- Third, enter the text packets file name. As a recommended short cut, the default text packets file has the same file name with a *.tpf file extension, which is used by *FEACrack*.

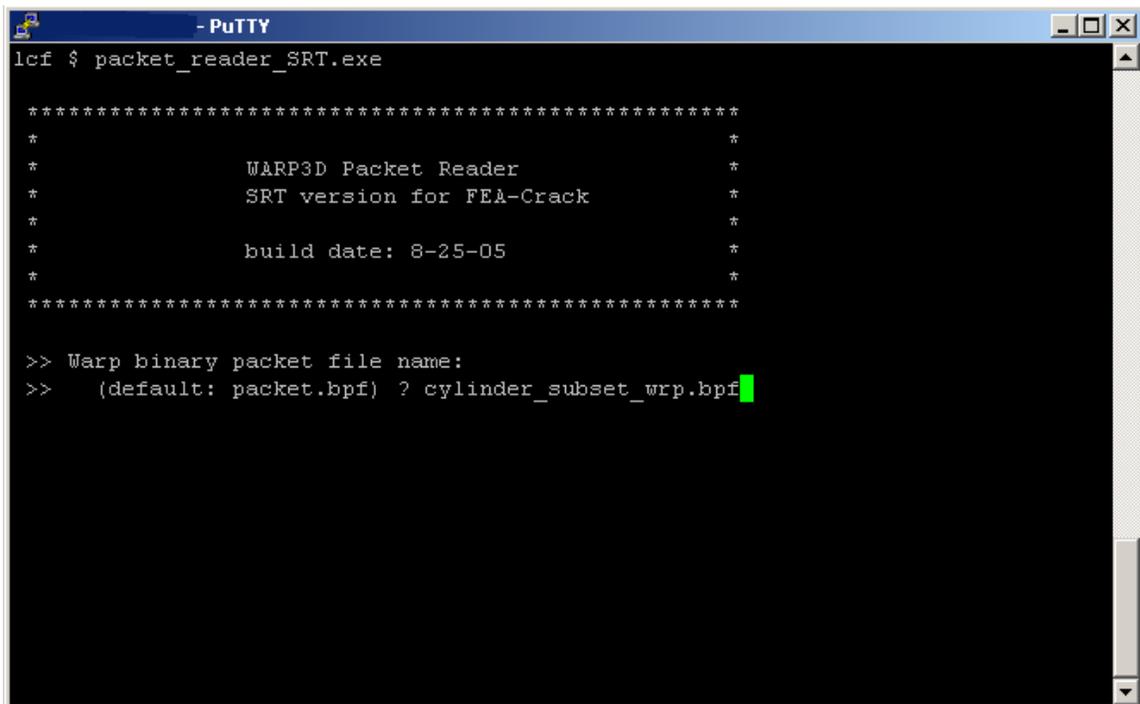


Figure 124 Step 1, run the packet reader from the command line, enter the binary packets file name

```
- PuTTY
ctoa constant front step summary 8
ctoa front step summary 9
killed cohesive element 10
element displacements 11
element velocities 12
element acclerations 13
element reactions 14
element stresses 15
element strains 16
domain J-integral results 17
ctoa non-constant front release 18
ctoa constant front release 19
killed gurson element 20
killed stress-mod crt strain elm 21
smcs elem status 22
smcs elem status (auto ld red) 23
accumulated step pattern factors 24
I-integral SIF results 27
I-integral T-stress results 28

** Select all available packets, enter 0 **

>> Enter desired packet number: █
```

Figure 125 Step 2, use short cut, enter “0” to select all available packets

```
- PuTTY
killed stress-mod crt strain elm 21
smcs elem status 22
smcs elem status (auto ld red) 23
accumulated step pattern factors 24
I-integral SIF results 27
I-integral T-stress results 28

** Select all available packets, enter 0 **

>> Enter desired packet number: 0

>> Name of text packet file containing results:
>> (default: cylinder_subset_wrp.tpf)?

>> Results file opened

>> Packets file closed

>> Results file closed

>> END OF FILE...program terminating in main

lcf $ █
```

Figure 126 Step 3, use default text packets file name, write the text packets file

Cylinder Subset Mesh Example

A quarter symmetric cylinder with an internal surface crack is used as an example to show the details of the mesh subset method. The model and input files are available for download from the Quest Integrity Group web site ([cylinder_subset_mesh.zip](#)). This example uses an SGI UNIX computer as the remote computer to run the *WARP3D* analysis. The cylinder crack mesh is shown in the figure below.

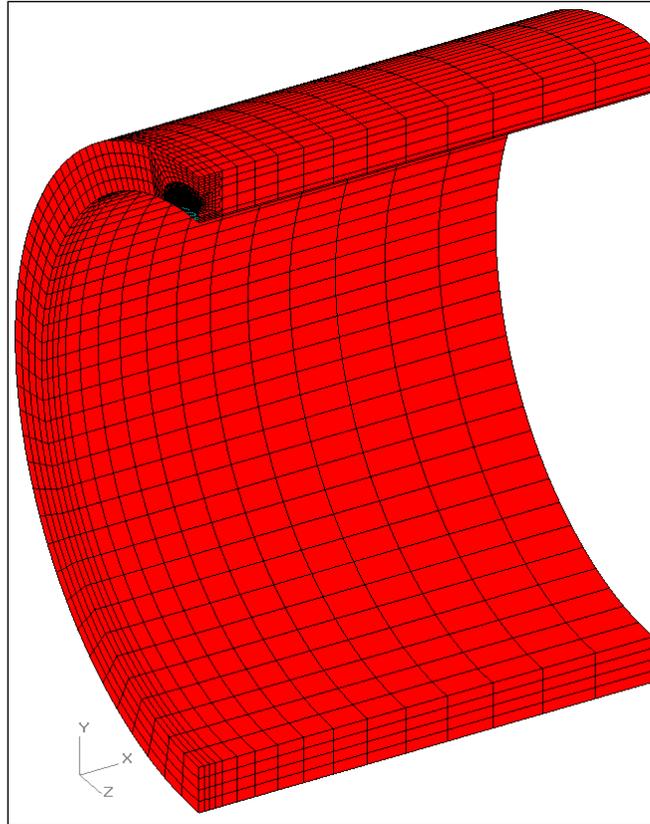


Figure 127 Internal circumferential surface crack in a cylinder

The custom subset sizes are: delta x right = 0.2, delta y top = 0.5, delta z front = 0.1, delta x left = 0.1, delta y bottom = 0.1, delta z back = 0.3 in. These custom subset sizes are entered in the Mesh Subset form, shown below.

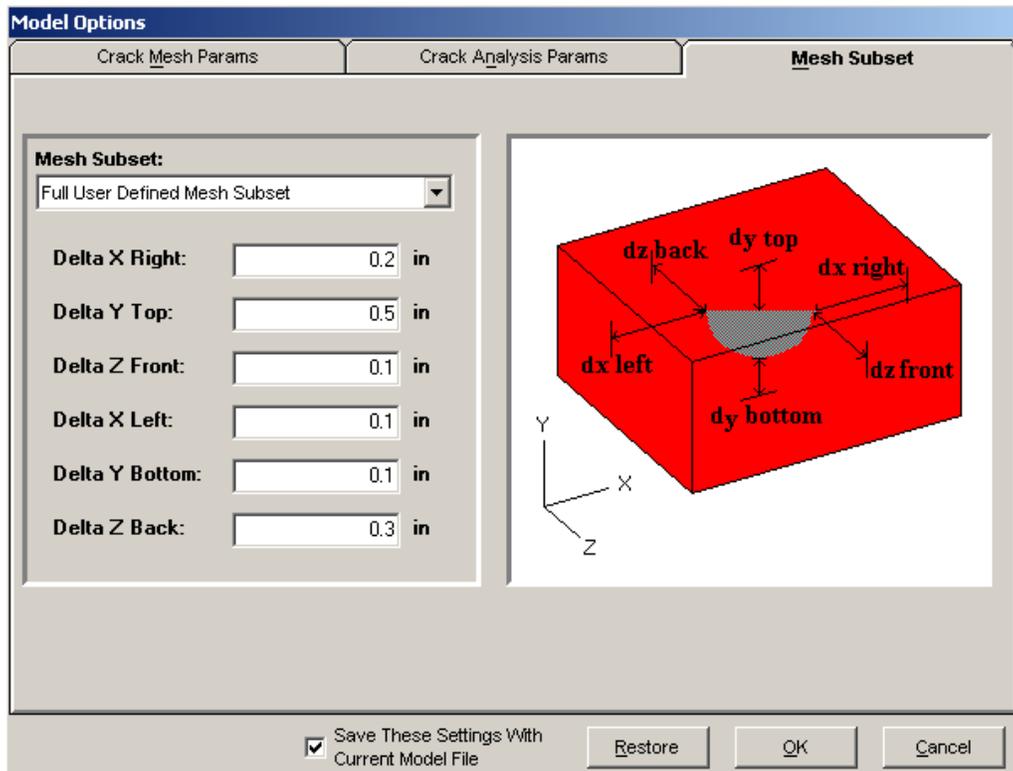


Figure 128 Enter 6 size values for the subset region

Steps for the subset mesh analysis:

1. Build the *WARP3D* crack mesh input file using *FEACrack* (cylinder_subset.elt model file); activate the *WARP3D* binary packets output check box in the Tools/Program Options/Analysis Options form. Activate and enter the subset mesh sizes in the Tools/Model Options/Mesh Subset form.
2. Move the *WARP3D* input file (cylinder_subset_wrp.inp) from the PC to the UNIX computer using a file transfer program, or secure FTP if required.
3. On the UNIX computer, use the “to_UNIX” command to convert the Windows/DOS format *WARP3D* input file to UNIX text format:
`$to_UNIX cylinder_subset_wrp.inp cylinder_subset_UNIX_wrp.inp`
4. Run the *WARP3D* analysis on the UNIX computer; it may be necessary to run the analysis from the command line, and it may be convenient to use a script to run the *WARP3D* analysis, such as the “warp_ser” script:
`$warp_ser < cylinder_subset_UNIX_wrp.inp > cylinder_subset_UNIX_wrp.out`
5. After the analysis is finished, run the packet_reader_QR.exe program (available on request) to read the *WARP3D* binary packets file and write the text packets file; the binary packets file name is cylinder_subset_wrp.bpf, which is given in the *WARP3D* input file; the cylinder_subset_wrp.tpf text packets file is created by the packet_reader program; convert the UNIX format text packets file to Windows/DOS format using the “to_dos” UNIX command:
`$to_dos cylinder_subset_wrp.tpf cylinder_subset_wrp_dos.tpf`
6. Move the Windows/DOS format text packets file from the UNIX computer back to the PC for post processing using *FEACrack*; check that the packets file name matches the

base file name of the input file, and put the text packets file from the remote computer in the same directory as the input file.

The subset mesh sigma X (cylinder axial stress) results and deformed shape due to internal pressure are shown in the figure below. The J-integral results from the standard *WARP3D* output file, the binary packets file (from the PC), and the text packets file (from the SGI UNIX computer) are compared in the plot, shown below. The J-integral results are the same from each results file since the same input file was used for the analysis on the PC and SGI computers.

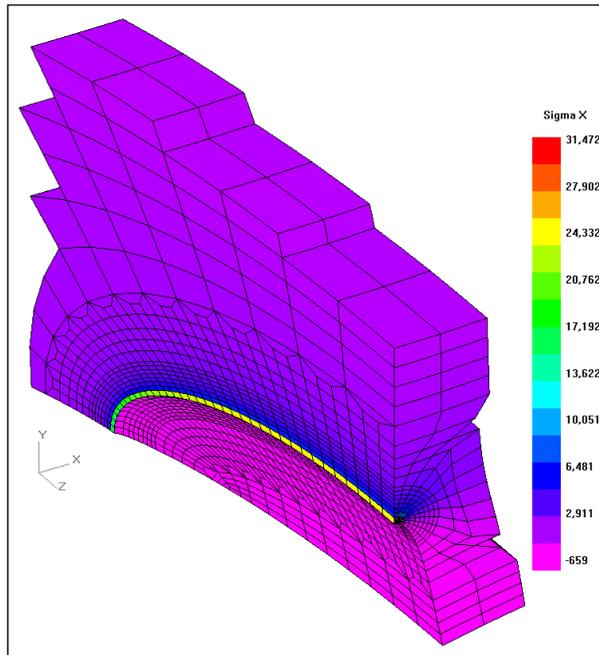


Figure 129 Cylinder internal surface crack subset set *WARP3D* results.

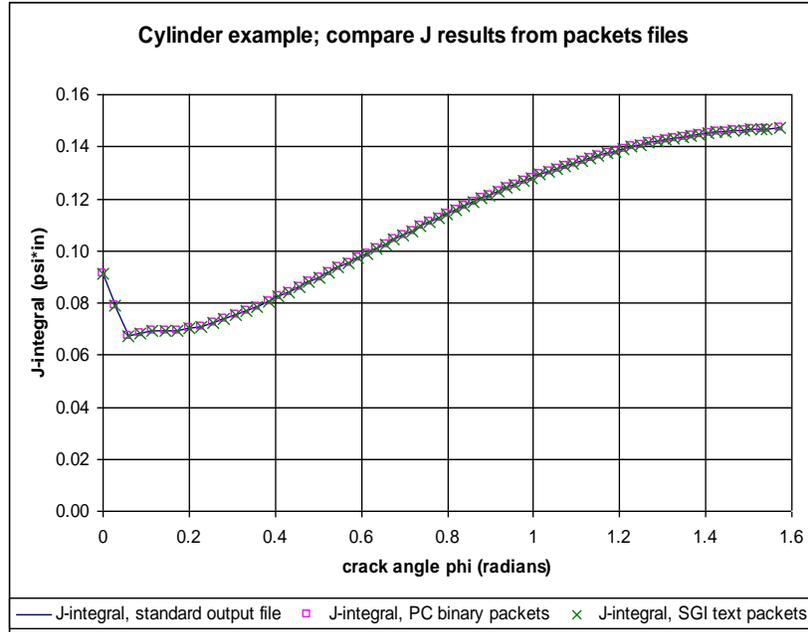


Figure 130 The J-integral results are the same from the WARP3D result and packets files.

Abaqus Subset Mesh

The subset mesh options can also be used with an *Abaqus* mesh. The selected mesh subset is identified by the “node_subset” *NSET node set and the “elem_subset” *ELSET element set in the *Abaqus* input file. These sets are used to limit the requested node and element output to just the subset region of the mesh. During post processing any data outside the mesh subset is ignored; only the subset mesh results and crack front results are displayed.

When transferring the *.fil *Abaqus* results file between computer platforms, the ASCII format result file should be used. The subset mesh results are also written to the *.dat *Abaqus* output file; the *.dat file also contains the crack data blocks used during post processing. Both the *.fil and *.dat files should be returned from a remote computer for *FEACrack* post processing.

The above cylinder subset mesh example was meshed for *Abaqus*. The J-integral results are compared in the plot below. The crack results are the same for the full mesh and for the subset mesh.

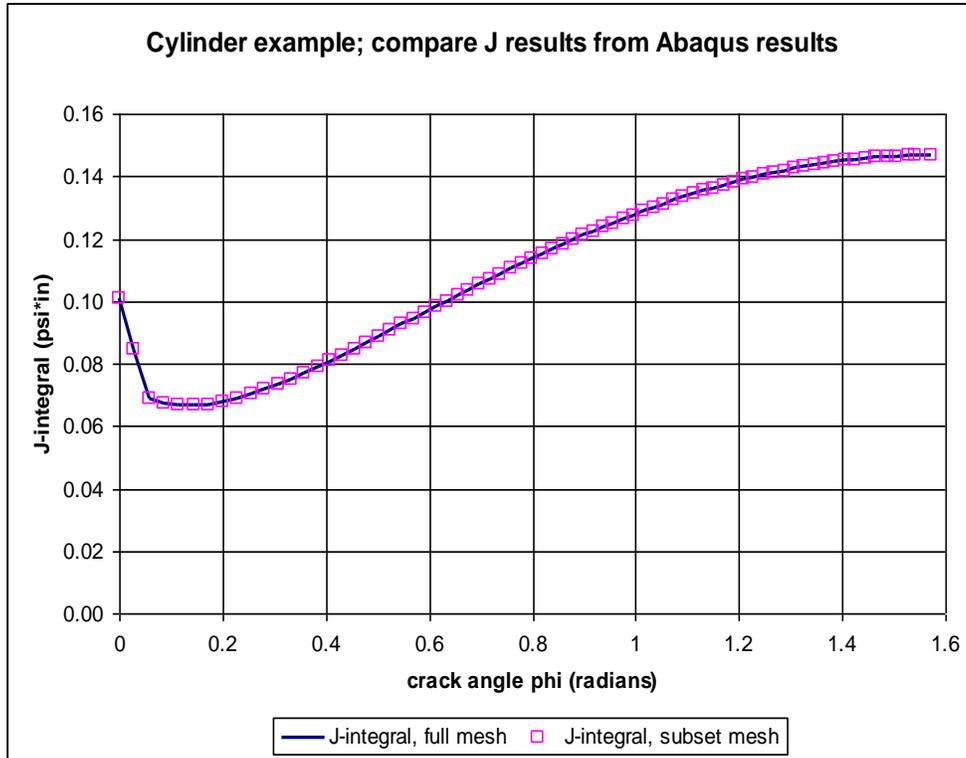


Figure 131 Cylinder subset J-integral results from Abaqus

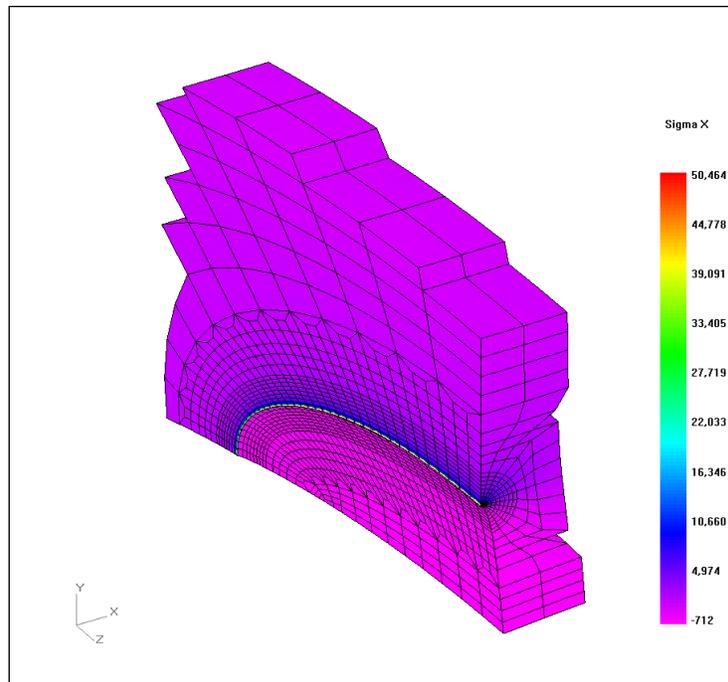


Figure 132 Cylinder internal surface crack subset set Abaqus results.

A flat plate with a surface crack and elastic-plastic material also gives the same J-integral results for either the full mesh or subset mesh results. The J-integral is plotted versus the applied sigma Z tension load in the plot below.

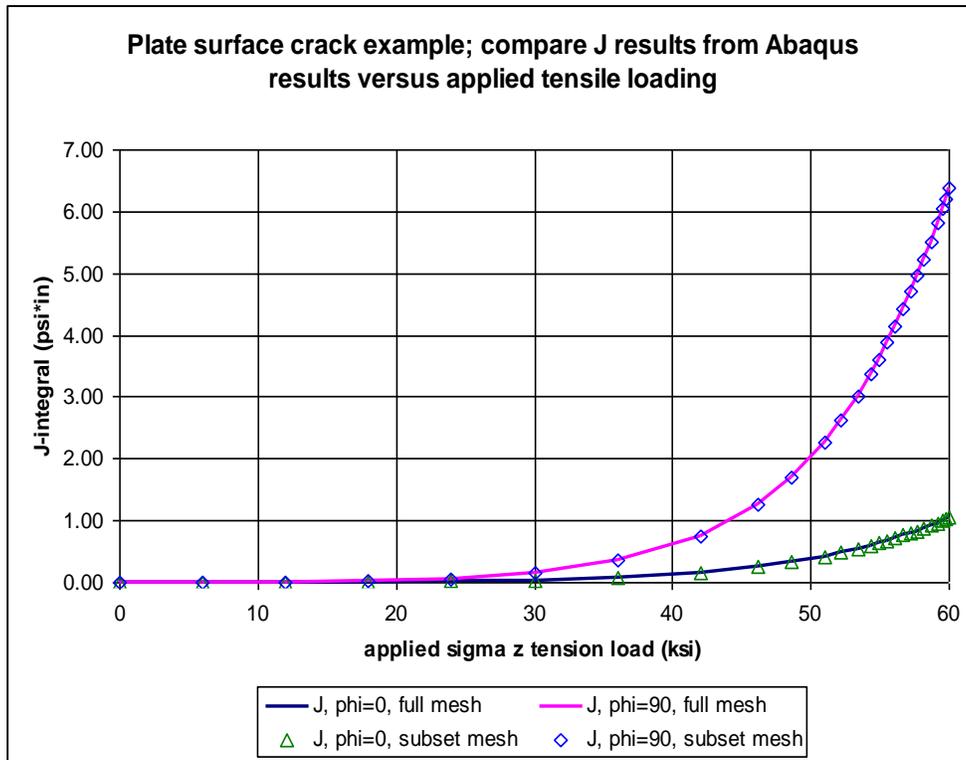


Figure 133 Plate with surface crack, plasticity J results, subset mesh comparison.

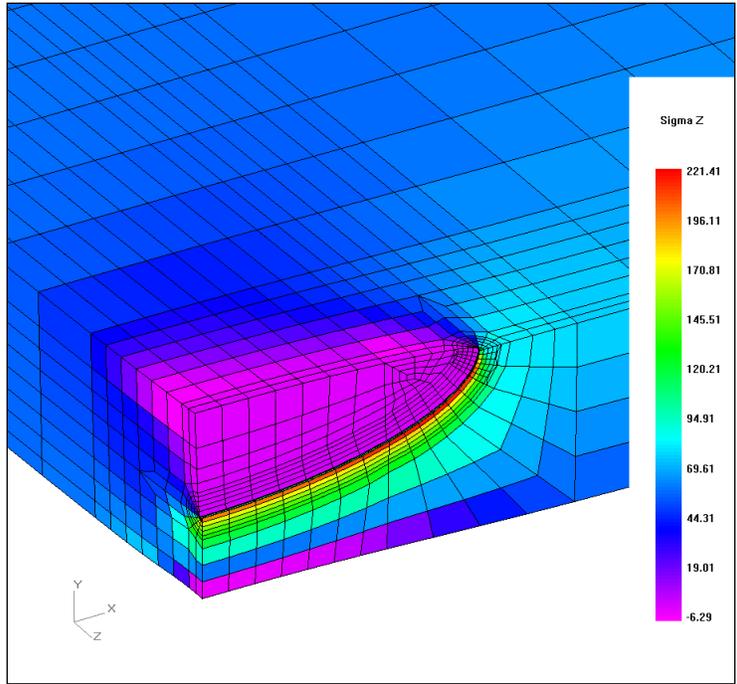


Figure 134 Sigma Z results for the full plate mesh

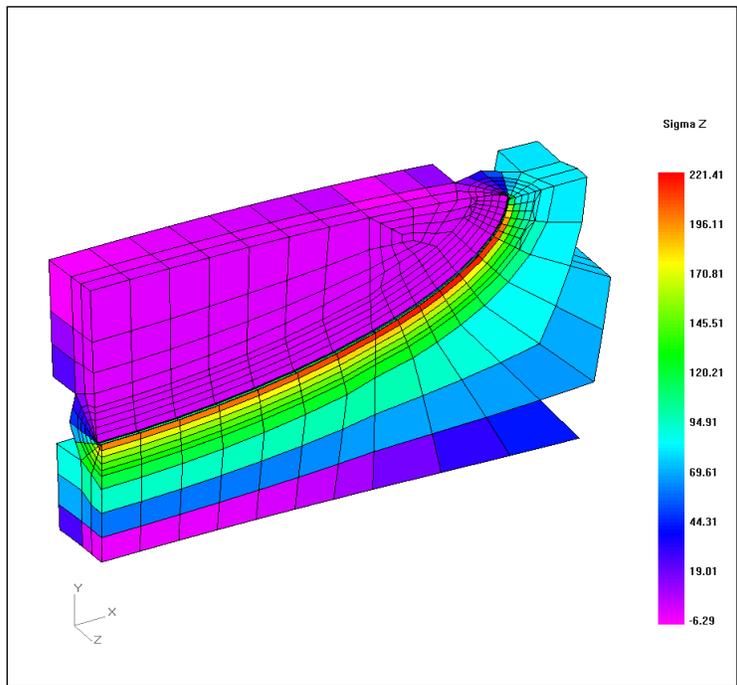


Figure 135 Sigma Z results for the subset plate mesh

Cell Crack Mesh

Cell Crack Mesh Introduction

An example of a cell type semi-elliptical surface crack mesh in a quarter symmetric flat-plate is shown below. Many of the mesh features, such as the cell crack zone, mesh refinement, and optional mesh transitions, can be easily controlled by the various mesh parameters described in the following sections. In the figures below, the crack face elements are colored light blue to show where the initial crack front is located. The through-thickness crack shape is also available in the cell mesh. The cell mesh uses the PRO (parameterized refinement objects) mesh pattern to increase the possible mesh refinement.

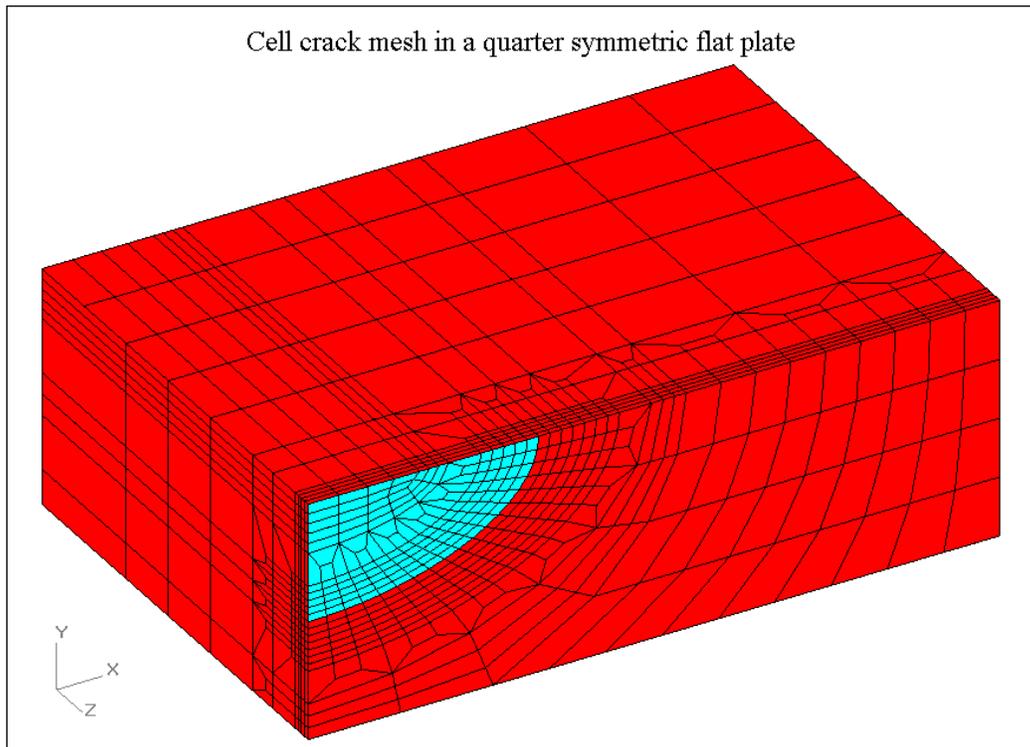


Figure 136 Example of a cell-type crack mesh

The focused crack mesh with collapsed elements at the crack front to compute J-integral values is also available in the PRO mesh as shown in the figure below; the typical focused mesh pattern is used along the crack front instead of the cell mesh pattern. The mesh outside of the crack front region is the same for both the cell and focused PRO crack meshes.

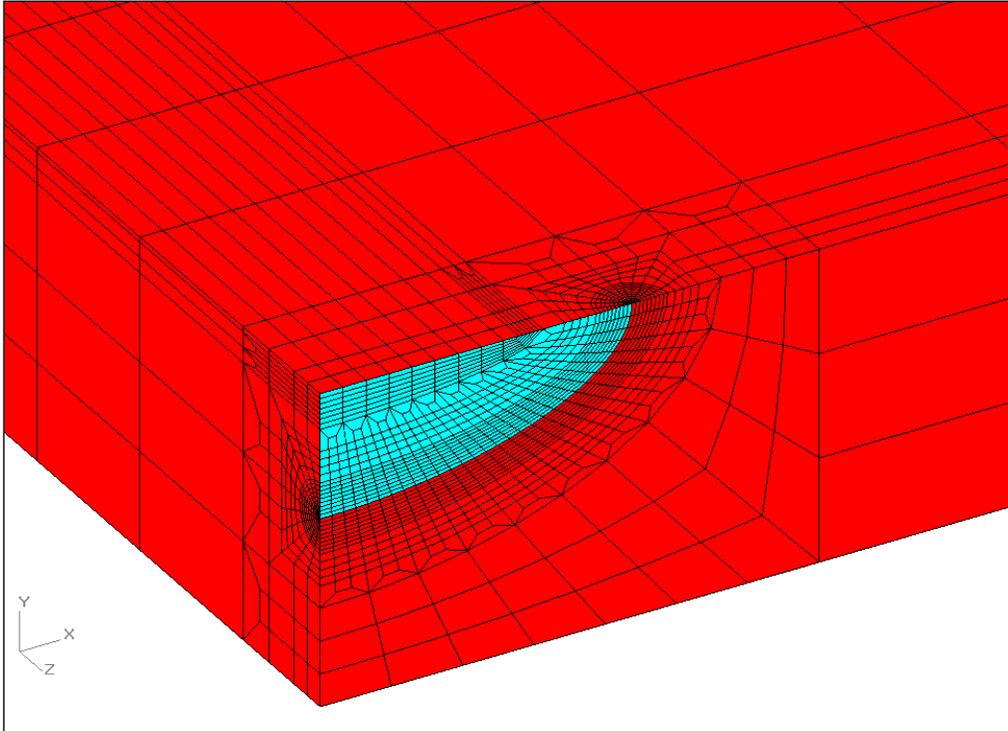


Figure 137 Focused crack in the cell mesh

The PRO crack mesh pattern is controlled by the many available mesh parameters. The parameters are grouped in eight mesh zones within the cell crack mesh. The eight mesh zones are shown in the figures below for the surface and through-thickness crack shapes. The mesh zones are described in the table below. The PRO crack mesh zones are numbered from the crack front zone (cell or focused crack mesh) to the surrounding mesh zones and then to the mesh zones farther away from the crack front zone. The mesh zone numbering also follows the input order of the mesh control parameters.

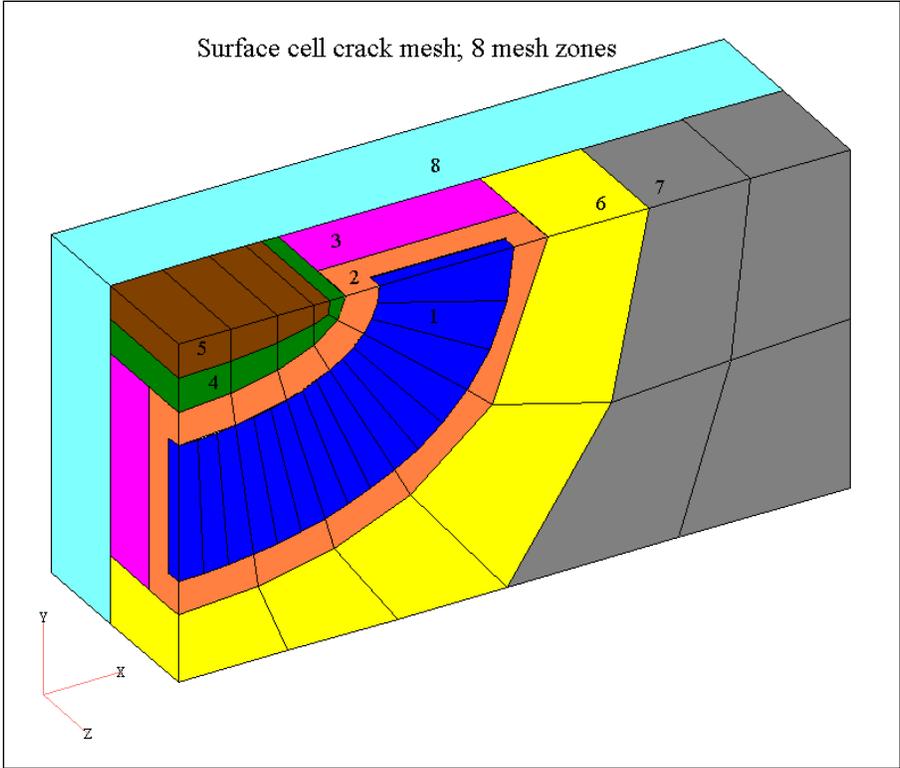


Figure 138 PRO crack mesh zones for the surface crack shape.

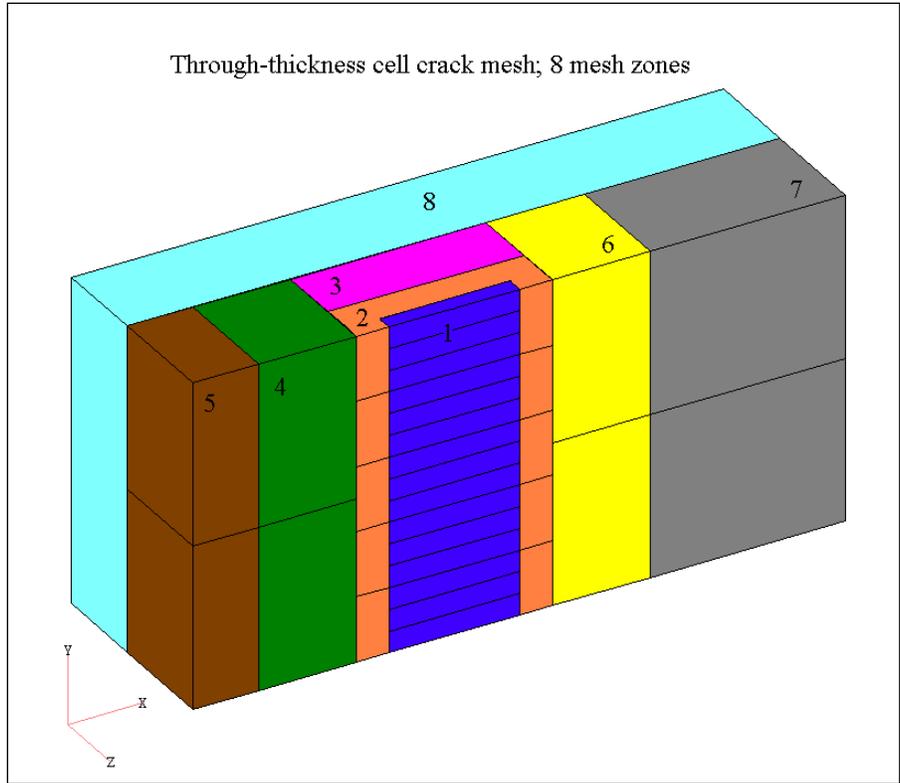


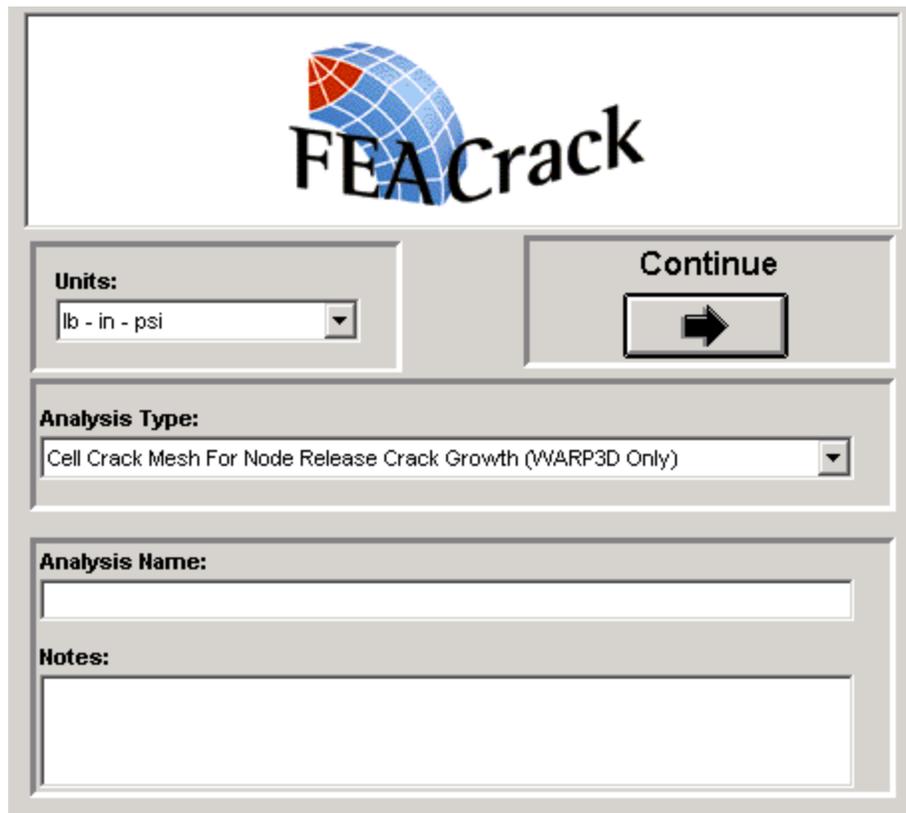
Figure 139 PRO crack mesh zones for the through-thickness crack shape.

<u>Mesh Zone</u>	<u>Mesh Zone Description</u>
1	crack front mesh zone; has a curved grid mesh pattern for a semi-elliptical surface crack; a through thickness crack shape has a vertical crack front; uses the cell or focused mesh pattern
2	crack front transition zone; reduces the number of elements along the crack front direction in the remainder of the mesh; optional
3	crack growth transition zone; reduces the number of elements along the crack growth direction in the remainder of the mesh; allows other depth direction transitions; optional
4	center of the crack face; allows the number of crack face elements to be adjusted independently of the crack front zone; has an optional transition to reduce the number of crack face elements in the remainder of the mesh
5	top of the surface crack face; or left side of the through crack face; affects the mesh pattern on the crack face; no mesh transition possible in this mesh zone; setting the number of elements to zero on the left side of the through crack face deactivates mesh zone 5 in the through thickness crack leaving just mesh zone 4 on the through crack face
6	beside the crack; completes the remaining mesh thickness below the crack front zone for a surface crack; to the right side of the crack front mesh zone for a through thickness crack; has an optional transition to reduce the number of elements in the remaining depth of the mesh; the mesh pattern can be curved or vertical on the right side
7	remaining width; completes the remaining mesh width and allows the number of elements through the thickness on the right edge of the mesh to be chosen for a surface crack; the number of elements through the

- thickness for a through crack is determined by the number of mesh transitions, if any, along the through crack front; has an optional transition to reduce the number of elements in the remaining depth of the mesh; if the mesh width dimension is too small this mesh zone will not be used leaving just mesh zone 6 to the right of the crack
- 8 remaining depth; mesh zone behind the crack front zone and any mesh transitions; set the number of elements to complete the remaining depth of the mesh; no further mesh transitions available in this mesh zone

The parameters for the crack front mesh zone and nearby mesh zones will affect the possible parameter choices for mesh zones further away from the crack. For example, if there are few elements along the crack front, there must then be a smaller number of elements in the crack face, and also in the thickness on the right edge of the crack mesh. Currently, the transition in mesh zone 3 behind the crack zone must be active to allow other mesh depth direction transitions to also be included.

To choose the cell type crack mesh, use the Analysis Type pull down menu on the home form. Currently, the two available choices are for the previously available focused type crack mesh (to compute J-integral and stress intensity K), or the new cell type crack mesh (for node release crack growth). To choose the focused PRO crack mesh select the focused crack mesh option on the home form then select the focused PRO crack mesh in the Model Options form, Crack Block Type menu.



The image shows a software interface for FEACrack. At the top is the logo, which consists of a blue and red globe icon followed by the text 'FEACrack'. Below the logo is a form with several sections: 1. 'Units:' section with a dropdown menu showing 'lb - in - psi'. 2. 'Continue' section with a button containing a right-pointing arrow. 3. 'Analysis Type:' section with a dropdown menu showing 'Cell Crack Mesh For Node Release Crack Growth (WARP3D Only)'. 4. 'Analysis Name:' section with a text input field. 5. 'Notes:' section with a text area.

Figure 140 Select the cell crack mesh on the home form

PRO Mesh Parameters

The PRO crack mesh parameter input is divided into two groups in *FEACrack*. The user must always enter the basic input values for the number and size of the elements in the crack front mesh zone. The remaining advanced mesh parameters are optional; the user can use the default parameter values or modify selected parameters to control the PRO crack mesh.

The PRO crack mesh controls begin on the Geometry/Cracks/Cell Mesh form, shown below. For example, the element sizes for a cell mesh crack front mesh zone are entered as Delta C (in the crack length direction) and Delta A (in the crack depth direction). The number of elements in the crack growth direction is constant along the crack length and depth directions and defines the number of elements across the cell crack mesh zone. The number of elements along the crack front is also constant across the cell crack mesh zone, from the initial to final crack fronts. If the Advanced Mesh Refinement Options is left unchecked at the bottom of the form, the remaining cell crack mesh parameters will be set to default values. Click on the check box to activate the Advanced Mesh form to view or adjust the other cell crack mesh parameters.

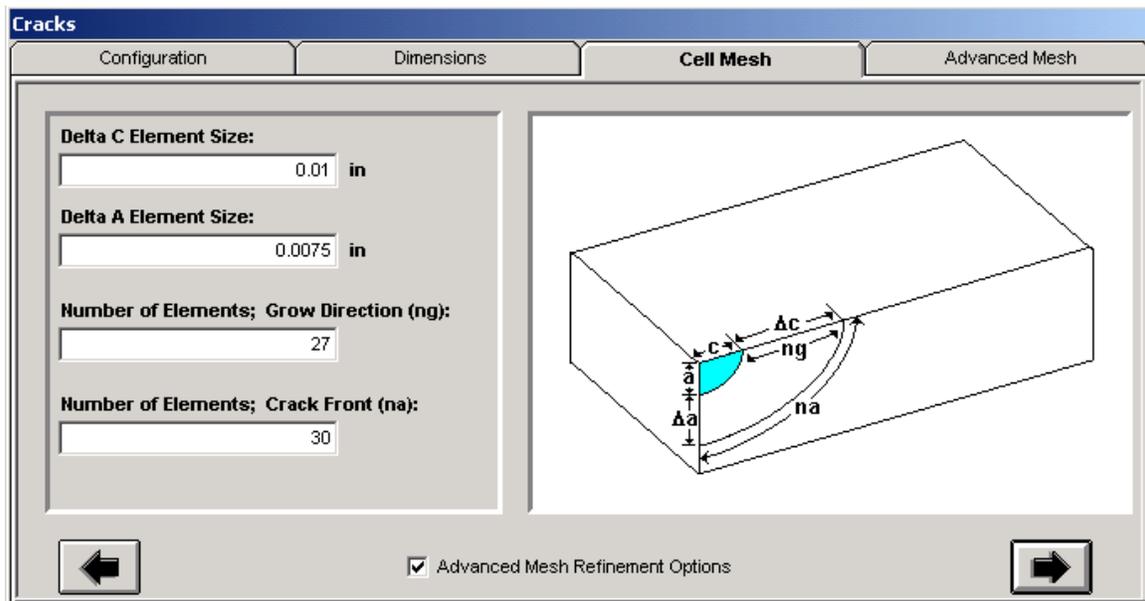


Figure 141 Cell meshcrack front basic parameters.

The sum of the initial crack length, C , and the size of the cell crack zone, $\Delta C * ng$, must be less than the mesh width. Likewise, the sum of the initial crack depth, A , and the size of the cell crack zone, $\Delta A * ng$, must be less than the mesh thickness.

If the Delta C and Delta A element sizes are different, the crack front C/A aspect ratio will change for each subsequent crack front in the cell crack mesh zone. The flat plate cell crack mesh shown below has a Delta C twice as large as the Delta A (Delta C is 0.06 in and Delta A is 0.03 in).

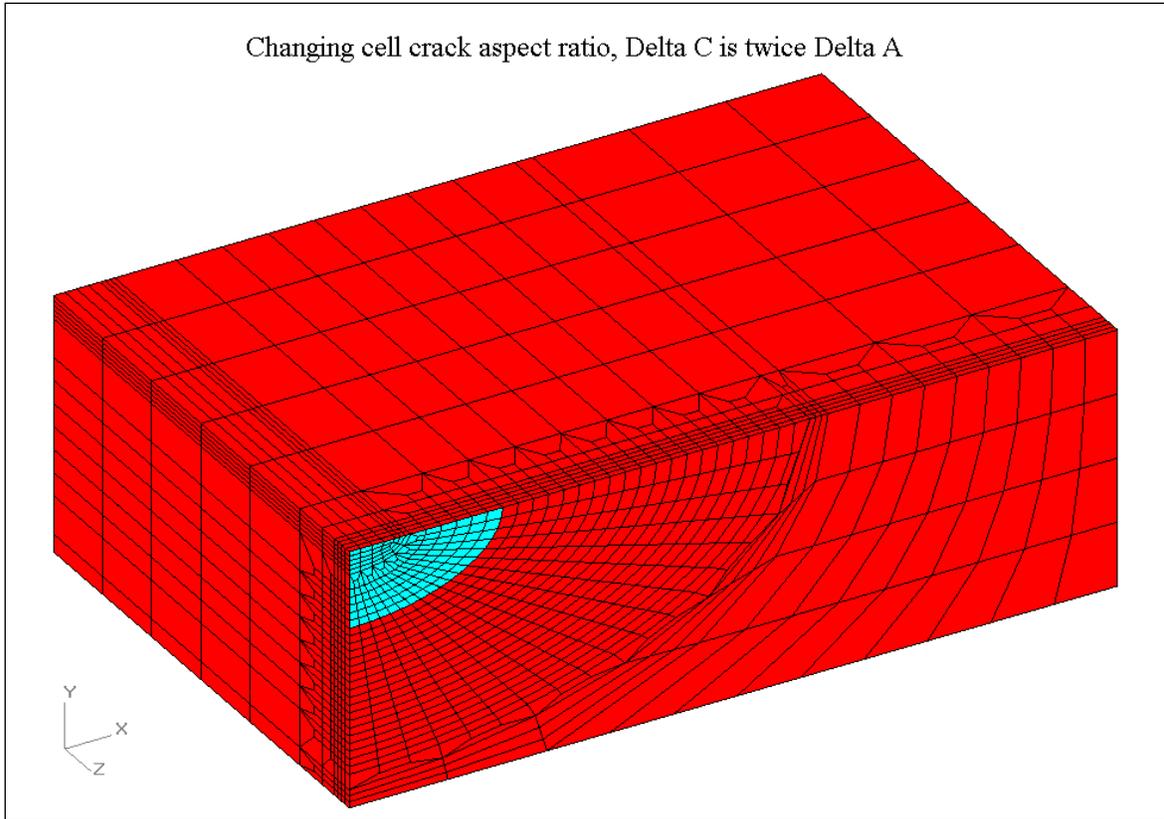


Figure 142 Effect on the cell crack mesh of setting Δc to twice Δa .

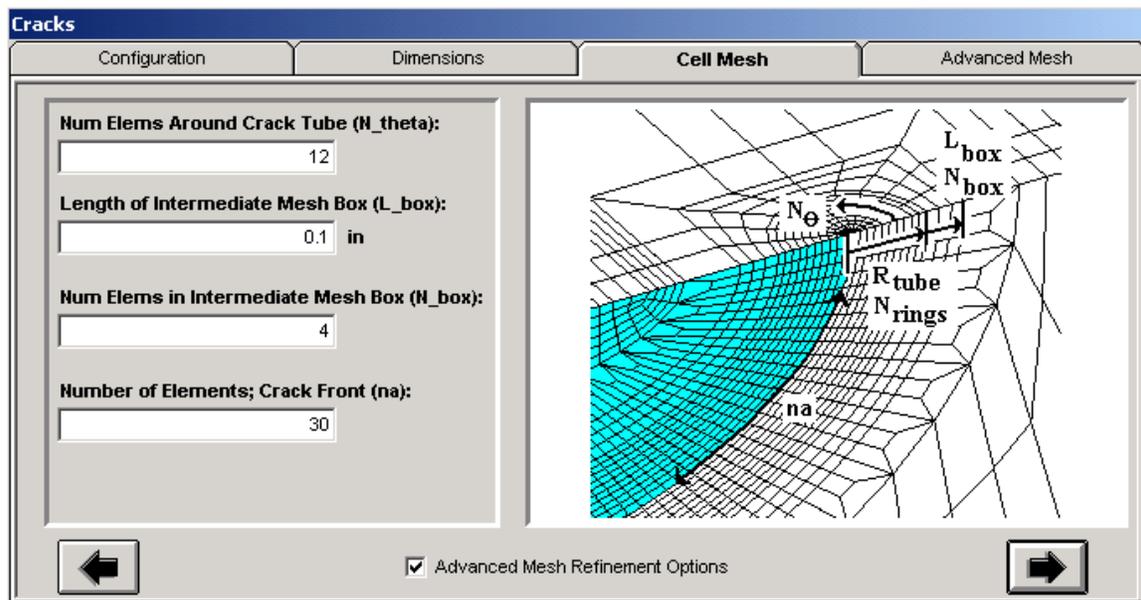


Figure 143 Focused mesh crack front basic parameters.

For a focused crack front in the PRO mesh, the number of elements around the crack tube, N_{θ} , is typically 8 to 12 (enter an even number). The length of the intermediate mesh box extends from the outside of the crack tube to the edge of the intermediate mesh box. The number

of elements in the intermediate mesh box, N_{box} , is in addition to the number of concentric rings of elements in the crack tube. The number of concentric rings of elements in the crack tube is set in the Model Options. The number of elements along the crack front, n_a , is the same as for the cell crack mesh.

Advanced Refinement Form

The advanced cell crack mesh parameters are given on four forms; use the tabs on the right side of the form or the right-arrow button at the bottom of the form to navigate. On the Refinement tab, shown below, is the element size Delta Z in the mesh depth direction, behind and normal to the crack face. The default Delta Z is the average of the given Delta C and Delta A element sizes. The number of elements in the crack zone depth direction can be greater or equal to 1. The default number of elements in the crack zone depth is 1 or the number of contours for the J-integral.

The crack front element size bias controls the relative element size along the crack front. The default bias of 1.0 gives uniform size elements along the crack front. A bias value larger than 1.0 gives smaller elements at the crack tip and larger elements at the crack depth. For example, a bias of 2.0 gives elements that are twice as long at the crack depth than at the crack tip along the crack front direction.

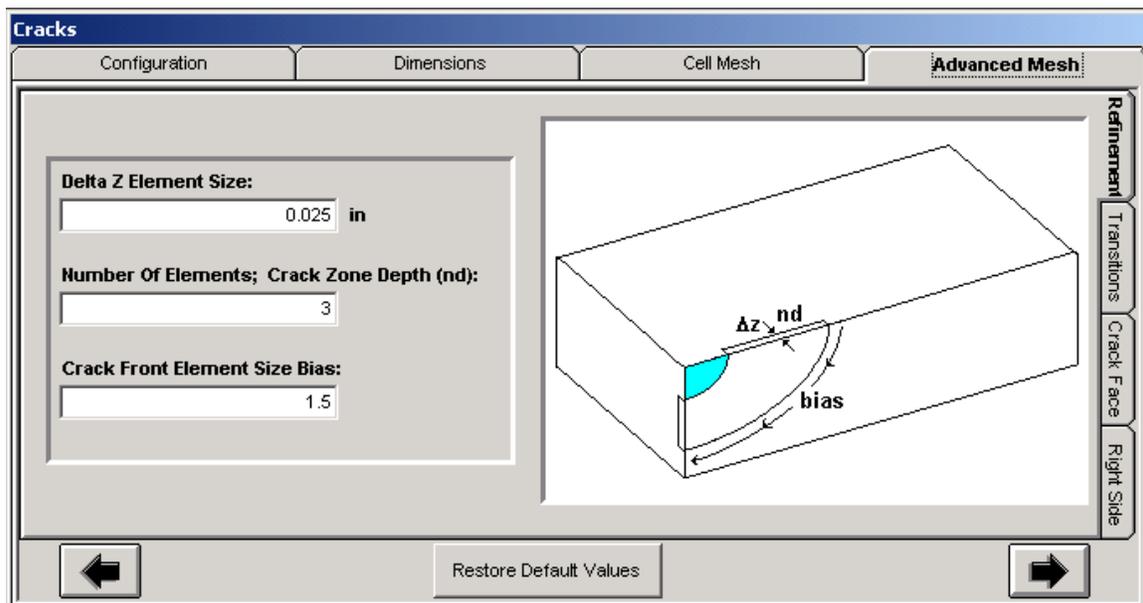


Figure 144 Advanced PRO/cell mesh parameters

Advanced Transitions Form

The number of mesh transitions for the crack front and crack growth directions are input on the Transitions tab, shown below. If a transition zone is included, its size can be adjusted by setting its multiplier value for the element size in that direction. For example, the size of the crack front transition mesh zone is given with a multiplier of the Delta C element size; a larger multiplier

gives a larger mesh zone size. A mesh transition zone can be omitted by setting the number of mesh transitions to zero.

Currently, the mesh transitions use a 3-to-1 element pattern. To use a mesh transition, set the number of elements in that mesh direction to a number that is divisible by 3. Or, for two sets of transitions, set the number of elements in that mesh direction to a number that is divisible by 9. In general, the number of elements should be set to a value that is divisible by 3 raised to a power that is the number of transitions. Future development will add other mesh transition patterns to allow any number of elements along a mesh direction.

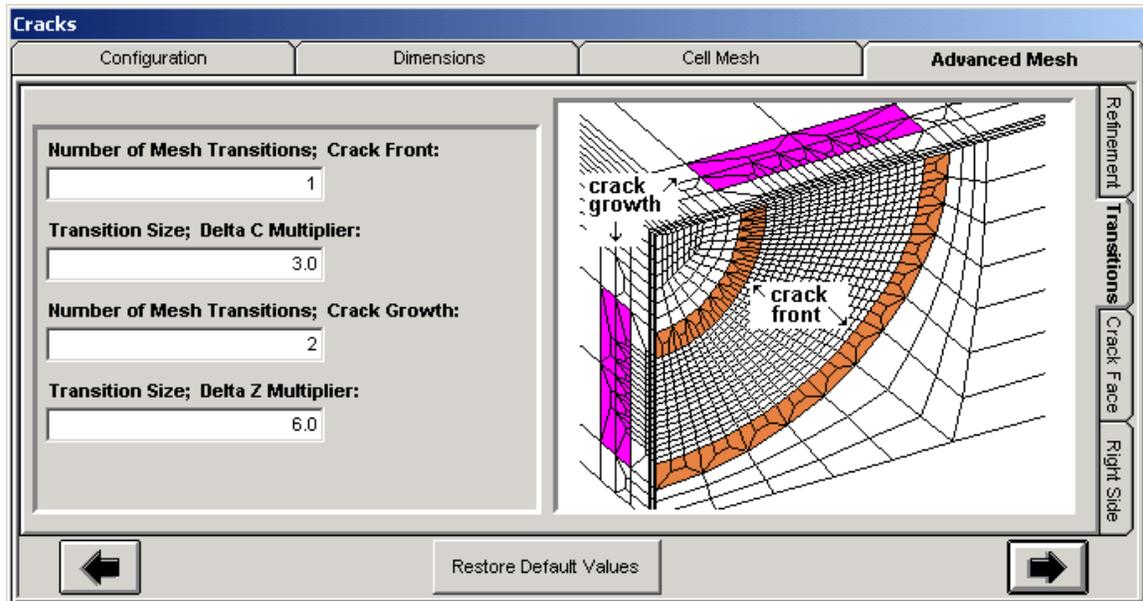


Figure 145 More advanced mesh parameters

The crack front transition mesh zone is shown by the elements colored green in the figure below. The number of elements along the crack front is reduced by one third for each transition set. If the crack front transition mesh zone is omitted, the number of crack front elements will continue into the neighboring mesh zones. The crack front transition can be used independently of other mesh zones or other mesh transitions.

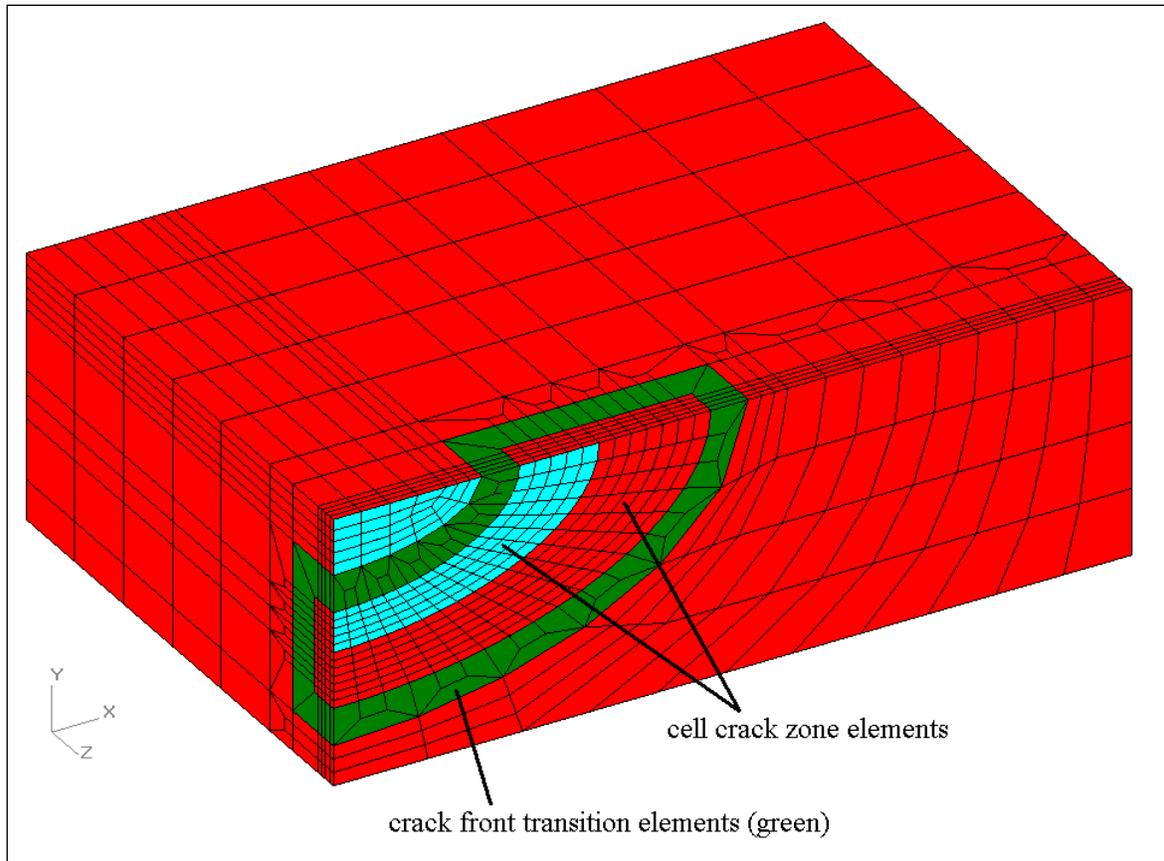


Figure 146 Crack front transition zone

The crack growth direction transition mesh zone is shown by the elements colored green in the figure below. The number of elements along the crack growth direction is reduced by one third for each transition set. If the crack growth transition mesh zone is omitted, the number of crack growth elements will continue through the remaining mesh depth.

Currently, the crack growth transition mesh zone must be active for other mesh zone transitions in the depth direction to be available.

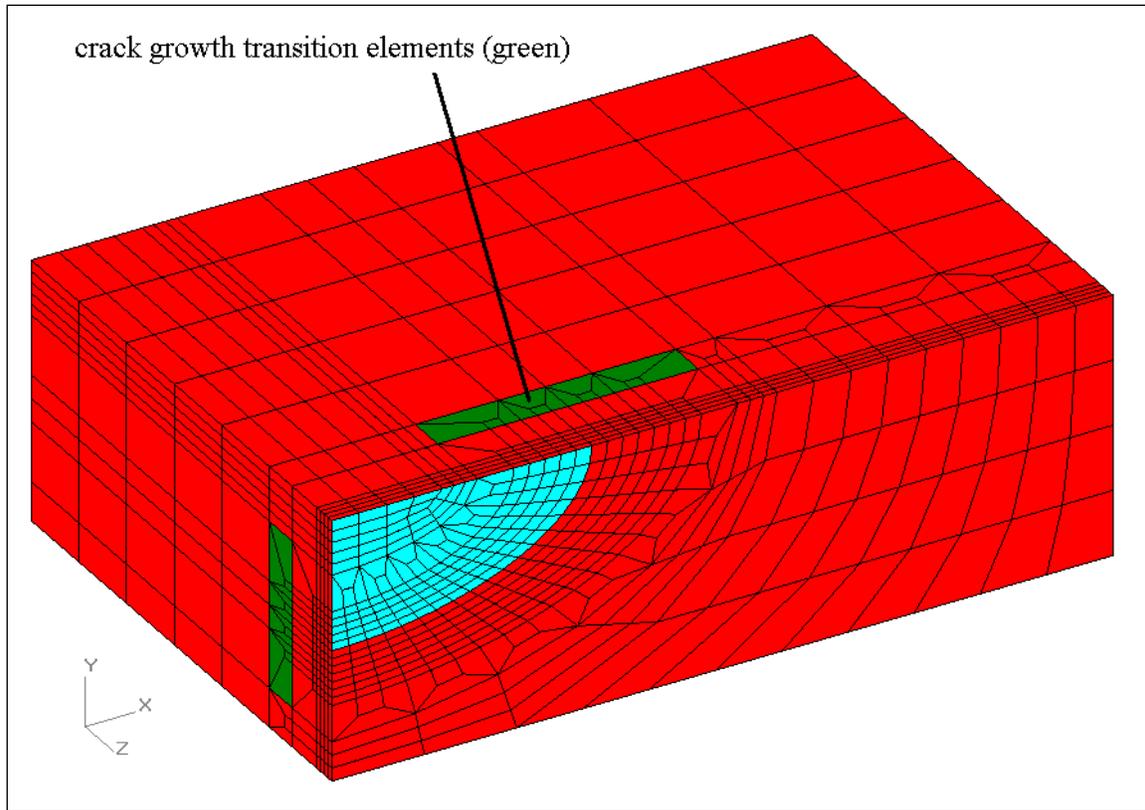


Figure 147 Crack growth transition zone

Advanced Crack Face Form

The remaining mesh depth elements and crack face mesh parameters are input on the Crack Face tab, shown below. The number of elements in the remaining mesh depth are located behind the cell crack mesh zone and any transition zones. The existing mesh pattern from the cell crack and any transition is extruded to fill the remaining mesh depth. There are currently no further mesh transitions in the remaining depth mesh zone.

A typical value for the total number of elements on the crack face in the crack depth direction is about half of the elements along the crack front. If the crack front transition mesh zone is included, the crack depth can typically have about half the remaining number of elements along the crack front. The total number of elements along the crack depth can be increased to larger values to increase the crack face mesh refinement.

The number of elements at the top of the crack face can be up to half the number of elements along the crack front. Or if the crack front transition is included, the top of the crack face can have a number of elements up to half the remaining elements along the crack front direction. The number of elements at the top of the crack face will affect the mesh pattern on the crack face. The minimum is 1 element at the top of the crack face. Since there are no mesh transitions possible in the top of the crack face, the mesh pattern will be continued through the remaining depth of the mesh.

The center of the crack face will then have a number of elements equal to the given total number of crack depth elements minus the number of top crack face elements minus the number of crack front transition sets. The number of elements in the center of the crack face is needed if the mesh transition behind the crack face is included. Currently, the number of crack face center elements must be divisible by 3 for the mesh transition to be included. The size of the crack face center transition will match the size of the crack growth direction transition zone.

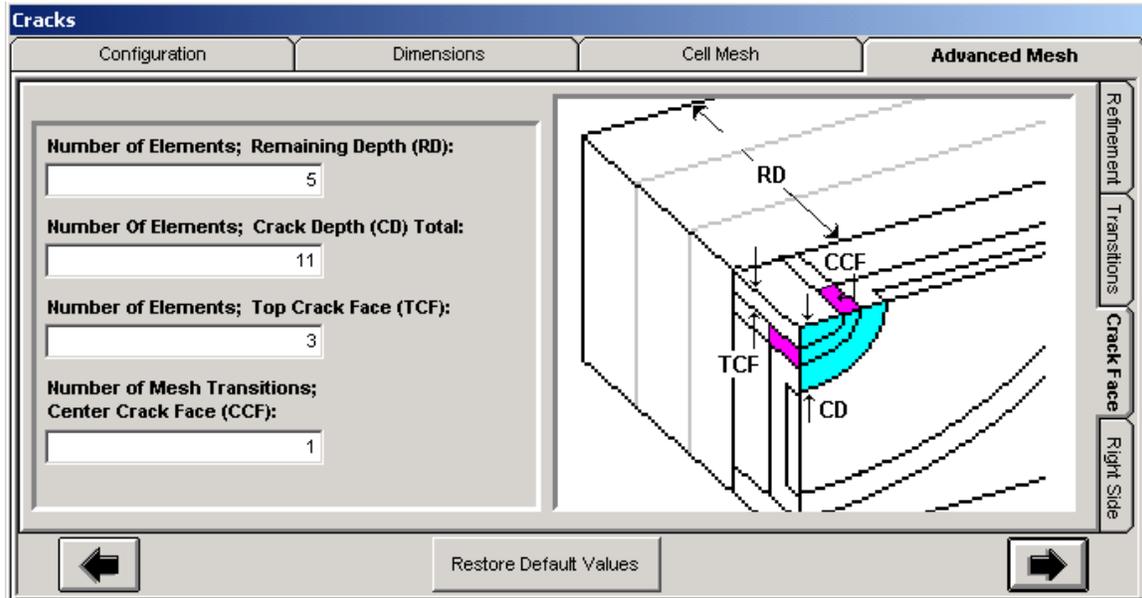


Figure 148 Advanced mesh parameters

If the crack face center transition is included, the transition elements will be several elements behind the crack face, colored green in the figure below. To omit the crack face center transition set the number of transitions to zero; the crack face mesh pattern will be continued through the remaining depth of the mesh.

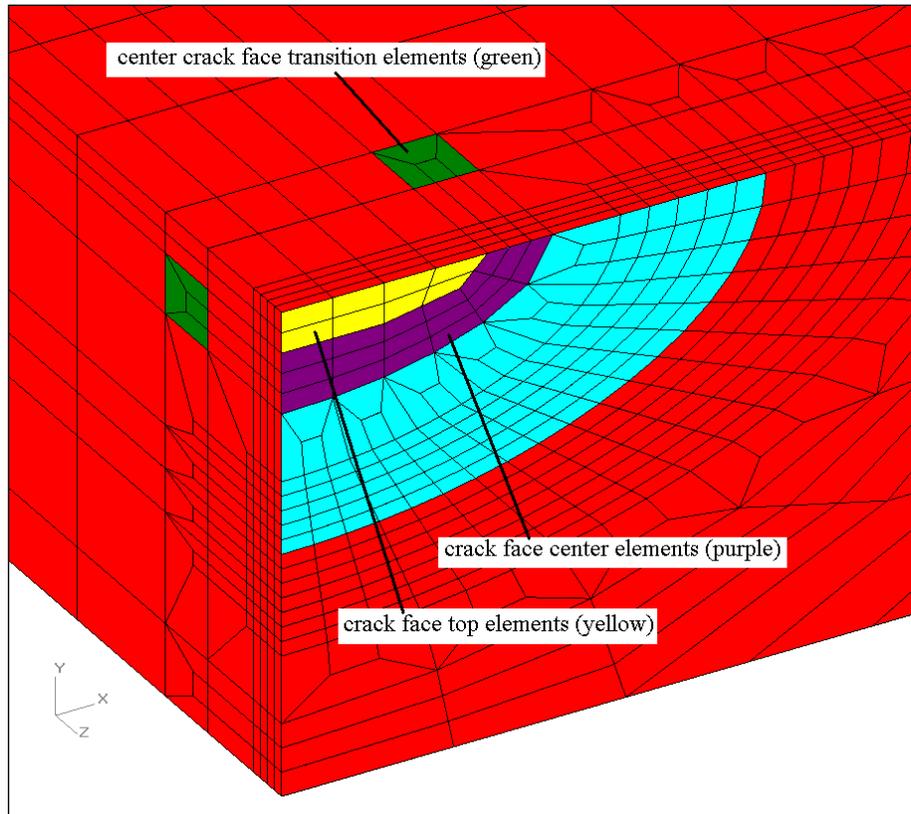


Figure 149 Crack face mesh zones and crack face transition zone

Advanced Right Side Form

The mesh zone beside the cell crack zone wraps around the cell crack zone (below and to the right for a surface crack; to the right for a through crack) to complete the remaining mesh thickness and begins to adjust the mesh pattern for the right edge of the mesh.

The number of elements beside the crack zone will be between the cell crack zone or the crack front transition zone and the remaining thickness of the mesh for a surface crack. The number of elements beside the crack is constant as the mesh zone wraps around to the top of the mesh. The optional mesh transition will reduce the number of elements beside the crack zone in the remaining depth of the mesh. Use a number of elements divisible by 3 when including the mesh transition. The minimum is 1 element to complete the remaining mesh thickness.

The mesh pattern can be selected as a vertical line or a curved line (circle or ellipse arc) on the right side of the mesh zone beside the crack. Depending on the number of elements through the thickness on the right edge of the mesh; the mesh pattern may automatically be set as a vertical line. The mesh pattern can be curved when the number of elements through the thickness is increased.

The remaining mesh width is completed from the mesh pattern line to the left and to the right edge of the mesh. The number of elements in the remaining mesh width should be divisible by 3 if the transition will be included. The mesh zone on the right can be omitted from the cell crack

mesh by setting the number of elements in the remaining width to zero. Likewise, if the cell crack zone is near the right edge of the mesh, the remaining width mesh zone will be omitted from the crack mesh. When the remaining width mesh zone is omitted, the mesh zone beside the crack will complete the mesh width.

The number of elements on the right edge of the mesh is typically half or less of the number of elements along the crack front. Or if the crack front transition is included, the number of elements through the thickness is about half the remaining elements along the crack front. The number of elements through the thickness must be given even if the remaining width mesh zone is omitted. The minimum is 1 element through the thickness.

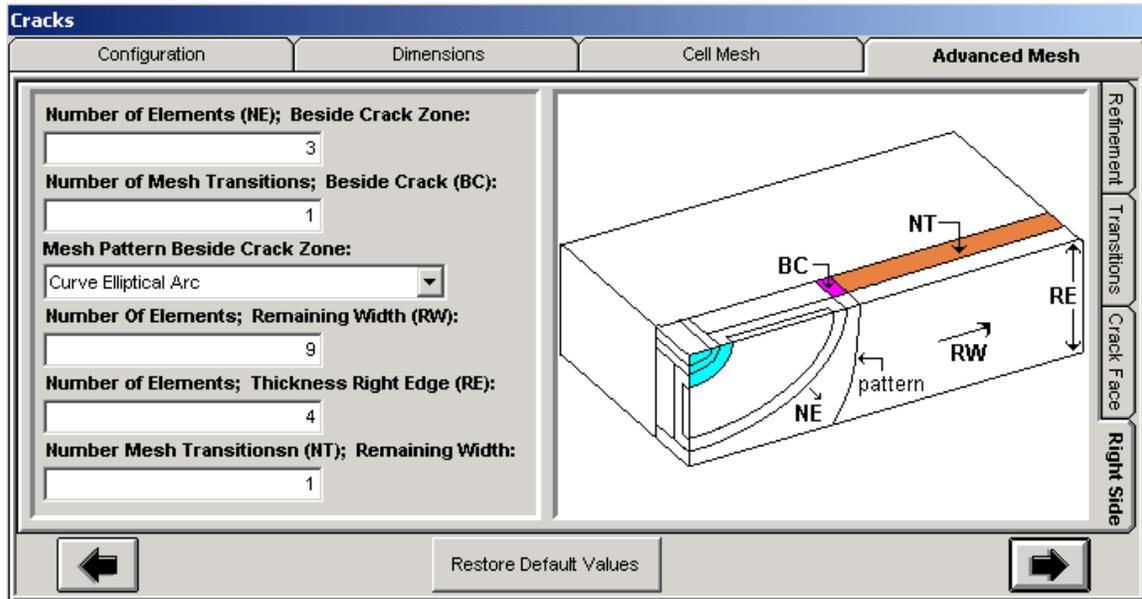


Figure 150 Advanced crack mesh parameters on the right side of the crack front

The optional transition beside and behind the crack zone is shown by the elements colored green in the figure below. This transition reduces the number of elements in the remaining mesh thickness and beside the crack zone in the remaining mesh depth direction. The size of the transition will match the crack growth transition zone.

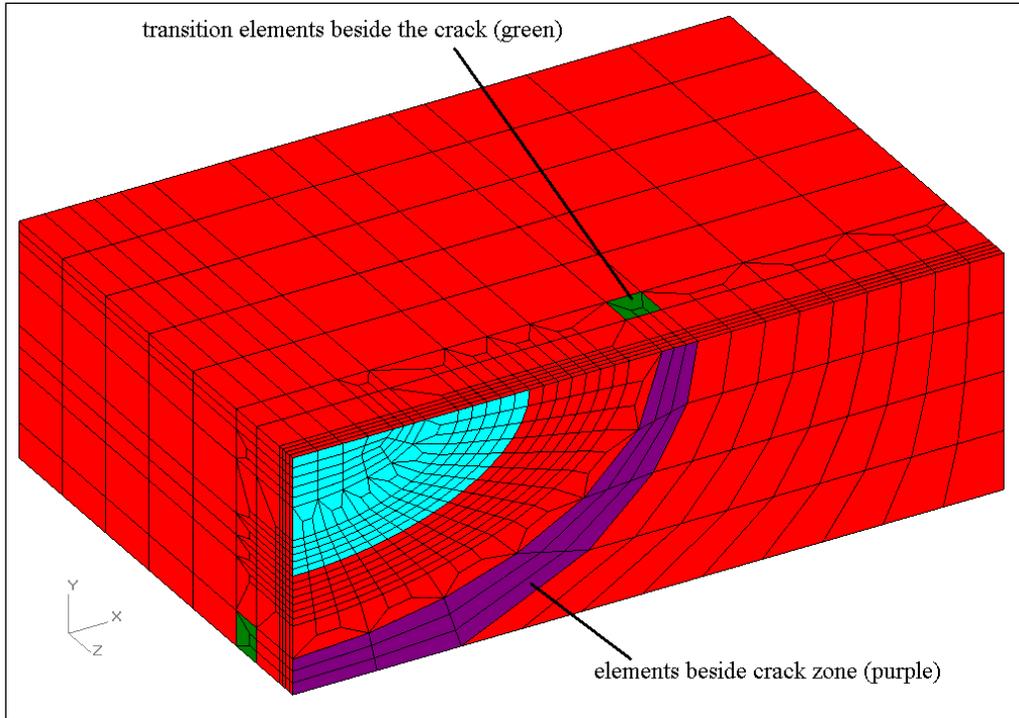


Figure 151 Right size mesh zone and transition zone

The optional transition behind the remaining width mesh zone is shown by the elements colored green in the figure below. This transition reduces the number of elements in the remaining mesh width through the remaining mesh depth direction. The size of this transition will match the crack growth transition zone. This transition can be used independently of the transition behind and beside the crack zone.

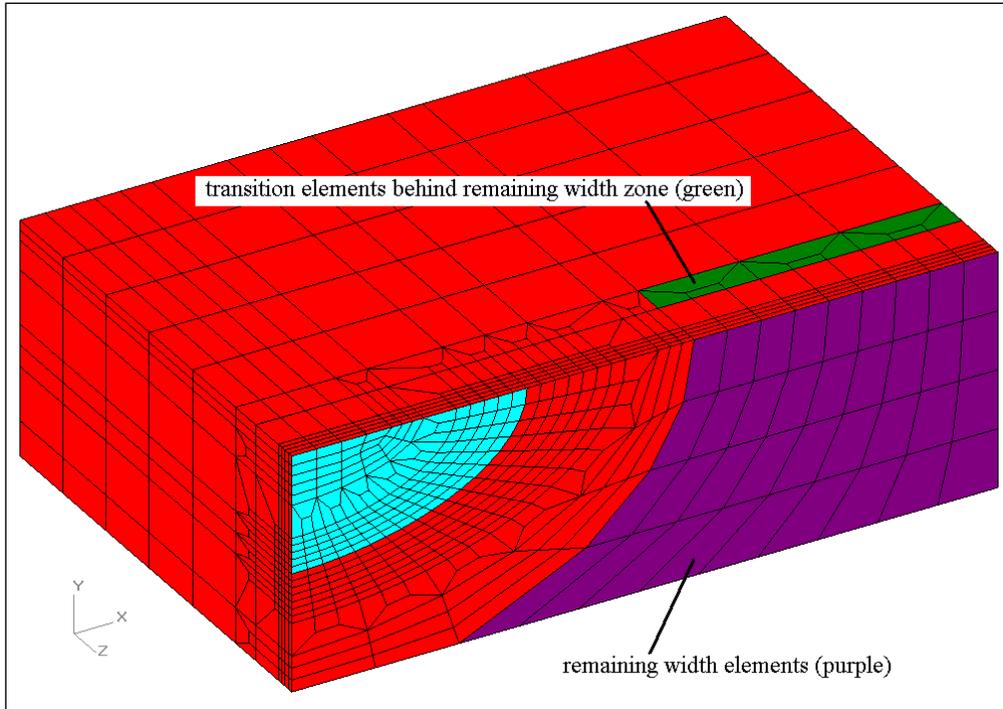


Figure 152 Remaining width mesh zone and transition zone

Additional PRO Mesh Parameters

Vertical Mesh Lines Option

The PRO mesh lines in the ligament region below a surface crack match the crack front element sizes by default. For longer surface cracks, the mesh lines in the ligament region can be set to be vertical along part of the crack by using key word text in the Home/Notes form. This option is useful for long surface cracks when the elements in the ligament below the surface crack can become skewed. The key word text for this option is:

*use PRO num uniform elem = <number>

Where <number> is an integer number of elements, less than the total number of elements along the bottom of the crack. The figure below shows an example of the key word text in the Home/Notes form. This key word text can be used with other text by separating with a semi-colon “;”.

The image shows the FEACrack software interface. At the top is the logo, which consists of a blue and red globe icon followed by the text 'FEACrack'. Below the logo are several input fields and buttons. On the left, there is a 'Units:' dropdown menu currently set to 'lb - in - psi'. To the right of this is a 'Continue' button with a right-pointing arrow. Below these is an 'Analysis Type:' dropdown menu set to 'Focused Crack Mesh For J-Integral Analysis (Standard)'. Further down are two text input fields: 'Analysis Name:' containing 'PRO mesh plate example' and 'Model Label:' containing 'plate'. At the bottom is a 'Notes:' text area. The text '*use PRO num uniform elem = 3' is entered in this field and is circled in red. A red line points from this circle to the caption below.

Use key word text for the PRO mesh option

Figure 153 Key word text in the Home/Notes form

The uniform size elements are near the end of the crack, below the crack front, shown in the figure below. For comparison, the following figure shows the standard PRO mesh without this option.

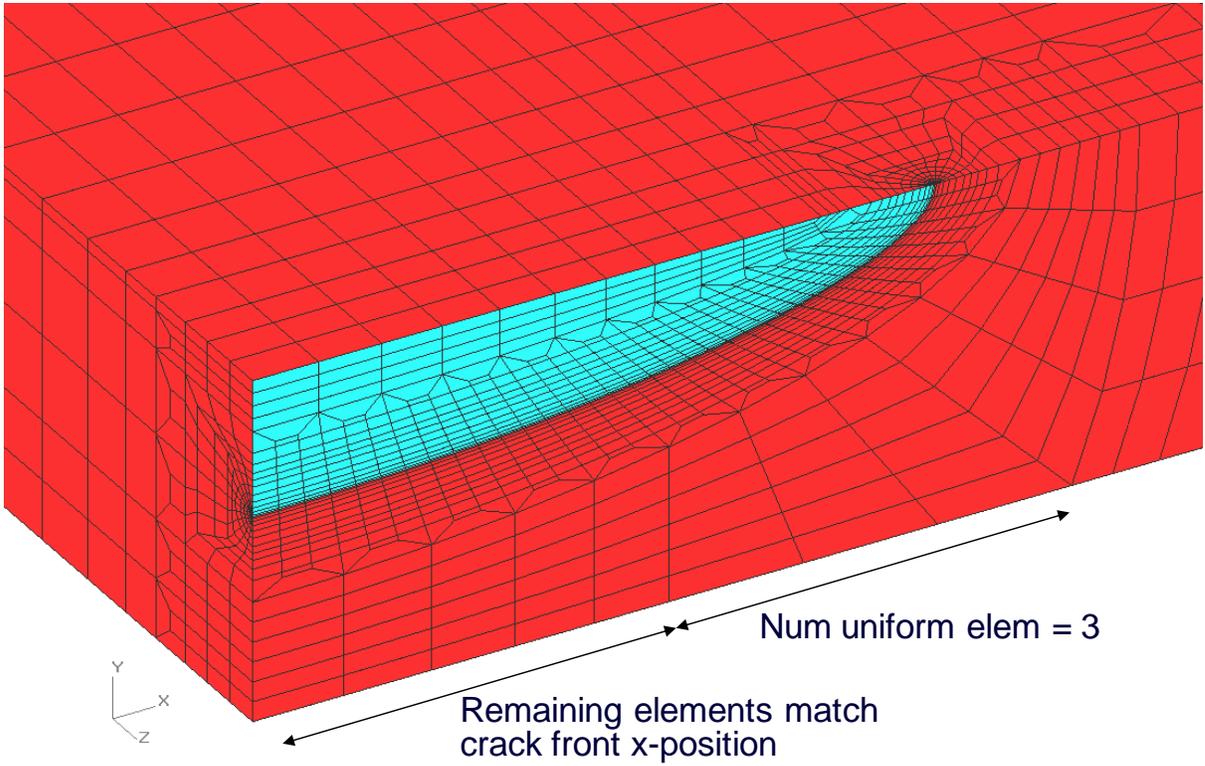


Figure 154 Example of using the PRO mesh option to set the number of uniform size elements and get vertical mesh lines in the ligament below the surface crack

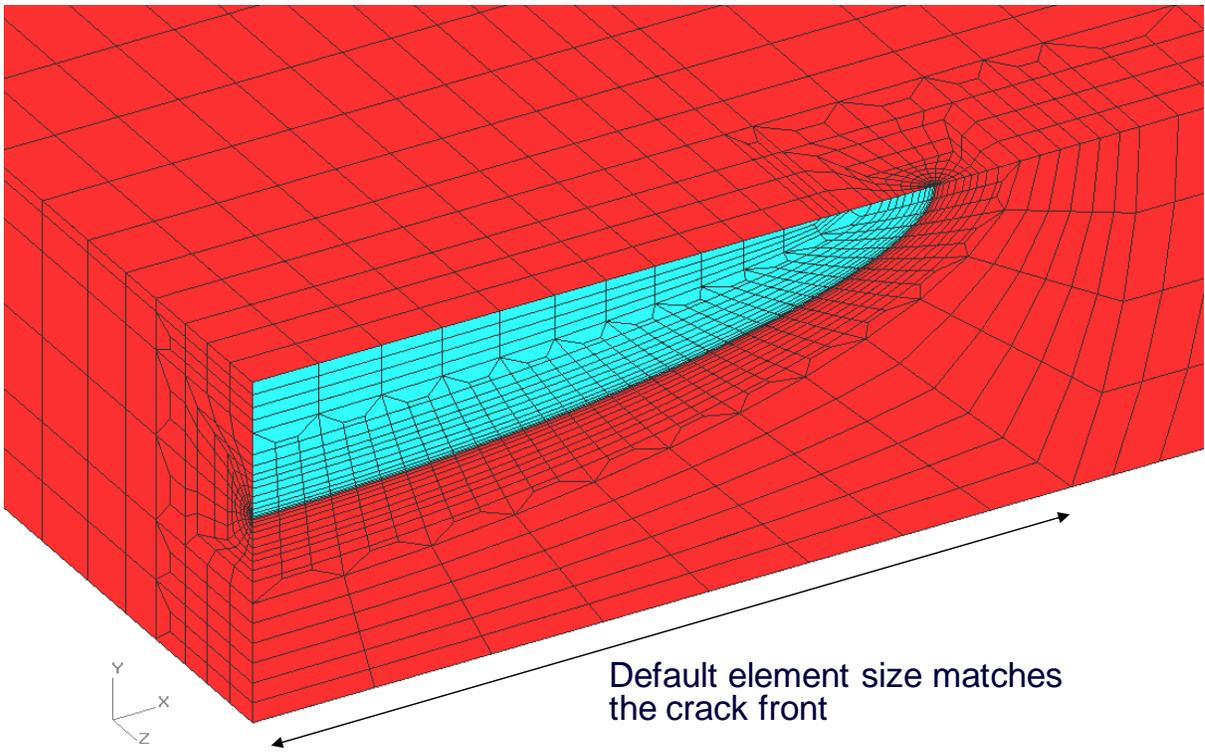


Figure 155 The standard PRO mesh in the ligament below the surface crack

Inner Crack Tube Length Option

When using the finite radius crack front option, the size of the inner quadrant of the crack tube can be adjusted by setting the inner crack tube length. Use the prototype *use key word in the Home/Notes form:

*use inner tube length = <value>

Where <value> is the length along the edge of the inner part of the crack tube along the crack face, as shown in the figure below, or omit the key word to use the default inner crack tube size.

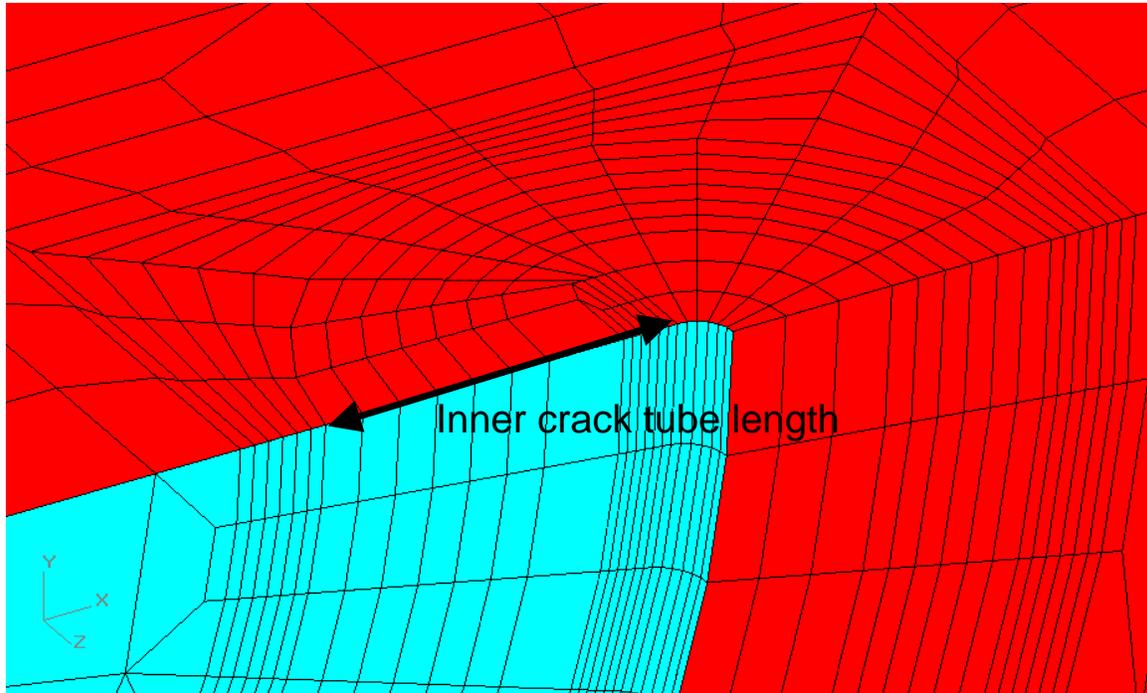


Figure 156 The inner crack tube length

Inner Mesh Box Multiplier Option

When using the finite radius crack front option, the size of the inner mesh box can be adjusted using the multiplier. The figure below shows the inner region of the intermediate mesh box. The prototype *use key word in the Home/Notes form:

*use inner box mult = <value>

Where <value> is the multiplier to adjust the intermediate mesh box length; to reduce the size of the inner mesh box zone use a multiplier value less than 1.0.

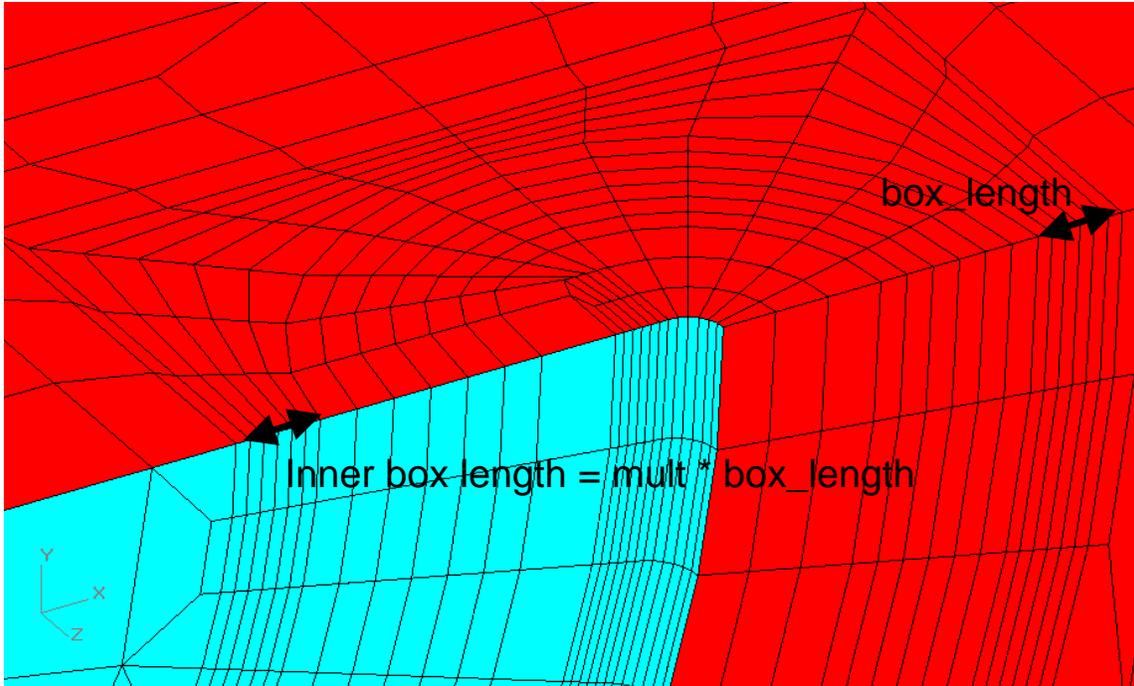
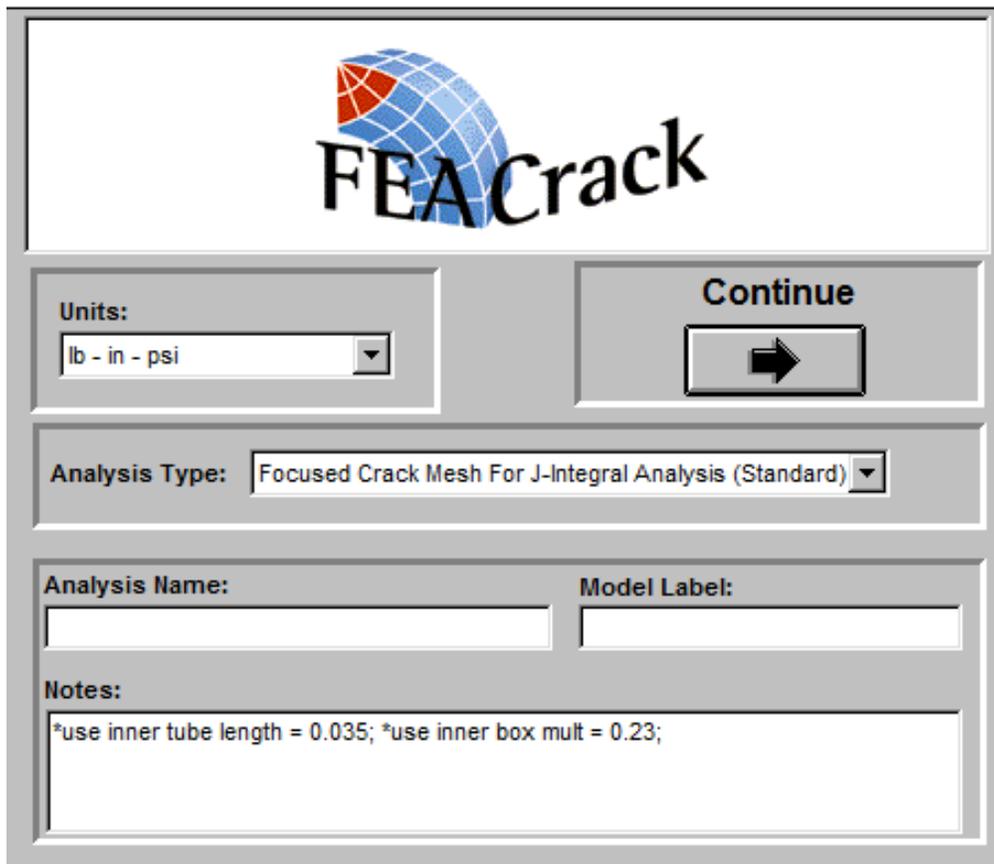


Figure 157 The inner mesh box length and multiplier



The image shows the FEACrack software interface. At the top is the logo, which consists of a blue and red globe icon followed by the text "FEACrack". Below the logo are several input fields and buttons. On the left, there is a "Units:" label above a dropdown menu showing "lb - in - psi". To the right of this is a "Continue" button with a right-pointing arrow. Below these is an "Analysis Type:" label above a dropdown menu showing "Focused Crack Mesh For J-Integral Analysis (Standard)". Further down are two empty text input fields labeled "Analysis Name:" and "Model Label:". At the bottom is a "Notes:" label above a text area containing the text: "*use inner tube length = 0.035; *use inner box mult = 0.23;".

Figure 158 Example of the inner crack tube length and inner box multiplier key words

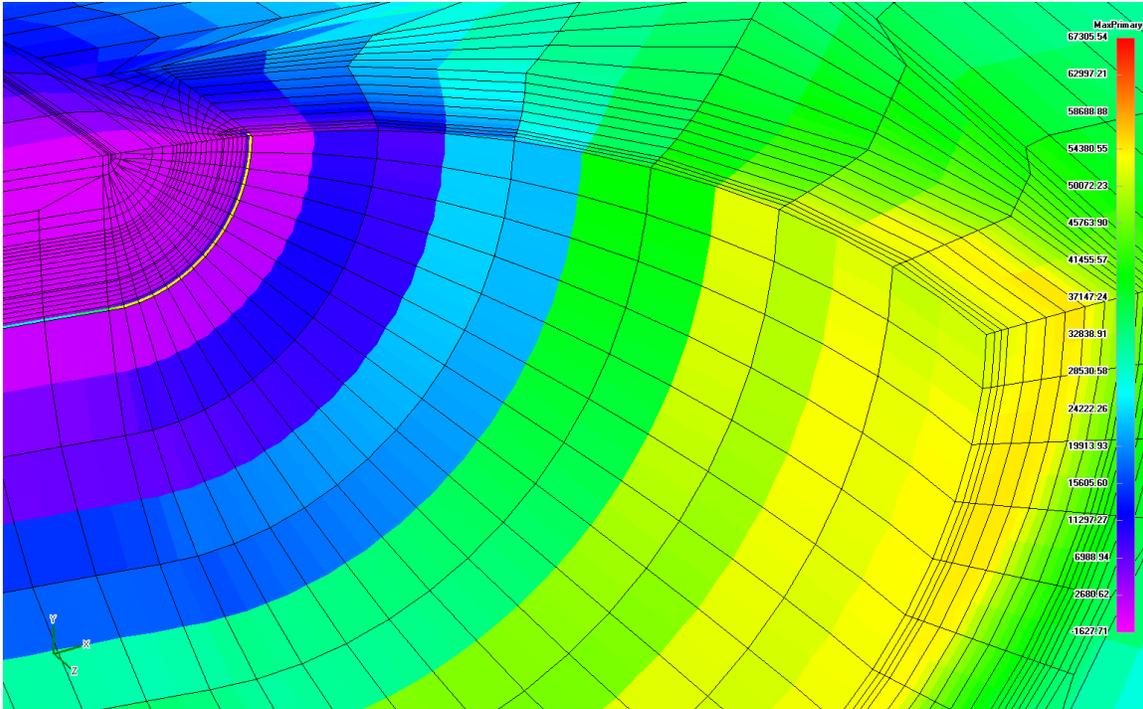


Figure 159 Example of a large radius groove using the inner mesh zone options

Node Release Crack Growth

The cell crack mesh can be used for a crack growth analysis where the crack front nodes are released at user selected analysis steps. As mentioned above, select the node release crack growth from the Analysis Type menu on the Home form. After selecting the geometry and setting the crack mesh parameters, enter the node release data on the Boundary Conditions form as shown below.

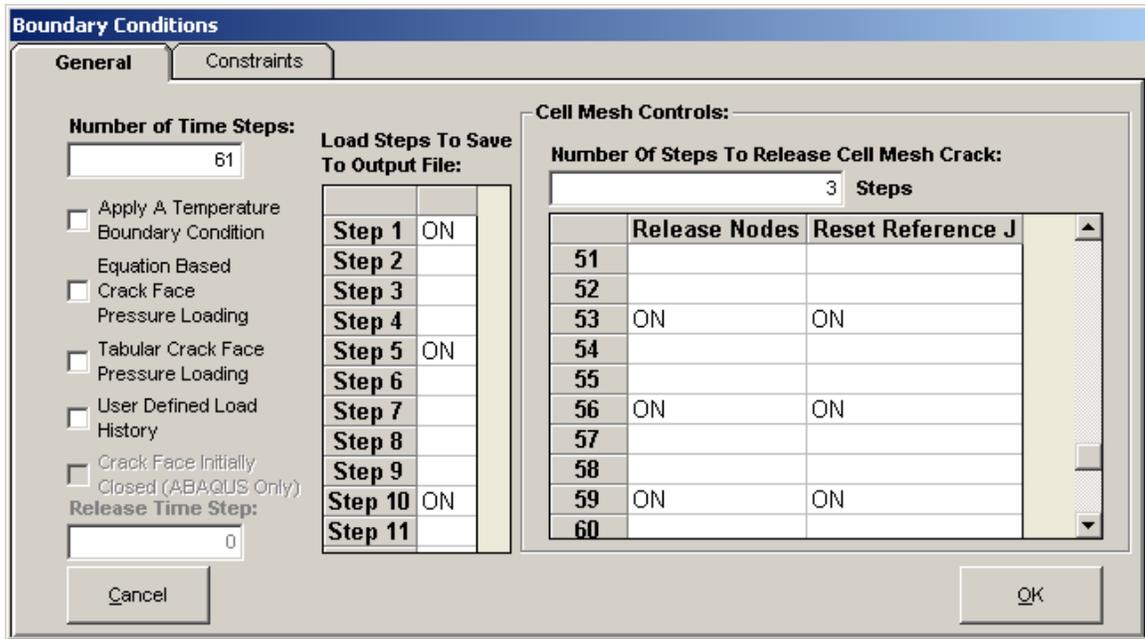


Figure 160 Control the output steps and crack front release

A node release crack growth analysis will typically require many analysis steps to permit multiple crack front node releases, and sufficient steps between node releases to maintain the solution convergence. Enter the number of analysis steps in the upper left corner text box. Then enter the number of analysis steps to release the crack front nodes in the top right text box. The crack front node release can be distributed over several analysis steps to improve convergence.

The desired analysis steps for output are selected in the center column of the form. Selecting fewer steps for output reduces the size of the result file and may be necessary when there are more than 30 steps to avoid memory problems during post processing. Enter “on” or a “1” in the column rows for the steps with output.

The steps when the crack front node release occur are selected the Release Nodes column. Enter “on” or a “1” to select the steps for the node release. The cell mesh crack front will advance by one element every time a node release is selected. Future development will support ΔJ results, and the Reset Reference J column has been included to select when the mesh reference state should be reset during the *WARP3D* crack growth analysis. Currently, a comment line is being written to the *WARP3D* input file for the reset reference J command but is otherwise unused.

More details about the node release crack growth are available in the *WARP3D* manual, which is available as a PDF file in the FEACrack install directory or by download from the www.questintegrity.com web page, or by contacting technical support.

Results are shown below for a node release crack growth analysis. The initial crack has had 14 node releases during the analysis. The crack has grown to the larger size shown at the end of the analysis.

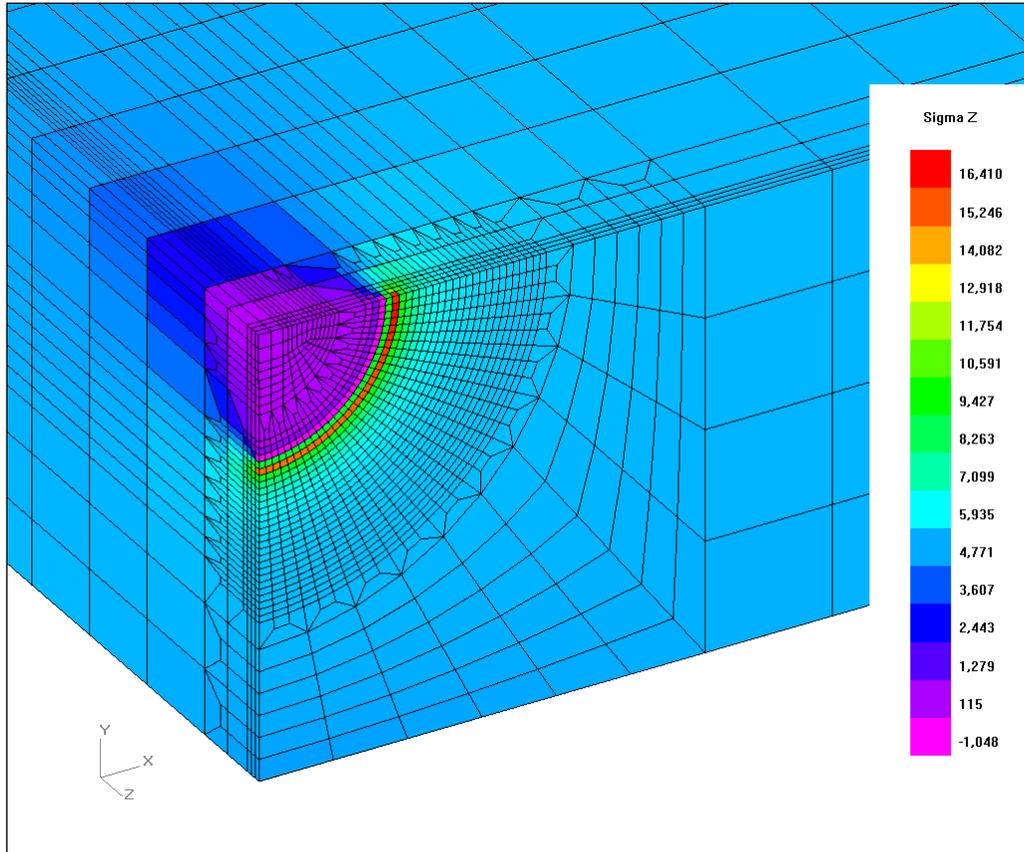


Figure 161 Cell crack mesh, analysis step 1, initial crack size before node release.

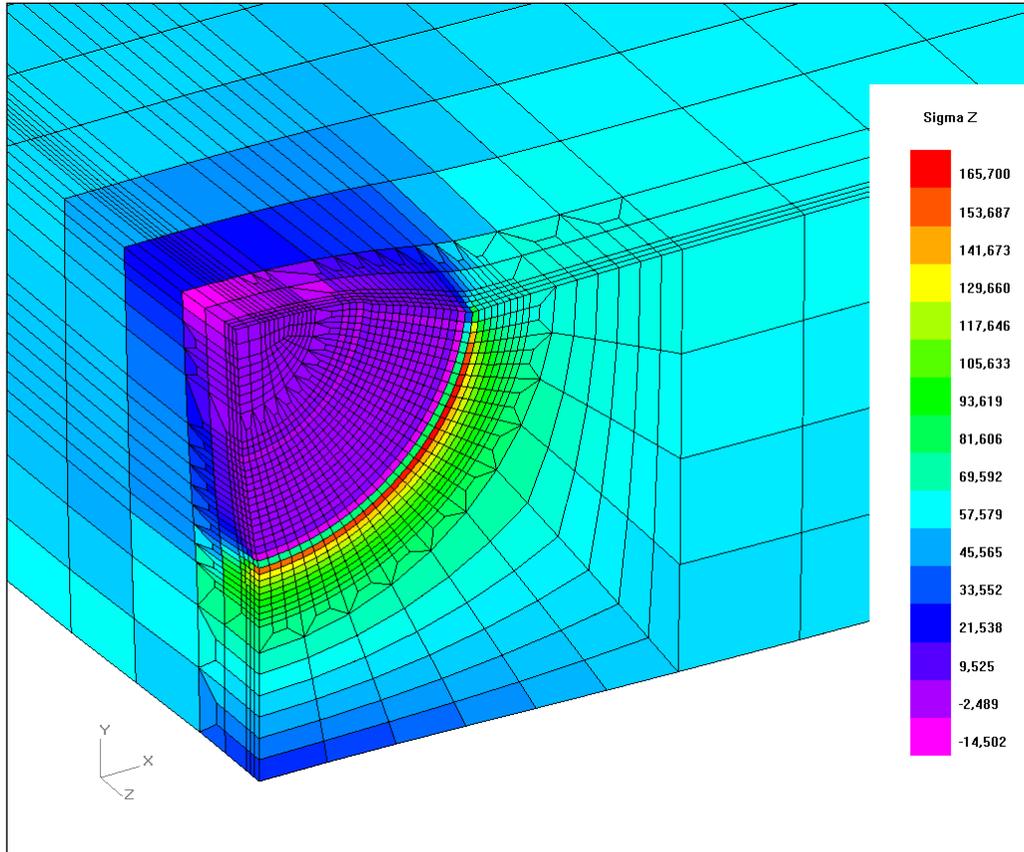


Figure 162 Cell crack mesh, analysis step 61, final crack size after 14 crack front node releases.

Plots of the J-integral versus the crack extension (J-R curve) are available in the crack results forms when the “Create J vs R curve” check box is selected on the FEA Result File Selection form. The “Create A FAD Curve” option can be unchecked to avoid warning messages from the FAD calculations for cell crack mesh results.

The crack extension distance is computed as the distance from the crack node on the initial crack front to the matching crack node on each new crack front after the node release. The crack extension is computed for all the crack front nodes, but only selected crack front locations are shown in the plot to limit the number of curves per mesh. All the crack front extension values for every crack front node are available in the extracted crack result file (check the option box on the FEA Result File Selection form). The J-integral values are from a remote domain that surrounds the cell mesh crack growth zone. This is a different J domain definition than is used in the focused crack mesh.

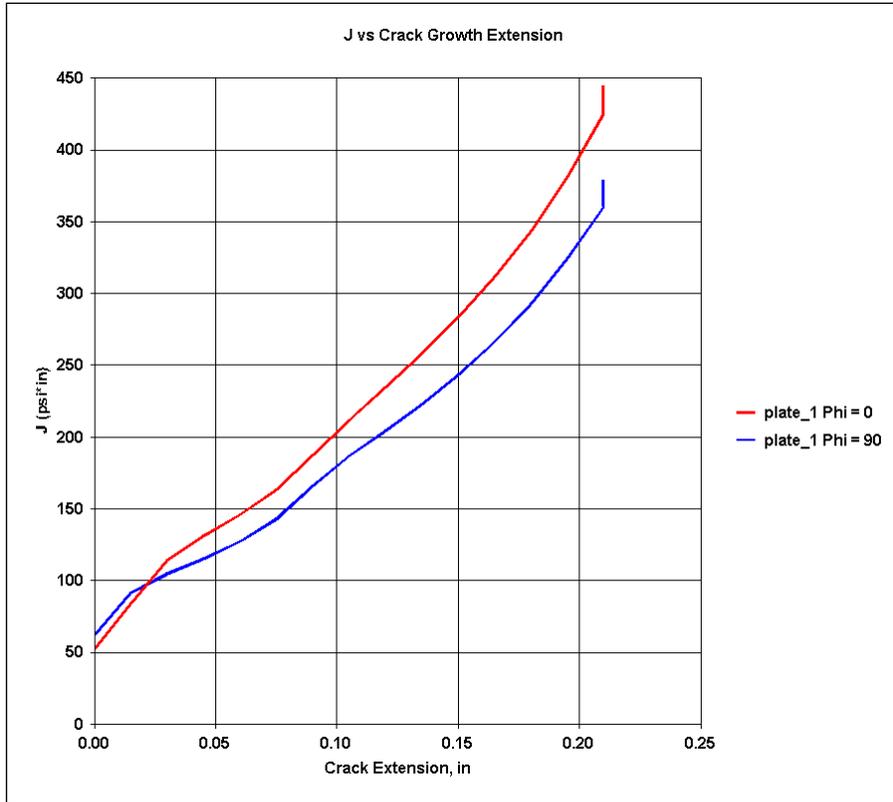


Figure 163 J versus crack extension result plot

Compare J-integral Results

A flat plate unit cube crack mesh is used to compare the J-integral results from a focused crack mesh and a cell crack mesh using J domains at the crack front rather than the remote J domain. The crack front does not advance in this analysis comparison. The unit cube focused and cell crack meshes are shown below. The cube dimensions are 1 inch per side; the crack is semi-circular with the crack length and depth of 0.5 inches. A membrane stress is applied to the backside of each mesh; sigma Z is 10,000 psi. The Young's modulus of elasticity is 30×10^6 psi and the Poisson ratio is 0.3.

The focused crack mesh is generated with 8 and 20 node brick elements. The cell crack mesh is generated with 8 node brick elements. *WARP3D* input files are created and used to run the crack mesh analyses to obtain J-integral results for each crack mesh.

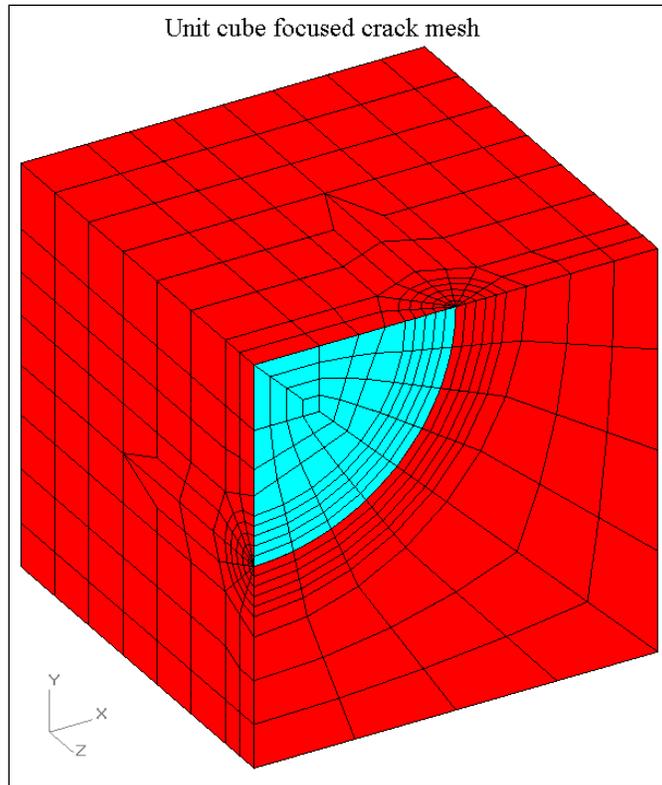


Figure 164 Example of a focused crack mesh

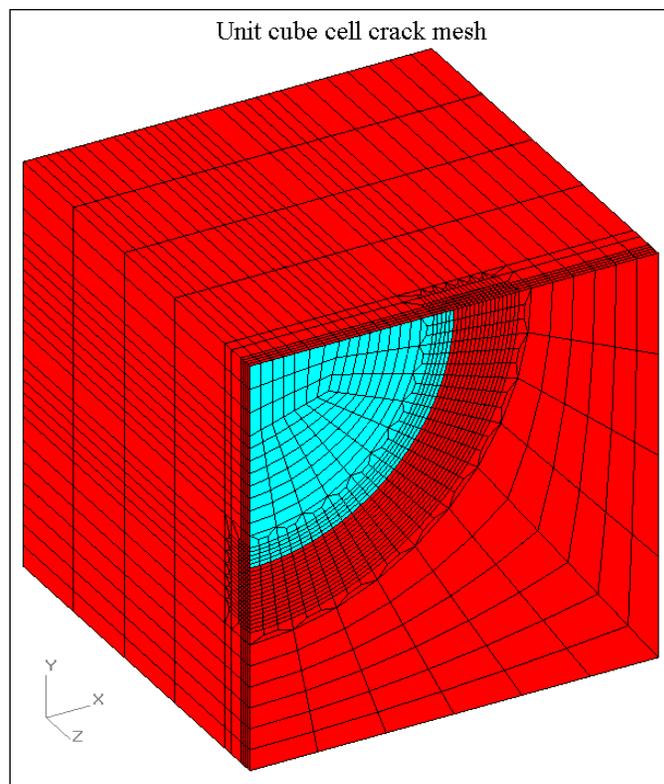


Figure 165 Example of a cell-type crack mesh

Cell Crack Mesh J-integral Results

The cell crack mesh J-integral results are extracted from the *WARP3D* result file using the *FEACrack* post processor. The plot of each contour integral along the crack is shown in the plot below. It appears that the first two contours underestimate and overestimate the J value, but the third, fourth, and fifth contours have converged to a consistent J value. The first two contours do not include very many elements around the crack front and would likely have higher path dependence than later contours that include more elements around the crack front. In general it is better to use one of the focused crack meshes with collapsed elements at the crack front when J-integral values are desired.

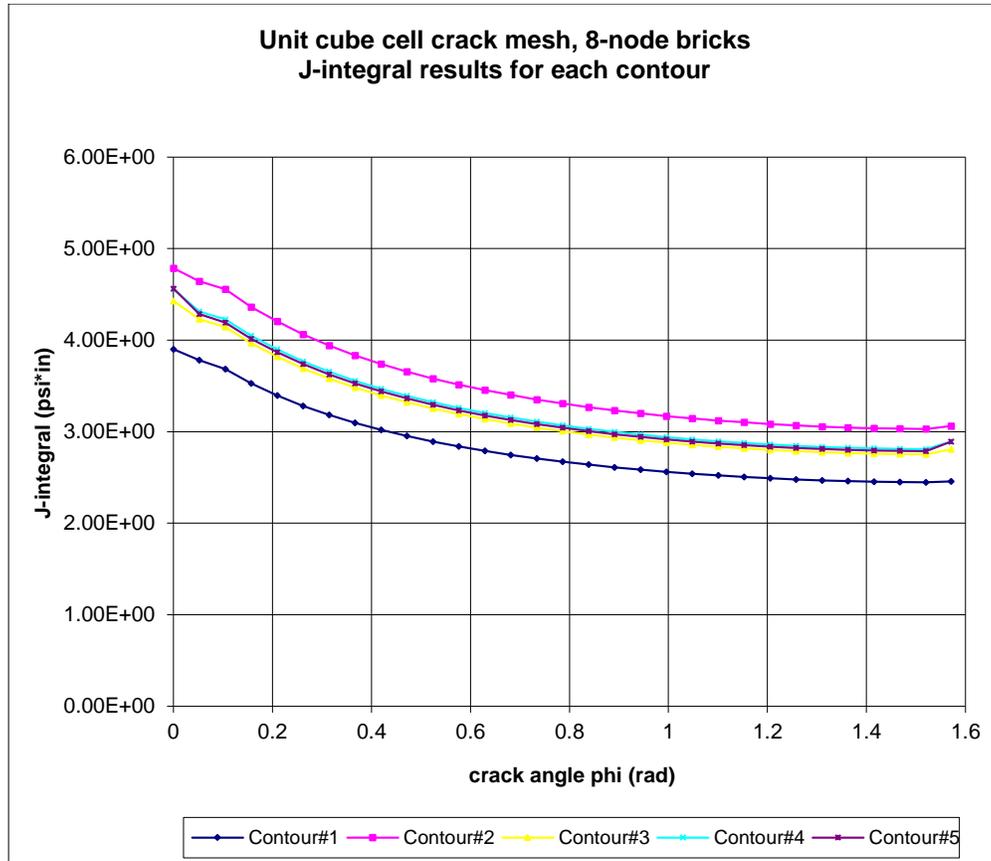


Figure 166

The 30 elements, colored green, in the fifth J-integral contour in the cell crack mesh are shown in the figure below.

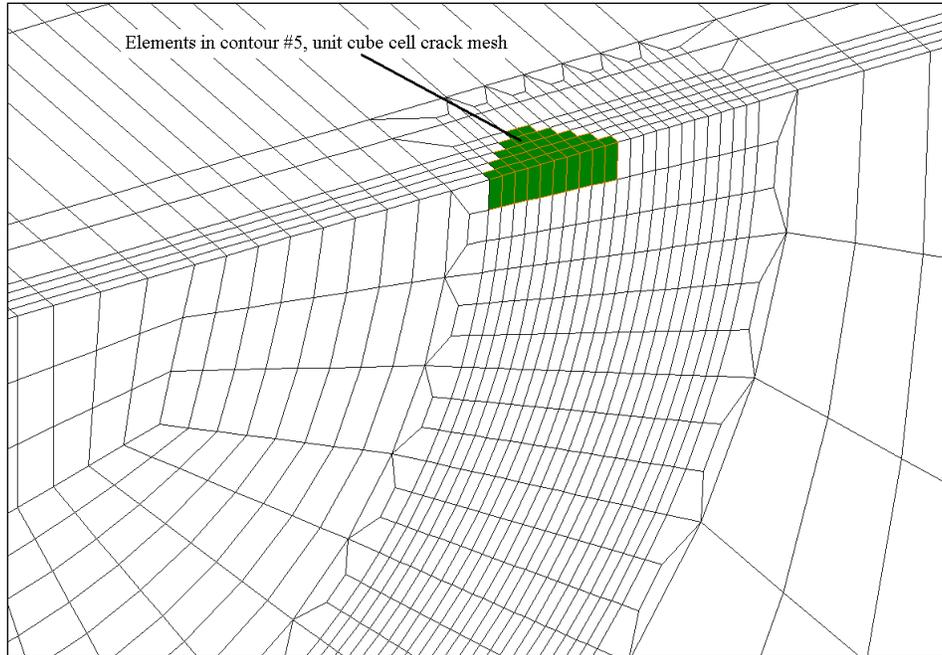


Figure 167 Elements in contour #5 in the cell crack mesh.

Focused Crack Mesh J-integral Results

The plot of each contour integral along the crack for the 8 node brick focused crack mesh is shown in the figure below. It appears that the first contour underestimates the J value, but later contours have converged to a consistent J value. Typically the first contour is not used for a focused crack mesh J-integral result. In *FEACrack* post processing, the second and higher contours are used to get an averaged J-integral value at each node along the crack front.

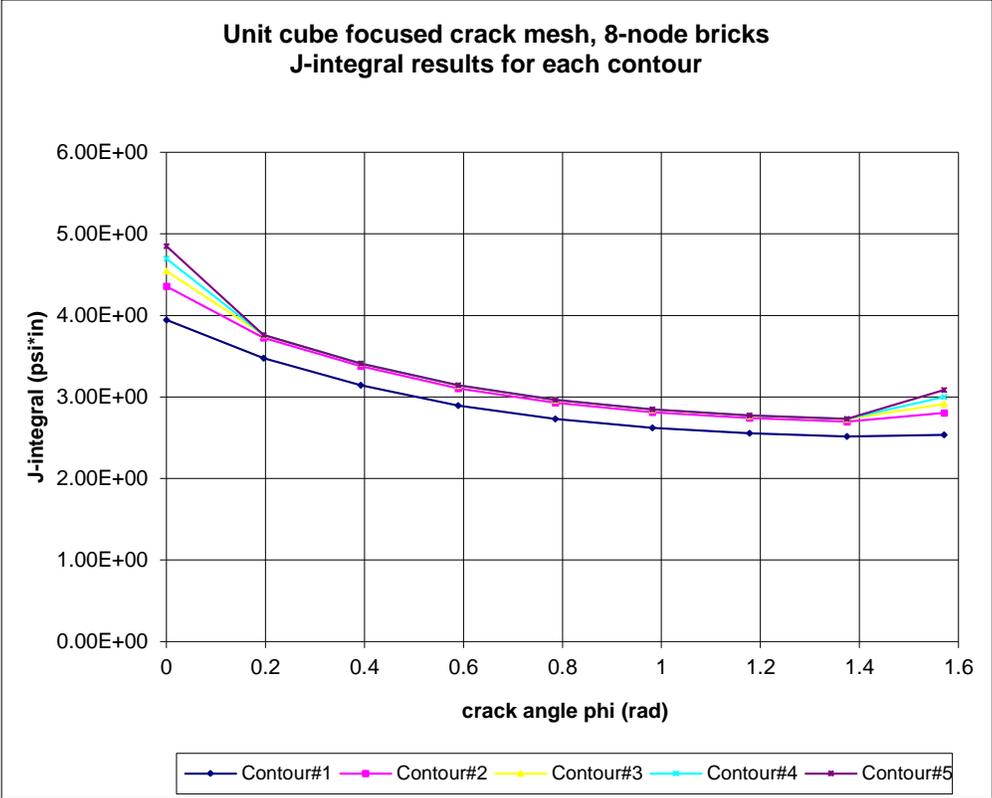


Figure 168 Comparison of J-integral results for the unit cube crack mesh example

The plot of each contour integral along the crack for the 20 node brick focused crack mesh is shown in the figure below. It appears that all the contours have converged to a consistent J-value.

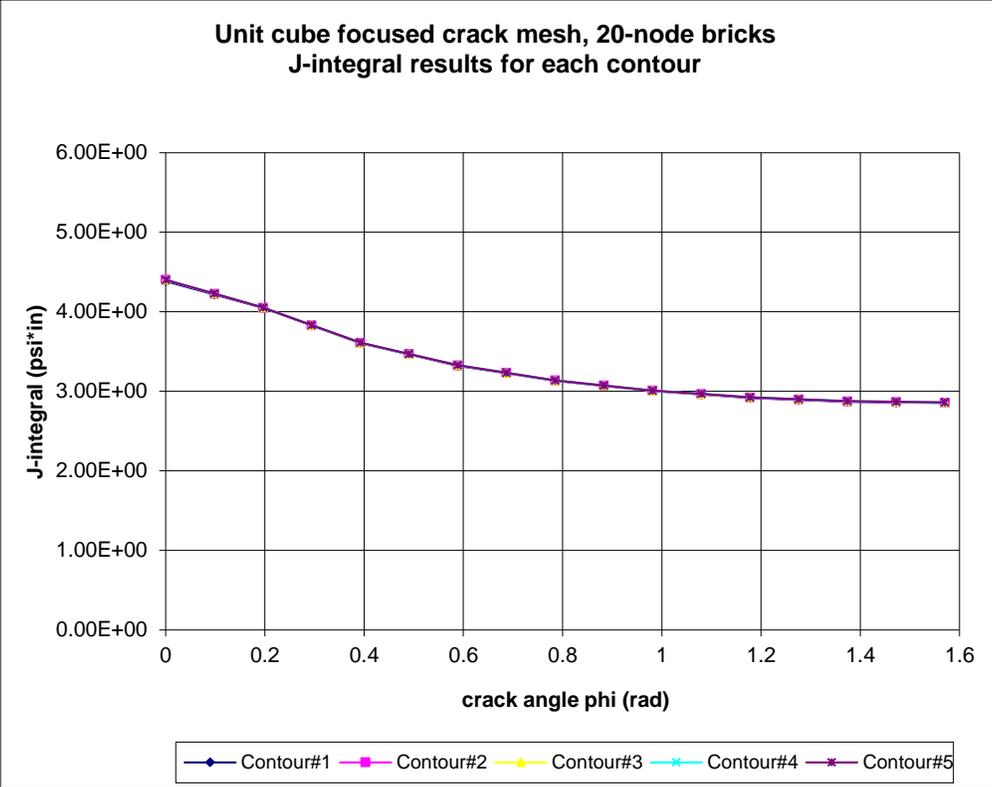


Figure 169 Comparison of individual J-integral contour results

Compare Crack Mesh J-integral Results

To compare the J-integral results from the cell and focused crack meshes, the plot of the fifth contour value from each crack mesh is shown in the figure below. The cell crack mesh J-integral results compares very well to the 8 node and 20 node brick focused crack mesh results.

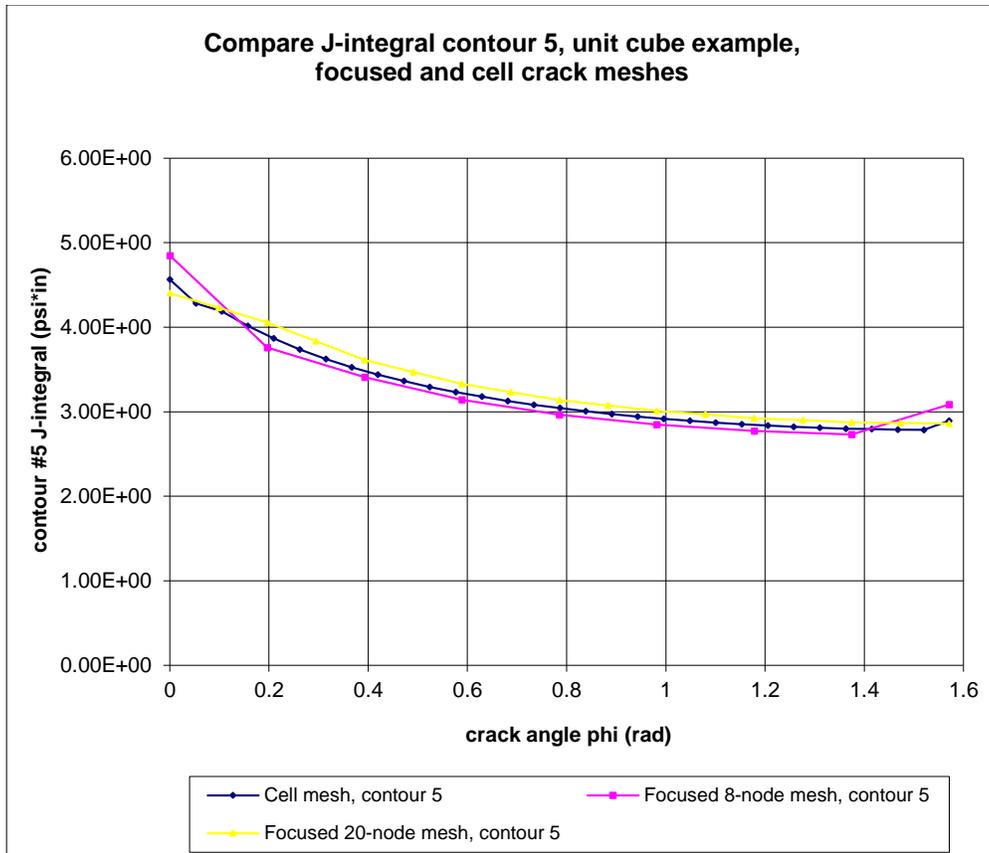


Figure 170 Comparison of contour 5 J-integral results

Complex Crack Shape

The complex crack shape is selected from the Crack Shape menu in the Geometry/Cracks/Dimensions form; see the figure below. The custom crack front node c,a coordinates are defined in the spreadsheet table. For the complex custom crack the first crack front node is defined at the center of the crack front. The first crack front node must be at $c = 0$, and $a < 0$. For the cylinder geometry, the first crack node is at the bottom of the cylinder to correspond to the first crack front node in a 360-degree part-depth crack. At the end of the complex crack the crack depth must equal the negative mesh thickness, $a = -T$.

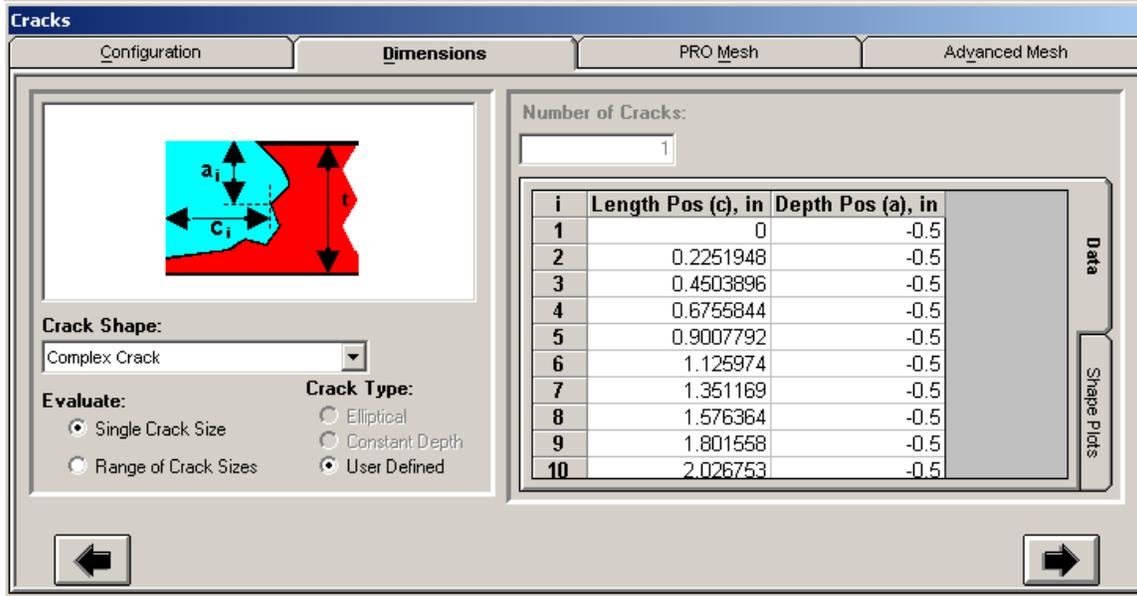


Figure 171 Select the “complex crack” shape and define the custom crack length and depth data

As a complex crack grows (when using the Fatigue module), the crack depth increases while the crack length decreases. The crack length decrease at the end of the complex crack occurs since the through-wall part of the crack is getting longer and the remaining un-cracked ligament is getting shorter. For the complex custom crack, the final length and final depth end conditions cannot be evaluated, so the other fatigue growth options of “Grow for a Number of Cycles” or “Grow for a Number of Steps” must be used. See the figure below. For the number of cycles option, the fatigue analysis will stop when the total number of cycles is greater or equal to the given number of fatigue cycles. The number of steps option will run the fatigue analysis for a set number of fatigue steps, generating that number of intermediate meshes.

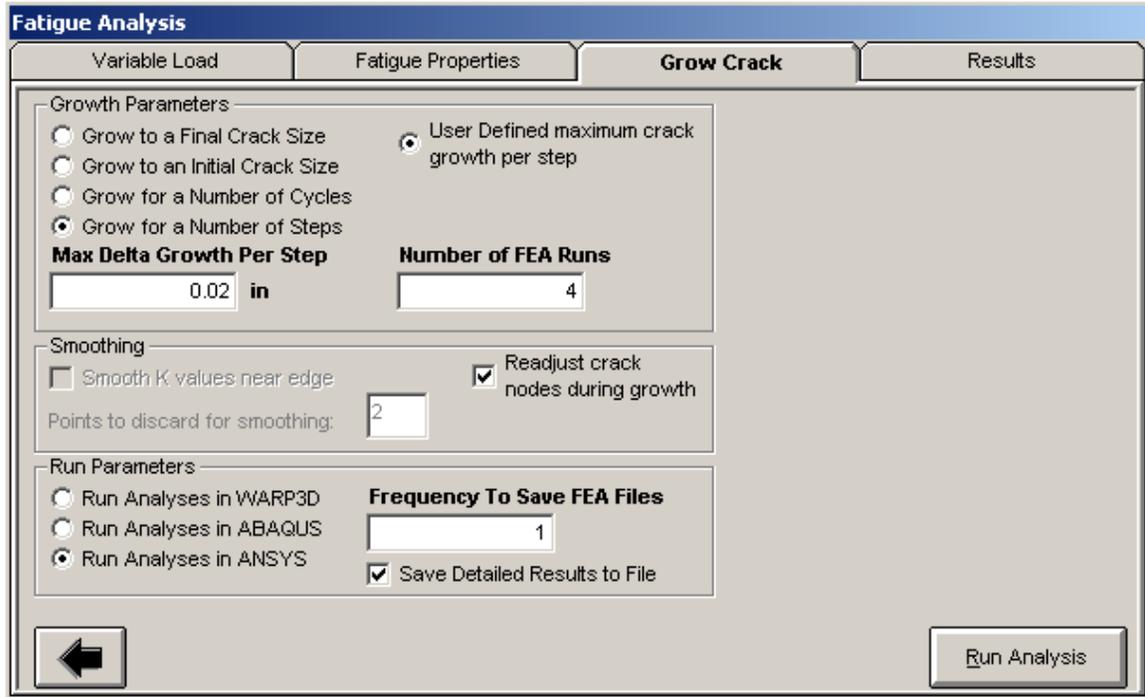


Figure 172 Use the number of cycles or number of steps to grow the complex crack

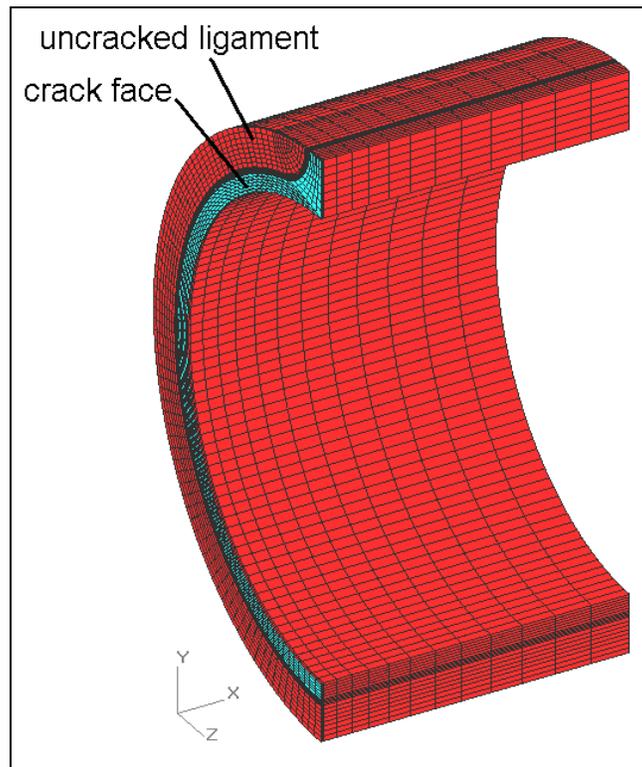


Figure 173 Complex custom crack shape; the part-through crack front around the circumference curves to the outside surface for a through-thickness segment of crack at the top of the cylinder geometry.

User-Defined Geometry

The user-defined geometry provides a general crack meshing capability in *FEACrack*. A complete crack mesh in an arbitrary six-sided primitive is generated by the mesh generator; consider the similarity to a full flat plate crack mesh but in an arbitrary shaped volume. A variety of crack shapes and symmetry choices are available for the user-defined geometry crack mesh. For example, a full surface crack can include both crack faces, from $\phi = 0$ (right crack tip) to $\phi = \pi$ (left crack tip), in the crack mesh ($\phi = \pi/2$ at the deepest point along the crack).

A definition mesh gives the user-defined geometry shape. The definition mesh can be created in a general meshing program, exported to an input text file, and then imported into *FEACrack*. In general, a brick (hex) element mesh with a mapped grid mesh pattern is used for the definition mesh. The number of elements on opposite edges must be equal; no mesh transitions are allowed in the definition mesh as a grid mesh is required to correctly define the transformation objects. A tetrahedron definition mesh can be given, but a corresponding brick definition mesh will be automatically generated using the shape of the tetrahedron mesh edges; however, the internal shape of the tetrahedron mesh may be lost, so the mapped grid brick definition mesh is preferred.

The brick elements in the user-defined definition mesh are used to form the transformation objects that will be used to transform the crack mesh into the desired shape. This gives the user direct control over the crack mesh transformation. In general 20-node brick elements are preferred to get the best transformation of curved surfaces, but 8-node brick elements in the definition mesh work well too. If possible, locate the 20-node brick mid side nodes on the geometry surface to better capture any surface curvature.

The user-defined crack mesh can be used as the final structure for a stand-alone analysis, or the transformed crack mesh can be added to another mesh as part of a larger structure and connected with tied contact (see below for the automatic combining of meshes). Boundary conditions can be applied to the user-defined crack mesh for direct analysis of the structure. No node constraints for symmetry planes are automatically defined for the user-defined geometry. The user can select the appropriate boundary conditions for each crack mesh surface. To add the crack mesh to a larger structure the tied contact capability in *Warp3D*, *Abaqus*, or *ANSYS* (bonded contact) can be used to combine and connect the meshes (requires a separate license for running *Abaqus* or *ANSYS*). *Warp3D* version 15 and later supports the tied contact capability and is included with *FEACrack*. To convert a larger model to *Warp3D* format, the *patwarp_windows.exe* program can be used or use the Tools/*Abaqus* to *Warp3D* Translator form. More information on the *patwarp* converter program is available in the *Warp3D* manual. A task list for combining meshes is given below. For more information and detailed instructions on combining the crack mesh with another mesh you may contact technical support (refer to the telephone number and e-mail address on the title page).

Additional custom boundary conditions can be added to the crack mesh using the “macro text” option, which may help obtain a stand-alone analysis model. Some node sets are pre-defined by the mesh generator that could be used to add boundary conditions to the crack mesh.

Building A User-Defined Crack Mesh

The user-defined geometry is typically used when the structural shape does not already exist among the geometry library choices. To begin, select the user-defined geometry from the

Structural Shape menu on the Geometry/Configuration form. On the Cracks/Configuration form the location of the crack within the user-defined shape is chosen. The crack can be located at the center of the shape or shifted to an off-center location given by setting the distances from the left and front edges of the geometry to the center of the crack. To locate the crack, select the User Controlled Location choice from the menu and then enter the W1 and L1 distances. The offset distances are measured from the corner 1 “C1” node in the definition mesh, which is listed at the bottom of the Geometry/Dimensions/Dim and Pic form. For a new model, it may be easier to start with the crack at the default center location and build the crack mesh once, then use the C1 reference corner to adjust the crack location within the shape and rebuild the crack mesh.

On the Cracks/Dimensions form enter the crack dimensions as usual; single or multiple crack dimensions can be entered. On the Cracks/Angle form, the orientation of the crack can be modified by entering a crack bevel angle (change the crack face angle relative to the surface normal), or by rotating the crack relative to the surface centerline.

Since the user-defined geometry can be a general shape, a definition mesh must provide detailed information describing the shape. The user-defined geometry is described by using brick (or tetrahedron) finite elements as the transformation objects and the node coordinates to give the user-defined geometry shape. The element objects can be 8, 20, or 27 node bricks; 20 node bricks are recommended to give the best transformed shape. Using 8-node brick elements can give a piece-wise planar transformed surface since the 8-node brick element faces do not have any curvature. The user-defined geometry data is entered in the Geometry/Dimensions forms.

Importing the Definition Mesh

The preferred method is to create a definition mesh using general FEA meshing software and export it to an *Abaqus*, *ANSYS*, *Warp3D* input file or a Femap neutral file. The definition mesh can then be imported into *FEACrack* to provide the user-defined geometry data. Use the Import button at the bottom of the Geometry/Dimensions/Number of Objects form; then select one of the file types that contain the definition object mesh. Use a *.inp file extension for the input file name that contains the definition mesh. Ansys .ans or .cdb (coded data base files) and Femap neutral files (*.neu) can also be imported. The number of objects in each direction along the mesh edges, the node coordinates, and the object connectivity tables will be filled in from the input file. An alternative method is to enter the geometry data using the spread sheet tables for node coordinates and the object connectivity. The data can be pasted into the spread sheet tables. Once the definition mesh data is entered in *FEACrack* it is saved in the model file so that no future import or entry of the object data is needed.

When importing the definition mesh, the default algorithm that finds the corner nodes and edge nodes uses the Tools/Model Options/Crack Analysis/Imported Model Tolerance Angle value to compare the brick element face outward normal direction vector. The default import tolerance angle is 45 degrees, but can be modified if needed. A valid definition mesh should have the number of objects (ie brick elements) along each edge updated, and the eight corner node ID numbers list in the Dim and Pic (Dimensions and Picture) form. If adjusting the import tolerance angle does not allow the definition mesh import to succeed (too many surfaces or corners are found), try the comment data block method described below.

Definition Mesh Corner Nodes Comment Data Block

The alternative to using the import tolerance angle is to provide the eight corner nodes in a comment block in the definition mesh file. In the general mesh software used to create the definition mesh; also use it to identify the eight corner node ID numbers. For example, the definition mesh shown in the figure below (from the 4-point bending load pin example) has corner node ID numbers: 10, 78, 79, 11, 45, 113, 112, 44.

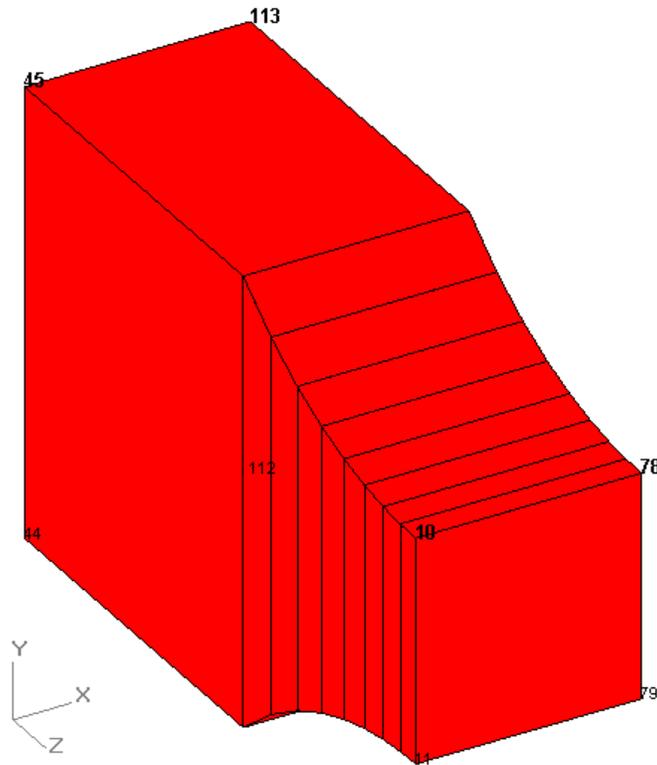


Figure 174 Identify the definition mesh corner node ID numbers for the comment data block

Near the top of the definition mesh input file, list the eight corner nodes within the “definition mesh corner nodes” comment data block (after exporting the definition mesh to an input file). For example using an Abaqus format input file the comment data block syntax and corner node list are given as comment lines near the top of the file:

```
**  
** definition mesh corner nodes  
** 10 78 79 11 45 113 112 44  
** definition mesh corner nodes  
**
```

Note that the comment data block begins and ends with the specific key words. If this comment data block is found when reading the definition mesh file, the given corner node ID numbers will be used instead of the import tolerance angle to find the edge nodes from the element connectivity. The comment data block should be near the top of the input file, within the first 20

key words or data lines (non-comment lines) of the file. The eight corner nodes can be entered in any order. If a particular corner node ID is convenient for locating, orienting, or shifting the crack location enter it as the first corner node ID number in the comment data block.

For a Femap neutral file (*.neu) the “model notes” can be used to give the definition mesh corner nodes data block. In Femap, use the File/Notes to enter the key word syntax and eight corner node ID numbers. Or add data block 540 to the Femap neutral file with the key words and eight corner nodes.

Updating the Definition Mesh Data

The numbers of objects in each direction of the primitive are defined at the top of the form. The product of the three numbers gives the total number of brick element objects in the user-defined geometry. Several brick object connectivity choices are available in the Element Numbering Scheme menu, and the brick element diagram shows the node numbering order to define the object. On the Coord List tab the node coordinates and object connectivity are listed in the spreadsheet tables. After entering the object data or making changes click on the “Update” button to refresh the object length dimensions and corner node list on the Dim and Pic form.

Based on the node coordinates, the surface length along each edge of the geometry will be computed. The average arc length of the four edges along the same direction are listed as the initial length estimates. If these estimated values are inaccurate they can be modified by the user.

The surface length dimensions should be as accurate as possible to avoid stretching the crack mesh relative to the user-defined geometry. For example, if the surface width (W) was entered as twice the actual width (too large), then the crack will be half as long relative to the user-defined object. The mesh transformation maps the primitive crack mesh into the user-defined geometry. The initial crack mesh is built using the given surface lengths. The initial surface length dimensions should match the definition object mesh size as closely as possible to avoid stretching of the crack during the mesh transformation.

The crack location relative to the object can be selected from the Crack Location menu. The corner node list is updated from the crack location choice to define the 8 corner node ID numbers. The corner node ID numbers are used to orient the crack location and direction relative to the primitive crack mesh for the user-defined geometry transformation. Refer to the bold corner node ID numbers in the preview screen to see the reference points for the crack location.

The Preview form shows the user-defined geometry definition object mesh that will be used for the crack mesh transformation. If the mesh picture looks incorrect, the object data will need to be corrected. When using the node, element, or corner labels the wire frame picture is recommended. Using the labels can help to correct the object data by finding the node or element or corner that is incorrectly defined.

Boundary conditions can be chosen for each face of the crack mesh. The mesh face menus (front face, right face, etc.) are defined relative to the crack location in the object. The front face is always in front of the crack, the right face is always to the right side of the crack, the crack opens onto the top face, etc. The boundary condition labels use the definition mesh corner nodes as reference for each of the six mesh surfaces. Also use the definition mesh node ID numbers to locate each crack mesh surface.

A user-defined geometry example of a generic doubly curved plate is available in the Example Files download (contact us to request the download). Load the model file (*.elt) into *FEACrack* to view the mesh and user-defined object data. Two definition meshes are available in *Abaqus* input file format. The same doubly curved plate object mesh is given using 20-node brick elements and with 27-node brick elements. The doubly curved plate user-defined geometry and the crack mesh after transformation are shown below. Other user-defined crack mesh examples are available from the *FEACrack* web page.

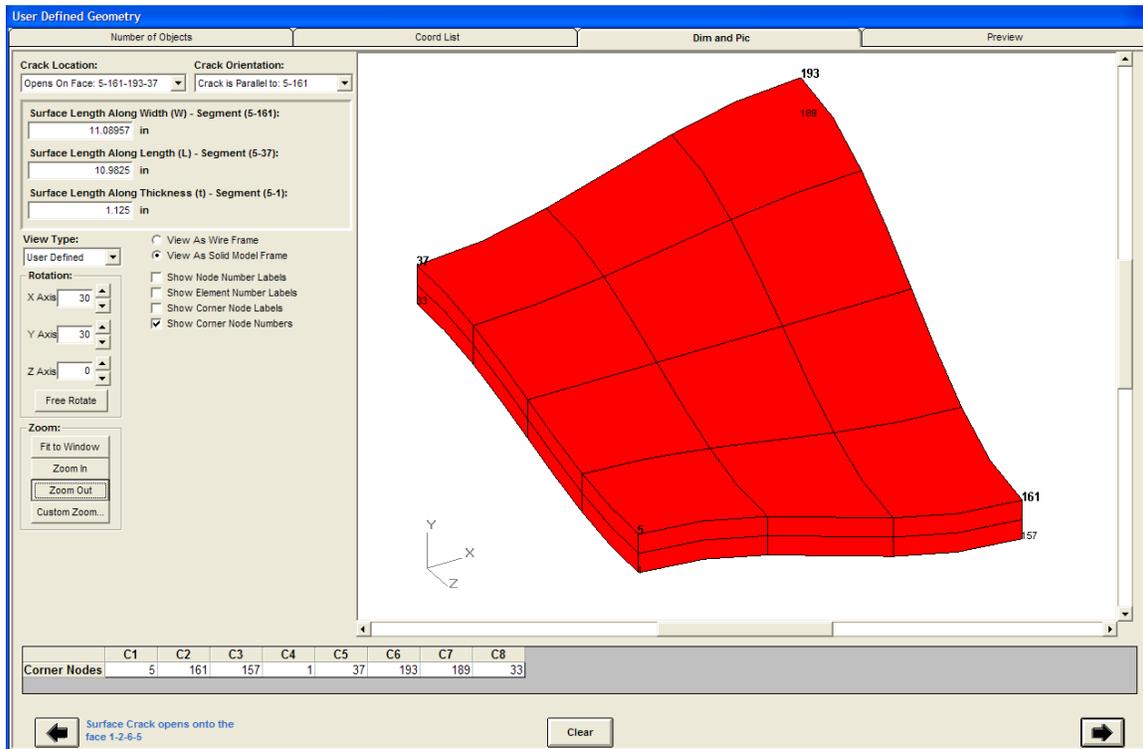


Figure 175 Doubly curved plate user-defined geometry imported definition mesh; 24 objects.

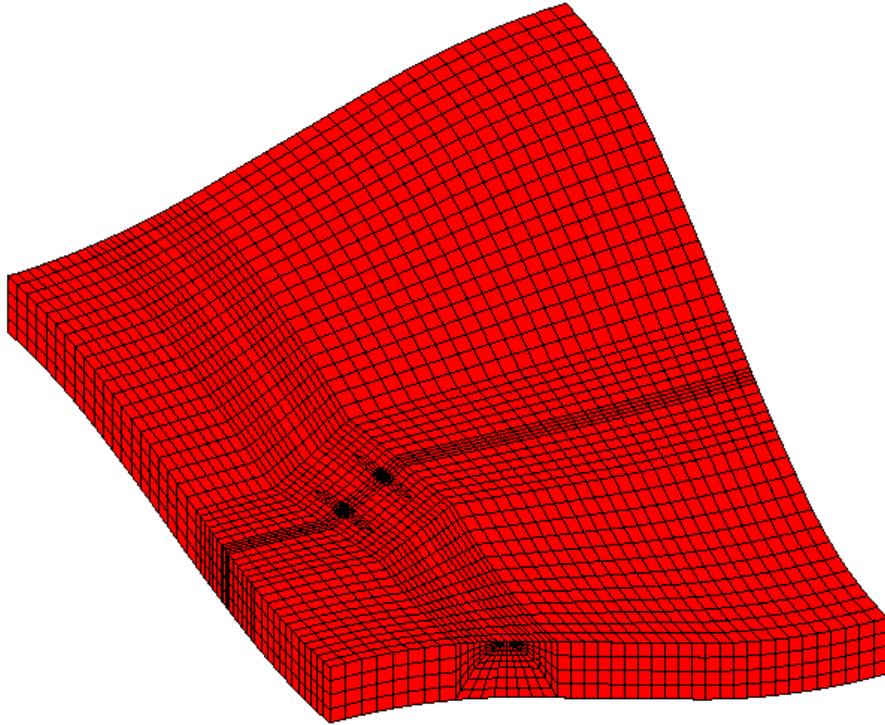


Figure 176 Crack mesh in the user-defined geometry; the crack is shifted and rotation from the center of the model.

Automatic Combining of Crack Mesh and Master Mesh

The user-defined crack mesh and master mesh input files can be automatically combined. The crack mesh will fill the volume given by the definition mesh, which should be removed from the master mesh leaving an open region for the crack mesh. Typically the crack mesh and master mesh will be connected using tied contact on their adjacent mesh surfaces. Additional data can also be provided using the macro text option. This automatic mesh combining option is currently available for Abaqus, ANSYS, and Warp3D input files.

This option can be used with the fatigue analysis module so that the updated crack mesh is combined with the master mesh for each increment of the fatigue crack growth.

The automatic file combining occurs during crack mesh generation when the input file is written, and the option is currently activated using key word text in the Home/Notes form (see Figure below):

*use combine master mesh = <file name>

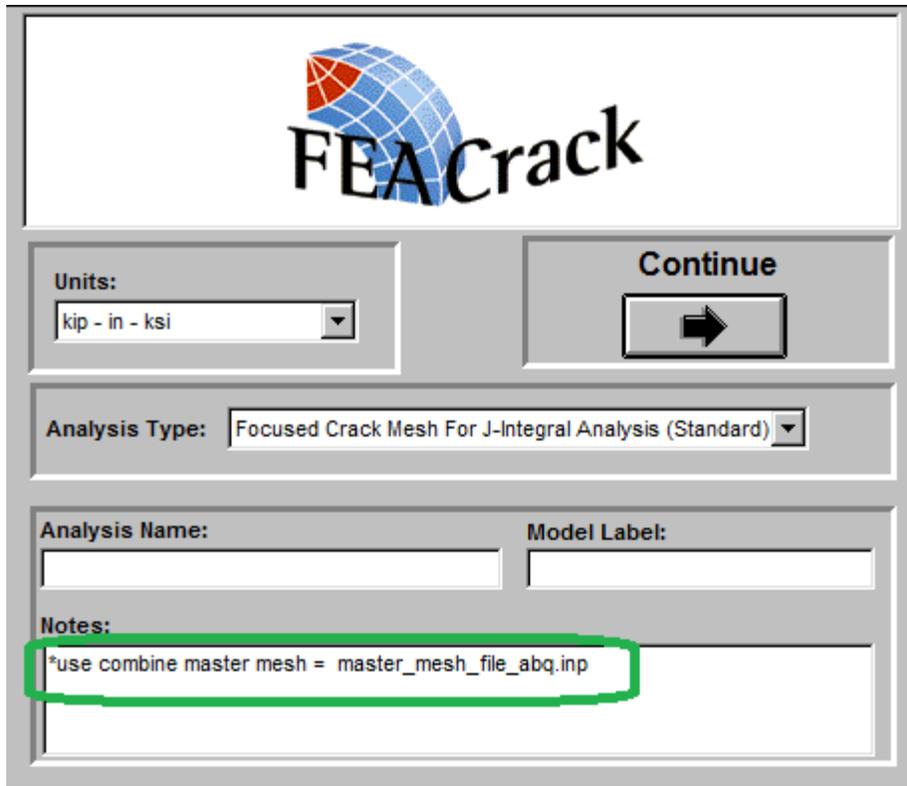


Figure 177 Identify the master mesh input file name after the key word.

The master mesh input file and generated crack mesh input file need to be for the same FEA solver choice; check the Tools/Program Options/Analysis form for the FEA solver choice. If the master mesh input file is in the same directory as the *.elt model file, just the master mesh file name can be given, or include the full path to the master mesh input file if it is located in another directory. If there are spaces in the file path or file name use single quotes around the path and file:

*use combine master mesh = 'c:\case 1\master mesh file.inp'

The master mesh input file needs to be prepared with key words to identify each section of mesh data that will be copied from the master mesh to the combined mesh input file. The key word is used at the beginning and end of the section of data to indicate where to start and stop copying data from the master mesh to the combined input file. The key words are added as comments in the master mesh input file, and have the format:

master mesh <key>

The key words for each section of data are:

- master mesh coordinates
- master mesh element connectivity
- master mesh material
- master mesh element types (used with Warp3D, see note below)
- master mesh surface
- master mesh tied contact
- master mesh model data
- master mesh constraints (within the load step data)

- master mesh traction load (within the load step data)
- master mesh step data <n> (general step and boundary data for step <n>)
 - Where <n> is the analysis step number
 - Allows different data to be added in specific analysis steps
 - Example for step 3: ** master mesh step data 3

The available master mesh key words can also be listed in the crack mesh <model>.log file by setting the Tools/Program Options/Analysis log file output level ≥ 3 , and view the log file with a text editor and search for “master mesh key words”. A summary of the master mesh data sections is also written to the log file to confirm that the data needed for the combined mesh has been copied.

When tied contact is used to connect the crack mesh and master mesh, start by building the crack mesh once to get the crack mesh surfaces labels that will be needed for the tied contact. Add the tied contact syntax to the master mesh input file and build the crack mesh a second time to get the ready-to-run combined model.

Note about Warp3D element type

When combining a master mesh and crack mesh for Warp3D analysis, confirm that the output location (centroid, Gauss points or nodes) is consistent between both meshes to avoid post-processing errors.

Here is an example of the master mesh node coordinates with the key words in an Abaqus input file:

```

** master mesh coordinates          <- first key word starts the data section
*NODE, NSET=GLOBAL
  1, 1.2096011, 26.933862, 2.96101
  2, 1.2113917, 26.973733, 2.6317364
...
14432, 20.906407, 0., 3.0935928
14433, 20.641242, 0., 3.3587585
** master mesh coordinates          <- second key word finishes the data section

```

Here is an example of the master mesh node coordinates with the key words in an ANSYS input file (APDL format):

```

! master mesh coordinates
N, 1, 1.2096011, 26.933862, 2.96101
N, 2, 1.2113917, 26.973733, 2.6317364
...
N, 14432, 20.906407, 0., 3.0935928
N, 14433, 20.641242, 0., 3.3587585
! master mesh coordinates

```

Run the combined mesh analysis to confirm that the model contains all the needed data and behaves as expected.

Step-By-Step Tasks To Combine Meshes

The following list outlines the necessary tasks to create a crack mesh in a user-defined geometry. After generating the crack mesh, combine the crack mesh and master mesh (larger mesh) input files to create the combined input file used to run the analysis. For this example a curved pipe (a piping elbow) with a flank surface crack is used for the master mesh and definition mesh. This example is available from the *FEACrack* web page or by contacting technical support. In the Figure below the definition mesh region is in red; this is the region that will be removed from the model and replaced with a crack mesh. The rest of the master mesh is in blue; this mesh will be connected to the crack mesh.

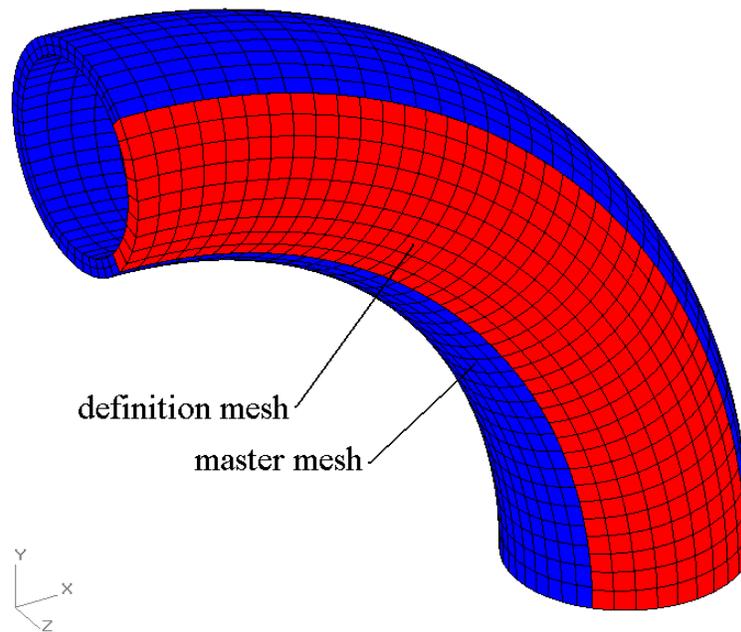


Figure 178 Piping elbow model; two mesh regions for the user-defined geometry.

The definition mesh removed from the curved tube model is shown in the figure below. Sufficient mesh refinement is needed to capture any curvature in the structure. In this case only one element is needed through the thickness since there isn't any curvature in that direction. The definition mesh is used only for geometry information and is not part of the generated crack mesh.

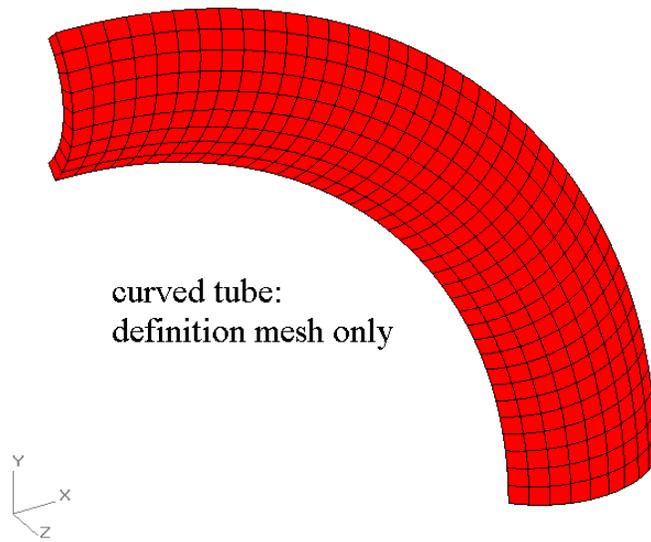


Figure 179

Definition mesh used to describe the geometry of the crack region; the definition mesh is imported into FEACrack.

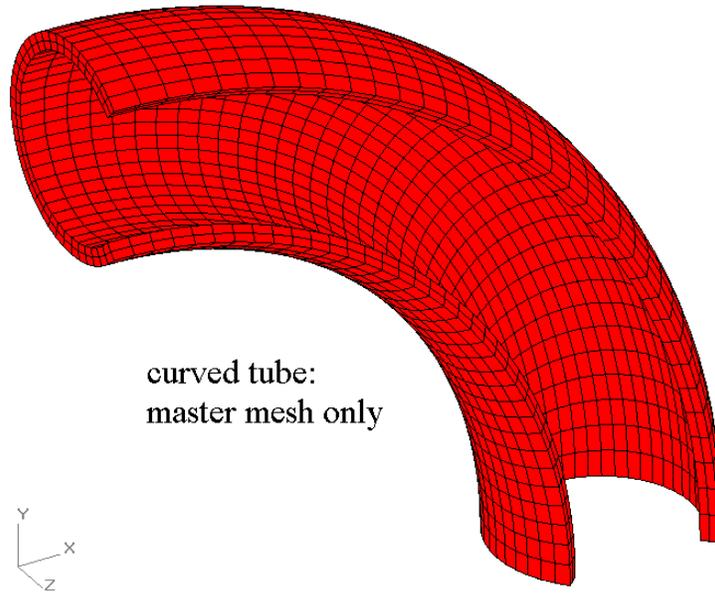


Figure 180

Definition mesh removed from the elbow model before the crack mesh is inserted.

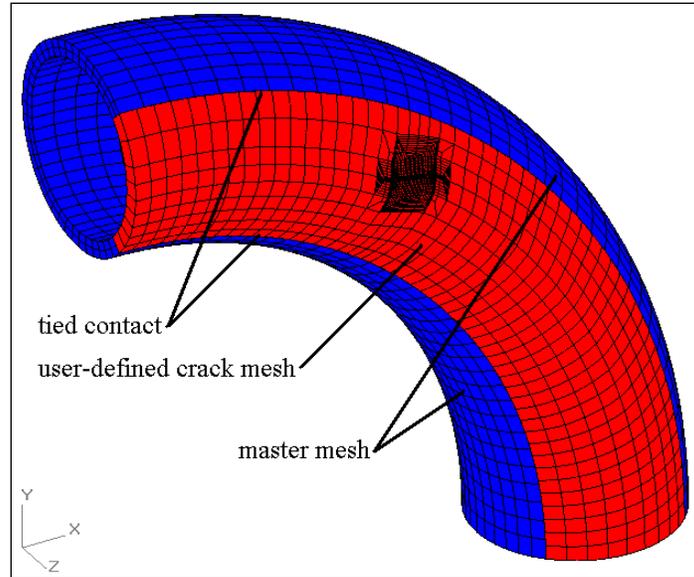


Figure 181 Piping elbow with a flank surface crack; the crack mesh (red) has been added to the rest of the elbow model (blue) and the two mesh regions are connected by tied contact.

1. As shown in the figures above, the full piping elbow mesh is divided into two regions: the master mesh (blue region), and the definition mesh where the user-defined crack mesh is inserted (red region). Note that the surface crack can be shifted and rotated in the user-defined crack mesh relative to the elbow centerline.
2. The master mesh has the user defined geometry region removed. In this example, remove the flank region on the front side of the elbow. Save this model as the master mesh as its nodes and elements may need to be renumbered later. Using “master_mesh” as part of the file name should help organize the many input files that will be used for the crack mesh model.
3. A definition mesh gives the user-defined geometry shape. In this example, the flank region on the front side of the elbow is the definition mesh. Before exporting the definition mesh to a file (*Abaqus*, *ANSYS*, or *FEMAP* format files can be used), renumber the nodes and elements in the definition mesh to start at 1 without gaps. Renumbering the definition mesh minimizes the memory used by *FEACrack* to store and process the user-defined geometry. Also, the data spreadsheets are limited to about 16,000 rows. The exported file can then be imported into *FEACrack*.
4. Export the definition mesh to an *Abaqus* (flattened format input file), *ANSYS* (input file or .cdb “coded data base” file), or *FEMAP* neutral format file.
5. In *FEACrack* select the user-defined geometry choice. Select the appropriate mesh symmetry and enter a crack shift location if needed. It may be easiest to start with the crack in the default center location and later shift the crack location once the mesh is getting generated. Then enter the crack dimensions. Import the definition mesh using the Import button at the bottom of the Geometry/Dimensions form. The same definition mesh can be used for more than one crack size.
6. Locate the crack relative to the definition mesh using the pull down menus on the Dim and Pic form. Note that the definition mesh corner node ID numbers are in bold on the mesh surface where the free surface of the crack opens. The definition mesh corner nodes are used as the reference points to locate the crack and apply boundary conditions.
7. Check the Surface Length Dimensions (W, L, t) in the Dim and Pic form. These values may need to be updated from the default estimates to the correct surface arc length depending on the crack location and orientation. The default values are computed from the average of the

definition mesh arc lengths on the four mesh edges in each direction. For example, if the crack is on the outer surface of curved geometry, the arc length on the outer surface should be used to update the mesh length dimension. Incorrect mesh length values will cause distortion of the crack dimensions in the transformed shape.

8. Select boundary conditions on the user defined crack mesh. In this example, two mesh surfaces have tied contact surfaces defined to connect with the master mesh. The left and right ends have node constraints applied. The inside surface has the internal pressure applied to match the pressure applied in the master mesh. Use the definition mesh corner node ID numbers as a reference to apply the boundary conditions to each mesh surface.
9. Build the crack mesh in the user-defined geometry. Make any adjustments to the crack location or mesh refinement and re-mesh if necessary. Also check that the desired format input file is being generated (in the Tools menu, Program Options form). For this example a *WARP3D* input file was created.
10. Either the crack mesh or the master mesh needs to be renumbered. For example, if the master mesh is to be listed first in the combined input file, then the crack mesh needs to be renumbered. Or if the crack mesh is to be listed first, then the master mesh needs to be renumbered (externally of *FEACrack*); use the number of nodes and elements from the crack mesh to renumber the master mesh. If there is more than one crack inserted into a larger global model, then each crack mesh may need to be renumbered. In general the node and element ID numbers of one of the meshes needs to be renumbered. For *WARP3D* the combined mesh must have consecutive node and element numbering, starting at 1, without gaps. To renumber the crack mesh use the Tools/Model Options/Numbering form to set the starting node, element, and crack ID numbers. For *Abaqus* it may be useful to use the parts and assemblies syntax. Select the parts and assemblies option in the Tools/Program Options form. Refer to the *Abaqus* user's manual for more information about parts and assemblies syntax.
11. Export or save the master mesh to an input file, renumbered if necessary, (*Abaqus* or *ANSYS* format), or to a *PATRAN* neutral file. Since most model building programs will not directly export a *WARP3D* format input file, export to a *PATRAN* neutral file and then use the "patwarp.exe" program to convert the master mesh *PATRAN* neutral file to a *WARP3D* input file. Refer to the *WARP3D* manual for more details about the patwarp.exe program and *WARP3D* syntax.
12. Use the crack mesh and master mesh input files to assemble the combined full model. A recommended method is to make a copy of the crack mesh input file, and rename it with the word "combine" in the file name to help organize the many files. This will become the complete input file containing the master mesh model and crack mesh for the analysis. By copying the crack mesh all the comment data block information (needed for crack result post processing) will be included in the combined model. The goal is to combine both meshes, add the necessary tied contact syntax to have a complete model containing a crack. A basic text editor like Notepad can be used when editing and combining the crack mesh input files.
13. Update the maximum node and element ID numbers for the combined mesh in the Mesh Size Data comment block at the top of the input file; these values can be rounded up for convenience. The post-processing module uses the maximum ID values. For *ANSYS* leave extra room in the maximum number of elements for the bonded contact elements that will be generated during the analysis; usually an extra thousand elements is sufficient. For *ANSYS* update the /FILENAME label at the top of the file to match the new combined input file name; the file name given in the /FILENAME line controls the *ANSYS* analysis output file names. The /FILENAME parameter may be limited to 32 characters.
14. Add the master mesh nodes to the combined mesh file. Copy the nodes from the master mesh input file and insert them either before or after the crack mesh nodes so that all the node ID numbers are in ascending order.

15. Add the master mesh elements to the combined mesh file. Copy the elements from the master mesh input file and insert them either before or after the crack mesh elements so that all the element ID numbers are in ascending order. The master mesh elements can use the same or different material data set as the crack nodes. In general, there can be numerous material groups in the combined model.
16. Add any material data used by the elements in the master mesh. Check that the material data for the crack mesh region is correct.
17. Add the master mesh contact surface data to the combined input file. Depending on the input file format the contact surface data may be a list of elements or a node set. The *WARP3D* input file for this example uses a list of elements and element face ID numbers to define the contact surfaces on the master and crack meshes. Also add the appropriate tied contact command that references the defined contact surfaces to connect the mesh regions. In *WARP3D* the command is “tie mesh”, or in *Abaqus* it is “*TIE”. In *ANSYS* the contact surfaces are defined by node lists, then use the TARGE170 and CONTA174 elements to generate the contact surfaces. Use KEYOPT(12)=5 for bonded contact. KEYOPT (5)=3 can be used to close any initial gap between mesh surfaces, and KEYOPT(9)=1 can be used to exclude any initial gap of offset between the mesh surfaces.
18. Add boundary conditions to the combined mesh input file: node constraints and loading. Some of the constraints may come from the master mesh and some from the crack mesh. In this example both meshes have internal pressure and node constraints at each end.
19. Save the combined mesh input file, shown in the figure above.
20. Run the combined crack mesh file analysis using the analysis program corresponding to the input file format. For this example, *WARP3D* was used to run the analysis.
21. Use *FEACrack* to load the results and extract the J-integral and related crack results. Check the deformed shape and stress results to check that the tied contact is working correctly, and that the mesh regions remain connected. Use the deformed shape to check that the applied boundary conditions cause the crack to open to get positive crack front stress intensity values.

Another approach to combine the crack mesh and master mesh input files is to use the “sub-files” option with the Abaqus **include* or *Warp3D *input* command. The sub-file option is in the Tools/Program Options form, and the crack mesh is written to several separate files, which each contain one section of the mesh data like the nodes, elements, constraints, loads, etc. By add the **include* command in each section of data in the Abaqus input file, the crack mesh can be included with the master mesh for the analysis. Add or create another sub-file to contain the tied contact data.

Fatigue Analysis Module

FEACrack can perform crack growth analyses under linear elastic conditions. Crack growth rate is inferred from the stress intensity solution for a given finite element run combined with a user-specified growth law. Crack dimensions are then updated, the model is re-meshed, and the process is repeated until a final crack size is reached.

Fatigue: Preliminary Steps

To perform a fatigue analysis on a geometry of interest, first input the geometry and dimensions as in a standard analysis. At the crack dimensions form, select the Single Crack Size option.

Input the appropriate loads, stresses, or displacements and the Boundary Conditions window. The number of time steps must be set to 1. For a single load or displacement, a unit value should be input. The resulting K solution will then be scaled by one or more amplitude values, as described below. For combined loading, such as bending + axial loading, one of the loads should be set to a unit value and the other(s) scaled accordingly. For example, if there is a 2:1 ratio between bending stress and axial stress, the former can be set to 1.0 and the latter to 0.5.

At the Material Properties window(s), set the material behavior to elastic and input the appropriate Young's modulus and Poisson's ratio.

Although not absolutely necessary, we recommend that you click the Build Mesh button to ensure that the dimensions are correct and that the desired mesh refinement is achieved. Refer to the sections above for instructions on adjusting the mesh parameters. In this instance, use the Refinement Indexes option to adjust the mesh surrounding the crack rather than the Mesh Stretch parameters. This will ensure a relatively constant mesh outside of the crack region as the crack grows.

Once the geometry, dimensions, boundary conditions, and elastic constants have been input, click on the Fatigue Analysis button.

Fatigue: Cyclic Loading

Both constant amplitude and variable amplitude options are available. In the former case only a single amplitude factor is required. If unit loads were specified at the Boundary Conditions window, enter the appropriate amplitude to scale the $FEACrack K$ solution to obtain ΔK for the fatigue analysis. If the imposed loading has been input as the actual cyclic value(s), enter 1.0 in the amplitude field.

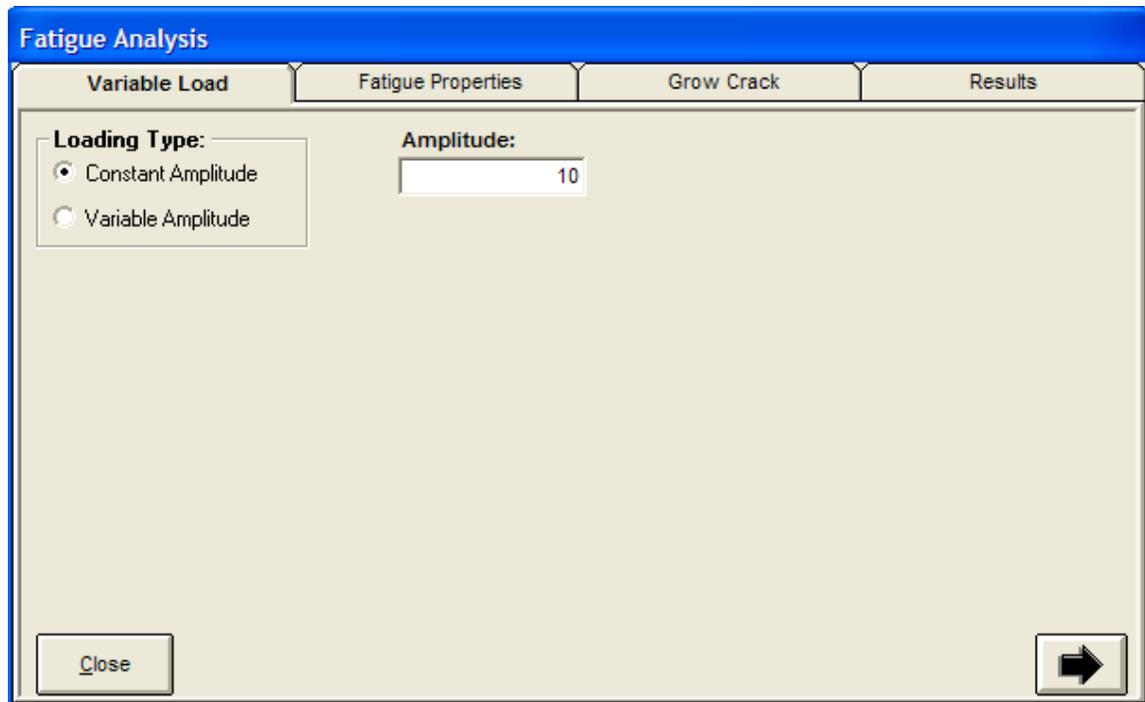


Figure 182 Constant amplitude fatigue analysis; enter the loading amplitude value

For variable amplitude loading, the amplitude spectrum can be specified by a Rayleigh distribution or by a histogram table. The procedure for handling variable amplitude loading is illustrated below. For a given crack size, the amplitude histogram is converted into a ΔK histogram by multiplying the former by the K solution from the corresponding finite element run. Next, the ΔK histogram is then converted to a crack growth rate (da/dN) by applying the growth law. A weighted average growth rate is then computed as follows:

$$\frac{d\bar{a}}{dN} = \frac{\sum \left(\frac{da}{dN} \right)_i N_i}{N_{tot}}$$

Equation 183 Weighted average growth rate

Where N_i is the number of cycles in a given histogram bin and N_{tot} is the total number of cycles in the histogram. This process is repeated for each crack growth increment. More equations are given later in this section.

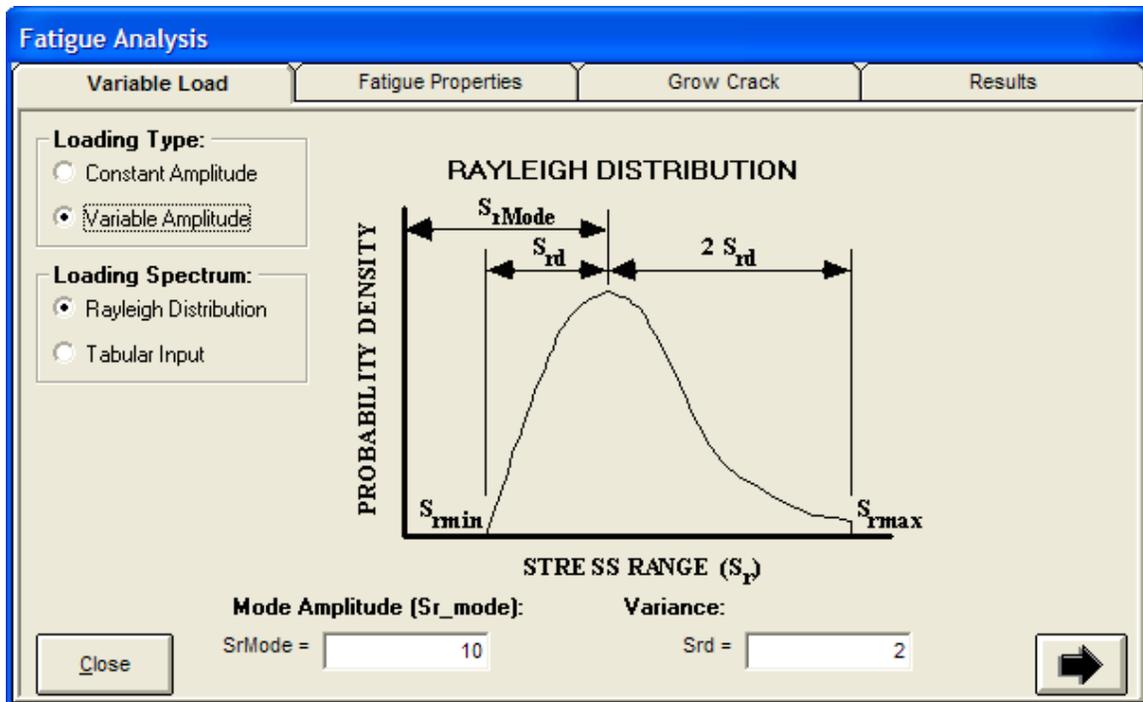


Figure 184 Variable amplitude fatigue analysis; enter the Rayleigh values

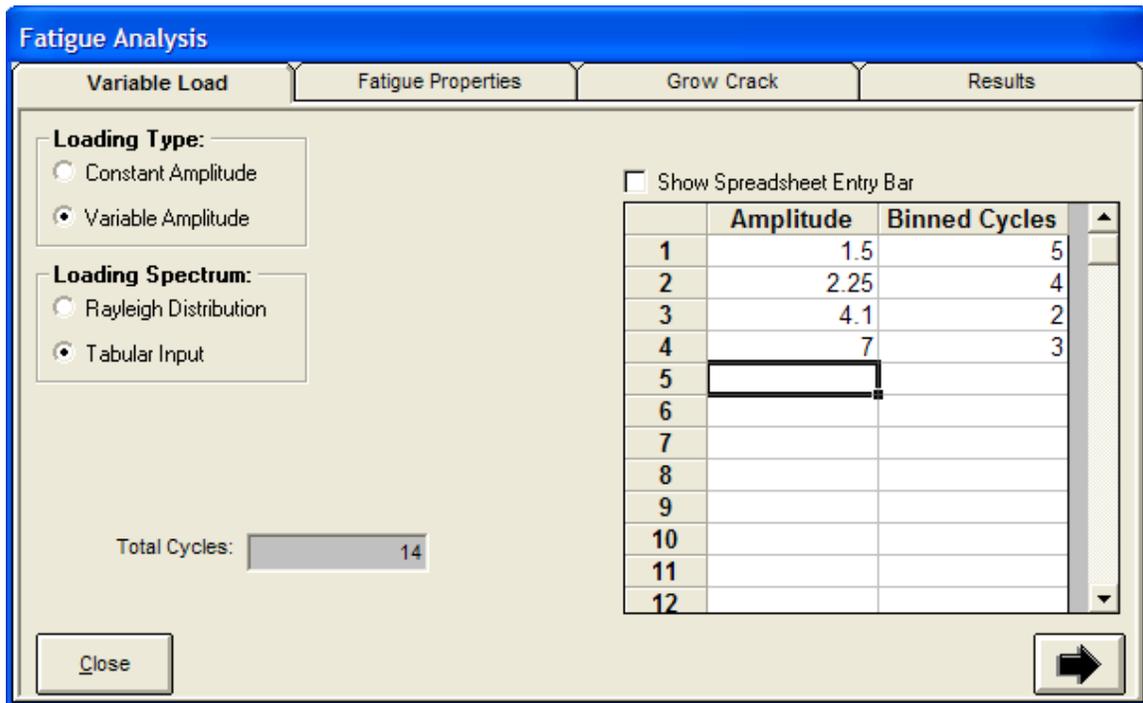


Figure 185 Variable amplitude fatigue analysis; enter the Tabular Histogram values

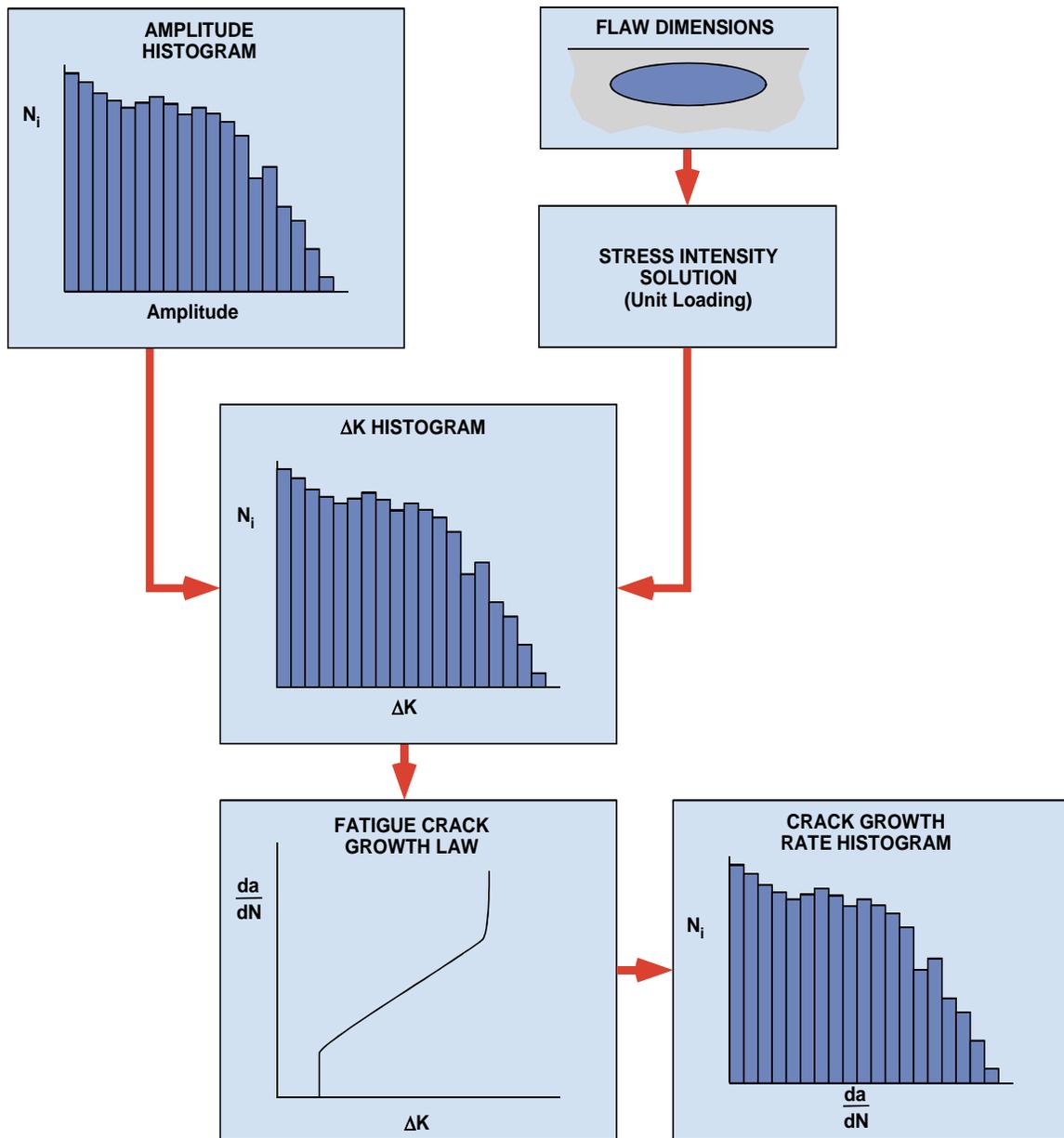


Figure 186 Histogram calculations for variable amplitude fatigue analysis

Fatigue Properties

Click on the Fatigue Properties tab to specify the desired growth law. Several common fatigue crack growth equations are available for which several material parameters must be input. Alternatively, you may select User-Defined Growth Law and enter a table of ΔK versus da/dN values. Note that this table is a functional spreadsheet, so the growth rate table may be generated by entering a spreadsheet formula and using the **Edit → Copy Down** menu command.

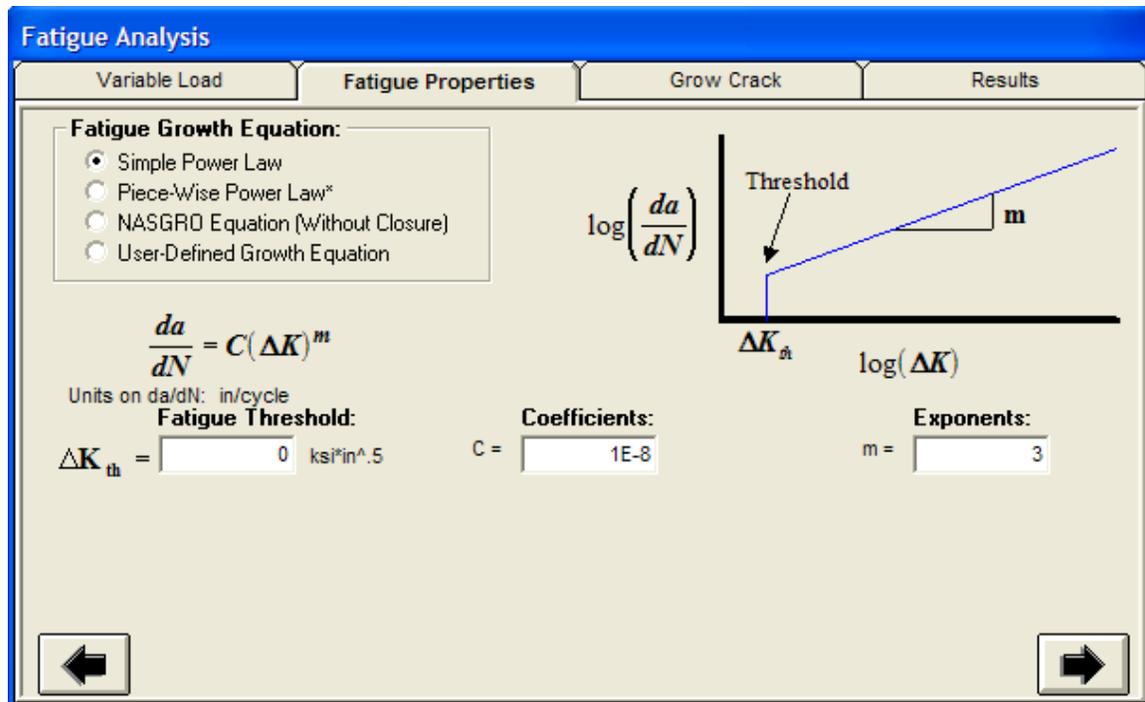


Figure 187 Select the fatigue growth equation and enter the Fatigue values

Fatigue: Running the Analysis

The starting crack size for the fatigue analysis is the value specified in the crack dimensions window, accessed through the Geometry button on the navigation strip. The fatigue analysis grows the crack to a specified size, or for a specified number of fatigue cycles, or for a specified number of analysis steps. Note that the analysis can be run backwards to determine the number of cycles to achieve an initial crack size given the final dimensions. The crack growth increment and the crack size at the end of the analysis can be specified either in terms of the depth (a) or length (c) dimension.

On the Run Parameters tab, specify the frequency that finite element input and results files are saved. For example, enter 1 to save the mesh and result files for every step, or enter 5 to save the files every 5th step. Finite element files from intermediate steps that are not saved will be overwritten. Select the FEA program to run the analysis. The “Save Detailed Results to File” option activates the fatigue log file, which is located in the same directory as the mesh and FEA results files. The fatigue log file name has the format: <model name>_fatigue.log and can be viewed with a text editor. The FEA Run Time Commands text box allows additional command line syntax to be entered; for example, a user subroutine can be specified for Abaqus.

Click on the Run Analysis button to start the fatigue analysis (on the Growth Parameters tab). A mesh will be generated with the starting crack dimensions, and the finite element analysis will be run. At the conclusion of the finite element analysis, the results file will be automatically post-processed and a new mesh will be created. Because a trapezoid integration scheme is used (see below), the incremental number of cycles, ΔN , cannot be computed until after the second step is completed.

Clicking the PAUSE Analysis button will cause the analysis to be suspended at the conclusion of the finite element analysis that is currently running. You may then resume the analysis at the point it was interrupted. If you wish to start over from the beginning crack size, click on the Clear Results button and then click on the Run Analysis button.

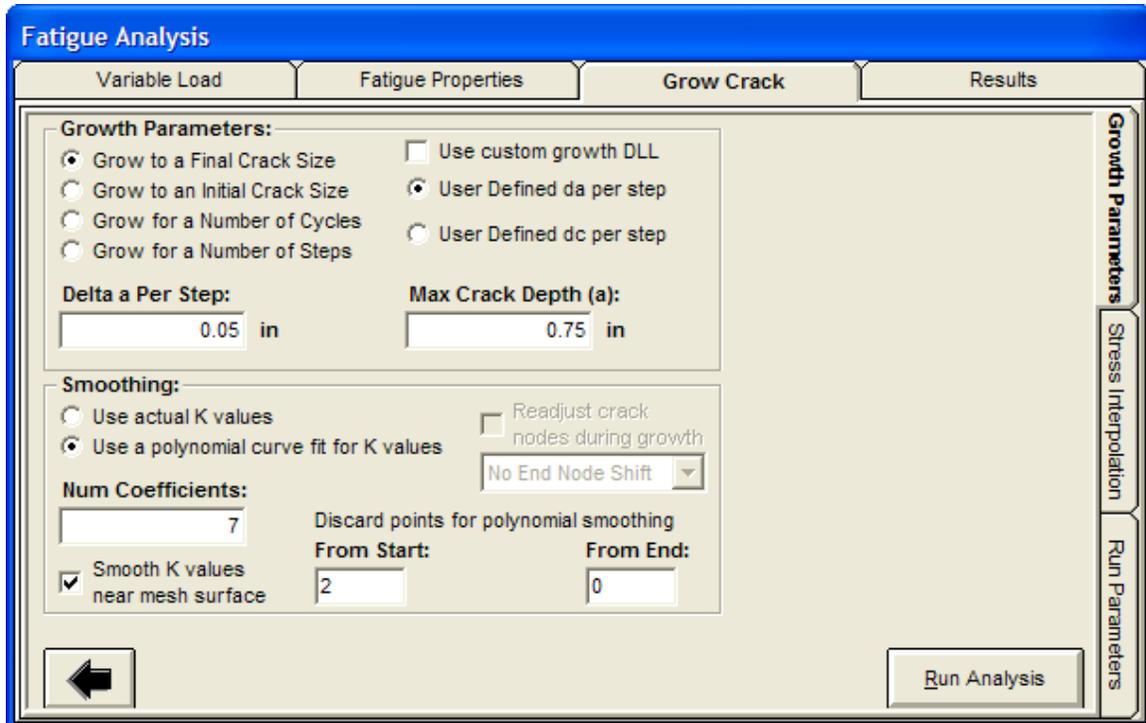


Figure 188 Fatigue analysis controls; enter the crack increment and final size; select crack front K options

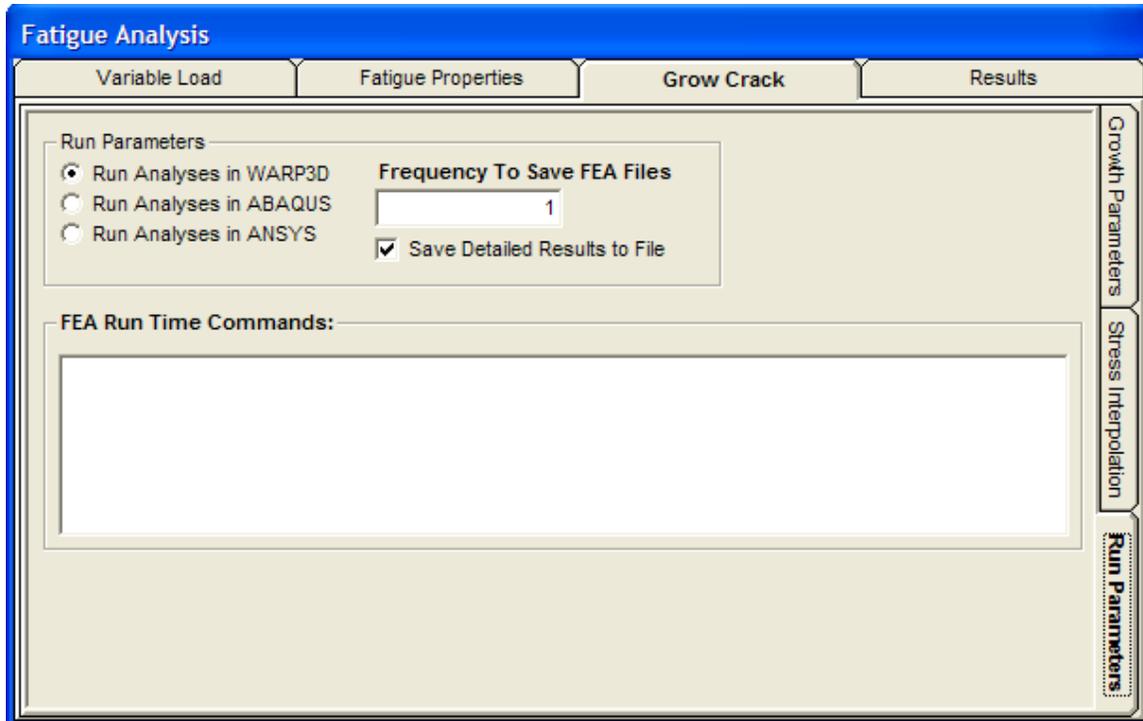


Figure 189 Fatigue analysis run-time controls; select the FEA program, enter the file save frequency, activate the fatigue log file option

Step: 7 ← a

1312204.42 Step Cycles } b
 12171626.11 Total Cycles }
 Crack Front: 1
 Crack Length (2c): = 0.7222224 in } c
 Crack Depth (a): = 0.32 in }
 Polynomial curve fit used to compute Fatigue Growth K Values. 5 coefficients.
 0.762507170718672 -0.252309421537259 0.322886829242329 -0.170472974167875 3.27292445900119E-02 } d

Row	Norm Crack Pos	Crack Front X	Crack Front Y	Crack Front Z	K From FEA	K Used For Growth
1	0	0.361111194	0	0	0.796834398812894	0.762507200241089
2	0.065611	0.360334225	-0.0209803429	0	0.743765296874087	0.747295320034027
3	0.130967	0.358018672	-0.0417896801	0	0.735295412894795	0.734627962112427
4	0.19635	0.354172544	-0.0624289016	0	0.723088615803452	0.72417277097702
5	0.262165	0.348772439	-0.0829350793	0	0.715833490272815	0.715635597705841
6	0.32752	0.341915652	-0.102942600	0	0.709124210680107	0.708894073963165
7	e	f	f	f	g	h
8						
9	0.52401	0.312657279	-0.160113812	0	0.69748370568143	0.696893930435181
10	0.589049	0.300252984	-0.177702471	0	0.694483761860379	0.695017278194427
11	0.655065	0.286364249	-0.194947375	0	0.694404640953046	0.693889677524567
12	0.720417	0.271386143	-0.211103399	0	0.693692146691597	0.693394362926483
13	0.785398	0.255344174	-0.226274165	0	0.692582365378293	0.69338071346283
14	0.85139	0.237949878	-0.240703124	0	0.69392972606269	0.6937335729599
15	0.916743	0.219703011	-0.253959757	0	0.694483761860379	0.694340407848358
16	0.981748	0.20062263	-0.26607027	0	0.694246372086955	0.695107161998749
17	1.047647	0.180414999	-0.277200019	0	0.696301061346226	0.695972204208374
18	1.113001	0.159600862	-0.287049301	0	0.697247337290904	0.696853995323181
19	1.178097	0.138191271	-0.295641444	0	0.697089713837485	0.697707653045654
20	1.243855	0.115970245	-0.303049247	0	0.699057452505438	0.6985102891922
21	1.30921	0.0933881626	-0.30911389	0	0.699685960119474	0.699228584766388
22	1.374447	0.0704492991	-0.313851283	0	0.699214632373081	0.699860990047455
23	1.440033	0.0470855268	-0.317268055	0	0.700627664395067	0.700420260429382
24	1.50539	0.0236022635	-0.319315751	0	0.700862892759726	0.700924336910248
25	1.570796	0	-0.319999993	0	0.726046045797733	0.701413214206696

Figure 190 Fatigue analysis log file lists detailed results

The following list describes the labels, by alphabetic letter, in the fatigue output log file figure:

- a. Fatigue analysis step
- b. Step and total cycles
- c. Updated crack growth dimensions
- d. Coefficient of the polynomial used for curve fit
- e. Normalized crack position along the crack front
- f. Cartesian coordinates along the crack front
- g. Unaltered K values
- h. K values from the polynomial used for curve fit

To compare the unaltered K values and the polynomial curve-fit, copy the fatigue log file values to a spreadsheet program to make the desired comparison plots.

Fatigue: K Smoothing Options

The crack front K smoothing options give more control of the fatigue analysis. In the figure below, the smoothing options are highlighted in the red box. It is recommended to first run the fatigue analysis with the “Use actual K values” to observe the crack front K values as the crack grows, and to check the J path dependence along the crack front. If local peaks or “spikes” occur due to numerical reasons, the polynomial curve fit and end point options can be activated to improve the fatigue analysis. The polynomial order is set by the number of coefficient values; for example, 7 coefficients gives a 6th order polynomial. The “discard points” option allows a given number of crack front nodes to be discarded from the polynomial curve fit, which is useful if there is high J path dependence for crack nodes near a mesh surface.

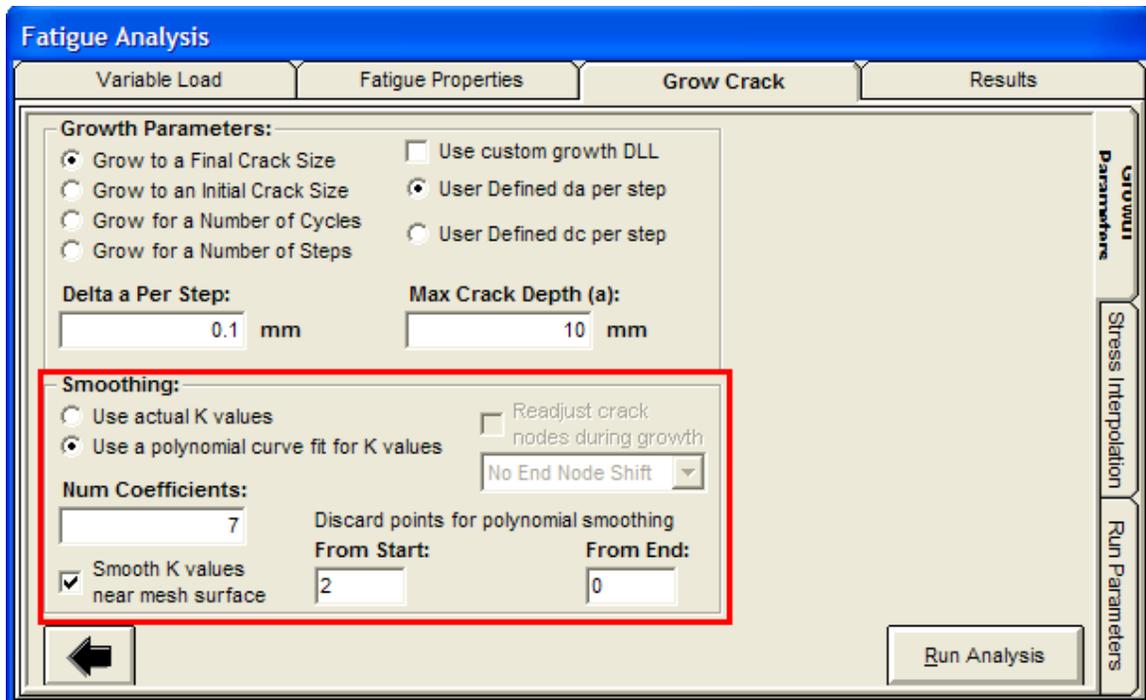


Figure 191 Fatigue analysis K smoothing controls

Consider a semi-elliptical surface crack, $c/a = 1$, and the crack front K results in the figures below. Since the K values at the crack tip free surface nodes often have high J path dependence values, there may be a local spike in the crack front K results. This is an example of when to use the polynomial curve fit and to consider discarding first one or two crack front K results.

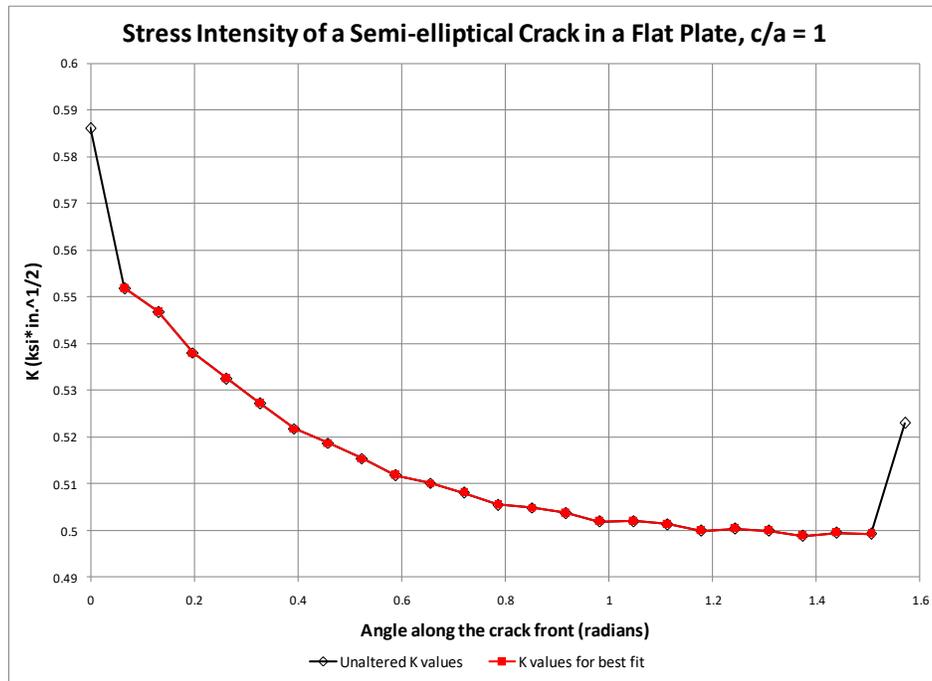


Figure 192 Example of a numerical spike in the crack front K results at each mesh surface; semi-circular surface crack, $c/a = 1$

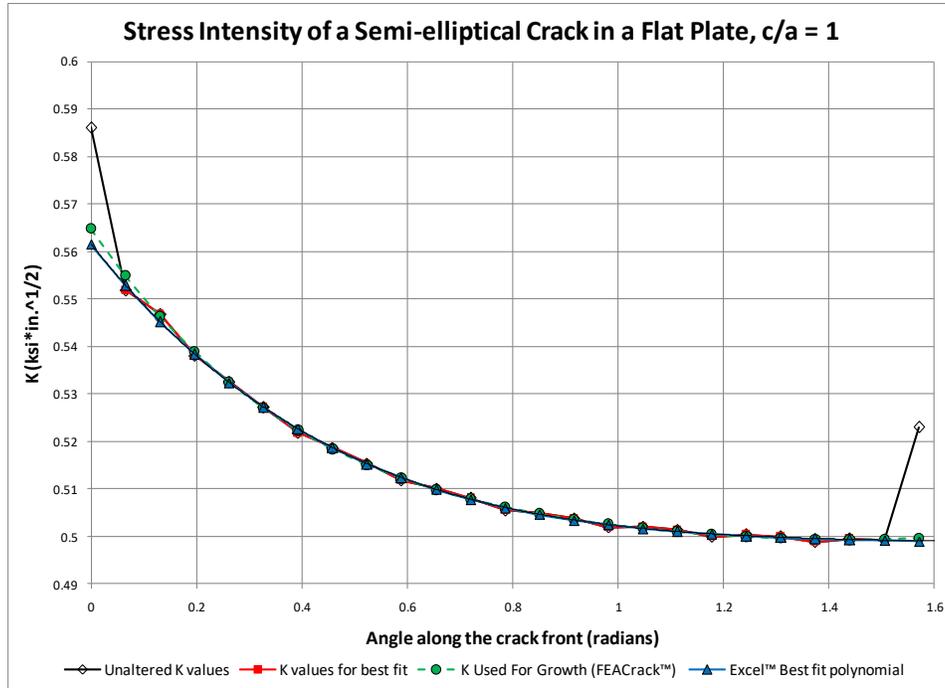


Figure 193 Comparison of crack front K values using the polynomial smoothing and discarded end point values

Fatigue: Life Prediction Methodology

The standard approach to fatigue life prediction is to integrate the crack growth rate from an initial crack size to a final size:

$$N = \int_{a_o}^{a_f} \frac{da}{(d\bar{a}/dN)}$$

Equation 194 Crack growth integration

Since $d\bar{a}/dN$ is a function of crack size, it is re-evaluated at each step of the numerical integration. For the special case of constant amplitude loading, a single crack growth rate, da/dN , is used in place of the weighted average. In expressions that follow, the weighted average crack growth rate is used to represent the most general case.

The above expression pertains to one-dimensional crack growth. In typical 3D cases, however, cracks can grow in both the length and depth directions. That is, the crack growth is two-dimensional. Examples of two-dimensional crack growth for surface and subsurface flaws are shown below.

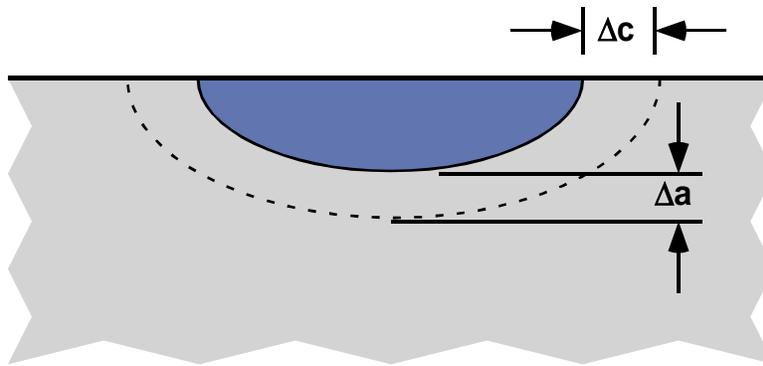


Figure 195 Surface flaw crack growth

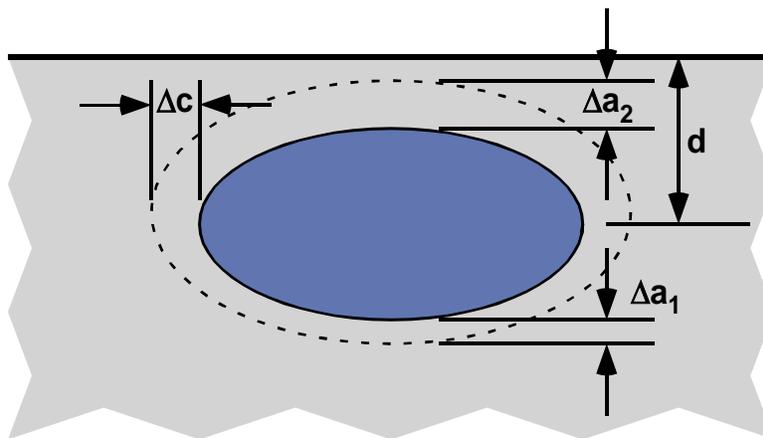


Figure 196 Subsurface flaw crack growth

Fatigue Crack Growth Equations

For surface cracks and corner cracks, *FEACrack* computes the growth rate at the deepest point of the crack and free surface (da/dN and dc/dN , respectively) from the K solutions at those locations. The crack shape is assumed to remain elliptical with growth (unless the custom crack is used), so crack growth rates at only two locations are necessary to define the evolution of the crack. Either the crack length increment, Δc , or the crack depth increment, Δa , is specified by the user, and the other is computed from the relative growth rates. For example, if crack depth increment is user-specified, the crack length increment for the i th increment is computed using the equations given below.

The crack length, $2c$, and crack depth, a , are updated during fatigue crack growth analysis using the following equations. The fatigue equation can have a general form, $f(\Delta K, R, \phi)$; for this example the simple Paris equation is used [1].

$$\frac{dc}{dN} = f(\Delta K, R, \phi) = C \Delta K_{\phi=0}^m$$

$$\frac{da}{dN} = f(\Delta K, R, \phi) = C \Delta K_{\phi=\pi/2}^m$$

Equation 197 **Crack growth rates**

Where a is the crack depth, c is the surface crack half length, N is the number of fatigue cycles, K is the stress intensity at each position on the crack front, and C and m are fatigue coefficients. The crack front position is given by the angle ϕ are 0 at the free surface and $\pi/2$ at the crack depth.

The fatigue differential equations are converted to incremental form for the numerical analysis.

$$\frac{\Delta c}{\Delta N} = f_{\phi=0}$$

$$\frac{\Delta a}{\Delta N} = f_{\phi=\pi/2}$$

Equation 198 **Incremental form**

These equations are integrated to compute the number of fatigue cycles, ΔN , for one analysis increment, i , to advance the crack depth and crack length.

$$\Delta N_i = \int_{c_i}^{c_i+\Delta c} \frac{\Delta c}{f_{\phi=0}}$$

$$\Delta N_i = \int_{a_i}^{a_i+\Delta a} \frac{\Delta a}{f_{\phi=\pi/2}}$$

Equation 199 **An Increment number of cycles**

Since the crack values needed are available from the current and previous increment FEA results, the trapezoidal rule is used to evaluate the integrals. The trapezoidal integration rule for a single increment Δx is given by [2]:

$$\int_x^{x+\Delta x} f(x)dx \cong \frac{1}{2}(f(x) + f(x + \Delta x))\Delta x$$

Equation 200 **Trapezoidal integration**

Using the trapezoidal rule, the equations to integrate the fatigue crack growth from the previous increment, $i-1$, to the current increment, i , at the crack tip and crack depth locations are given by:

$$\Delta N(c)_i = \int_{c_i}^{c_i+\Delta c} \frac{\Delta c}{f_{\phi=0}} \cong \frac{1}{2} \left(\frac{1}{f_{\phi=0,i-1}} + \frac{1}{f_{\phi=0,i}} \right) \Delta c_i = \frac{1}{2} \left(\frac{1}{\frac{dc}{dN}_{i-1}} + \frac{1}{\frac{dc}{dN}_i} \right) \Delta c_i$$

$$\Delta N(a)_i = \int_{a_i}^{a_i+\Delta a} \frac{\Delta a}{f_{\phi=\pi/2}} \cong \frac{1}{2} \left(\frac{1}{f_{\phi=\pi/2,i-1}} + \frac{1}{f_{\phi=\pi/2,i}} \right) \Delta a_i = \frac{1}{2} \left(\frac{1}{\frac{da}{dN}_{i-1}} + \frac{1}{\frac{da}{dN}_i} \right) \Delta a_i$$

Equation 201 Fatigue crack growth from the previous increment, i

Since the number of fatigue cycles ΔN for each increment must be consistent for both the crack length and crack depth, the average (an equal weighting from both crack positions) is used to compute ΔN .

$$\Delta N = \frac{1}{2} (\Delta N(c)_i + \Delta N(a)_i)$$

$$\Delta N = \frac{1}{2} \left(\frac{1}{2} \left(\frac{1}{\frac{dc}{dN}_{i-1}} + \frac{1}{\frac{dc}{dN}_i} \right) \Delta c_i + \frac{1}{2} \left(\frac{1}{\frac{da}{dN}_{i-1}} + \frac{1}{\frac{da}{dN}_i} \right) \Delta a_i \right)$$

$$\Delta N = \frac{1}{4} \left(\frac{1}{\frac{dc}{dN}_{i-1}} + \frac{1}{\frac{dc}{dN}_i} \right) \Delta c_i + \frac{1}{4} \left(\frac{1}{\frac{da}{dN}_{i-1}} + \frac{1}{\frac{da}{dN}_i} \right) \Delta a_i$$

Equation 202 Average number of cycles

In the fatigue analysis one of the crack increment sizes must be selected as an input value. For example, the crack depth increment size Δa is given a constant value, and then the Δc increment size needs to be computed in order to compute ΔN . Since the number of cycles must be consistent along the crack, the two fatigue equations are set equal to each other to solve for Δc . The Δc value is then computed using the ratio of the crack growth rate values.

$$\Delta N = \frac{\Delta c}{\Delta f_{\phi=0}} = \frac{\Delta a}{\Delta f_{\phi=\pi/2}}$$

$$\Delta c = \Delta a \frac{\Delta f_{\phi=0}}{\Delta f_{\phi=\pi/2}}$$

$$\Delta c_i = \Delta a_i \frac{\left(\frac{dc}{dN} \right)_{i-1}}{\left(\frac{da}{dN} \right)_{i-1}}$$

Equation 203 Given the depth increment, a , compute the length increment, c

The crack length is updated from the computed increment:

$$2c_i = 2c_{i-1} + 2\Delta c_i$$

Equation 204 Update crack length

An underlying assumption is that the ratio of the crack growth rates does not change significantly from the previous increment to the current increment. As is usual in numerical analysis, the increment size is important for accuracy. A sufficiently small Δa increment size should be used to get accurate fatigue results.

For a crack mesh with the full surface crack profile, for example the back-half symmetry or full model cases, the crack growth rates and Δc increment at each free surface (each end of the crack) are used to update the fatigue crack growth values. The full crack length $2c$ is updated with values from each crack tip at $\phi = 0$ and π .

$$\Delta c(\phi = 0)_i = \Delta a_i \frac{\left(\frac{dc(\phi = 0)}{dN}\right)_{i-1}}{\left(\frac{da}{dN}\right)_{i-1}}$$

$$\Delta c(\phi = \pi)_i = \Delta a_i \frac{\left(\frac{dc(\phi = \pi)}{dN}\right)_{i-1}}{\left(\frac{da}{dN}\right)_{i-1}}$$

Equation 205 Crack length contributions from both crack tips

Use the crack length increment at each end of the crack front to update the total length:

$$2c_i = 2c_{i-1} + \Delta c(\phi = 0)_i + \Delta c(\phi = \pi)_i$$

Equation 206 Update total crack length

The number of fatigue cycles is computed using the crack growth rates at three positions on the surface crack: the two crack tips and the crack depth.

$$\Delta N = \frac{1}{8} \left(\frac{1}{\frac{dc(\phi = 0)}{dN}_{i-1}} + \frac{1}{\frac{dc(\phi = 0)}{dN}_i} \right) \Delta c(\phi = 0)_i + \frac{1}{8} \left(\frac{1}{\frac{dc(\phi = \pi)}{dN}_{i-1}} + \frac{1}{\frac{dc(\phi = \pi)}{dN}_i} \right) \Delta c(\phi = \pi)_i + \frac{1}{4} \left(\frac{1}{\frac{da}{dN}_{i-1}} + \frac{1}{\frac{da}{dN}_i} \right) \Delta a_i$$

Equation 207 Update number of cycles in the increment

Fatigue Equations References

1. Anderson, Ted L., Fracture Mechanics, Fundamentals and Applications, 3rd ed., CRC Press Taylor and Francis Group, 2005; equations 10.1 and 10.5, p. 453
2. Kreyszig, Erwin, Advanced Engineering Mathematics, 6th ed., John Wiley and Sons, Inc., 1988, p. 979.

Fatigue Results

At the conclusion of a fatigue analysis, the number of cycles will be displayed. Click on the Results tab to view the detailed output in both graphical and tabular form. You should save the current *.elt model file to preserve these results. Click on the “Show Variable Selections” to customize the fatigue result plots. On the main graphics window, you can also view the meshes that were saved during the analysis; select a mesh file using the menu at the top of the graphics window.

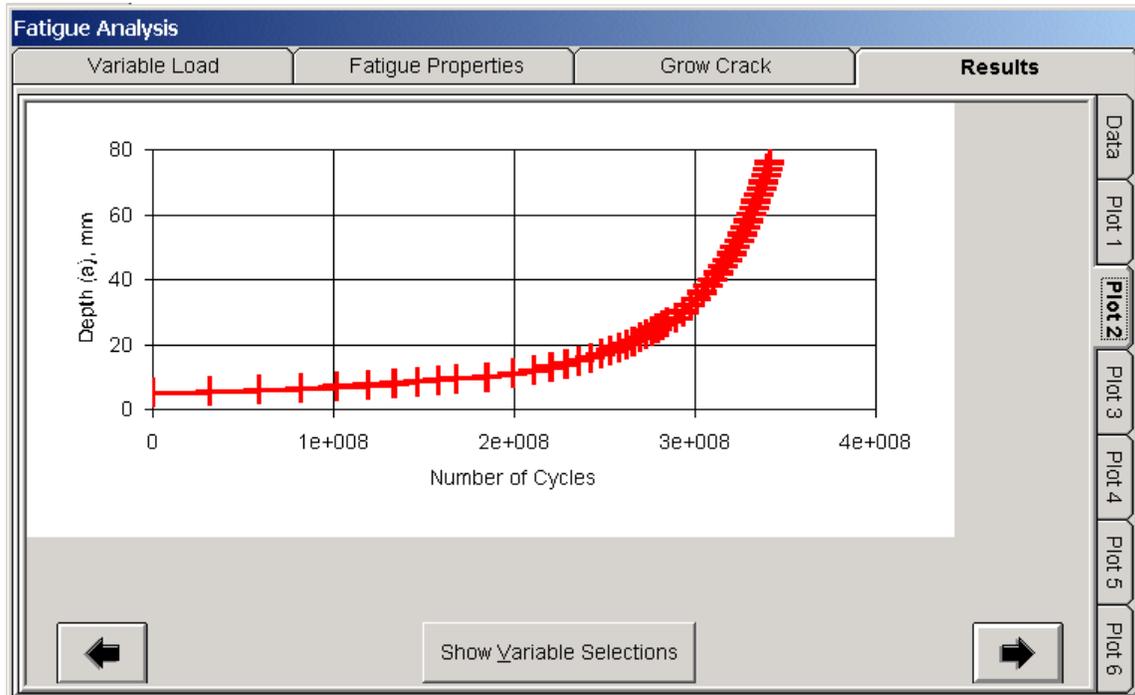


Figure 208 Crack depth versus cycles fatigue results

Fatigue Validation: Semi-Circular Crack

Fatigue analysis results for a surface crack in a flat plate can be compared to an approximate closed-form solution of a semi-circular surface crack in a semi-infinite flat plate by setting the crack length equal to the crack depth: $c = a$. See the figure and equations below.

Fatigue Validation: Approximate Closed-Form Solution

To begin, the semi-elliptical surface crack solution in a semi-infinite flat plate is given by [1]:

$$K_I = \lambda_s \sigma \sqrt{\frac{\pi a}{Q}} f(\phi)$$

Equation 209 **K solution for semi-elliptical surface crack**

Where K_I is the stress intensity factor along the crack front, λ_s and $f(\phi)$ are geometry shape factors, σ is the applied uniform membrane stress, a is the crack depth, and Q is the semi-elliptical shape factor, each given by these subsidiary equations:

$$Q = 1 + 1.464 \left(\frac{a}{c}\right)^{1.65} \quad (\text{for } a \leq c)$$

$$\lambda_s = \left[1.13 - 0.09 \left(\frac{a}{c}\right)\right] [1 + 0.1(1 - \sin\phi)^2]$$

$$f(\phi) = \left[\sin^2(\phi) + \left(\frac{a}{c}\right)^2 \cos^2(\phi)\right]^{1/4}$$

Equation 210 **K solution geometry shape factors**

Following the method in Example 10.1 [2], the K_I equation is simplified for a semi-circular crack by setting $c = a$, so that the ratio $a/c = 1$. The equations simplify to the following values:

$$Q = 1 + 1.464 \left(\frac{a}{c}\right)^{1.65} = 1 + 1.464 = 2.464$$

$$\lambda_s = \left[1.13 - 0.09 \left(\frac{a}{c}\right)\right] [1 + 0.1(1 - \sin\phi)^2] = 1.04[1 + 0.1(1 - \sin\phi)^2]$$

Equation 211 **Simplify for a semi-circular crack**

At the free surface crack tip, $\phi = 0$, and at the crack depth, $\phi = \pi/2$, giving two possible values for λ_s :

$$\lambda_s = 1.04[1 + 0.1] = 1.144 \quad (\phi = 0)$$

$$\lambda_s = 1.04[1 + 0] = 1.04 \quad (\phi = \pi/2)$$

Equation 212 **Simplified geometry factor, λ_s**

At either the free surface or the crack depth for the semi-circular crack, the $f(\phi)$ equation reduces to a value of 1:

$$f(\phi = 0) = 1$$

$$f(\phi = \pi/2) = 1$$

Equation 213 **Simplified geometry factor, f**

Substituting these simplified values for the semi-circular crack the stress intensity equation becomes, where λ_s is a constant value:

$$K_I = \lambda_s \sigma \sqrt{\frac{\pi a}{2.464}}$$

Equation 214 Simplified K solution for semi-circular crack

Fatigue Validation: Paris Equation

The Paris equation is used to describe the fatigue crack growth:

$$\frac{da}{dN} = C \Delta K^m$$

Equation 215 Paris Fatigue Equation

Where a is the crack depth, N is the number of fatigue cycles, $\frac{da}{dN}$ is the crack growth rate, and C and m are material constants. Substituting the semi-circular stress intensity equation into the Paris fatigue equation gives:

$$\frac{da}{dN} = C \left(\lambda_s \Delta \sigma \sqrt{\frac{\pi a}{2.464}} \right)^m$$

Equation 216 Substitute semi-circular K solution

Where $\Delta \sigma$ is the cyclic stress. Grouping the constant terms together and arranging in terms of the number of cycles gives an equation that can be directly integrated to a closed-form solution:

$$\begin{aligned} \frac{da}{dN} &= C \left(\lambda_s \Delta \sigma \sqrt{\frac{\pi}{2.464}} \right)^m a^{m/2} \\ dN &= \left(\frac{1}{C \left(\lambda_s \Delta \sigma \sqrt{\frac{\pi}{2.464}} \right)^m} \right) a^{-m/2} da \\ N &= \left(\frac{1}{C \left(\lambda_s \Delta \sigma \sqrt{\frac{\pi}{2.464}} \right)^m} \right) \int_{a_0}^{a_f} a^{-m/2} da \\ N &= \left(\frac{1}{C \left(\lambda_s \Delta \sigma \sqrt{\frac{\pi}{2.464}} \right)^m} \right) \frac{[a_f^{1-m/2} - a_0^{1-m/2}]}{(1 - m/2)} \end{aligned}$$

Equation 217 Closed-form solution for the number of fatigue cycles, semi-circular crack example

Fatigue Validation: Comparison Example

An *FEACrack* flat plate with an initial semi-circle surface crack is used to compare the fatigue results to the approximate closed-form solution. The values used in this example are listed in the table below. The plate dimensions are chosen so that the crack is much smaller than the plate to approximate the surface crack in a semi-infinite volume.

Parameter	Value
Initial crack depth, a_0	0.1 in
Final crack depth, a_f	0.5 in
Initial crack length, $2c$, set $c = a$	0.2 in
Plate half width, W	10 in
Plate thickness, T	5 in
Cyclic stress, $\Delta\sigma$	1 ksi
Fatigue, C coef.	1×10^{-9}
Fatigue, m exponent	3

Table 218 Parameters for Plate Surface Crack Fatigue Example

The quarter symmetric initial and final crack meshes are shown in the figures below. Note that the crack is small compared to the size of the plate.

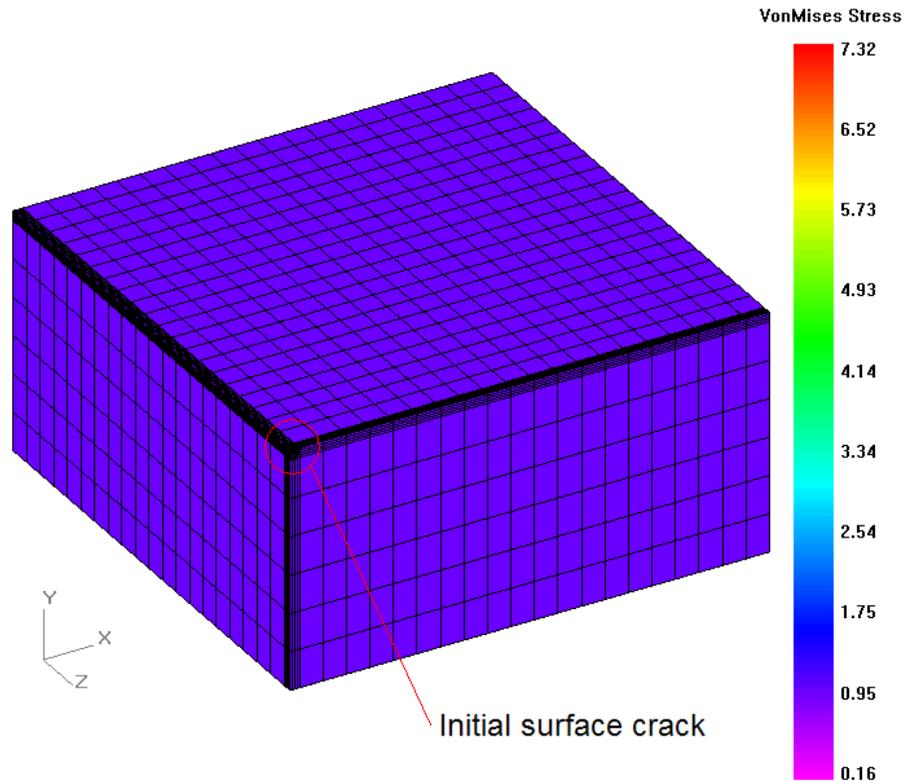


Figure 219 Initial semi-circular surface crack; quarter symmetric plate mesh

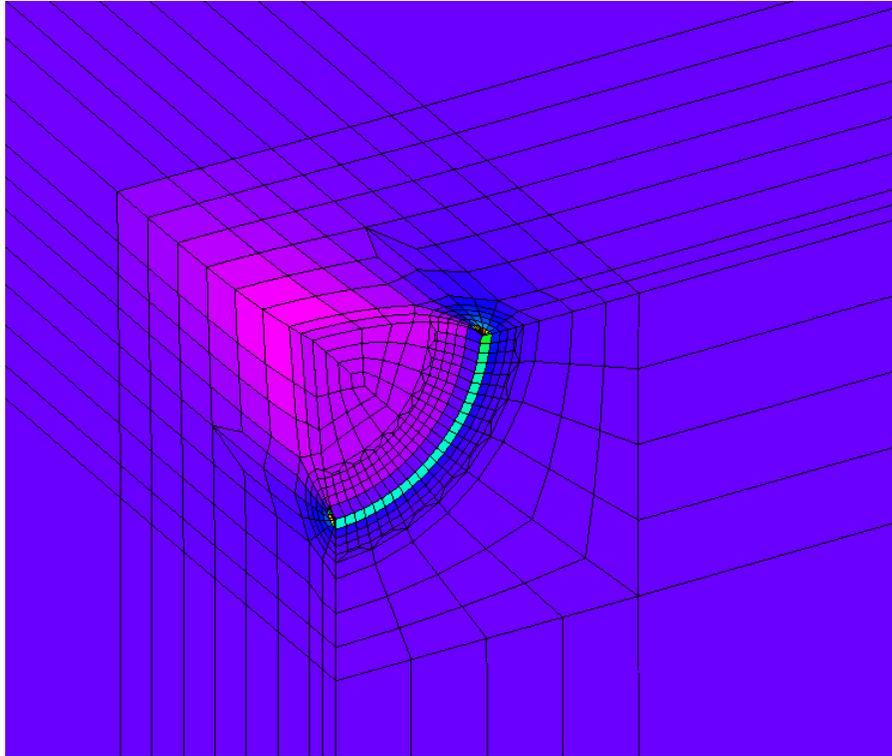


Figure 220 Close up of the initial semi-circular surface crack

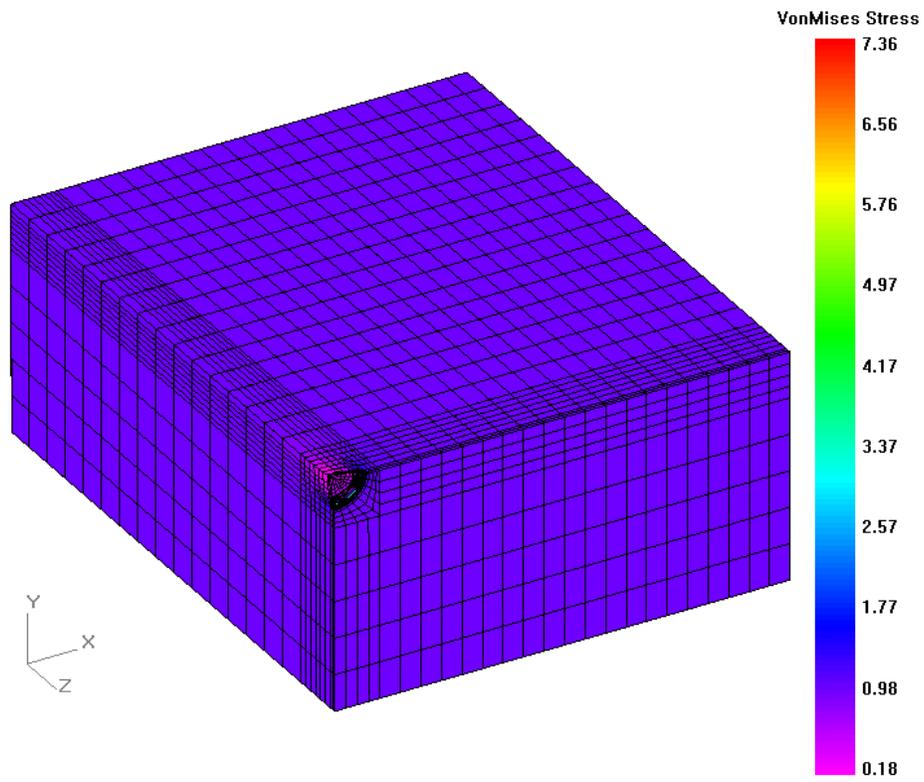


Figure 221 Final surface crack mesh after fatigue crack growth

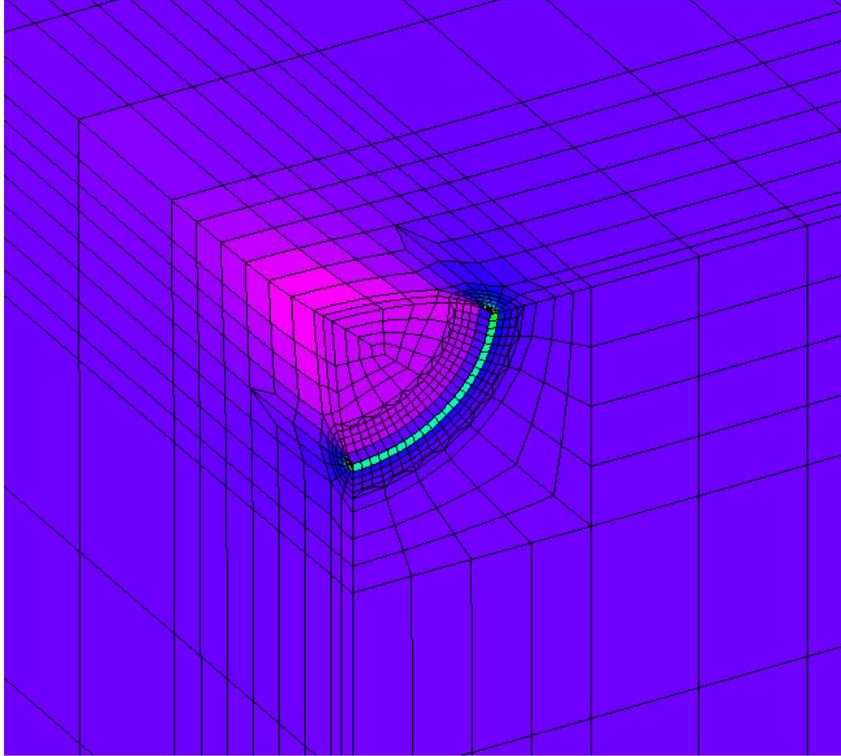


Figure 222 Close up of final surface crack mesh

The crack depth and crack length results versus the number of computed fatigue cycles are shown in the figures below. The fatigue results are computed by FEACrack, Signal Fitness-for-Service™ (a Quest Integrity Group fracture mechanics software using a library of K solutions), and the semi-circular closed-form solution. Since the crack shape is kept semi-circular in the closed-form solution, there are three possibilities for the number of fatigue cycles N : use $\lambda_s = 1.144$ at the crack tip, or use $\lambda_s = 1.04$ at the crack depth, or use the average as a general crack front value $\lambda_{s, \text{avg}} = 1.092$. In the crack depth plot, the crack tip and crack depth λ_s curves give an upper and lower bound to the crack depth fatigue growth. It is observed that the initially semi-circular crack becomes more elliptical in the FEACrack and Signal calculations; the crack has no condition applied or requirement to maintain the semi-circular crack shape. These crack depth values compare most closely with the average $\lambda_{s, \text{avg}}$ value, which perhaps better represents the approximate nature of the semi-circular closed-form solution along the entire crack front. In the crack length results plot, the curves show the crack becoming more elliptical compared to the semi-circular closed-form results; the crack length growth rate is greater at the free surface than at the crack depth.

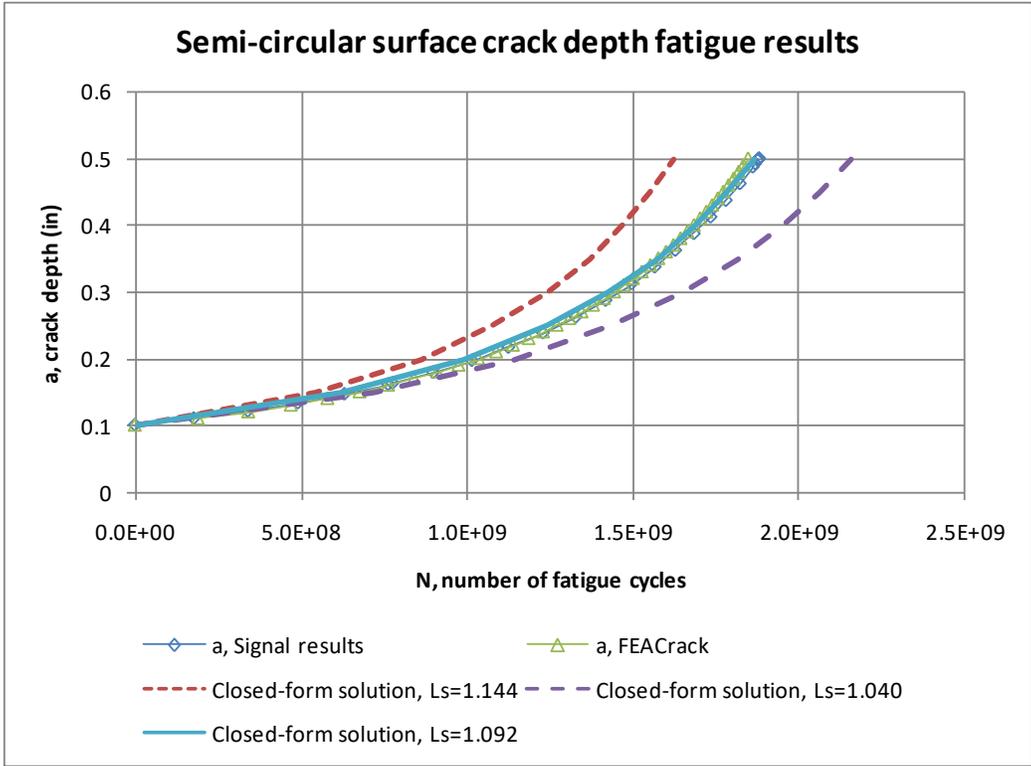


Figure 223 Crack depth fatigue results, semi-circular example

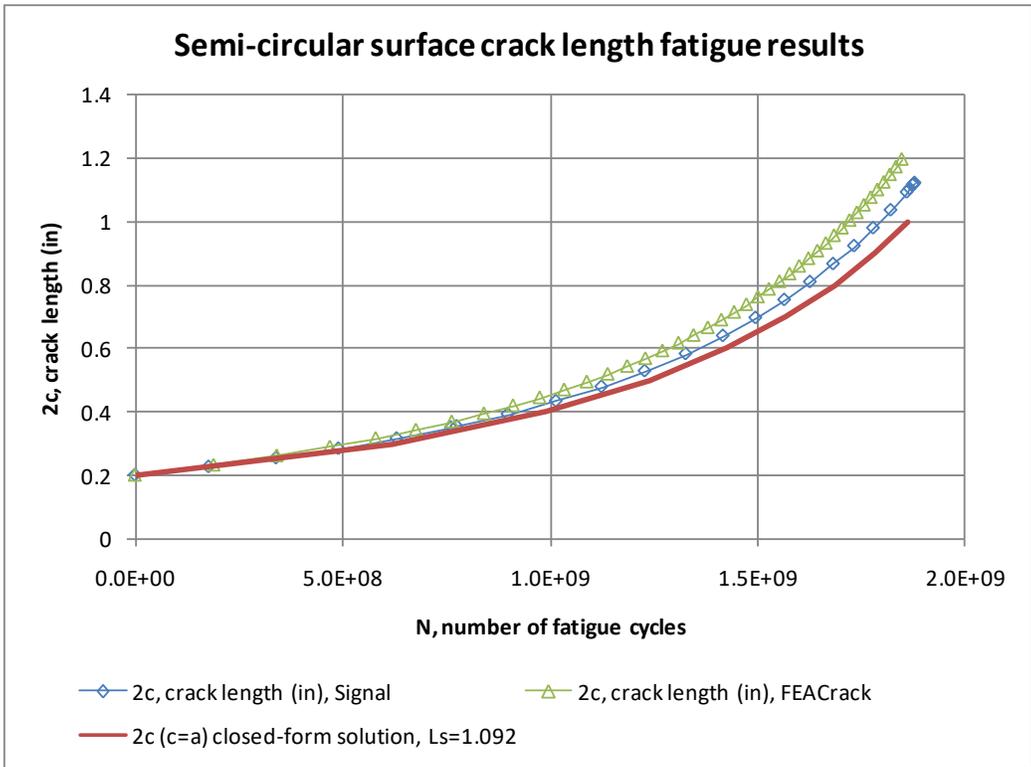


Figure 224 Crack length fatigue results, semi-circular example

Initial Stress Interpolation

For interpolation of stress values from a previous analysis to a new crack mesh, select Show Stress Interpolation in the Tools menu in *FEACrack*. This method allows initial stresses to be applied to a crack mesh that vary point by point through the mesh.

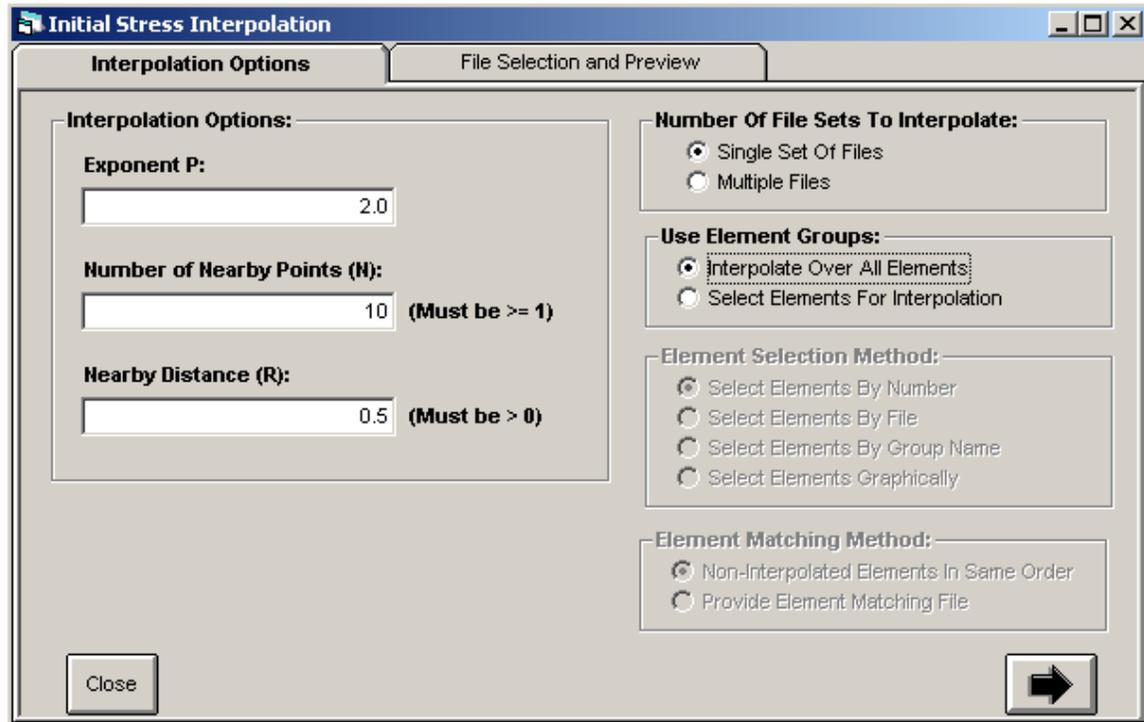


Figure 225 Stress interpolation parameters

On the Interpolation Options window set the interpolation parameters and processing options. The exponent P gives the interpolation weighting versus distance. The typical value is 2.0 for an inverse distance squared weighting in the interpolation. To limit the number of nearby points used to interpolate the stress values, set a small number of nearby points N ; a typical value is 10 to 20 points. To limit the size of the interpolation region around each point, set the nearby distance R . The minimum required distance R can be estimated from half the longest element edge length from the previous model in the element interpolation region. A smaller distance R will include fewer points in the interpolation calculation and speed up the time elapsed to complete the interpolation calculations; a larger distance R will include more points and also increase the total time for the interpolation. The nearest N points within distance R will be used to compute the interpolated stress values at each element integration point in the new crack mesh. If the distance R is too small no nearby points will be found for some of the element integration points in the new crack mesh, and the interpolation will stop and exit with an error message.

A single set of files can be used for an interpolation of stress values, or multiple sets of files can be selected to compute the interpolation of stress values in batch processing mode. When Multiple Files is selected, the files are selected in the Multiple File Selection window. If a single set of files will be used, the files are selected in the File Selection and Preview window.

The stress interpolation can be computed for all elements in the new crack mesh or for a selected sub region. To select a group of elements as the sub region for stress interpolation choose Select Elements By File. Separate input data files will be used to give the element numbers for each element group in the previous model and new crack mesh. The elements outside the interpolation sub-region will have their stress values reused. Re-using stress values requires that the region outside the element groups have the same mesh. The number of elements and the node coordinates must be the same from the previous model to the new crack mesh. The elements can be renumbered from the previous model to the new crack mesh to accommodate the different number of elements and nodes in the interpolation sub region in the new crack mesh (where the crack is located). If the renumbered elements are in the same order, the change in element number from the previous mesh to the new crack mesh is automatically detected within the program. The element numbers in the new crack mesh are used when writing the reused stress values to the new initial stress file.

In the Multiple File Selection window, start with the number of sets at 1. Select the files for the previous model input file, as well as the previous model stress file, the new crack mesh input file, and set a name for the new crack mesh stress file. Select the element group input files, if the interpolation will be computed for a sub region of the mesh. As described above, an input data file is used to list the elements for the previous model element group and the new crack mesh element group. These element groups must be in the same region of the model. For example, the previous model element group file could be the definition mesh for a user defined geometry, and the new crack mesh element group file could be the user defined crack mesh generated by *FEACrack*. To automatically fill in the file selections for more interpolation sets, change the number of sets to the desired value.

For example, changing the number of sets from 1 to 8 will add more file selection boxes for the additional sets and automatically fill in the file choices. Modify any of the file choices that change for each set; usually the new crack mesh file, the new crack mesh stress file, and the new crack mesh element group file will change for each interpolation set. To view the mesh and stress data for a particular set of files, click on the Preview Files button for that set. That set of files will be filled in the File Selection and Preview window. When files for all the sets have been selected and verified, click on the Interpolate Batch button at the bottom of the window to begin the batch interpolation processing.

Every set of files will have the stress values interpolated from the previous model to the new crack mesh; any set with an error will be reported after all the sets are complete. Common errors are due to having element integration points in the new crack mesh not finding any nearby data points from the previous model. The problem can usually be corrected by adjusting the nearby distance *R* to a larger value.

In the File Selection and Preview window the mesh and stress files can be viewed to verify the file selection and element group data. Click the check box next to a file selection to view the mesh or the stress data. When element groups have been defined, the elements in the group are shown in yellow for the previous mesh and new crack mesh files. Use the Result Type menu to select the stress component to view for the previous model stress file, or for the new crack mesh stress file after the interpolation calculations are complete.

In *Abaqus* the initial stress values are applied using the **INITIAL CONDITIONS* command. This command must be added to the crack mesh input file before running the analysis. Typically this command is added just before the first **STEP* command in the input file. For initial stress values applied at the element centroids, the command is: **INITIAL CONDITIONS*,

TYPE=STRESS. Copy the initial stress values from the data file created the stress interpolation into the input file. For stress values applied at the element Gauss integration points the command includes the USER parameter and is given as: *INITIAL CONDITIONS, TYPE=STRESS, USER. Include the Fortran user subroutine created by the stress interpolation in the analysis.

In ANSYS the initial stress values can be read from an **.ist** initial stress file using ISFILE, READ, <file name>, ist, , 2,. When running the un-cracked model use the ISWRITE, ON command in the analysis step to export the stresses for the un-cracked mesh to an initial stress file; this is the file that will be read to obtain the stresses for interpolation onto the crack mesh. ANSYS does not support mixed element types in the initial stress file, and the collapsed brick elements at the crack front are converted within ANSYS to wedge elements. To avoid this element type problem when using the ANSYS initial stress file, use a small radius key hole at the crack front so that all the elements in the crack mesh remain brick elements.

Stress Interpolation During Fatigue

The Inverse Distance Weighted (IDW) interpolation can be used during a fatigue analysis to apply initial stress to each crack mesh. The figure below shows the Stress Interpolation side tab in the Fatigue/Grow Crack form. To activate the stress interpolation during fatigue, check the “Perform Stress Interpolation ...” option box. This form is similar to the Tools/Show Stress Interpolation form, except that the “new crack” mesh is not selected. The new crack mesh name is known, since it is the next input file in the fatigue analysis. Enter the interpolation parameters, and select the previous analysis mesh and stress files. Also select a file name for the initial stress file.

For ANSYS, the initial stress file has a *.ist file extension and will contain the stress tensor at each crack mesh element integration point. Also for ANSYS, the ISFILE command must be included in the crack mesn input file so that the new stress *.ist file is used during the analysis. Include the appropriate ISFILE syntax in the Tools/Model Options/Macros form. Since ANSYS does not support mixed element types for the initial stress file, the small radius key hole option must also be used so that the crack front elements remain bricks rather than getting converted by ANSYS to wedge elements. The key hole option is in the Tools/Model Options/Crack Analysis Params form.

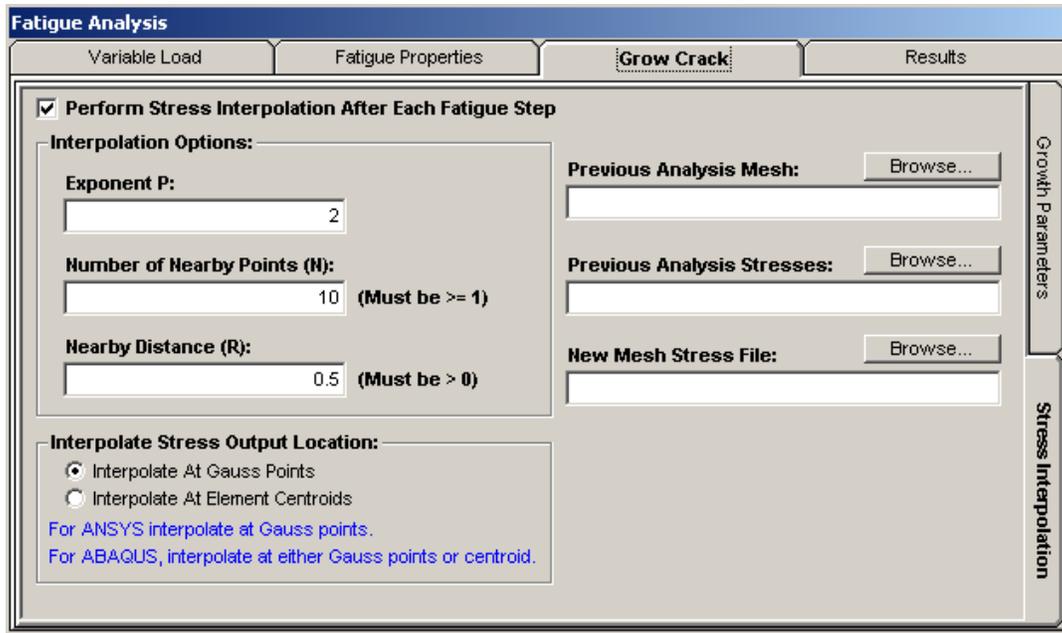


Figure 226 Set the interpolation parameters and select the file names for the stress interpolation

It is recommended that a single stress interpolation be run from the Tools/Show Stress Interpolation form before starting the fatigue analysis to check the parameters, files, and time needed for the interpolation calculations. If the parameters and files have been selected in the Tools/Show Stress Interpolation form, the same values and files will be filled into the Grow Crack/Stress Interpolation form.

DLL for Fatigue with a Custom Crack

A custom crack front can be used for fatigue analysis with a user-defined geometry in *FEACrack* by providing an updated external DLL module. For a standard geometry, a cylinder for example, the custom crack front reverse transformations are automatically done during a fatigue analysis. The reverse transformation calculation converts the updated crack front node global coordinates back to the crack front c,a coordinates to generate the next mesh.

For a user-defined geometry, the reverse transformation calculations must be provided. The default UserReverseTransform.dll in the *FEACrack* install directory must be replaced by an updated version that provides the necessary crack front reverse transformation. To activate use of the DLL during the fatigue analysis, check the “Use custom growth DLL” box on the Fatigue Analysis/Grow Crack form (circled in red in the figure below).

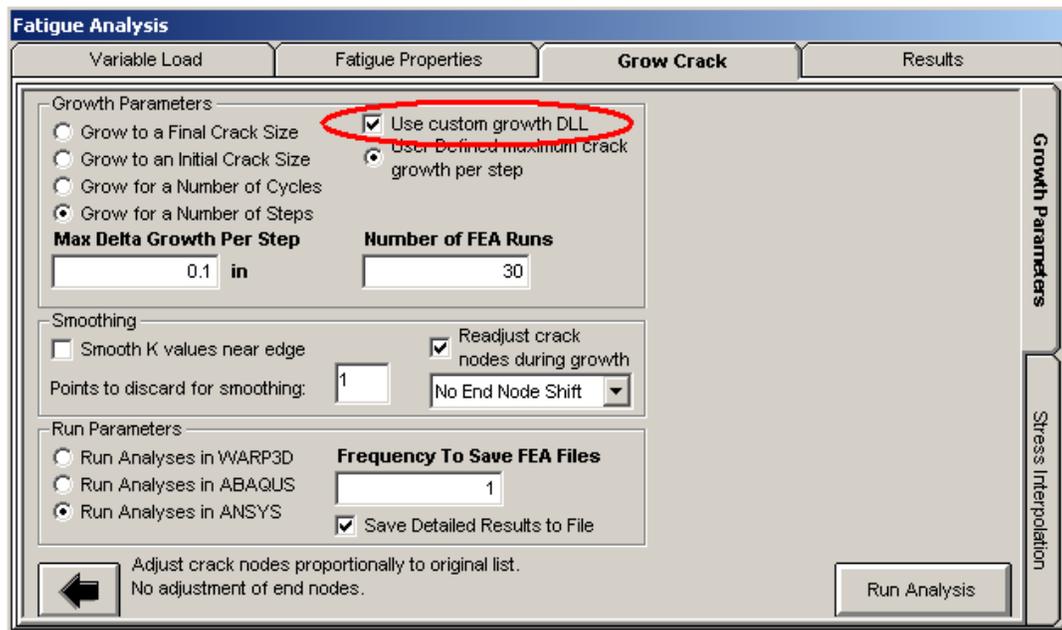


Figure 227 Activate the external DLL module for custom crack fatigue analysis in a user-geometry

Cylinder Reverse Transformation Example

Using a cylinder with a circumferential crack as an example, the custom crack front global coordinates must be related to the crack front c , a coordinates by reverse transformation equations. For an internal circumferential custom crack the crack depth is from the inside surface to the crack front, and the crack length is the circular arc length along the cylinder to the crack front node. The figure below shows an example of the custom crack. For a crack in the global y, z plane, the radius from the cylinder centerline to a crack front node, R_n , is:

$$R_n = \sqrt{y_n^2 + z_n^2}$$

And the angle θ around the cylinder is computed by the inverse cosine function so that the angle value is from 0 to π :

$$\theta = \cos^{-1} \left(\frac{y_n}{R_n} \right)$$

The custom crack front c , a locations are then given from the cylinder inside radius, R_{in} , the radius to the node, R_n , and the angle θ :

$$c = R_{in} \theta$$

$$a = R_{in} - R_n$$

Note that the crack length c is positive along the crack front, but the crack depth a is a negative value to match the expected input for a custom surface crack in the Geometry/Cracks/Dimensions form.

An example of the Fortran subroutine for this particular reverse transformation is given in the Appendix.

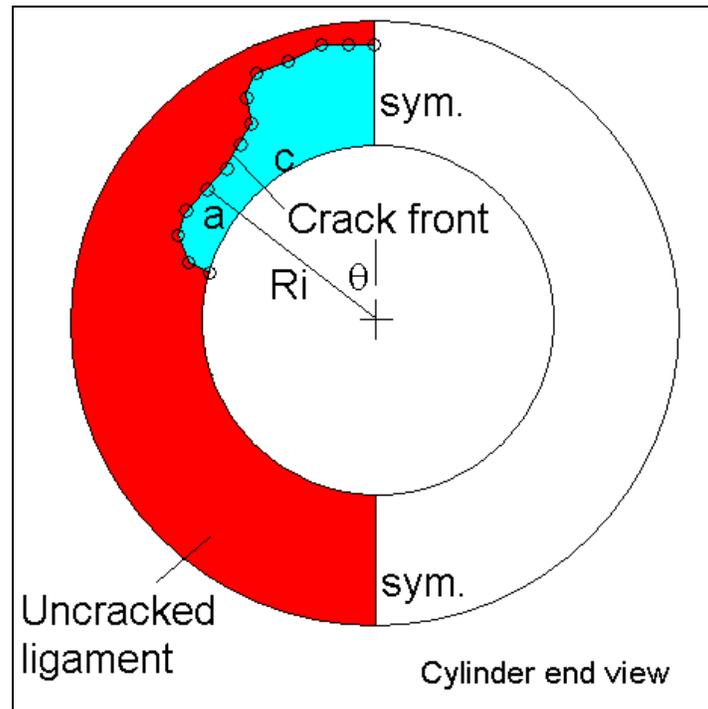


Figure 228 Cylinder custom crack reverse transformation calculation example; the crack depth a is radial, the crack length c is the arc length

Abaqus Part and Assembly Introduction

In an *Abaqus* input file, each component can be described as a distinct part by using the “part and assembly” syntax. The parts can be used one or more times by specifying an instance for each part’s use. The entire model is the assembly of the part instances. Refer to the *Abaqus* user’s manual and the *Abaqus* /CAE manual for more information. In *FEACrack* the crack mesh can be written to an *Abaqus* input file that has the part and instance syntax added to the mesh data and commands. Select the option from the Tools/Program Options form in the “Type of *Abaqus* File To Write” pull-down menu; see the figure below.

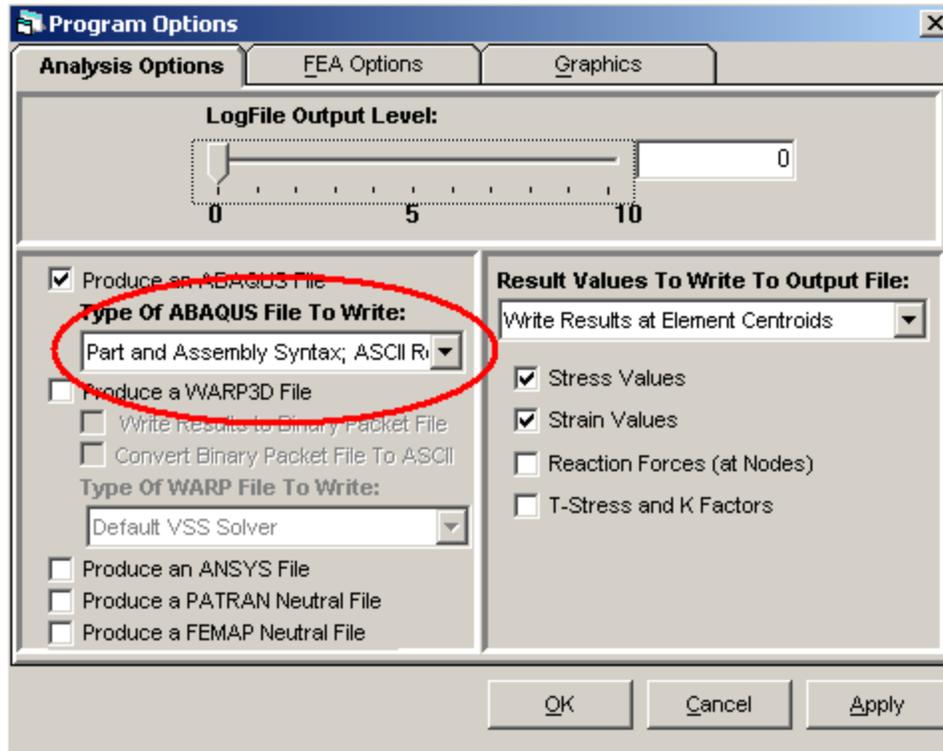


Figure 229 Select the Abaqus part and assembly syntax option

The part and assembly syntax is usually used with the user-defined geometry crack mesh to combine the crack mesh with the larger surrounding model. Using the crack mesh as a part permits both the crack mesh and larger model to have overlapping node and element numbering. For example, the nodes in each part can be numbered starting at 1. *Abaqus* rennumbers the parts to a global numbering during pre processing; the global numbering is used to report the results to the *.fil result file. A local to global renumbering map for the nodes and elements is listed in the *.dat *Abaqus* output data file, and is read during post processing in *FEACrack*. The *Abaqus* input files for the crack mesh and user-defined geometry can be combined using a text editor (Notepad or Wordpad) or *Abaqus* /CAE.

The part and assembly syntax is not needed if the crack mesh and larger model are numbered sequentially to avoid overlapping node and element ID numbers. With sequentially numbered mesh regions, the combined model can use just the standard commands in the *Abaqus* input file.

The crack mesh is defined as a part using the *PART command in the *Abaqus* input file. To add the crack mesh to the model, use the *ASSEMBLY command, followed by the *INSTANCE command to reference the crack mesh part.

For *FEACrack* post processing, use the word “crack” as part of the crack mesh instance label, and avoid the word “crack” as part of any non-crack instance label. The “crack” key word will be part of the local to global renumbering map listing that is read from the *.dat output file. This requirement can be done automatically by including the word “crack” in the *FEACrack* *.elt model file name; the model file name is used to form the default part and instance labels.

The subset mesh option can also be used with the part and assembly syntax. The subset mesh node and element sets are defined in the crack mesh part, and referenced in the output commands.

When post processing a crack mesh using *FEACrack*, the *Abaqus* input (*.inp), output (*.dat), and result (*.fil) files must be present in the same directory so that the crack data blocks and global renumber map can be obtained.

Part and Assembly Syntax Summary

The *Abaqus* part, instance, and assembly commands are listed below as a summary of the type of data given within each command. Also refer to the *Abaqus* user's manual for more details.

Define a part, such as the crack mesh or larger model:

*PART, NAME=part

- Node coordinates
- Element connectivity
- Section properties
- Parts must end with *END PART

Define and instance of a part:

*INSTANCE, NAME=instance, PART=part

- Node sets
- Element sets
- Surface definitions
- An instance must contain one and only one unique part; nodes and elements referenced in the instance are unique to that instance's part
- Instances are defined within *ASSEMBLY
- Instances must end with *END INSTANCE

Define the entire model as the assembly, reference all the instances to be included in the model:

*ASSEMBLY, NAME=assembly

- All instances
- *TIE command
- Defined in model data
- References to nodes, elements, sets must follow *assembly.instance.ID* naming syntax
- One assembly per model
- Assembly ends with *END ASSEMBLY

Analysis step data; the boundary conditions and output commands can reference node and elements sets from a particular instance.

*STEP

- Output commands
- Contour commands
- Loads
- Boundary conditions
- References to nodes, elements, sets must follow *assembly.instance.ID* naming syntax

Material properties, initial conditions, amplitude tables, boundary conditions and interactions appear outside the *ASSEMBLY command

J-integral node set names in the *CONTOUR command must also follow *assembly.instance.nodeset* naming syntax. Since the combined label can become long, just one node set is written per line to the input file.

Radial Mesh Lines Option for PRO mesh

The PRO focused mesh allows the user to control many of the crack mesh parameters and mesh refinement. The default PRO mesh does not enforce that the radial mesh lines from the crack front in the crack tube remain perpendicular (normal) to the crack front. To have the crack tube radial mesh lines remain normal to the crack front, use the “Perpendicular Radial Mesh Lines in Crack Tube” option in the Tools/Model Options/Crack Mesh Params. The option check box is circled in red in the figure below.

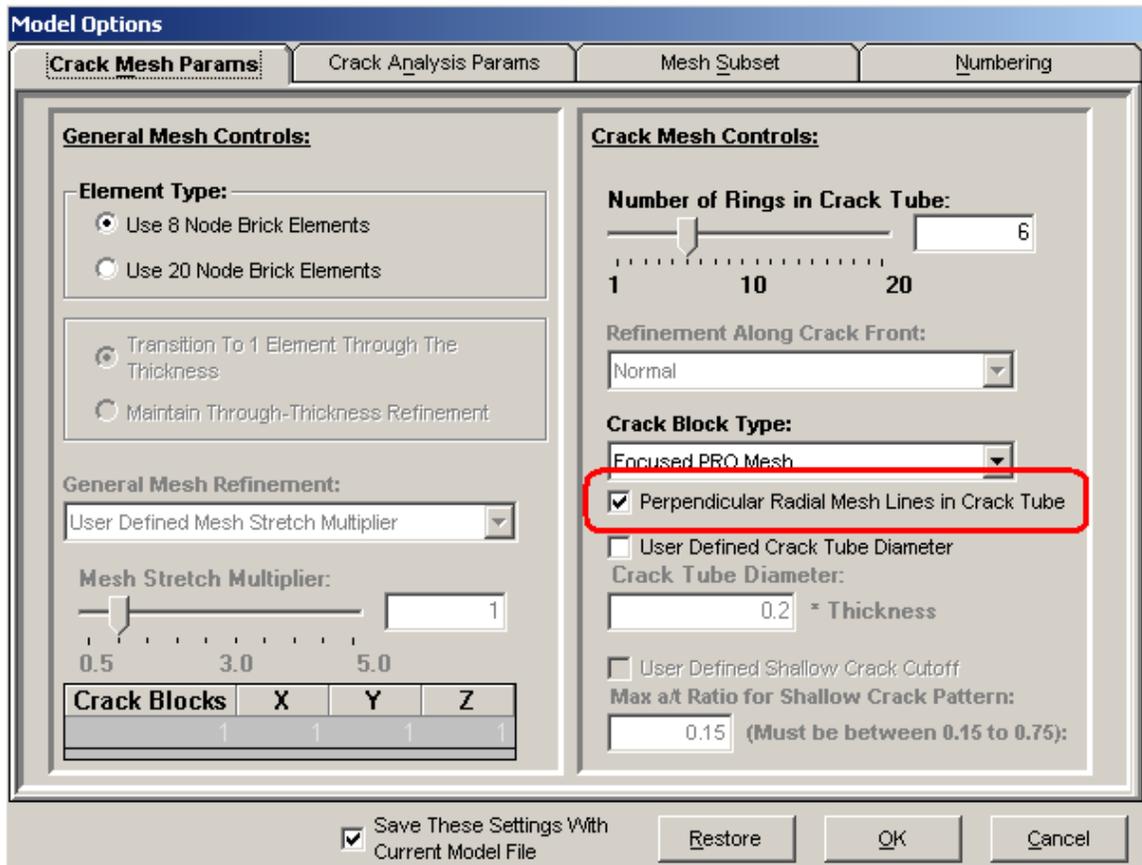


Figure 230 Select the “Focused PRO Mesh” and then check the “Perpendicular Radial Mesh Lines in Crack Tube” option

If there are just 1 or no crack front transition zones, the normal radial mesh lines are automatically extended past the crack tube and into the mesh box zone and the mesh adjustment zone. If there are 2 or more crack front transition zones, the radial mesh lines outside the crack

tube are adjusted to accommodate the additional crack front transition elements. The figures below compare the extended radial mesh lines, the mesh with a second crack front transition zone, and the standard PRO mesh without the radial mesh lines.

Depending on the crack dimensions the crack tube radius can be limited when the normal radial mesh lines are used. If the current crack tube radius is too large, the maximum allowed crack tube radius is automatically computed and used to generate the PRO crack mesh.

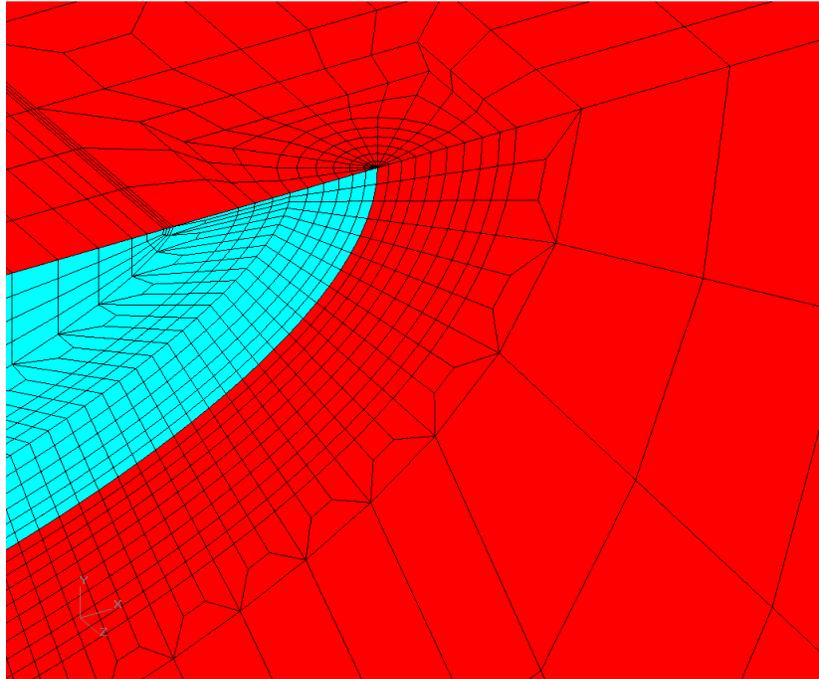


Figure 231 Normal radial mesh lines extend past the crack tube ahead of the crack.

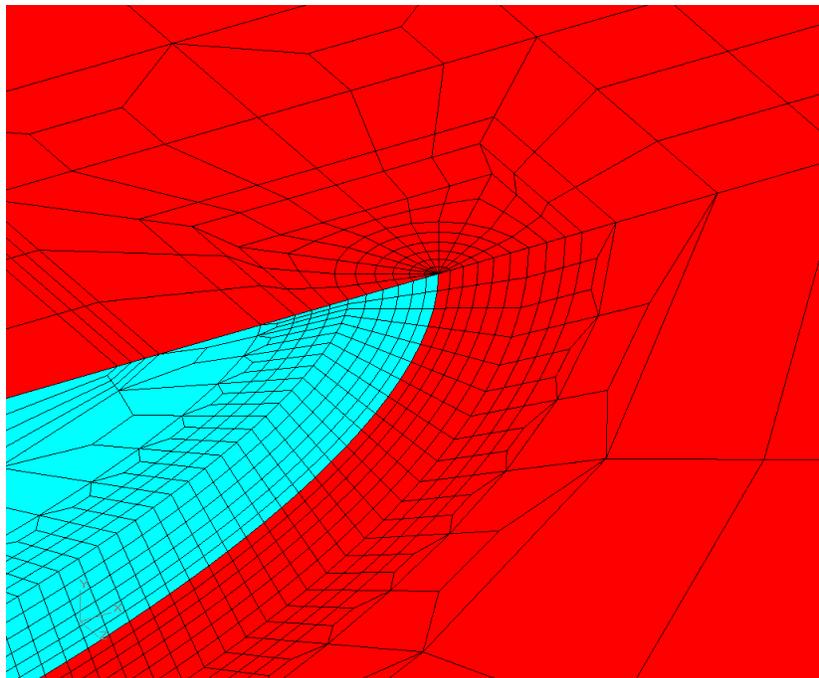


Figure 232 Normal radial mesh lines in the crack tube only with two crack front transition zones.

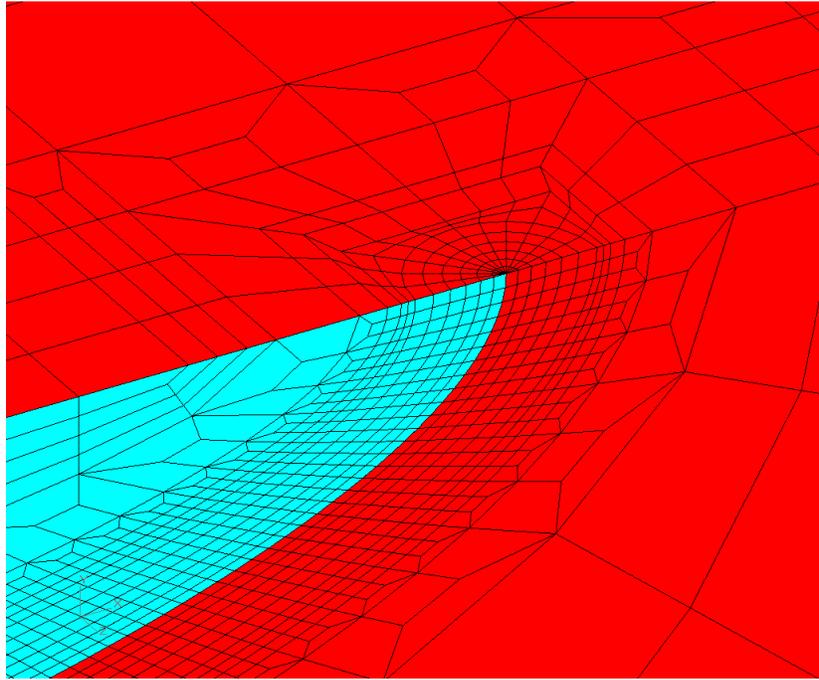


Figure 233 Standard PRO mesh, the radial mesh lines do not remain normal to the crack front.

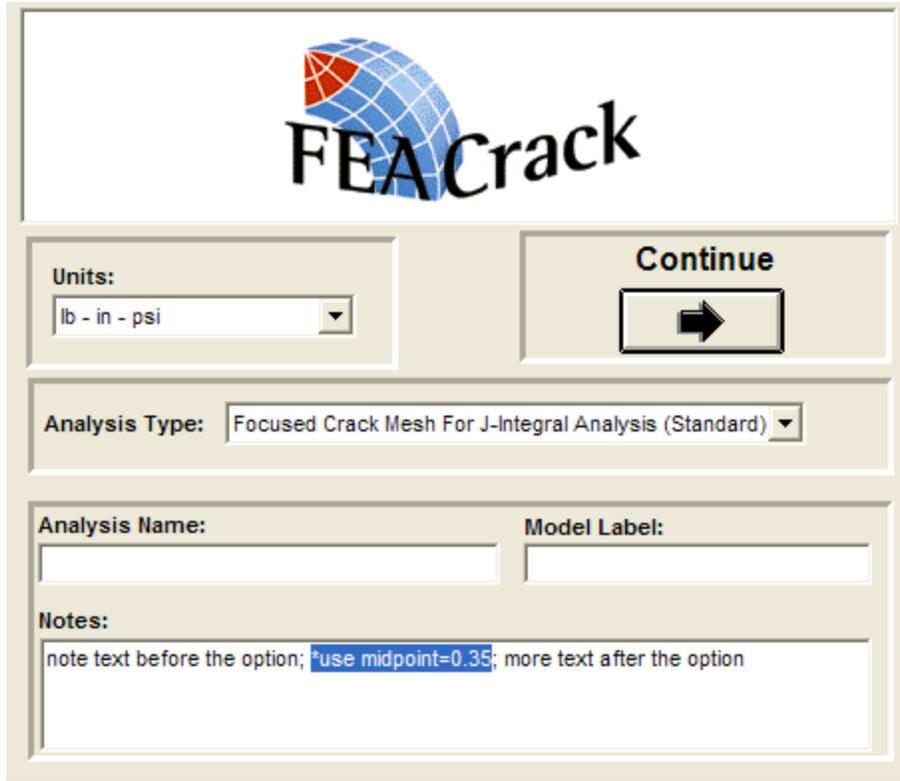
Mid Side Node Position

In an elastic analysis the crack front collapsed elements typically have their mid side nodes located at the quarter point position to give the desired $1/\sqrt{R}$ strain singularity in the collapsed element. An option is available to adjust the default mid side node quarter point position.

Enter the key word and value in the Home form Notes text box; see the highlighted text shown in the picture below. The key word text begins with an asterisk "*", followed by "use midpoint". The equal sign "=" separates the key word from the position value. A semi-colon ";" can be used after the value to separate the option from other text in the Notes. For example, the key word for a position value of 0.35 is:

```
*use midpoint = 0.35
```

The position value must be greater than zero and less than one. The recommended position value is from 0.25 to 0.5. To deactivate this option just remove the key word and value from the Notes.



The image shows the FEACrack software interface. At the top is the logo, which consists of a blue and red globe icon followed by the text "FEACrack". Below the logo is a "Units:" dropdown menu set to "lb - in - psi". To the right is a "Continue" button with a right-pointing arrow. Below that is an "Analysis Type:" dropdown menu set to "Focused Crack Mesh For J-Integral Analysis (Standard)". Underneath are two input fields: "Analysis Name:" and "Model Label:". At the bottom is a "Notes:" text area containing the text "note text before the option; *use midpoint=0.35; more text after the option". The option key word and value, "*use midpoint=0.35", is highlighted in blue.

Figure 234 Mid side position option key word and value given in the Home/Notes form

An example crack mesh with the mid side node position at 0.4 from the crack front is shown in the figures below.

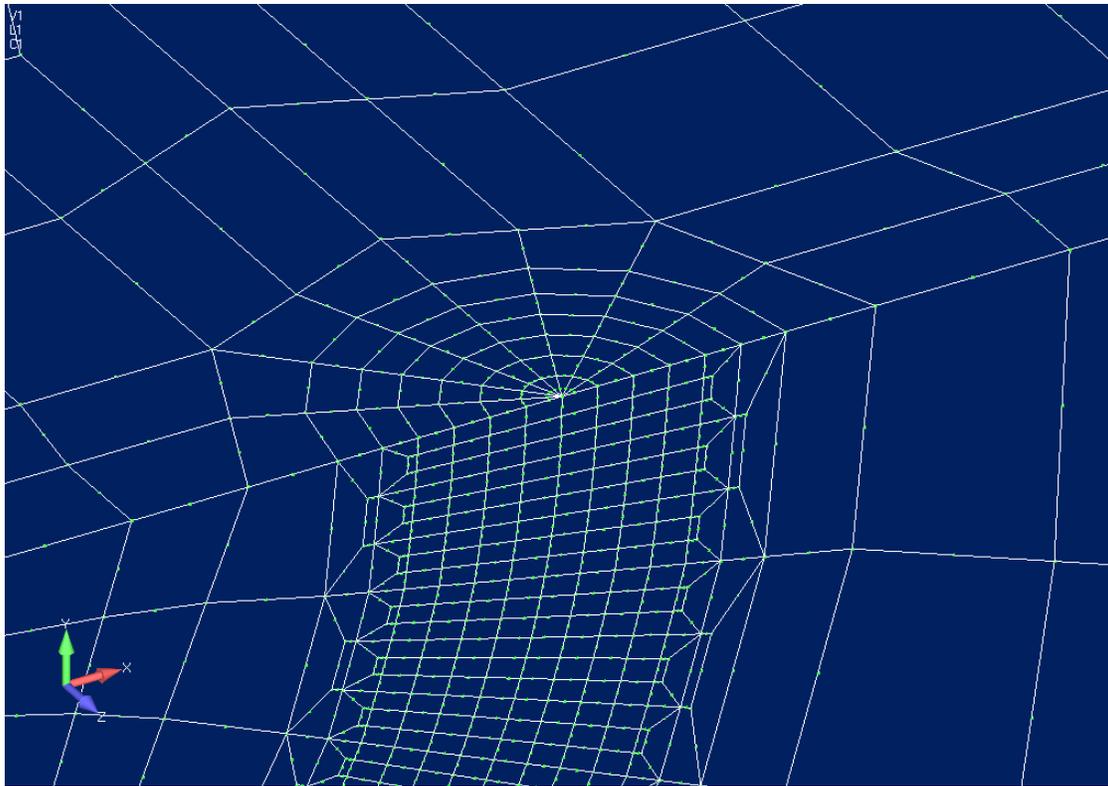


Figure 235 Example crack mesh with the mid side node position at 0.4

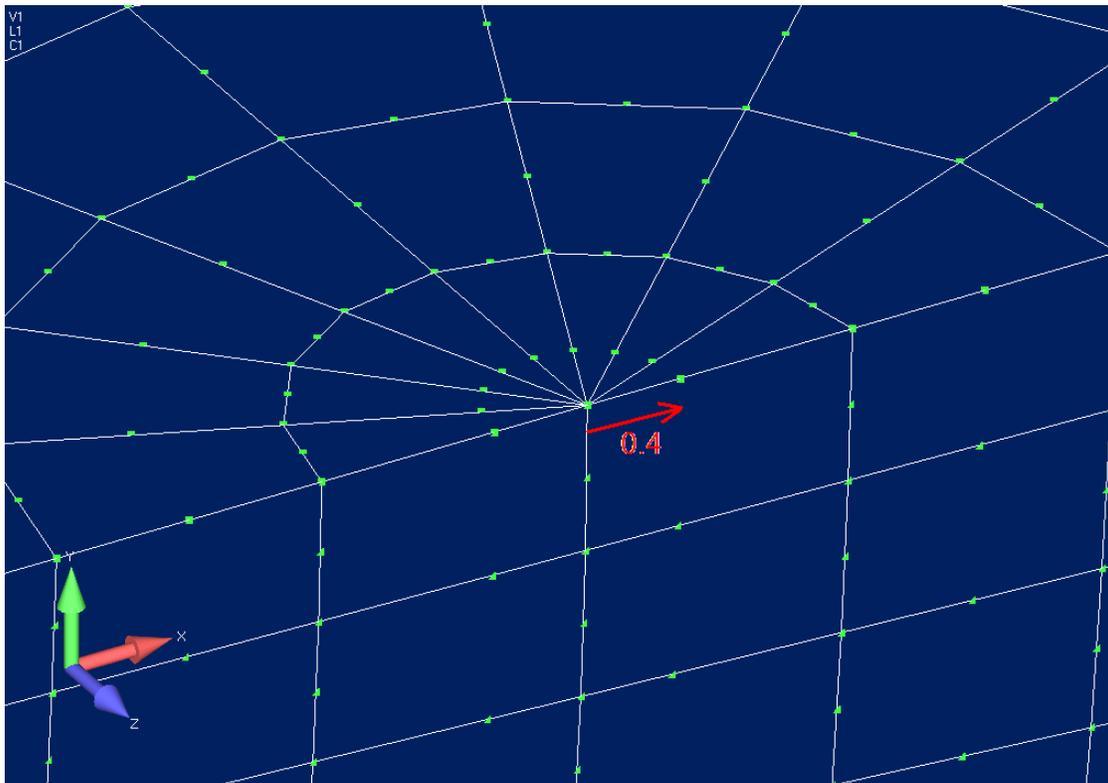


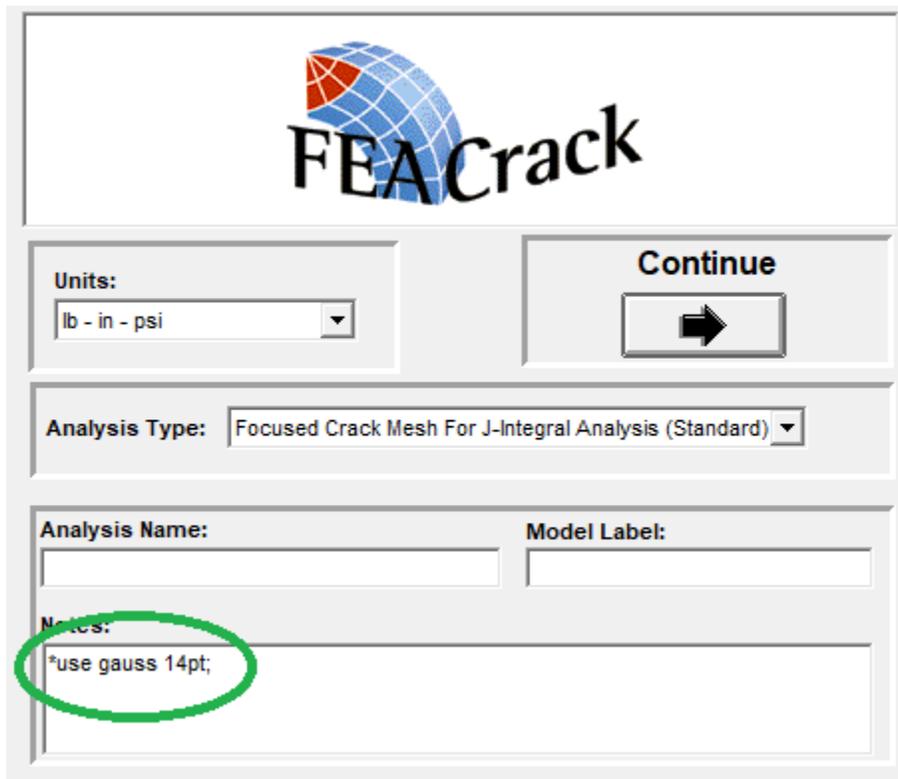
Figure 236 Example: close-up of a crack mesh with the mid side node position at 0.4 from the crack front

Gauss Integration Rule for Warp3D

The Gauss integration rule for Warp3D input files can be set to the 9-point rule or 14-point rule using two choices of syntax in the Home/Notes form:

- *use gauss 9pt
- *use gauss 14pt

The “*use gauss 9pt” syntax will write the “order 9pt_rule” to the Warp3D element type data, and the “*use gauss 14pt” syntax will write the “order 14pt_rule” to the Warp3D element type data in the input file. The figure below shows an example of the key word syntax for the 14-point rule.



The screenshot shows the FEACrack software interface. At the top is the FEACrack logo. Below it are several input fields and buttons. A 'Continue' button with a right-pointing arrow is on the right. A 'Units' dropdown menu is set to 'lb - in - psi'. An 'Analysis Type' dropdown menu is set to 'Focused Crack Mesh For J-Integral Analysis (Standard)'. There are two empty input fields for 'Analysis Name' and 'Model Label'. A 'Notes' field contains the text '*use gauss 14pt;', which is circled in green.

Figure 237 Example: close-up of a crack mesh with the mid side node position at 0.4 from the crack front

Radial Point Results

Stress and strain results can be obtained along radial lines from the crack front nodes using the “Radial Crack Front Calculator” in the Tools menu. Two methods are available: radial lines on the crack plane and in 3D at an arbitrary angle around the crack front. The crack plane method uses the element shape functions to interpolate the stress and strain results at the radial point locations. The 3D arbitrary angle method uses Inverse Distance Weighted (IDW) interpolation to obtain result values at the radial point locations.

The radial points are located along the crack propagation direction in the crack plane (the crack plane is at $\theta = 0^\circ$ for 3D IDW), which is the outward normal to the crack front at each crack front node. During the crack mesh generation, the crack propagation direction and crack normal

vectors are saved to the comment data block in the input file. During the radial point results post processing, the crack propagation direction vector is used to generate the radial point coordinates on the crack plane. In the 3D arbitrary angle method, the angle theta around the crack front is used to rotate the direction of the radial points. The stress and strain results are interpolated at the radial point locations using the selected method. The user enters the number of radial points and the radial distance where the results are interpolated; the radial points are evenly spaced. After obtaining the radial point results, the result values for plot x and y axes can be normalized by given denominator values.

For the crack plane only method, each radial point is assigned to an element face, and the local element iso-parametric coordinates are computed for the radial point. The element's stress and strain results are evaluated at the radial point using the element shape functions at the local element iso-parametric coordinates. Either Gauss integration point or element centroid results can currently be used to compute the radial point results; nodal results are not supported. Using Gauss point results appears to give the best radial point results using the crack plane method.

For the 3D arbitrary angle method, theta is the angle around the crack front from the crack plane. Theta is right-hand positive about the crack front tangent vector. For example, in the flat plate geometry at the first crack node at the free surface the tangent vector is outward from the mesh face in the y-axis direction. Theta is 0° on the x-y plane, theta is 90° in the negative z-axis, and theta is 180° on the crack face. The IDW interpolation algorithm is described in more detail in the section below. The 3D arbitrary angle method supports interpolation using nodal, Gauss point, and centroid results. Using nodal results appears to give the best interpolated radial point values using the IDW interpolation. IDW interpolation is also used to transfer residual stresses from an un-cracked mesh to a crack mesh in the Tools/Stress Interpolation form.

When radial mesh lines from the crack front are available, such as at the mesh surfaces, or if the radial mesh line option is used, an alternative method is to use the Select/Select Nodes tool to obtain results directly at the nodes. Use results at the nodes with this method.

To begin the radial point interpolation, load a crack mesh result file as usual using the FEA Results/File Selection form. After the results have been loaded, open the Tools/ Radial Crack Front Calculator form, shown in the figure below. Select the interpolation method on the Options tab from the top pull-down menu. Modify the number of radial points and the radial vector length (distance from the crack front) from the default values if desired. A default radial length value is obtained from the result file and given the text box. The default value can be changed by typing in a new value, the default radial length value is the crack tube radius for the particular mesh. To obtain all the radial point result values in a text file, check the "Create Extracted Results File" box and the radial point results are written to the <model name>_res.out file.

For the 3D arbitrary angle method, enter the angle theta (in degrees, from -180° to +180°), and modify the exponent P, the number of nearby points N, and the nearby point distance R. The exponent P value affects the weighting of nearby point values in the IDW interpolation. The number of nearby points N, and nearby point distance R, are used to limit the number of result points used to interpolate the result values at the radial point locations. The nearest N points are used for the IDW interpolation. Comparing radial point values, N = 2 nearby points gives good values. A small nearby distance R can speed up the interpolation by limiting the number of nearby points that are found and sorted for the number of points N. If the nearby distance R is too small, no result points will be found for interpolation.

Click the “Load Data” button to generate the radial points, and interpolate the results at the radial points. After the radial point results are initialized, the crack front nodes and result values can be selected using the check box lists at the left and right side of the form, as shown in the figure below.

The “Always on top” check box option will keep the Radial Crack Front Data form above all other windows to aid in selecting crack front nodes.

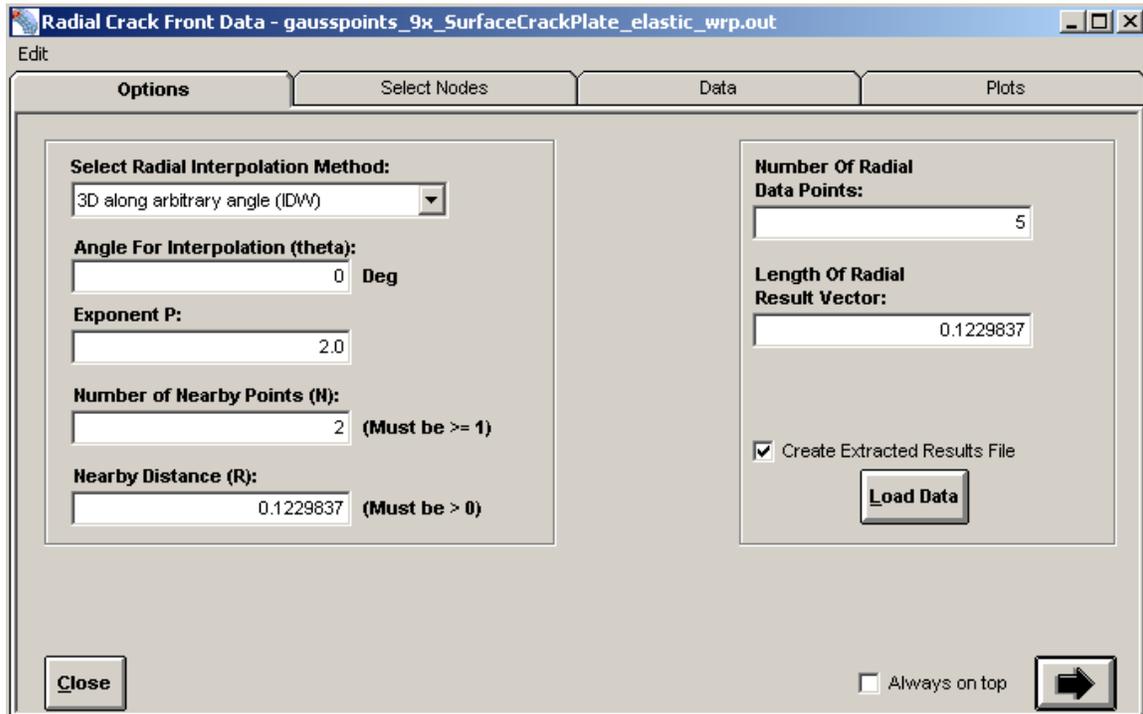


Figure 238 Radial point results form; select the method; click on Load Data after loading an FEA results file.

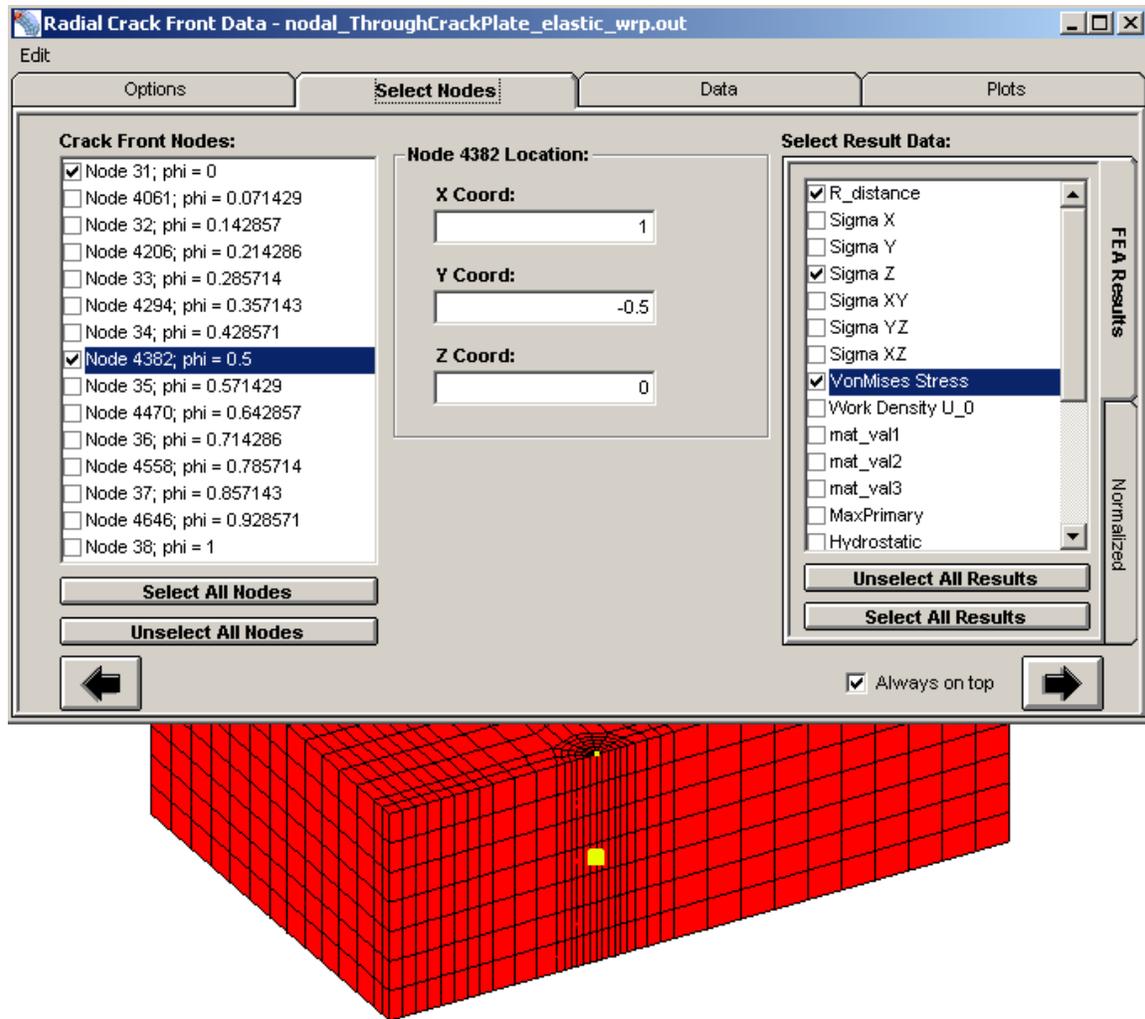


Figure 239 Check boxes to select the crack front node and result value; selected crack front nodes highlighted in yellow.

When the interpolation is finished, the Select Nodes tab will be activated automatically. Select the desired crack front nodes in the left column. The selected crack front nodes are highlighted in the mesh view in yellow. The highlighted node in the list is indicated by the larger yellow dot in the mesh view window. Select the stress or strain results in the right column in the FEA Results tab. The stress components, stress invariants, material state variables (for *WARP3D*), maximum primary stress, and the hydrostatic stress (the sum of the principal stresses divided by 3) are available. The selected results are listed in the Data sheet (second tab at the top of the form); the result selection can be changed instantly in the Data sheet without reloading the data. An example of the data sheet values is shown in the figure below. The data only needs to be reloaded if the number of radial points or the radial vector length is changed. When interpolation parameters are changed, then the radial points must be recomputed.

The “Show Spreadsheet Edit Bar” check box at the bottom of the Data tab allows individual cell values to be edited or to add or change a cell equation (see normalized results below). The Edit menu allows the selected data cells to be copied to the clipboard to paste into other programs for more calculations or plots; key board short cuts like “Ctrl+C” can also be used.

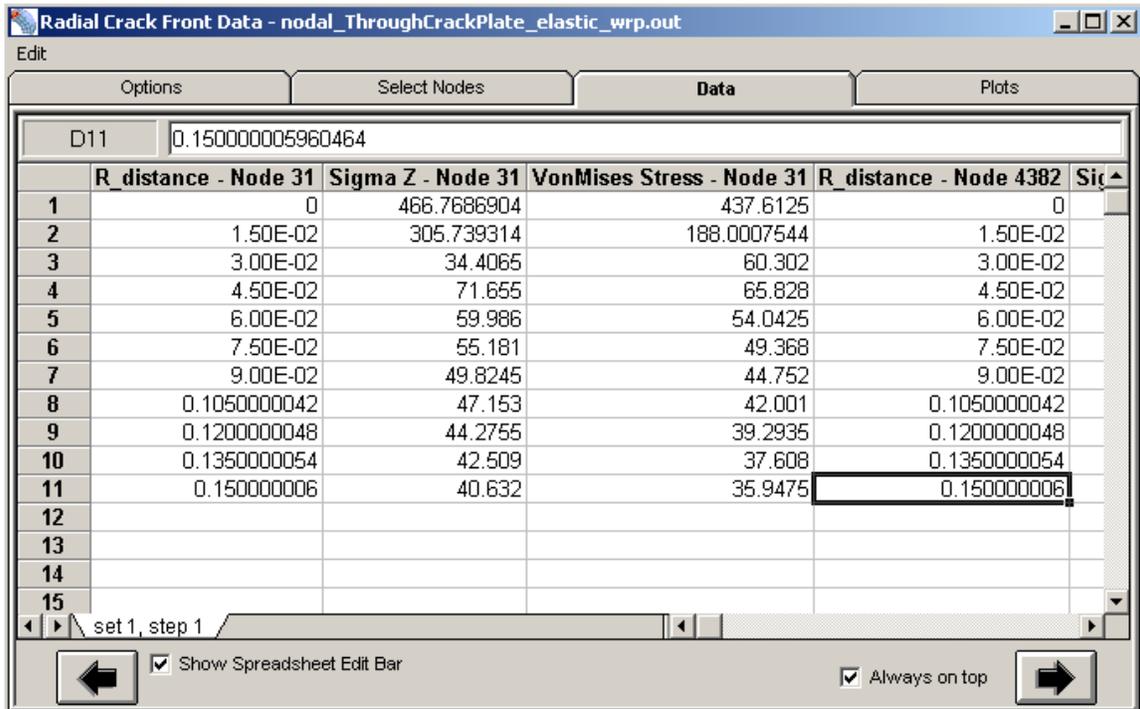


Figure 240 Data sheet contains the selected data values, one page per result set. The spreadsheet edit bar allows cell values to be viewed or changed.

The results are shown in x-y plots in the Plots tab. Click the “Show Plot Choice” button at the bottom of the tab to select the particular result values from the data sheet to plot (the standard *FEACrack* plot data selection form). One or many result values can be selected for a plot. Six plot tabs are available (along the right side of the form) for plotting a variety of the radial result and normalized values. For results with many output sets, the output set can be selected using the check box list on the right side of the plot. The data values and plots can be copied and pasted to other software for plotting and reports.

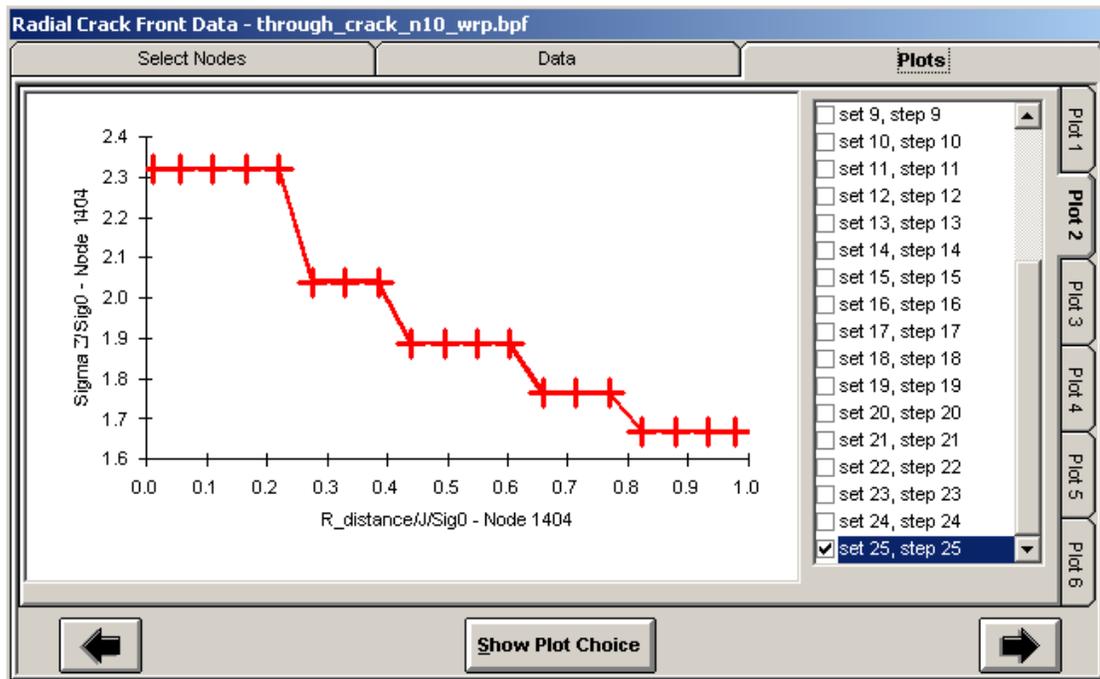


Figure 241 X-Y plot of the Sigma Z stress versus the radial distance for a selected result set (centroid results).

Figure 242 Plot variable selection form. Choose the x-axis variable from the left menus; choose the y-axis variable from the center menus.

For the crack plane method, depending on the mesh (full mesh or subset mesh), the crack dimensions, and the radial distance given by the user, some of the radial points may extend past the edge of the mesh. When this occurs, the radial points outside the mesh surface are discarded from the list, and a warning message is given. The available radial point results for each selected crack front node are listed in the data sheet.

Normalized Results

To normalize the radial point result data values click on the “Normalized” tab on the right side of the Select Nodes form; see the figure below. Enter the number of normalized data columns. Additional columns will be added on the Data sheet to contain the normalized values. To normalize the x and y axes of a plot, at least 2 data columns are needed. For example, to normalize the radial distance, the Sigma Z stress, and the maximum primary stress, 3 data columns are needed for the additional normalized data.

For each normalized result value, select the result to be normalized using the pull-down menu. Select the result label from the next pull-down menu (or type in a label instead), and then enter the denominator value to divide the result column values by. The denominator label menu contains a selection of frequently used labels like “J/Sig0”, “YS” for yield strength, or “Flow Stress”. The denominator label is combined with the result label to give the normalized column labels: “R_distance/J/Sig0” or “Sigma Z/Sig0”.

To normalize the radial distance by J/Sig0, the appropriate J value for the selected crack front node is automatically obtained from the crack results for each analysis step, but the J value is not shown separately since it changes for each crack node and result step. The J value being used to normalize the R_distance can be seen by using the spreadsheet edit bar on the Data tab and selecting one of the J/Sig0 cells. The Sig0 value is given in the “Divide By J over:” text box. If default normalizing values are used, the Sig0 value estimate is entered and shown in blue; see the figure below. The user can change the Sig0 value by selecting the “User Defined Values” radio button and then entering the Sig0 value in the text box. User defined normalizing values are shown in black.

Other result values such as stress or strain values are normalized by a single denominator value. In the figure below the Sigma Z stress is normalized by the Sig0 value. The Sig0 value can be the default estimate or can be entered by the user. If the same Sig0 value is desired for both the stress and R_distance normalization, it must be entered twice with the same value to be consistent for each result selection.

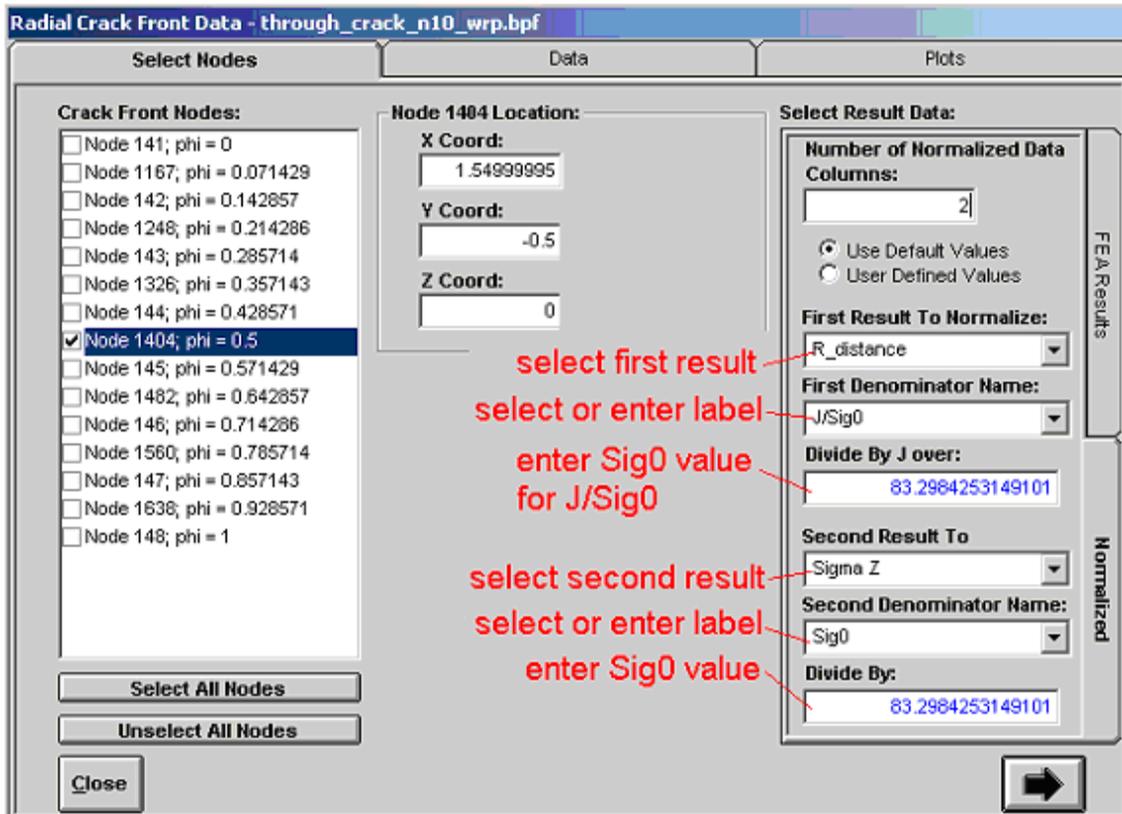


Figure 243 Select results, labels, and enter values to normalize the radial point result values.

The default Sig0 values shown in blue can be changed by selecting the “User Defined Value” and entering a new value.

The normalized data columns in the Data tab are shown in the figure below. The spreadsheet edit bar is also shown with the intermediate $J/Sig0$ calculation. The column labels are formed from the selected results and the given normalized value labels: “result label”/“normalized label”. The normalized result columns are now available to be selected for an x-y plot. Use the “Show Plot Choice” button to select the normalized result columns to plot in one of the plot tabs. Take care when selecting the data column for the x-y plot since there are extra “work” columns like “ $J/Sig0$ ” used for intermediate calculations. The normalized results can be copied from the Data sheet into another form, such as the Q calculator for additional post processing calculations.

Radial Crack Front Data - through_crack_n10_wrp.bpf

Select Nodes Data Plots

C1 =15.9666037994346/83.29842

	Sigma Z - Node 1404	J/Sig0 - Node 1404	R_distance/J/Sig0 - Node 1404	Sigma Z/Sig0 - Node 1404
1	193.2687373	0.1916795517	0.0109832435	2.320196917
2	193.2687373	0.1916795517	0.05491621749	2.320196917
3	193.2687373	0.1916795517	0.109832435	2.320196917
4	193.2687373	0.1916795517	0.1647486525	2.320196917
5	193.2687373	0.1916795517	0.2196648699	2.320196917
6	169.8805822	0.1916795517	0.2745810874	2.039421423
7	169.8805822	0.1916795517	0.3294973049	2.039421423
8	169.8805822	0.1916795517	0.3844135224	2.039421423
9	157.1667248	0.1916795517	0.4393297399	1.886791188
10	157.1667248	0.1916795517	0.4942459574	1.886791188
11	157.1667248	0.1916795517	0.5491621749	1.886791188
12	157.1667248	0.1916795517	0.6040783923	1.886791188
13	146.9542736	0.1916795517	0.6589946098	1.764190408
14	146.9542736	0.1916795517	0.7139108273	1.764190408
15	146.9542736	0.1916795517	0.7688270448	1.764190408
16	138.9303612	0.1916795517	0.8237432623	1.667863102
17	138.9303612	0.1916795517	0.8786594798	1.667863102
18	138.9303612	0.1916795517	0.9335756972	1.667863102
19	138.9303612	0.1916795517	0.9884919147	1.667863102

set 23, step 23 set 24, step 24 set 25, step 25

Show Spreadsheet Edit Bar

Figure 244 Normalized result values with column labels. Values in a cell are shown in the edit bar.

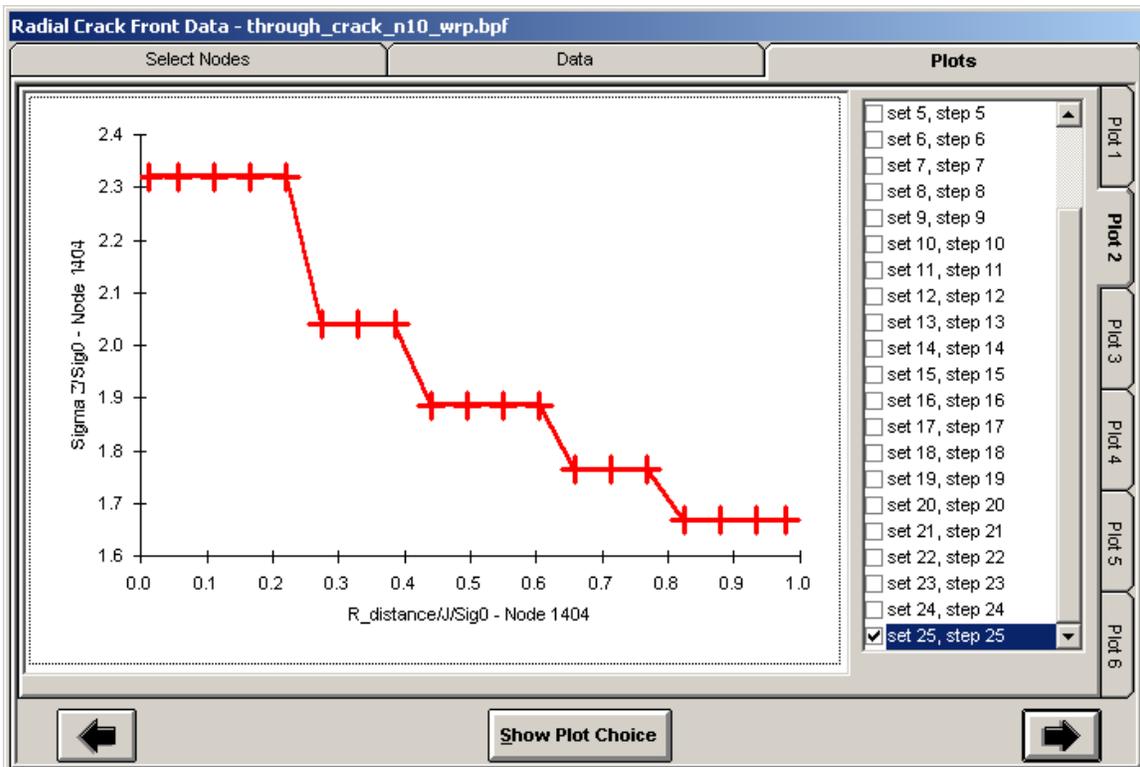


Figure 245 Plot the normalized result values.

IDW Interpolation

The inverse distance weighted (IDW) 3D interpolation method (Shepard's method) [D. Shepard, "A two-dimensional interpolation function for irregularly-spaced data", Proceedings of the 23rd National Conference ACM, ACM 517-524] is used to interpolate from FEA stress and strain result values to the radial point locations around the crack front. IDW interpolation permits use of irregularly spaced data in 3D Cartesian space. The IDW interpolation is given by the following equations:

$$F(x, y, z) = \sum_{i=1}^n w_i f_i \quad (\text{IDW 1.1})$$

$$w_i = \frac{d_i^{-p}}{\sum_{j=1}^n d_j^{-p}} \quad (\text{IDW 1.2})$$

$$d_i = \sqrt{(x - x_i)^2 + (y - y_i)^2 + (z - z_i)^2} \quad (\text{IDW 1.3})$$

Where the value, $F(x,y,z)$, is interpolated from the given n data values, f_i ; from the FEA results. For example the various stress and strain component values give individual f_i values, which are interpolated to the F value at the radial point location. The w_i weighting values are computed using the distance, d_i , from the interpolation point position (the radial point location) to the surrounding data points (FEA result locations). The exponent value, p , is a positive number, typically 2.0, and is used to adjust the relative weighting of near and far data points. A higher value of p gives more weighting to nearer points and decreases weighting from farther data points.

A local subset of data points can be used by selecting the nearby distance R and the number of nearby points N to limit the stress interpolation to a smaller volume around each interpolation point and speed up the interpolation calculations.

The FEA result points are listed at their global Cartesian x,y,z locations so that the same interpolation method can be used for nodal, Gauss point, and centroid results. For nodal results, the FEA results are listed at the node x,y,z coordinates. For Gauss point results, each Gauss point is evaluated using the element shape functions and element corner node coordinates to get the Gauss point x,y,z global Cartesian coordinates. The Gauss point result values are listed with the Gauss point coordinates to make the list of result data points for the IDW interpolation. For centroid results, the element centroid Cartesian x,y,z coordinates are computed using the element shape functions to get the list of results used for interpolation.

Nodal results appear to give the best interpolated values at the radial points since the result locations are usually closer to the radial points. The second best results are from Gauss point results since there are more result points (typically 8 Gauss points per element) as compared to centroid results (1 point per element). Increased mesh refinement can also help improve the radial point results by giving more result points, but can slow down the interpolation calculations due to more crack front nodes and more data points.

A2 Crack Front Constraint

Using normalized crack opening stress values extending radially ahead of the crack front, the A2 term from the power-series stress equation can be used to fit the equation to the data. The A2 term is computed using the equations from references 1 and 2. The A2 value is used as a measure of crack front constraint when the J-integral does not fully describe the crack front stresses to the model configuration. The material must be described by the Ramberg-Osgood strain-hardening power-law equation:

$$\frac{\varepsilon}{\varepsilon_0} = \frac{\sigma}{\sigma_0} + \alpha \left(\frac{\sigma}{\sigma_0} \right)^n \quad (\text{A2 1.1})$$

Where σ_0 is the yield stress, α is a fitting coefficient, n is the strain-hardening exponent, and $\varepsilon_0 = \sigma_0/E$ where E is the Young's modulus of elasticity.

The first three terms of the crack front normalized stress power-series equation are given by:

$$\begin{aligned} \frac{\sigma_\theta}{\sigma_0} &= \bar{A}_1 \left[\left(\frac{r}{L} \right)^{s_1} \mathcal{O}_\theta^{(1)} + A_2 \left(\frac{r}{L} \right)^{s_2} \mathcal{O}_\theta^{(2)} + A_2^2 \left(\frac{r}{L} \right)^{s_3} \mathcal{O}_\theta^{(3)} \right] \\ \bar{A}_1 &= \left(\frac{J}{\alpha \varepsilon_0 \sigma_0 I_n L} \right)^{-s_1} \\ s_1 &= - \left(\frac{1}{n+1} \right) \end{aligned} \quad (\text{A2 1.2})$$

Where σ_θ is the crack opening stress (stress component normal to the crack plane), r is the radial distance ahead of the crack front, and L is a characteristic length (1.0 is a default). The $\mathcal{O}_\theta^{(k)}$ non-dimensional field values, s_k stress exponents, and non-dimensional integral value I_n are tabulated functions of the given strain-hardening exponent n (ref. 2). Tabulated values are available for $3 \leq n \leq 50$. Note that s_1 can also be computed directly from n . To solve for the A2 value, the power-series stress equation can be rearranged in terms of the A2 coefficient and solved as a quadratic.

$$aA_2^2 + bA_2 + c = 0 \quad (\text{A2 1.3})$$

Where the quadratic equation a , b , c coefficient terms are given by the equations:

$$\begin{aligned} a &= \left(\frac{r}{L} \right)^{s_3} \mathcal{O}_\theta^{(3)} \\ b &= \left(\frac{r}{L} \right)^{s_2} \mathcal{O}_\theta^{(2)} \\ c &= \left(\frac{r}{L} \right)^{s_1} \mathcal{O}_\theta^{(1)} - \frac{1}{\bar{A}_1} \frac{\sigma_\theta}{\sigma_0} \end{aligned} \quad (\text{A2 1.4})$$

The A2 value is computed using the positive root from the quadratic equation:

$$A_2 = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \quad (\text{A2 1.5})$$

Using the given normalized radial stress values σ_θ/σ_0 versus the normalized radial distance ahead of the crack front $r\sigma_0/J$, an A2 value is computed for each data point in the range $1.0 \leq r\sigma_0/J \leq 5.0$ and then averaged to get the final A2 value.

A2 Calculator

To compute the A2 term, normalized radial stress values must first be obtained ahead of the crack. Use the Tools/Radial Crack Front Calculator to obtain the normalized crack opening stress values. Also obtain the J-integral value that corresponds to the same crack front node and analysis step as the normalized radial stress data.

Open the Tools/A2 Calculator form to enter the data, which includes the characteristic length, the J-integral value corresponding to the normalized stress data (same crack front node, same analysis step), and the Ramberg-Osgood power-law material values. To compute multiple A2 values, enter the number of A2 values to be computed in the top left corner text box, and multiple tabs will be available to enter the data. The figure below shows the form with typical data values.

A report text file "A2Report.txt" containing all the data and intermediate values used to compute A2 is created when the check box in the top right corner of the form is checked. The A2 report text file is written to the *FEACrack* install folder.

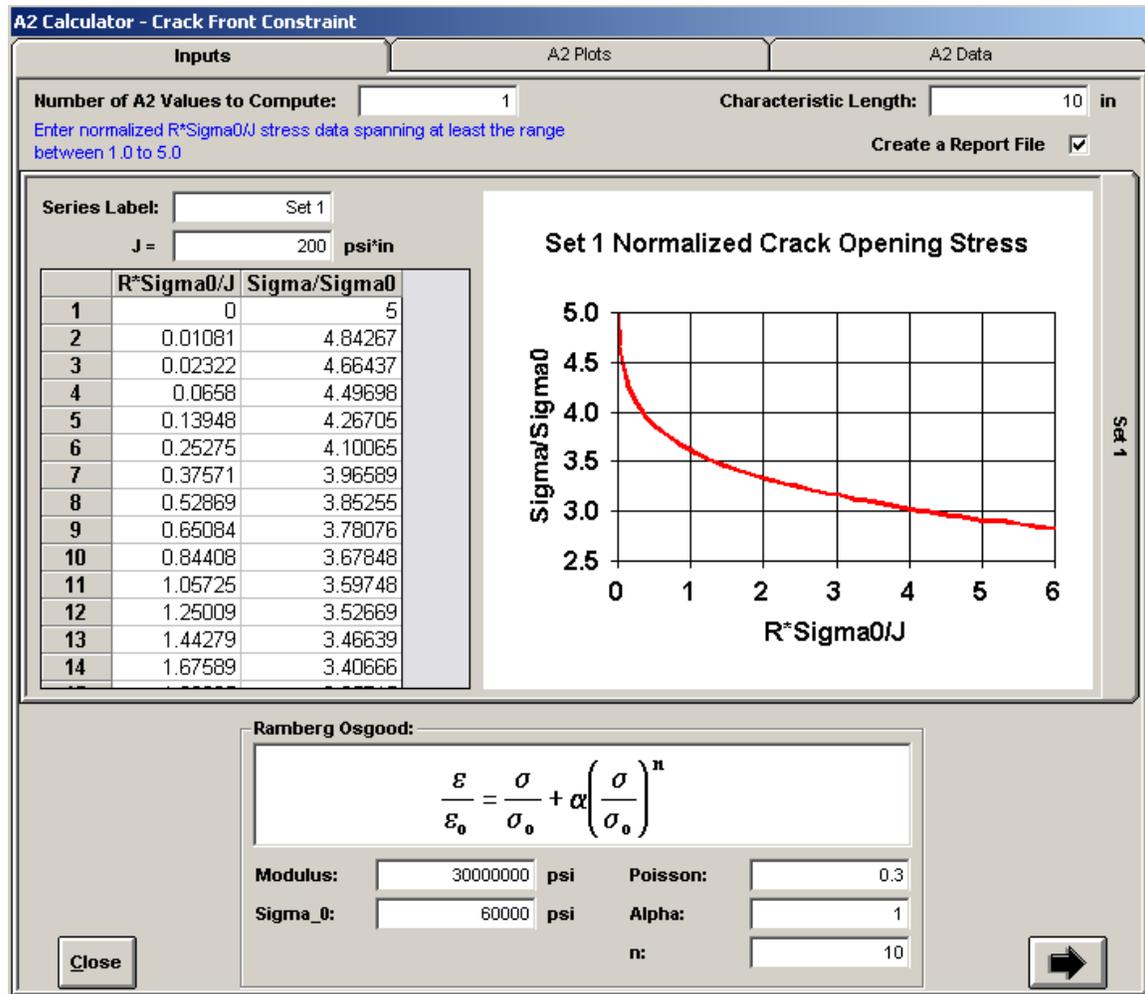


Figure 246 A2 Calculator, enter normalized stress profile

The normalized stress data is plotted to allow quick checking that the data is correct. Several data points within the $1.0 \leq r\sigma_0/J \leq 5.0$ normalized radius range are recommended. The normalized stress data can extend beyond that range for convenience, but only the values within the 1.0 to 5.0 normalized radius range are used in the average to compute A2. Once the data is entered, click the right arrow button in the bottom right corner of the form to compute A2 and plot the curve-fit stress trends.

To test how well the computed A2 value gives a curve-fit to the normalized stress data, the A2 calculator evaluates the power-series stress equation using the given J and computed A2 values. The evaluated curve will match the data most closely within the $1.0 \leq r\sigma_0/J \leq 5.0$ range. The first term in the power-series equation is the HRR solution (Hutchinson, Rice, and Rosengren, 1968) and is also plotted for comparison.

The computed A2 value is given in the text box in the lower left corner of the A2 Plots tab. The evaluated power-series stress values are available in the A2 Data tab.

The figure below shows the comparison plots and the computed A2 value.

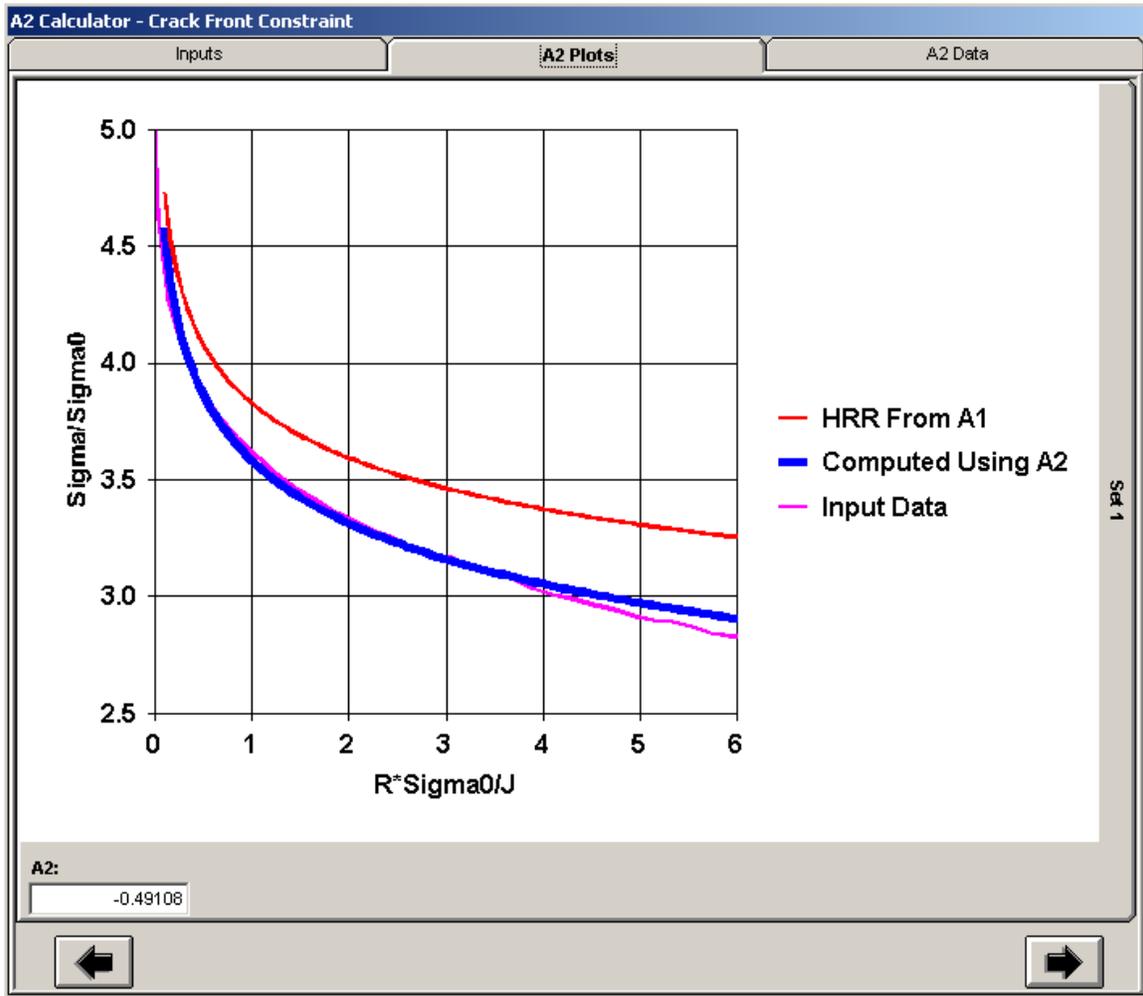


Figure 247 Compare the input data to the power-series stress equation evaluated using J and A2

Q calculator

The Q stress difference field is defined as the difference between a reference stress solution at $T=0$, $(\sigma_{yy})_{T=0}$, (a modified boundary layer analysis with the elastic T-stress = 0) and the crack plane normal stress, σ_{yy} , divided by the yield strength, σ_0 :

$$Q = \frac{\sigma_{yy}}{\sigma_0} - \frac{(\sigma_{yy})_{T=0}}{\sigma_0} \quad (\text{Q 1.1})$$

The equation for Q is given in this form to emphasize the difference between the two normalized stress fields. The normal stresses are given as functions of the normalized radial distance from the crack front. Refer to section 3.6.2 “J-Q Theory” in the Fracture Mechanics textbook [Fracture

Mechanics, Fundamentals and Applications, T. L. Anderson, 2nd ed.]. Note that the crack plane normal stress solution and the reference solution must use the same stress-strain material data.

In *FEACrack*, Q can be calculated using the Tools/Q Calculator form, shown in the figure below. On the left side of the Input Data form enter the normalized radial distance values, $R*\Sigma_0/J$, and the normalized reference solution values, Σ_{YY}/Σ_0 . On the right side of the form enter the second analysis solution (a 3D crack mesh solution). Enter the normalized radial distance and normalized crack plane normal stress values, Σ_N/Σ_0 . For a crack mesh with the crack in the x-y plane the normal crack plane stress is Σ_Z , or for a crack mesh with the crack in the y-z plane the normal crack plane stress is Σ_X . Using normalized values, permits many analysis cases to use the same reference solution when both the reference and analysis solutions use the same stress-strain curve or power law. The crack plane normal stresses for the analysis solution can be obtained using the Tools/Radial Crack Front Calculator.

Reference Solution:			Second Analysis Solution:		
	$R*\Sigma_0/J$	Σ_{YY}/Σ_0		$R*\Sigma_0/J$	Σ_N/Σ_0
1	0	5	1	0.005144358	3.220816667
2	0.01081	4.84267	2	0.025721791	3.220816667
3	0.02322	4.66437	3	0.051443583	3.220816667
4	0.0658	4.49698	4	0.07716556	3.220816667
5	0.13948	4.26705	5	0.102887165	3.220816667
6	0.25275	4.10065	6	0.128609143	3.220816667
7	0.37571	3.96589	7	0.154330748	3.220816667
8	0.52869	3.85255	8	0.180052726	3.220816667
9	0.65084	3.78076	9	0.20577433	2.831516667
10	0.84408	3.67848	10	0.231496308	2.831516667
11	1.05725	3.59748	11	0.257217913	2.831516667
12	1.25009	3.52669	12	0.282939891	2.831516667
13	1.44279	3.46639	13	0.308661496	2.619733333
14	1.67589	3.40666	14	0.334382472	2.619733333

Figure 248 Enter the normalized stress versus normalized radial distance to compute Q

A reference solution for hardening exponent $n = 10$ is available in the “Figure3_33_ModBoundaryLayerStressField.xls” EXCEL file, or the “ModifiedBoundaryLayerResults_Fig3_33.txt” text file. These Files can be found as examples that are available when installing your *FEACrack* 3.0. The appropriate reference solution should be obtained before using the Q Calculator.

To compare and verify the normalized stress solutions, the given data is plotted on the Input Plots form, shown in the figure below. Use the plot to verify that the normalized stress results are the correct order of magnitude.

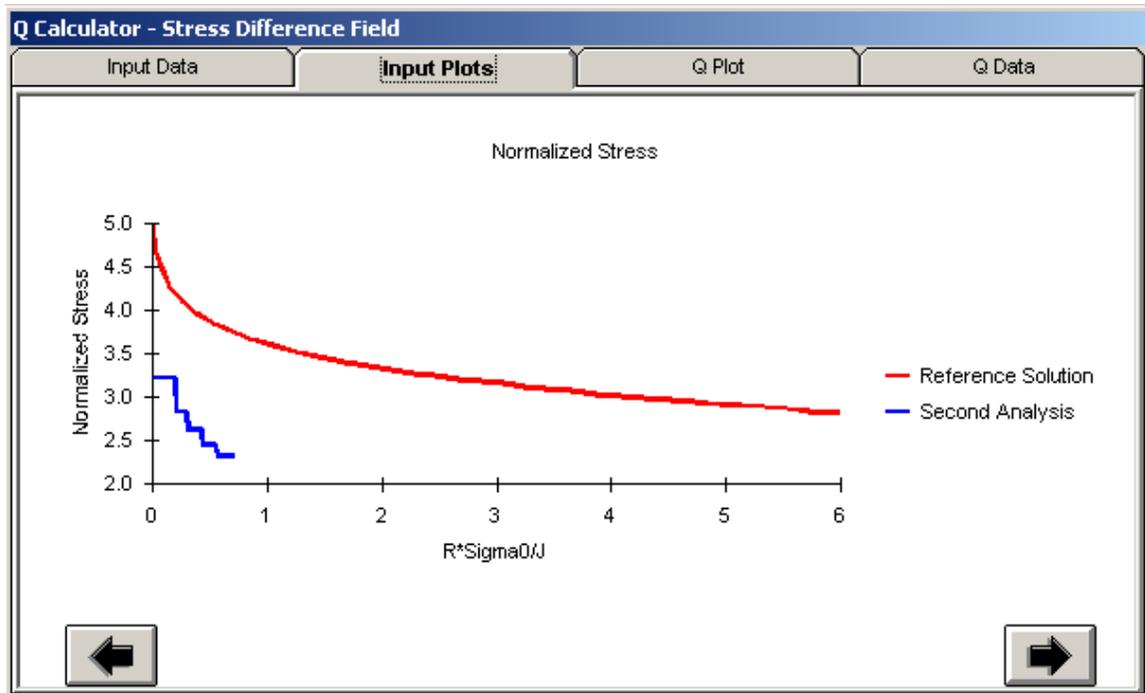


Figure 249 Plot the reference and second analysis normalized solutions

To compute Q as a function of the normalized radial distance, click on the right-arrow or the Q Plot tab. Q will be computed at the same $R*\sigma_0/J$ x-data points as the analysis solution by interpolating the reference solution data to the same x-data points as the analysis solution. In general the $R*\sigma_0/J$ x-data points will not be the same between the reference solution and the analysis solution, so interpolation is used to get normalized stress values at the same x-data points to permit the stress field difference to be computed. If any analysis solution data points are outside the range of the reference solution, those points from the analysis solution will be ignored and not included in the computed Q values. The computed Q values are shown in the Q Plot tab and the values are listed in the Q Data tab, see the figures below.

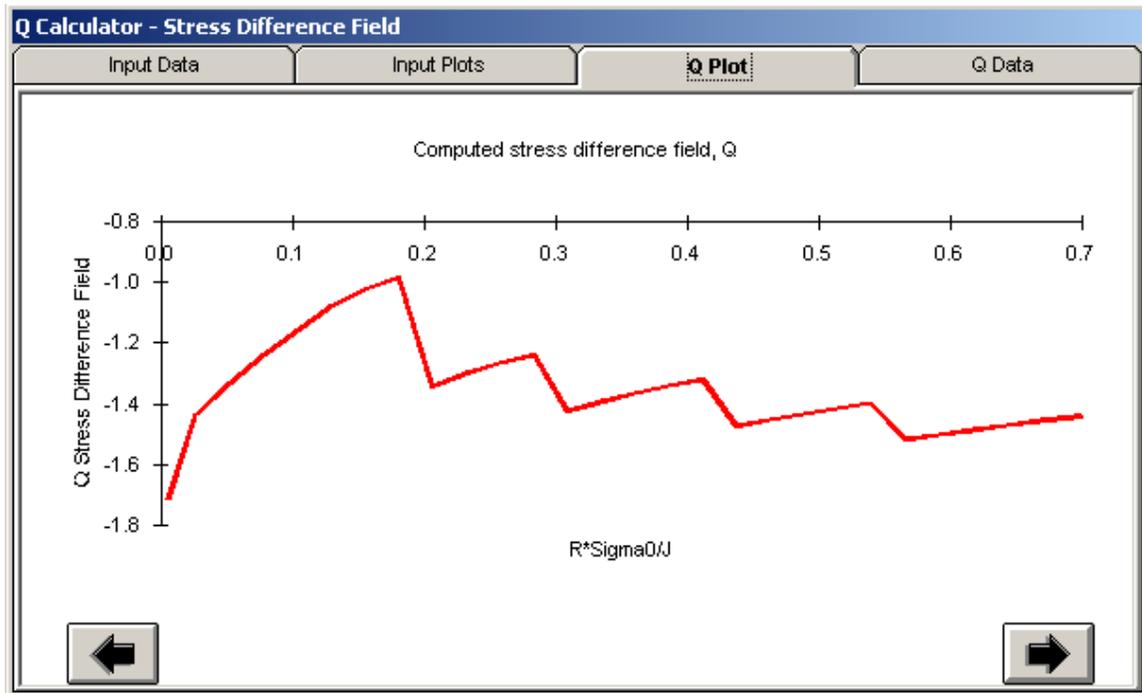


Figure 250 Plot the compute Q values versus normalize radial distance from the crack front.

The figure shows a table with two columns: "R*sigma0/J" and "Q Difference". The table contains 15 rows of data. The values for "Q Difference" are all negative, ranging from approximately -1.70 to -1.39.

	R*sigma0/J	Q Difference
1	0.005144358	-1.704311747
2	0.025721791	-1.433718321
3	0.051443583	-1.332601112
4	0.07716556	-1.24069532
5	0.102887165	-1.160427016
6	0.128609143	-1.080157548
7	0.154330748	-1.024416749
8	0.180052726	-0.9866297168
9	0.20577433	-1.338143234
10	0.231496308	-1.300356202
11	0.257217913	-1.264236652
12	0.282939891	-1.236046234
13	0.308661496	-1.419639559
14	0.334383473	-1.391449142
15	0.360105451	-1.363258714

Figure 251 Computed Q values

Results and Reference Solution for Q-stress

The results from the modified boundary layer model can be normalized using the Tools/Radial Crack Front Calculator in *FEACrack*. To compute the Q-stress, the crack plane normal stress (σ_{zz}) is typically normalized by the σ_0 value, and the radial distance ahead of the crack front is normalized by J/σ_0 . This normalized result gives the “reference solution”. The crack plane stress results from another analysis normalized in the same way are then used with the boundary layer model reference solution to compute Q. In *FEACrack* use the Tools/Q Calculator form. The plot in Figure 6 shows a comparison of boundary layer model results for two cases: $T/\sigma_0 = 0$ and $T/\sigma_0 = -1$ ($T_{\max} = -60,000$ psi). The normalized stress results are compared to Figure 3.33 from Ted Anderson’s Fracture Mechanics textbook.

As the applied load increases (the applied K and T), the size of the plastic zone near the crack front will also increase. To obtain a good reference solution the outer region of the boundary layer model should remain in the elastic range to limit the effect of the small plastic zone near the crack front. To examine the effect of the plastic zone, the K from J results can be computed and compared to the applied K values. If the K from J result closely matches the applied K value, then the plastic zone is not affecting the overall elastic behavior of the boundary layer model. The plot in Figure 178 shows that the K from J results begins to diverge at higher applied K values. If the K values diverge too much, then the model radius should be increased, or the applied K should be decreased. To compute K from J, copy the J vs step plot to a spreadsheet and convert J to K using the equation:

$$K = \sqrt{\frac{JE}{1-\nu^2}} \quad (\text{Q 1.2})$$

Where K is the stress intensity, J is the J-integral analysis result, E is the modulus of elasticity, and ν is the Poisson ratio.

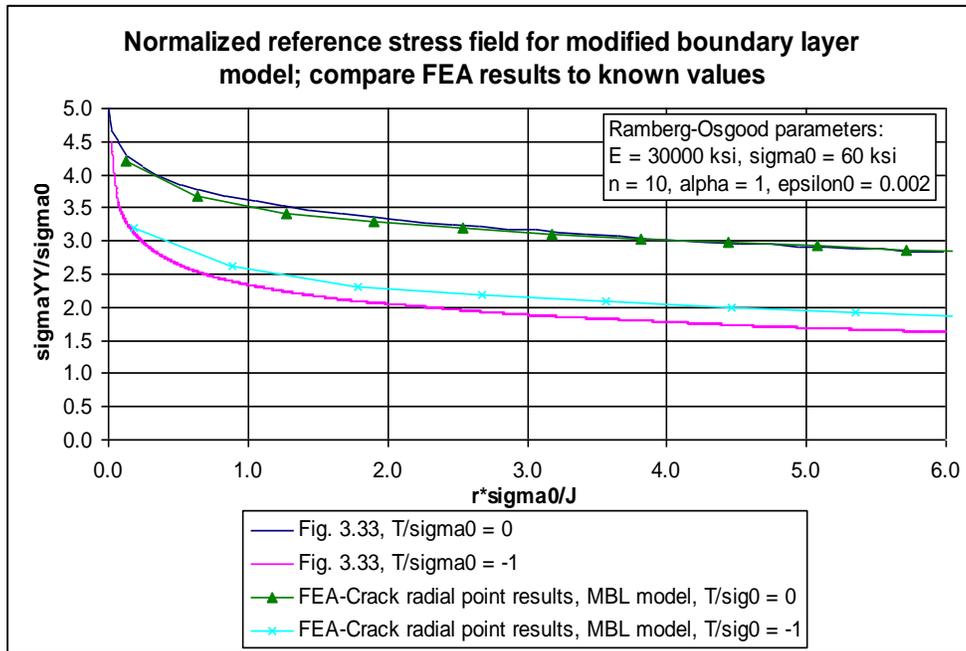


Figure 252 Normalized crack plane stress results compared to text book values; the reference solution used to compute the Q-stress for another analysis geometry

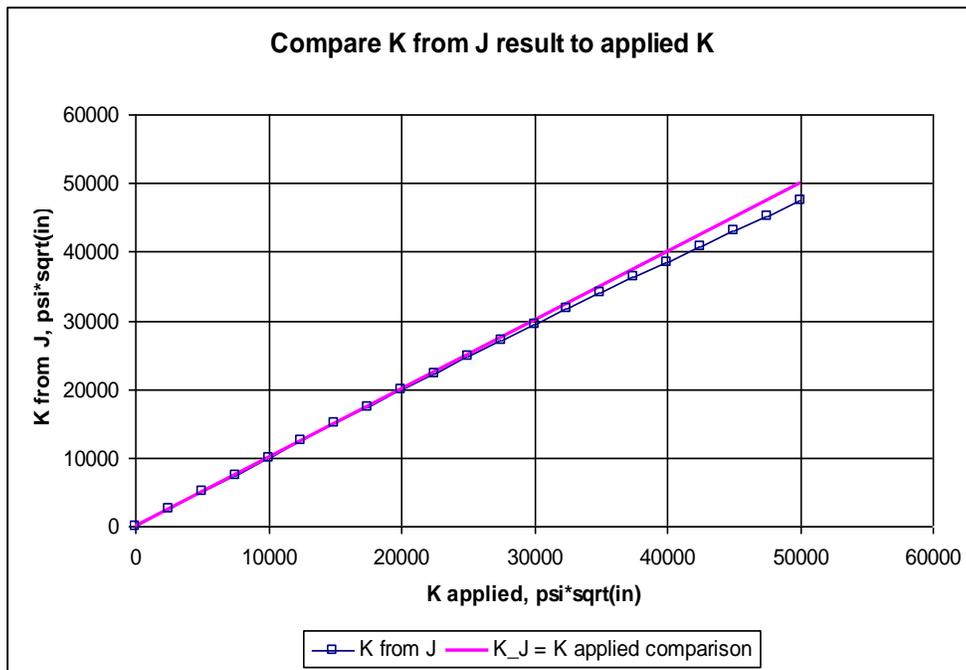


Figure 253 Compare K from J and applied K values to examine the crack front plastic zone effect; K from J begins to diverge at about 25,000 psi*sqrt(in)

Alpha-H calculator

The α_h value, Newman’s hyper-local constraint factor, is computed from a weighted average of the crack plane normal stresses along the outward normal to the crack front. The α_h method is described in the report “Fracture Analysis Of Semi-Elliptical Surface Cracks In Ductile Materials” [“NAG8 1883 final report.pdf”, August 25, 2004]. From equation (5-1) in the report (page 28), α_h is given by:

$$\alpha_h(\phi) = \frac{1}{S(\phi)} \int_0^{s(\phi)} \left(\frac{\sigma_{zz}}{\sigma_0} \right) ds \quad (\text{A 1.1})$$

Where ϕ is the crack front position angle; (0 at the free surface to $\pi/2$ at the deepest point of the crack.); S is the size of the plastic zone, σ_{zz} is the crack plane normal stress (crack opening) in the plastic zone, σ_0 is the yield strength, and ds is the increment along the crack front normal radial line. The plastic zone is defined by the von Mises stress being greater than σ_0 at the radial points ahead of the crack front. From the report, crack front “constraint refers to the buildup of stresses around a crack front due to restraint against in-plane and out-of-plane deformation” (page 8).

The α_h values along the crack front are computed in *FEACrack* using the Tools/alpha-H calculator form to select the crack plane normal stress and von Mises stress from the FEA results. Select an analysis step, and enter the σ_0 and radial point values. In the figure below, the crack plane normal stress and von Mises stress results are selected from the pull-down menus on the left side of the form; the appropriate crack plane normal stress for the current crack orientation must be chosen. For example, for an axial crack in the x-y plane the Sigma Z results are normal to the crack plane, or for a circumferential crack in the y-z plane the Sigma X results are normal to the crack plane. The analysis output step is also selected on the left side of the form; typically later analysis steps should be used to have sufficiently high von Mises stress at the crack front that exceeds the given σ_0 . The yield strength value, σ_0 , the number of radial points, and the radial length are entered in the text boxes on the top, right side of the form. The number of radial points and the radial length has the same definition as in the Tools/Radial Crack Front Calculator. Since the α_h constraint factor is computed for all the crack front nodes, the radial points are recomputed for this calculation. After selecting and entering the values, click the Load Values button to generate the radial points along the crack outward normal lines and to compute the α_h constraint factor.

To determine the size of the plastic zone ahead of the crack front, the radial point von Mises stress results greater than σ_0 are included in the plastic zone. When the von Mises stress drops below σ_0 the distance to where the von Mises stress is equal to σ_0 between two radial points is computed by interpolation. The crack plane normal stress at the radial points inside the plastic zone and the last interpolated point at the edge of the plastic zone are used to compute the weighted average, given in the equation above. To evaluate the integration of the normal stress over the plastic zone, trapezoidal integration is used. The computed integral value is normalized by the given σ_0 , and divided by the plastic zone size to get α_h at each crack front node.

The Alpha H Calculator form is divided into three tabs: Inputs, FEA Results, and Alpha H. The Inputs tab is active and contains the following fields:

- Crack Plane Normal Stress:** A dropdown menu set to "Sigma Z".
- Sigma0:** A text input field containing the value "1329.7".
- Von Mises Stress:** A dropdown menu set to "VonMises Stress".
- Number Of Radial Data Points:** A text input field containing the value "30".
- Load Step:** A dropdown menu set to "set 40, step 40".
- Length Of Radial Result Vector:** A text input field containing the value "2.5".

At the bottom of the form, there are three buttons: "Close", "Load Values", and a right-pointing arrow button.

Figure 254 Alpha-H Calculator form to select stress results and enter radial point values

After the α_h values are computed, they are plotted versus the normalized crack front angle $2*\phi/\pi$ (0 at the crack tip to 1.0 at the deepest point of the crack front) in the Alpha H/Plot tab; see the figure below. The computed values including the number of radial points in the plastic zone and the plastic zone size, S , are listed in the Alpha H/Data tab. The radial point distances from the crack front and FEA results (crack plane normal and von Mises stress) used to compute the constraint factor are listed in the FEA Result tabs.

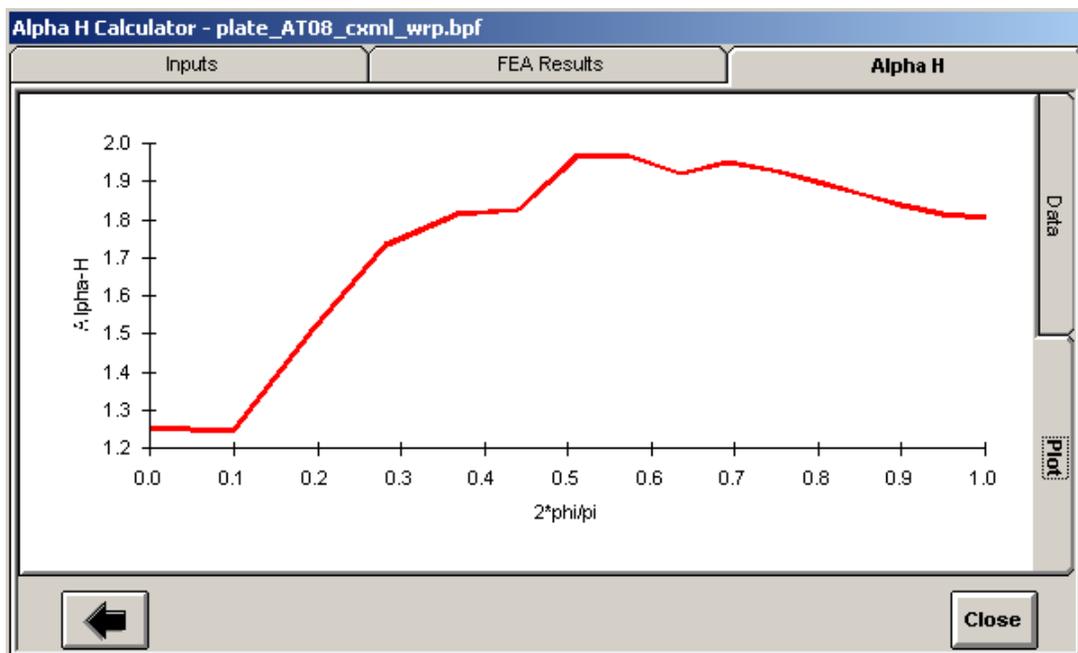


Figure 255 Computed α_h values along the crack front

Alpha H Calculator - plate_AT08_cxml_wrp.bpf

Inputs		FEA Results		Alpha H	
	phi	2*phi/pi	Alpha-H	Number Of Radial Points In Plastic Zone	S Zone Size
Node 170	0	0	1.253859009	15	1.467883202
Node 171	0.154135	9.81E-02	1.245075672	16	1.571331453
Node 172	0.303788	0.193397446	1.507985857	15	1.504731858
Node 173	0.444896	0.2832295904	1.73426447	10	1.034011976
Node 174	0.576986	0.3673206952	1.813342812	10	0.9609462233
Node 175	0.69281	0.4410565316	1.825905484	10	0.931516746
Node 176	0.800899	0.5098681432	1.964359021	7	0.6604486105
Node 177	0.901567	0.5739553511	1.965741871	6	0.6168813556
Node 178	0.998038	0.6353707028	1.920023836	6	0.5991288131
Node 1501	1.086142	0.6914594179	1.951597677	6	0.579864051
Node 1502	1.17215	0.7462138555	1.928800511	6	0.5537798631
Node 1503	1.255092	0.7990163769	1.897778785	6	0.56438225
Node 1504	1.335803	0.8503985984	1.867299653	6	0.578839018
Node 1505	1.414955	0.9007883176	1.837680834	6	0.5993681137
Node 1506	1.493116	0.9505471551	1.81482125	6	0.6185737387

Close

Figure 256 Computed α_h values and plastic zone data

Alpha H Calculator - plate_AT08_cxml_wrp.bpf

Inputs		FEA Results					Alpha H	
	Node	Phi	Point 1	Point 2	Point 3	Point 4	Point 5	
1	170	0	2.07E-02	0.1034481382	0.2068967134	0.3103446864	0.4137929641	
2	171	0.154135	2.07E-02	0.103447737	0.2068966963	0.3103448828	0.4137931664	
3	172	0.303788	2.07E-02	0.103448165	0.2068964655	0.3103448286	0.4137931124	
4	173	0.444896	0.02068971377	0.10344799	0.2068965118	0.3103448141	0.4137930926	
5	174	0.576986	2.07E-02	0.1034481051	0.2068965398	0.3103440323	0.413793106	
6	175	0.69281	2.07E-02	0.1034482219	0.2068970741	0.3103446402	0.4137928446	
7	176	0.800899	2.07E-02	0.1034482485	0.206896814	0.310344736	0.413792978	
8	177	0.901567	2.07E-02	0.103447829	0.2068966907	0.3103447844	0.4137930445	
9	178	0.998038	2.07E-02	0.1034486163	0.2068971	0.3103451239	0.4137932759	
10	1501	1.086142	2.07E-02	0.1034485677	0.2068964719	0.310344969	0.4137931048	
11	1502	1.17215	2.07E-02	0.1034481933	0.2068965875	0.3103447927	0.4137930654	
12	1503	1.255092	2.07E-02	0.103448225	0.2068966198	0.3103448058	0.4137930795	
13	1504	1.335803	2.07E-02	0.103448244	0.2068966236	0.3103448135	0.4137930879	
14	1505	1.414955	0.02068968489	0.1034482563	0.2068966262	0.3103448183	0.4137930933	
15	1506	1.493116	2.07E-02	0.1034479918	0.2068966293	0.310343938	0.413793097	

Normal Stress
VonMises Stress

Figure 257 Radial point FEA results to compute α_h

ASTM J Calculator: C(T) and SE(B)

The equations from ASTM E1820 to compute J using the load line displacement (LLD) and the total reaction force for a compact tension C(T) and single-edge bending SE(B) test specimens are implemented in the *FEACrack* ASTM J calculator. In the Tools menu, select the “Show ASTM J Calculator” to open the form, shown in the figure below.

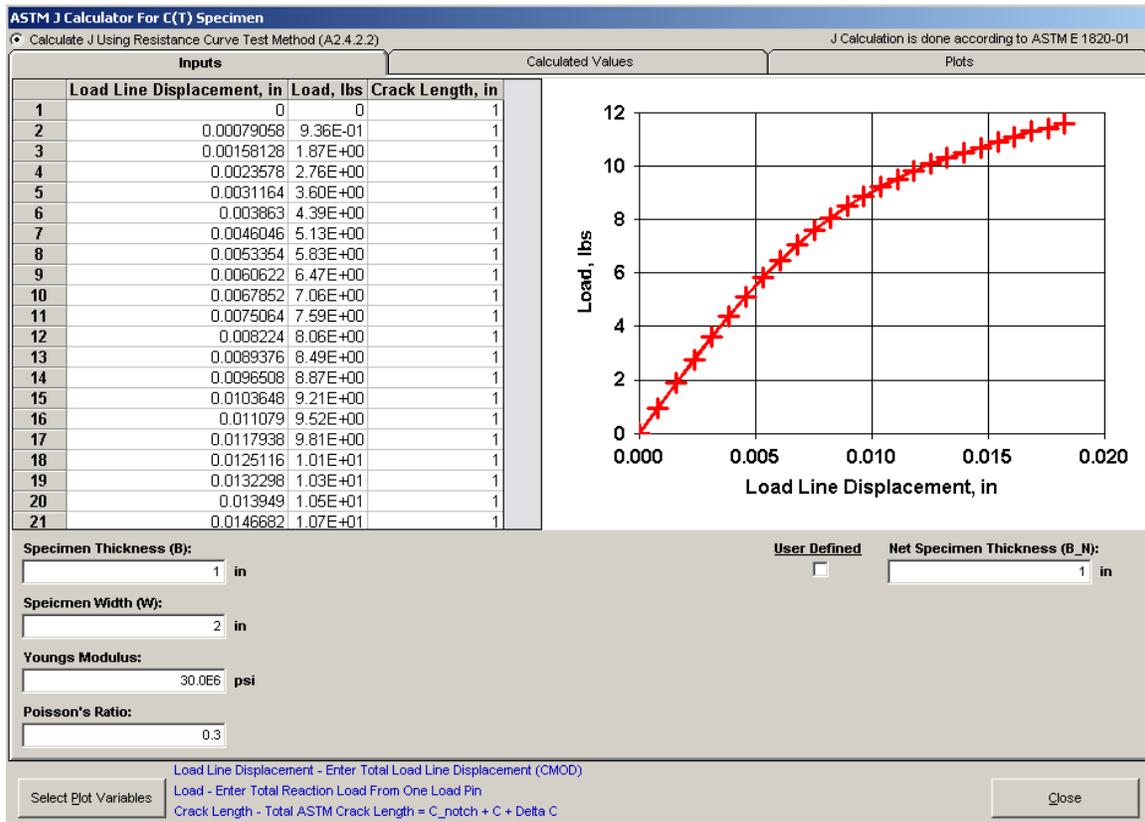


Figure 258 ASTM J Calculator in FEACrack

The load line displacement and total reaction force for the C(T) specimen can be obtained using predefined node sets from the FEA results (discussed below). The load line displacement is entered in the first column, and the total reaction force is entered in the second column for each analysis step. The crack length can be obtained from the crack results forms for a growing crack. For a stationary crack (using a focused crack mesh), enter the constant crack length for all analysis steps. After entering the data, the load-displacement curve is shown in the plot to verify that the data is entered correctly. At the bottom of the form enter the C(T) specimen thickness, B, the width, W, the modulus of elasticity, E, and the Poisson’s ratio. The net specimen thickness, B_N, is automatically set. The default values available for the C(T) dimensions are derived from ASTM 1820-01 but it is recommended that the user review all input values.

The intermediate calculations and final ASTM J values are listed in the Calculated Values tab. On the Plots tab, various result values can be selected for x-y plotting. For example, click on the Plots tab, and then click the Select Plot Variables button (lower left corner). Select the Time Step for the x-axis, and the ASTM J for the y-axis; click OK to close the form. The J versus load step

plot is then shown; see the figure below. Other combinations of result values can be selected on the other plots. For a growing crack, (results from a cohesive, Gurson, or node release analysis), the ASTM J can be plotted versus the Delta Crack Length to obtain a J-R plot.

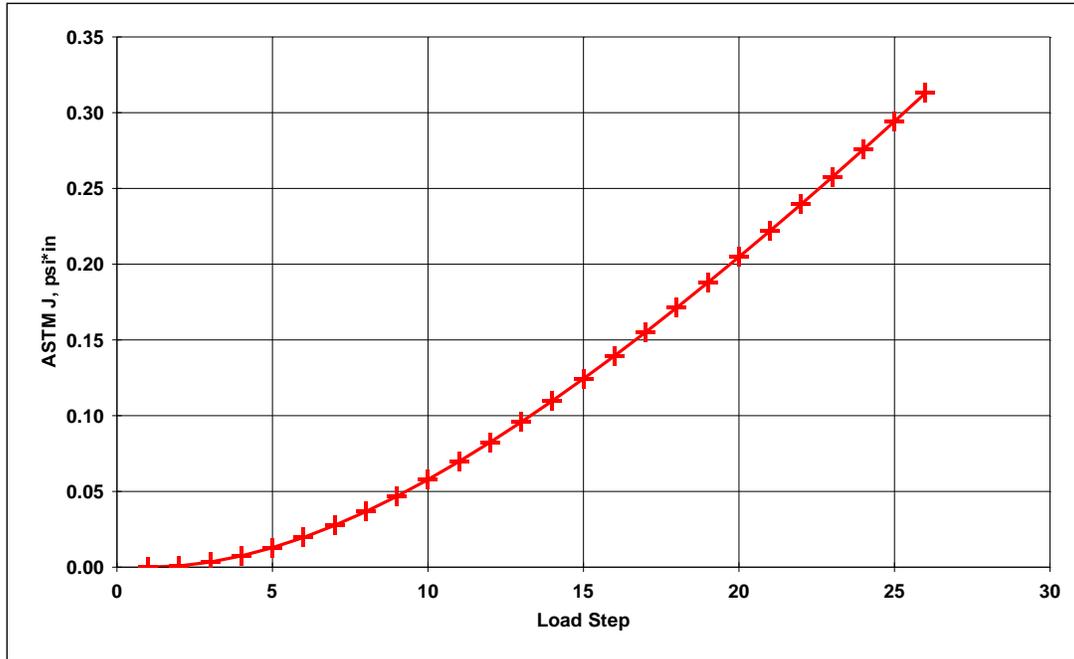


Figure 259 J versus Load Step from the ASTM J calculator

Obtaining Load Line Result Values

The result values needed for the ASTM J calculator can be easily obtained in *FEACrack* by using the predefined node sets and data sheets. Examples will demonstrate the methods to extract the needed results. The first example is a quarter symmetric C(T) specimen mesh with a static crack front, refer to the “ASTM J static crack example” files.

To get the load line displacement, use the Select/Select Nodes menu. In the Select Nodes form, use the pull-down menu to choose one of the “Load Disp nodes” node sets. The nodes are located along the C(T) specimen load line (load pin center line) at various locations through the thickness. The selected nodes are highlighted by yellow dots on the mesh. Highlight an individual node ID number in the “Selected Nodes” list to identify a particular node on the mesh picture. To most easily get the load line displacement, keep only one node, delete the other node from the node list by highlighting the node ID number and clicking on “Remove Node”. See the figure below for a selected node on the load line.

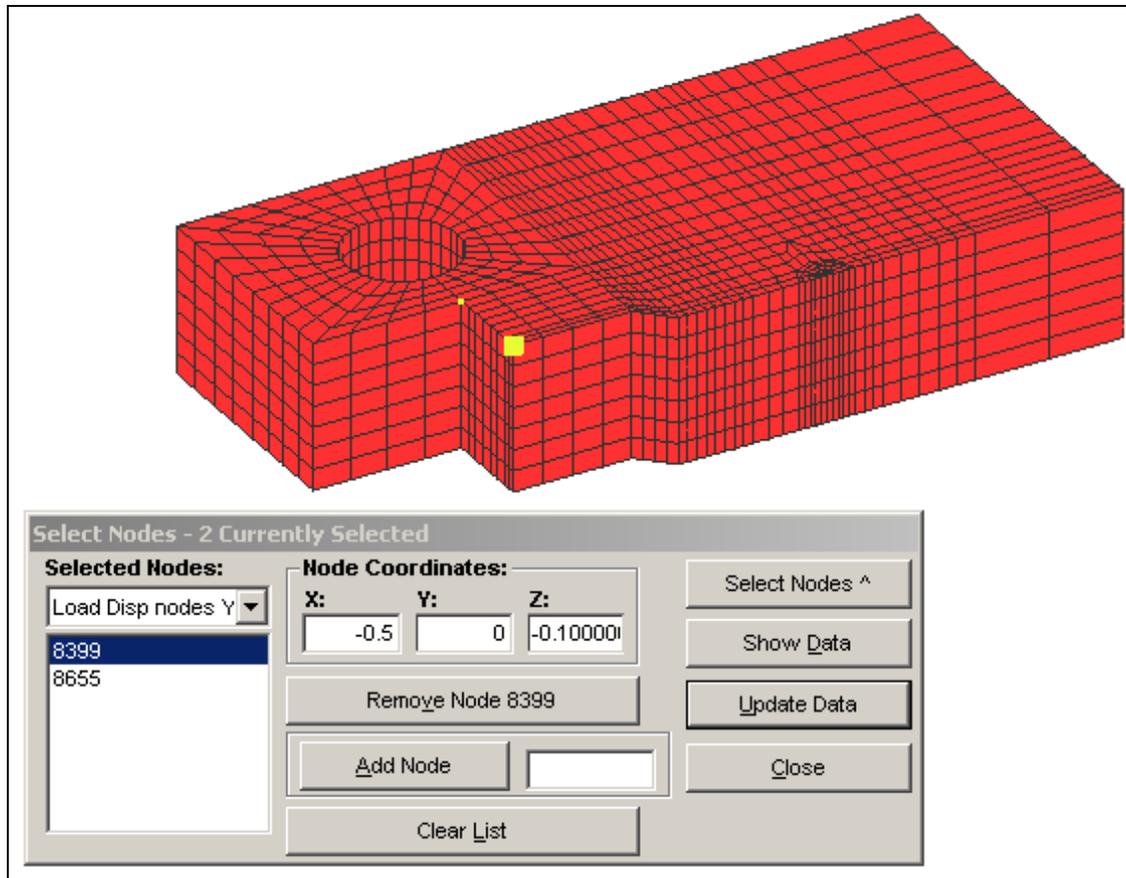


Figure 260 Use the predefined node set to select the node for the load line displacement.

Then click the “Show Data” button to automatically fill the data sheets with the available results at the selected node. In this case, the Z displacement along the load line is one of the result values available. There will be a row of data per node, and a data page per analysis output set. To plot the result, click the “Show Variable Selections” button. For example, choose the Time Step for the x-axis, and choose the Delta Z Per Step for the y-axis. Click OK, then click on the Plots tab. Click the check boxes in the right side column to add curves to the plot. Just one check box for a single node. Use a right click to quickly select or deselect all the check boxes.

Click on the “By Load Step” tab to get the data from each load step sheet listed by column on one data sheet for a selected result value. For a single node, there will be one row of data, and for many nodes there will be one row of data per node. At the bottom of the sheet the result value per step is listed in row and transposed column format, to make copy and pasting of the data to the ASTM J calculator easier. Use the “Data To Show” menu at the top right corner to select the Delta Z results. This is the load line node displacement of the selected node in the -Z loading direction. The sum and absolute sum columns at the bottom of the data sheet contain just the single node values (the sum of a single result). See the figure below for part of the Delta Z data sheet.

Since the ASTM J calculator requires the total load line displacement, multiply the delta Z displacement from the quarter symmetric C(T) mesh results by 2 (one crack face in the quarter symmetry mesh). Additional calculations can be made in the data sheet, or copy the delta Z values to another spreadsheet.

Selected Results							
Plots				Data			
Data To Show:							
Delta Z							
	Node Index	set 1, step 1	set 2, step 2	set 3, step 3	set 4, step 4	set 5, step 5	set 6
1	8399	-0.00039529	-0.00079064	-0.0011789	-0.0015582	-0.0019315	-0.0023023
2							
3	SUMS:	-0.00039529	-0.00079064	-0.0011789	-0.0015582	-0.0019315	-0.0023023
4	ABS(SUM):	0.00039529	0.00079064	0.0011789	0.0015582	0.0019315	0.0023023
5	AVGS:	-0.00039529	-0.00079064	-0.0011789	-0.0015582	-0.0019315	-0.0023023
6	MINS:	-0.00039529	-0.00079064	-0.0011789	-0.0015582	-0.0019315	-0.0023023
7	MAXS:	-0.00039529	-0.00079064	-0.0011789	-0.0015582	-0.0019315	-0.0023023
8							
9							
10		Transposed Totals					
11		SUMS:	ABS(SUM):	AVGS:	MINS:	MAXS:	
12	set 1, step 1	-0.00039529	0.00039529	-0.00039529	-0.00039529	-0.00039529	-0.00039529
13	set 2, step 2	-0.00079064	0.00079064	-0.00079064	-0.00079064	-0.00079064	-0.00079064
14	set 3, step 3	-0.0011789	0.0011789	-0.0011789	-0.0011789	-0.0011789	-0.0011789
15	set 4, step 4	-0.0015582	0.0015582	-0.0015582	-0.0015582	-0.0015582	-0.0015582
16	set 5, step 5	-0.0019315	0.0019315	-0.0019315	-0.0019315	-0.0019315	-0.0019315
17	set 6, step 6	-0.0023023	0.0023023	-0.0023023	-0.0023023	-0.0023023	-0.0023023
18	set 7, step 7	-0.0026677	0.0026677	-0.0026677	-0.0026677	-0.0026677	-0.0026677
19	set 8, step 8	-0.0030311	0.0030311	-0.0030311	-0.0030311	-0.0030311	-0.0030311
20	set 9, step 9	-0.0033926	0.0033926	-0.0033926	-0.0033926	-0.0033926	-0.0033926
21	set 10, step 10	-0.0037532	0.0037532	-0.0037532	-0.0037532	-0.0037532	-0.0037532
22	set 11, step 11	-0.004112	0.004112	-0.004112	-0.004112	-0.004112	-0.004112
23	set 12, step 12	-0.0044688	0.0044688	-0.0044688	-0.0044688	-0.0044688	-0.0044688
24	set 13, step 13	-0.0048254	0.0048254	-0.0048254	-0.0048254	-0.0048254	-0.0048254
25	set 14, step 14	-0.0051824	0.0051824	-0.0051824	-0.0051824	-0.0051824	-0.0051824
26	set 15, step 15	-0.0055395	0.0055395	-0.0055395	-0.0055395	-0.0055395	-0.0055395
27	set 16, step 16	-0.0058969	0.0058969	-0.0058969	-0.0058969	-0.0058969	-0.0058969
28	set 17, step 17	-0.0062558	0.0062558	-0.0062558	-0.0062558	-0.0062558	-0.0062558
29	set 18, step 18	-0.0066149	0.0066149	-0.0066149	-0.0066149	-0.0066149	-0.0066149

Figure 261 Part of the selected node result data sheet.

To extract the total reaction force, return to the Select/Select Nodes menu. This time pick the “Top Reaction Nodes” node set to select all the nodes that were constrained at the top load pin hole. (Or the top load pin when included in the mesh). All the nodes are highlighted by yellow dots, see the figure below. Use all the nodes in this node set. Again click the “Show Data” button to automatically fill the data sheets with the extracted node results. This time the Force Z results will give the total load on the C(T) mesh (remember to select the Reaction Forces in the Tools/Program Options form when building the mesh). On the “By Load Step” tab, pick the Force Z results in the “Data To Show” menu. The Z reaction force for all the nodes in the node set will be put on the single data sheet for each analysis step (one row per node, one column per set). At the bottom of the form, the sum of the Z Force for each step is given in row and transposed column format. The total reaction force sum must be multiplied by 2 for the quarter symmetric C(T) specimen to get the total load (half the thickness in the mesh). The multiplication can be done in the data sheet, or copy the Force Z sum to another spreadsheet. If the mesh were half symmetric the Z Force sum would not need to be multiplied by 2, since the full thickness is present in the mesh.

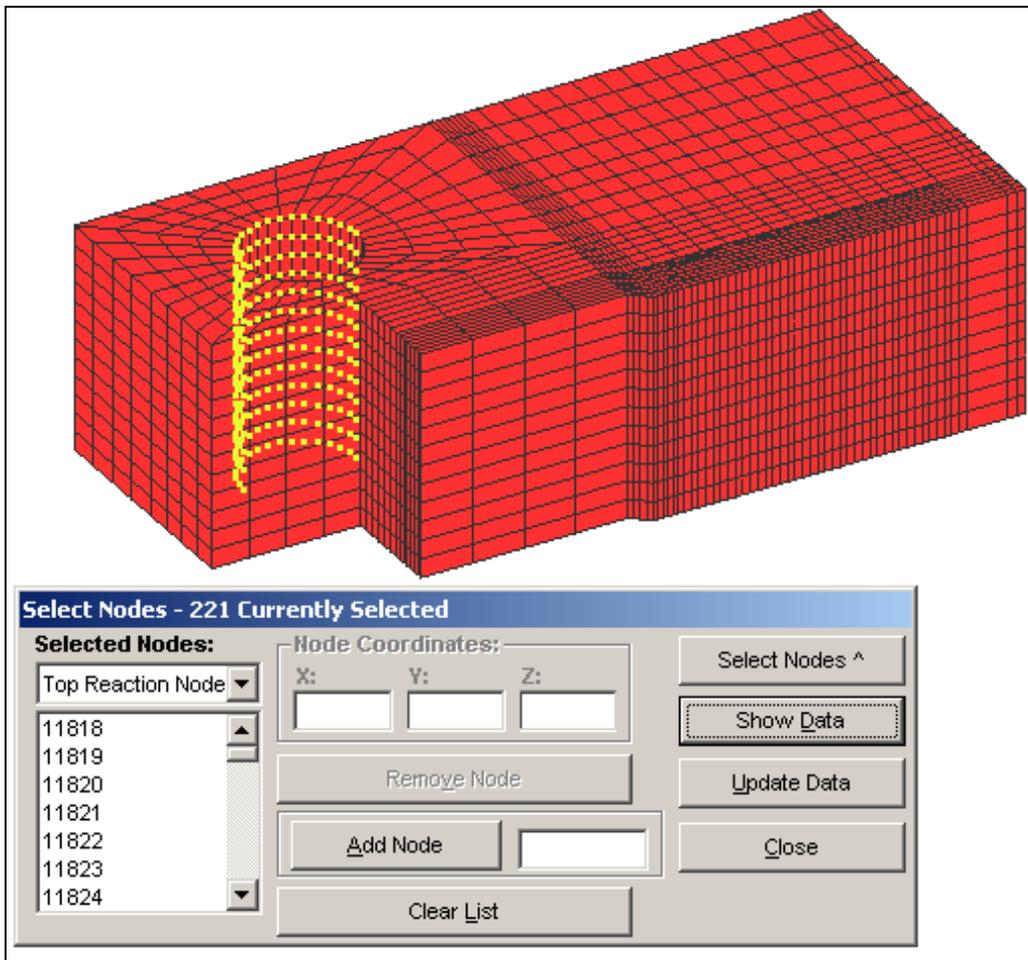


Figure 262 Node set with all constrained nodes in the load pin to obtain the total load.

When the total load line displacement and total load values per step are obtained (and modified by a factor of 2 if necessary), copy the values into the ASTM J calculator. All the data for the two C(T) mesh examples needed in the ASTM J calculator can be found in the *EXCEL* spreadsheets with part of the file name “data_for_ASTM_J”.

When pasting data from the data sheet into the ASTM J calculator, use the Edit/Paste Special menu to paste only the values. The default is to paste the data sheet cell formulas.

For a growing crack, the crack length results are also needed.

Obtaining Crack Length Result Values

In the FEA Results/File Selection form click the “Create J vs R Curve” box when loading cohesive, Gurson, or node release results to include the crack extension results in the crack results post processing. For the second C(T) mesh example refer to the “ASTM J cohesive example” files.

After loading the results file, the J-R curve is available in the Crack Results/J Extension tab. The delta_C crack extension values are available in the Data/J Extension data sheet. For a through crack, the average delta_C is reported in the data sheet. To obtain specific delta_C crack extension values for each crack front node position, also check the “Create Extracted Crack Results File” box before loading the results. A *_res.out file matching the model file name will be created in the same directory as the result file. The extracted crack results file contains all the crack result values and data listed in various formats for additional post processing (view the file using a text editor like Notepad). For the crack extension values at each crack front node position go to the “Crack extension results” section of the text file.

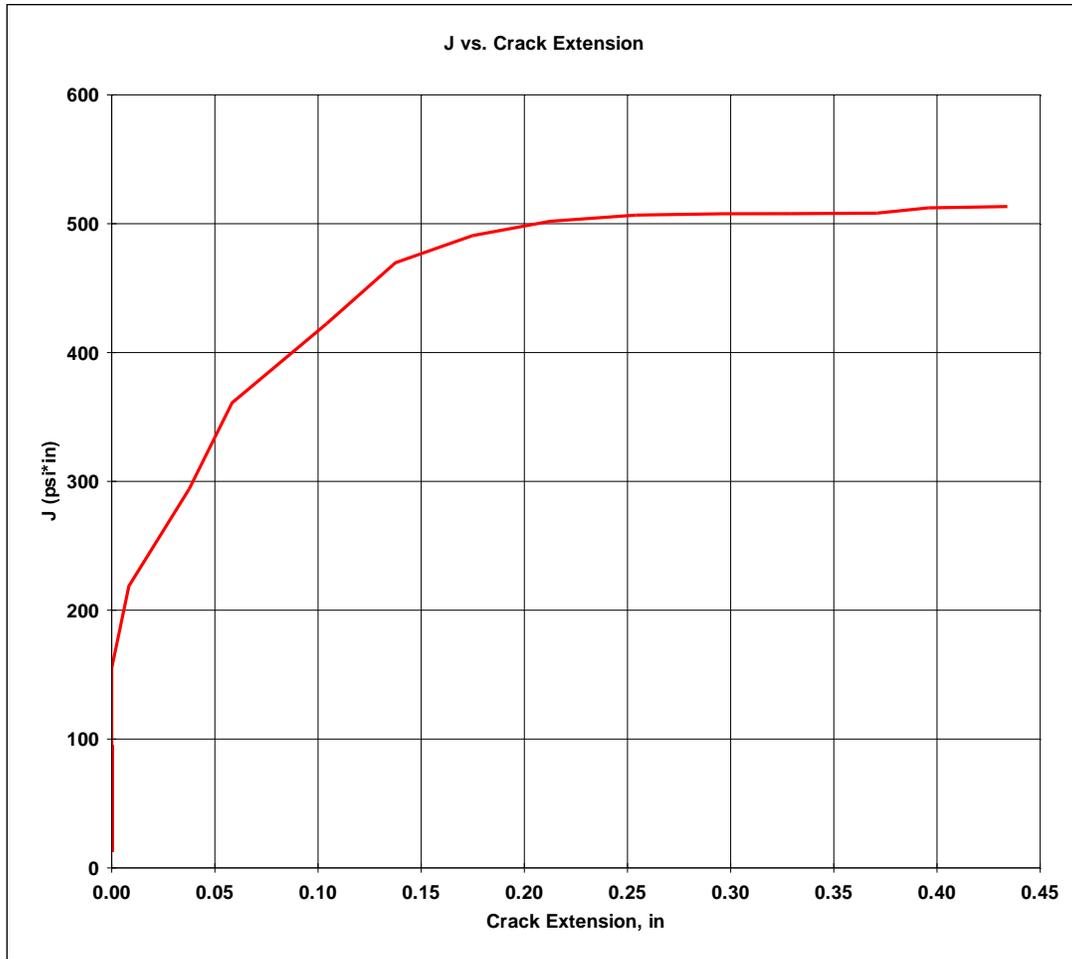


Figure 263 J-R curve from the cohesive C(T) specimen

The ASTM initial crack length $a_0 = C_{\text{notch}} + C$ from the C(T) specimen mesh. The total crack length at each result set is $C_{\text{notch}} + C + \delta_C$. This calculation can be done in one of the data sheets, or in another spreadsheet. For a growing crack, paste the crack length values for each analysis step into the third column in the ASTM J calculator form.

Compare J results

The J results for the two C(T) mesh examples are compared in the figures below.

The quarter symmetric C(T) mesh with a static crack front (focused crack mesh) gives good comparison between J from the FEA results and the ASTM J at $y=T/2$ (mid thickness, symmetry plane). The J from FEA results at $y=0$ (top surface) show 3D effects and do not agree with the ASTM J values.

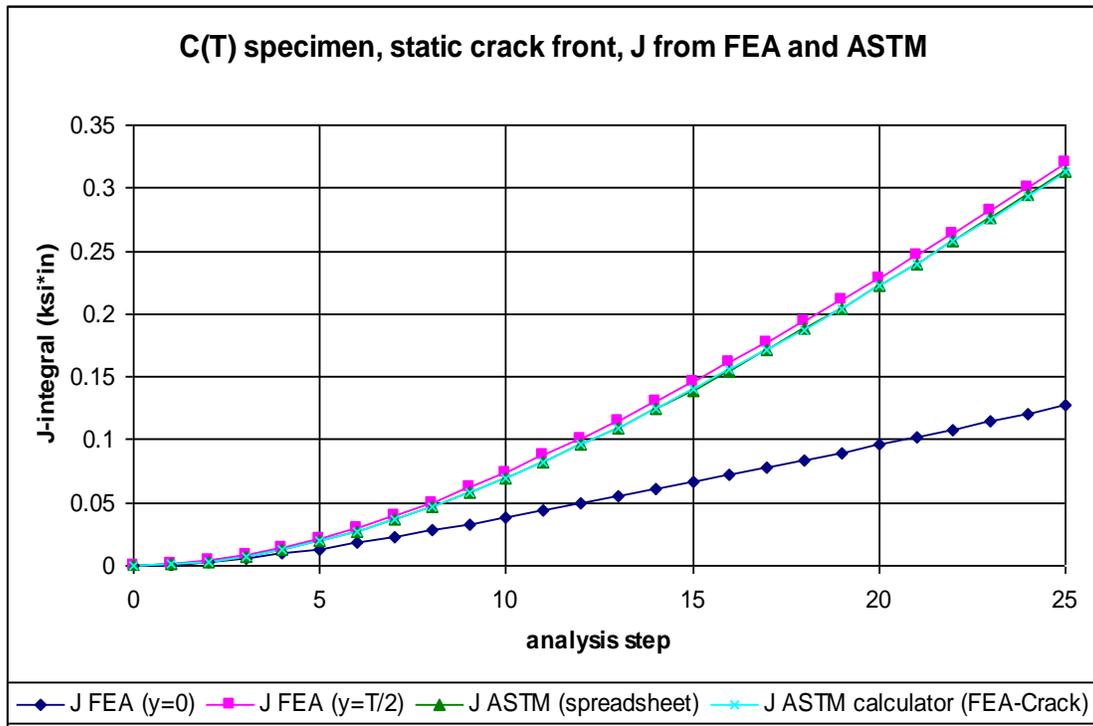


Figure 264 J results for the quarter symmetric C(T) specimen with a static crack front

The half symmetric C(T) mesh with cohesive material on the crack plane has a growing crack. The ASTM J results are between the J from FEA results at $y=0$ and $y=T/2$; see the figure below. The $y=0$ and $y=T$ results are reported in the Crack Results form. The $y=T/2$ J results can be obtained from the extracted crack results file.

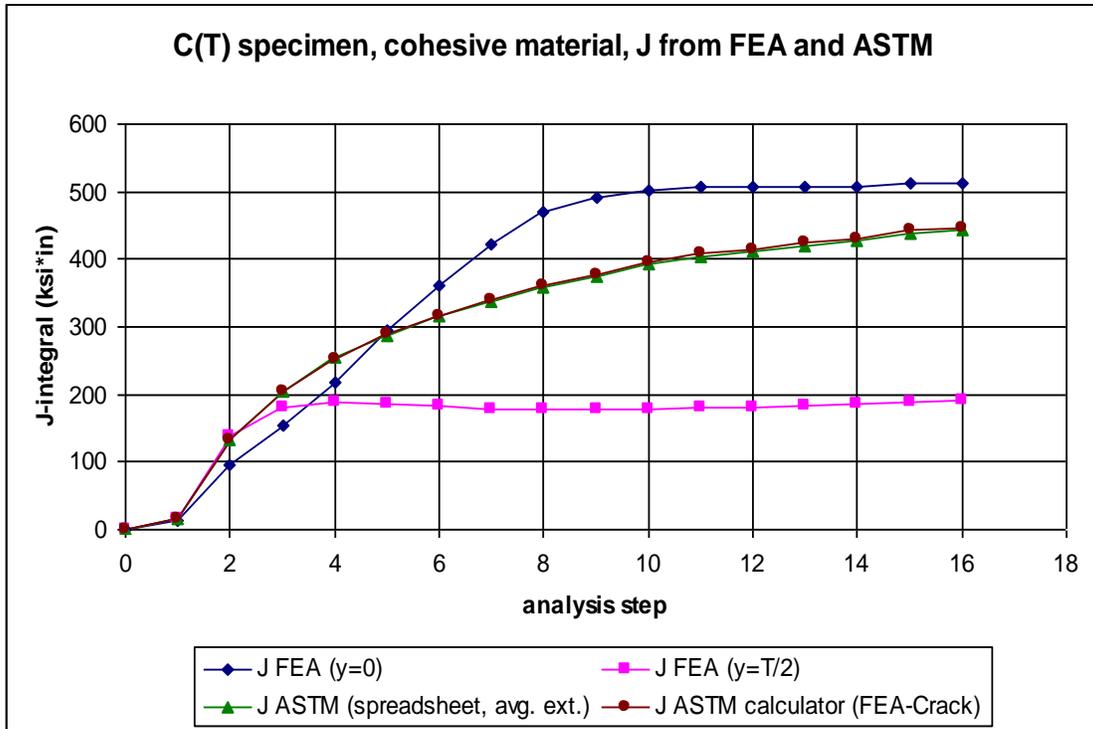


Figure 265 J results for the half symmetric C(T) specimen with cohesive material and a growing crack front

SE(B) Specimen Geometry

The single-edge bend SE(B) test specimen geometry is available in the *FEACrack* geometry library. The SE(B) specimen crack mesh geometry follows the ASTM E1820 specification. The focused mesh (standard or PRO) and the various cell meshes are available for the SE(B) specimen.

The SE(B) specimen is selected from the “Structural Shape menu” on the “Geometry/Configuration form” (toward the bottom of the menu). The SE(B) specimen mesh can be generated with three symmetry choices: quarter, left half, or the full mesh. The quarter and half symmetry meshes can be located in the +Z direction for node release or Gurson crack growth analysis as required by *WARP3D*.

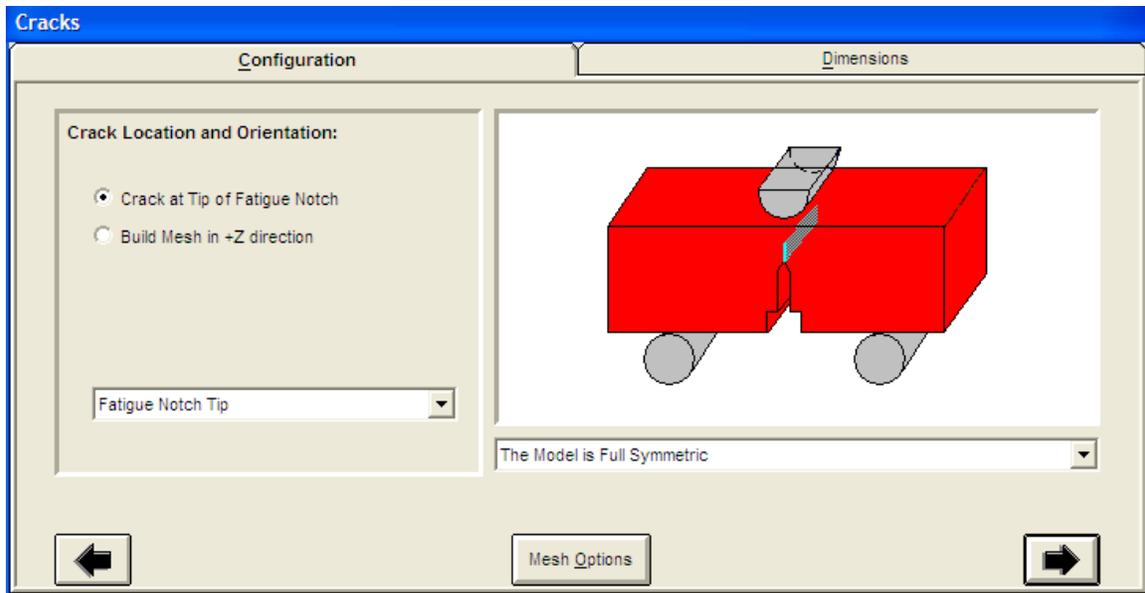


Figure 266 SE(B) specimen

The crack length, c , is measured from the end of the notch to the crack front. The total ASTM crack length, a_0 , is the notch length, C_{notch} , plus the crack length, c . All of the SE(B) specimen dimensions can be individually entered, or enter just the W and C_{notch} dimensions and the other dimensions are set using the ASTM E1820 defaults. When using the quarter symmetry, enter half the thickness, $B/2$, in the “Geometry/Dimensions form”.

The PRO mesh parameters control the mesh region around the crack front. It may be possible to add more mesh controls for other regions of the SE(B) mesh, such as the number of elements in the notch.

The boundary conditions are applied to the top edge of the mesh where the ram would strike. The ram radius is available as a variable to control the surface area used for the imposed load. The diameter of the support rollers is also a variable to allow the user to control the area of the applied boundary conditions. The forces or displacements from the ram should be applied in the $-X$ to bend the bar and open the crack.

SE(B) Zero Size Load and Ram Pins

The SE(B) model boundary conditions are available for zero width constraints and loads can be selected. The Roller Diameter, D , and the Ram Radius, R , can be set to zero to specify a zero width constraint size in the Geometry/Dimensions form. A single line of nodes across the mesh thickness are selected for the constraint or load applied to the SE(B) mesh. Either or both constraint zone sizes can be set independently to zero or a finite size.

An optional second material group can be activated at the boundary condition locations in the Geometry/Configuration form. The elements in the second material group are chosen if they are connected to any of the nodes in the constraint or load. A zero size constraint will have elements on each side of the constraint in the second material group. A finite width constraint will include

more elements in the second material group. For elastic-plastic analysis the second material group can be set to elastic material to reduce the indentation of the mesh at the constrained and loaded nodes. Either set a higher yield stress value for the second material group, or select the “Assume Elastic Behavior” option at the top of the Material Properties/Load Location Material form. The primary model material type for the analysis is still set in the Material Properties/Material Model Type form, but individual material groups can be set to elastic properties.

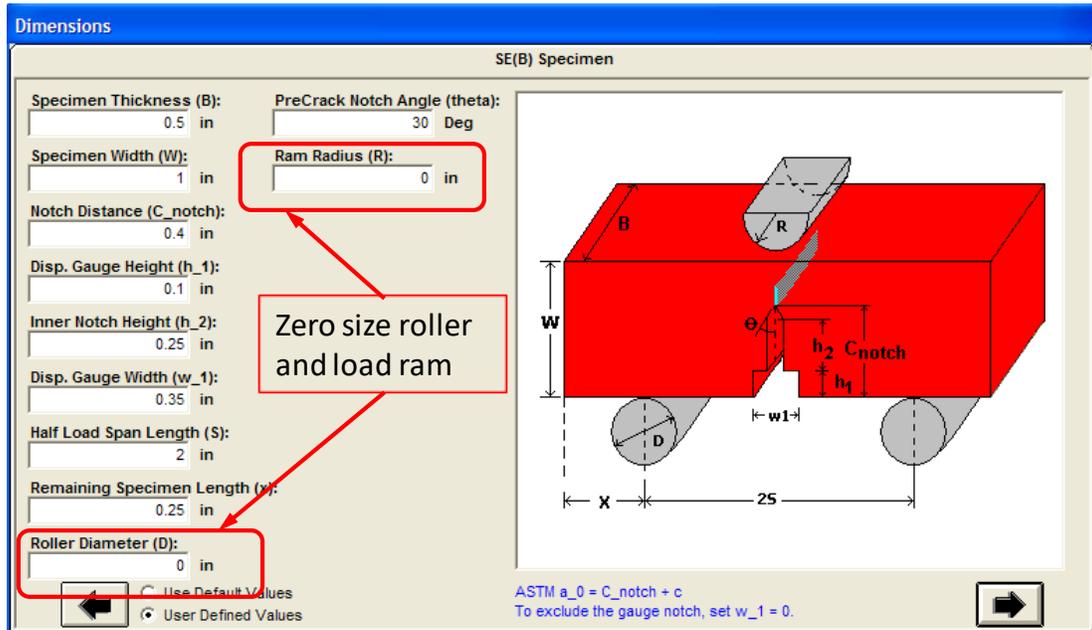


Figure 267 Enter a zero size roller diameter or load ram radius for the SE(B)

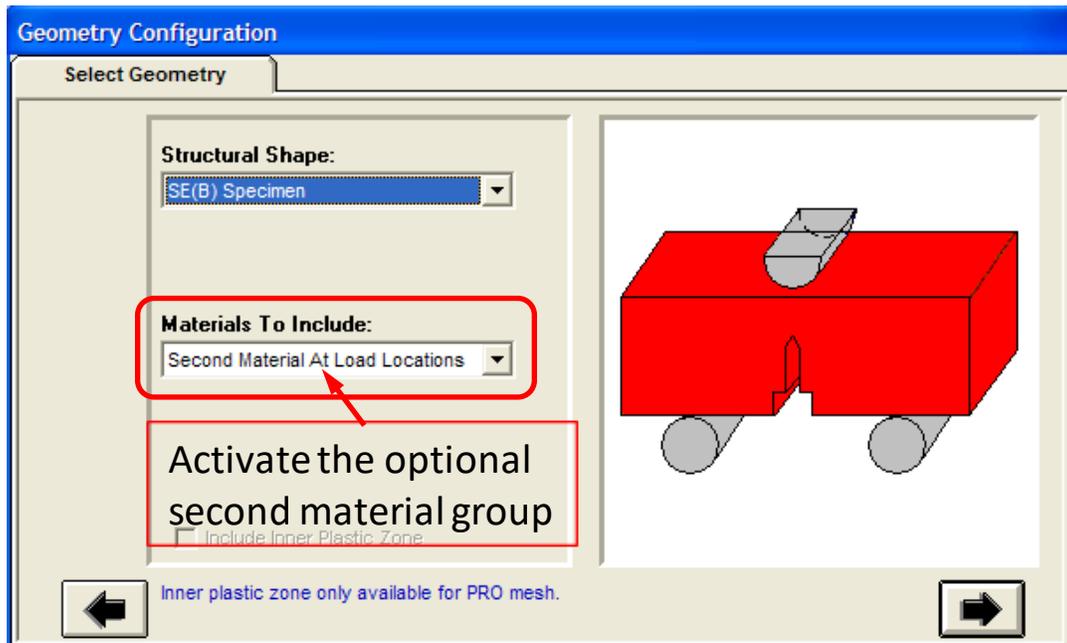


Figure 268 Activate the optional second material group at the boundary constraint and load locations

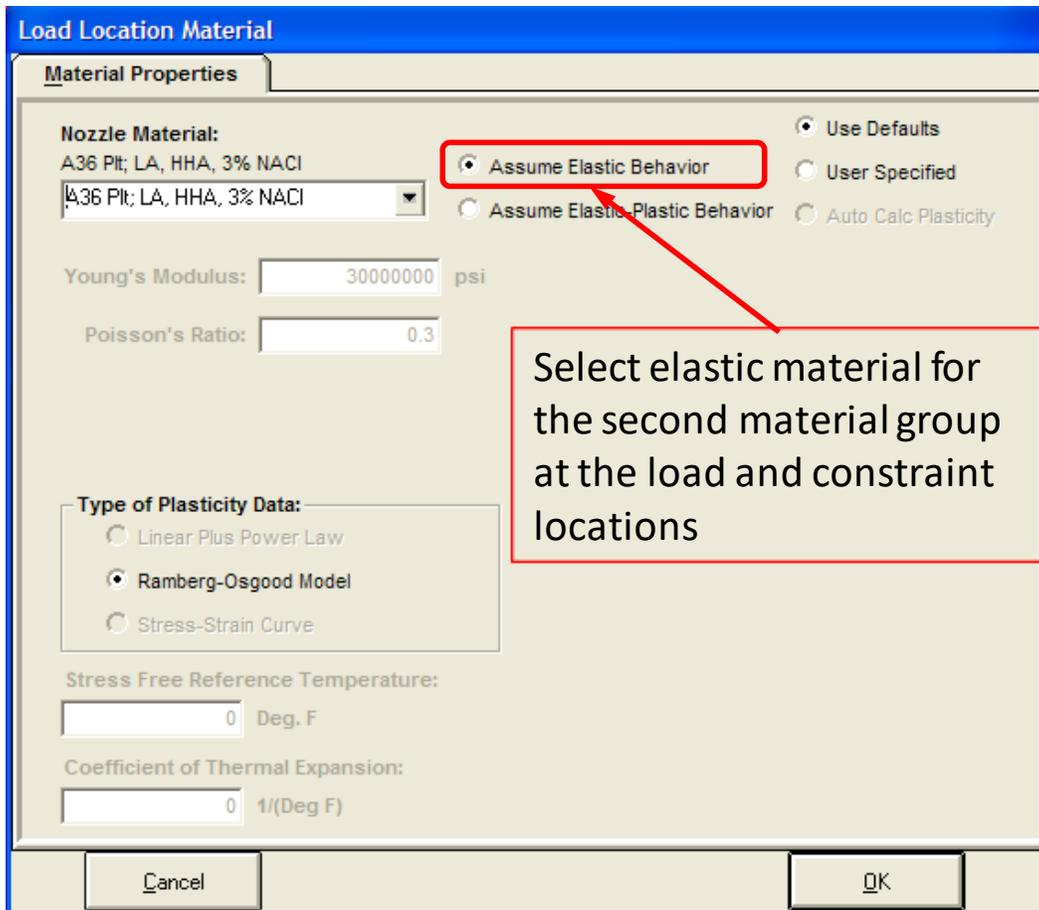


Figure 269 Elastic material can be selected for the SE(B) second material group at the constraint and load locations

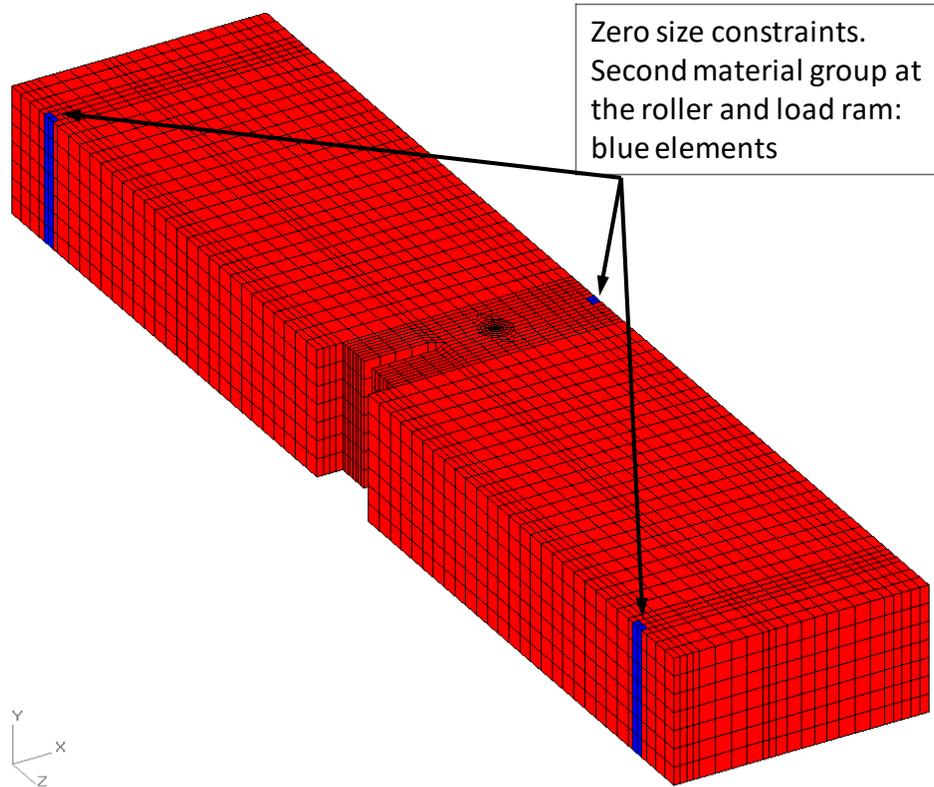
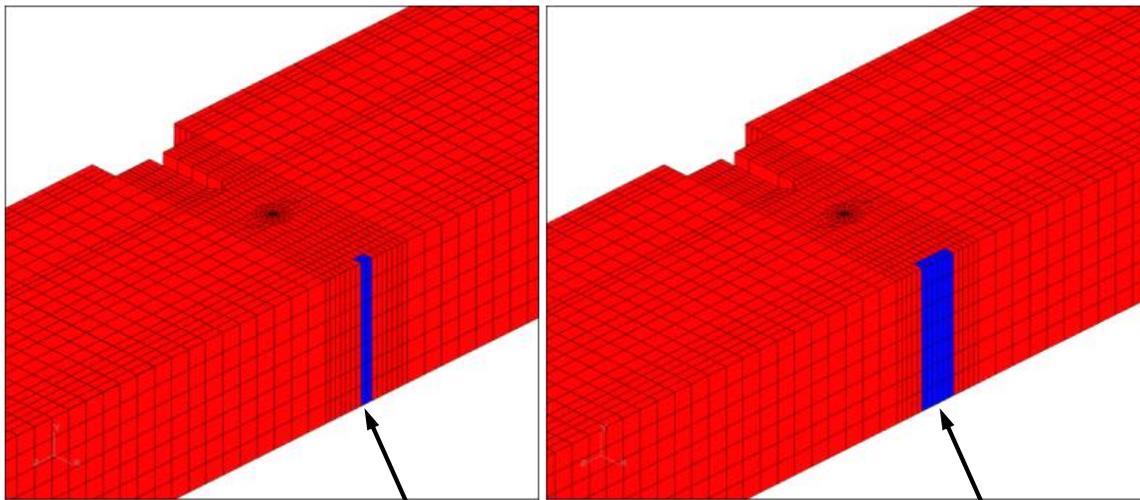


Figure 270 SE(B) mesh with zero size constraints at the rollers and load ram; the blue elements show the second material group



Zero width load ram radius

Finite width load ram radius

Figure 271 Comparison of the zero width and finite width load ram radius and the number of elements in the second material group

SE(B) Ram Load Pin force option

The SE(B) ram load pin x-direction force is applied to the mesh by equivalent element face pressures by default. Using element face pressure within the ram load pin region allows for varying element sizes, such as using the crack front element size bias option in the PRO mesh. Since this is the default, the user only needs to enter the x-direction ram force in the Boundary Conditions form.

The other option is to use distributed nodal forces to apply the ram load pin force. This option is activated using key word text in the Home/Notes form. The key word text is:

*use SEB nodal forces for ram load pin

SE(B) PRO Mesh Options

When using the PRO mesh for the SE(B) model, additional SE(B) specific PRO mesh controls are available by entering key word text in the Home/Notes form. Any of the mesh options can be used, and any not defined will use a default value to generate the SE(B) mesh. As with other “*use” key word syntax, each option is separated by a semi-colon “;”. An example is shown in the figures below.

- Available SE(B) specimen prototype PRO mesh options key word text:
 - *use SEB num elem span = <integer value>
 - *use SEB depth bias = <decimal value>
 - *use SEB num elem small notch = <integer value>
 - *use SEB num elem precrack notch = <integer value>
 - *use SEB num elem caliper notch = <integer value>
 - *use SEB num elem support = <integer value>
 - *use SEB num elem end = <integer value>
- Enter an integer value for the number of elements in each mesh zone
- The element size bias along the SE(B) depth z-direction can be a decimal value greater than zero



Units: <input type="text" value="lb - in - psi"/>	Continue <input type="button" value="➔"/>
Analysis Type: <input type="text" value="Focused Crack Mesh For J-Integral Analysis (Standard)"/>	
Analysis Name: <input type="text" value="SE(B) specimen"/>	Model Label: <input type="text" value="SEB01"/>
Notes: <input type="text" value="*use SEB num elem span = 15; *use SEB depth bias = 2.8; *use SEB num elem small notch = 2; *use SEB num elem precrack notch = 7; *use SEB num elem caliper notch = 5; *use SEB num elem support = 2; *use SEB num elem end = 3"/>	

Figure 272 Additional SE(B) specific PRO mesh options

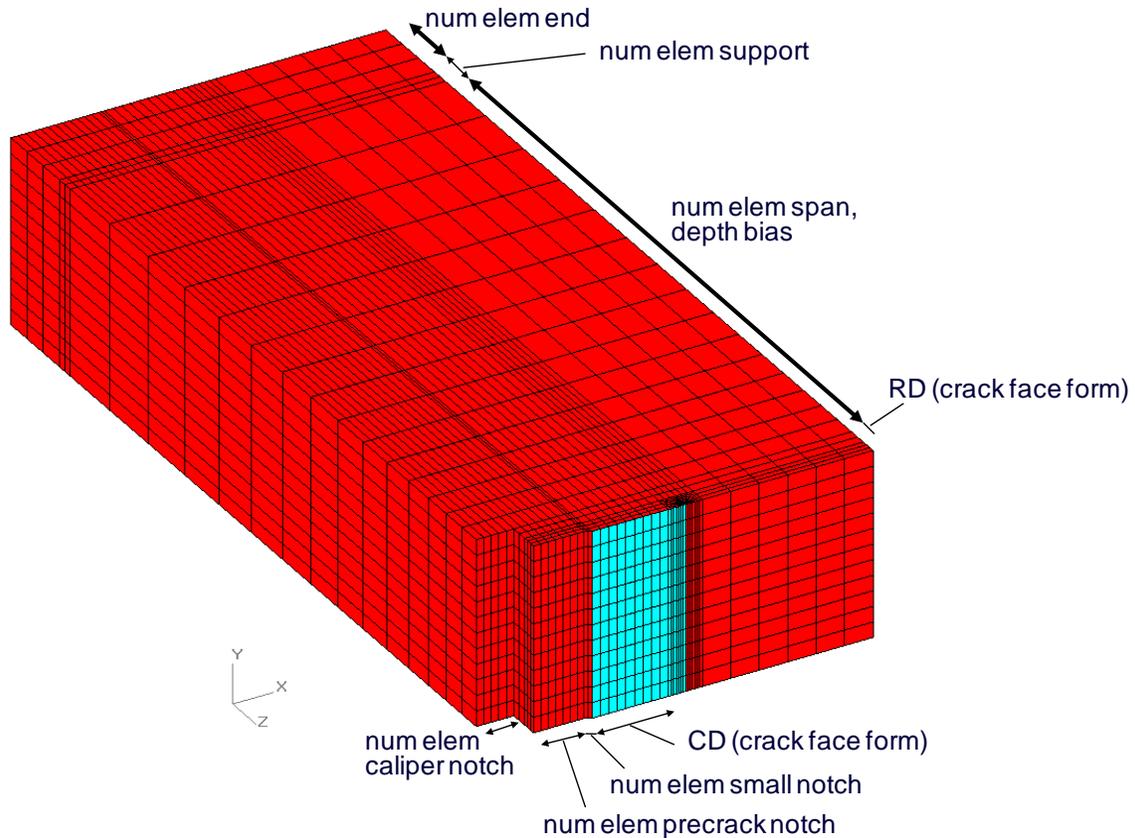


Figure 273 Locations of the SE(B) specific PRO mesh options; the RD and CD options are standard PRO mesh options and are located on the Crack Face form

SE(B) Side Groove

The SE(B) side groove is located in the crack plane along the specimen width x-direction. The SE(B) side groove option is activated using key word text in the Home/Notes form. An example is shown in the figure below.

- The SE(B) side groove options are entered as key word text in the Home/Notes form:
- *use SEB side groove option
- *use SEB side groove width = <Wg value>
- *use SEB side groove thickness Bn = <Bn value>

Where the Wg and Bn values are entered as decimal values, and are greater than zero. Each key word parameter is separated by a semi-colon “;”. Wg is the half groove width measured from the crack plane, and Bn is the ligament thickness between the top and bottom side groove; see the figure below.

FEACrack

Units:
lb - in - psi

Continue
➔

Analysis Type: Focused Crack Mesh For J-Integral Analysis (Standard)

Analysis Name: SE(B) side groove **Model Label:** SEB

Notes:
*Use SEB side groove option; *use SEB side groove width $W_g = 0.08$; *use SEB side groove thickness $B_n = 0.84$

Figure 274 Activate the SE(B) side groove option using key word text in the Home/Notes form

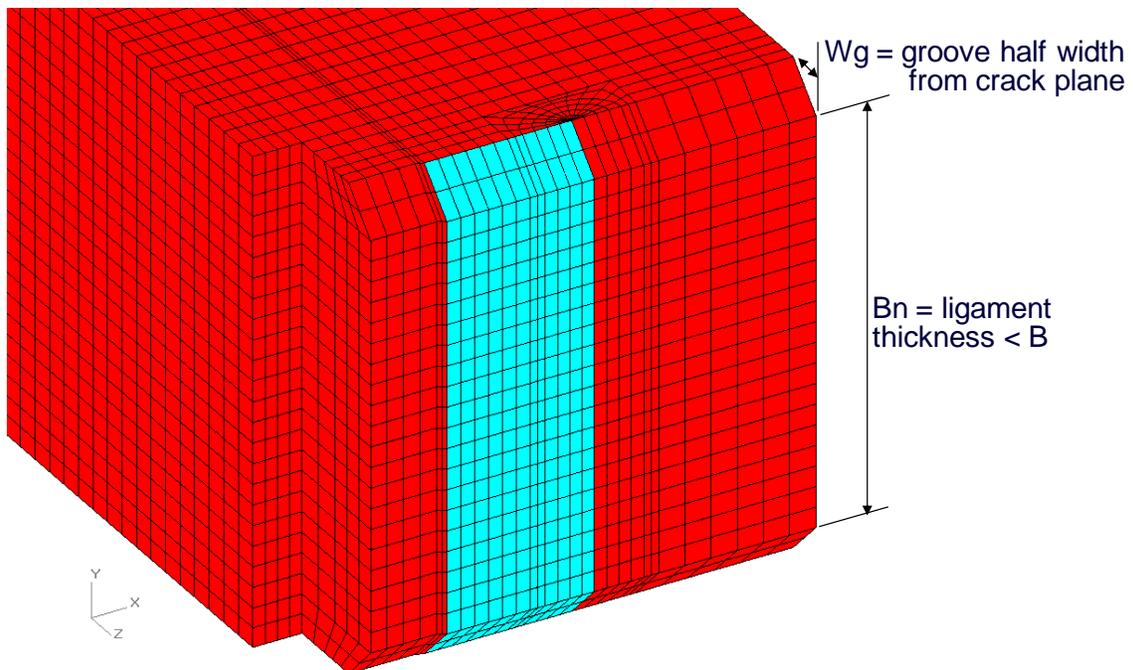


Figure 275 SE(B) side groove option locations for the groove half width and ligament thickness

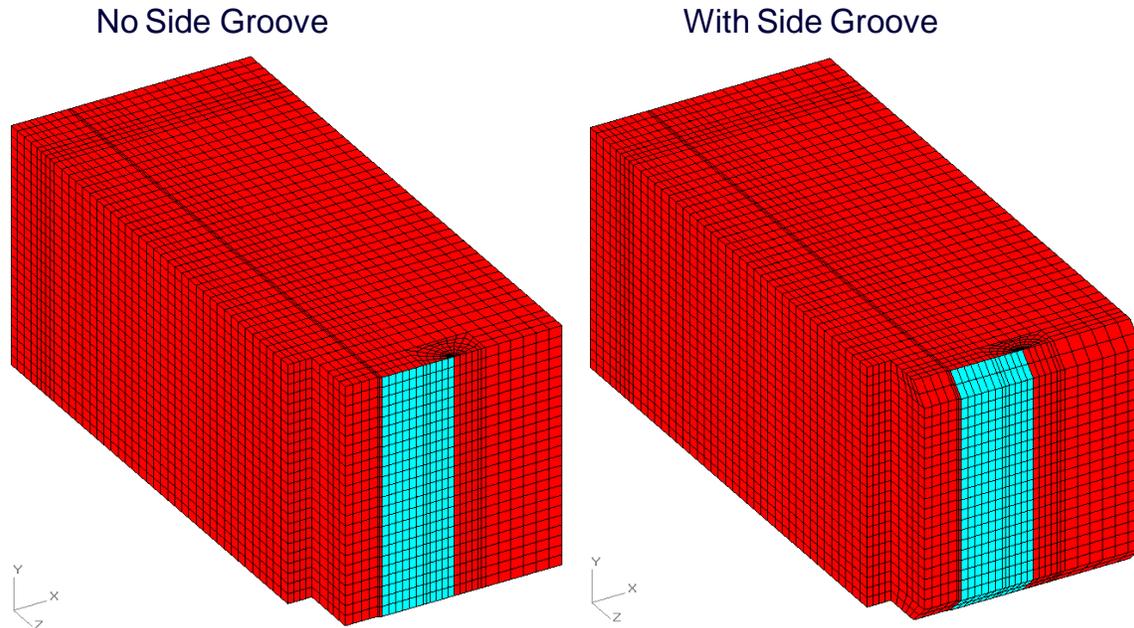


Figure 276 Comparison of the SE(B) model with and without the side groove option

SE(B) Post Processing

For post processing, pre-defined node sets contain nodes on the notch edges for easy selection of particular node displacement and reaction force results. For example, select load line nodes to create a load-displacement curve, or obtain values to compute J using the ASTM E1820 equations.

To obtain specific node or node set results, such as crack mouth opening displacements (CMOD), use the “Select menu”. After loading the results file, go to the “Select/Select Nodes menu” to open the form to select node sets (or individual nodes) to extract result values to the spreadsheet and x-y plots. In the “Selected Nodes pull-down menu” several node sets are available by name, for example “Load Disp nodes Y=0”. When one of the node sets is selected the nodes in the set are highlighted in the mesh by yellow dots. Several load line node sets are available at various locations through the mesh thickness. Click the Show Data button to have the selected node results extracted and put into the spreadsheet. There will be one row of data per node in the set, and one page per result step. To get an x-y plot of the node results, click the Show Variable Selections button at the bottom of the spreadsheet, and then select the result values for the x-axis and y-axis of the plot. For example, select the “Delta Z per Step” for the plot x-axis (the node delta Z displacement results for all the result steps) and select the “Force X per Step” (the node reaction force X results for all the result steps). After selecting the node result values click the OK button and then click on the Plots tab (top of the form); then select individual node check boxes to add the result curves to the plot. To get the plot data to another program such as an *EXCEL* spreadsheet, right click on the plot and select copy in the pop-up menu, then in the separate *EXCEL* spreadsheet use paste to put the columns of data from the node result plot, or use paste special to put the plot picture.

The load-displacement curve created from the extracted results from the *FEACrack* post processing is shown in the plots below. The results are from a left half symmetric SE(B)

specimen example with elastic-plastic material (available for download from our web site)
 Notice that the extracted result values are **negative since the left half of the SE(B) specimen is displacing in the $-Z$ direction**. Copying the plot values to an *EXCEL* spreadsheet allows the more customary load-displacement curve to be created. The X reaction force from the node in the load pin is used as the load value, and twice the delta Z from the node at the corner is used for the total CMOD.

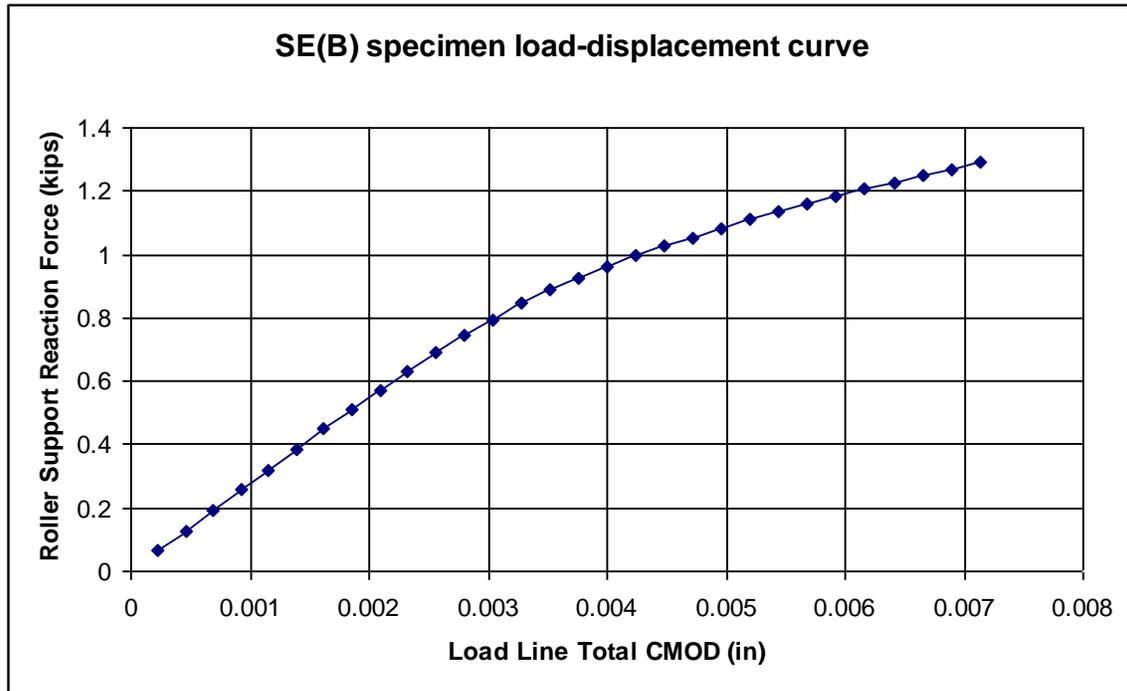


Figure 277 Load-displacement curve using the extracted node results.

An ASTM J calculator to use the reaction force, load line displacement, and crack growth extension results to compute J according to the ASTM E1820 equations is available in Tools menu in *FEACrack*.

C(T) Specimen

The compact tension C(T) test specimen geometry is available in *FEACrack* geometry library. The C(T) specimen crack mesh geometry follows the ASTM E1820 specification. The focused mesh (standard or PRO) and the various cell meshes are available for the C(T) specimen.

The C(T) specimen is selected from the Structural Shape menu on the Geometry/Configuration form (toward the bottom of the menu). There is an option to include or omit the load pin from the mesh. If the load pin is included in the mesh, the boundary conditions (forces or displacements) are applied to the load pin center-line. If the load pin is omitted from the mesh, the boundary conditions are applied to the inside of the load pin hole. The load pin can have different material properties than the specimen, for example to keep the load pin elastic but have elastic-plastic material for the specimen. To model the load pin interaction with the specimen, only the nodes around the top half of the load pin are merged with the specimen nodes; this allows the load pin to separate from the bottom of the hole as loading is applied and the mesh deforms. This allows

the load pin to be included in any of the FEA input files. Future load pin options may include the use of contact surfaces between the load pin and the specimen in an *Abaqus* analysis. An *Abaqus* analysis would also permit the use of rigid elements or rigid surfaces in the load pin.

The C(T) specimen mesh can be generated with three symmetry choices: quarter, top half, or the full mesh. The quarter and half symmetry meshes can be located in the +Z direction for node release or Gurson crack growth analysis as required by *WARP3D*.

The crack length, c , is measured from the end of the notch to the crack front. The total ASTM crack length, a_0 , is the notch length, C_{notch} , plus the crack length, c . All of the C(T) specimen dimensions can be individually entered, or enter just the W , B , C_{notch} dimensions and the other dimensions are set using the ASTM E1820 defaults. When using the quarter symmetry, enter half the thickness, $B/2$, in the Geometry/Dimensions form.

The PRO mesh parameters control the mesh region around the crack front. It may be possible to add more mesh controls for other regions of the C(T) mesh, such as the number of elements in the notch and load pin.

The boundary conditions are applied to the load pin or the inside of the load pin hole. The forces or displacements should be applied in the $-Z$ direction for the top load pin and in the $+Z$ direction for the bottom load pin to open the crack.

When viewing the C(T) specimen crack mesh, a pre-defined view has been added at the bottom of the View Type menu (left side of the mesh view), called “C(T) Specimen”. The view rotation angles for this pre-defined view are set to: 75° (x-axis), 0° (y-axis), 330° (z-axis). A mesh using the C(T) Specimen pre-defined view is shown below.

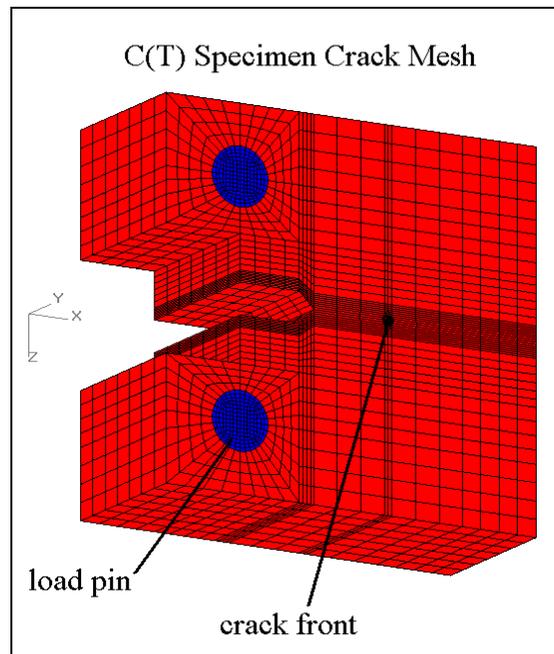


Figure 278 Crack mesh using the pre-defined “C(T) Specimen” View Type.

For post processing, pre-defined node sets contain nodes along the load line for easy selection of particular node displacement and reaction force results. For example, select load line nodes to

create a load-displacement curve, or obtain values to compute J using the ASTM E1820 equations. The load line is defined between the centers of the load pins in the Z direction.

After loading the results file, go to the Select/Select Nodes menu to open the form to select node sets (or individual nodes) to extract result values to the spreadsheet and x-y plots. In the Selected Nodes pull-down menu, several node sets are available by name, for example “Load Disp nodes Y=0”. When one of the node sets is selected the nodes in the set are highlighted in the mesh by yellow dots. Several load line node sets are available at various locations through the mesh thickness. Click the “Show Data button” to have the selected node results extracted and put into the spreadsheet. There will be one row of data per node in the set, and one page per result step. To get an x-y plot of the node results, click the “Show Variable Selections button” at the bottom of the spreadsheet, and then select the result values for the x-axis and y-axis of the plot. For example, select the “Delta Z per Step” for the plot x-axis (the node delta Z displacement results for all the result steps) and select the “Force Z per Step” (the node reaction force Z results for all the result steps). After selecting the node result values click the OK button and then click on the Plots tab (top of the form). Then select individual node check boxes to add the result curves to the plot. To get the plot data to another program such as an *EXCEL* spreadsheet, right click on the plot and select copy in the pop-up menu. Then in the separate spreadsheet use paste to put the columns of data from the node result plot, or use paste special to put the plot picture.

The extracted load line results from the *FEACrack* post processing and the load-displacement curve created from the extracted results are shown in the plots below. The results are from a top half symmetric C(T) specimen example with elastic-plastic material (available for download from our web site) Notice that the extracted result values are negative since the top half of the C(T) specimen is displacing in the $-Z$ direction. Copying the plot values to a spreadsheet allows the more customary load-displacement curve to be created. The Z reaction force from the node in the load pin is used as the load value, and twice the delta Z from the node at the corner is used for the total CMOD.

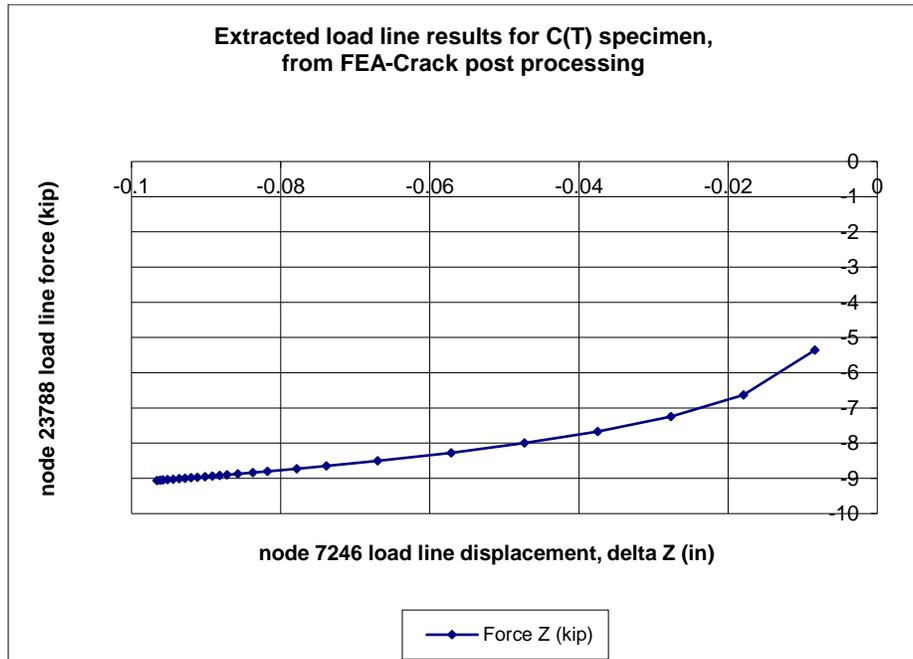


Figure 279 C(T) specimen extracted node force and displacement results.

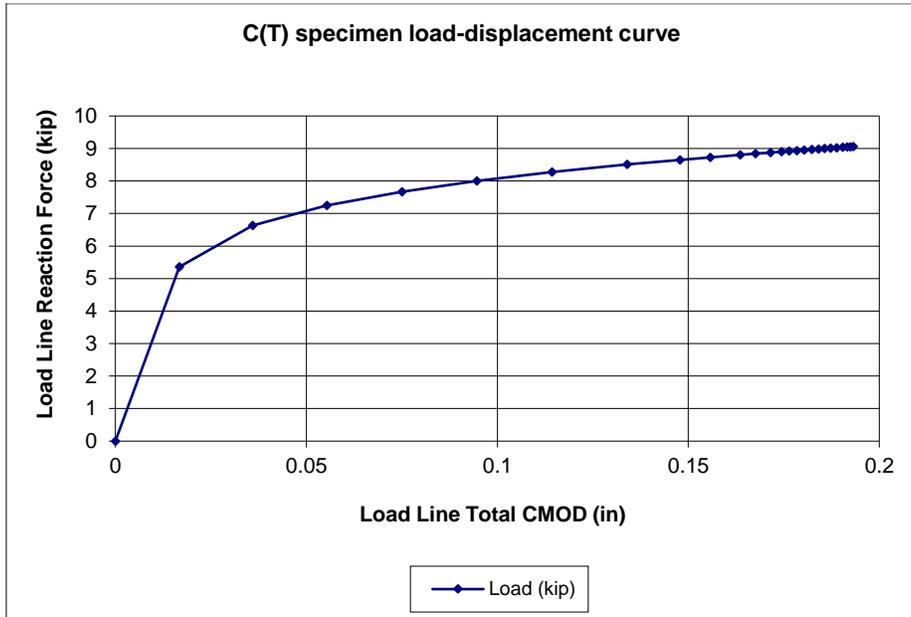


Figure 280 Load-displacement curve using the extracted node results.

Also see the ASTM J calculator in the Tools menus, which uses the reaction force, load line displacement, and crack growth extension results to compute J according to the ASTM E1820 equations.

C(T) Specimen Side Groove

The C(T) specimen geometry has a side groove option, which is selected in the Geometry/Crack/Configuration form, shown in the figure below.

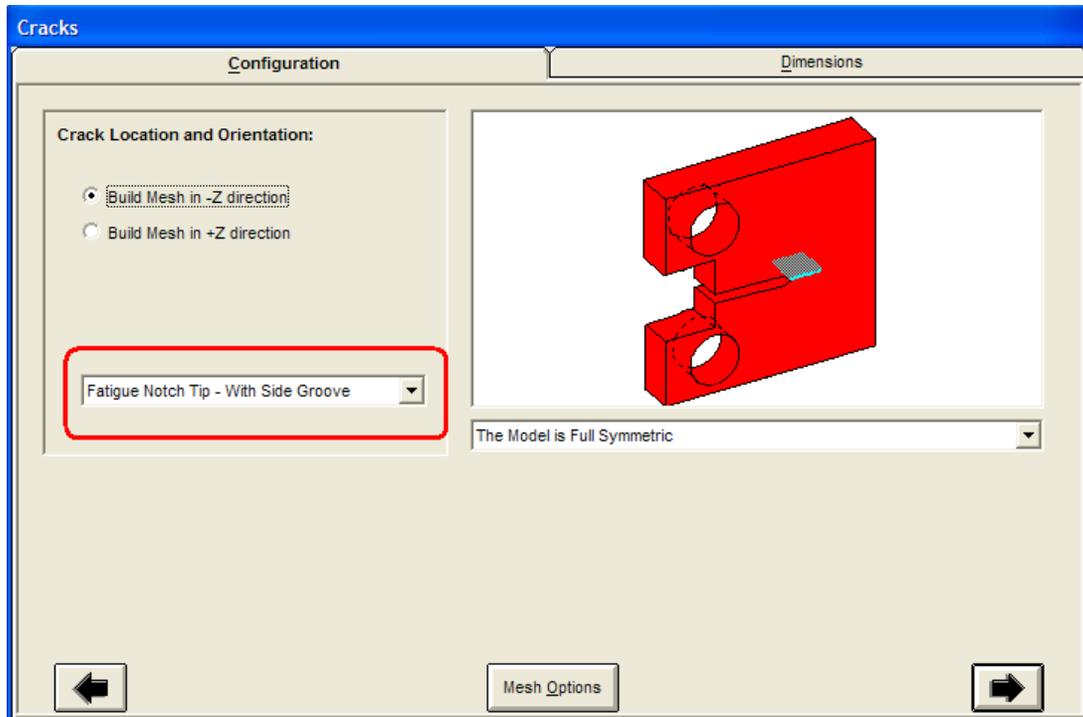


Figure 281 Select the C(T) specimen side groove option in the Geometry/Crack/Configuration form

The side groove height is set by the h_g parameter in the Geometry/Dimensions form. The “User Defined Values” option will need to be selected at the bottom of the form to activate the h_g input and other custom dimensions. The side groove height can vary from the precrack notch height h_4 to the main notch height h_3 . If the given side groove height is less than h_4 or greater than h_3 , a default value will be used to generate the C(T) specimen mesh. See the figure below.

The side groove thickness B_n can be less than or equal to the specimen thickness B . This allows the side groove mesh to be selected to generate the C(T) specimen with or without an actual side groove.

The precrack notch can be deactivated by setting h_4 or θ to zero. The main notch can be deactivated by setting h_3 to zero. And both notches can be deactivated by setting those dimensions to zero.

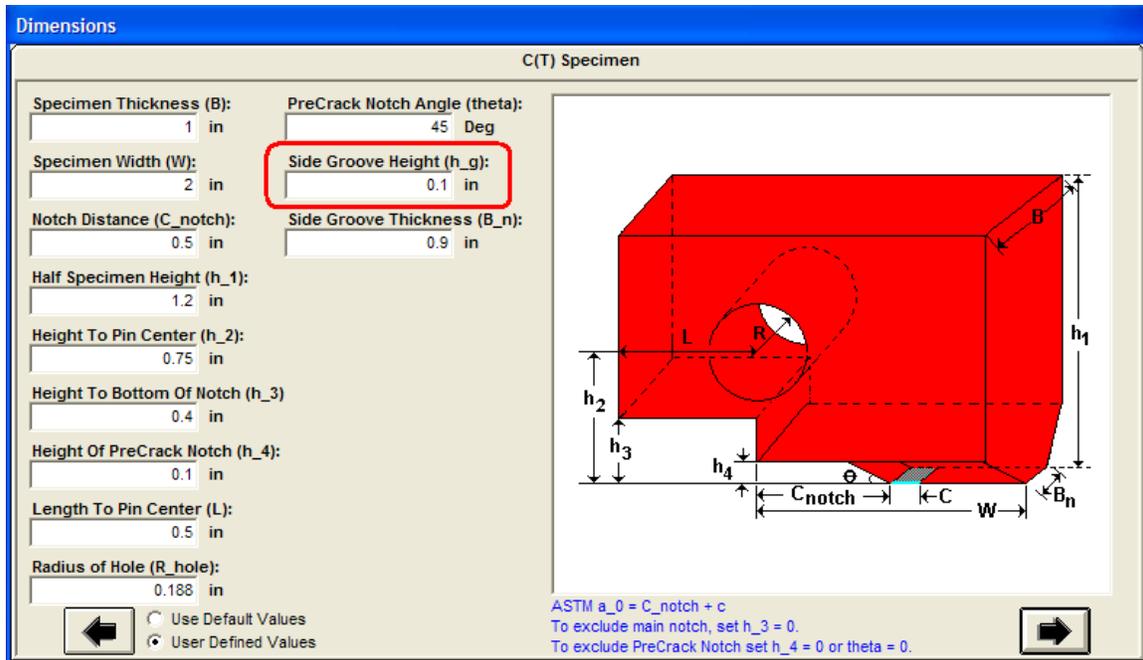


Figure 282 Select the User Defined Values option to activate the side groove height h_g input

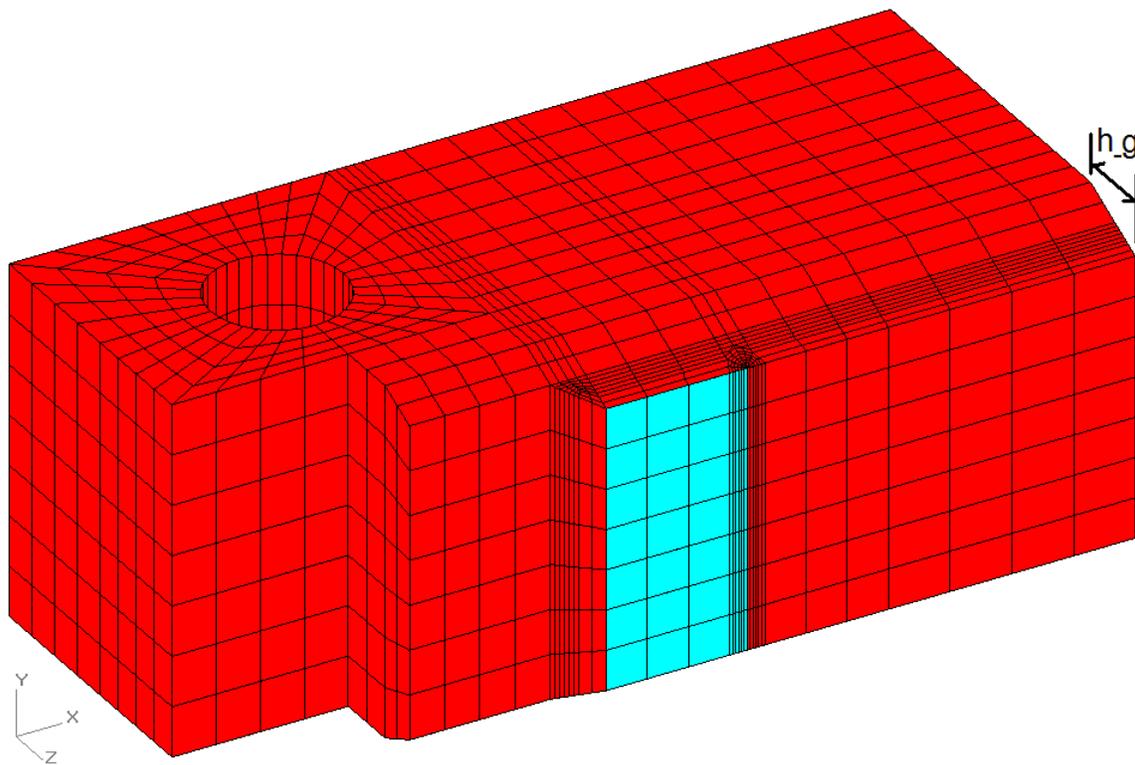


Figure 283 Select C(T) specimen half model with a side groove; dimension h_g is within: $h_4 \leq h_g \leq h_3$ main notch

Modified Boundary Layer Model

The modified boundary layer model (BLM) represents the crack front region in an infinite body and is used to compute the σ_{yy}/σ_0 normalized reference solution for the Q-stress difference field. Use the radial result calculator to extract the sigma_z values versus distance from the crack front.

The quarter and half symmetric models include the back half cylinder of the boundary layer model and include the back crack face. A much greater amount of mesh refinement is needed near the crack front to capture the stress gradient in the radial direction ahead of the crack, see the figures below. Use the PRO mesh controls to adjust the radial element size bias and the thickness size bias values.

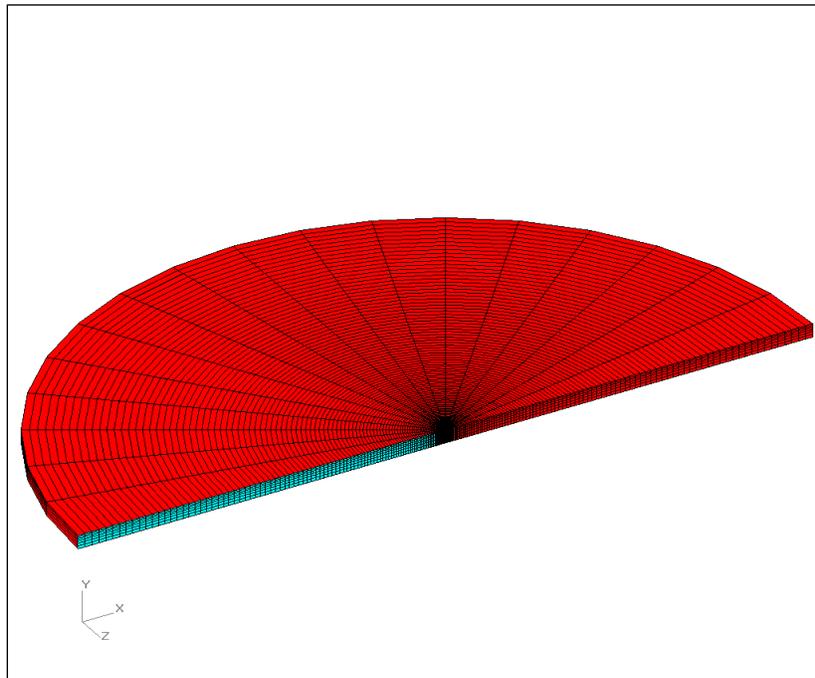


Figure 284 Modified boundary layer model, half symmetric mesh

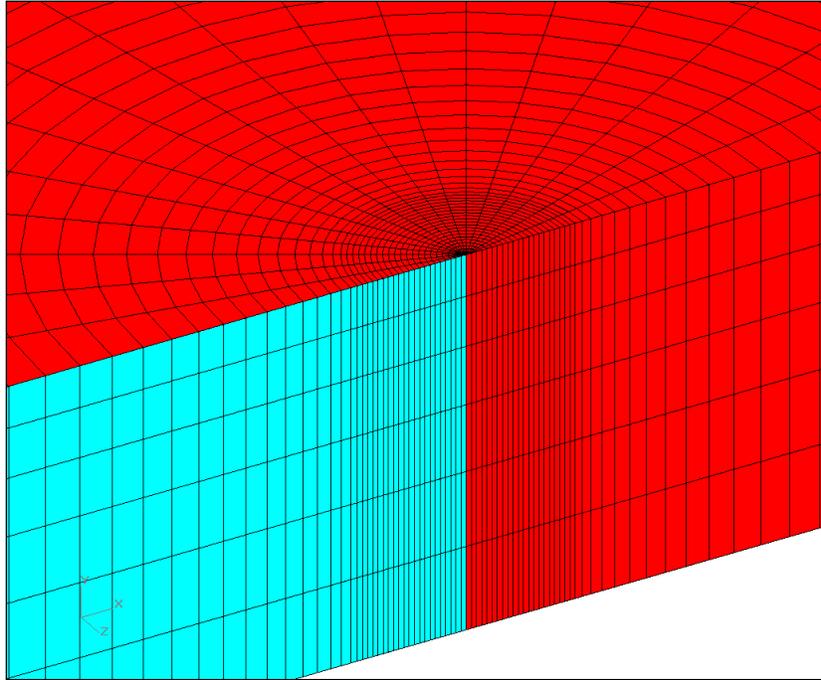


Figure 285 Close up of the uniform mesh size refinement near the crack front in the estimated plastic zone

To obtain more mesh refinement, the boundary layer model has two zones: the estimated plastic zone near the crack front, and the region outside the plastic zone to the edge of the model at radius R . In the PRO mesh controls enter the number of elements inside and outside the plastic zone, and enter the element size bias factors. There are separate radial size bias factors for the plastic zone and the region outside the plastic zone. The number of elements inside the plastic zone can be greater than the selected number of J-integral contours. The model thickness B and the estimated plastic zone radius R_p are entered in the “Geometry/Dimension” form, see the figure below. In the outer zone, 50 to 100 elements or more are recommended. The number of elements inside the plastic zone should be great enough to obtain the desired radial stress results ahead of the crack; usually 20 to 40 elements are sufficient.

The estimated radius of the plastic zone can be computed using the “Calculate Radius” form. Click the “Calculate Radius” button on the Dimensions form to open the calculator. Enter the applied far-field stress intensity K , and enter the reference (yield stress) value σ_0 from the material-stress-strain curve; the estimated plastic zone radius is computed in the “ $R_{plastic}$ ” text box. Click the “Use $R_{plastic}$ ” button to have the radius value entered in the dimensions form. Any plastic zone radius value can be entered to control the size of the high refinement region near the crack tip. The plastic zone radius must be less than the model radius: $R_p < c = R$.

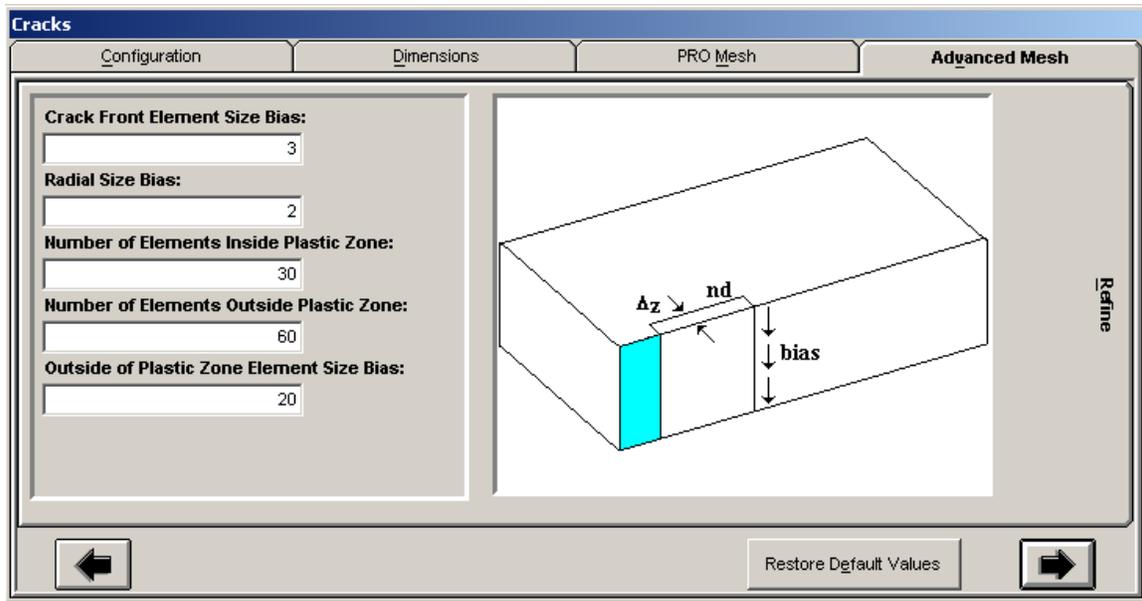


Figure 286 Enter the size bias factors, and the number of elements inside and outside the plastic zone

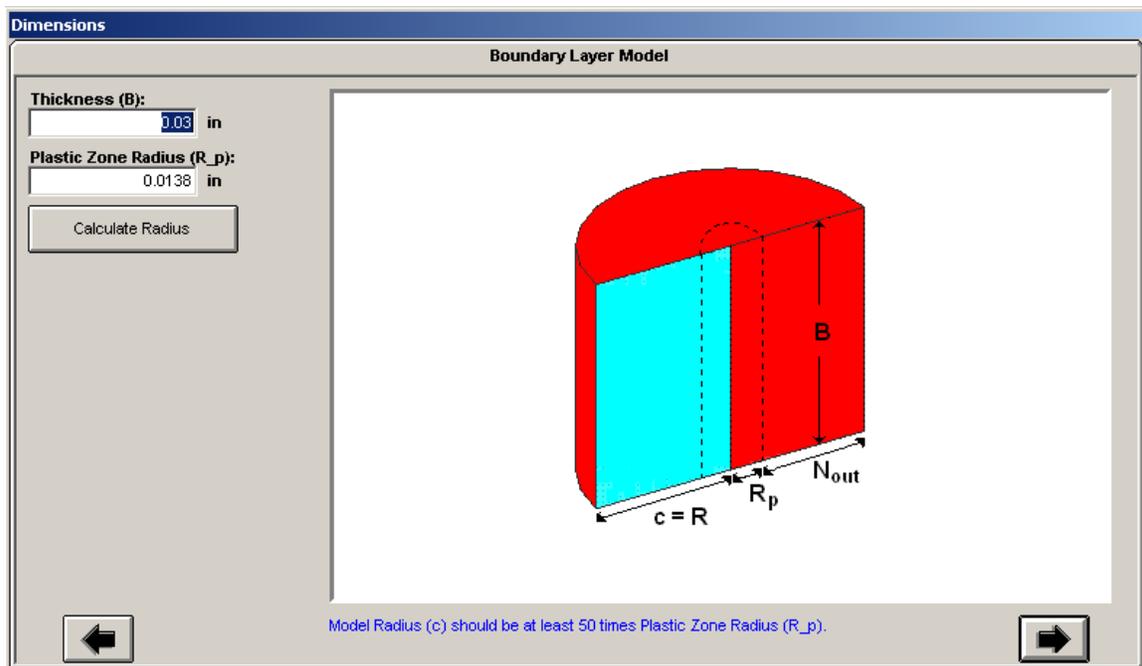


Figure 287 Enter the model thickness, B, and the estimated plastic zone radius, R_p

Calculate Plastic Zone Radius

$$R_{plastic} = \frac{0.25}{\pi} \left[\frac{K_{applied}}{\sigma_0} \right]^2$$

K_applied: 25000 **psi*in^{0.5}**

Sigma_0: 60000 **psi**

R_plastic: 0.0138 **in**

Cancel Use R_plastic

Figure 288 Estimated plastic zone radius calculator; enter K and σ_0

BLM Constraints: K and T-stress

The imposed node displacement constraints needed to simulate a given far-field stress intensity K and T-stress are computed using the K and T values entered in the boundary conditions form. The $u(R,\theta)$ displacement is in the global x-axis direction, and the $v(R,\theta)$ displacement is in the negative global z-axis direction on the mesh. Enter the applied K and T-stress values in the form and the corresponding displacements will be computed and applied around the outside surface (outer circular edge) of the mesh. The current boundary displacement equations are appropriate to model plane-strain or plane-stress conditions. Use the radio button on the constraints form to select either the plane-stress or plane-strain; the corresponding equation is displayed. Refer to the “Modified_boundary_layer_model_displacement_equations.pdf” document for the derivation of the $u(R,\theta)$ and $v(R,\theta)$ displacement equations.

Fixed constraints can also be applied to the top and bottom of the mesh (bottom of the half-symmetric mesh is available for constraint) to better model plane-strain conditions. For example, constrain the vertical y-axis displacement as a symmetry plane and avoid a free surface in the mesh.

A multi-step plasticity analysis must be used to obtain accurate crack plane stresses near the crack tip. The estimated plastic zone radius should be much smaller than the model radius. Typically the model radius should be 50 to 100 times larger than the plastic zone radius. A stress-strain curve or power-law should be entered in the material data form, and 20 to 30 analysis steps are recommended for accuracy and convergence. The given K and T values are applied as displacement constraints around the outer edge of the model, rather than stresses, to improve convergence of the elastic-plastic solution (displacement controlled plasticity analysis).

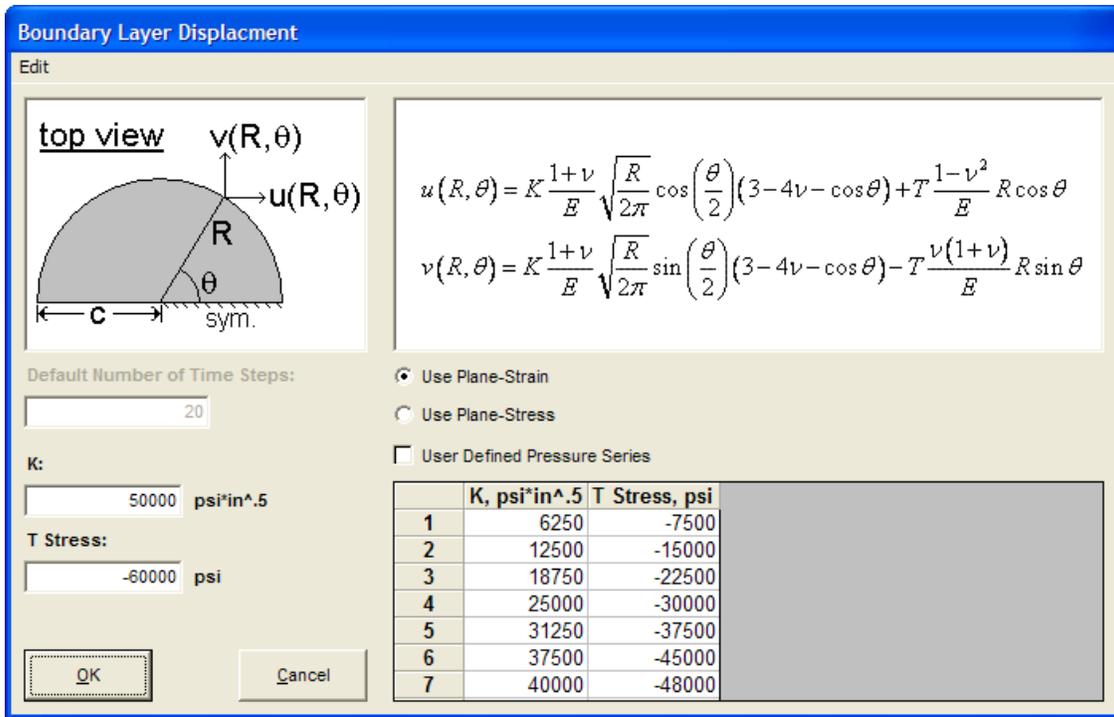


Figure 289 Enter K and T-stress to compute the imposed perimeter displacement for plane-strain

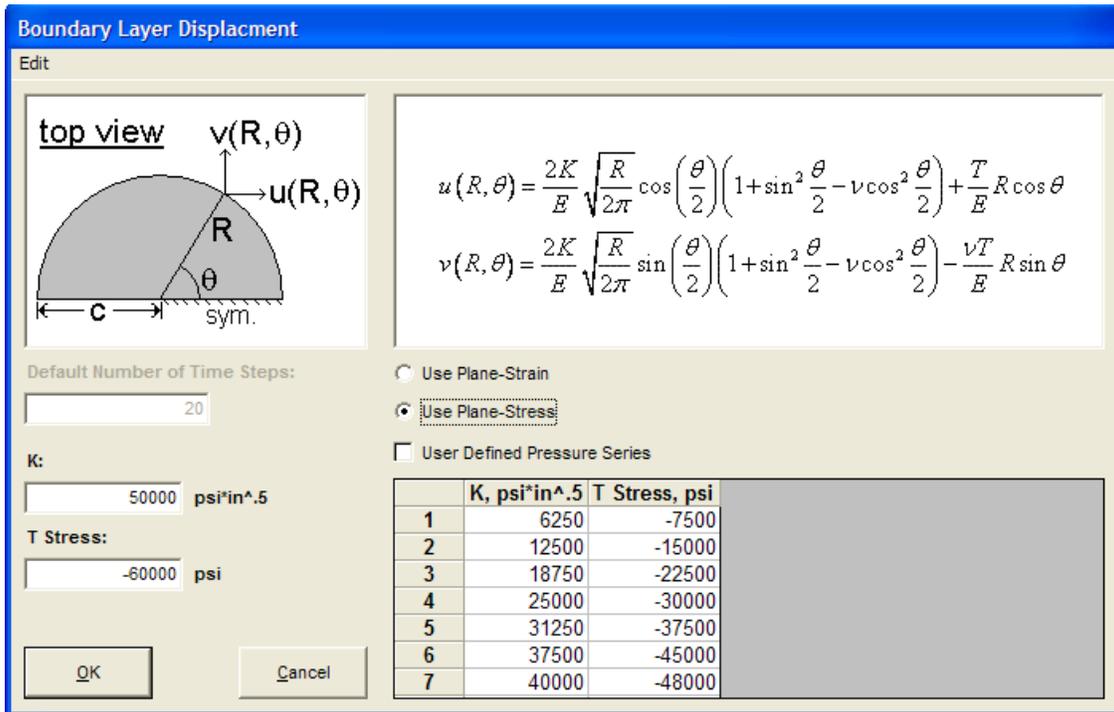


Figure 290 Enter K and T-stress to compute the imposed perimeter displacement for plane-stress

For the plane-stress case, if the top and bottom surface (half symmetry) have a y-constraint a warning message will be returned from the mesh generator, but the perimeter displacements will be applied to the BLM mesh, and the analysis could be run. For the quarter symmetry model, the

warning is given if the top surface has a y-constraint. The plane-strain case does not have this warning check.

BLM Cell Mesh

The standard boundary layer model (BLM) crack mesh uses the focused crack front mesh, which has a radial mesh pattern around the crack front shown below. The focused crack front mesh is suitable for computing the crack front J-integral, but not for crack growth analysis. The crack face is shown in light blue; there are symmetry constraints on the right side of the crack front in the X-Y plane.

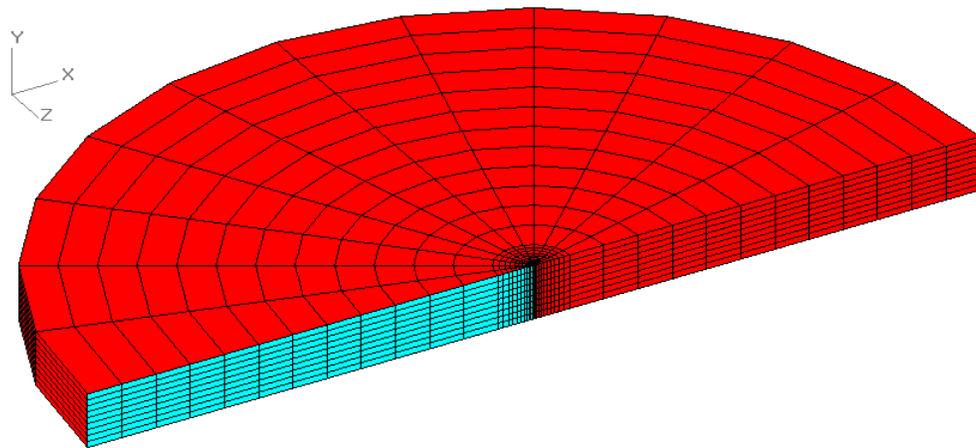


Figure 291 Example of the standard boundary layer model with a focused mesh

To model crack growth within a single finite element analysis a cell type crack mesh can be used, for example using the node release method in the Warp3D FEA solver. An example of a cell type BLM crack mesh is shown below. Remote domains around the cell zone can be used to compute the J-integral.

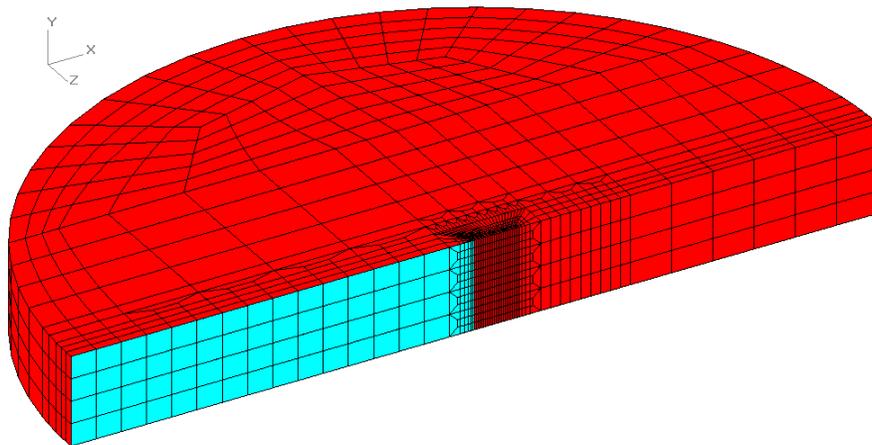


Figure 292 Example of the standard boundary layer model with a cell type mesh

The example shown above is a relatively coarse mesh to show how the flat plate region is across the front width of the BLM, and how the additional mesh zones are added behind the plate zone to create the circular BLM model. An example of a BLM with more mesh refinement and smaller size cell elements at the crack front is shown in the figures below, which zoom in toward the crack front. When using the cell mesh for node release crack growth, the Warp3D FEA solver requires that the mesh be on the +Z side of the X-Y plane, which is a mesh option.

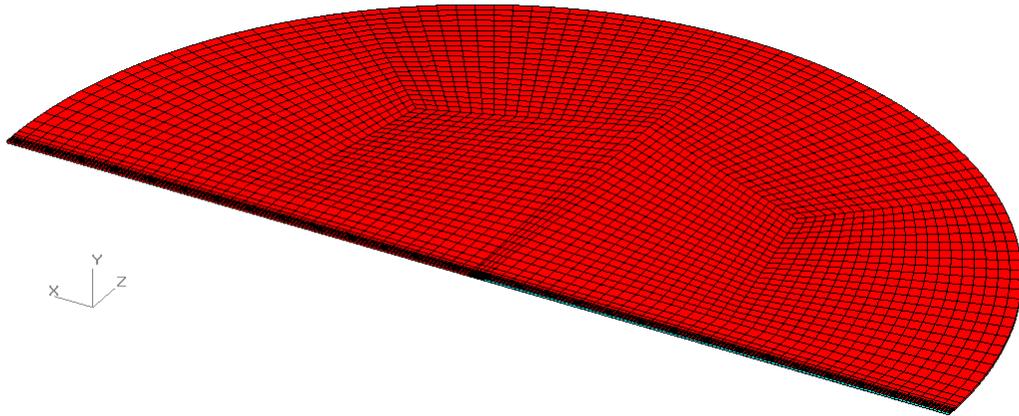


Figure 293 BLM with more mesh refinement and smaller cell element size

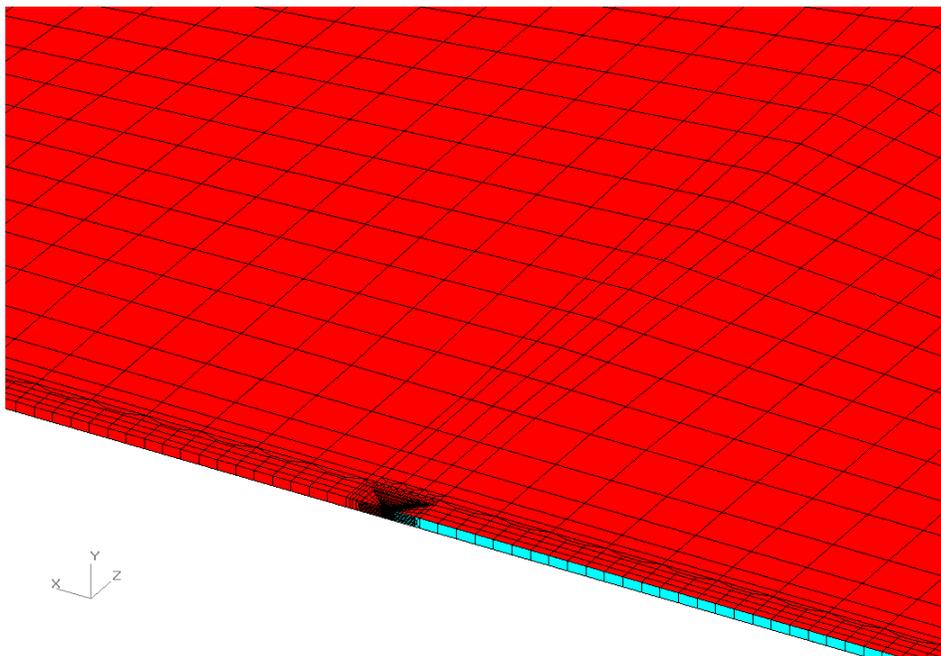


Figure 294 BLM with more mesh refinement and smaller cell element size; close up

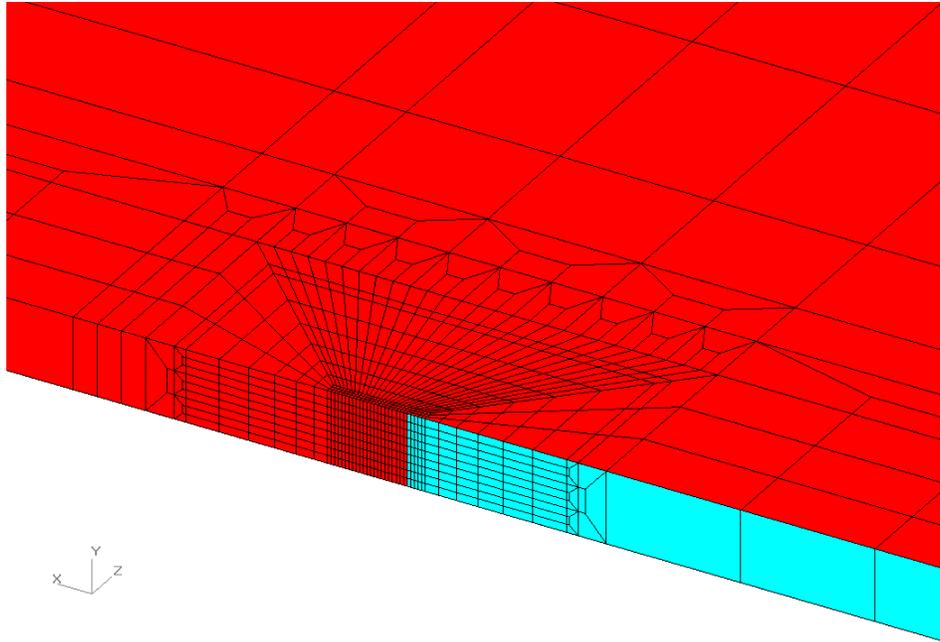


Figure 295 BLM with more mesh refinement and smaller cell element size; close up at the crack front

BLM Cell Mesh Activation

The BLM with a cell mesh is activated in the Home/Notes form with key word syntax. The mesh begins as a flat plate with an edge crack and uses the PRO mesh parameters to control the cell mesh refinement. The two key words used with the BLM cell mesh in the Home/Notes form are:

- *use BLM cell mesh
- *use BLM KT file = <file name>

Where the key word “*use BLM cell mesh” activates the BLM to start as a flat plate with an edge crack. And the second key word “*use BLM KT file = <file name>” provides the text data file name that contains the list of stress intensity K and T-stress values used to compute the perimeter node displacements, and to select the plane stress or plane strain equation choice. The KT data file should be located in the same directory as the FEACrack *.elt model file. The BLM cell mesh can be generated without the BLM KT file information. The figure below shows an example of the BLM key words in the Home/Notes form. The BLM KT file name given in the figure below is: “K and T - boundary values.txt”. These key words can be used with other “*use” key words or general comments; each key word or comment is separated by a semi-colon.



Units:
lb - in - psi

Continue
➔

Analysis Type: Cell Crack Mesh For Node Release Crack Growth (WAR)

Analysis Name:

Model Label:

Notes:
*use BLM cell mesh; *use BLM KT file = K and T - boundary values.txt; *Use
c Intel MKL solver out-of-core off;

Figure 296 Example of BLM key word text in the FEACrack Home/Notes form

BLM KT data file

To create the K and T stress data text file, refer to the “boundary conditions K and T stress value.xlsx” Excel spread sheet, provided with the examples. The maximum K and T stress values are entered and a load history is created. The table of values can be copied from this spread sheet into a data text file to quickly update the BLM perimeter displacement constraints. The number of analysis steps in the BLM KT file should be the same as the number of steps in the Boundary Conditions form. The figure below shows an example of the BLM KT data text file.

```

K and T - boundary values.txt - Notepad
File Edit Format View Help
Boundary Layer Model K and T stress values
perimeter displacement type = plane strain
number of steps = 20
step, K, T stress
1, 1000.0000, 250.0000
2, 2000.0000, 500.0000
3, 3000.0000, 750.0000
4, 4000.0000, 1000.0000
5, 5000.0000, 1250.0000
6, 6000.0000, 1500.0000
7, 7000.0000, 1750.0000
8, 8000.0000, 2000.0000
9, 9000.0000, 2250.0000
10, 10000.0000, 2500.0000
11, 11000.0000, 2750.0000
12, 12000.0000, 3000.0000
13, 13000.0000, 3250.0000
14, 14000.0000, 3500.0000
15, 15000.0000, 3750.0000
16, 16000.0000, 4000.0000
17, 17000.0000, 4250.0000
18, 18000.0000, 4500.0000
19, 19000.0000, 4750.0000
20, 20000.0000, 5000.0000

```

Figure 297 Example of the BLM KT data file containing the K and T-stress values

Create a BLM cell mesh

The BLM cell mesh begins as a flat plate geometry to provide all the PRO mesh refinement controls in the FEACrack user interface. To start a new model, select the “Cell Crack Mesh for Node Release” choice in the Home/Analysis Type menu.

In the Geometry/Configuration form select the “Flat Plate” geometry in the Structural Shape menu. In the Geometry/Crack/Configuration form select the $\frac{1}{2}$ symmetry choice and select the “Build mesh in +Z direction”. In the Geometry/Crack/Dimensions form select an edge crack, and enter the crack length, c . A BLM typically has the crack length equal to the BLM radius, which is half the plate width. In general, the crack length can be any length within the plate width. In the Geometry/Crack/Cell Mesh and Advanced Mesh forms enter the cell element sizes and set the mesh refinement options. The Adjust Zone is used to provide the remote contours around the cell zone for the J-integral. In the Geometry/Dimensions form enter the flat plate length, width, and thickness dimension values. The plate width should be twice the desired BLM radius, typically twice the crack length. The plate width is used to set the BLM cell mesh radius. The plate length should be less than half of the BLM radius, or smaller, to allow room behind the plate to add the mesh zones that give the circular BLM. A plate length that is several times the plate thickness is often a good starting value.

In the Boundary Conditions form, enter the number of analysis steps. In the “Load Steps To Save” column, remove the word “ON” to omit output from any steps. In the Constraints form the top and bottom face constraints can be set for the BLM cell mesh. The back, right, and front face constraints on the plate will not be used in the BLM cell mesh. For a quarter symmetric BLM cell mesh, apply custom displacements to constrain the y DOF (degree of freedom) on the bottom

face. Even though the plate mesh is initially created as half symmetric, the final BLM cell mesh will be quarter symmetric with that type of applied constraint. In the Cell Mesh tab, enter the crack tip opening angles for release and initiation; these control the node release crack growth.

In the Material Properties form, select the material plasticity type, typically incremental with a stress strain curve. In the Shell Material form (the base material), select elastic-plastic behavior and enter the Modulus, Poisson ratio, and stress-strain data.

In the Tools/Program Options and Tools/Model Options forms, select the FEA input file type, usually Warp3D for node release, and make any other changes needed for the model. Save the model file and build the crack mesh. Return to the Geometry/Crack/Cell Mesh and Advanced Mesh forms to adjust the mesh refinement if needed.

BLM cell mesh log file information

The mesh log file reports BLM cell mesh information to confirm the Home/Notes key word text input. In the Tools/Program Options form set the “Logfile Output Level” to 1 before building the crack mesh. The <model name>.log file can be viewed using a text editor like Notepad. The log file contains “INFO:” messages that the BLM cell mesh has been activated, that the BLM KT data file name has been set, and the K and T-stress values have been read and listed with the plane strain/stress equation choice.

Friction Stir Plug Weld Model

The friction stir plug weld is available in FEACrack, and the following information describes using the plug weld model. In the FEACrack Geometry/Configuration form, select the “Friction Stir Plug Weld” model. Use the “Material Group case” menu to select all 12 materials, or one of the weld material grouping choices (reduces the total number of material groups). To get two material groups in the plug HAZ, select the “Unique Materials” option on the Configuration form. This gives the option to have a uniform material in the plug HAZ or to have two material groups on each side of the crack surface; see the figure below.

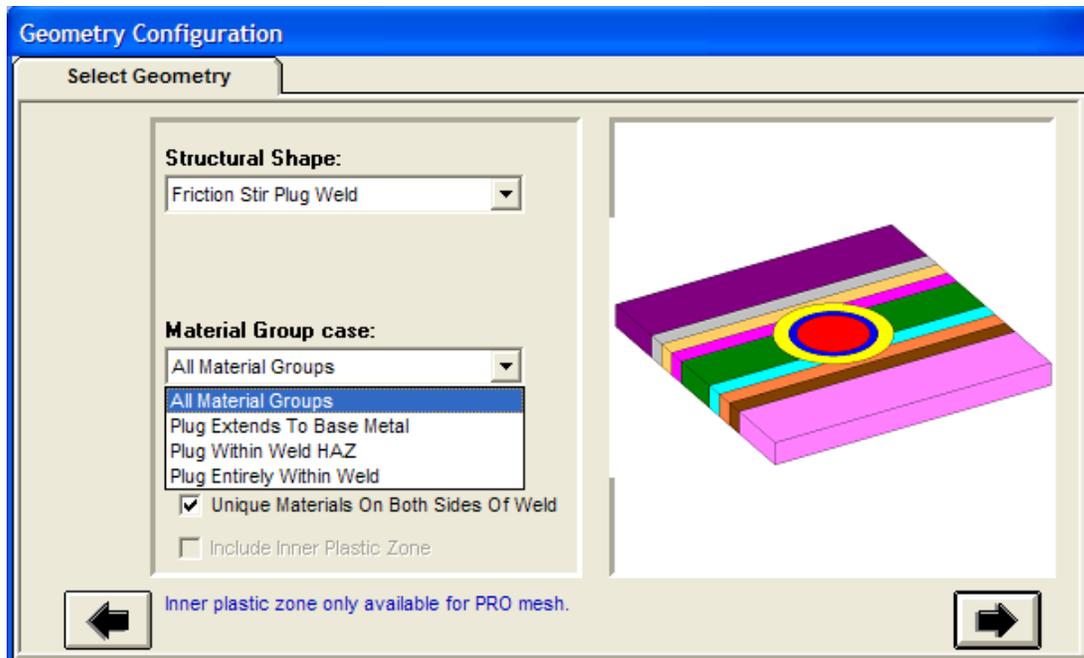


Figure 298 Select the “Material Group” and “Unique Materials” options for the plug weld model

In the Geometry/Crack/Configuration form there is a single crack location choice: the crack is in the plug weld HAZ. And there a single model symmetry option for the full mesh since the crack can be located at any position around the plug.

In the Geometry/Crack/Dimensions form there are three crack shape choices: surface, through-wall, and partial-depth. The crack length and position are given by two arc angles around the plug. The angles are measured from the model centerline on the back side of the plug (-Z side), counter-clockwise around the plug circle; see the figure below. Any angle value, positive or negative in degrees, can be entered for the start and end of the crack. In the mesh generation module, the angle values are converted to be within the range: $-180^{\circ} \leq \theta \leq 180^{\circ}$. For example, giving a start angle of 0.0° and an ending angle of 90.0° will position the crack in the back left quadrant of the plug. Or entering a start angle of -135.0° and an ending angle of -45.0° will position the crack on the right (+X) side of the plug.

For the partial-depth crack shape, entering a full 360° arc will give a continuous crack front around the plug circumference. For example, enter a start angle of 0.0° and an ending angle of 360° ; or enter angles of -180.0° to 180.0° . A small angle range will give a partial-arc and partial-depth crack. In this case the partial-arc crack has crack front nodes along the bottom of the crack, but not along the vertical mesh lines at each end of the crack.

The crack depth, a , is given from the top surface of the plug weld plate and is measured perpendicular to the top surface through the plate thickness. The crack depth is not affected by the plug bevel.

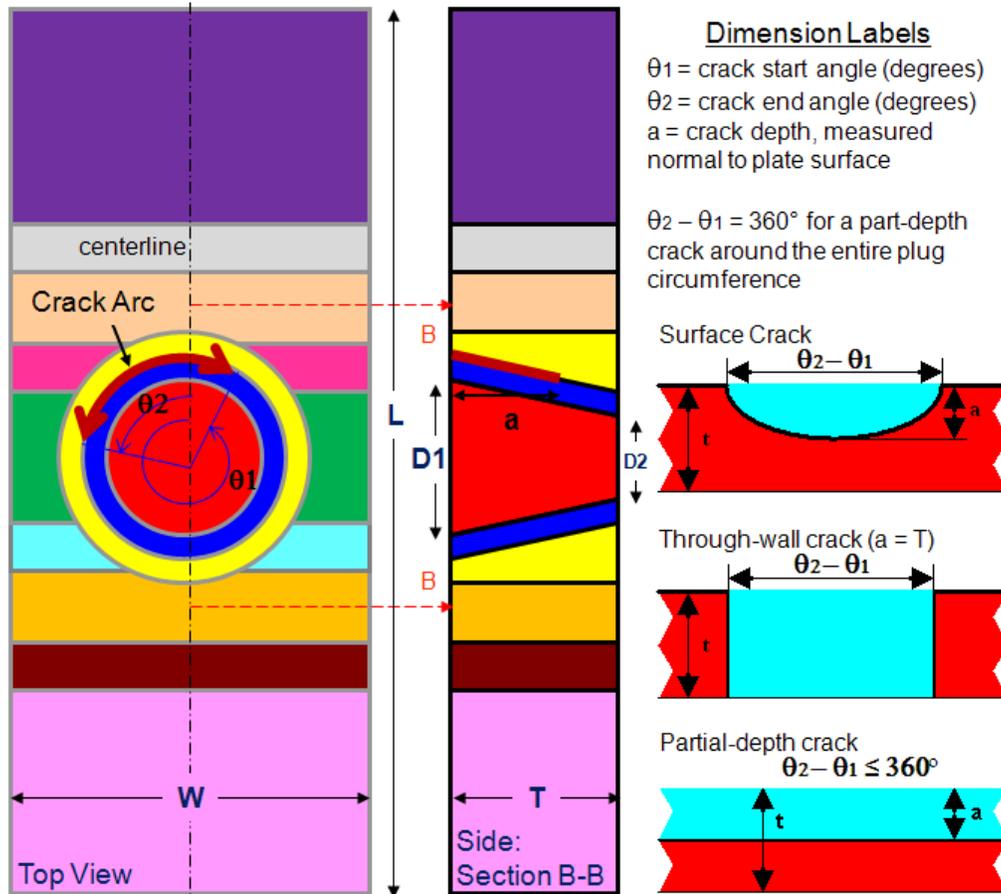


Figure 299 Crack arc angles are measured counter-clockwise from the centerline on the back side (-Z) of the plug

The through-wall and partial-depth crack shapes the surrounding mesh is generated to match the plug mesh zone so that the nodes can be merged to connect the mesh zones together. The surface crack uses automatically defined tied contact surfaces to connect the mesh zones. For the surface crack shape 20-node brick elements are recommended to improve the tied contact. When using the automatic tied contact between the plug and surrounding mesh zones it is also recommended to have sufficient mesh refinement to capture the circular arc of the plug region. Higher mesh refinement helps improve the tied contact.

In the Geometry/Dimensions/Friction Stir Plug Weld form enter the plate, plug, and weld zone dimensions. The “Use Default Values” option at the bottom of the form will fill in most of the plug weld model dimensions based on the given plate thickness, t , and plug weld diameter, $D1$; see the figure below. The default values may be helpful when starting a new model since there are several dimension limits on the various weld zones. Selecting the “User Defined Values” option allows all the plug weld model dimensions to be entered.

Dimensions

Friction Stir Plug Weld

Mesh Options

Plate Length (L): 5.52 in

Plate Width (W): 3.68 in

Plate Thickness (t): 0.32 in

Plug Top Diameter, D1 (A): 0.92 in

Plug Bottom Diameter, D2: 0.92 in

Plug Inner HAZ, T1 (B): 0.092 in

Plug Outer HAZ, T2 (C): 0.092 in

Weld Thickness, T3 (D): 0.69 in

Weld Zone 1 Thickness, T4 (E): 0.184 in

Weld Zone 2 Thickness, T5 (F): 0.184 in

Weld Zone 3 Thickness, T6 (G): 0.184 in

Weld Zone 4 Thickness, T7 (H): 0.184 in

Weld Zone 5 Thickness, T8 (I): 0.184 in

Weld Zone 6 Thickness, T9 (J): 0.184 in

Bevel options: No Bevel

Use Default Values (selected) / User Defined Values

Figure 300 Use default or user defined values for the plug weld model dimensions

In the Geometry/Dimensions/Mesh Options form enter the plate, plug, and weld zone mesh values to set the number of elements and element size bias in each mesh zone. See the figure below. The “Use Default Values” on this form will set all the default mesh option values. Selecting the “User Defined Values” option allows the number of elements and element size bias to be entered for all the mesh zones. The number of elements in each mesh zone, such as NA, NDx, NDz, should be entered as integer values. The element size bias values (parameters ending with “b”), such as NAb, NDb, can be floating-point values like 1.5, and are generally greater or equal to 1.0. A bias of 1.0 gives uniform size elements in a mesh zone. The number of elements in each zone can also be entered as zero for the mesh generator to set a default value for a particular mesh.

bevel option gives a cone segment surface between the plug and surrounding transverse mesh zones. Depending on the amount of outer bevel, given by the bottom plug radius D2, the outer profile of the plug will be within the outer weld mesh zones, within the weld HAZ mesh zones, or within the center weld zone. Tied contact is required for the outer bevel option.

On the Mesh Options form the “Plug Automatic Tied Contact” option is activated for the no bevel and inner bevel cases for the partial-depth and through-wall crack shapes; see the figure below. Other choices will require the automatic tied contact by default. When available, checking the Plug Automatic Tied contact will use tied contact between the plug and surrounding mesh zones, unchecking the option is recommended for the nodes to be merged between mesh zones.

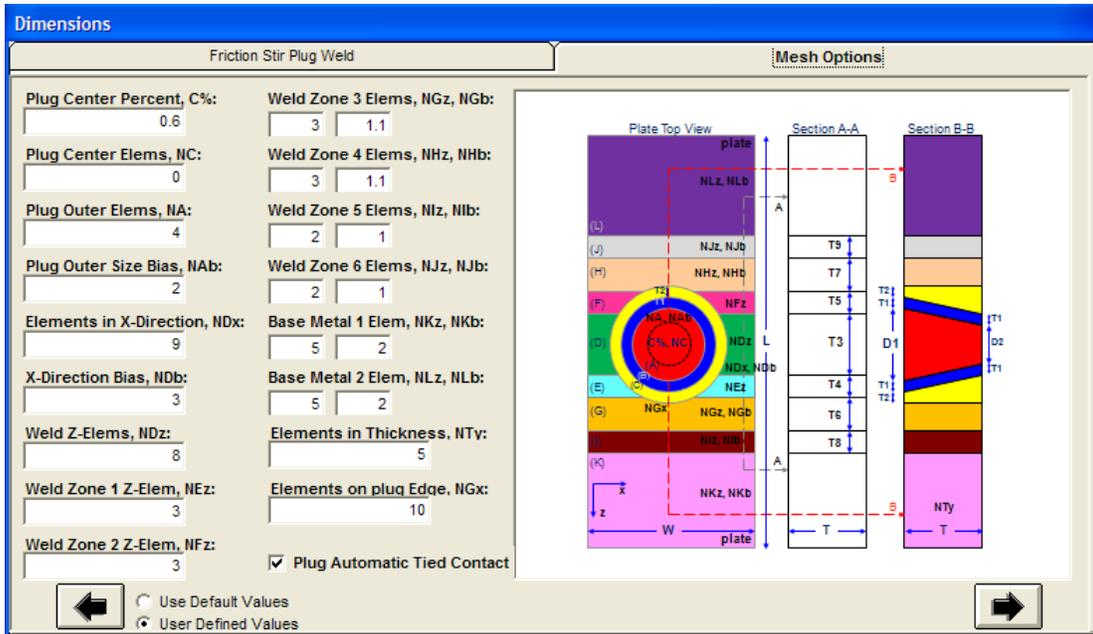


Figure 303 Some plug bevel and crack shapes activate the Plug Automatic Tied Contact option

General boundary conditions are available on the six exterior mesh surfaces of the plug weld model, and crack face pressure. Use the Boundary Conditions/Constraints form to select the node constraints and loading on the plug weld model; see the figure below.

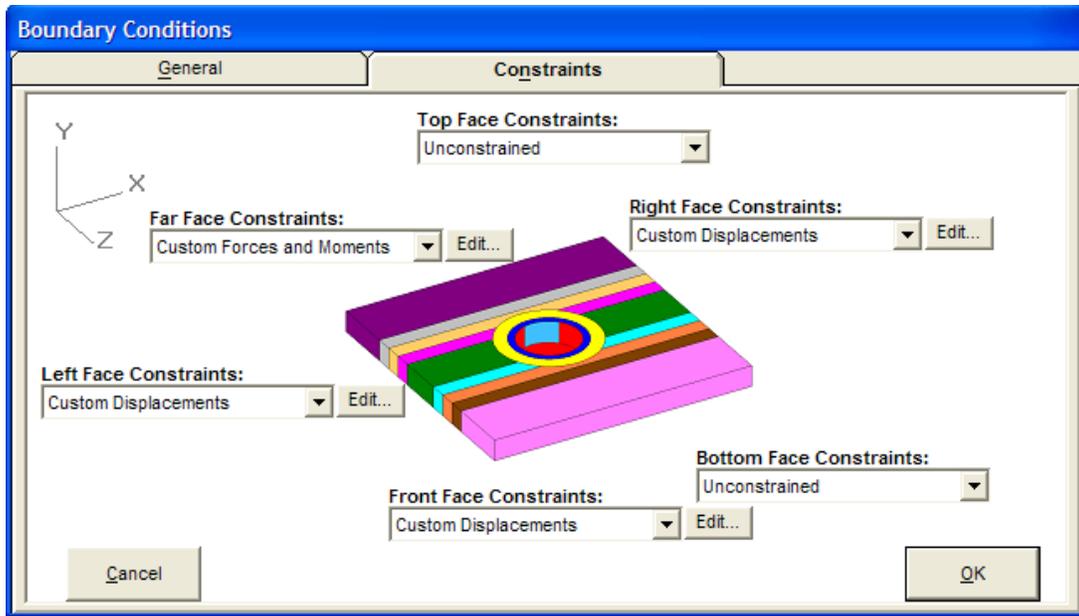


Figure 304 Select constraints and loads on the plug weld surfaces

The plug weld model has up to 12 material groups in the plug, weld, and base metal mesh zones. In the Material Properties/Material Model Type form use the 12 buttons to select the material values for each mesh zone; see the figure below.

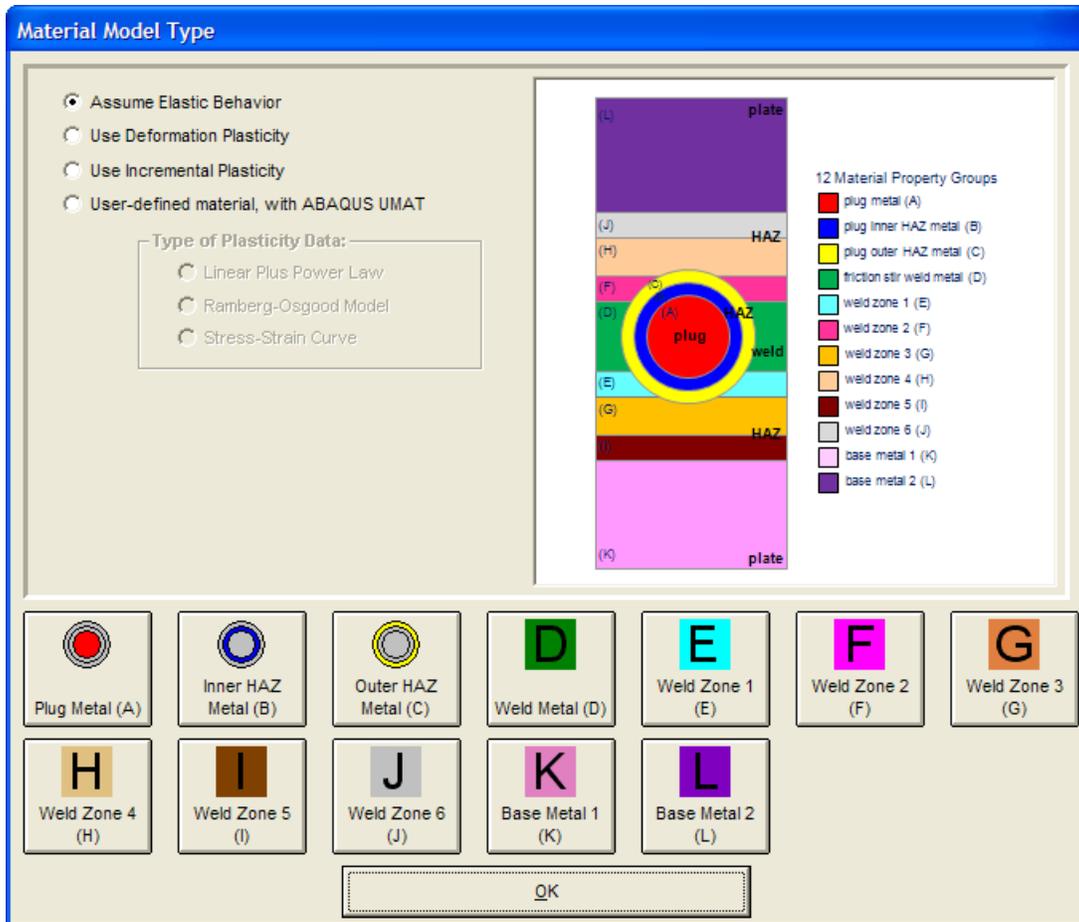


Figure 305 The plug weld model has 12 material groups

As described above, the number of active material groups is controlled by the “Material Group” case menu in the Geometry/Configuration form. For example, the selection in the figure below is for the plug entirely within the weld. And on the Material Properties form, the number of active materials is now 8 since some of the mesh zones will use the same material property values.

If the unique materials option is unchecked on the Configuration form, the inner plug HAZ material group will also be deactivated on the Material Properties form. This could give 7 or 11 active material groups depending on the weld material group case.

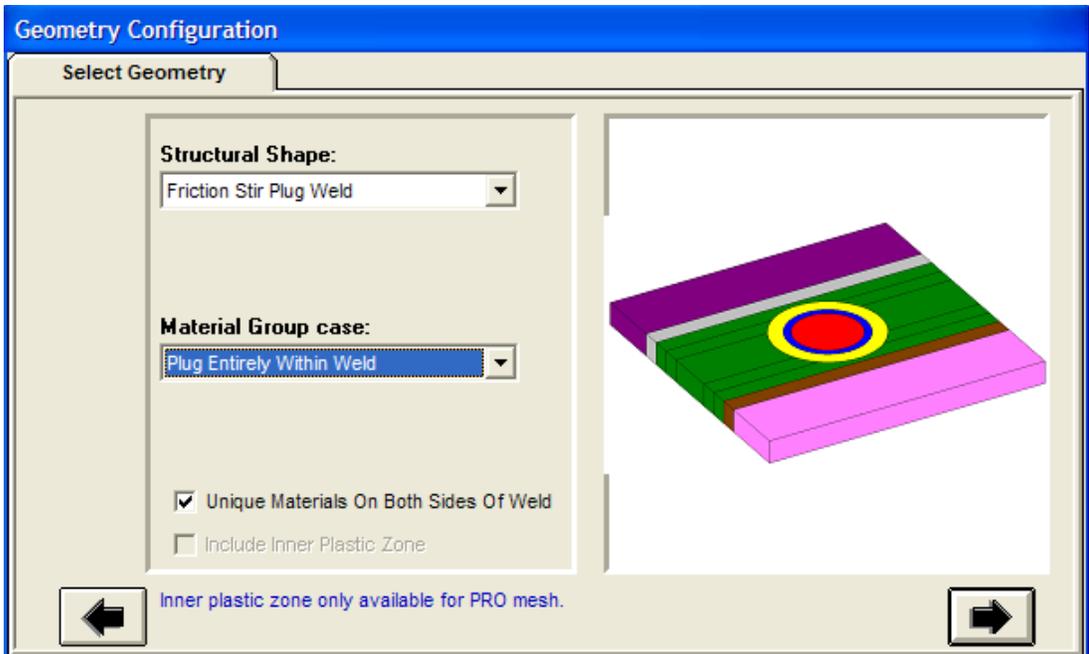


Figure 306 Material Group choice: the plug is entirely within the weld, 8 material groups active

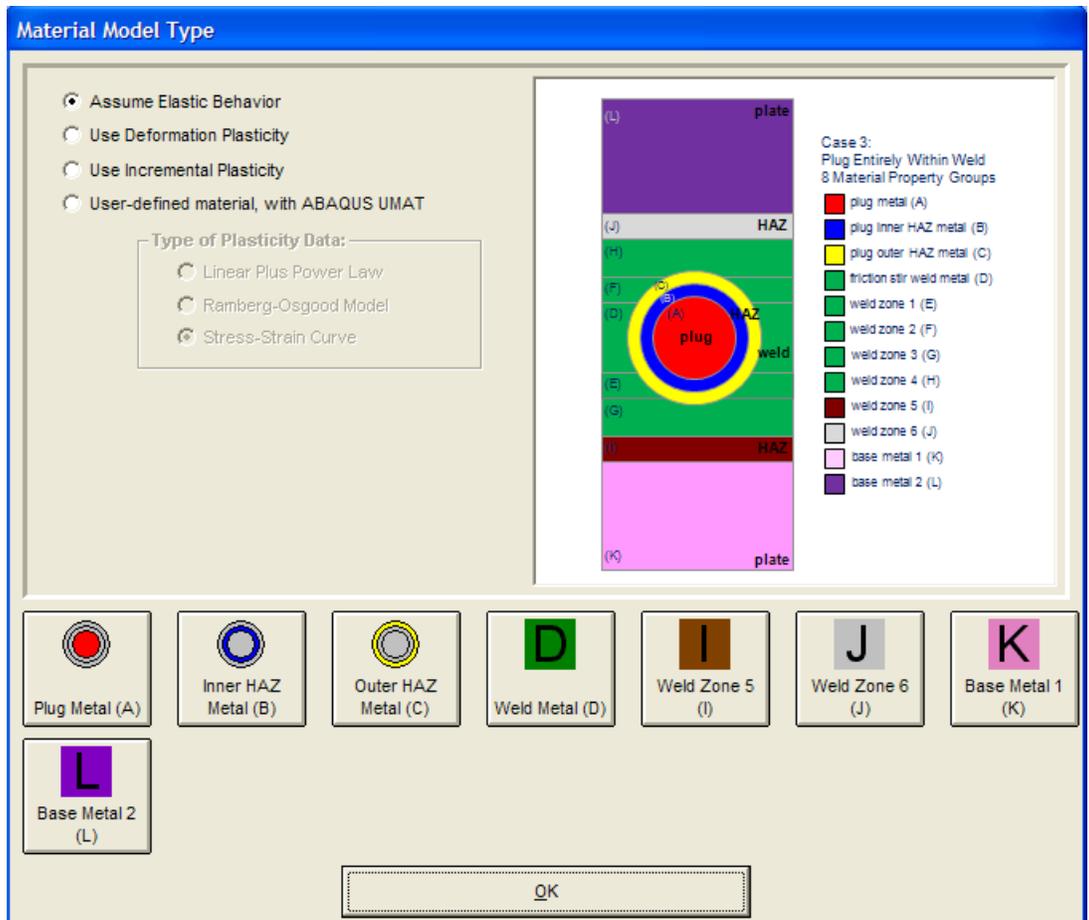


Figure 307 Materials form: plug entirely with the weld, 8 material groups active

Sub-files Option

The *Abaqus* and *Warp3D* input files generated by *FEACrack* can use sub-files. Select the “Use Sub-Files for FEA Input” option in the Tools/Program Options/Analysis form, shown below.

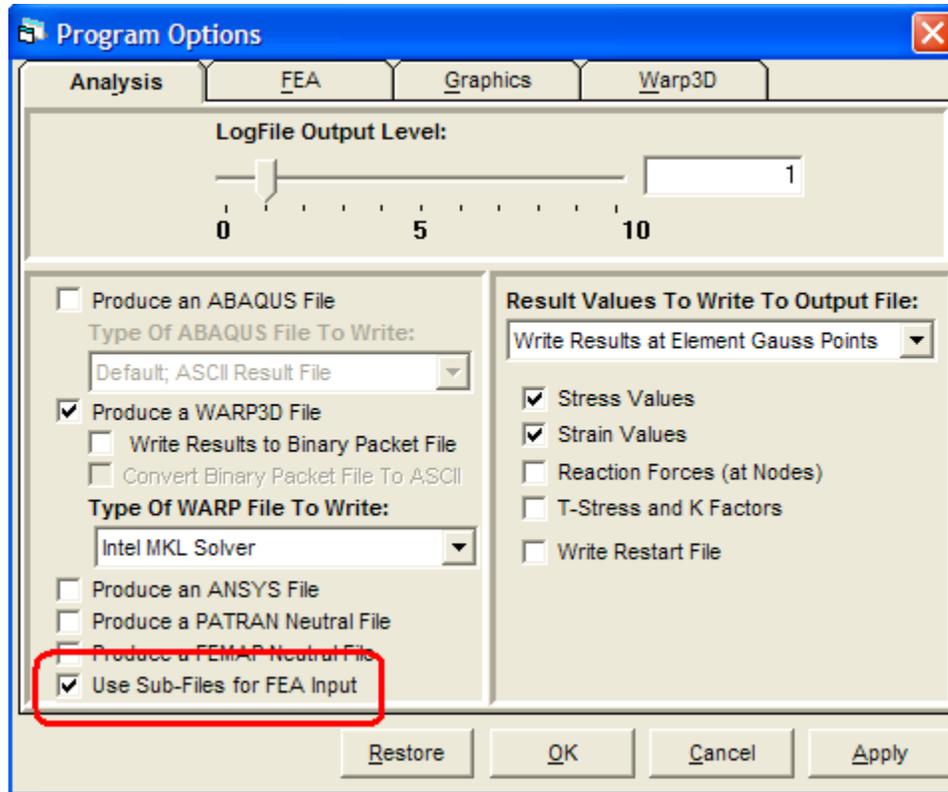


Figure 308 Sub-Files option in the Tools/Program Options/Analysis form

In the *Abaqus* input file, the `*INCLUDE` command is used to specify the name of each sub-file. In the *Warp3D* input file, the `*input` command is used to specify the name of each sub-file.

The sub-files separately contain the following blocks of data:

- Crack comment data blocks (used for crack result post processing),
- Node coordinates
- Element connectivity (and the element blocking for *Warp3D*)
- Material data
- Initial conditions (currently nodal temperatures)
- Node and element sets (for *Abaqus*)
- Analysis step data

The analysis step data sub-file also uses sub-files for the following data:

- Loads (element tractions and nodal forces)
- Node constraints
- J-integral domains

Using the sub-files gives a very brief master input file that just references the appropriate sub-files.

The sub-file names use a keyword appended to the given model name, and have a *.txt file extension. The *.txt extension was chosen instead of *.inp so that extra files do not appear in the Run FEA file control wizard form when selecting input files to run. The file name keywords include “_commentdata”, “_nodes”, “_elements”, etc. For post-processing *Abaqus* results the <model>_commentdata_abq.txt file must be located in the same directory as the *Abaqus* *.fil results file so that the crack comment data block information is available. The sub-file containing the crack comment data block is echoed to the *Warp3D* output file, so there is no change to the post processing for *Warp3D* results.

***Warp3D* Utility Commands**

The *Warp3D* utility commands “*echo on” and “*echo off” are written as comment lines before and after each section of data in the *Warp3D* input file. To activate these commands, use a text editor to make a global find-and-replace of the command to quickly remove the comment character.

Warp3D Function Type “d” For Through-Thickness Cracks

In *Warp3D* through-thickness cracks can be analyzed using function type “d” along the crack front to get a crack front average J value rather than point wise J values at each crack front node. The function d can be selected for *Warp3D* input files in *FEACrack* by entering this text in the Home/Notes form (key text is case insensitive):

```
*use warp3d function type=d
```

The crack mesh generator will check the Notes text for this key text, and if found will activate the function type d option.

The option can be verified in the *Warp3D* input file by looking for the text: “function type d”, which is an option with the “domain” calculation command, typically located at the bottom of the input file.

Figure 309 Enter key text in the Notes for function type=d for through-thickness cracks

Warp3D automatic element blocking

The *Warp3D* analysis uses element blocking to process groups, the “blocks”, of elements that have similar properties. The crack mesh generator automatically divides the mesh into blocks and writes the “blocking” data to the *Warp3D* input file. *Warp3D* also provides an automatic blocking option since version 17.3.2 (refer to the *Warp3D* manual). To use the automatic blocking instead of the built-in standard blocking, enter this keyword text in the Home/Notes form:

```
*use warp3d auto blocking
```

The option can be verified in a *Warp3D* input file by looking for the syntax: “blocking automatic”.

The element blocking size for both choices is set in the Tools/Program Options/FEA form in the “Number of Elements Per Warp3D Element Block” text box. Typical block size values are 32, 64, and 128 elements.

FEACrack

Units:
kip - in - ksi

Continue
➔

Analysis Type: Focused Crack Mesh For J-Integral Analysis (Standard)

Analysis Name:

Model Label:

Notes:
*use warp3d auto blocking

Figure 310 Enter keyword text in the Home/Notes for the automatic element blocking option

View A Femap Neutral File

The FEACrack/Tools/Show Stress Interpolation/File Selection and Preview form can be used to view a mesh. This option can be useful when viewing incomplete meshes associated with an error message.

For example the *Femap* neutral file format can be selected in the Tools/Program Options/Analysis form. When a mesh has been generated, the file name is selected by browsing for the “Previous Analysis Mesh” file name; see the figure below. The “Show” check box will load the mesh file and show the mesh. The window size can be increased using the maximize button at the upper right corner of the form.

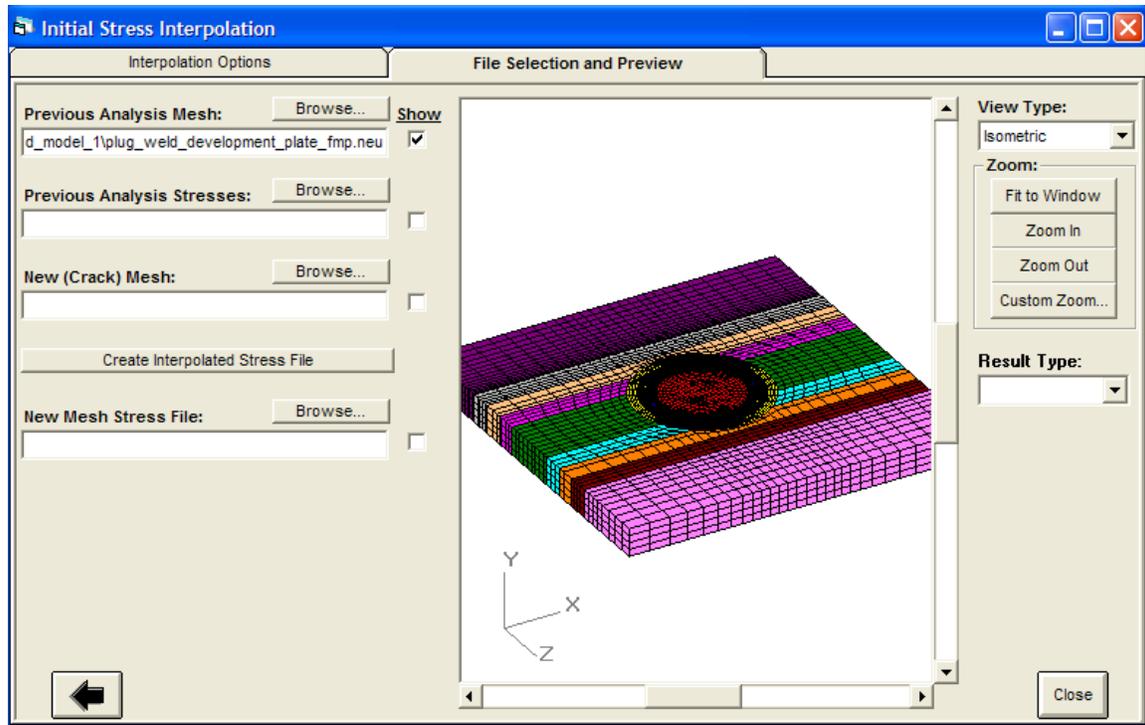


Figure 311 Use the File Selection and Preview form to view a mesh file

Crack Face View Option

For a full crack mesh model or a right-half symmetry crack mesh model, the crack face is not visible, since the crack is within the model volume. To view the crack face mesh, the “view crack face” option can be used to create several *Femap* neutral files. Each *Femap* neutral file shows part of the crack mesh: the back crack face, the front crack face, the right side of the crack, and the left side of the crack. The *Femap* neutral file can be viewed in *FEACrack* using the stress interpolation tool as a mesh viewer. *Femap* is a general purpose commercial mesh generation program and its neutral file format is supported in *FEACrack*.

To activate the prototype view crack face option, add key word text in the Home/Notes form:

*use view crack face femap files

Figure 312 Add key word text in the Home/Notes form to activate the view crack face option

When the view crack face option is active the *Femap* files created for each mesh region will use the given model file name with a description name appended. For an example model name of “modell”, the *Femap* neutral file names are:

modell_ViewCrackBack_fmp.neu
modell_ViewCrackFront_fmp.neu
modell_ViewCrackLeft_fmp.neu
modell_ViewCrackRight_fmp.neu

To view these *Femap* files, open the Tools/Show Stress Interpolation form; then click on the File Selection and Preview tab. Since the form is used just for mesh viewing, the interpolation values can be left unchanged. It may be helpful to maximize the form size using the maximize button at the upper right corner of the form. Use the Previous Analysis Mesh file name and browse to locate one of the Femap neutral files. Then click the Show option box to show the mesh.

For example, a surface crack in a plate is shown below. The crack has been shifted within the plate volume, so each crack face will have a different size mesh region.

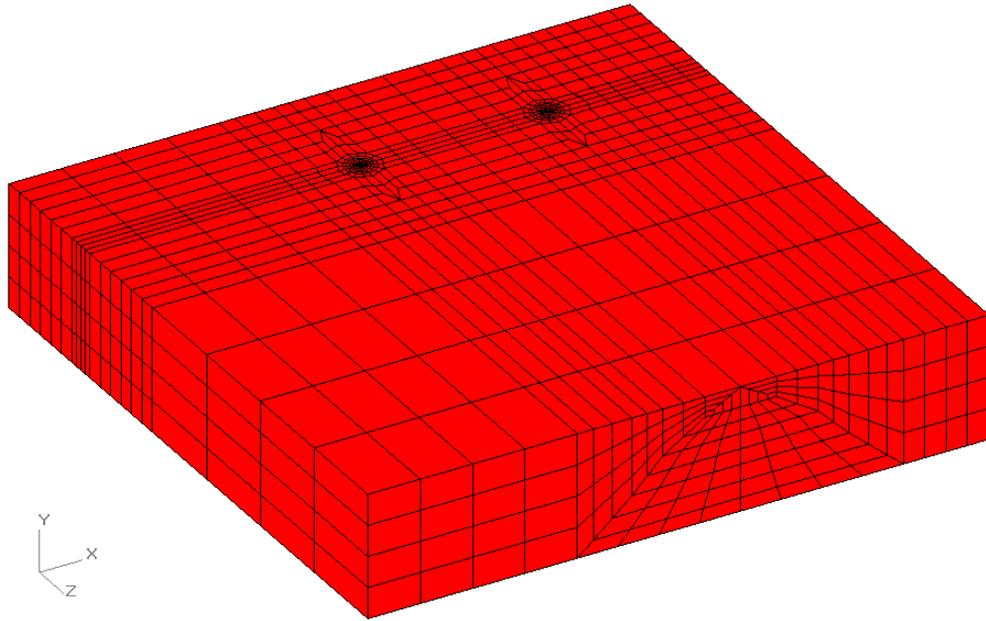


Figure 313 The standard full plate surface crack mesh picture in the main graphics window

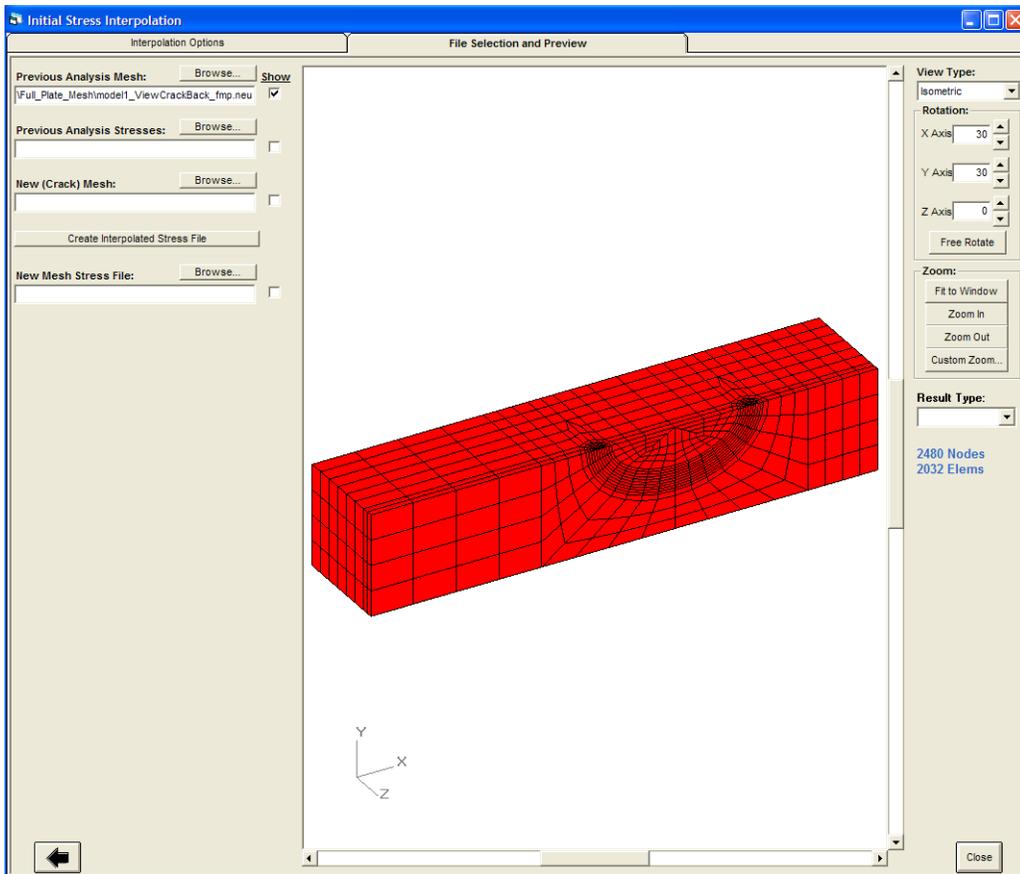


Figure 314 View of the back crack face from the “model1_ViewCrackBack_fmp.neu” Femap file using the Tools/Show Stress Interpolation form; the Femap file is selected with the Browse button; click the Show check box to show the mesh

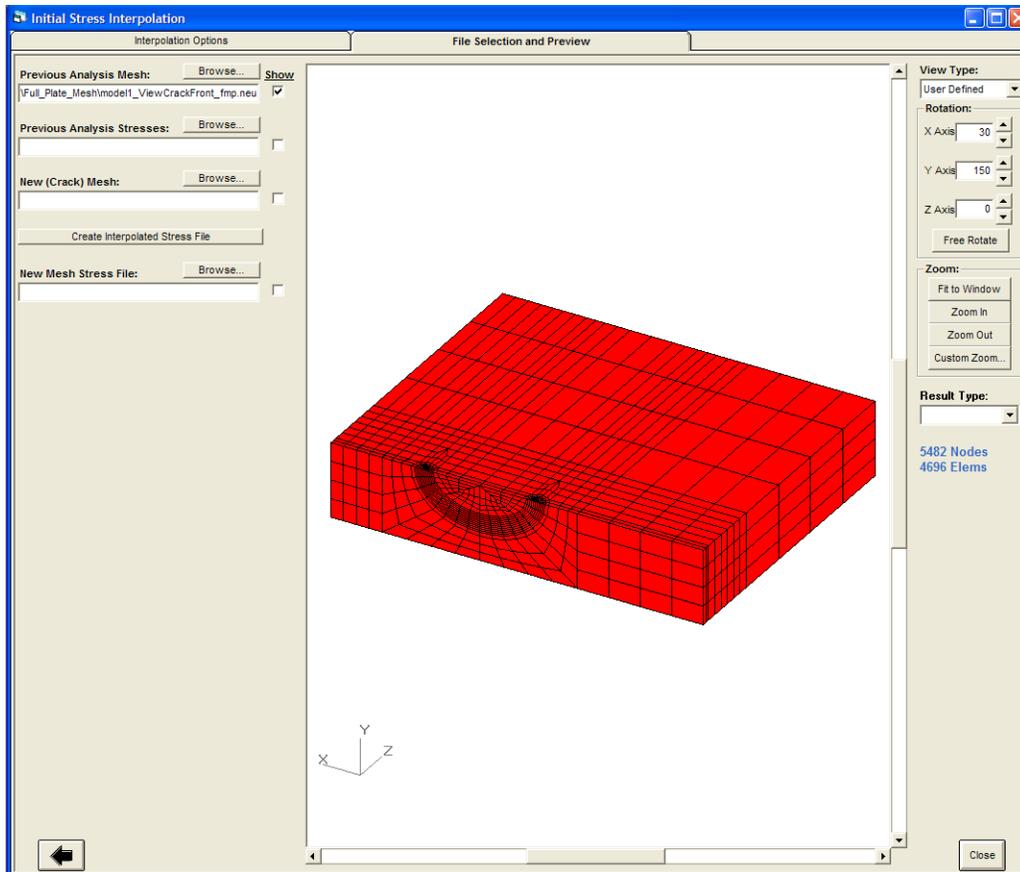


Figure 315 The View of the front crack face from the “model1_ViewCrackFront_fmp.neu” Femap file; the mesh has been rotated by setting the Y-axis rotation angle to 150o to view the front crack face

Crack face area option

The crack face area calculation is activated by key word text in the Home/Notes form. Enter the text (see the figure below):

*use crack face area compute on

This key word text can be used with other key words by separating each item with a semi-colon “;”.

The area of each element face on the crack face is computed and summed to get the total mesh area, which includes both crack faces if present in the mesh. This option is currently available for the flat plate geometry and user-defined geometry choices.

The image shows the FEACrack software interface. At the top is the logo, which consists of a blue and red globe icon followed by the text "FEACrack". Below the logo are several input fields and buttons:

- Units:** A dropdown menu currently set to "N - m - Pa".
- Continue:** A button with a right-pointing arrow.
- Analysis Type:** A dropdown menu set to "Focused Crack Mesh For J-Integral Analysis (Standard)".
- Analysis Name:** An empty text input field.
- Model Label:** An empty text input field.
- Notes:** A text area containing the text: "*use ansys node file; *use crack face area compute on;".

Figure 316 Enter key word text in the Notes field to activate the crack face area calculation.

The calculated crack face area value is written to the crack mesh log file. Activate the log file in the Tools/Program Options form by setting the "LogFile Output Level" to 1 or higher. After generating the crack mesh, open the log file: <model name>.log with a text editor like Notepad. The total mesh area, crack face area, and symmetry multiplier values are given near the end of the log file. Search for the text "crack face area:" in the log file. See the figure below for an example of text from the log file.

The total mesh area and the crack face area values may be multiples of each other depending on the selected mesh symmetry. If the mesh has right half symmetry or a full model, then the two crack faces will give a total mesh area double the crack face area, otherwise the mesh area will equal the crack face area. Increase the log file level to get more output of the crack face area calculations.

```

crack face area:
    total mesh area =    1.54164850589230
    crack face area =   0.770824252946148
    symmetry multiplier =  2.000000000000000

```

Figure 317 Open the log file with a text editor to view the calculated area value.

Shift Node Coordinates

The mesh node coordinates can be shifted in the x, y and z directions by entering the key word text in the Home/Notes form:

```
*use shift coord x = <x_value>, y = <y_value>, z = <z_value>
```

See other “*use” key words for similar examples of syntax in the Home/Notes form. Each shift distance value is entered after the coordinate label and equal sign as a decimal number, for example:

```
*use shift coord x = 12.5, y = -2.25, z = 0.1
```

Any of the coordinate labels can be omitted and the shift distance will be set to zero, for example omitting the x and z labels to shift just the y distance:

```
*use shift coord y = 1.75
```

This key word can be used with other key words in the Home/Notes form by separating each key word syntax with a semi-colon “;”. The node shift values can be confirmed in the mesh log file; look for the text “Mesh coordinates shift distances”, followed by the shift distance values.

References

1. Yuh J. Chao and X. K. Zhu, “Constraint-modified J-R curves and its application to ductile crack growth”, International Journal of Fracture, 106: pp. 135-160, 2000, Department of Mechanical Engineering, University of South Carolina, Columbia, SC 29208
2. Yuh J. Chao and Li Zhang, “Tables of plane strain crack tip fields: HRR and higher order terms”, ME-Report 97-1, 1997, Department of Mechanical Engineering, University of South Carolina, Columbia, SC 29208
3. Anderson, T. L., Fracture Mechanics, Fundamentals and Applications, 3rd ed., CRC Press, 2005

Example Files

Please contact support@questintegrity.com for more information about example files for .
Abaqus, *ANSYS*, and *WARP3D*

Appendix:

Example Fortran Source Code

This example of Fortran Source Code will be loaded with the examples at the time of installation for your reference. By locating the electronic version of the subroutine you shouldn't need to retype the Fortran source code. Below is a listing of the Fortran subroutine used to compile the UserReverseTransform.dll for the cylinder example; the Fortran source code is listed here for reference. To avoid incorrect calculations for other models, the *.elt model file name must match the expected model name, otherwise an error is returned. A Fortran compiler is required to compile the UserReverseTransform subroutine, such as the Intel Fortran compiler. The Fortran compiler must be purchased separately from FEACrack.

```
!  
! Introduction:  
! Compute the reverse transformation of the custom crack front node coordinates  
! for a user-defined geometry crack mesh in FEACrack.  
!  
! When using a custom crack in a user-defined geometry for a fatigue analysis  
! in FEACrack, the incremental crack growth is updated at each crack front node  
! position along the custom crack front. To generate the crack mesh for the  
! next analysis step in a fatigue analysis, the global crack front x,y,z  
! coordinates must be converted to the crack plane c,a coordinates; the initial  
! c,a crack front coordinates are entered in the Geometry/Cracks/Dimensions  
! form in FEACrack for the custom crack.  
!  
! In this routine compute the reverse transformation from the global  
! coordinates to the crack plane coordinates corresponding to the particular  
! user-defined geometry.  
!  
! For more information refer to the FEACrack user's manual, the web site, or  
! contact technical support at Quest Integrity:  
! Main line: 303-415-1475  
! FAX line: 303-415-1847  
! E-mail: support@questintegrity.com  
! web site: http://www.questintegrity.com  
! 2465 Central Avenue, Suite 110  
! Boulder, CO, USA 80301  
!  
subroutine UserReverseTransform(dimIntArray,IntArray, &  
                                GlobalCrackCoord,CrackCAcoord,StringArray)  
!  
! Declare Variables  
!  
    implicit none  
    integer, intent(in) :: dimIntArray  
    integer, intent(inout) :: IntArray(dimIntArray)  
    character (len=512), intent(inout) :: StringArray(IntArray(3))  
    real(8), intent(in) :: GlobalCrackCoord(IntArray(4),IntArray(5))  
    real(8), intent(out) :: CrackCAcoord(IntArray(7),IntArray(8))  
!  
! Expose this subroutine for external access in this DLL  
!  
!DEC$ ATTRIBUTES DLLEXPORT::UserReverseTransform  
!DEC$ATTRIBUTES ALIAS : 'UserReverseTransform' :: UserReverseTransform  
!DEC$ATTRIBUTES STDCALL :: UserReverseTransform  
!DEC$ATTRIBUTES REFERENCE :: dimIntArray,IntArray  
!DEC$ATTRIBUTES REFERENCE :: GlobalCrackCoord,CrackCAcoord,StringArray  
!  
! Variables:  
! dimIntArray = the IntArray() row dimension  
! IntArray() = a list of integer values, such as flag values to indicate an  
! error, options to control the calculations, or array dimensions:  
! row 1 = mflag = return the integer flag to indicate if a warning or error  
! message should be shown; 0=no message, 1=warning message (analysis
```

```

!           continues and show the message), 2=error message (analysis stops, and
!           show the message); the message is returned in the StringArray()
! row 2 = logFlag = an integer flag to activate output to the log file for
!           debugging and testing; the log file name is set in the StringArray()
! row 3 = maxString = the maximum number of strings in the StringArray(), the
!           array row dimension
! row 4 = maxRowGlobalCrackCoord = the GlobalCrackCoord() array row dimension
! row 5 = maxColGlobalCrackCoord = the GlobalCrackCoord() array column
!           dimension; expecting 3 for the x,y,z coordinate values
! row 6 = numCrackNode = the number of crack front nodes
! row 7 = maxRowCrackCAcoord = the CrackCAcoord() array row dimension
! row 8 = maxColCrackCAcoord = the CrackCAcoord() array column dimension
! row 9 = step_index = the current fatigue analysis step number
! row10 = crack_shape = the crack shape integer ID
! row11 = crack_location = the crack location integer ID
! row12 = geom_type = the geometry choice integer ID
! row13 = symmetry_flag = the mesh symmetry choice integer ID
! row14 = [unused]
! StringArray() = array of string variables for file names and other strings:
! row 1 = message string, initially blank; if return a warning or error
!           message if mflag = 1 or 2
! row 2 = logFile, the log file name to open the log file and output testing
!           and debugging information to a text file
! row 3 = modelFile, from the *.elt file name for the current model; use to
!           check that the reverse transformation matches the user-defined geometry
!           crack mesh model
! GlobalCrackCoord() = the global cartesian x,y,z coordinates for the crack
!           front nodes; use these global coordinates to compute the crack plane c,a
!           coordinates; the rows contain the data for each crack front node; the
!           columns correspond to 1=x, 2=y, 3=z coordinates
! CrackCAcoord() = return the crack plane c,a coordinates computed from the
!           reverse transformation corresponding to the particular user-defined
!           geometry crack mesh; the rows match the rows in the GlobalCrackCoord()
!           array for the same crack front nodes; the columns correspond to
!           1=c (crack length position), 2=a (crack depth position)
!
!
! Declare Local Variables
!
integer :: mflag,logFlag,maxString,maxRowGlobalCrackCoord,maxColGlobalCrackCoord
integer :: numCrackNode,maxRowCrackCAcoord,maxColCrackCAcoord
integer :: step_index,crack_shape,crack_location,geom_type,symmetry_flag
integer :: logUnit = 20           ! set the log file unit number
integer :: debug_level = 1      ! set the log file local debug level
integer :: stat_err,row
character (len=512) :: message,logFile,modelFile
character :: today*8,now*10     ! string variables for the date and time to write to
the log file
real(8) :: c_length,a_depth

!
! Declare additional local variables for each particular reverse transformation
! calculation; these variables may be specific to a particular model
!
real(8) :: radius_in,thickness,radius_out,radius_node
real(8) :: node_y,node_z,theta
character(len=120) :: model_name_1,model_name_2     ! string variables for the
expected model names
!
! Local Variables:
! logUnit = the log file unit number
! debug_level = set the log flag debug level to activate or deactivate the log
! file; this value corresponds to the value set by the log file slider bar in
! the Tools/Program Options/Analysis Options form; when the given logFlag
! value is greater than or equal to debug_level then the log file will be
! written
! stat_err = file status flag to trap file errors
! row = array row index for do-loops
!
!-----
! Extract values from the arrays
!

```

```

mflag = 0          ! initialize, IntArray(1)
logFlag = IntArray(2)
maxString = IntArray(3)
maxRowGlobalCrackCoord = IntArray(4)
maxColGlobalCrackCoord = IntArray(5)
numCrackNode = IntArray(6)
maxRowCrackCAcoord = IntArray(7)
maxColCrackCAcoord = IntArray(8)
step_index = IntArray(9)
crack_shape = IntArray(10)
crack_location = IntArray(11)
geom_type = IntArray(12)
symmetry_flag = IntArray(13)

message = ' ' ! initialize, StringArray(1)
logFile = StringArray(2)
modelFile = StringArray(3)
!
! Get the date in the "today" string; and get the current time in the "now" string;
! write these values to the log file
!
      call DATE_AND_TIME(today,now)
!
! Open the log file; report testing and debugging information to the text file
!
      if(logFlag.ge.debug_level) then
          open(unit=logUnit, file=logFile, status='unknown', err=901, iostat=stat_err)
          ! jump to bottom on error
          write(logUnit,*,ERR=901) '----- Log file for UserReverseTransform ----
-----'
          write(logUnit,*,ERR=901)
          write(logUnit,3,ERR=901) 'current date =
',today(5:6), '/',today(7:8), '/',today(1:4)          ! month/day/year
          write(logUnit,3,ERR=901) 'current time =
',now(1:2), ':',now(3:4), ':',now(5:6)                ! hour:minute:second
          write(logUnit,*,ERR=901)
          write(logUnit,*,ERR=901) 'Input values:'
          write(logUnit,*,ERR=901) '                dimIntArray =',dimIntArray
          write(logUnit,*,ERR=901) '                logFlag =',logFlag
          write(logUnit,*,ERR=901) '                maxString =',maxString
          write(logUnit,*,ERR=901) '                maxRowGlobalCrackCoord
=',maxRowGlobalCrackCoord
          write(logUnit,*,ERR=901) '                maxColGlobalCrackCoord
=',maxColGlobalCrackCoord
          write(logUnit,*,ERR=901) '                numCrackNode =',numCrackNode
          write(logUnit,*,ERR=901) '                maxRowCrackCAcoord =',maxRowCrackCAcoord
          write(logUnit,*,ERR=901) '                maxColCrackCAcoord =',maxColCrackCAcoord
          write(logUnit,*,ERR=901) '                step_index =',step_index
          write(logUnit,*,ERR=901) '                crack_shape =',crack_shape
          write(logUnit,*,ERR=901) '                crack_location =',crack_location
          write(logUnit,*,ERR=901) '                geom_type =',geom_type
          write(logUnit,*,ERR=901) '                symmetry_flag =',symmetry_flag
          write(logUnit,*,ERR=901) '                debug_level =',debug_level
          write(logUnit,*,ERR=901) '                logFile =',TRIM(logFile)
          write(logUnit,*,ERR=901) '                modelFile =',TRIM(modelFile)
          write(logUnit,*,ERR=901) ' GlobalCrackCoord() values'
          write(logUnit,1,ERR=901) 'row','X-coord','Y-coord','Z-coord'
          do row=1,numCrackNode

          write(logUnit,2,ERR=901) row,GlobalCrackCoord(row,1:maxColGlobalCrackCoord)
          end do
          close(unit=logUnit)
      end if

1      format(a10,3a16)
2      format(i10,3es16.6)
3      format(6a)
!
! Initialize variables
!
      CrackCAcoord = DBLE(0) ! initialize all the values in the array

```

```

model_name_1 = 'user_cyl_ext'
model_name_2 = 'user_cyl_int'

if(logFlag.ge.debug_level) then
  open(unit=logUnit,file=logFile,position='append',err=901,iostat=stat_err)
! jump to bottom on error
  write(logUnit,*,ERR=901)
  write(logUnit,*,ERR=901) 'available model names:'
  write(logUnit,*,ERR=901) '   model_name_1 =',TRIM(model_name_1)
  write(logUnit,*,ERR=901) '   model_name_2 =',TRIM(model_name_2)
  close(unit=logUnit)
end if

!
! Initial error checking
!
  if(numCrackNode.le.0) then
    mflag = 2
    message = 'ERROR: zero crack front nodes are defined, numCrackNode = 0;
cannot compute the custom crack c,a crack coordinates in the user-defined geometry
reverse transformation routine (UserReverseTransform).'

```



```

        if(logFlag.ge.debug_level) then

            open(unit=logUnit,file=logFile,position='append',err=901,iostat=stat_err) ! jump
to bottom on error
                write(logUnit,*,ERR=901)
                write(logUnit,*,ERR=901)'2. begin cylinder internal crack example'
                write(logUnit,*,ERR=901)'    radius_in =',radius_in
                write(logUnit,*,ERR=901)'    thickness =',thickness
                close(unit=logUnit)
            end if

            !
            !           get the crack front node global y and z coordinates;
            !           compute the radius from the x-axis (cylinder center line) to the
            !           node; compute the angle theta around the cylinder;
            !           use the radius and angle to compute the crack c,a position
            !
            do row=1,numCrackNode
                node_y = GlobalCrackCoord(row,2) ! get the node global y-
coordinate
                node_z = GlobalCrackCoord(row,3) ! get the node global z-
coordinate
                radius_node = DSQRT(node_y**2 + node_z**2) ! radius from the
cylinder center to the node
                if(radius_node.le.DBLE(0)) then ! check for a positive radius
to the node
                    mflag = 2
                    message = 'ERROR: the radius to the custom crack front node
is zero (internal crack example); check the custom crack front global coordinates; cannot
compute the custom crack c,a crack coordinates in the user-defined geometry reverse
transformation routine (UserReverseTransform).'

```

```

        write(logUnit,*,ERR=901)
        write(logUnit,*,ERR=901)'Final values:'
        write(logUnit,*,ERR=901)'          numCrackNode =',numCrackNode
        write(logUnit,*,ERR=901)' CrackCAcoord() values'
        write(logUnit,1,ERR=901)'row','c_length','a_depth'
        do row=1,numCrackNode
            write(logUnit,2,ERR=901) row,CrackCAcoord(row,1:maxColCrackCAcoord)
        end do
        close(unit=logUnit)
    end if

!
!-----
! Exit procedures; jump here on an error to gracefully exit the routine
!
900 continue

    if(mflag.gt.0) then
        IntArray(1) = mflag
        StringArray(1) = message
    end if

    if(logFlag.ge.debug_level) then
        open(unit=logUnit,file=logFile,position='append',err=901,iostat=stat_err)
! jump to bottom on error
        write(logUnit,*,ERR=901)
        write(logUnit,*,ERR=901)'          mflag =',mflag
        if(mflag.gt.0) then
            write(logUnit,*,ERR=901) TRIM(message)
        end if
        write(logUnit,*,ERR=901)'----- End of UserReverseTransform -----'
    end if
    close(unit=logUnit)

end if

return
!
!-----
! Jump here on a file error
!
901 continue
    mflag = 2          ! error
    select case(stat_err)
    case(9) ! Permission to access file denied
        message = 'Error while opening the log file in the UserReverseTransform
DLL routine: file access denied (run-time error 9); the file permission or drive could be
read-only; if the file is on a CD-ROM copy to the hard drive and rebuild the mesh so that
the mesh files can be saved.'
    case default
        message = 'Error while opening the log file in the UserReverseTransform
DLL routine; check if the file permission or drive is read-only, or if the file is on a
CD-ROM (cannot write the files to the CD).'
    end select
    logFlag = 0          ! deactivate further use of the log file
    goto 900            ! jump up to exit procedure
!
end subroutine UserReverseTransform

```

TABLE OF FIGURES

Figure 1 FEACrack home form; select the units and analysis type.....	9
Figure 2 navigation bar.....	10
Figure 3 Geometry form, select the geometry model.....	11
Figure 4 Configuration form to select the crack location and model symmetry	12
Figure 5 enter the crack dimensions.....	13
Figure 6 enter a list of crack sizes to generate multiple cracks	13
Figure 7 enter the model dimensions.....	14
Figure 8 enter additional model dimensions.....	14
Figure 9 select nodes to get nodal result values in the data table.....	15
Figure 10 General through-thickness crack profile table	16
Figure 11 A custom thumbnail shape through crack.....	16
Figure 12 An S-shape through crack based on the fu 1	17
Figure 13 A jagged shape front through crack. 1.....	17
Figure 14 Von Mises stress results for a cell through crack mesh with a thumbnail shape.....	18
Figure 15 Von Mises stress results for a focused through crack mesh with a thumbnail shape.	19
Figure 16 J-integral results for the thumbnail shape through crack. The focused crack meshes use the crack front contours; the cell crack meshes use the remote domain.	19
Figure 17 enter the crack shift distances relative to the model.....	20
Figure 18 enter the crack bevel and orientation angles to rotate the crack relative to the model..	21
Figure 19 crack location relative to corner node “C1” in a user-defined geometry	22
Figure 20 the crack has been shifted and rotated relative to the center of the model.....	23
Figure 21 Change the color scale bar value range.....	24
Figure 22 Update the display controls.....	25
Figure 23 Help/About form shows program data including the host ID number and log file location.....	26
Figure 24 On the Home form enter optional text for the analysis name, model label, and notes .	27
Figure 25 General analysis controls for FEA type and result choices.....	28
Figure 26 Analysis parameter controls for Warp3D	29
Figure 27 Enter minimum and maximum iteration values used in an analysis step.....	30
Figure 28 Check the option by item “4” to use the “-i” and “-o” options on the ANSYS command line	31
Figure 29 Select one of four functions to automatically fill the load step data in the Constraints forms.....	33
Figure 30 Load history increment sizes computed by each function choice; example using 25 steps	34
Figure 31 Load history for each function choice; example using 25 steps	34
Figure 32 File Translator GUI in the FEACrack/Tools menu	36
Figure 33 Example of running the file translator interactively from the console window command line	37
Figure 34 Enter key word text in the Notes field to adjust the Intel MKL out-of-core options	39
Figure 35 Tools/Program Options form to set FEA choices	40
Figure 36 Set file extensions and program locations.....	42
Figure 37 General graphics controls.....	43
Figure 38 Crack Analysis Params to set maximum mesh size and other options	45
Figure 39 activate the weld heat affected zone.....	51
Figure 40 An example of a PRO focused crack mesh with the weld and HAZ material is shown below.	52

Figure 41 example of a crack in the center of the weld (green) with HAZ (tan) and base material (red)	52
Figure 42 In the Load Pins Options menu select the load pins location or hide the load pin menus	54
Figure 43 Load pin boundary condition menus on the right and left sides of the plate geometry	54
Figure 44 Load pin analytic rigid cylindrical surface data	55
Figure 45 Edge crack plate model results with 3-point bending using the load pin constraints ...	56
Figure 46 Corner crack plate with 3-point bending using load pins on the top and bottom surfaces	56
Figure 47 set the number of time steps and select steps with output.....	57
Figure 48 use the default load step values or enter load step values	58
Figure 49 enter crack face pressure polynomial values.....	59
Figure 50 Example of the initial crack face pressure table.....	60
Figure 51 Transposed crack face pressure table	61
Figure 52 Transposed Crack Orientation	61
Figure 53 Transposed crack face pressure table to enter in FEACrack.....	61
Figure 54 example of the pressure table for linear crack face pressure.....	62
Figure 55 example of the transposed tabular crack face pressure to enter in FEACrack	62
Figure 56 enter constraint values at available locations	63
Figure 57 enter displacement constraints; enter a value for a non-zero constraint.....	64
Figure 58 user defined load history values, used in post-processing plot x-axis	65
Figure 59 constraint example	66
Figure 60 constraint example	66
Figure 61 constraint example	66
Figure 62 constraint example	67
Figure 63 constraint example	67
Figure 64 constraint example	67
Figure 65 Select the “Submodel Boundary” for one or more mesh surfaces	68
Figure 66 Include key text: “*use ansys node file” in the Notes to activate the ANSYS node file	69
Figure 67 Key word text in the Home/Notes form to activate the tabular temperature option..	71
Figure 68 Example of a tabular temperature data file.....	72
Figure 69 Check the temperature option to activate the temperature boundary condition	73
Figure 70 Use the temperature values as a multiplier for each analysis step.....	73
Figure 71 Key word text in the Home/Notes form to activate the tabular stress option.....	74
Figure 72 Example of a tabular stress data file.....	75
Figure 73 Select Custom Stresses to activate the stress boundary condition.....	76
Figure 74 Use the membrane stress values as a multiplier for each analysis step	76
Figure 75 material data form for Warp3D to select the Mises material option	78
Figure 76 Warp3D cyclic material parameters.....	78
Figure 77 Select incremental plasticity and the stress-strain curve for Abaqus hardening options	79
Figure 78 Select the hardening key word for the Abaqus input file.....	80
Figure 79 Select the user-defined material option on the Material Properties form	81
Figure 80 Enter the number of dependent variables and the material constants.....	82
Figure 81 mesh view	84
Figure 82 mesh view close up	85
Figure 83 Run FEA batch control form; select input files and add to the queue; select the option to pipe the FEA solution status file to the bottom window	86
Figure 84 FEA Results form, select result files for post-processing	88
Figure 85 Crack front position by <input type="checkbox"/> angle or d	90
Figure 86 FEA results, color map of stress results on the mesh picture.....	91

Figure 87 Crack results, plot of J-integral results along the crack front.....	92
Figure 88 Crack results, plot of the non-dimensional G value.....	92
Equation 89 J path dependence index	92
Figure 90 Crack results, elastic-plastic multi-step J results.....	93
Figure 91 Crack results, FAD plot computed from elastic-plastic J-integral results.....	95
Figure 92 Crack results, reference stress computed from elastic-plastic J-integral results	95
Figure 93 Check the option box to activate the inner plastic zone mesh option.....	97
Figure 94 Enter the number of elements and the element size bias for the inner plastic zone. Enter the number of J-integral contours.	98
Figure 95 Enter the crack tube radius and inner plastic zone radius.....	98
Figure 96 Circumferential surface crack with the inner plastic zone mesh option	99
Figure 97 Close up of the crack front showing the inner plastic zone	99
Figure 98 C(T) specimen mesh with the inner plastic zone.....	100
Figure 99 Close up of the crack front and inner plastic zone.....	100
Figure 100 J Contour Plotting tool; plot the J-integral versus the contour index to examine the path dependence	101
Figure 101 The average J along the crack front is computed using the starting and ending contour values	102
Figure 102 J-integral contour data available to copy from the Data tab.....	103
Figure 103 Normal crack front refinement.	104
Figure 104 3x refinement along the entire crack front.....	105
Figure 105 9x refinement along the entire crack front.....	105
Figure 106 3x refinement near the free surface.	106
Figure 107 9x refinement near the free surface. Note the 3x transition zone.	106
Figure 108 Standard pattern for $a/t = 0.25$. The shallow crack pattern is not used in this case. 107	
Figure 109 Shallow crack pattern for $a/t = 0.25$	108
Figure 110 General mesh controls.....	109
Figure 111 Transitioning to(1) element through the thickness.....	110
Figure 112 Maintaining through-thickness refinement by extruding the mesh pattern away from the crack.	110
Figure 113 All mesh stretch parameter set to 1.0.....	112
Figure 114 All mesh stretch parameters set to 4.0.	112
Figure 115 Variable mesh stretch multipliers. Crack Block = 1.0, X Multiplier = 4.0, Y Multiplier = 0.5, Z Multiplier = 4.0.....	113
Figure 116 Refinement indices. $N_x = N_y = N_z = 1.0$	113
Figure 117 Refinement indices. $N_x = N_y = N_z = 4.0$	114
Figure 118 Custom macro text to replace the ANSYS SOLVE command in the input file. .	114
Figure 119 Mesh subset sizes given from the ends of the crack.	117
Figure 120 All of the surface crack mesh.....	118
Figure 121 Default subset mesh, subset uniform region size based on the crack size.	119
Figure 122 Uniform subset region defined by a single value.....	120
Figure 123 Custom subset mesh size defined by 6 values; large Delta Y to include the mesh through the thickness.....	121
Figure 124 Step 1, run the packet reader from the command line, enter the binary packets file name	122
Figure 125 Step 2, use short cut, enter “0” to select all available packets	123
Figure 126 Step 3, use default text packets file name, write the text packets file	123
Figure 127 Internal circumferential surface crack in a cylinder	124
Figure 128 Enter 6 size values for the subset region	125
Figure 129 Cylinder internal surface crack subset set WARP3D results.....	126

Figure 130	The J-integral results are the same from the WARP3D result and packets files.	127
Figure 131	Cylinder subset J-integral results from Abaqus.....	128
Figure 132	Cylinder internal surface crack subset set Abaqus results.....	128
Figure 133	Plate with surface crack, plasticity J results, subset mesh comparison.	129
Figure 134	Sigma Z results for the full plate mesh.....	130
Figure 135	Sigma Z results for the subset plate mesh	130
Figure 136	Example of a cell-type crack mesh	131
Figure 137	Focused crack in the cell mesh.....	132
Figure 138	PRO crack mesh zones for the surface crack shape.	133
Figure 139	PRO crack mesh zones for the through-thickness crack shape.	134
Figure 140	Select the cell crack mesh on the home form.....	135
Figure 141	Cell meshcrack front basic parameters.	136
Figure 142	Effect on the cell crack mesh of setting $\square c$ to twice $\square a$	137
Figure 143	Focused mesh crack front basic parameters.	137
Figure 144	Advanced PRO/cell mesh parameters.....	138
Figure 145	More advanced mesh parameters	139
Figure 146	Crack front transition zone.....	140
Figure 147	Crack growth transition zone	141
Figure 148	Advanced mesh parameters	142
Figure 149	Crack face mesh zones and crack face transition zone	143
Figure 150	Advanced crack mesh parameters on the right side of the crack front	144
Figure 151	Right size mesh zone and transition zone	145
Figure 152	Remaining width mesh zone and transition zone.....	146
Figure 153	Key word text in the Home/Notes form	147
Figure 154	Example of using the PRO mesh option to set the number of uniform size elements and get vertical mesh lines in the ligament below the surface crack.....	148
Figure 155	The standard PRO mesh in the ligament below the surface crack.....	148
Figure 156	The inner crack tube length	149
Figure 157	The inner mesh box length and multiplier.....	150
Figure 158	Example of the inner crack tube length and inner box multiplier key words.....	151
Figure 159	Example of a large radius groove using the inner mesh zone options.....	152
Figure 160	Control the output steps and crack front release	153
Figure 161	Cell crack mesh, analysis step 1, initial crack size before node release.	154
Figure 162	Cell crack mesh, analysis step 61, final crack size after 14 crack front node releases.	155
Figure 163	J versus crack extension result plot	156
Figure 164	Example of a focused crack mesh.....	157
Figure 165	Example of a cell-type crack mesh	157
Figure 166	The 30 elements, colored green, in the fifth J-integral contour in the cell crack mesh are shown in the figure below.	158
Figure 167	Elements in contour #5 in the cell crack mesh.	159
Figure 168	Comparison of J-integral results for the unit cube crack mesh example.....	160
Figure 169	Comparison of individual J-integral contour results	161
Figure 170	Comparison of contour 5 J-integral results	162
Figure 171	Select the “complex crack” shape and define the custom crack length and depth data	163
Figure 172	Use the number of cycles or number of steps to grow the complex crack	164
Figure 173	Complex custom crack shape; the part-through crack front around the circumference curves to the outside surface for a through-thickness segment of crack at the top of the cylinder geometry.	164

Figure 174	Identify the definition mesh corner node ID numbers for the comment data block	167
Figure 175	Doubly curved plate user-defined geometry imported definition mesh; 24 objects.	169
Figure 176	Crack mesh in the user-defined geometry; the crack is shifted and rotation from the center of the model.	170
Figure 177	Identify the master mesh input file name after the key word.	171
Figure 178	Piping elbow model; two mesh regions for the user-defined geometry.	173
Figure 179	Definition mesh used to describe the geometry of the crack region; the definition mesh is imported into FEACrack.	174
Figure 180	Definition mesh removed from the elbow model before the crack mesh is inserted.	174
Figure 181	Piping elbow with a flank surface crack; the crack mesh (red) has been added to the rest of the elbow model (blue) and the two mesh regions are connected by tied contact.	175
Figure 182	Constant amplitude fatigue analysis; enter the loading amplitude value	179
Equation 183	Weighted average growth rate	179
Figure 184	Variable amplitude fatigue analysis; enter the Rayleigh values	180
Figure 185	Variable amplitude fatigue analysis; enter the Tabular Histogram values	180
Figure 186	Histogram calculations for variable amplitude fatigue analysis	181
Figure 187	Select the fatigue growth equation and enter the Fatigue values	182
Figure 188	Fatigue analysis controls; enter the crack increment and final size; select crack front K options	183
Figure 189	Fatigue analysis run-time controls; select the FEA program, enter the file save frequency, activate the fatigue log file option	184
Figure 190	Fatigue analysis log file lists detailed results	184
Figure 191	Fatigue analysis K smoothing controls	185
Figure 192	Example of a numerical spike in the crack front K results at each mesh surface; semi-circular surface crack, $c/a = 1$	186
Figure 193	Comparison of crack front K values using the polynomial smoothing and discarded end point values	187
Equation 194	Crack growth integration	187
Figure 195	Surface flaw crack growth	188
Figure 196	Subsurface flaw crack growth	188
Equation 197	Crack growth rates	189
Equation 198	Incremental form	189
Equation 199	An Increment number of cycles	189
Equation 200	Trapezoidal integration	189
Equation 201	Fatigue crack growth from the previous increment, i	190
Equation 202	Average number of cycles	190
Equation 203	Given the depth increment, a , compute the length increment, c	190
Equation 204	Update crack length	190
Equation 205	Crack length contributions from both crack tips	191
Equation 206	Update total crack length	191
Equation 207	Update number of cycles in the increment	191
Figure 208	Crack depth versus cycles fatigue results	192
Equation 209	K solution for semi-elliptical surface crack	193
Equation 210	K solution geometry shape factors	193
Equation 211	Simplify for a semi-circular crack	193
Equation 212	Simplified geometry factor, \square	193
Equation 213	Simplified geometry factor, f	193
Equation 214	Simplified K solution for semi-circular crack	194

Equation 215	Paris Fatigue Equation.....	194
Equation 216	Substitute semi-circular K solution	194
Equation 217	Closed-form solution for the number of fatigue cycles, semi-circular crack example	194
Table 218	Parameters for Plate Surface Crack Fatigue Example.....	195
Figure 219	Initial semi-circular surface crack; quarter symmetric plate mesh	195
Figure 220	Close up of the initial semi-circular surface crack	196
Figure 221	Final surface crack mesh after fatigue crack growth.....	196
Figure 222	Close up of final surface crack mesh.....	197
Figure 223	Crack depth fatigue results, semi-circular example.....	198
Figure 224	Crack length fatigue results, semi-circular example.....	198
Figure 225	Stress interpolation parameters	199
Figure 226	Set the interpolation parameters and select the file names for the stress interpolation	202
Figure 227	Activate the external DLL module for custom crack fatigue analysis in a user-geometry	203
Figure 228	Cylinder custom crack reverse transformation calculation example; the crack depth a is radial, the crack length c is the arc length.....	204
Figure 229	Select the Abaqus part and assembly syntax option.....	205
Figure 230	Select the “Focused PRO Mesh” and then check the “Perpendicular Radial Mesh Lines in Crack Tube” option	207
Figure 231	Normal radial mesh lines extend past the crack tube ahead of the crack.	208
Figure 232	Normal radial mesh lines in the crack tube only with two crack front transition zones.	209
Figure 233	Standard PRO mesh, the radial mesh lines do not remain normal to the crack front.	209
Figure 234	Mid side position option key word and value given in the Home/Notes form	210
Figure 235	Example crack mesh with the mid side node position at 0.4.....	211
Figure 236	Example: close-up of a crack mesh with the mid side node position at 0.4 from the crack front	211
Figure 237	Example: close-up of a crack mesh with the mid side node position at 0.4 from the crack front	212
Figure 238	Radial point results form; select the method; click on Load Data after loading an FEA results file.	214
Figure 239	Check boxes to select the crack front node and result value; selected crack front nodes highlighted in yellow.....	215
Figure 240	Data sheet contains the selected data values, one page per result set. The spreadsheet edit bar allows cell values to be viewed or changed.	216
Figure 241	X-Y plot of the Sigma Z stress versus the radial distance for a selected result set (centroid results).....	217
Figure 242	Plot variable selection form. Choose the x-axis variable from the left menus; choose the y-axis variable from the center menus.....	217
Figure 243	Select results, labels, and enter values to normalize the radial point result values.	219
Figure 244	Normalized result values with column labels. Values in a cell are shown in the edit bar.	220
Figure 245	Plot the normalized result values.....	220
Figure 246	A2 Calculator, enter normalized stress profile	224
Figure 247	Compare the input data to the power-series stress equation evaluated using J and A2	225
Figure 248	Enter the normalized stress versus normalized radial distance to compute Q.....	226

Figure 249	Plot the reference and second analysis normalized solutions	227
Figure 250	Plot the compute Q values versus normalize radial distance from the crack front. 228	
Figure 251	Computed Q values	228
Figure 252	Normalized crack plane stress results compared to text book values; the reference solution used to compute the Q-stress for another analysis geometry.....	230
Figure 253	Compare K from J and applied K values to examine the crack front plastic zone effect; K from J begins to diverge at about 25,000 psi*sqrt(in)	231
Figure 254	Alpha-H Calculator form to select stress results and enter radial point values ...	232
Figure 255	Computed σ_h values along the crack front.....	233
Figure 256	Computed σ_h values and plastic zone data.....	233
Figure 257	Radial point FEA results to compute σ_h	233
Figure 258	ASTM J Calculator in FEACrack.....	234
Figure 259	J versus Load Step from the ASTM J calculator	235
Figure 260	Use the predefined node set to select the node for the load line displacement....	236
Figure 261	Part of the selected node result data sheet.	237
Figure 262	Node set with all constrained nodes in the load pin to obtain the total load.	238
Figure 263	J-R curve from the cohesive C(T) specimen	239
Figure 264	J results for the quarter symmetric C(T) specimen with a static crack front	240
Figure 265	J results for the half symmetric C(T) specimen with cohesive material and a growing crack front	241
Figure 266	SE(B) specimen.....	242
Figure 267	Enter a zero size roller diameter or load ram radius for the SE(B)	243
Figure 268	Activate the optional second material group at the boundary constraint and load locations.....	243
Figure 269	Elastic material can be selected for the SE(B) second material group at the constraint and load locations.....	244
Figure 270	SE(B) mesh with zero size constraints at the rollers and load ram; the blue elements show the second material group	245
Figure 271	Comparison of the zero width and finite width load ram radius and the number of elements in the second material group	245
Figure 272	Additional SE(B) specific PRO mesh options	247
Figure 273	Locations of the SE(B) specific PRO mesh options; the RD and CD options are standard PRO mesh options and are located on the Crack Face form.....	248
Figure 274	Activate the SE(B) side groove option using key word text in the Home/Notes form	249
Figure 275	SE(B) side groove option locations for the grove half width and ligament thickness	249
Figure 276	Comparison of the SE(B) model with and without the side groove option	250
Figure 277	Load-displacement curve using the extracted node results.	251
Figure 278	Crack mesh using the pre-defined “C(T) Specimen” View Type.	252
Figure 279	C(T) specimen extracted node force and displacement results.	254
Figure 280	Load-displacement curve using the extracted node results.	254
Figure 281	Select the C(T) specimen side groove option in the Geometry/Crack/Configuration form	255
Figure 282	Select the User Defined Values option to activate the side groove height h_g input	256
Figure 283	Select C(T) specimen half model with a side groove; dimension h_g is within: precrack notch $h_4 \leq h_g \leq h_3$ main notch	256
Figure 284	Modified boundary layer model, half symmetric mesh.....	257

Figure 285	Close up of the uniform mesh size refinement near the crack front in the estimated plastic zone	258
Figure 286	Enter the size bias factors, and the number of elements inside and outside the plastic zone	259
Figure 287	Enter the model thickness, B, and the estimated plastic zone radius, R _p	259
Figure 288	Estimated plastic zone radius calculator; enter K and σ_0	260
Figure 289	Enter K and T-stress to compute the imposed perimeter displacement for plane-strain	261
Figure 290	Enter K and T-stress to compute the imposed perimeter displacement for plane-stress	261
Figure 291	Example of the standard boundary layer model with a focused mesh.....	262
Figure 292	Example of the standard boundary layer model with a cell type mesh	262
Figure 293	BLM with more mesh refinement and smaller cell element size	263
Figure 294	BLM with more mesh refinement and smaller cell element size; close up	263
Figure 295	BLM with more mesh refinement and smaller cell element size; close up at the crack front	264
Figure 296	Example of BLM key word text in the FEACrack Home/Notes form	265
Figure 297	Example of the BLM KT data file containing the K and T-stress values.....	266
Figure 298	Select the “Material Group” and “Unique Materials” options for the plug weld model	268
Figure 299	Crack arc angles are measured counter-clockwise from the centerline on the back side (-Z) of the plug	269
Figure 300	Use default or user defined values for the plug weld model dimensions	270
Figure 301	Use default or user defined values for the plug weld mesh options	271
Figure 302	Select the plug bevel case in the Bevel Options menu	271
Figure 303	Some plug bevel and crack shapes activate the Plug Automatic Tied Contact option	272
Figure 304	Select constraints and loads on the plug weld surfaces	273
Figure 305	The plug weld model has 12 material groups.....	274
Figure 306	Material Group choice: the plug is entirely within the weld, 8 material groups active	275
Figure 307	Materials form: plug entirely with the weld, 8 material groups active.....	275
Figure 308	Sub-Files option in the Tools/Program Options/Analysis form	276
Figure 309	Enter key text in the Notes for function type=d for through-thickness cracks	278
Figure 310	Enter keyword text in the Home/Notes for the automatic element blocking option	279
Figure 311	Use the File Selection and Preview form to view a mesh file	280
Figure 312	Add key word text in the Home/Notes form to activate the view crack face option	281
Figure 313	The standard full plate surface crack mesh picture in the main graphics window	282
Figure 314	View of the back crack face from the “modell_ViewCrackBack_fmp.neu” Femap file using the Tools/Show Stress Interpolation form; the Femap file is selected with the Browse button; click the Show check box to show the mesh.....	282
Figure 315	The View of the front crack face from the “modell_ViewCrackFront_fmp.neu” Femap file; the mesh has been rotated by setting the Y-axis rotation angle to 150o to view the front crack face.....	283
Figure 316	Enter key word text in the Notes field to activate the crack face area calculation.	284
Figure 317	Open the log file with a text editor to view the calculated area value.....	284