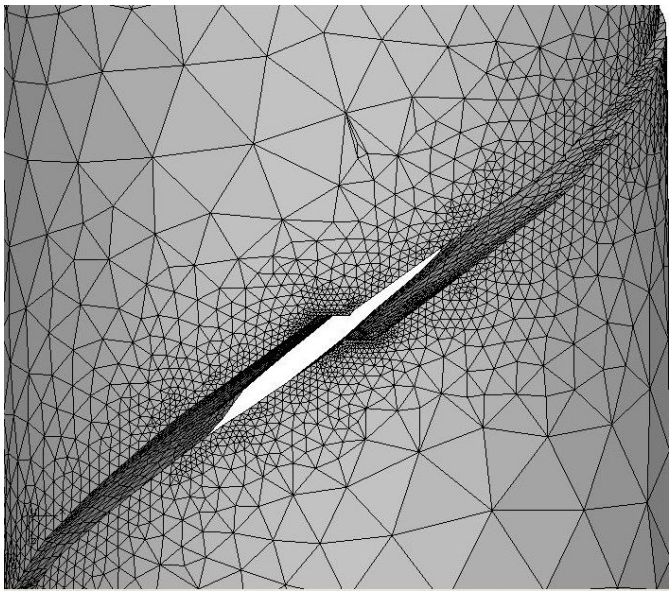
FRANC3D

Reference Manual



Version 9.2

Fracture Analysis Consultants, Inc
https://www.fracanalysis.com/

Revised: Aug 2026

Table of Contents:

1. Introduction

FRANC3D is a program that inserts cracks or voids in pre-existing finite element (FE) meshes. This manual describes the components of the graphical user interface (GUI) of the program, specifically for Version 9. It also includes underlying concepts and theory where appropriate; the User's Guide provides more details on concepts, and the (future) Theory Reference will provide more details on theory. A separate FRANC3D Command Language & Python Extensions reference provides a listing of all the commands to run in batch-mode along with the corresponding Python functions and a description of the Python user-crack-growth capabilities.

There are four FRANC3D Tutorial documents that describe the program usage when combined with the three supported commercial FE codes: ANSYS, ABAQUS and NASTRAN. There are preliminary Tutorials for each of the three analysis codes to describe the interface between FRANC3D and the analysis codes. A more generic Tutorial#2-14 document describes additional example simulations.

There is also a FRANC3D Benchmark document that describes crack configurations for which there are analytical or handbook solutions for stress intensity factors (SIFs), and the FRANC3D SIFs are compared to these values.

To get started using FRANC3D, one can choose one of the preliminary Tutorials and follow the steps. You can consult this Reference document for more details at any of the steps in the tutorial. You should also try to reproduce the Benchmark models or choose your own models for validation.

In this document, menu buttons, dialog or wizard panel titles, and buttons on these panels are indicated by **bold** text. Fields, labels and selectable options inside the dialog or wizard panels are indicated by <u>underlined</u> text. Model and file names will be indicated by *italic text.*

2. General and 3D View Manipulation

An image of the main FRANC3D window is shown in Fig 2.0.1. Most of the window consists of a 3D graphics space that displays the current model.

When a model is first displayed, the 3D view is set such that the viewer is looking toward the center of the model from a position in the positive z direction. The distance from the viewer to the model is set so that the full model is visible.

The 3D view is changed by moving the mouse in the graphics window with the appropriate combination of mouse buttons and keyboard keys. There are five basic functions for view manipulation. A unique combination of mouse buttons and keyboard keys is defined for each of the functions. The button and key assignments can be changed using the **3D View** tab in the **Preferences** dialog box (see Section 5.2.4).

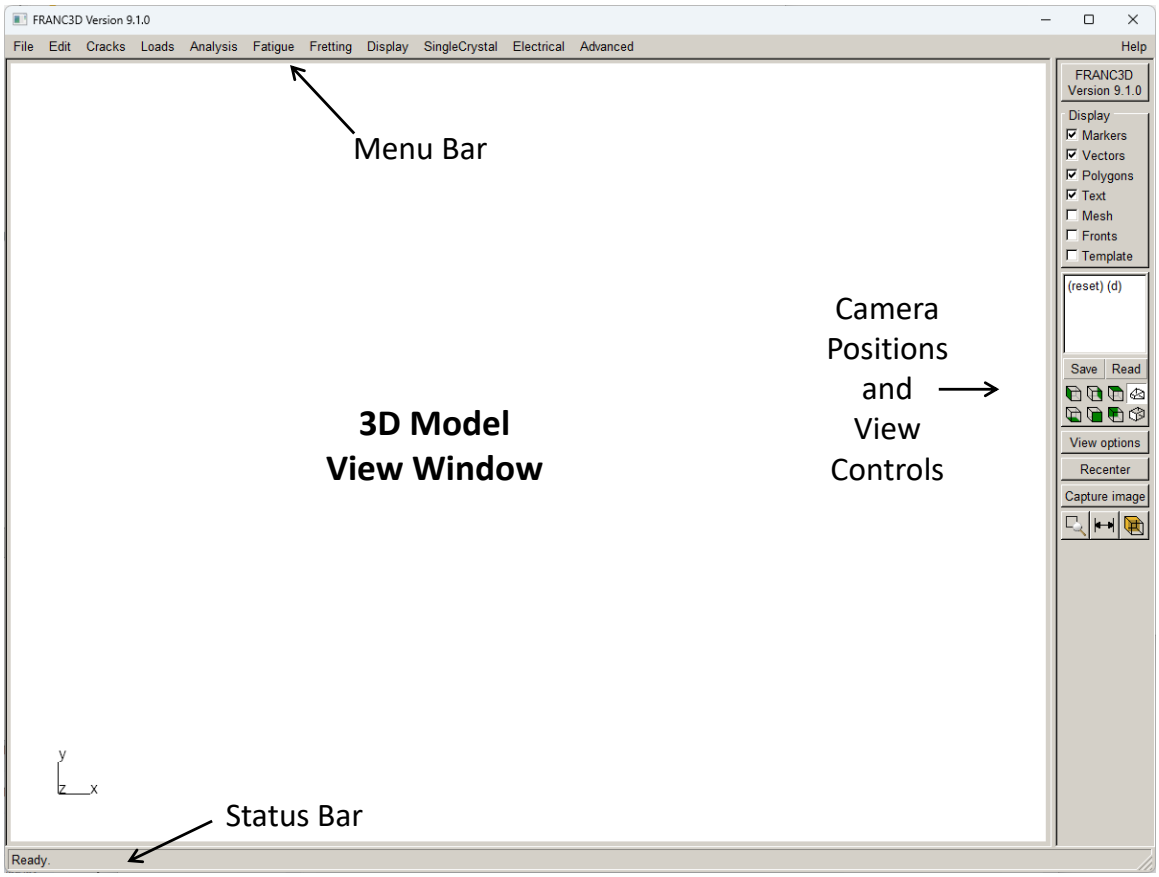


Figure 2.0.1 FRANC3D main window.

The view functions are (using the FRANC3D default mouse and keyboard key assignments):

Rotate (left mouse button, no keyboard keys): Mouse motion rotates the model on the screen. The mouse can be moved in any direction. The axis of rotation is defined to be perpendicular to the mouse motion and passes through the current center of rotation. The center of rotation is defined as the horizontal and vertical center of the graphics window at a location midway between the front and back clipping planes (described below). The center of rotation can be changed, Fig 2.0.2, using the Ctrl key and left mouse button (or with the **Recenter** button).

Pan (center mouse button or wheel, no keyboard keys): Mouse motion pans or drags the model around the screen. The mouse can be moved in any direction, and the model moves in such a way as if it were following the mouse.

Zoom/Spin (right mouse button, no keyboard keys): These are two distinct types of motion activated by the same mouse key. If the mouse is moved up or down, it will appear as if the viewer is getting closer or farther, respectively, from the model. If the mouse is moved left or right, the model rotates about an axis that is perpendicular to the plane of the screen and passes through the horizontal and vertical centers of the window.

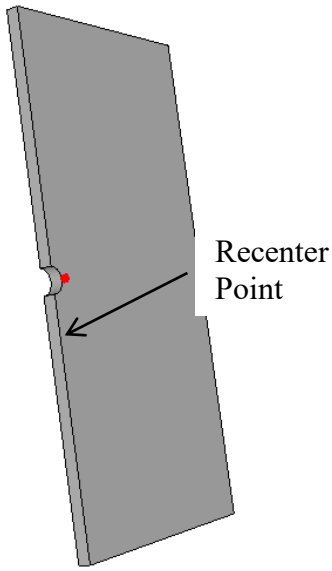


Figure 2.0.2 Recenter.

Front Clip (center mouse button or wheel plus shift key): A cutting (or clipping) plane can be moved parallel to the screen from the viewer toward the model. This can be used to clip away the front of a model to see internal features such as cracks. Moving the mouse upward "pushes" the clipping plane away from the viewer toward the model, cutting away the portion of the model closest to the viewer. Moving the mouse downward "pulls" the plane away from the model and toward the viewer.

Back Clip (right mouse button or wheel plus shift key): A cutting plane can be moved parallel to the screen from behind the model toward the viewer. This can be used to clip away parts of the model that might be making the view confusing (this is especially useful when the model is displayed in "wireframe" mode without polygons). Moving the mouse upward "pushes" the clipping plane away from the viewer. Moving the mouse downward "pulls" the clipping plane toward the viewer, which will cut away the back portion of the model.

2.1 View Controls

The view controls box, along the right-hand side of the FRANC3D main window, provides additional options for manipulating the view of the model, Fig 2.1.1.

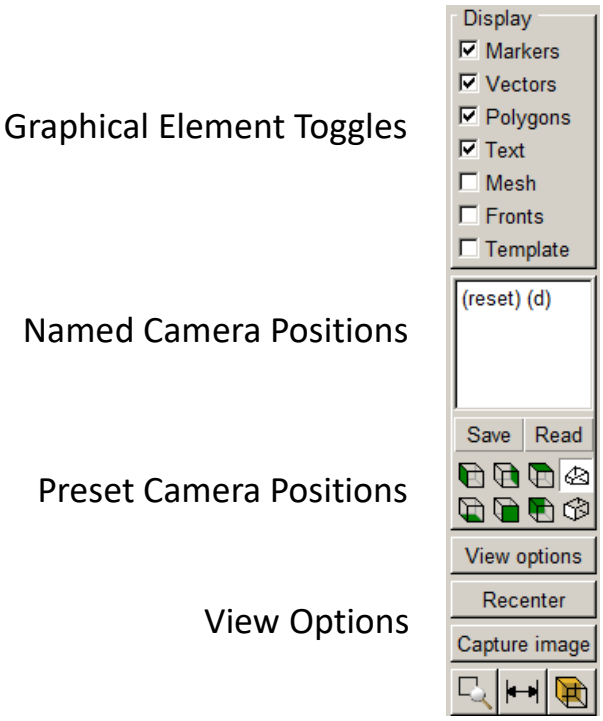


Figure 2.1.1 View controls side panel.

Graphical Element Toggles: Graphics in 3D display windows consist of a combination of markers, vectors, polygons, and text. The toggles in this box will turn these items on or off independently. The surface mesh, crack fronts and templates can be turned on or off as well.

Named camera positions: The list of named camera positions is displayed in the list box. There should always be a reset view provided as the default (d) view. A crack view will be saved automatically after a crack is inserted. User-named camera positions can be **Save**d to a file and **Read** from a file.

Preset Controls: These preset camera position icons provide quick access to preset views that place the viewer along one of the Cartesian axes. The two icons on the right switch the view between perspective and orthographic.

View Options: This button displays the dialog in Fig 2.1.2. In this dialog, the Show Axes turns the Cartesian axes display on or off. The Different Front/Back Colors turns the front/back coloring on or off. By default, the front and back sides of surfaces are shaded differently. This toggle is useful when looking at the crack surface, where the two surfaces coincide. The Set Speed sliders allow one to control rotation, translation, zoom, and spin speeds independently. The Cutting Planes option allows one to define cutting planes normal to the global Cartesian axes. These will cut away a portion of the model. These are independent from the front and back clipping planes.

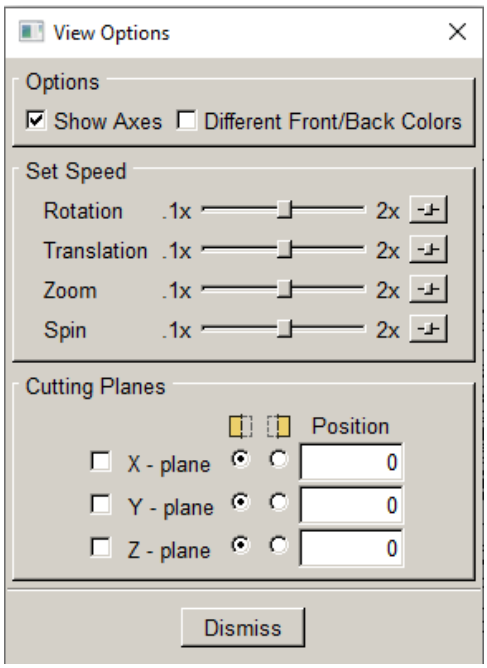


Figure 2.1.2 View Options dialog.

Recenter: This button displays the Set Rotation Center dialog, Fig 2.1.3. One can specify the center of rotation interactively, using the dialog in Fig 2.1.4; click the **Recenter** button to display this dialog.

A node ID or Cartesian coordinates can be specified. The Ctrl-key and left-mouse button is the quickest way to set the rotation center.

In the **Setcenter View** dialog, Fig 2.1.4. The center of rotation is set by dragging the red boxes left or right and up or down. The new center of rotation is the intersection of the corresponding three lines.

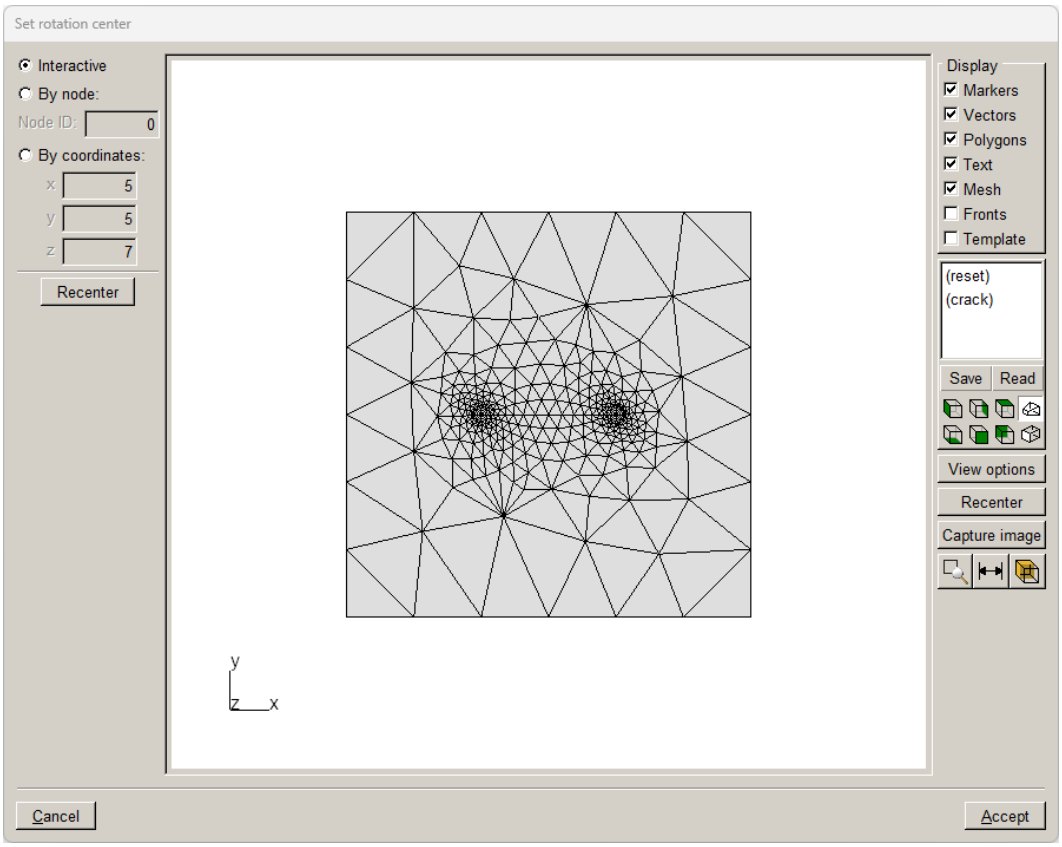


Figure 2.1.3 Set Rotation Center dialog.

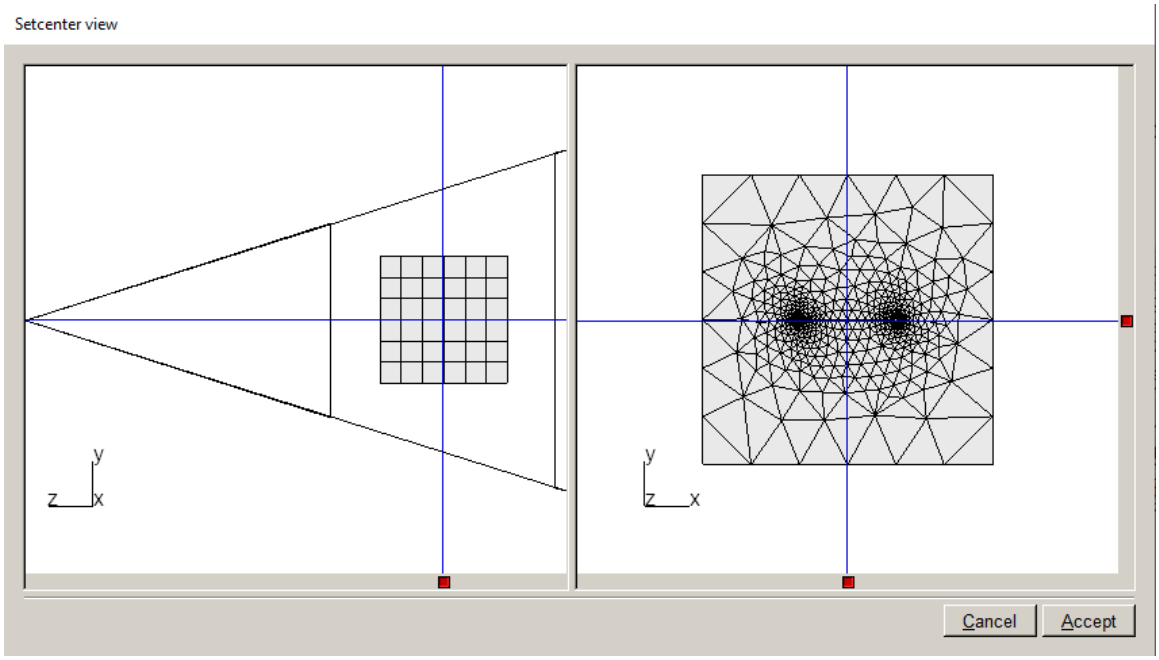


Figure 2.1.4 Setcenter View dialog.

Capture: This button displays the **Save File As** dialog, Fig 2.1.5. The current view of the model is captured and saved to a file. Either a *.png* or *.jpg* file can be saved.

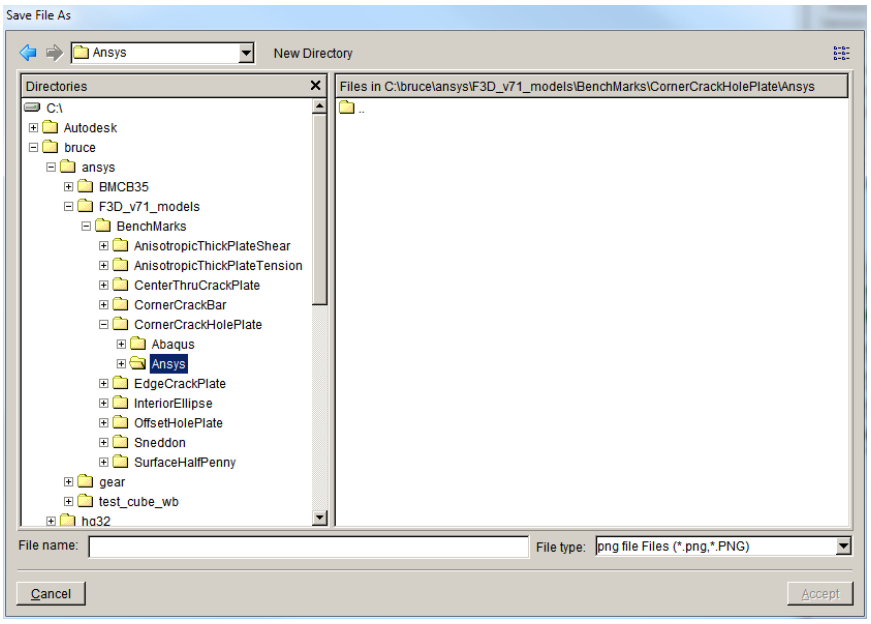


Figure 2.1.5 Save File As dialog.

The bottom of row of icons, Fig 2.1.6, allow one to do a box-zoom, to take measurements on the model surface, and to define front and back clipping planes.

Figure 2.1.6 Bottom row of icons: box-zoom, measurements, and clipping planes.

Box Zoom: This allows one to zoom-in on a portion of the model by dragging a box, Fig 2.1.7. Press the **Box Zoom** button and then, using the left mouse button, click and hold, then drag the mouse to create the box, and then release the mouse button.

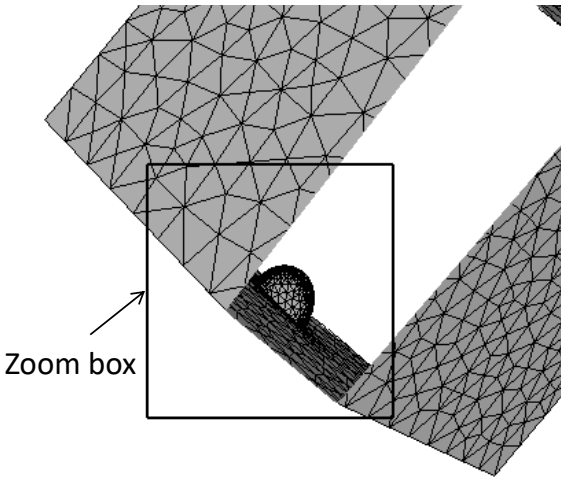


Figure 2.1.7 Box zoom used to zoom-in on a portion of the model.

Measurements: This allows one to find the Cartesian coordinates of a point on the model surface, or to find the distance between two points on the model surface, Fig 2.1.8. Using the left mouse button, click on the model surface to view the coordinates; click and hold and then drag the mouse to obtain the distance between two points on the model surface.

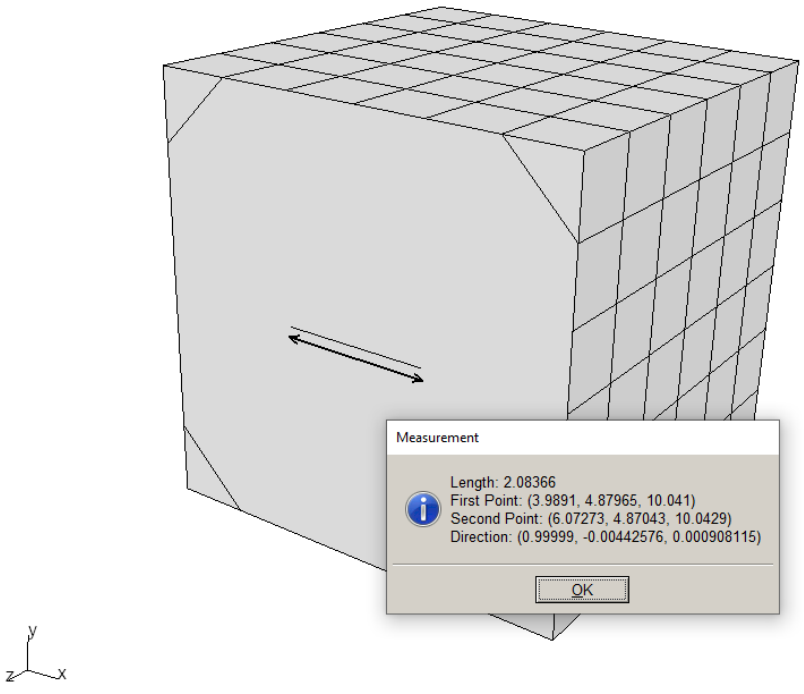


Figure 2.1.8 Surface measurement.

Clipping: This icon displays the **Clipping View** dialog, Fig 2.1.9. The front and back clipping planes can be moved by dragging the red boxes left or right. This is an alternative to the mouse-keyboard combinations described previously.

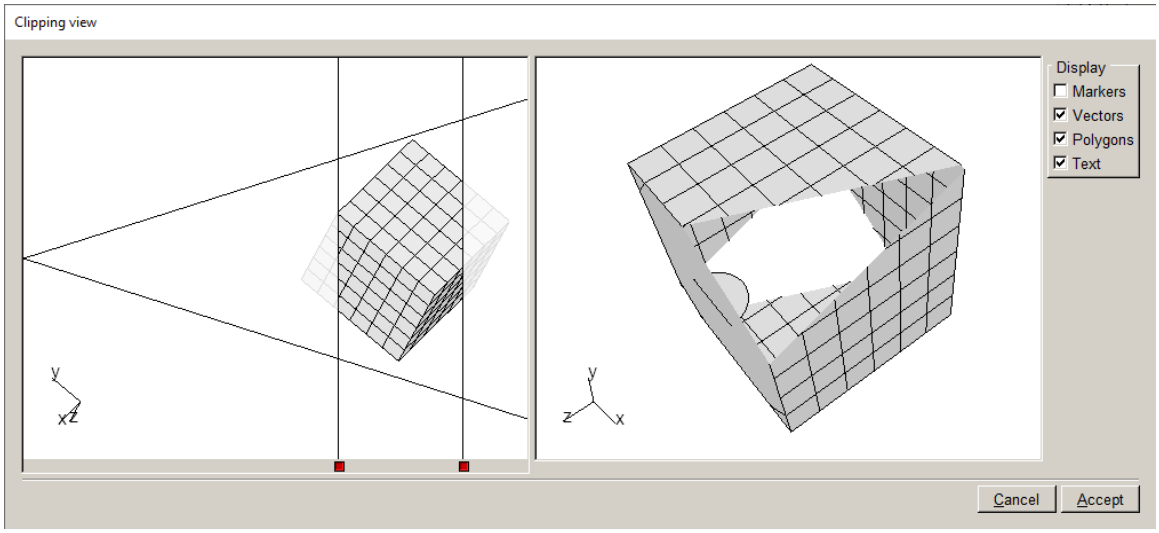


Figure 2.1.9 Clipping view dialog.

2.2 OpenGL Version

Click the FRANC3D nameplate, Fig 2.2.1, to display the OpenGL information dialog. Either version 4.5 or version 1.1 will be used depending on the operating system and graphics card. The version can be set in the Preferences (see Section 5.2.3).

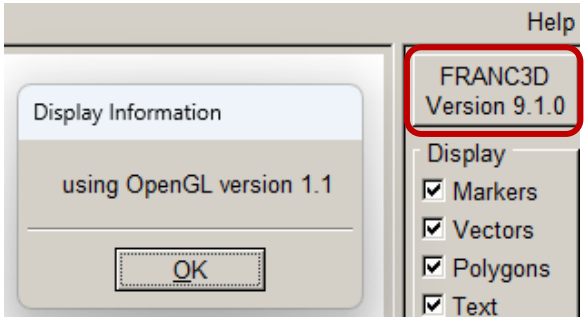


Figure 2.2.1 OpenGL display info.

3. Menus

A summary of the menus, from the menu bar shown in Fig 2.0.1, is provided here. In general, selecting a menu item leads to a wizard or dialog.

The **Help** menu is described here, while all other menu options are described in more detail in Sections 4 through 14.

3.1 Help

The **Help** menu, Fig 3.1.1, is on the far-right side of the menu bar and has options for accessing the FRANC3D documentation either locally, if installed, or on-line from FAC's web site (http:\\www.fracanalysis.com/software.html). The appropriate documentation will be displayed in a web browser if the files have been installed locally and the file path set. If the file path has not been set, the user is presented with a dialog box, Fig 3.1.2, and the appropriate *.pdf* file can be selected. Alternatively, if the **FRANC3D Website** menu option is selected, FAC's web site is accessed, and the appropriate documentation can be selected there.

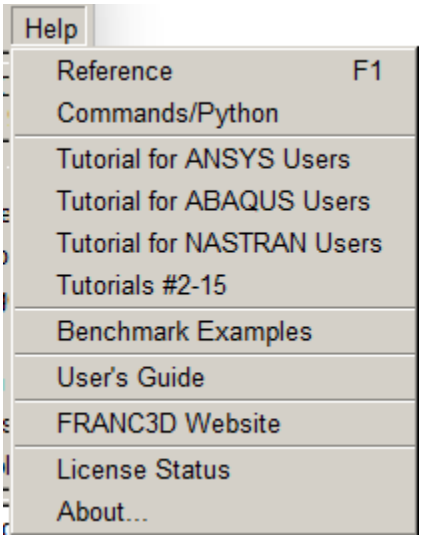


Figure 3.1.1 Help Menu.

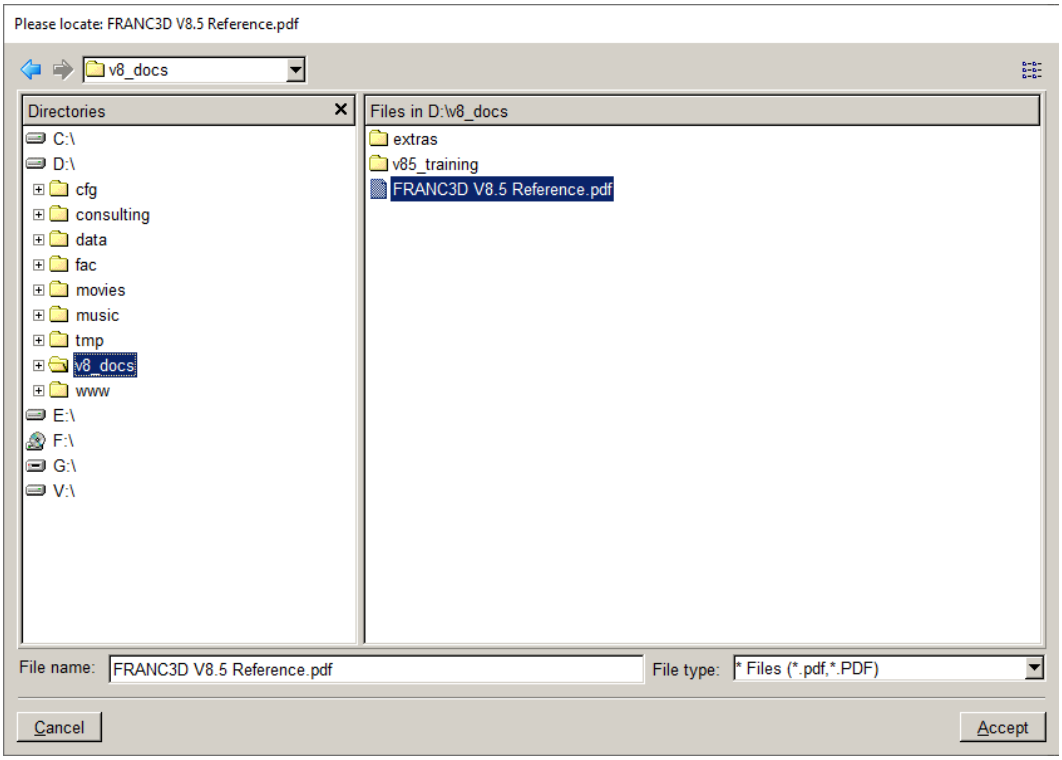


Figure 3.1.2 FRANC3D documentation file location dialog.

3.1.1 License Status

The second last menu item displays the **License Status** dialog, Fig 3.1.3, which shows the current license expiration.

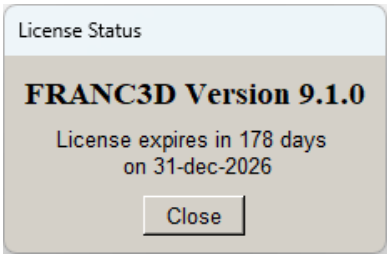


Figure 3.1.3 License status dialog box.

3.1.2 About Dialog

The last menu item displays the **About** dialog, which contains the version number, the build number and date, acknowledgements, and a list of supporting agencies, Fig 3.1.4. We also appreciate the help, support, and contributions of our users.

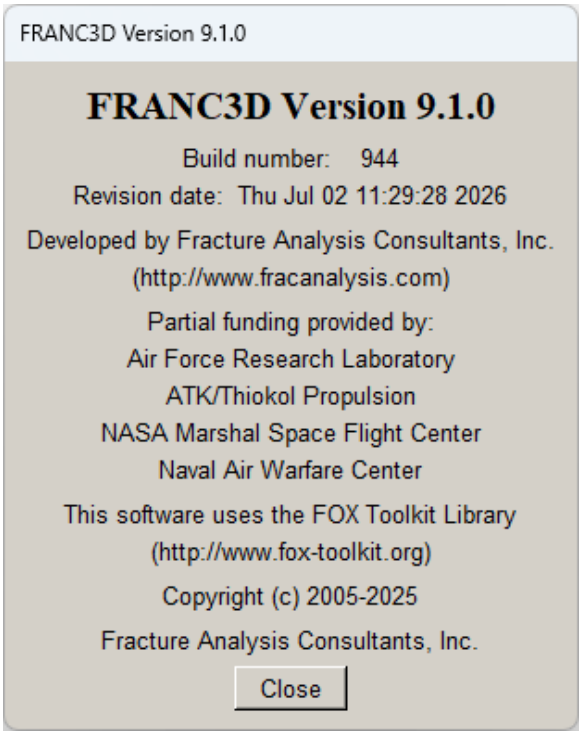


Figure 3.1.4 FRANC3D About dialog box.

3.2 File

The **File** menu is shown in Fig 3.2.1. Each of the menu items is listed below.

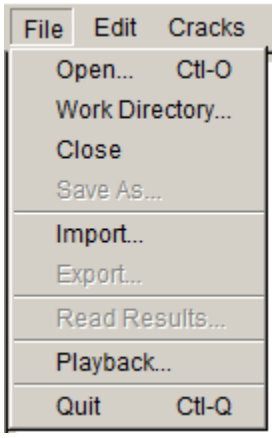


Fig 3.2.1 File Menu.

Open ... - Displays an **Open File** dialog box that allows a user to read a FRANC3D restart file; see Section 4.1.

Work Directory ... - Displays a dialog box that allows a user to set the working directory; see Section 4.2.

Close - Closes the current model so that a new model can be read or imported; see Section 4.3.

SaveAs ... - Displays a **Save File As** dialog box, which allows a user to save the current model to a FRANC3D restart file; see Section 4.4.

Import ... – Allows a user to import an uncracked FE model file. The file types that can be read include: *.cdb*, *.inp*, *.bdf*. The *.cdb* files are ANSYS database files in ASCII format. The *.inp* files are ABAQUS input files. The *.bdf* (or *.nas*) files are NASTRAN database files; see Section 4.5.

Export ... – Allows a user to export a FE model file. ANSYS *.cdb*, ABAQUS *.inp* and NASTRAN *.bdf* file formats can be written; see Section 4.6.

Read Results ... - Invokes the **Read Results File** dialog; see Section 4.7.

Playback ... - Invokes the **Playback Session File** dialog, which allows a user to read in a session log file and reproduce the commands in that file; see Section 4.8.

Quit - Closes the model and exits the program; see Section 4.9.

3.3 Edit

The **Edit** menu is shown in Fig 3.3.1. Each of the menu items is listed below.

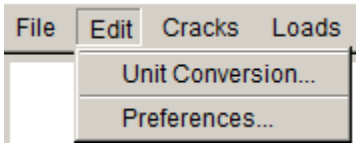


Figure 3.3.1 Edit Menu.

Unit Conversion... - Invokes the **Model Units** dialog box; see Section 5.1.

Preferences... - Invokes the **Preferences** dialog box; see Section 5.2.

3.4 Cracks

The **Cracks** menu is shown in Fig 3.4.1. Each of the menu items is listed below.

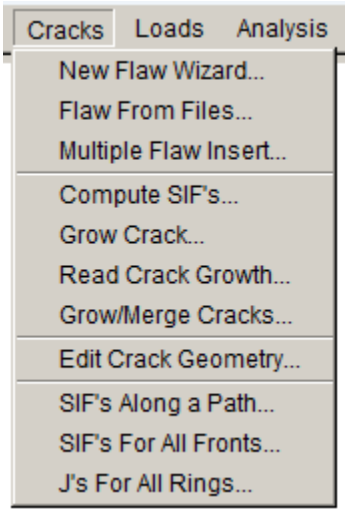


Figure 3.4.1 Cracks Menu.

New Flaw Wizard ... - Invokes the **New Flaw** wizard; see Section 6.1.

Flaw From Files ... - Invokes the **Locate Flaw File** dialog, described in Section 6.2, followed by a subset of the **New Flaw** wizard. This option allows a user to select a flaw description from a file, or from multiple files, and insert it into the current model. Note that one cannot insert a crack into an already-cracked model.

Multiple Flaw Insert ... Invokes the **Multiple Flaw** wizard; see Section 6.3.

Compute SIF's ... - Invokes the **Compute SIF's** wizard; see Section 6.4. Note that displacements from a completed analysis must be available for this option to be enabled.

Grow Crack ... - Invokes the **Crack Growth** wizard; see Section 6.5. Note that displacements from a completed analysis must be available for this option to be enabled.

Read Crack Growth ... - Invokes the **Read Crack Growth** wizard; see Section 6.6.

Grow/Merge Cracks ... - Invokes the **Crack Growth/Merge** wizard; see Section 6.7.

Edit Crack Geometry ... - Invokes the **Edit Crack Geometry** dialog; see Section 6.8.

SIFs Along a Path ... - Invokes the **SIFs Along a Path** dialog; see Section 6.9.

SIFs For All Fronts... - Invokes the **SIFs For All Fronts** dialog; see Section 6.10.

Js For All Rings... - Invokes the **Js For All Rings** dialog; see Section 6.11.

3.5 Loads

The **Loads** menu is shown in Fig 3.5.1. Each of the menu items is listed below.

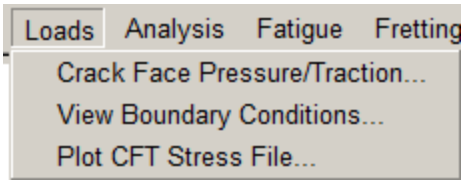


Figure 3.5.1 Loads Menu.

Crack Face Pressure/Traction ... - Invokes the **Crack-Face Tractions** dialog; see Section 7.1.

View Boundary Conditions ... - Invokes the **View Boundary Conditions** dialog; see Section 7.2.

Plot CFT Stress File ... – Displays mesh-based crack face tractions; see Section 7.3.

3.6 Analysis

The **Analysis** menu is shown in Fig 3.6.1. Each of the menu items is listed below.

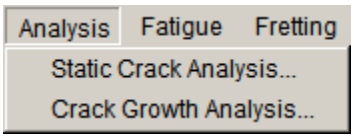


Figure 3.6.1 Analysis Menu.

Static Crack Analysis ... - Invokes the **Static Crack Analysis** wizard; see Section 8.1.

Crack Growth Analysis ... – Invokes the **Crack Growth Analysis** wizard; see Section 8.2.

3.7 Fatigue

The **Fatigue** menu is shown in Fig 3.7.1. Each of the menu items is listed below.

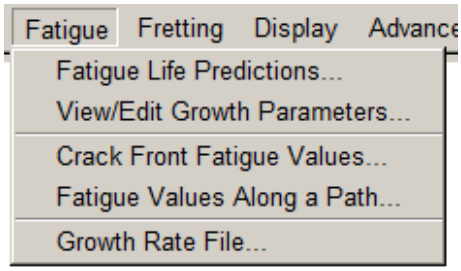


Figure 3.7.1 Fatigue Menu.

Fatigue Life Predictions ... - Invokes the **Fatigue Life** dialog; see Section 9.2.

View/Edit Growth Parameters ... – Allows one to open a growth parameters file and then edit it; see Section 9.3.

Crack Front Fatigue Values ... – Displays a plot of the crack growth data; see Section 9.4.

Fatigue Values Along a Path ... – Displays a plot of the crack growth data along a path; see Section 9.5.

Growth Rate File ... – see Section 9.6.

3.8 Fretting

The **Fretting** menu is shown in Fig 3.8.1. Each of the menu items is listed below. Note that the Fretting menu is not turned on by default; it can be turned on in the Preferences dialog.

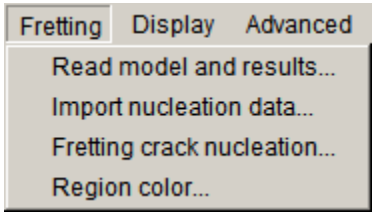


Figure 3.8.1 Fretting Menu.

Read Model and Results ... - Invokes the **Fretting Model Import** wizard; see Section 10.1.

Import Nucleation Data ... - Invokes the **Fretting Data Import** wizard; see Section 10.2.

Fretting Crack Nucleation ... - Invokes the **Fretting Crack Nucleation** wizard; see Section 10.3.

Region Color ... - Invokes the **Region Color** dialog; see Section 10.4.

3.9 Display

The **Display** menu is shown in Fig 3.9.1. Each of the menu items is listed below.

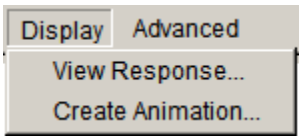


Figure 3.9.1 Display Menu.

View Response ... - Invokes the **View Response** dialog; see Section 11.1.

Create Animation ... - Invokes the **Animation** dialog; see Section 11.2.

3.10 Single Crystal

The **Single Crystal** menu is shown in Fig 3.10.1. Each of the menu items is listed below. Note that the Single Crystal menu is not turned on by default; it can be turned on in the Preferences dialog.

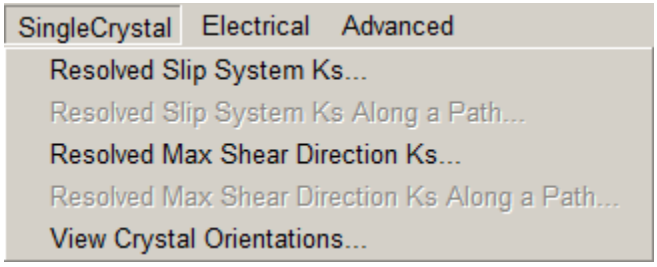


Figure 3.10.1 Single Crystal Menu.

Resolved Slip System Ks... - Invokes the **Resolved SIF's** wizard; see Section 12.1. Note that displacements from a completed analysis must be available for this option to be enabled.

Resolved Slip System Ks Along a Path... - see Section 12.2.

Resolved Max Shear Direction Ks... - see Section 12.3.

Resolved Max Shear Direction Ks Along a Path... - see Section 12.4.

View Crystal Orientations... - Invokes the **Crystal Orientations** dialog; see Section 12.5.

3.11 Electrical

The **Electrical** menu is shown in Fig 3.11.1. Each of the menu items is listed below. Note that the Electrical menu is not turned on by default; it can be turned on in the Preferences dialog.

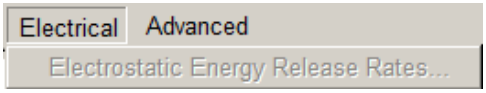


Figure 3.11.1 Electrical Menu.

Electrostatic Energy Release Rates... - see Section 13.1.

3.12 Advanced

The **Advanced** menu is shown in Fig 3.12.1. Each of the menu items is listed below.

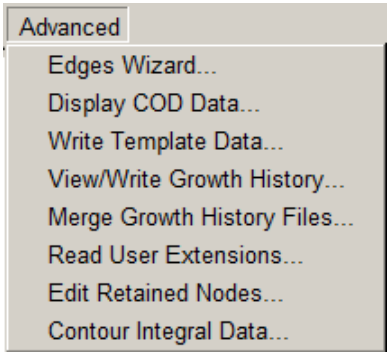


Figure 3.12.1 Advanced Menu.

Edges Wizard ... - Invokes the **Edge Extraction** wizard; see Section 14.1.

Display COD Data ... - Invokes the **Display COD Data** dialog; see Section 14.2.

Write Template Data ... - Invokes the **Write Template Data** dialog; see Section 14.3.

View/Write Growth History ... - Invokes the **View/Write Growth History** dialog; see Section 14.4.

Merge Growth History Files ... - Invokes the **Merge Growth History Files** dialog; see Section 14.5.

Read User Extensions ... - Invokes the **Read User Extensions** wizard; see Section 14.6.

Edit Retained Nodes ... – Invokes the **Edit Retained Nodes** dialog; see Section 14.7.

Contour Integral Data ... – Invokes the **Contour Integral Data** dialog; see Section 14.8.

4. File Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **File** menu options are described in this section.

4.1 Open…Ctrl-O

The **File →Open** menu option allows the user to read a FRANC3D restart file, with *.fdb* extension. The **Restart File** dialog allows one to select a *.fdb* file, Fig 4.1.1.

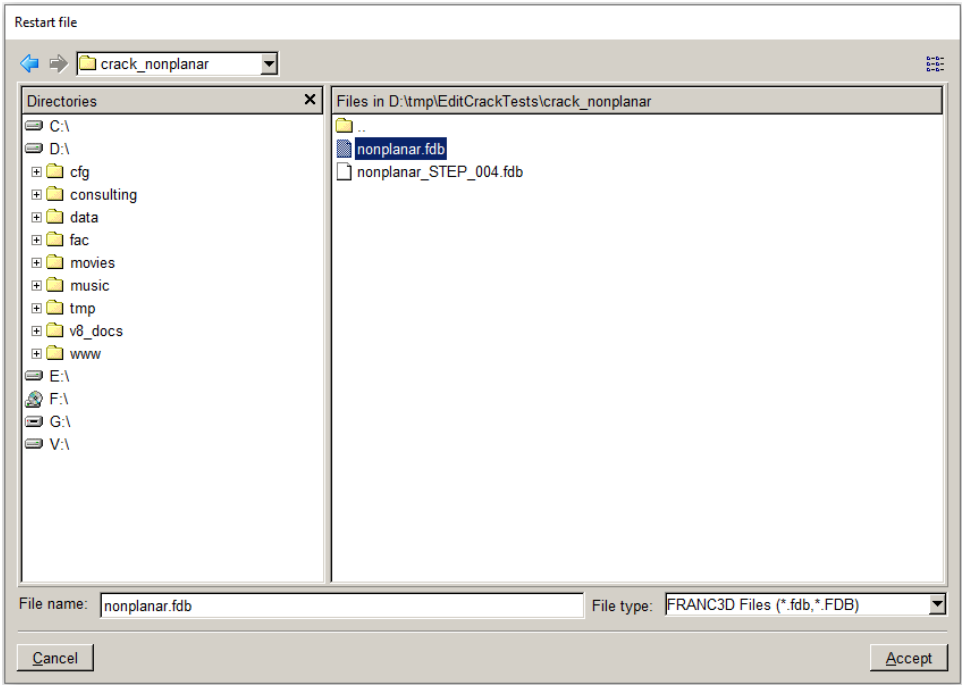


Figure 4.1.1 Model File dialog box.

4.1.1 FRANC3D Restart (.fdb) Files

FRANC3D reads the contents of the *.fdb* file along with reading any files that are referenced inside it. The *.fdb* file is organized in blocks, with each block having a title and version number. The blocks and their content are summarized here:

F3D_V8.7 (
Uncracked FE file name, type, and retained data.
)
HISTORY_SUMMARY (
Summary of the crack step file names.
)
STATICMETA (
Analysis type and input and results file names.
)
FLAWSURF (
Description of crack geometry: surface patches, crack front vertices, and template parameters.
Note that this block can be extracted and saved to a *.crk* file.
)
CRACKFRONTIDS (
List of crack front identifiers.
)
TEMPLATENODES {
List of crack front mesh template node ids.
}
TEMPLATENODECOORDS {
List of crack front mesh template node coordinates.
}
SIF_COMP_PARAMS (
Stress intensity factor computation parameters.
)
GROWTH_PARAMS (
Crack growth model parameters.
)
TEMPLATE_PARAMS (
Crack front mesh template parameters.
)
CRACKFACENODES {
List of crack face nodes with the local normal to the crack surface.
}
EDGE_EXTRACT_PARAMS (
Edge extraction parameters.
)
MESH_PARAMS (
Meshing parameters.
)
CRACK_GROWTH_DATA (
Crack growth history data.
)

UNITS (
Model units for length, stress, and temperature.
)
SUBMODEL_NODE_MAP (
Node id mapping between *.fdb* and FE model input.
)
SUBMODEL_ELEM_MAP (
Element id mapping between *.fdb* and FE model input.
)
RETAINED_NODES (
List of retained node IDs.
)
SAVED_VIEWS {
List of saved camera positions.
}

FRANC3D attempts to read all the files referenced in the *.fdb* file. If a file is missing, the user is prompted to locate that file, Fig 4.1.2. Note that FRANC3D needs to access the original uncracked FE model files as well as files associated with the current crack growth step.

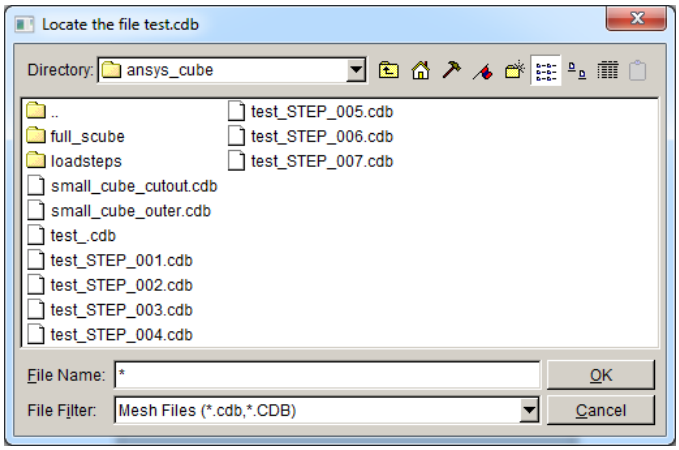


Figure 4.1.2 Locate File dialog box for finding missing files.

4.2 Work Directory

The **File →Work Directory** menu item allows the user to set a current working directory to save subsequent analysis files. The dialog, Fig 4.2.1, allows one to select the desired folder; press **Accept** once the folder is highlighted. This should be done first to ensure the session log and other files are saved to this folder.

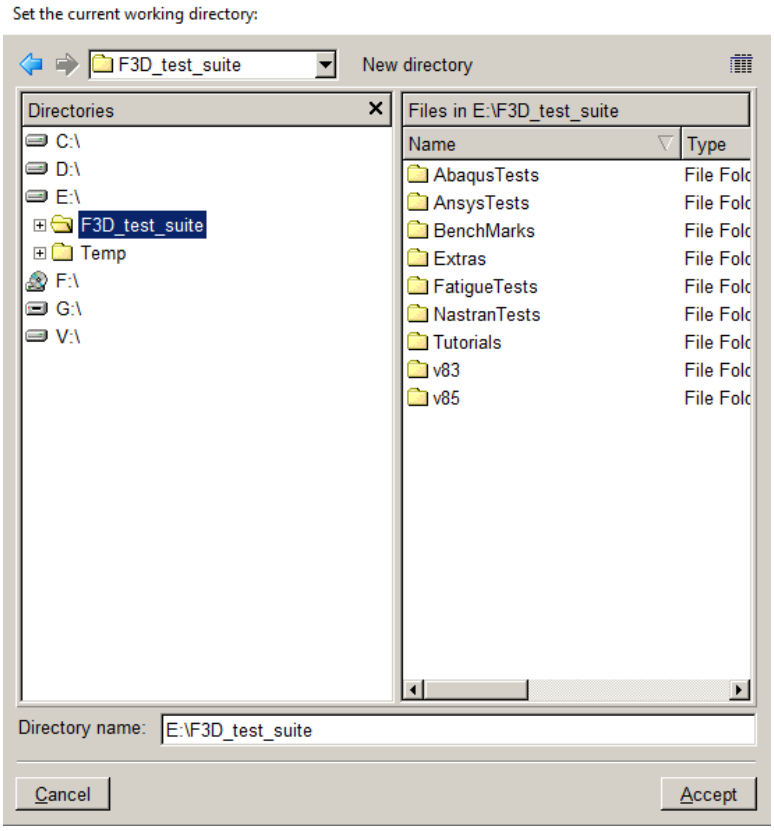


Figure 4.2.1 Set Working Directory dialog box for setting the current directory.

4.3 Close

The **File →Close** menu item closes the current model without quitting FRANC3D. If a model is open and has not been saved, the **Save Model Warning** dialog, Fig 4.3.1, is displayed. If the user wishes to save the model, the **Cancel** button can be selected. Selecting the **OK** button will close the model without saving.

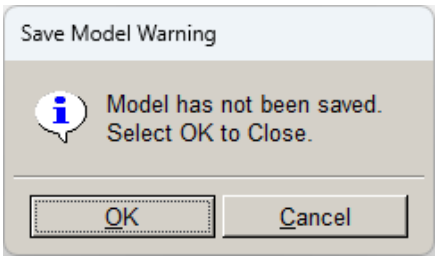


Figure 4.3.1 Save Model Warning dialog.

4.4 Save As…

The **File →Save As** menu item can be used to save the FRANC3D restart (*fdb*) file of a cracked model if the user does not want to perform an immediate analysis of the model. In addition to the *.fdb* file, an FE analysis file will be saved; the FE file type will be the same as the original input file type. The user is prompted to enter the *.fdb* file name, Fig 4.4.1, and press **Accept**.

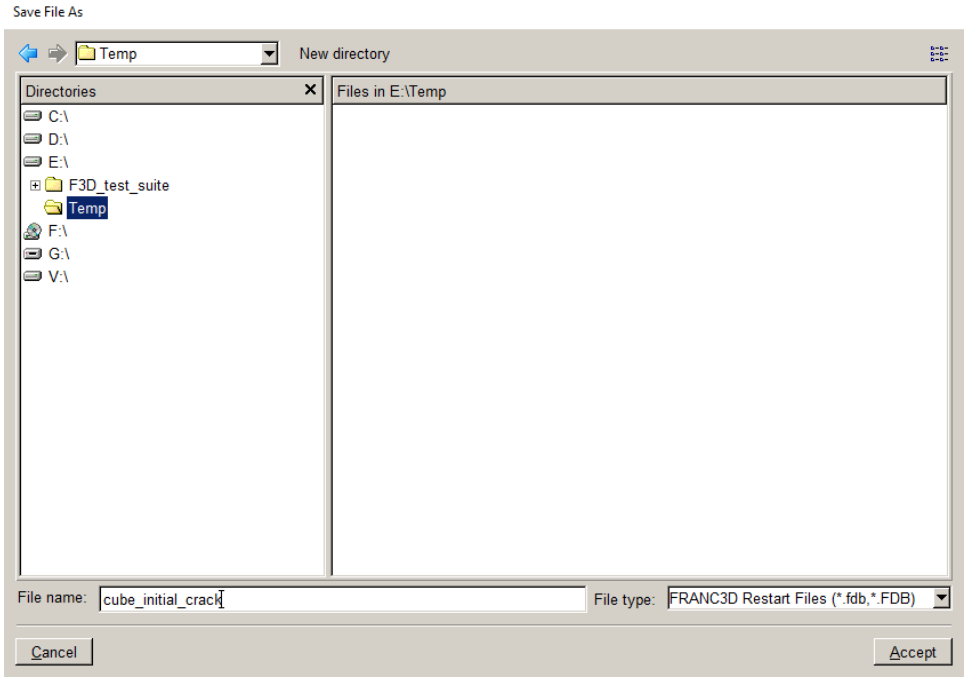


Figure 4.4.1 Save File As dialog.

Note that the files saved with the **Save As** option can differ from the files saved with the **Static Crack Analysis** and **Crack Growth Analysis** options (see Section 8). The **Save As** option does not write the merged local-cracked + global FE file.

4.5 Import…

The **File →Import** menu item is used to import uncracked FE models into FRANC3D. The dialog shown in Fig 4.5.1 is presented. The user can choose: 1) import a complete model into FRANC3D, 2) import and divide so that FRANC3D works on a local submodel, or 3) import an already divided model. Depending on the radio button that is chosen, different dialogs will be displayed next.

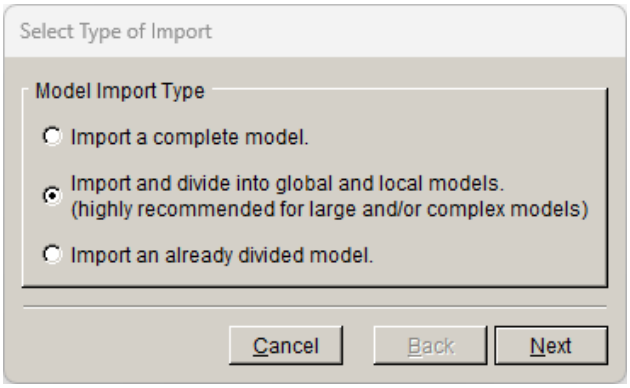


Figure 4.5.1 Select Model Import Type dialog.

4.5.1 Import a Complete Model

The dialog shown in Fig 4.5.2 is displayed. The user chooses the FE model type: ANSYS, ABAQUS or NASTRAN using the radio button at the top. If ANSYS is chosen, the user can also check the Extra load files box, if ANSYS load step files (*.s##*) are to be imported as well.

If the ANSYS Extra load files box is checked, the dialog shown in Fig 4.5.3 is displayed. The user can select one or more *.s##* files. Use the Shift or Ctrl key to select multiple file names.

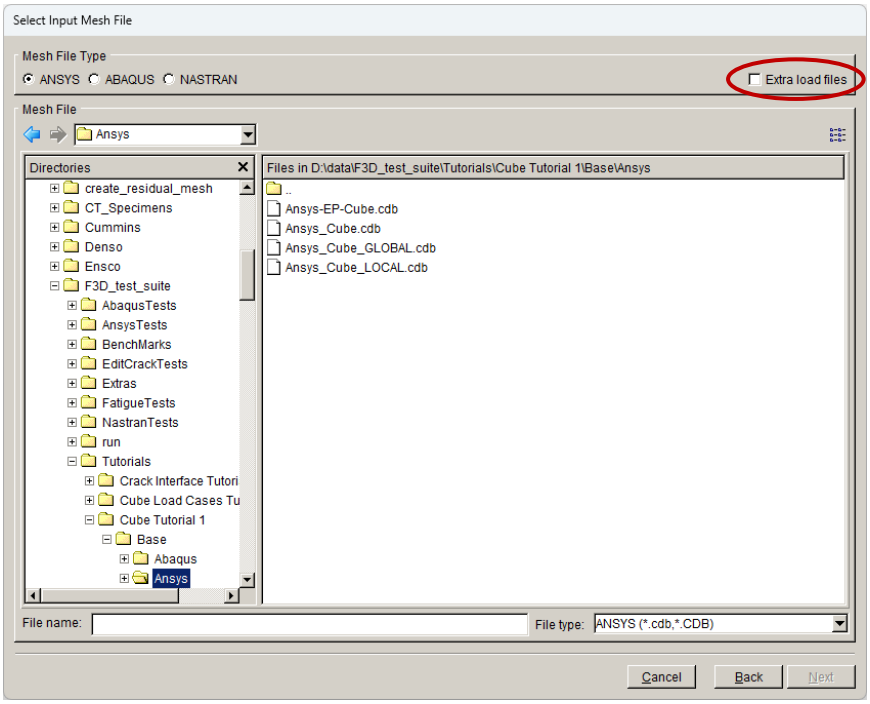


Figure 4.5.2 Select Model to Import dialog.

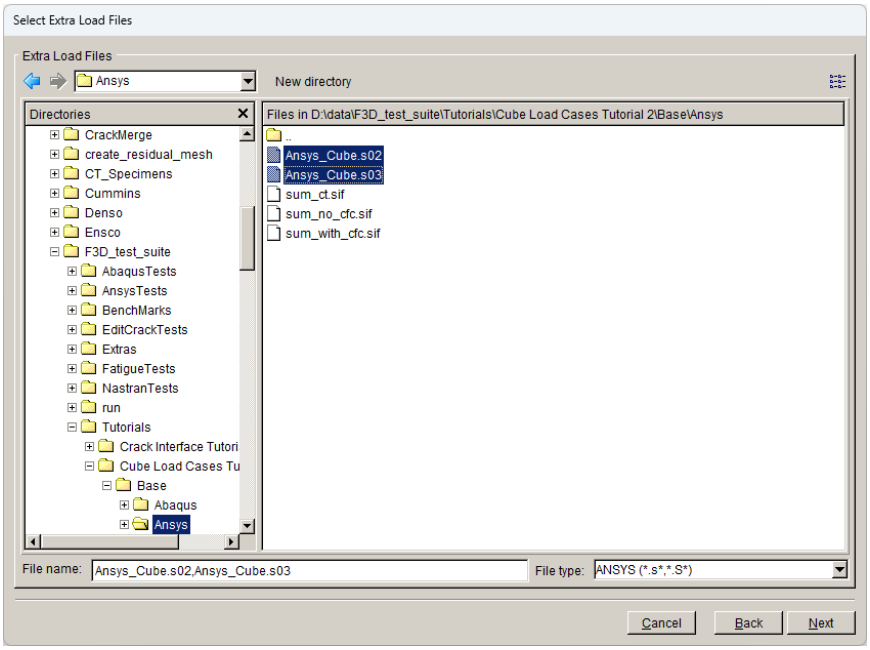


Figure 4.5.3 Select ANSYS Extra load step files dialog.

Once the FE model file has been selected, the **Select Retained BC Surfaces** dialog is displayed, Fig 4.5.4. Any surface that has boundary conditions will be highlighted in blue. These surfaces can be selected using Select all, and the blue surfaces will become red. Other node sets, element surface sets and surfaces can also be selected.

Note that cracks cannot be inserted into or propagated into a surface mesh that has been retained. If a surface has boundary conditions and a crack must be inserted into this surface, do not retain the mesh facets on this surface; the boundary conditions will be transferred to the remeshed surface. For surfaces without cracks, it is more efficient and more exact to retain the mesh so that boundary conditions are transferred directly.

The lower left section of the dialog has some additional display options. The Transparent check box makes the surfaces slightly transparent so that you can see into and through the model. The cut-surfaces are colored green by default; this can be turned off using the Global/Local cut surfaces checkbox. The FE mesh can be displayed using the Mesh overlay checkbox.

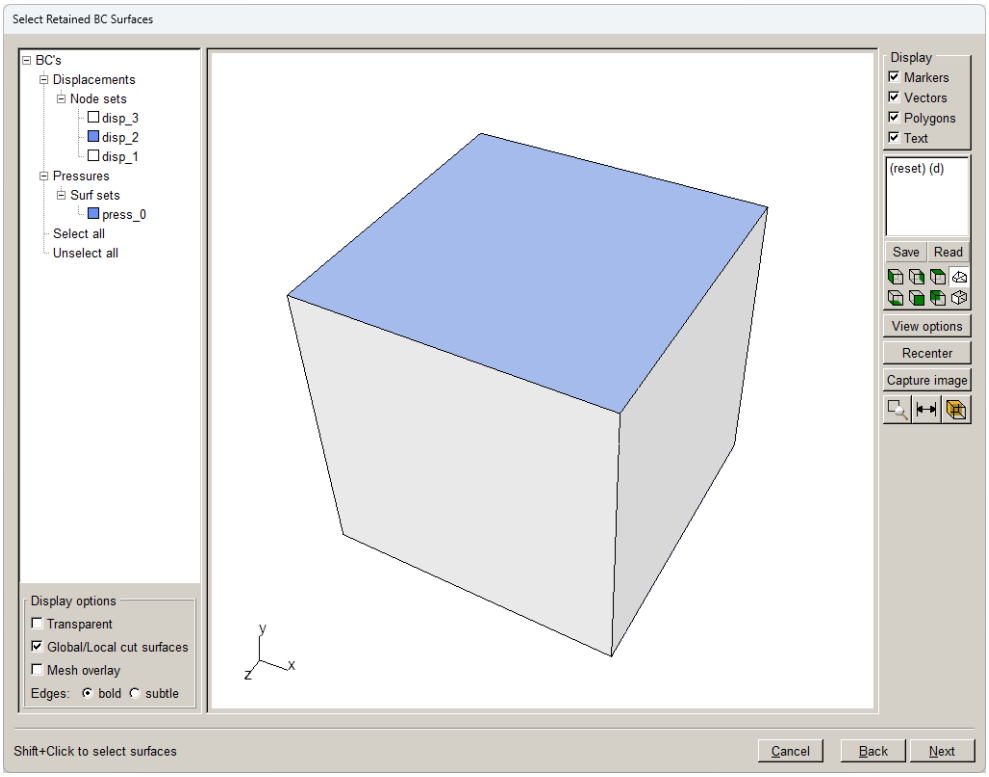


Figure 4.5.4 Select Retained BC Surfaces dialog.

4.5.2 Import and Divide Model

If the user chooses to import and divide, the first dialog that is presented is the same as that shown in Fig 4.5.2. Once the model file has been selected, the **Submodel** dialog is displayed, Fig 4.5.5. There are options for selecting a portion or portions of the model, and these are described in the subsections below. Once a submodel is defined and cropped, press **Next**, which leads to the **Select Retained BC Surfaces** dialog (see Fig 4.5.5), if the submodel has boundary conditions or sets/surfaces.

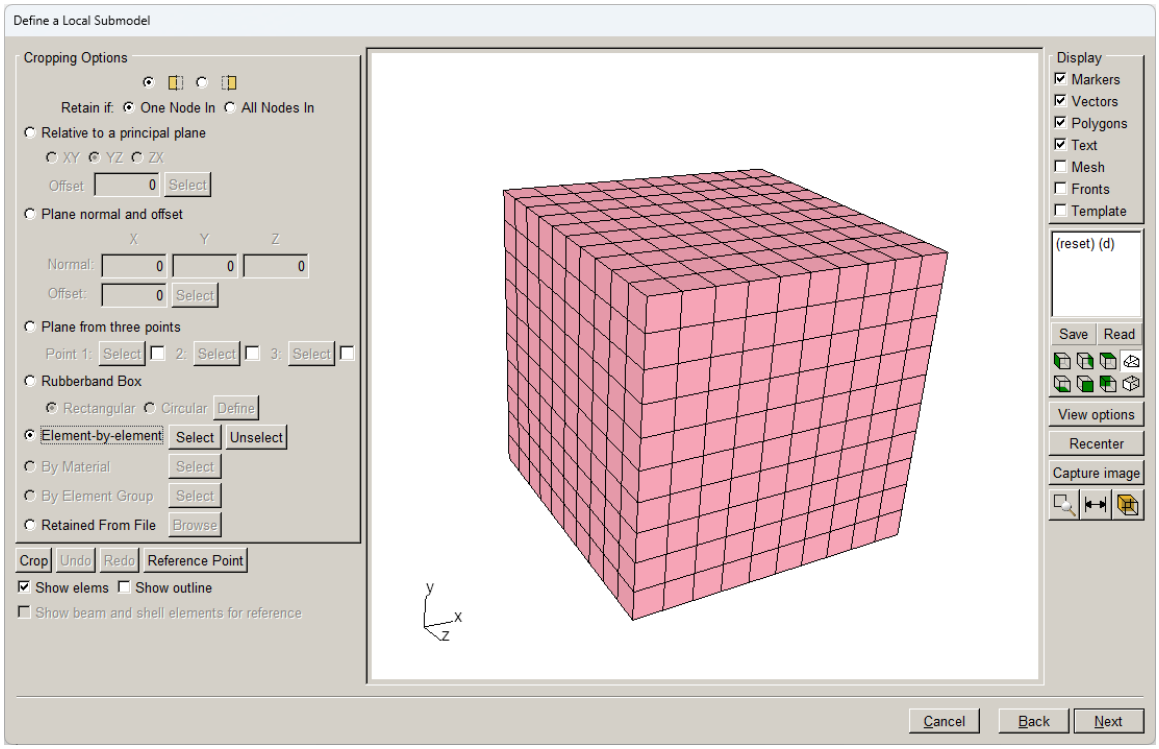


Figure 4.5.5 Define a Local Submodel dialog.

4.5.2.1 Cropping Selection Options

The first setting in the Cropping Options allows one to specify which 'side' of the selection plane or box is selected. The top row of radio buttons in Fig 4.5.6 lets one choose the 'left' or 'right' side (Fig 4.5.7); this is straightforward to describe when using the planar options below, but it also applies to the other options. For instance, when using the Rubberband Box, the selection will be either inside or outside of the rubberband box depending on which radio button is selected.

The second row of radio buttons allows one to select elements with either one or all nodes of the element collected. Fig 4.5.8 shows a case where the cutting plane is half-way through the elements; choosing the first option includes these cut-elements, while choosing the second option discards the elements that are cut.

Note that selected elements are colored red.

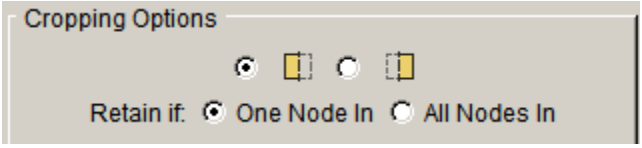


Figure 4.5.6 Cropping Options selection options dialog.

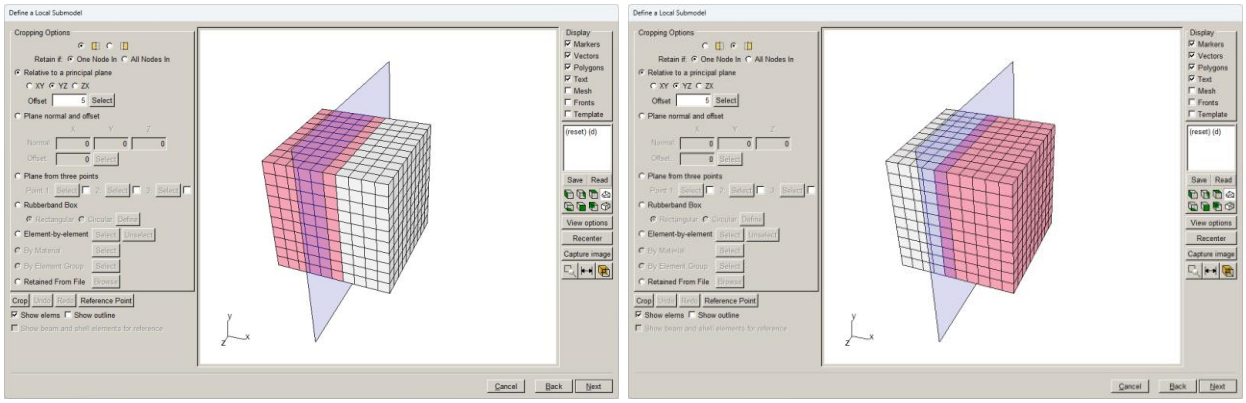


Figure 4.5.7 Selection based on ‘left’ and ‘right’ selection.

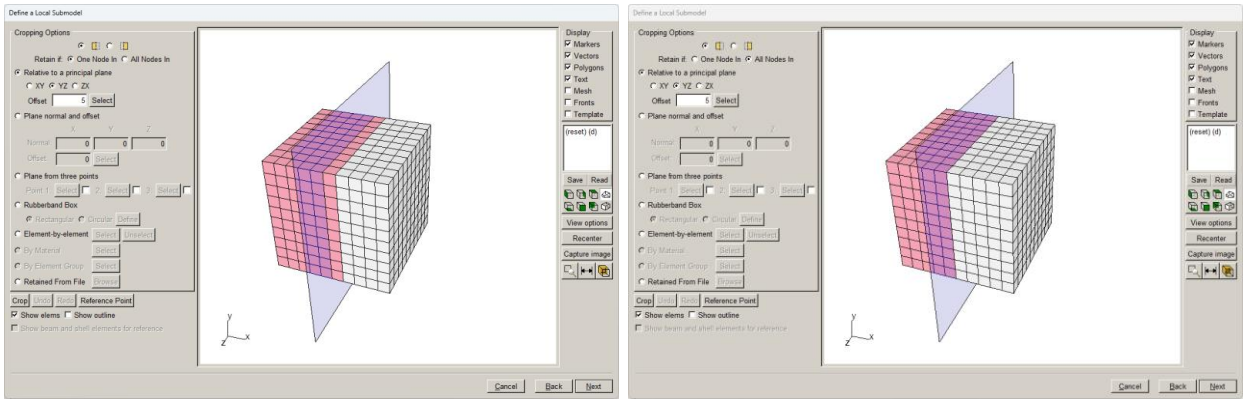


Figure 4.5.8 Selection based on Retain if: ‘one node in’ and ‘all nodes in’ selection.

4.5.2.2 Relative to a principal plane

The first crop-type allows one to define a plane that is aligned with the Cartesian axes, Fig 4.5.9. The plane can be offset from the origin, and the elements on either the ‘left’ or ‘right’ side of the plane are selected. In this case and in all cases below, the radio buttons at the top (see Section 4.5.2.1) are set to the defaults, which means left-side selection and one-node-in, Fig 4.5.10.

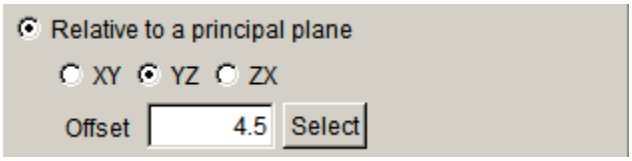


Figure 4.5.9 Relative to principal plane cropping option.

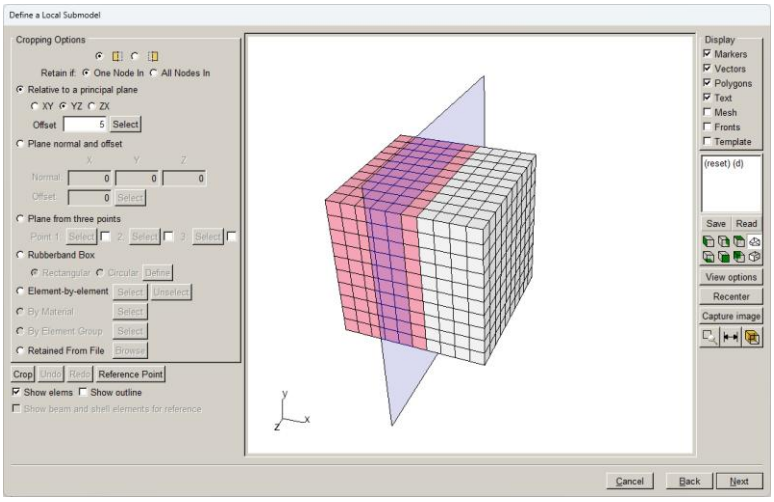


Figure 4.5.10 Selection based on offset YZ principal plane and left-side selection.

4.5.2.3 Plane normal and offset

The second crop-type allows one to define a plane that is normal to the unit vector defined by the x, y and z coordiantes, Fig 4.5.11. The plane can be offset from the origin. Fig 4.5.12 shows a plane that is normal to the vector (1,0,0), which is the same as the YZ principal plane in Section 4.5.2.2.

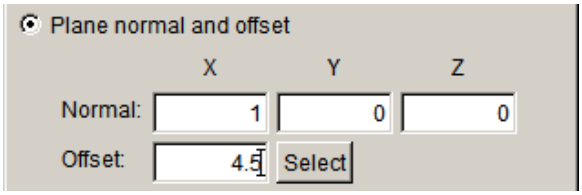


Figure 4.5.11 Plane normal and offset cropping option.

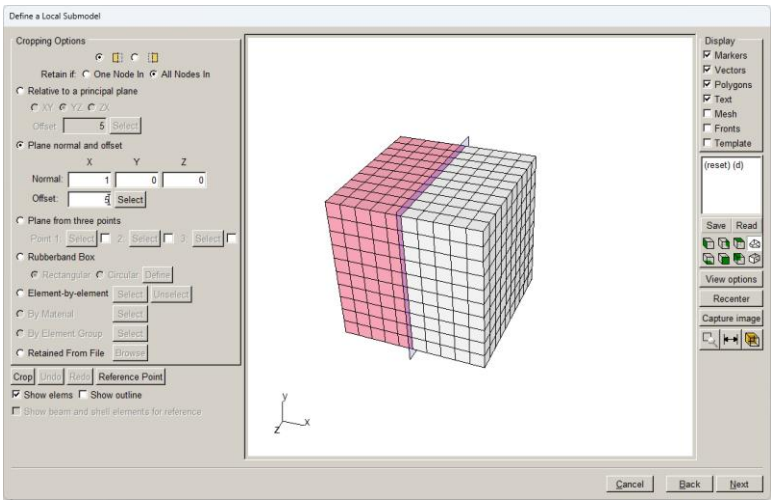


Figure 4.5.12 Selection based on a plane normal and offset.

4.5.2.4 Plane from three points

The third crop-type allows one to define a plane based on the selection of three points, Fig 4.5.13. The user presses the **Select** button for each point, and then uses the (cross) cursor to pick the point; three points of a triangle define the plane. Fig 4.5.14 shows the second (blue) point that was selected here. Fig 4.5.15 shows the resulting plane that was created by picking 3 points at x=5 coordinates; this is the same as the YZ principal plane in Section 4.5.2.2.

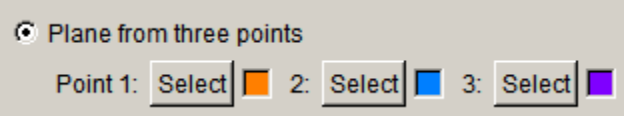


Figure 4.5.13 Plane from three points cropping option.

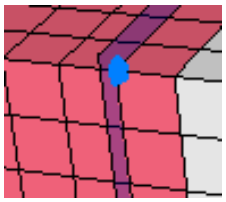


Figure 4.5.14 Second (blue) selected point used to define a plane.

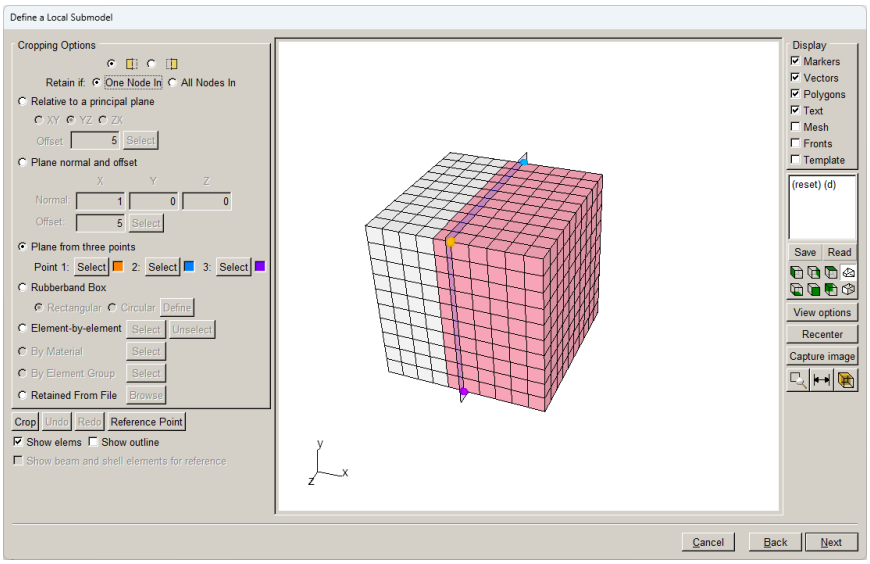


Figure 4.5.15 Selection based on a plane from three points.

4.5.2.5 Rubberband box

The fourth crop-type allows one to define a box or cylinder, Fig 4.5.16. The user selects the Rectangular or Circular radio button, presses the **Define** button, and then uses the left-mouse

button to drag the outline of a box or cylinder on the screen, Fig 4.5.17; the right-side image in Fig 4.5.17 is rotated to show the box.

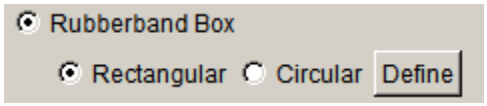


Figure 4.5.16 Rubberband box cropping option.

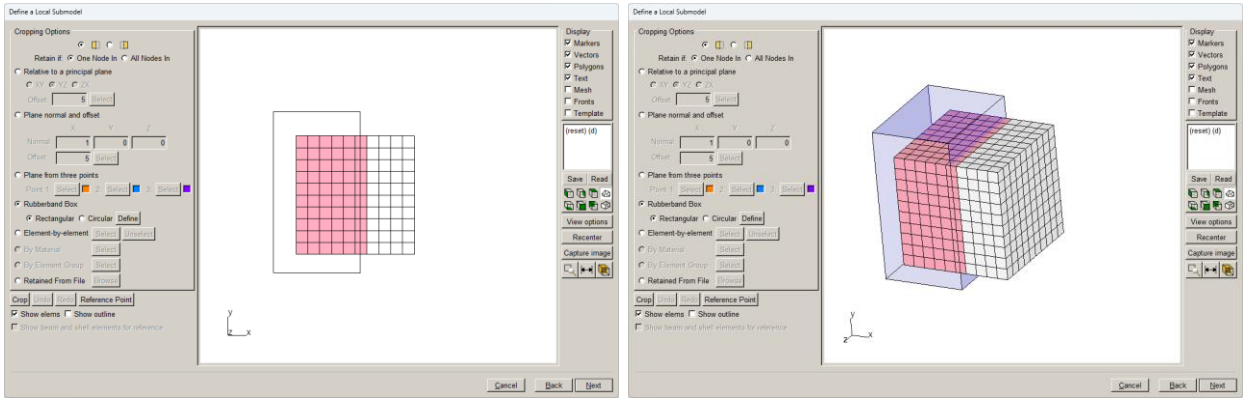


Figure 4.5.17 Selection based on a rectangular rubberband box.

4.5.2.6 Element by element

The fifth crop-type allows one to select (or unselect) individual elements, Fig 4.5.18. The user presses the **Select** button and then uses the left-mouse button to select elements one at a time, Fig 4.5.19. The default is to have all elements pre-selected and the **Select** button actually turns off the selection, while the **Unselect** button re-selects the element.

Figure 4.5.18 Element-by-element cropping option.

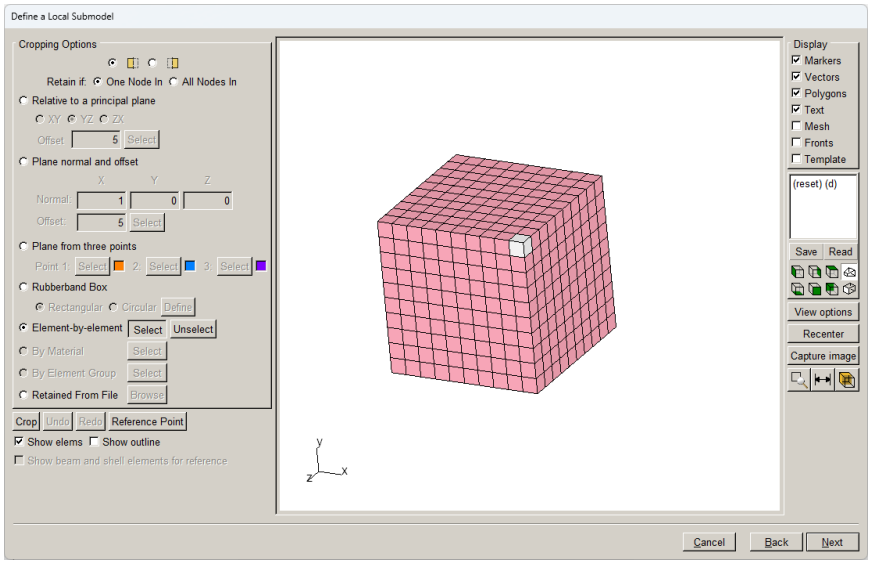


Figure 4.5.19 Selection based on element-by-element picking.

4.5.2.7 By material

The sixth crop-type allows one to select elements based on the material id, Fig 4.5.20. The user selects the material id # after pressing the **Select** button. The model in Fig 4.5.21 has two materials and material 1 is selected.

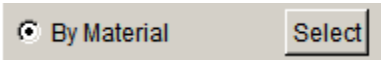


Figure 4.5.20 By material id cropping option.

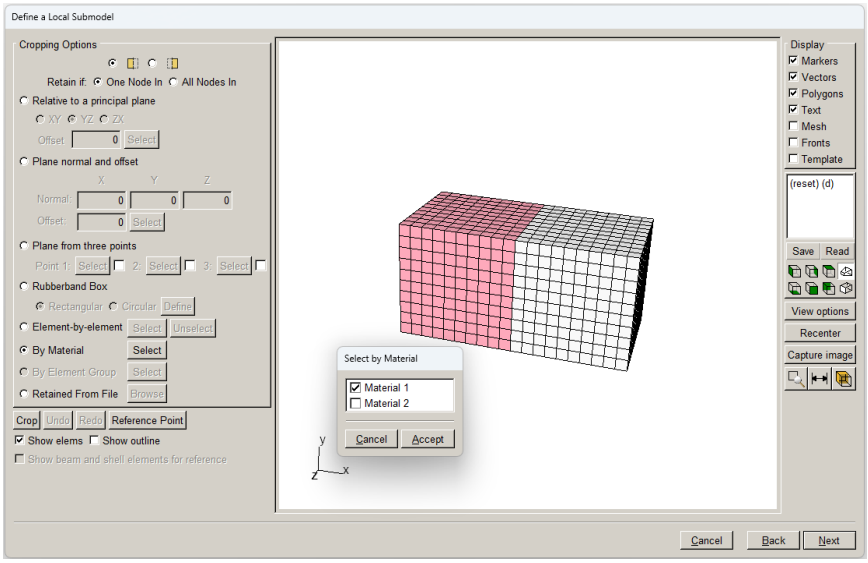


Figure 4.5.21 Selection based on material id.

4.5.2.8 By element group

The seventh crop-type allows one to select elements based on an element group name, Fig 4.5.22. The user selects the label after pressing the **Select** button. The model in Fig 4.5.23 has several element groups, one of which has been selected.

For ANSYS, element groups are defined by 'CMBL' data in the *.cdb* file, and for ABAQUS, the 'ELSET' data in the *.inp* file defines the groups.

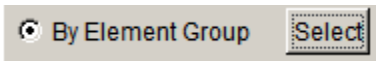


Figure 4.5.22 By element group label cropping option.

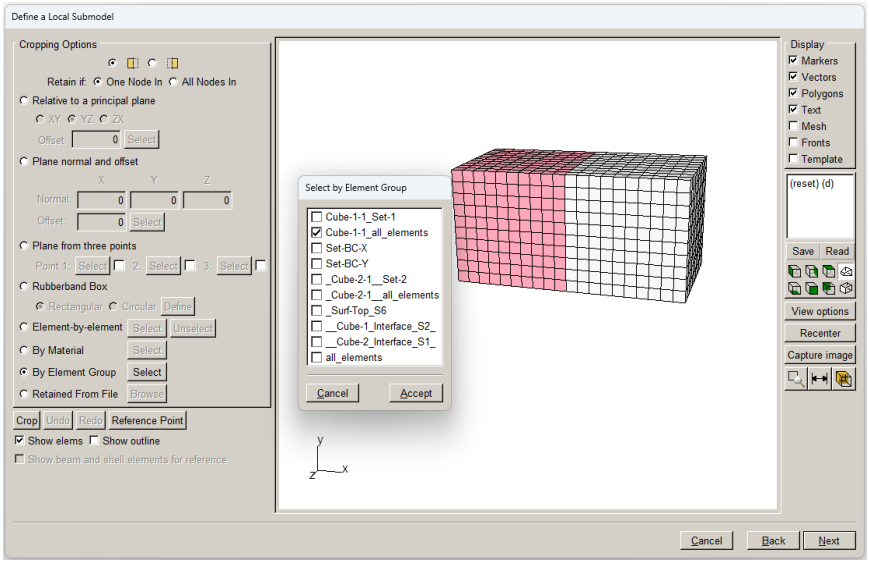


Figure 4.5.23 Selection based on element group label.

4.5.2.9 Retained from file

The final crop-type allows one to select element id's from a file, Fig 4.5.24. The user presses the **Browse** button to display the file selection dialog, Fig 4.5.25. This option is useful for re-selecting elements that were selected during a previous import. The *.txt* file is ASCII and contains a list of element ids; a range of ids is specified using a dash. Fig 4.5.26 shows the selected elements that were identified in the *.txt* file.

Figure 4.5.24 Retained from file cropping option.

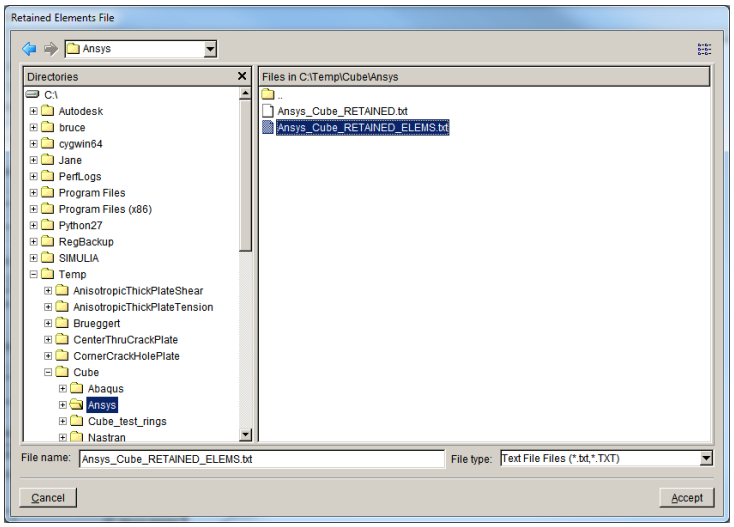


Figure 4.5.25 Retained element id file selection dialog.

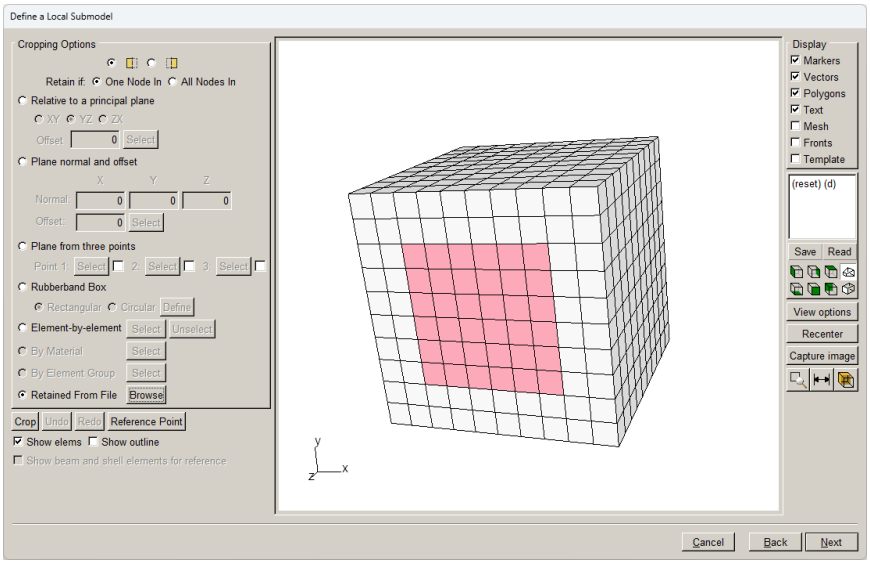


Figure 4.5.26 Selection based on element ids provided in an ASCII .txt file.

4.5.2.10 Crop, Undo and Redo

The three buttons near the bottom on the left side perform the actual cropping. Once the selection has been made using one of the options above, the selected elements are cropped by pressing the **Crop** button, Fig 4.5.27. The un-selected elements are removed. The user can **Undo** or **Redo** the cropping. Multiple selections and crops can be performed in sequence.

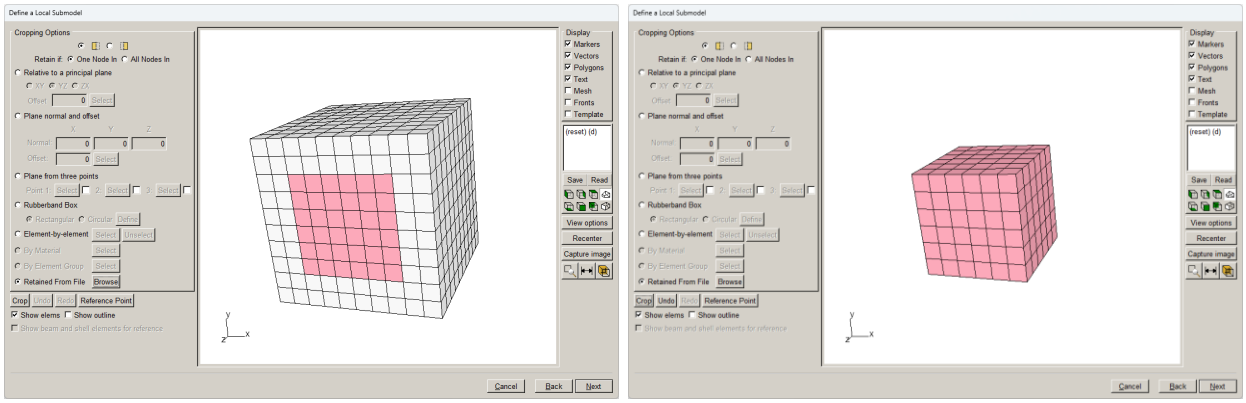


Figure 4.5.27 Crop button pressed to remove the un-selected elements.

4.5.2.11 Reference Point

The **Reference Point** button, next to the **Crop/Undo/Redo** buttons, allows you to specify a reference point, using either a node id or Cartesian coordinates, for the center of rotation, Fig 4.5.28. The reference point is displayed using a red dot. Double-click the (reset)(d) camera position to reset to the default.

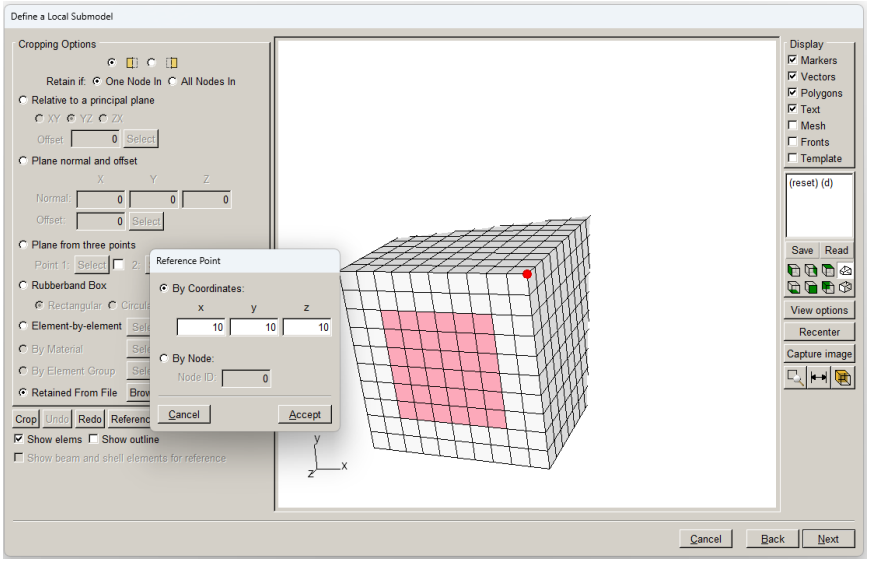


Figure 4.5.28 Reference Point dialog to set the rotation center.

4.5.2.12 Show elems and Show outline

The Show elems option at the bottom turns on or off the display of element edges, Fig 4.5.29. For some models, with many elements or with very refined meshes, the element edges can obscure the element selection. Un-checking this box turns off the black element edge coloring, which should make the red-colored elements more visible. The Show outline option is useful when selecting interior elements as it shows the exterior outline of the model.

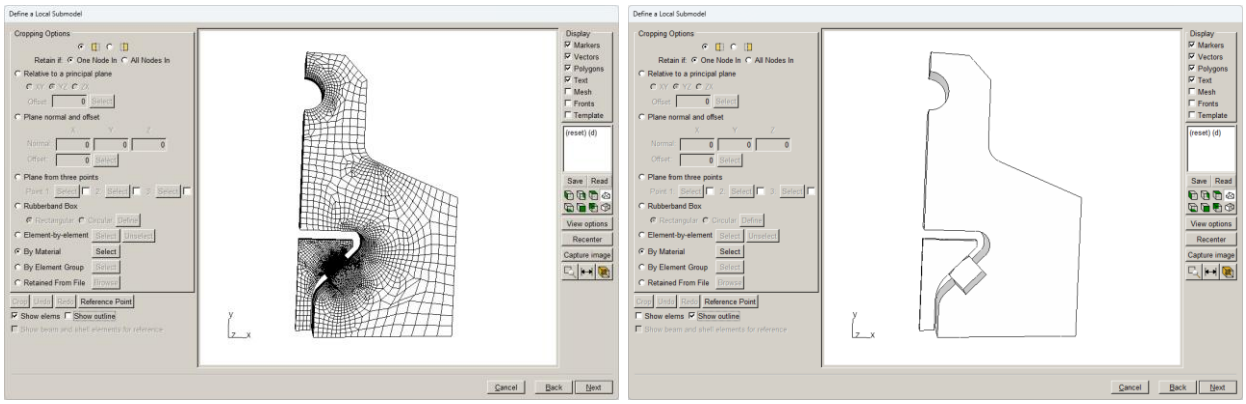


Figure 4.5.29 Show elems option.

4.5.2.13 Show beam and shell elements for reference

The Show beam and shell elements option at the bottom turns on or off the display of shell or beam elements, Fig 4.5.30. Shell and beam elements cannot be selected; they automatically are added to the global portion of the model.

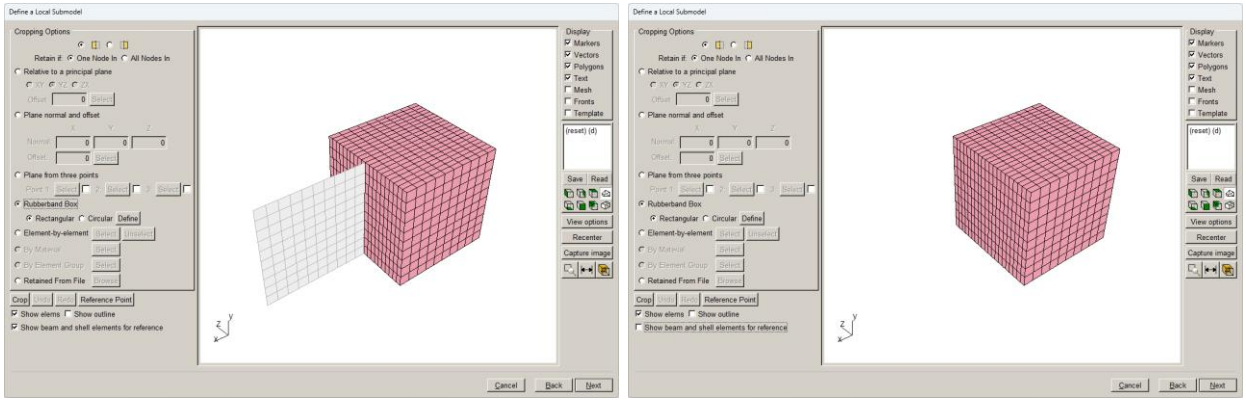


Figure 4.5.30 Show beam and shell elements option.

4.5.3 Import Already Divided Model

If the user has model files that have already been subdivided, the dialog shown in Fig 4.5.31 is displayed, which allows the user to select the global and local model files. These can be files created by FRANC3D (as in Section 4.5.2) or files created by the user using the FE analysis software. There is no option to select "extra load files"; all load step information must already be included in the local and global FE files.

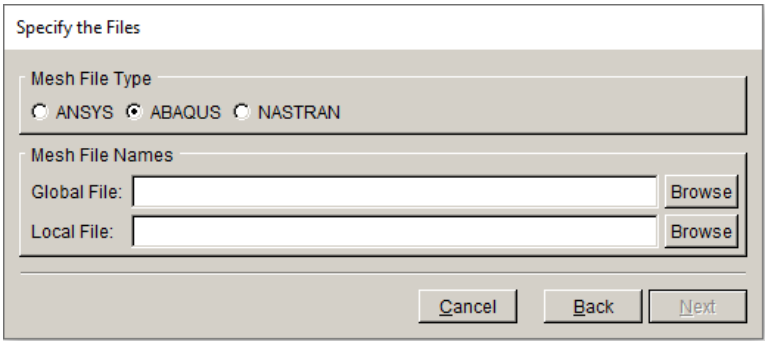


Figure 4.5.31 Select Global and Local files dialog.

4.6 Export…

The **File →Export** menu item can be used to save the FE model data without saving the FRANC3D *.fdb* file. The dialog shown in Fig 4.6.1 is presented; the user selects the analysis FE model type and enters the file name. This option can be used to convert FE model files from one type to another. FRANC3D passes as much of the original FE data through as possible, thus it might only be the solid elements and nodes that are converted correctly.

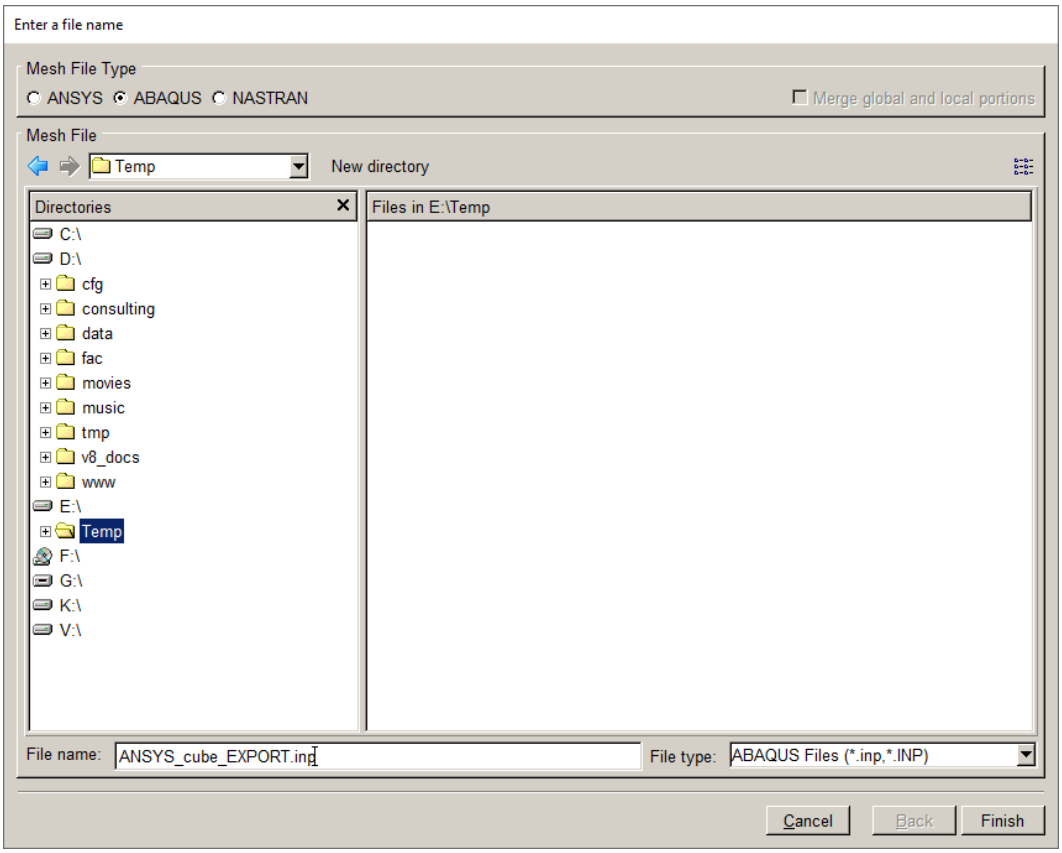


Figure 4.6.1 Export Model File dialog.

4.7 Read Results

The **File →Read Results** menu item invokes the dialog shown in Fig 4.7.1, which allows the user to choose the analysis results file to be read.

For ANSYS, the results file is a *.dtp* file, which is created using the FRANC3D generated ANSYS macros. The *.dtp* file will contain displacements, temperatures if those are not equal to the reference temperature and contact pressures on crack surfaces if such exist.

For ABAQUS, the results file will be a *.dtp* file, which is created using the FRANC3D generated ABAQUS Python script. The *.dtp* file will contain displacements, temperatures if those are not equal to the reference temperature and contact pressures on crack surfaces if such exist.

For NASTRAN, displacement results will be contained in a NASTRAN generated *.pch* file. Temperatures and contact pressures are not included in the *.pch* file. Temperatures are based on the applied temperatures. Contact pressures are not available, so NASTRAN users should not rely on M-integral or VCCT SIFs if crack face contact is included in the analysis.

The Do Map Results option is on by default. A user might turn it off if they manually combined the ANSYS local and global *.cdb* files in ANSYS WorkBench to run the analysis; this is the only known use-case for turning off the mapping. FRANC3D stores the node and element ID mapping for the local cracked mesh in the *.fdb* file, and the FE analysis results are mapped back into FRANC3D based on this mapping.

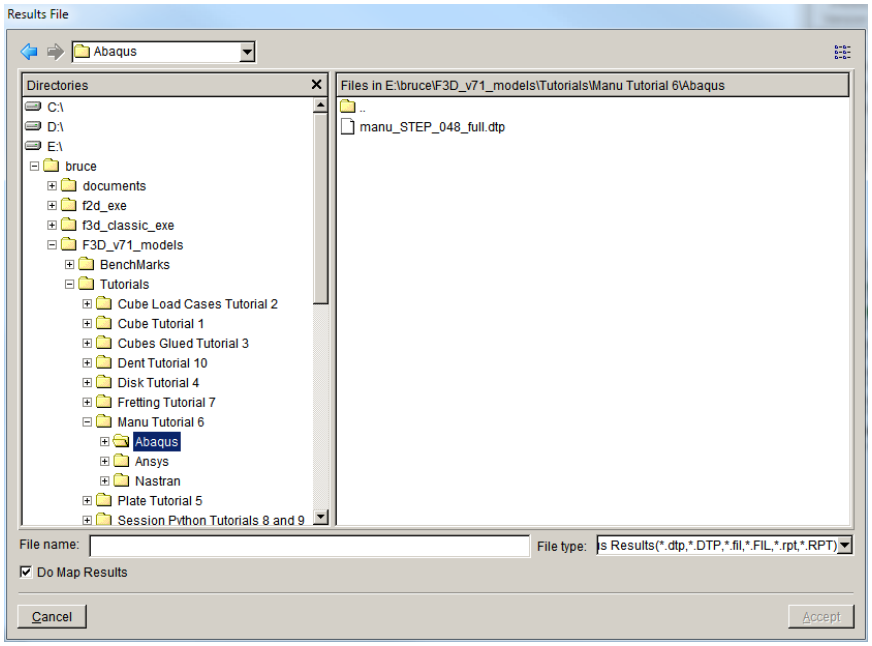


Figure 4.7.1 Read results file dialog.

4.8 Playback….

The **File →Playback** menu item allows playback of recorded session (log) files. Each time the FRANC3D program is executed, a *session##.log* file is saved. This file records the commands that the user executes using the menus and dialogs. A sample *.log* file might look like this:

```
OpenMeshModel (
   model_type=ANSYS,
   file_name='C:\cube\cube.cdb',
   retained_nodes_file=`cube_RETAINED.txt')

InsertFileFlaw (
   file_name='C:\cube\cube_05.crk')

RunAnsysAnalysis (
   file_name='C:\cube\cracked_cube',
   flags=[],
   command='"C:\ANSYS Inc\v252\ansys\bin\win64\ANSYS252.exe" -b -p
     ansys  -i "C:\cube\cracked_cube.cdb" -o "C:\cube\cracked_cube.out"')

ComputeSif ()

CloseModel()
```

A complete description of the FRANC3D command language is provided in the FRANC3D Command Language & Python Extensions document.

Selecting the **Playback** menu item invokes the **Playback Session File** dialog, Fig 4.8.1. The user selects the desired session file and then presses **Accept**. FRANC3D will read and execute the commands from the session file.

The Fatigue Life Predictions menu item (and dialog) does not write a session log command.

4.9 Quit - Ctrl-Q

The File **Quit** menu item exits FRANC3D. The **Save Model Warning** dialog (see Fig 4.3.1) will be displayed if the model has not been saved.

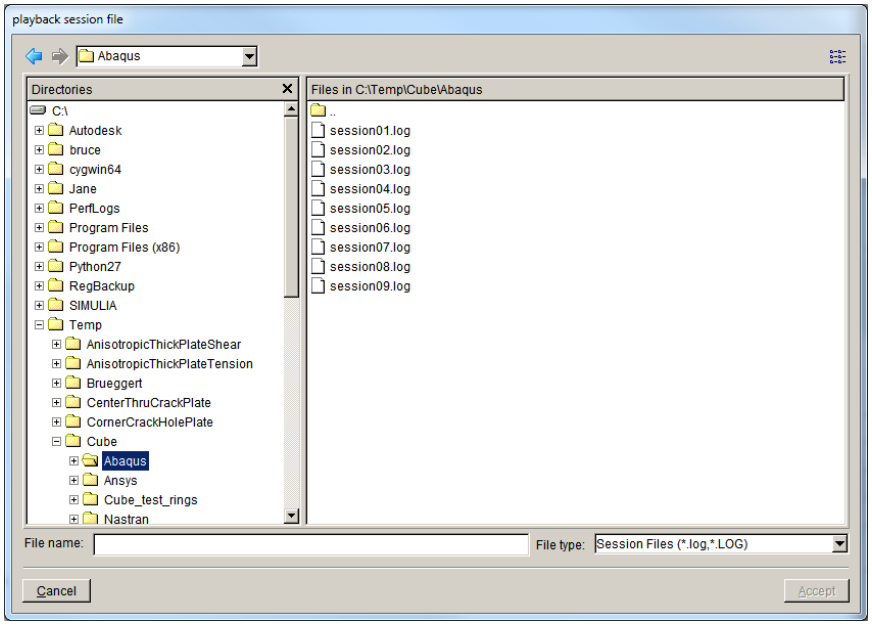


Figure 4.8.1 Playback session file selection dialog.

5. Edit Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Edit** menu options are described in this section.

5.1 Unit Conversion…

Selecting the **Edit →Unit Conversion** menu item invokes the **Model Units** dialog, which allows one to set the FE model units for length, stress, temperature and time, Fig 5.1.1. The units are used to label plots, and they are important when defining the crack growth model where additional material data is entered that is not typically part of the FE model. The original FE units can be set whenever you import a new model, but the display units will be based on your choice here.

FRANC3D will do unit conversions when matching the crack growth material data with the FE model results to compute crack growth or fatigue cycles.

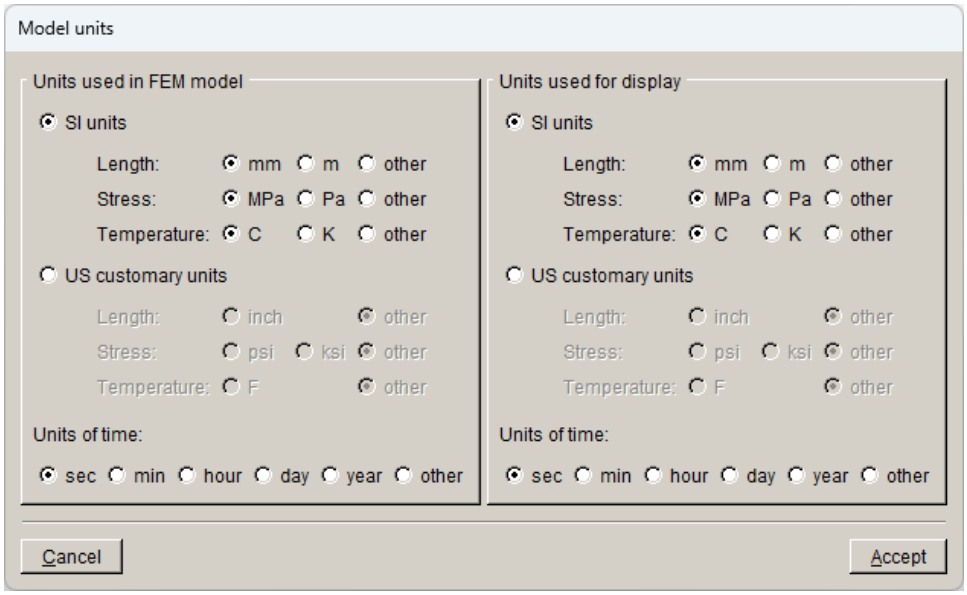


Figure 5.1.1 Model Units dialog.

If you consistently use the same set of units, those can be set in the Preferences, which is described in the next section.

5.2 Preferences…

Selecting the **Edit →Preferences** menu item invokes the **Preferences** dialog, which allows one to set program-wide configurations, Fig 5.2.1. These preferences are stored in a database that is read when FRANC3D is started. Some settings might not take effect until the program is restarted, so if you change settings, it is best to quit and restart. The **Preferences** dialog has ten tabs, which are described next.

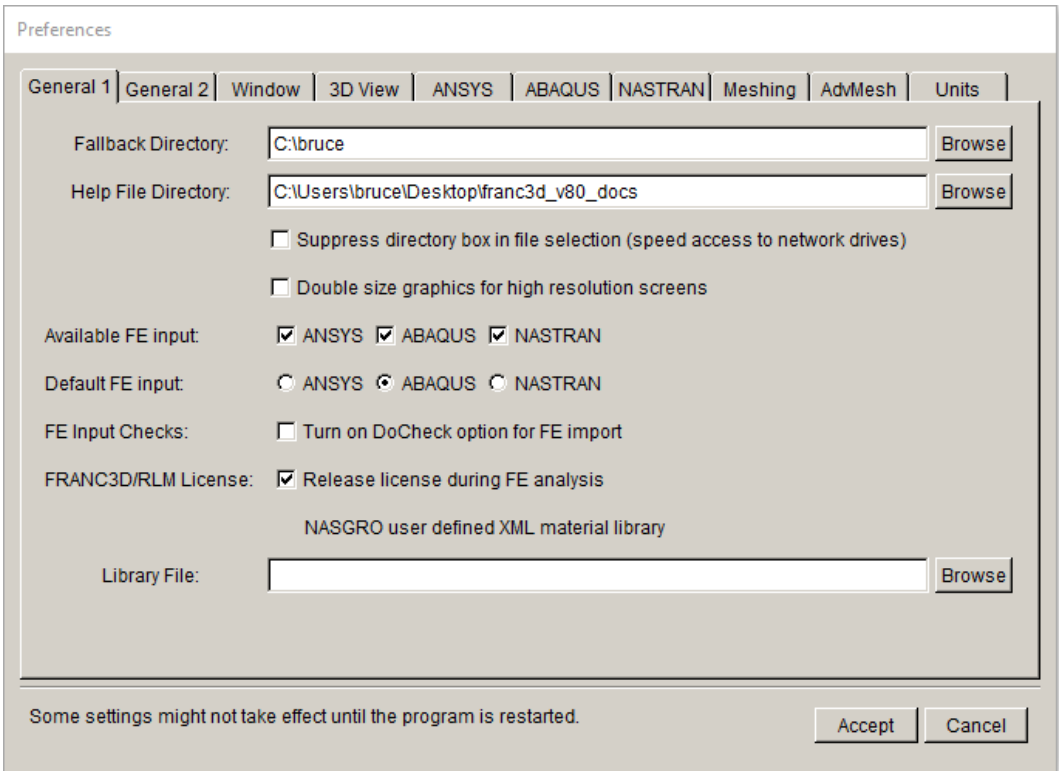


Figure 5.2.1 Preferences dialog with General tab displayed.

5.2.1 General 1 Tab

In the *General 1* tab, one can set the default work and help file directories (see Fig 5.2.1) and other generic settings.

The Fallback directory is the folder where the model files will be read from and written to by default if a work directory is not set or if it is invalid. The work directory should be set using **File →Work Directory**.

The Help File directory is the top-level directory for the documentation described in Section 3.1. Documentation is accessed from the **Help** menu.

The Suppress directory… option might be needed by users that must access network-mounted drives/directories. If you find that FRANC3D takes significant amounts of time to list files/folders when opening/saving, try turning this option on.

The Double size graphics… option might be needed by users with a very high-resolution display. Turning this on will double the size of the graphics, so the user might have issues displaying dialogs or plots for standard resolution displays.

The Available FE Input check box allows one to turn off GUI support for a FE program. For example, if you will only be using ABAQUS, turn off ANSYS and NASTRAN to limit the options when importing/saving/analyzing.

The Default FE Input radio button sets the user's default FE model type for importing a new model (see Section 4.5). For example , if you have both ANSYS and ABAQUS but mostly use ABAQUS, you can set the default to ABAQUS to simplify import and analysis.

The FE Input Checks option allows a user to check the FE model for potential errors/issues when importing. Turning this option on will alter the import dialog by adding an extra option, Fig 5.2.2. Press the **Choose Checks** button to turn on/off specific checks, Fig 5.2.3. The options are described in Section 5.4 of the User's Guide. In general, one should not need to run these checks, but if there are errors during crack insertion/meshing, the user can check the input FE model for errors before sending a bug report.

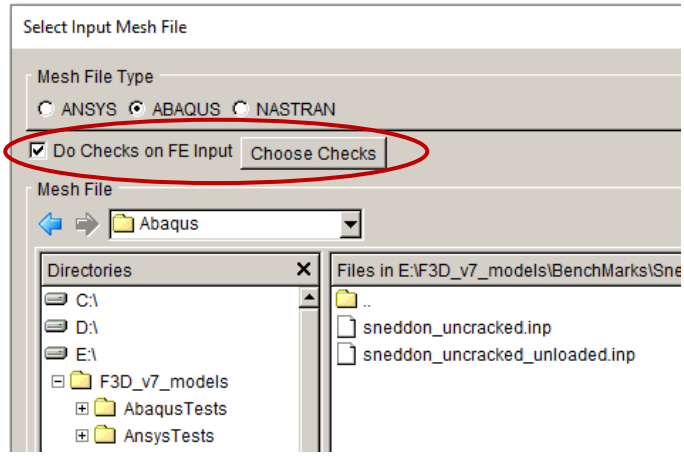


Figure 5.2.2 Import dialog with FE Input Checks turned on.

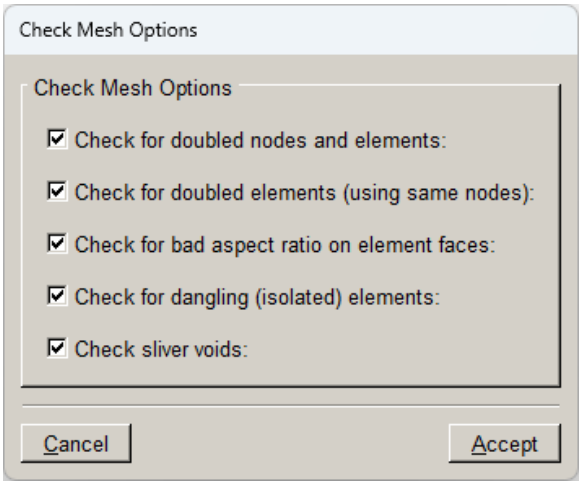


Figure 5.2.3 FE input checks.

The Release license… option allows one to turn off the license-release during the FE analysis. Typically, before the FE analysis starts, FRANC3D will release its license so another instance of the software can be used. If there are issues releasing and/or retrieving the license after the FE analysis has finished, this option can be turned off.

The NASGRO user defined XML material… option allows one to specify an *.xml* file containing NASGRO material data. An *.xml* file can be exported from NASGRO. The file can be used during subcritical crack growth and fatigue life computations.

5.2.2 General 2 Tab

The *General 2* tab, Fig 5.2.4, provides the option to disable the Fretting and Single Crystal menus. Symmetry crack capabilities can be turned on/off. There are additional settings for saving timing output and Python rerun scripts.

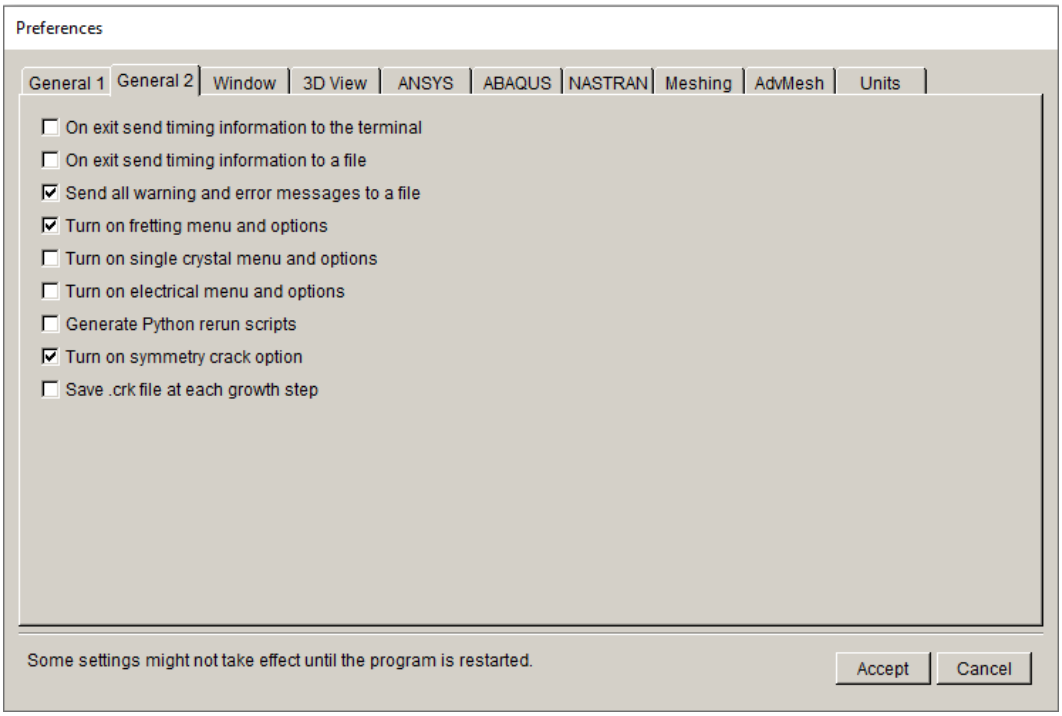


Figure 5.2.4 General 2 preferences tab.

There are two options for On exit send timing…; the first sends information to the terminal (or CMD) window, and the second sends information to a *timing.txt* file. If a user wants to understand where time is spent during a crack growth simulation, these options will provide insight.

The Send all warning and error messages to a file… will create a *f3d_warn_err.txt* file when running from the GUI.

The Turn on fretting menu… will make the Fretting menu available in the FRANC3D main menu. If you do not use the fretting module, you can turn it off.

The Turn on single crystal menu… will make the Single Crystal menu available in the FRANC3D main menu. If you do not use the single crystal module, you can turn it off.

The Turn on electrical menu… will make the Electrical menu available in the FRANC3D main menu. If you do not use the electrical module, you can turn it off.

The Generate Python rerun scripts is used to save Python scripts that are equivalent to the session log commands.

The Turn on symmetry crack… turns on the option to "insert" symmetry surface cracks in the flaw insertion wizard; see Section 6.1.

The Save .crk file at each growth step… will retain all of the *.crk* files as the crack is propagated. Note that the FLAWSURF block of data can be extracted from the *.fdb* file and saved as a *.crk* file also.

5.2.3 Window Tab

In the *Window* tab, one can set the font and colors used in the graphical user interface windows, Fig 5.2.5. The GUI font **Select** button will pop up a dialog box that allows one to select from the available fonts, Fig 5.2.6.

Clicking any of the color swatches (colored rectangles) will bring up a dialog box, Fig 5.2.7, that allows one to select a new color.

The GUI Equation Display option lets one adjust the equation scaling and display. Different graphics settings and monitors might display the equations or plot labels differently. These settings might help improve the display.

The Startup Window Size can be set by editing the width and height, or by selecting the **Current** button, which will measure the current window and then update the numbers.

The 'use OpenGL version 1.1' can be turned on to force the use of OpenGL 1.1 graphics. Depending on the operating system and graphics card, OpenGL version 4.5 might not be available, or might not function correctly.

The **Defaults** button restores all values to default settings.

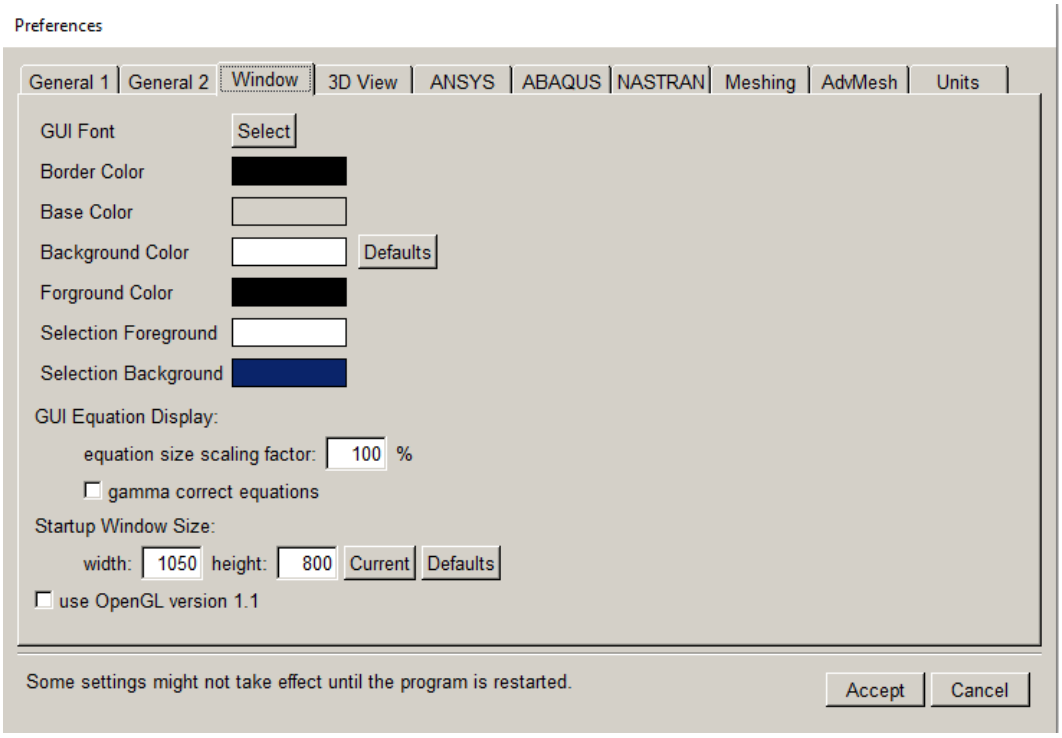


Figure 5.2.5 Window tab of Preferences dialog.

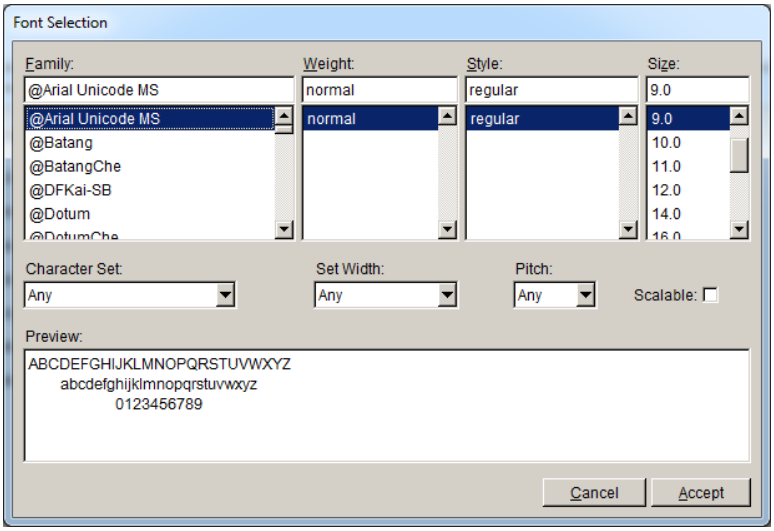


Figure 5.2.6 Font selection dialog.

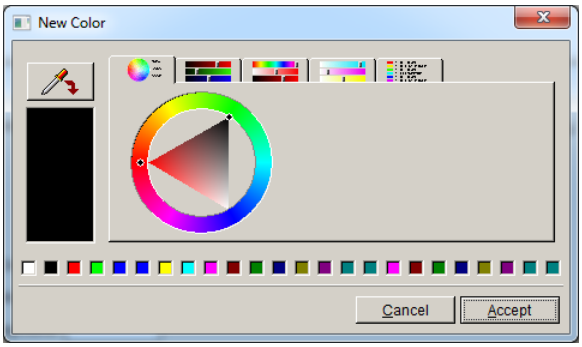


Figure 5.2.7 Color selection dialog.

5.2.4 3D View Tab

The *3D View* tab is used to set the combinations of mouse buttons and keyboard keys that are used to invoke the view and select functions, Fig 5.2.8. One should ensure that each function has a unique set of buttons and keys. The view manipulations are described in Section 2.

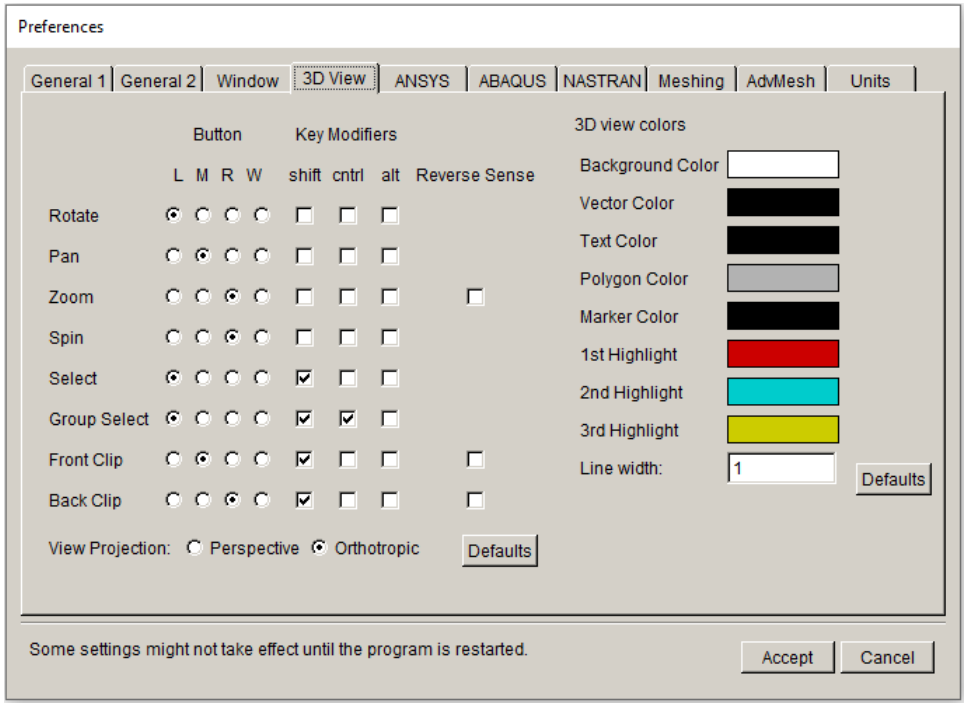


Figure 5.2.8 3D View tab of Preferences dialog.

The L, M, R and W correspond to the left, middle, right and wheel buttons of the mouse. Note that the Ctrl-key plus a left mouse button click provides a quick center of rotation selection.

The **Defaults** button (next to View Projection) restores all values to default settings. Rotate, pan, zoom and the front and back clipping options were described in Section 2.1. The Select option is used by the **Flaw Editor** (see Section 6.1.11).

In addition, the color settings in the Window tab, there are additional 3D view colors that can be set. Clicking any of the color swatches will bring up a dialog box (see Fig 5.2.7), which allows one to select a new color. The line width can be set also; the default is 1, but this value can be increased for thicker lines. The program can be restarted, or the model reread for these settings to take effect.

The **Defaults** button (next to Line width) restores all view colors to the default settings.

5.2.5 ANSYS Tab

The *ANSYS* tab allows the user to set default ANSYS analysis parameters, Fig 5.2.9. Configuration details that do not change frequently, such as the path to the ANSYS executable and the license type can be set here. These values will then appear as default values for every ANSYS analysis.

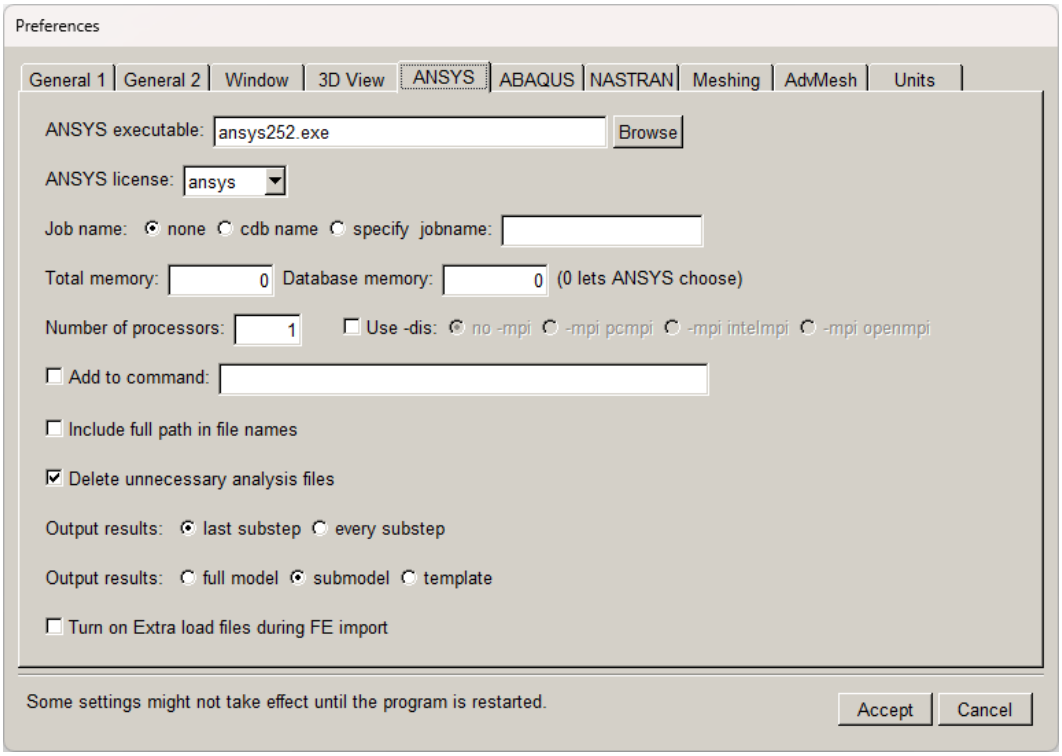


Figure 5.2.9 ANSYS tab of Preferences dialog.

ANSYS Executable stores the path to the ANSYS executable program. The **Browse** button will pop up a file browser that can be used to locate the program; Fig 5.2.9 shows the ANSYS executable for ANSYS 2021.

ANSYS License stores the ANSYS license type. Many ANSYS license strings have been encoded here. If you have a license string that is not included, you can add your string to the FRANC3D resource file. There is a *franc3D.ini* (or *.franc3d.rc* for Linux) file in the user's home folder. This file can be edited. Look for the [f3d_ansys] block and change the license string:

```
[f3d_ansys]
executable="C:\\Program Files\\ANSYS Inc\\v252\\ansys\\bin\\winx64\\ANSYS252.exe"
license=ansys
```

Job Name allows one to specify an ANSYS 'job' name.

Total and Database Memory options allow one to set the amount of memory that will be requested on the ANSYS command line (see ANSYS documentation for details).

Number of Processors is self-explanatory. The Use -dis option can be turned on for HPC license installations and analyses.

Add to command allows the user to add extra options to the ANSYS command line.

Include full path in file names adds the path to any file name for the ANSYS analysis files written by FRANC3D. Note that this option is usually needed if the analysis files are not all in the same folder.

Delete unnecessary analysis files allows FRANC3D to delete ANSYS analysis files that are not needed by FRANC3D. The ANSYS *.db* and *.rst* file are not deleted so that one can use ANSYS to visualize the model and results. Note however that these two files can be regenerated by rerunning the analysis.

Output results allows one to limit the amount of output that is written to the *.dtp* file. The default is to save results for all nodes of the local cracked submodel portion for the last substep of each load step.

Turn on Extra load files during FE import will automatically turn on the checkbox for extra load step files (see Fig 4.5.2). There are several ways to import multiple load steps for an ANSYS *.cdb* file. If the data is not included in the *.cdb* file, the .s## files should be imported when importing the *.cdb* file.

5.2.6 ABAQUS Tab

The *ABAQUS* tab allows the user to set default ABAQUS analysis parameters, Fig 5.2.10.

Abaqus Executable stores the path to the ABAQUS executable program. The **Browse** button will pop up a file browser that can be used to locate the program. For MS Windows, choose the *abaqus.bat* file from the ABAQUS *Commands* folder.

Number of Processors is self-explanatory.

Include full path in file names adds the path to any file name for the ABAQUS files written by FRANC3D. Note that this option is usually needed if the analysis files are not all in the same folder.

Delete unnecessary analysis files allows FRANC3D to delete ABAQUS analysis files that are not needed by FRANC3D. The ABAQUS *.odb* file is not deleted so that a user can use ABAQUS CAE to visualize the model and results. Note that this file can be regenerated by re-running the analysis.

Ask: 'Old job files exist. Overwrite? (y/n)' disables the ABAQUS prompt.

Output results allows one to limit the amount of output that is written to the *.dtp* file. The default is to save results for all nodes of the local cracked submodel portion for the last frame (substep) of each load step.

ABAQUS release… allows one to specify the ABAQUS version, which will affect the Python scripts that are generated to extract results from the *.odb* file. ABAQUS 2024 uses Python 3.

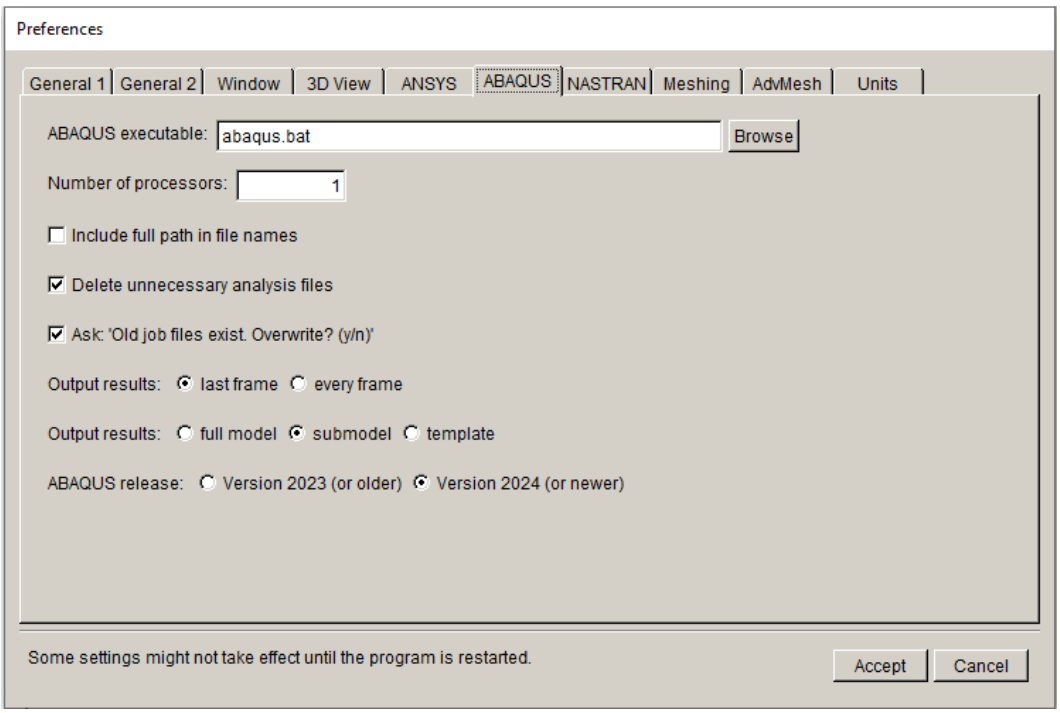


Figure 5.2.10 ABAQUS tab of Preferences dialog.

5.2.7 NASTRAN Tab

The *NASTRAN* tab allows the user to set default NASTRAN analysis parameters, Fig 5.2.11.

NASTRAN Executable stores the path to the NASTRAN executable program. The **Browse** button will pop up a file browser that can be used to locate the program.

Turn off GeomCheck allows one to turn off the GeomCheck option.
Crack front elements allows one to choose quarter-point or mid-side node locations for the crack front wedge elements. Mid-side nodes might be required for NX NASTRAN.

Include full path in file names adds the path to any file name for the NASTRAN files written by FRANC3D. Note that this option is usually needed if the analysis files are not all in the same folder.

Delete unnecessary analysis files allows FRANC3D to delete NASTRAN analysis files that are not needed by FRANC3D.

Output results allows one to limit the amount of output that is written to the *.pch* file. The default is to save results for all nodes of the local cracked submodel portion.

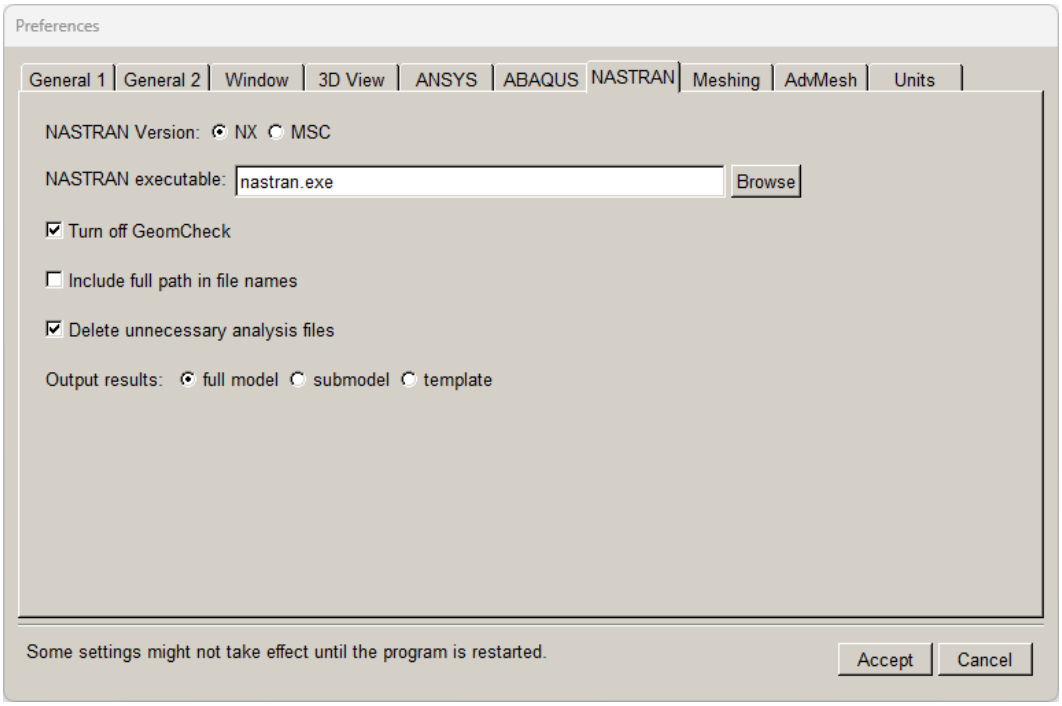


Figure 5.2.11 NASTRAN tab of Preferences dialog.

5.2.8 Meshing Tab

The *Meshing* tab allows the user to set default meshing parameters, Fig 5.2.12.

Number of element rings in template sets the number of rings; there must be a ring of singular wedge elements and at least one ring of brick elements.

Number of circumferential elements sets the number of elements per ring; there must be an even number to compute the M-integral SIFs.

Progression ratio sets the relative size of the rings.

Maximum aspect ratio sets the 'length' of the elements along the front.
Max volume elements sets a limit on the number of volume elements FRANC3D can generate.

Max backtrack restarts sets a limit on the number of FRANC3D volume meshing attempts. If the volume meshing gets stuck, the program will backtrack and restarts.

Template elements sets the default template element type(s).

Use quarter points sets the default node position for crack front elements; applies to wedge and collapsed-constrained brick elements.

Constrain bricks sets the default brick element type for all-brick templates.

Do coarsen crack mouth turns on/off the mesh coarsening along the crack mouth. For shallow cracks or for cases where the user wishes to have a higher mesh density, this box can be unchecked. The higher surface mesh density will result in a higher volume mesh density.

Do crack proximity refinement turns on/off the local surface mesh refinement for surfaces near the crack.

Do not coarsen more than uncracked mesh turns on/off the option to define the new surface mesh density based on the original surface mesh. In some cases, the default meshing algorithm creates a surface mesh that is too coarse to capture curved geometry features. Turning this option on produces a surface mesh that is more like the original uncracked surface mesh density. Further details can be found in Section 6.1.21.

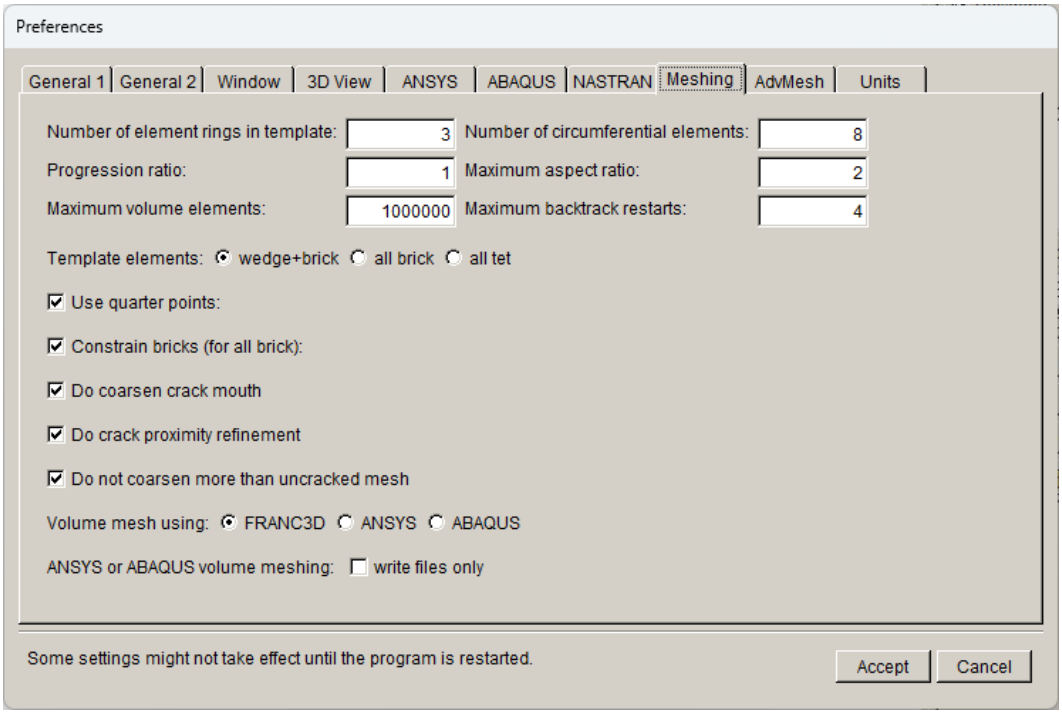


Figure 5.2.12 Meshing tab of Preferences dialog.

Volume mesh using allows the user to choose between FRANC3D, ANSYS and ABAQUS volume meshing. Note that the Max volume elements and Max backtrack restarts only applies to FRANC3D volume meshing.

ANSYS or ABAQUS volume meshing: write files only allows the user to write the surface mesh and the commands to generate the volume mesh from the surface mesh to files without running ANSYS or ABAQUS. This gives the user the option of sending the files to a different computer or modifying the commands. During the meshing phase of crack insertion, FRANC3D will prompt the user to save the surface mesh files and then wait for the volume mesh file.

5.2.9 Advanced Meshing Tab

The *Advanced Meshing* tab allows the user to set default advanced meshing parameters, Fig 5.2.13. The options are described in the dialog and are described more fully in the FRANC3D Users Guide. The volume meshing algorithm is described in journal articles; for example:

An Algorithm for Three-Dimensional Mesh Generation for Arbitrary Regions with Cracks, Cavalcante et al., Engineering with Computers (2001) 17: 75–91.

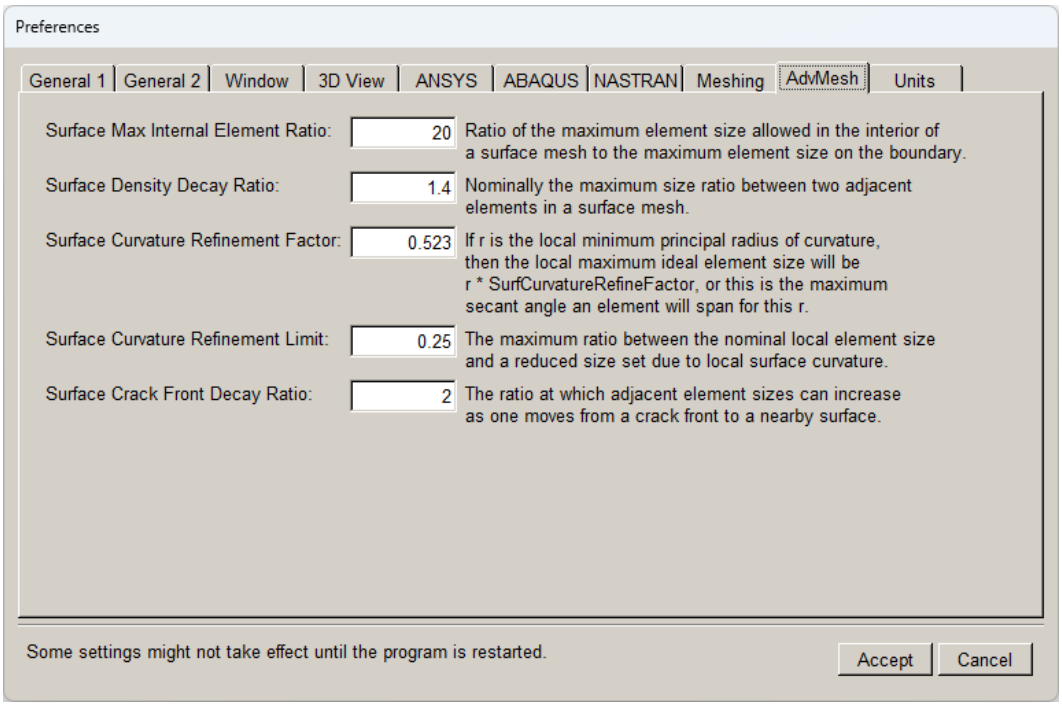


Figure 5.2.13 Advanced Meshing tab of Preferences dialog.

A brief description of each field is provided in the dialog. Decreasing the values will usually increase the mesh density.

5.2.10 Units Tab

The *Units* tab allows the user to set default units for the FE model and for display, Fig 5.2.14. The options are the same as for **Edit →Unit Conversion**; there is an extra option for changing the label for the number of passes through the load schedule. The passes are displayed in the Fatigue Life dialog (see Section 9).

The FEM and display units can be overridden when importing a new FE model or when displaying results.

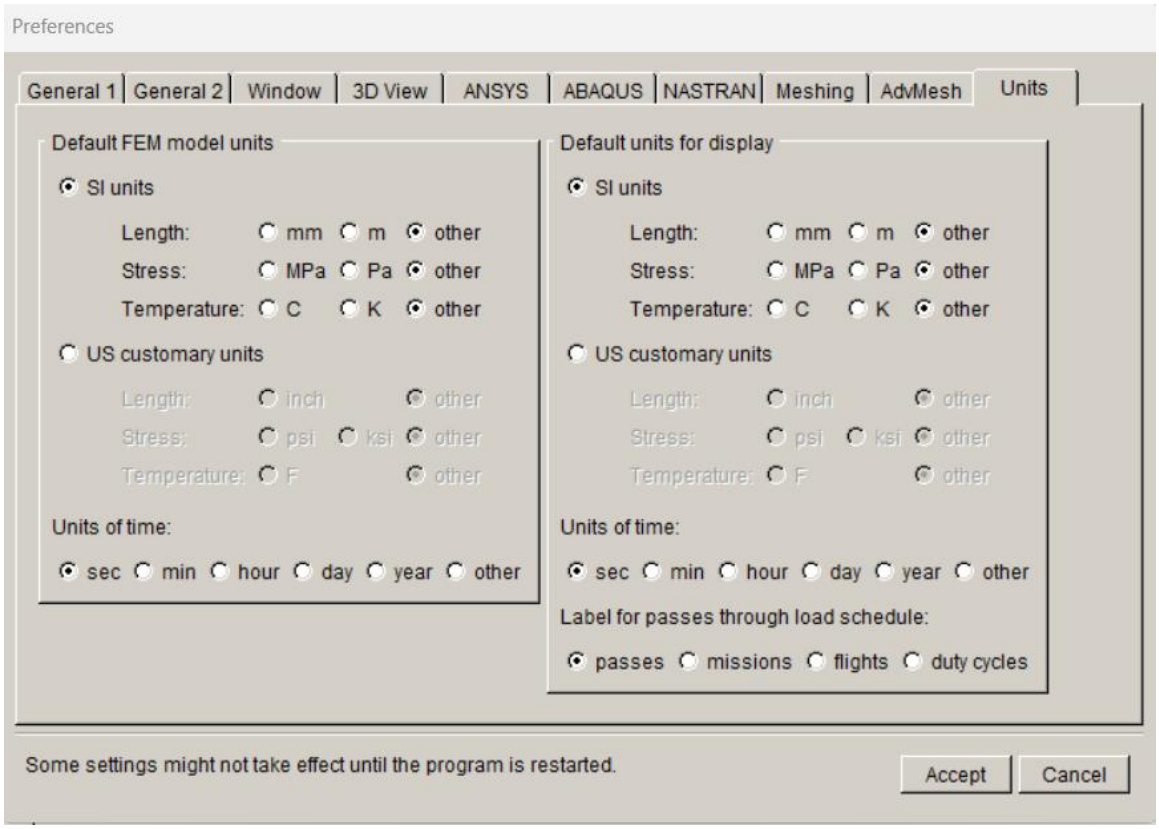


Figure 5.2.14 Units tab of Preferences dialog.

6. Cracks Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Cracks** menu options are described in this section.

The process of crack insertion (either the initial crack or subsequent steps of crack growth) requires geometric intersection of the crack surface with the model surface. The geometry for both surfaces is defined using triangular Bezier patches; this is briefly described in the User's Guide along with a summary of the geometric approach to crack extension. Surface and volume meshing is constrained by this underlying geometry; this includes the crack front template mesh described in Section 6.1.14.

6.1 New Flaw Wizard

The **New Flaw** wizard leads the user through the process of creating and orienting a parametrically defined flaw. This wizard contains several panels; however, normally one will

not see all panels. Choices in the panels determine which panels are shown next. The overall flow is illustrated in Fig 6.1.1. Note that the Elliptical and Through boxes contain multiple crack shapes.

The individual wizard panels are described below.

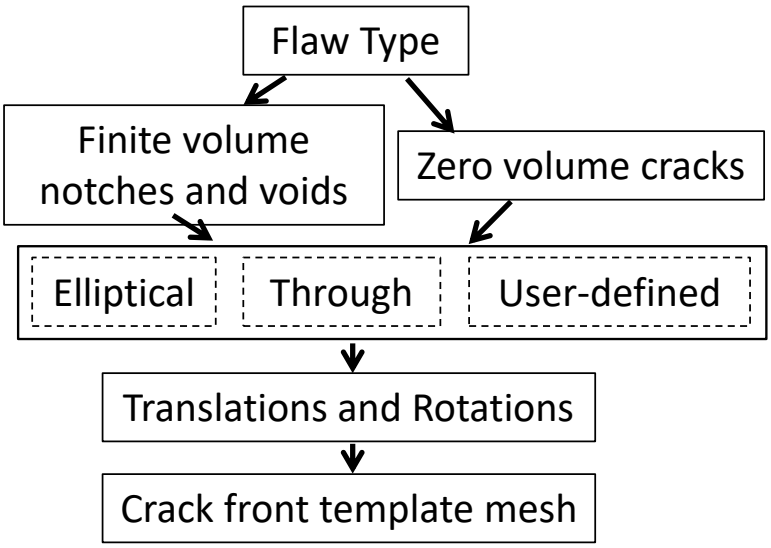


Figure 6.1.1 New Flaw wizard flow diagram.

6.1.1 Flaw type panel

The first panel determines the flaw type, Fig 6.1.2; either a crack (zero volume flaw), a notch, or a void (finite volume flaw) can be inserted.

There is the option to save the flaw description to a file. The default is to add the flaw to the model without saving to a (*.crk*) file. The second radio button (Save to file and add flaw) allows the user to save the flaw to a file and add it to the model. The third option (Save to file only) saves the flaw to a file without adding it to the model.

The last two options provide support for interface and symmetry cracks. If the user is inserting a crack that is embedded in a bi-material interface, this box should be checked. The second option will only be displayed if the "Symmetry crack" option is turned on in the Preferences (see Section 5.2.2). If the user is inserting a Symmetry surface crack, this box should be checked.

The crack types cannot be mixed; for example, you cannot insert a void and a crack into a model at the same time. Multiple flaw insertion is described in Section 6.3.

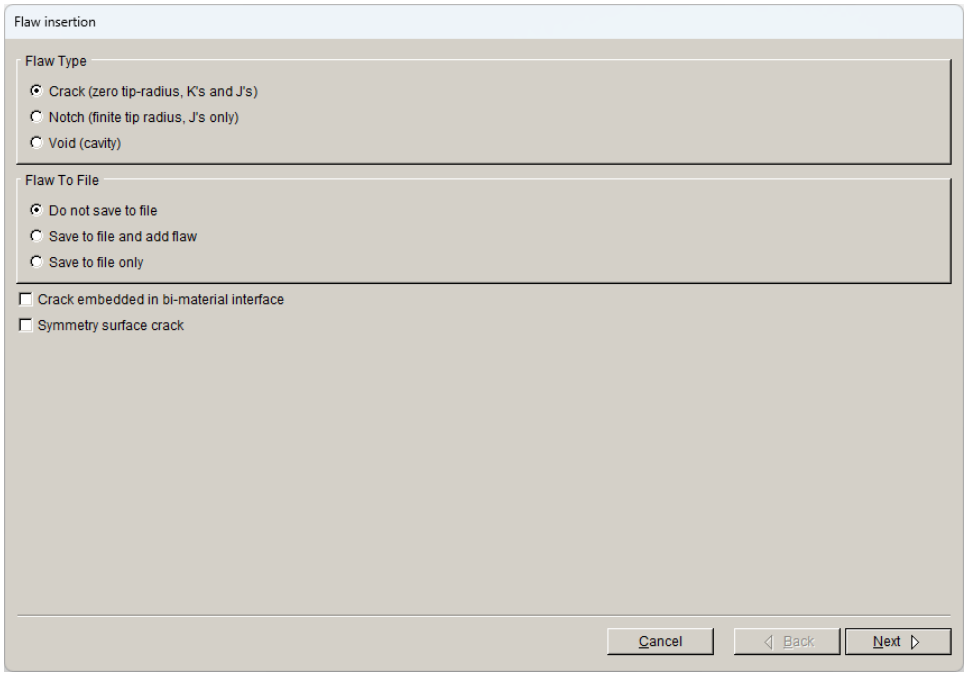


Figure 6.1.2 New Flaw wizard first panel selects flaw type.

6.1.2 Crack type panel

The current crack (zero volume flaw) types include:

1) elliptical crack with one crack front – top left icon in Fig 6.1.3,
2) through-the-thickness crack with one front – bottom left icon,
3) through-the-thickness crack with two fronts – top, second from the left,
4) long-shallow surface crack shape – bottom, second from the left,
5) elliptical crack shape with two fronts – top, third from the left,
6) long-shallow interior crack shape – bottom, third from the left,
7) curvilinear elliptical crack – top, fourth from the left,
8) user-defined crack – bottom, fourth from the left,
9) user-defined-surface mesh crack – top right, or
10) part-circular crack – bottom right.

The long shallow crack shapes can be used instead of long thin ellipses; they will produce better template elements at the ends of the major axis. The crack fronts are indicated by thicker lines. The dimension for the selected crack type are shown in the subsequent panel.

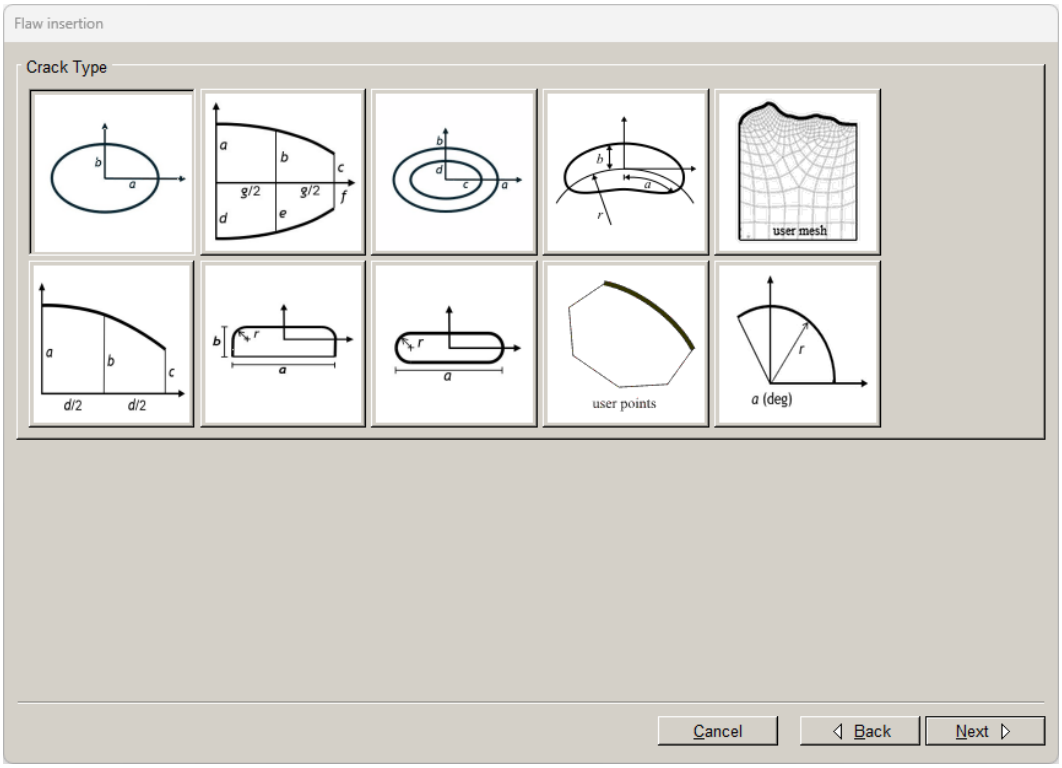


Figure 6.1.3 New Flaw Wizard crack (zero volume flaw) types.

6.1.3 Elliptical crack panel

Single front elliptical cracks are defined by entering the semi-axes lengths (a and b), Fig 6.1.4. Once values have been specified for the length of the axes, the ellipse is displayed in the 3D view window. The ellipse is displayed in its local orientation, which is in the *x*-*y* plane and centered at the global Cartesian origin.

The **Advanced Geometry** button allows one to add more boundary points, which might be needed for high aspect ratio ellipses. The associated dialog is described in Section 6.1.12. This button is available for all crack shapes except for the two user-defined cracks.

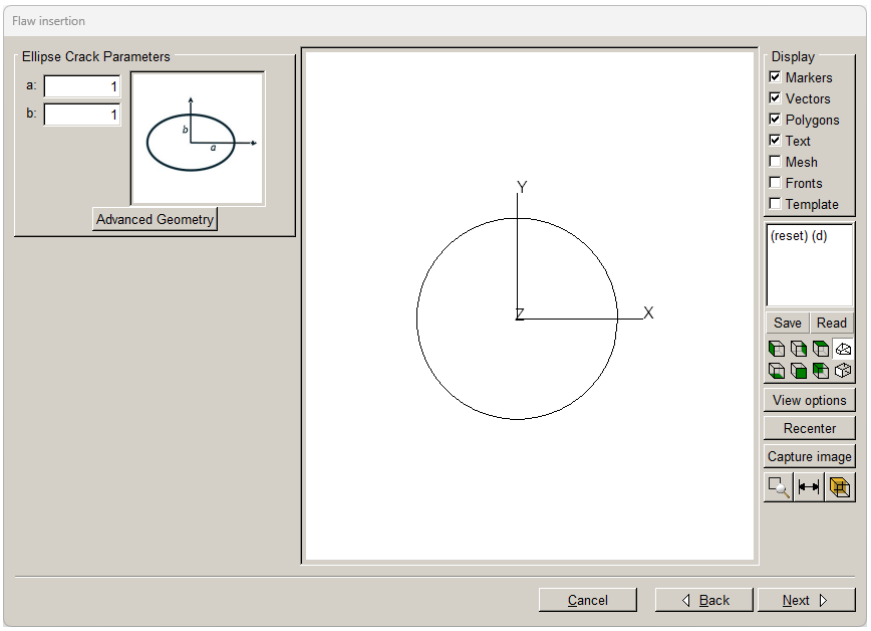


Figure 6.1.4 New Flaw Wizard elliptical crack parameters panel.

6.1.4 Single-front Through-the-thickness crack panel

Single-front through-thickness edge cracks are specified using three lengths (a – c) and the width (d), Fig 6.1.5. If set appropriately, the three lengths can be used to define a straight or a quadratic shape crack front. The crack is displayed in its local orientation, which is in the *x-y* plane with one corner at the global Cartesian origin.

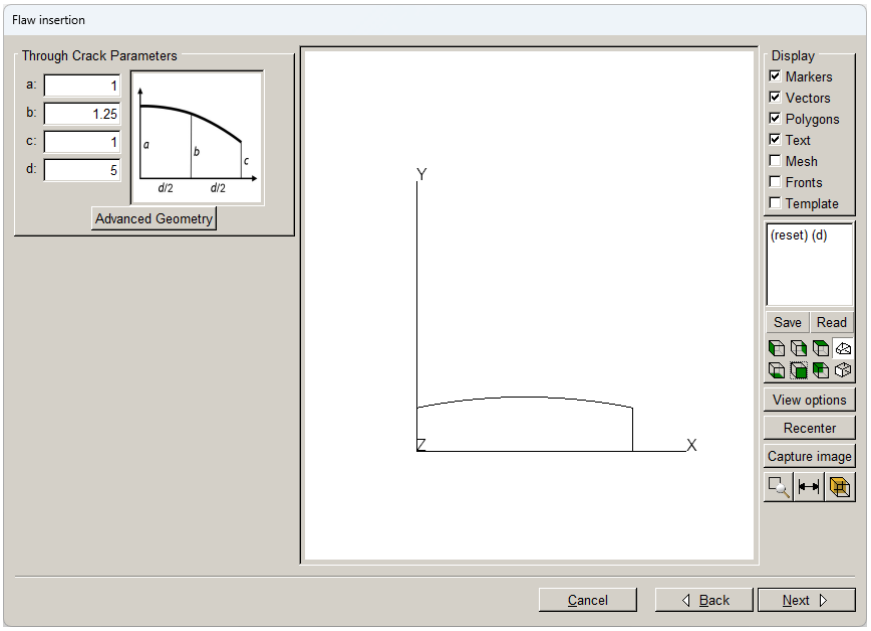


Figure 6.1.5 New Flaw Wizard single-front through-crack parameters panel.

6.1.5 Two-fronts Through-the-thickness crack panel

Two-front through-thickness center cracks are specified using six crack lengths (a – f) and the crack width (g), Fig 6.1.6. If set appropriately, the six lengths can be used to define either straight or quadratic shape crack fronts. The crack is displayed in its local orientation, which is in the *x-y* plane with one mid-side at the global Cartesian origin.

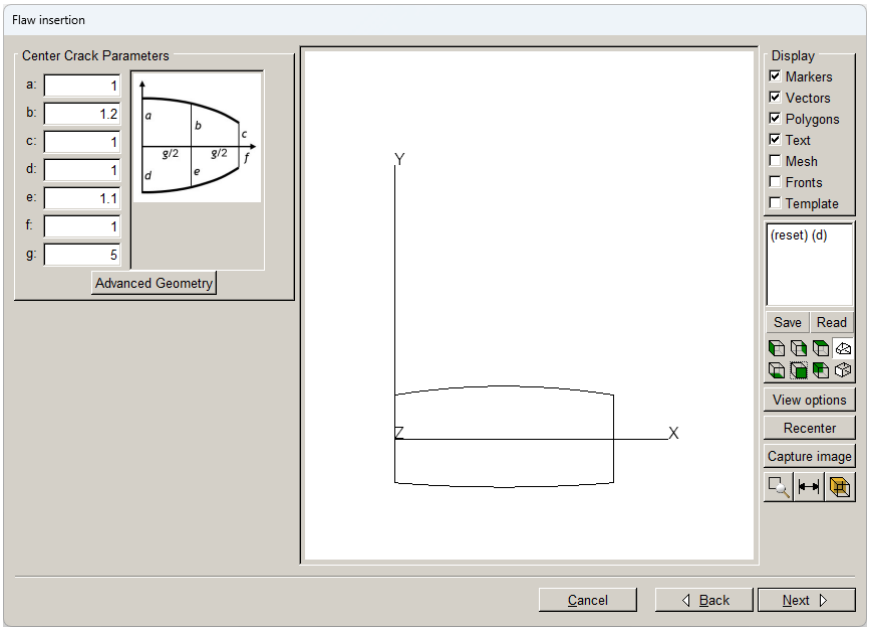


Figure 6.1.6 New Flaw Wizard two-front through-crack parameters panel.

6.1.6 Long shallow surface crack panel

Long-shallow-surface cracks are specified using the crack length (a), the crack width (b), and a corner radius (r), Fig 6.1.7. This crack shape can be used in place of high-aspect ratio elliptical surface cracks. The crack is displayed in its local orientation, which is in the *x-y* plane with the global Cartesian origin as shown in Fig 6.1.7.

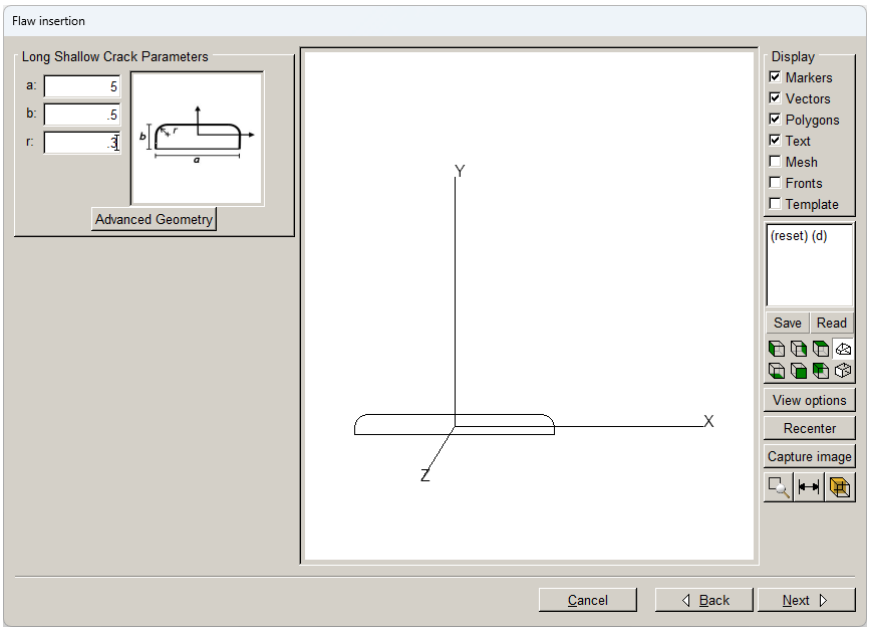


Figure 6.1.7 New Flaw Wizard long shallow surface crack parameters panel.

6.1.7 Two-fronts Elliptical crack panel

Two-front elliptical (ring) cracks, Fig 6.1.8, are defined by entering the outer semi-axes lengths (a and b) and the inner semi-axes lengths (c and d). The user also has the option of turning off the crack front for either the outer or inner front; this allows one to define a circumferential crack in the outer or inner wall of a pipe.

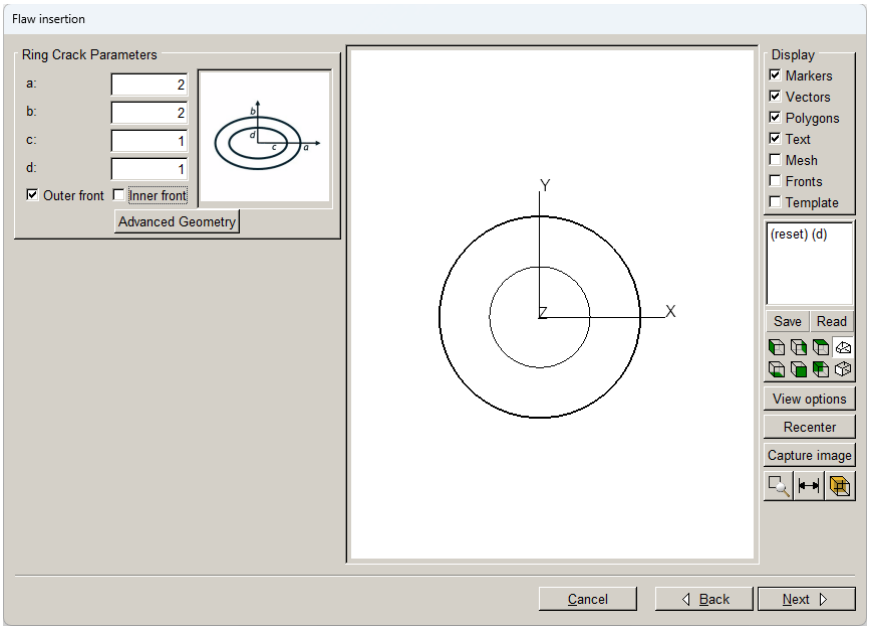


Figure 6.1.8 New Flaw Wizard two-front elliptical crack parameters panel.

Once values have been specified for the axes, the ellipses are displayed in the 3D view window, Fig 6.1.8. The crack is displayed in the local orientation, which is in the *x-y* plane and centered at the global Cartesian origin. Both outer axes must be larger than the inner axes to create a valid crack.

6.1.8 Long shallow interior crack panel

Long-shallow-interior cracks are specified using the crack length (a) and the crack corner radius (r), Fig 6.1.9. This crack shape can be used in place of high-aspect ratio elliptical interior cracks. The crack is displayed in its local orientation, which is in the *x-y* plane with the global Cartesian origin as shown in Fig 6.1.9.

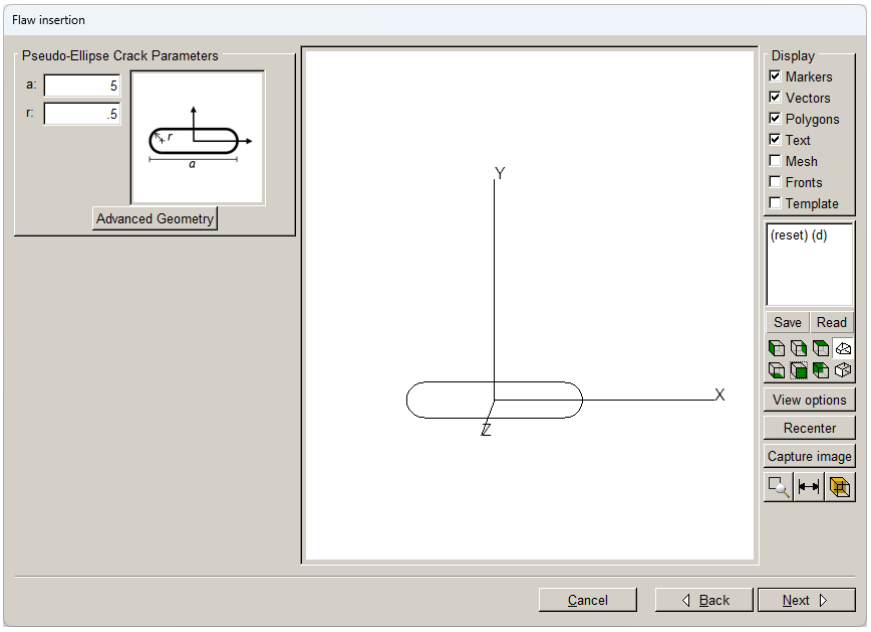


Figure 6.1.9 New Flaw Wizard long shallow interior crack parameters panel.

6.1.9 Curvilinear elliptical crack panel

Curvilinear elliptical cracks are specified by entering the semi-axes lengths (a and b) and the 'surface' radius (r), Fig 6.1.10. Once values have been specified, the ellipse is displayed in the 3D view window. The crack is displayed in its local orientation, which is in the *x-y* plane with the global Cartesian origin as shown in Fig 6.1.10. Note that there are limitations on the dimensions; you should not define a crack that has edges that overlap or intersect.

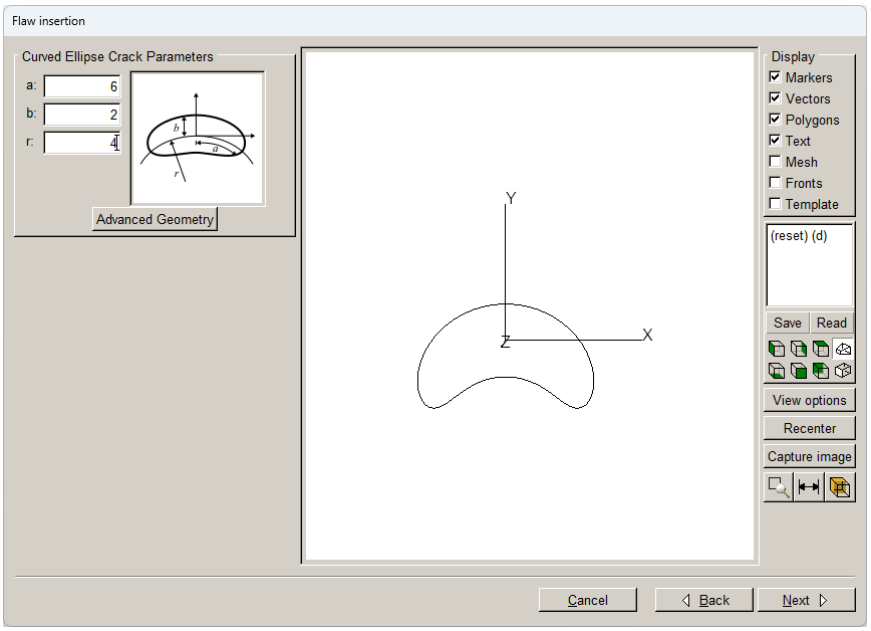


Figure 6.1.10 New Flaw Wizard long shallow interior crack parameters panel.

6.1.10 User-defined crack panel

The user-defined crack requires a set of points that defines the crack boundary. Click on **View/Edit Points**, Fig 6.1.11, to display a dialog that allows one to enter the geometry points, Fig 6.1.12. The front column in Fig 6.1.12 allows one to define points as crack front points using a value of 1; non-front points are set to 0. Note that unless one is defining an interior crack, there should be at least one point on the crack boundary that is not a crack front point. The front points are displayed and numbered corresponding to the listing shown in Fig 6.1.12. Points must be consecutive around the boundary.

One can read the points from a file using the **Read From File** button. The file should be a simple ASCII *.txt* file with the format:

x y z flag
x y z flag
x y z flag

The Smooth Front Points option in Fig 6.1.11 allows the user to smooth and re-parameterize the crack front points. This allows for more uniform crack surface geometry.

The boundary points are used with a surface triangulation algorithm to produce the interior crack surface geometry. Enough non-front points should be defined to produce a reasonable triangulation of the interior region. The **Generate Non-Front** button can be used to create non-front points if none exist. Note that the first and last boundary point should not be repeated.

Some non-planarity is possible; however, the user cannot define crack-surface-interior points, therefore, this crack type works best for planar or near-planar cracks.

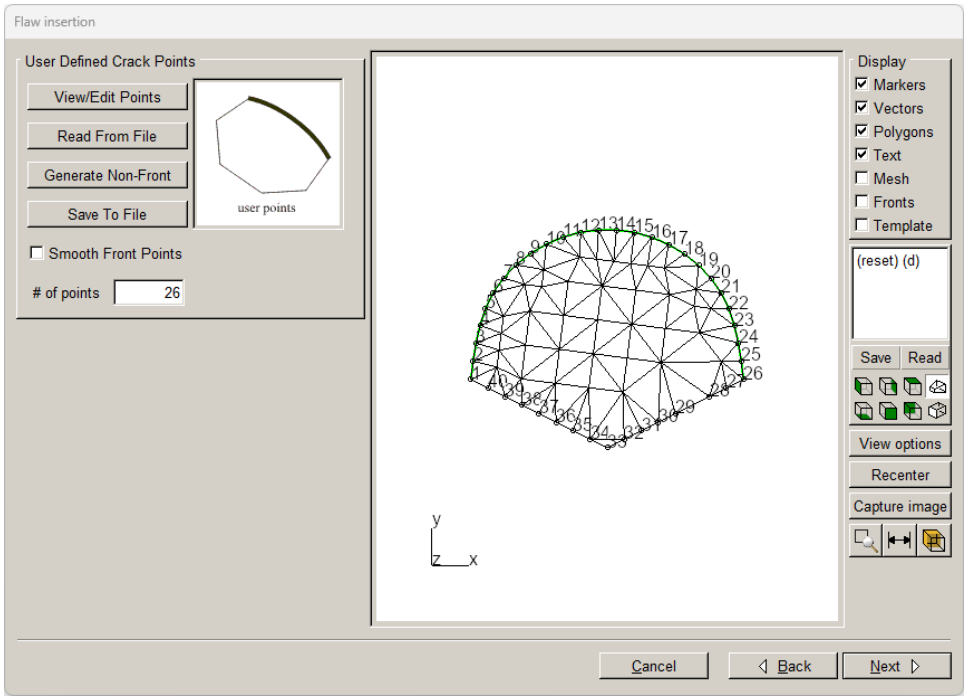
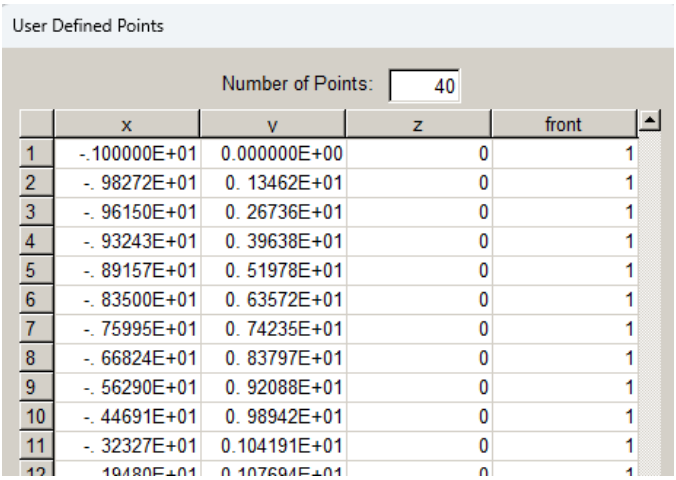


Figure 6.1.11 New Flaw Wizard user-defined crack panel.



	x	y	z	front
1	-.100000E+01	0.000000E+00	0	1
2	-. 98272E+01	0. 13462E+01	0	1
3	-. 96150E+01	0. 26736E+01	0	1
4	-. 93243E+01	0. 39638E+01	0	1
5	-. 89157E+01	0. 51978E+01	0	1
6	-. 83500E+01	0. 63572E+01	0	1
7	-. 75995E+01	0. 74235E+01	0	1
8	-. 66824E+01	0. 83797E+01	0	1
9	-. 56290E+01	0. 92088E+01	0	1
10	-. 44691E+01	0. 98942E+01	0	1
11	-. 32327E+01	0.104191E+01	0	1
12	-.19480E+01	0.107694E+01	0	1

Figure 6.1.12 User-defined crack points dialog.

6.1.11 User-mesh crack panel

The user-mesh crack allows one to convert a surface mesh into a crack. Click on **Read mesh File**, Fig 6.1.13, to display the file selector dialog, Fig 6.1.14. FRANC3D can read ABAQUS (*.inp*), ANSYS (*.cdb*) and NASTRAN (*.bdf*) files containing planar or shell elements. Select **Accept** and the surface will be displayed, Fig 6.1.15.

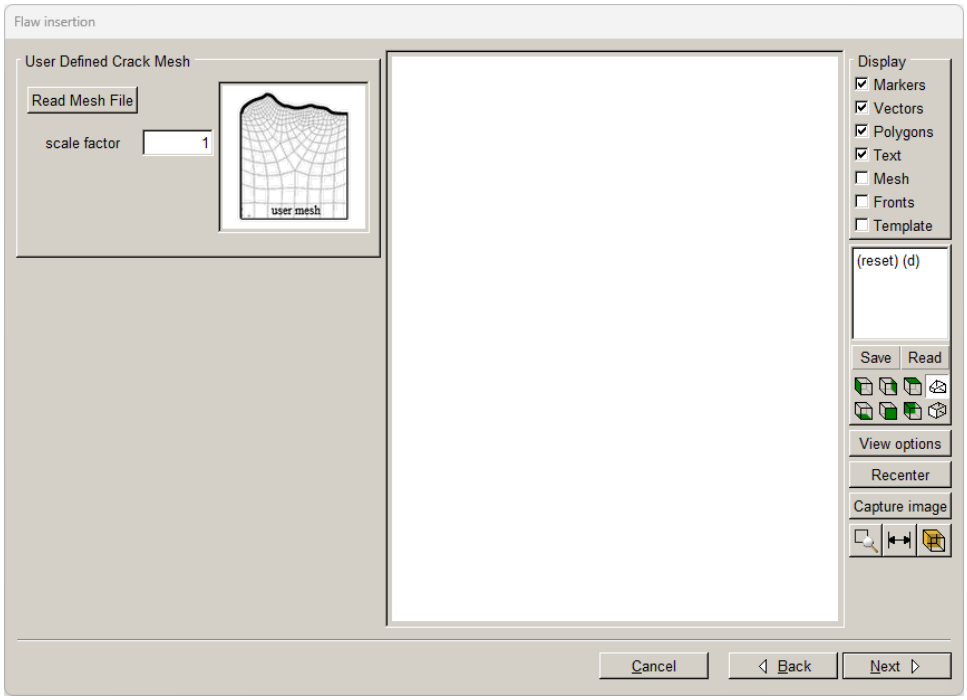


Figure 6.1.13 New Flaw Wizard user-mesh crack panel.

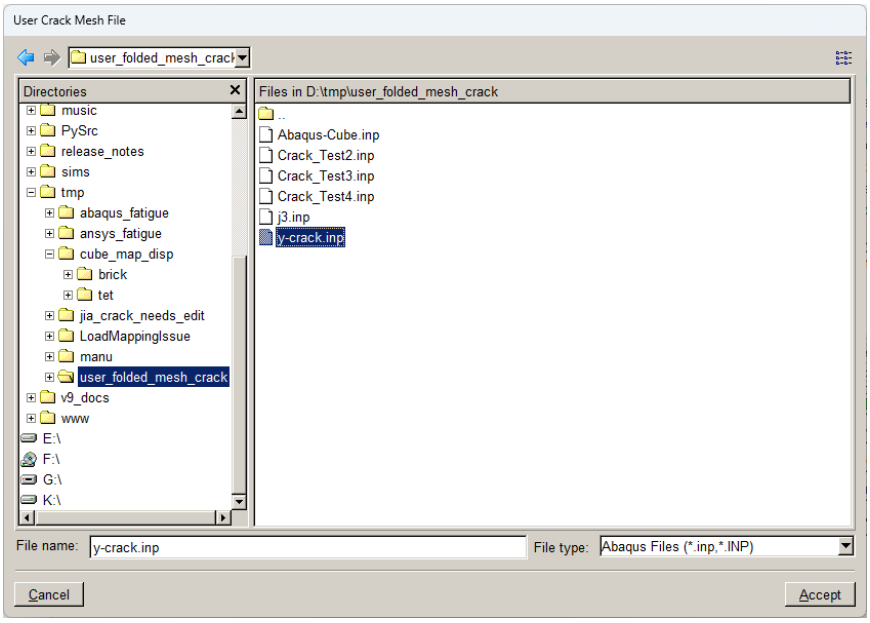


Figure 6.1.14 User-mesh crack mesh file dialog.

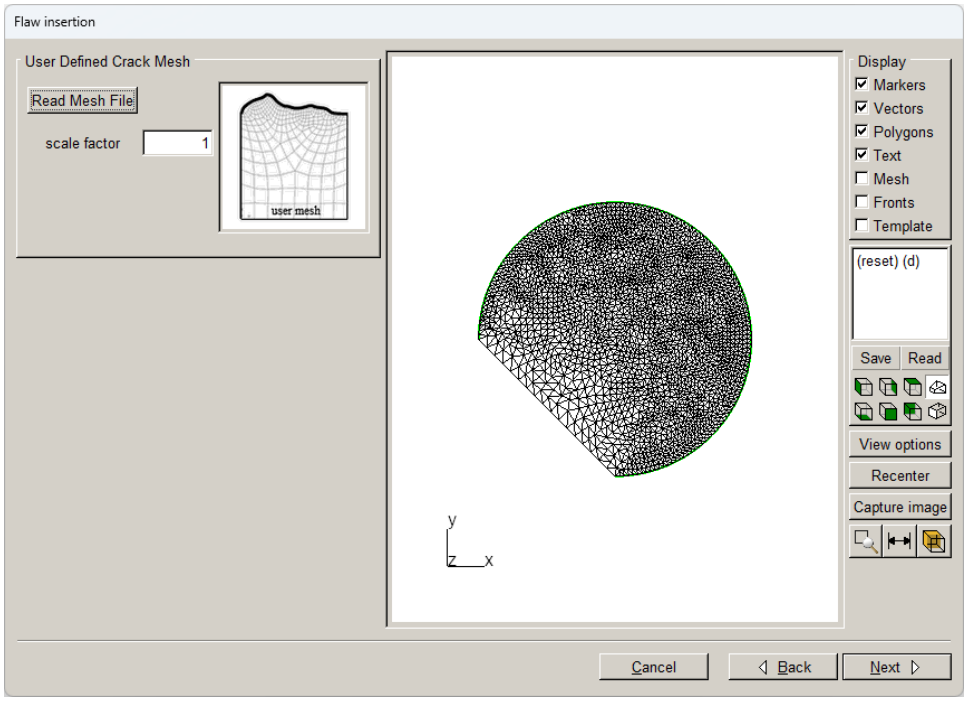


Figure 6.1.15 User-mesh crack surface.

The scale factor field, in Fig 6.1.15, allows one to scale the user-mesh crack. This can be especially useful for STL files that are saved using units of meters when the model is in millimeters.

In addition to the surface mesh files, FRANC3D can also import STL files, which is just a collection of triangles. Chapter 6 of the User's Guide provides additional information for user-defined crack shapes.

6.1.12 Part-circle crack panel

Part-circle cracks are defined by entering the degrees and radius (a and r), Fig 6.1.16. Once values have been specified, the portion of the circle is displayed in the 3D view window. The flaw is displayed in its local orientation, which is in the *x-y* plane and centered at the global Cartesian origin.

6.1.13 Advanced Geometry button

Selecting the **Advanced Geometry** button on the predefined crack shapes (where it exists) leads to the dialog shown in Fig 6.1.17.

There are two buttons on the top menu bar: **Refine** and **Reset**. The crack surface (triangular patches) or the boundary geometry is shown in the window. The two through-cracks are the only library flaws that use triangular patches, see Fig 6.1.18.

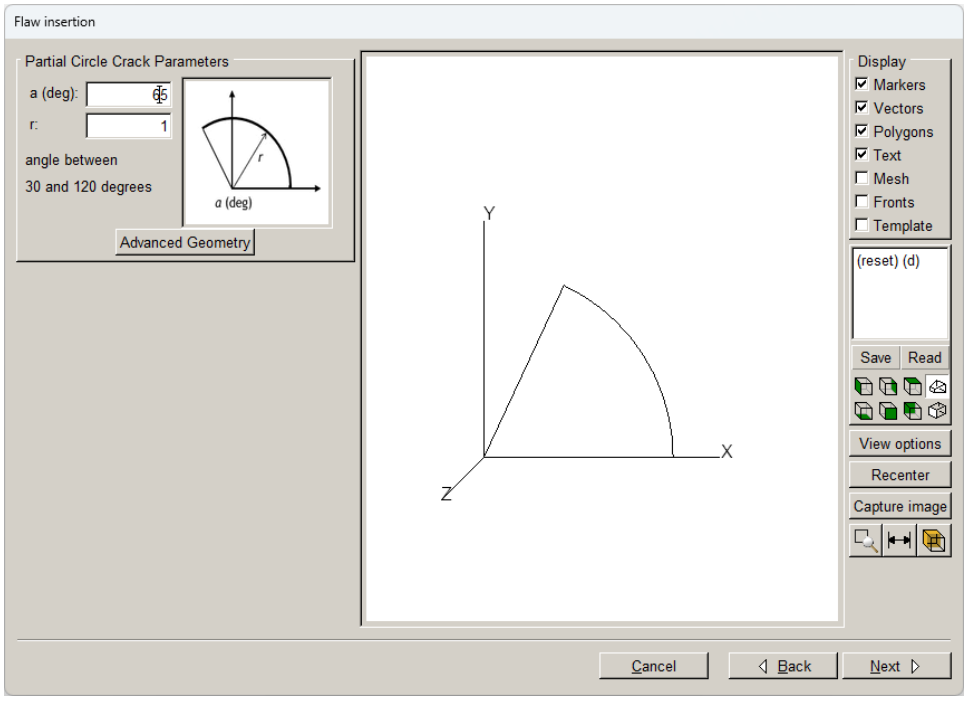


Figure 6.1.16 New Flaw Wizard long shallow interior crack parameters panel.

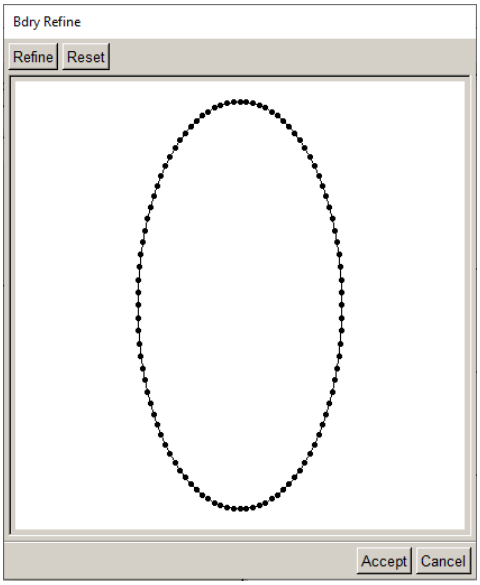


Figure 6.1.17 Crack Advanced Geometry dialog.

6.1.13.1 Refine

The **Refine** button increases the discretization of the crack surface or boundary geometry, Fig 6.1.18. Although cracks notionally have an analytical geometry (e.g., elliptical or quadratic),

they are defined geometrically by triangular cubic-Bezier patches. These patches only approximate the analytical form. As the number of geometry patches is increased, the approximation improves but at the cost of additional processing time required to insert the crack. The **Refine** button performs a refinement of the patches or the boundary points. The **Reset** button sets the geometry back to the default setting.

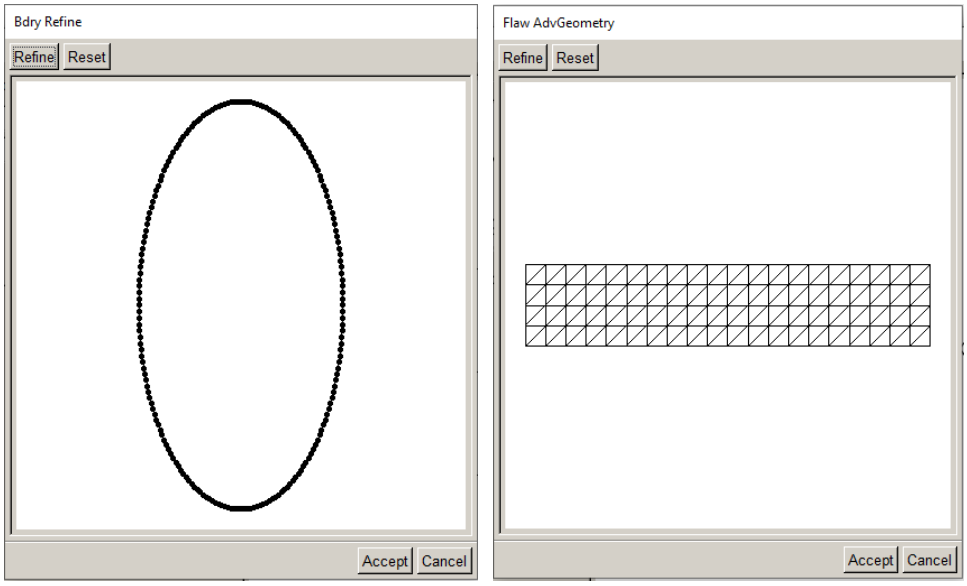


Figure 6.1.18 Example of a **Refined** geometry for the elliptical (left) and through (right) cracks.

6.1.14 Notch type panel

The current notch types include:

1) thick ellipse – top left icon in Fig 6.1.19,
2) thick through-thickness with two fronts – top middle,
3) thick through-thickness with one front – bottom left,
4) thick long-shallow surface – bottom middle, and
5) thick curvilinear ellipse – top right.

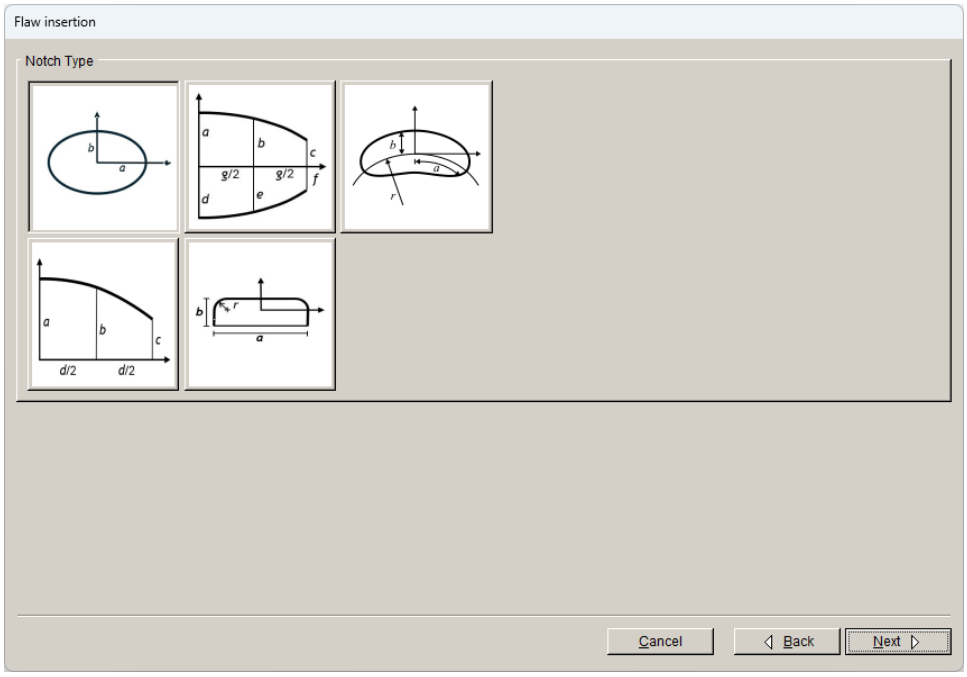


Figure 6.1.19 New Flaw Wizard notch types.

6.1.15 Thick (flat) elliptical void panel

Single-front thick ellipse notches are defined by entering the semi-axes lengths (a and b) and the thickness, Fig 6.1.20. Once values have been specified, the thick ellipse is displayed in the 3D view window. It is displayed in its local orientation, which is in the *x*-*y* plane and centered at the global Cartesian origin.

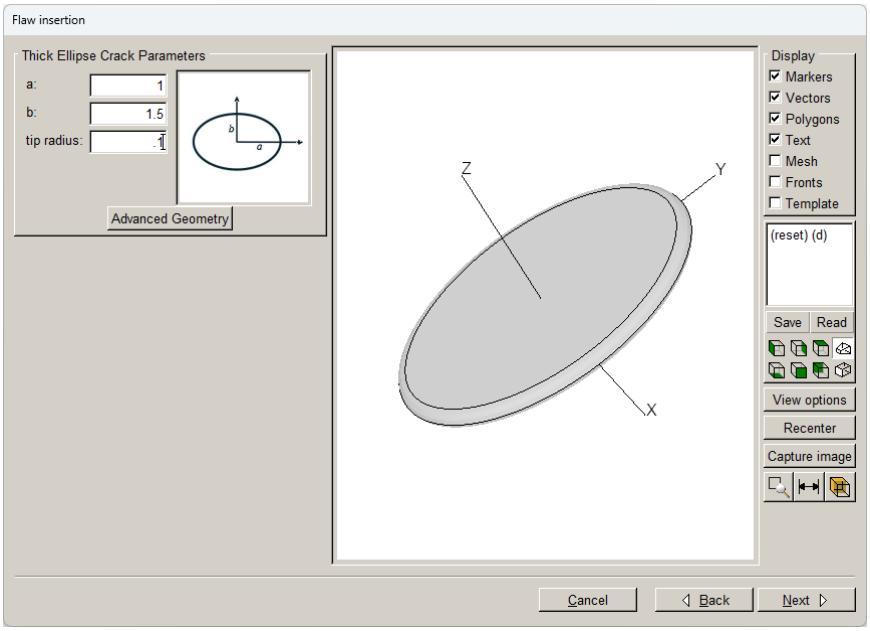


Figure 6.1.20 New Flaw Wizard elliptical crack parameters panel.

6.1.16 Two-fronts Through-thickness void panel

Two-front through-thickness notches are specified using six crack lengths (a – f), the crack width (g) and the thickness, Fig 6.1.21. If set appropriately, the six lengths can be used to define either straight or quadratic shape crack fronts. The crack is displayed in its local orientation, which is in the *x-y* plane with one mid-side at the global Cartesian origin.

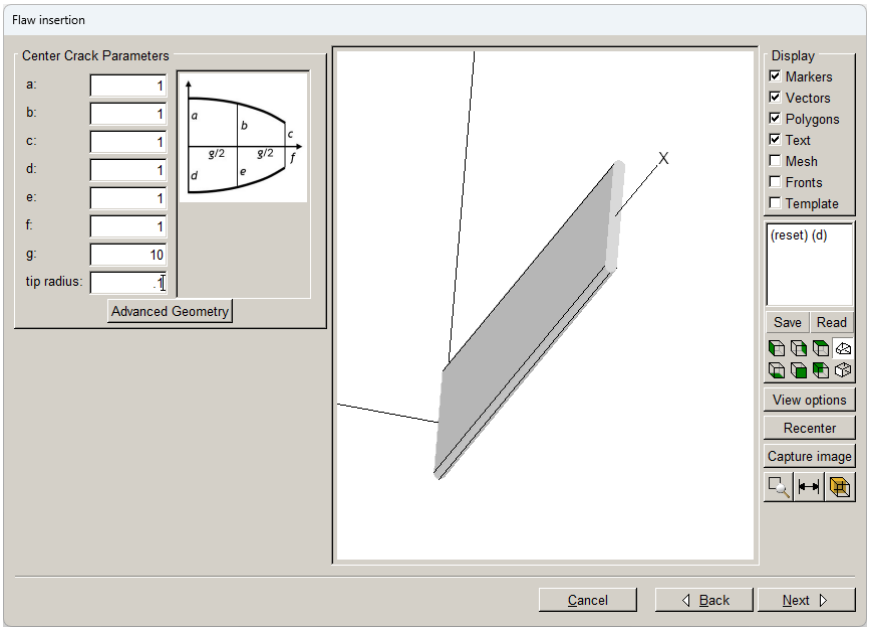


Figure 6.1.21 New Flaw Wizard two-front through-crack parameters panel.

6.1.17 Single-front Through-thickness void panel

Single-front through-thickness notches are specified using three lengths (a – c), the width (d) and the thickness, Fig 6.1.22. If set appropriately, the three lengths can be used to define a straight or a quadratic shape crack front. The crack is displayed in its local orientation, which is in the *x-y* plane with one corner at the global Cartesian origin.

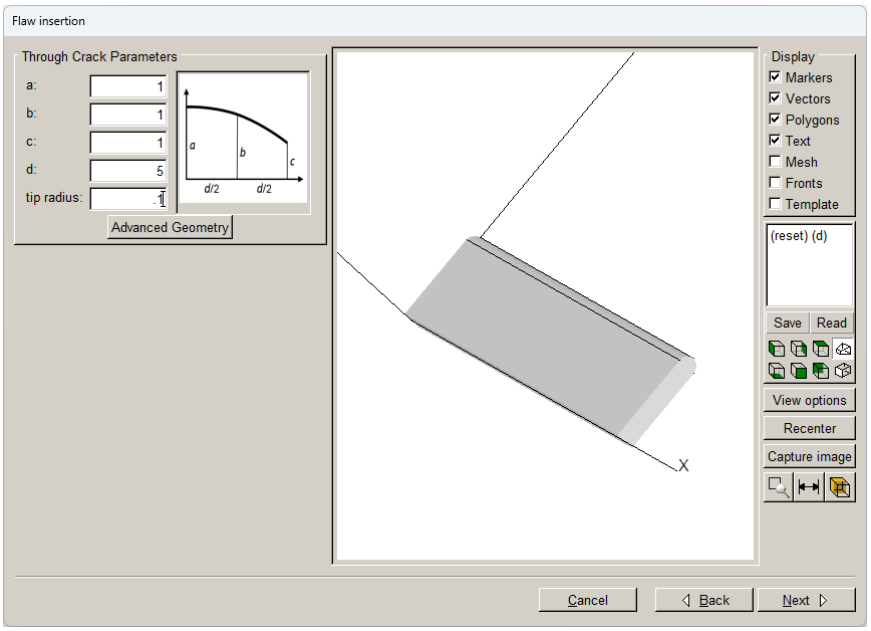


Figure 6.1.22 New Flaw Wizard single-front through-crack parameters panel.

6.1.18 Long shallow surface void panel

Long-shallow-surface notches are specified using the crack length (a), the crack width (b), a corner radius (r) and the thickness, Fig 6.1.23. The crack is displayed in its local orientation, which is in the *x-y* plane of the global Cartesian system.

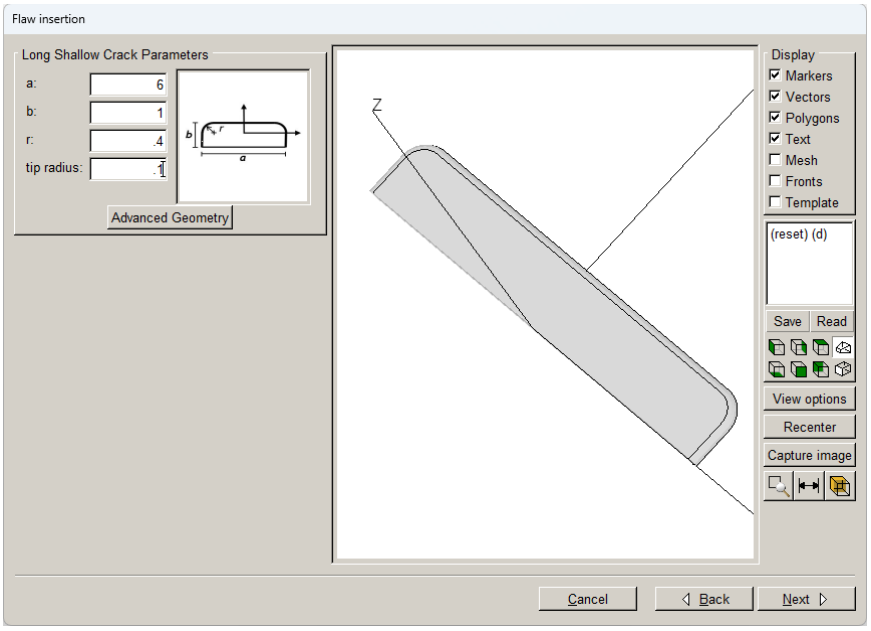


Figure 6.1.23 New Flaw Wizard long shallow surface crack parameters panel.

6.1.19 Curvilinear elliptical void panel

Curvilinear elliptical notches are specified by entering the semi-axes lengths (a and b), the 'surface' radius (r) and the thickness, Fig 6.1.24. The crack is displayed in its local orientation, which is in the *x-y* plane with the global Cartesian origin as shown in Fig 6.1.24. Note that there are limitations on the dimensions; you should not define a notch that has edges that overlap or intersect.

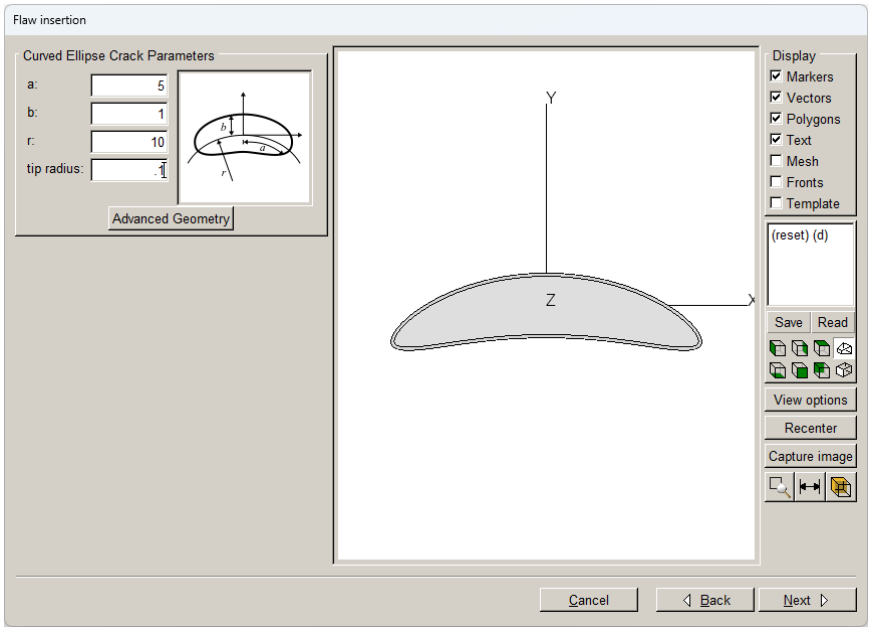


Figure 6.1.24 New Flaw Wizard long shallow interior crack parameters panel.

6.1.20 Void type panel

The current void types include:

1) ellipsoid – top left icon in Fig 6.1.25,
2) user-defined-volume mesh crack – top right.

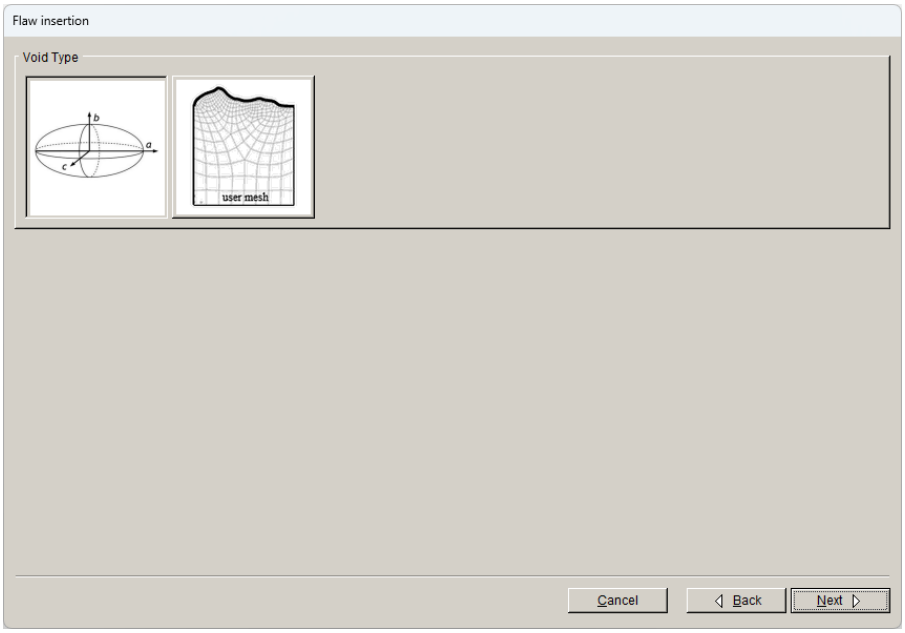


Figure 6.1.25 New Flaw Wizard void (finite volume flaw) types.

6.1.21 Ellipsoid panel

Ellipsoidal voids are defined by entering the three semi-axes lengths (a, b, and c), Fig 6.1.26. The void is displayed in the local orientation and centered at the global Cartesian origin. There are no "front" edges identified for an ellipsoid.

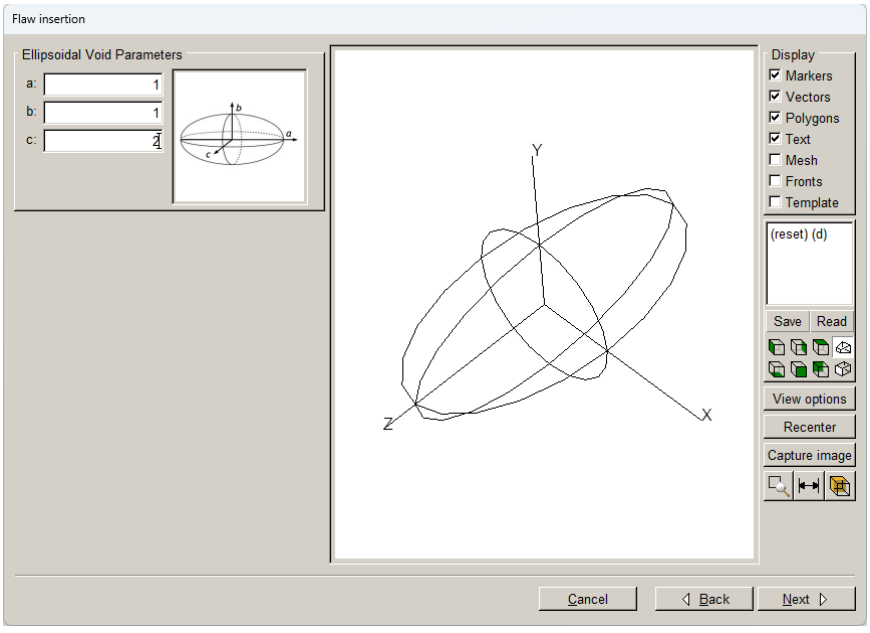


Figure 6.1.26 New Flaw Wizard ellipsoidal void parameters panel.

6.1.22 User-mesh void panel

User-mesh voids are added by importing a volume-mesh. Click on **Read mesh File**, Fig 6.1.27, to display the file selector dialog, Fig 6.1.28. FRANC3D can read ABAQUS (*.inp*), ANSYS (*.cdb*) and NASTRAN (*.bdf*) files containing volume elements.

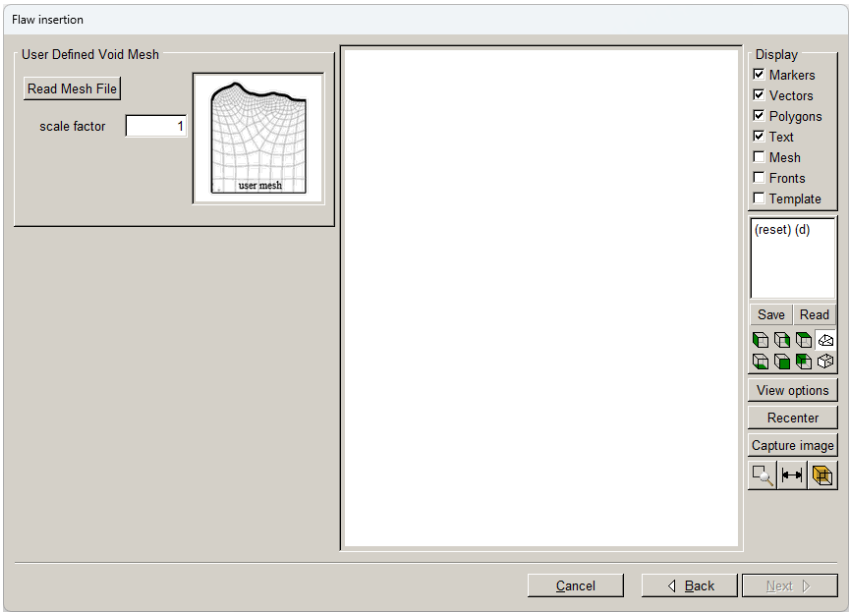


Figure 6.1.27 New Flaw Wizard user-mesh crack panel.

For example, a cylinder model is selected, Fig 6.1.29, and inserted into a cube to create a cylindrical void. The user-mesh void can be translated into position relative to the model, Fig 6.1.30; the resulting model surface mesh is shown in Fig 6.1.31.

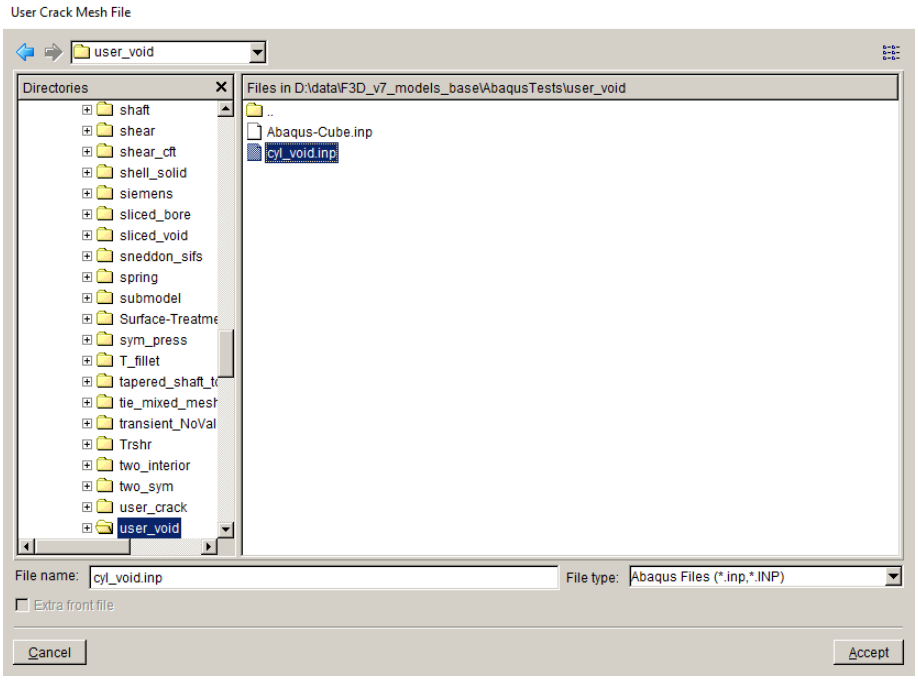


Figure 6.1.28 User-mesh void file dialog.

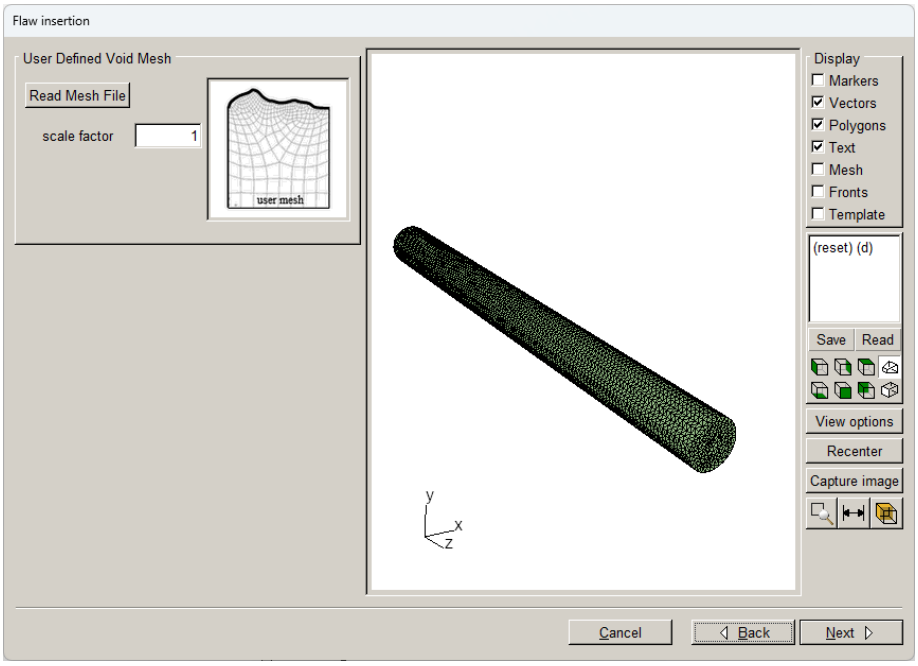


Figure 6.1.29 Cylinder volume mesh.

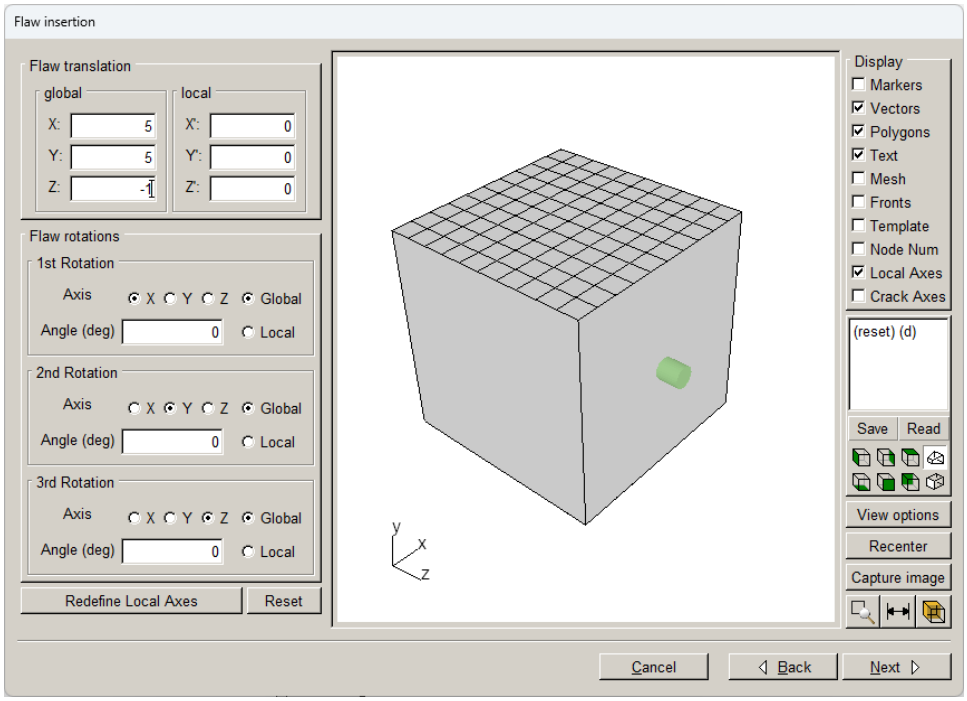


Figure 6.1.30 Cylinder volume mesh inserted into a cube.

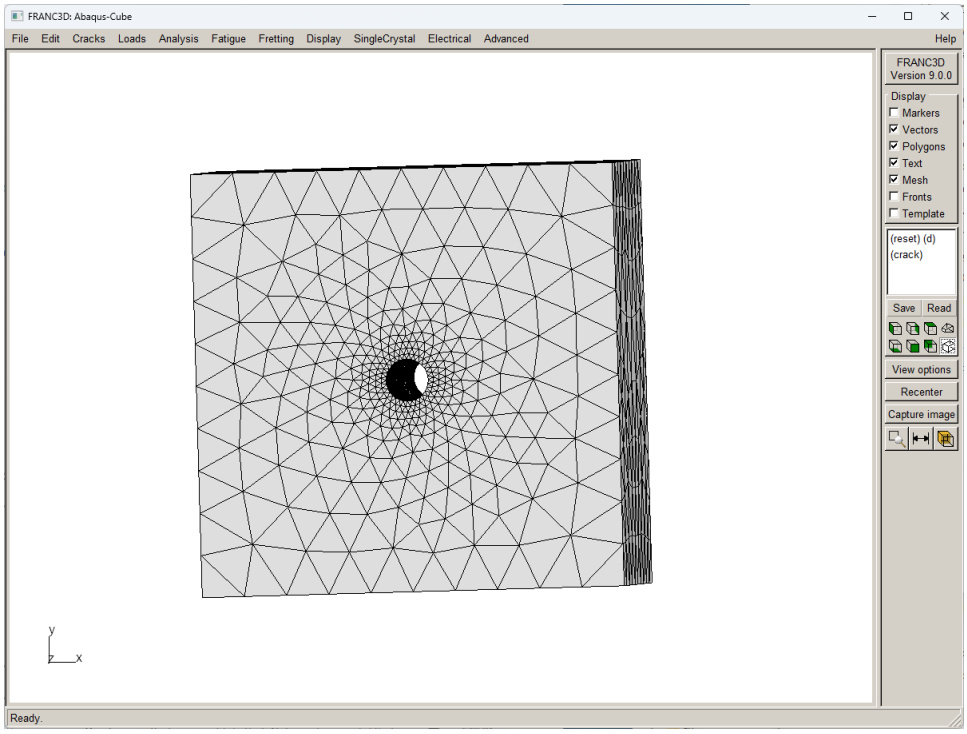


Figure 6.1.31 Cylindrical void in a cube.

6.1.23 Symmetry crack surface select panel

If a user checks the Symmetry surface crack box (see Fig 6.1.2), an extra dialog is displayed prior to selecting the crack type, Fig 6.1.32. Surfaces with symmetry-boundary conditions are displayed in blue. The user should choose the surface using the Shift key and the left mouse button (assuming default mouse settings are used). Click **Next** to continue defining the crack.

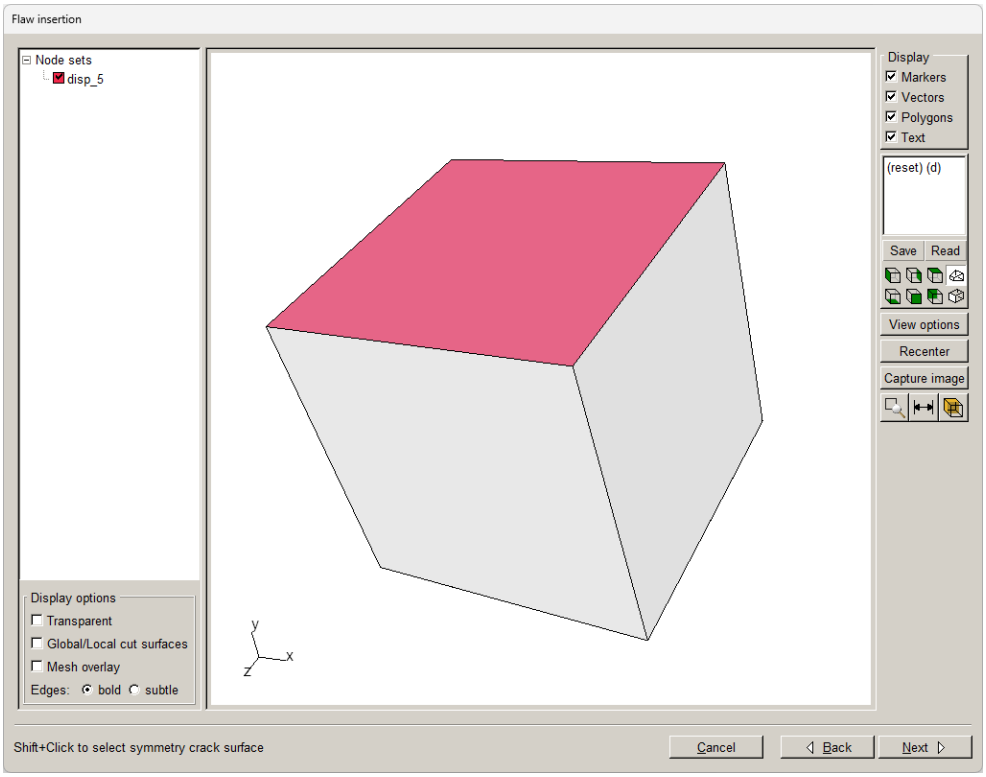


Figure 6.1.32 Select symmetry surface dialog.

6.1.24 Flaw translation and rotation panel

Once a flaw (crack, notch, or void) is defined, it must be translated and rotated into the proper location relative to the unflawed body. The flaw orientation panel displays the flaw and the model in a 3D graphics window, along with controls to translate and rotate the flaw, Fig 6.1.33. Translations and rotations can be specified in the global or in a local coordinate system.

The translations move the origin of the flaw. It is usually simpler to specify translations and then "zoom in" on the display of the flaw before specifying the rotations.

Up to three rotations about the (rotated) Cartesian axes can be specified. These are Euler angles, but the order of the rotation axes can be specified. The rotations follow a "right-hand-rule" where the thumb points in the positive direction along the axis and the fingers curl in the positive rotation direction.

The **Define Local Axes** button allows the user to specify a local Cartesian system to aid in placing the flaw; this is described in the next sub-section. The adjacent **Reset** button clears any local coordinate system and resets the original default crack translation and rotation.

The Display panel on the right side has two additional options to display the Local and Crack Axes.

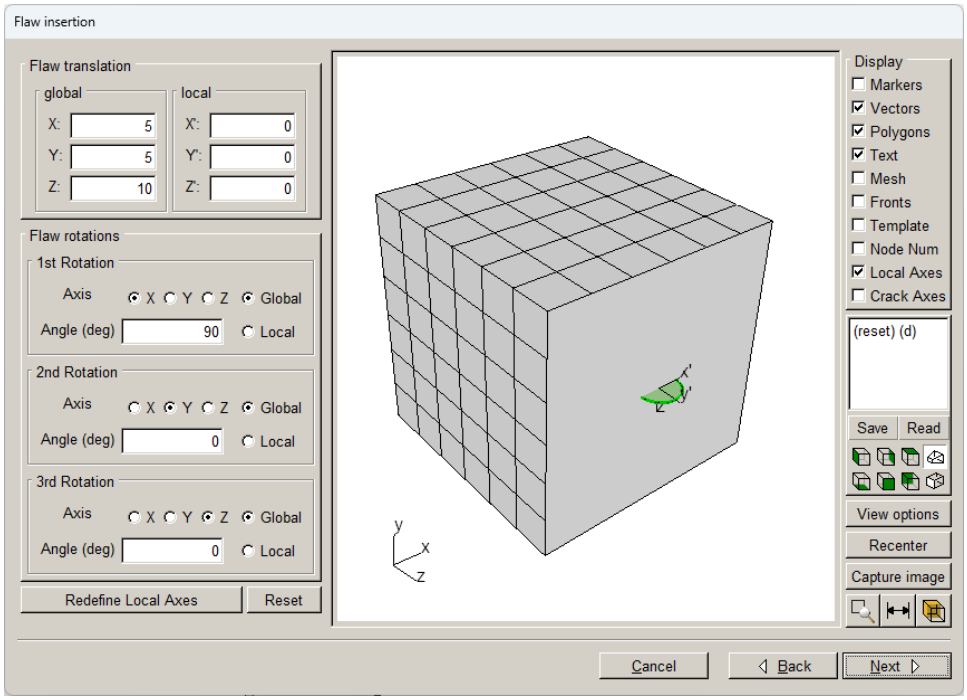


Figure 6.1.33 New Flaw Wizard orientation panel.

6.1.24.1 Define Local Axes

The **Redefine Local Axes** button displays a dialog, Fig 6.1.34, that allows one to specify a local Cartesian coordinate system. See Section 6.5 of the User's Guide for additional details on flaw placement and local axes.

The Anchor at node option allows one to specify a node ID, which will define the crack local origin. One can specify Normal to Surface or define the local x- and y-axes. For the first option, the surface-normal at the specified node in the model defines the crack plane.

The Anchor at point normal to surface is like the previous option, but the user must specify the global Cartesian coordinates of the point on the surface.

The Define by three nodes option allows one to specify three node IDs to define the origin and crack plane.

The Define by three points option is like the previous option, but the user must specify the global Cartesian coordinates of the three points.

The Define by angles option allows the user to specify the global Cartesian origin and three rotations.

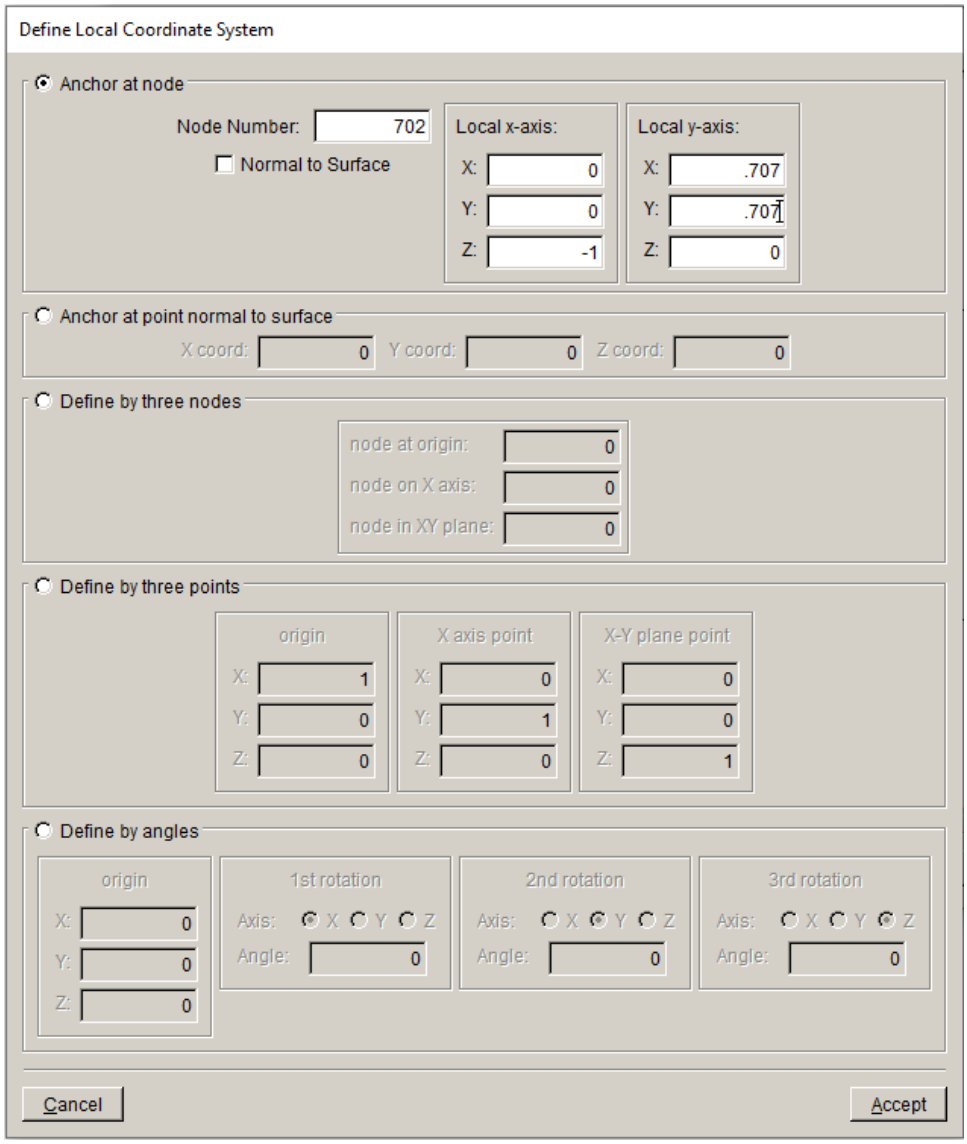


Figure 6.1.34 New Flaw Wizard orientation vectors dialog.

6.1.25 Crack Front Mesh Template Panel

For accurate stress-intensity factor computations, a pattern or "template" of elements with controlled sizes and shapes is placed about all crack fronts. The template takes the form of

generalized cylindrical tubes of elements with the crack fronts serving as the axes of the cylinders. Wedge shaped elements are placed immediately adjacent to the crack fronts. These are surrounded by rings of brick elements, Fig 6.1.35.

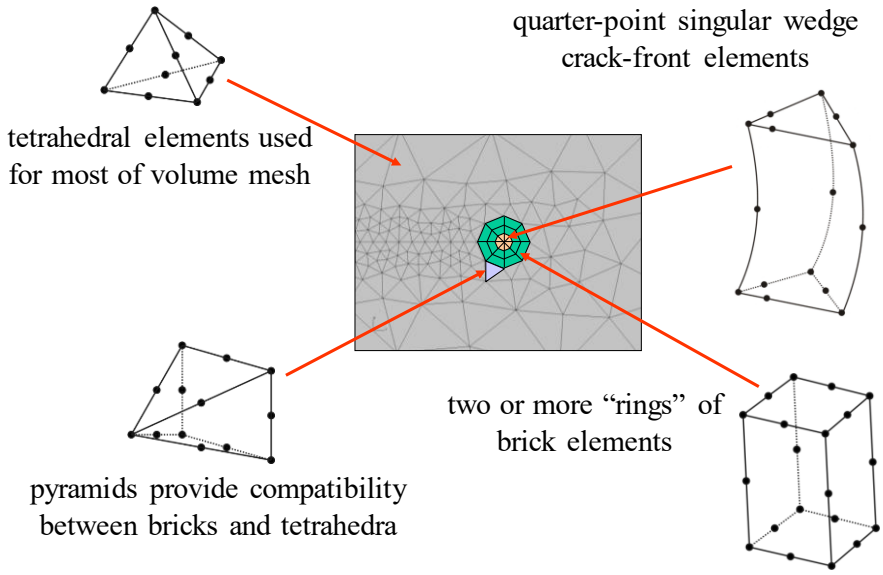


Figure 6.1.35 Crack front mesh element types.

Fig 6.1.36 shows the Crack Front Mesh Template wizard panel with an elliptical surface crack. The template elements are displayed in the 3D graphics window superimposed on the flaw and model geometry. By default, only half of the template elements are shown. This allows one to view the progression of element sizes as the elements transition from the crack front to the outer boundary of the template. One can view the full template by turning on Display full template.

A notch front will have a template, similar to the crack front; a void will not have a template.

The **Meshing parameters** button is used to specify surface and volume meshing parameters as well as choosing the volume mesher (see Section 6.1.25.1). The **Configure Template** button allows the user to adjust the template settings (see Section 6.1.25.2).

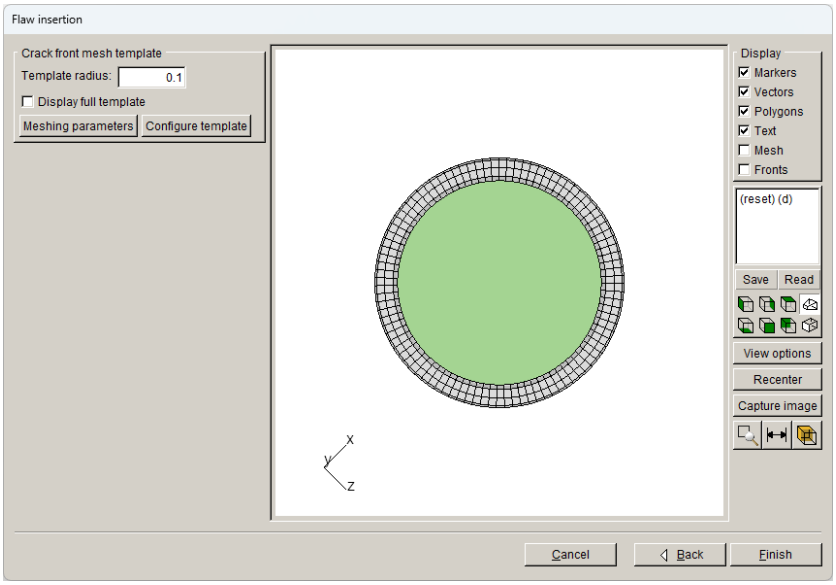


Figure 6.1.36 Crack front mesh template panel; *panel is not displayed for voids.*

6.1.25.1 Meshing Parameters Dialog

The **Meshing Parameters** dialog, Fig 6.1.37, has the same settings as the **Preferences** meshing tab described in Section 5.4.8. The user can control aspects of the surface and volume meshing. Section 7 of the User's Guide provides additional information.

The first parameter, Maximum Generated Elements, limits the total number of volume elements that FRANC3D will create during volume meshing. The second parameter, Maximum Volume Mesh Restarts, limits the number of volume meshing restarts. These two options only work when FRANC3D is used for volume meshing. FRANC3D uses an *advancing front* volume meshing algorithm; this algorithm can get stuck sometimes and the algorithm is designed to backtrack (*i.e.*, remove elements) and restart. If the program fails to create a volume mesh after several restarts, one might try volume meshing with ANSYS or ABAQUS.

The next two options allow the user to control surface mesh refinement: 1) on the crack surface, using the Do coarsen crack mouth mesh option, and 2) on the model surface in areas where adjacent region boundaries are near, using Do crack proximity refinement. The effect of the Do coarsen crack mouth mesh option is shown in Fig 6.1.38.

The Do not coarsen more than uncracked mesh option modifies the surface meshing algorithm to respect the original surface mesh density. Fig 6.1.39 shows an example where the option is turned off (left panel) and then turned on (right panel).

FRANC3D, by default, is used to mesh the volume, but ANSYS and ABAQUS can also be used by selecting the appropriate radio button for Volume mesh using… Note that the user must have a licensed working copy of these programs to use their meshing algorithms. The user can select

the ANSYS and ABAQUS executables, using the **Browse** button, if not already set in the **Preferences** (see Section 5.4).

For ANSYS or ABAQUS: write files only allows the user to write the surface mesh and the commands to generate the volume mesh from the surface mesh to files, without running ANSYS or ABAQUS. This gives the user the option of sending the files to a different computer or modifying the commands.

Fill inclusions allows the user to create inclusions instead of voids.

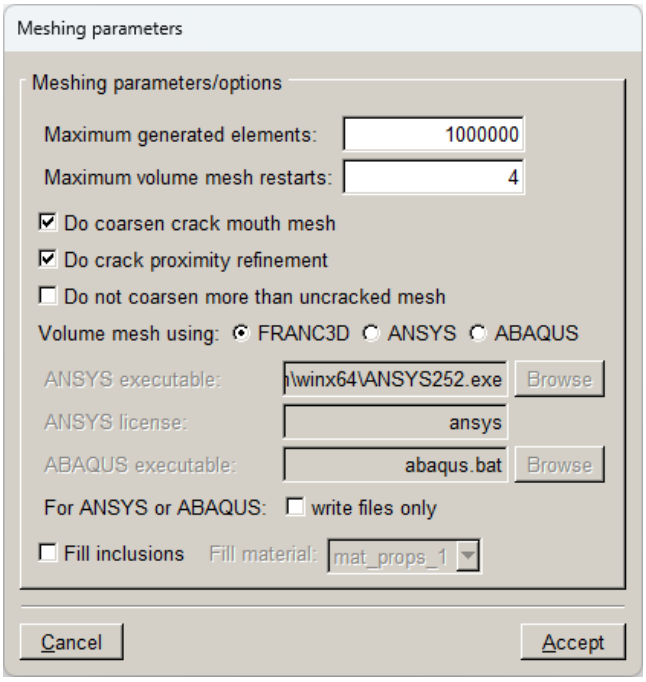


Figure 6.1.37 Meshing parameters dialog.

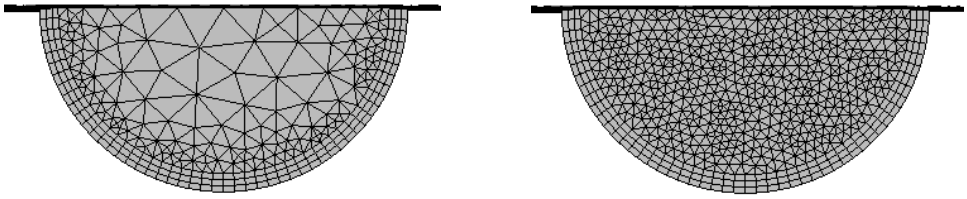


Figure 6.1.38 Effects of Do coarsen crack mouth option: on – left panel and off – right panel.

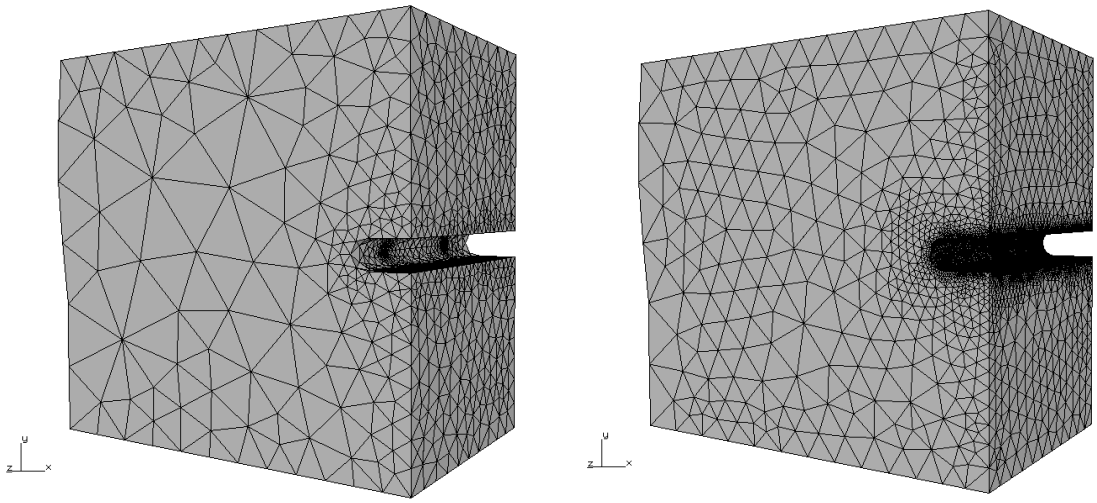


Figure 6.1.39 Effects of Do not coarsen more than uncracked mesh option: off – left panel and on – right panel.

6.1.25.2 Configure Template Dialog

The **Configure Template** dialog allows the user to adjust the default crack front mesh template parameters, Fig 6.1.40. The parameters are described below and illustrated in Fig 6.1.41, which shows a typical cross-section of a crack-front template. Section 7 of the User's Guide provides additional information.

Number of radial rings sets the number of rings of elements in the template.

Number of elements circumferentially sets the number of elements in the circumferential direction around the crack front.

Radia progression ratio sets the relative width of the element (in the direction perpendicular to the crack front) going from the crack front to the outer surface of the template. For example, a ratio of 1 means that all the rings of elements will have the same width. For a ratio of 1.5, the width of the elements in a ring will be 50% greater than the width of the elements in the next ring as we approach the crack front.

Max axial/radial aspect ratio controls the aspect ratio of the quadrilateral faces on the outer surface of the template that will trigger 1:2 or 1:3 transitions in the outer ring of elements.

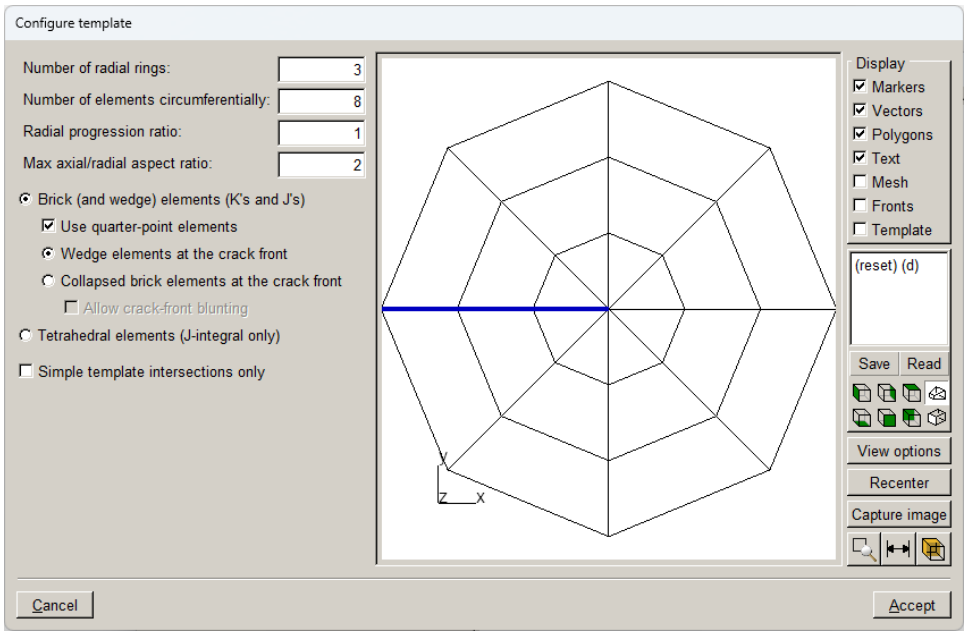


Figure 6.1.40 Configure crack front template dialog.

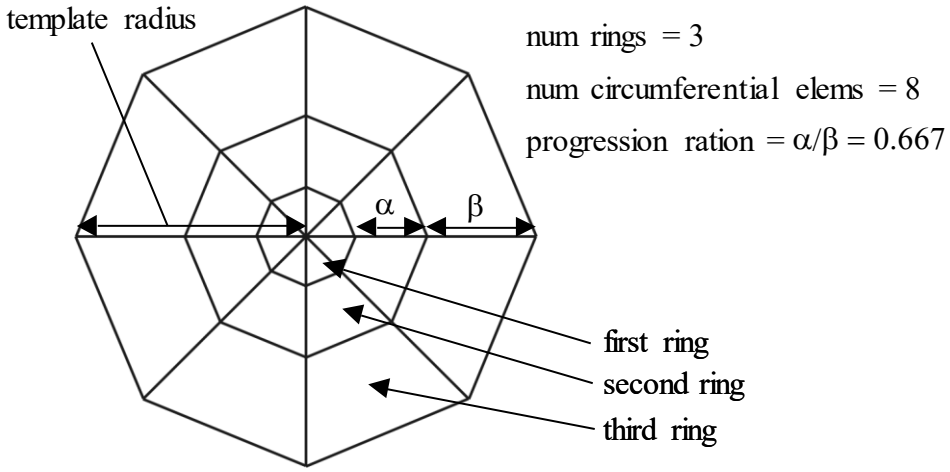


Figure 6.1.41 Crack front template cross-section.

Brick (and wedge) elements (K's and J's) sets the template element type to wedge surrounded by brick elements.

Use quarter-point elements sets the node position for the crack front elements; this applies to wedge and constrained-brick elements. If this option is un-checked, a mid-side node location is used.

Wedge elements at the crack front sets the crack front element as wedge.

Collapsed brick elements at the crack front sets the crack front element as brick.

Allow crack front blunting turns off the constrained nodes for crack front brick elements.

Tetrahedral elements (J-integral only) sets the template element type to all-tetrahedral elements.

Simple template intersections only toggle allows the user to terminate the template inside the model surfaces. For cases where the template will intersect the model surface at shallow angles or at corners, Fig 6.1.42, this option allows a crack to be inserted with template elements along the bulk of the crack front, Fig 6.1.43. FRANC3D attempts to recognize instances where the template will intersect model surfaces at poor angles and automatically uses this option as needed.

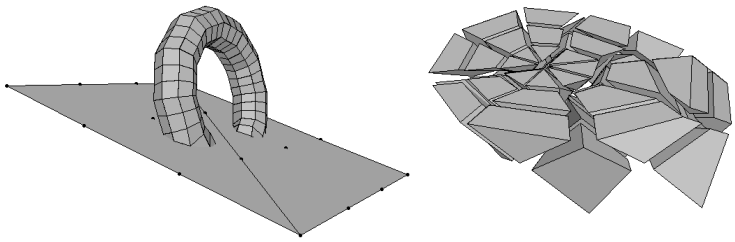


Figure 6.1.42 Crack front mesh template intersecting model surface at an angle.

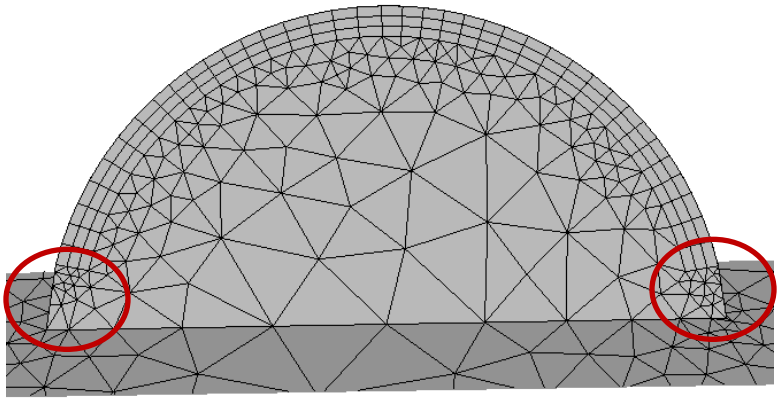


Figure 6.1.43 Crack front mesh template pulled back from the model surface.

6.1.25.3 Configure Notch Front Template Dialog

The **Configure Template** dialog for notches is shown in Fig 6.1.44; the parameters are described below.

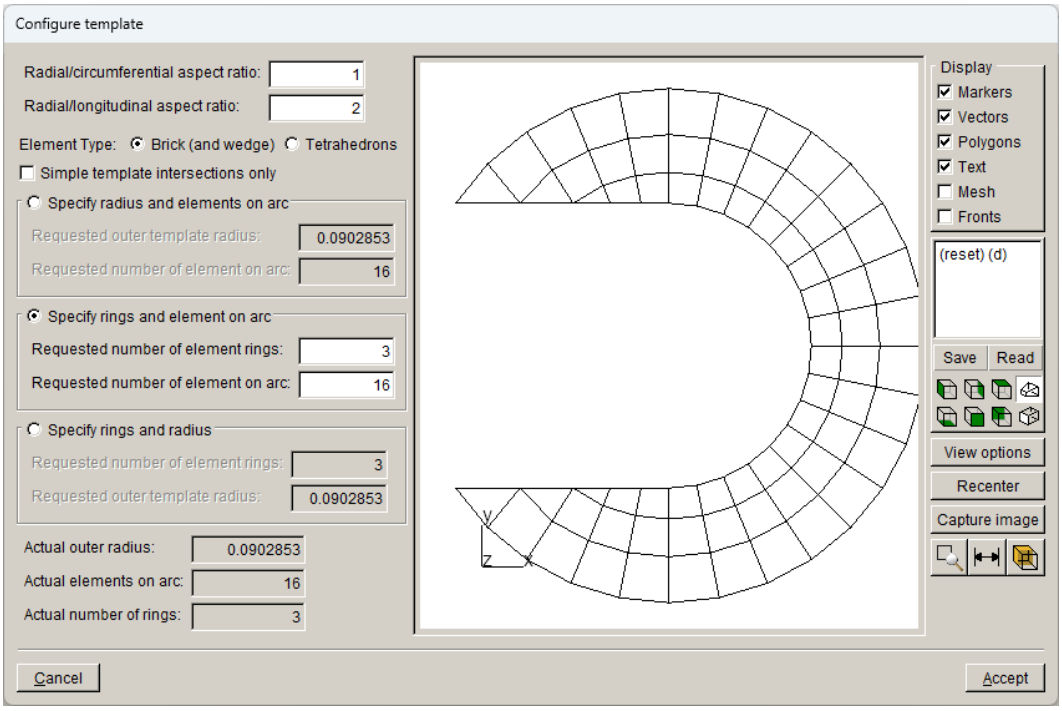


Figure 6.1.44 Configure notch front template dialog.

Radial/circumferential aspect ratio controls the aspect ratio.
Radial/longitudinal aspect ratio controls the aspect ratio.

Element type:

Brick (and wedge) sets the element type to brick and wedge.

Tetrahedrons sets the element type to all-tet.

Simple template intersections only toggle allows the user to terminate the template inside the model surfaces.

Specify radius and elements on arc allows the user to set the desired outer template radius and the number of elements on the arc (the circumferential dimension).

Requested outer template radius …

Requested number of elements on arc …

Specify rings and elements on arc allows the user to set the desired number of element rings and the number of elements on the arc (the circumferential dimension).

Requested number of element rings …

Requested number of elements on arc …

Specify rings and radius allows the user to set the desired number of element rings and the outer template radius.

Requested number of element rings …

Requested outer template radius …

The actual values might differ slightly from the requested values depending on the notch dimensions and the aspect ratio values. The dialog provides the actual values of template outer radius and number of elements.

Actual outer radius: the actual outer radius.
Actual elements on arc: the actual number of circumferential elements.
Actual number of rings: the actual number of rings.

6.1.26 Flaw Insertion

After selecting **Finish** on the crack front mesh template panel (see Fig 6.1.36), the flaw is inserted into the body and the model is remeshed. An information box is displayed to give the status of operations, Fig 6.1.45. The crack geometry is inserted into the model geometry first, represented by the Doing geometric intersections... status. Once the crack geometry has been inserted, trimmed, and tied to the model geometry, surface and then volume meshing occurs. If using the FRANC3D volume meshing, the final volume mesh is smoothed to improve the element quality.

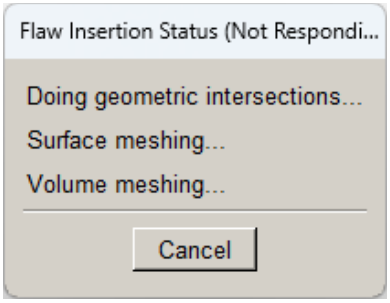


Figure 6.1.45 Flaw Insertion Status dialog.

6.2 Flaw from Files

The user has the option of saving the flaw to a *.crk* file when defining a new flaw (see Fig 6.1.2). The **Flaw from Files** menu option allows the user to read one or more *.crk* files and insert the crack(s) into the model. The **Locate .crk flaw file** dialog, Fig 6.2.1, is displayed, which allows the user to select one or more *.crk* files. Use Ctrl- or Shift-key to select multiple files. After selecting **Accept**, the flaw is displayed within the model, Fig 6.2.2. A flaw can be translated, but rotation is not allowed.

Select **Next** to define the crack front template mesh; this was described in Section 6.1.23.

The *.crk* file contains triangular Bezier patches that define the surface and a description of the crack front vertices. It also contains crack front template meshing parameters. This data is also stored in the FRANC3D restart *.fdb* file. As the crack grows, new geometry is added, and this

information is saved in the *.fdb* file. If needed, the FLAWSURF block of the *.fdb* file can be extracted and saved as a *.crk* file (see Section 4.1.1).

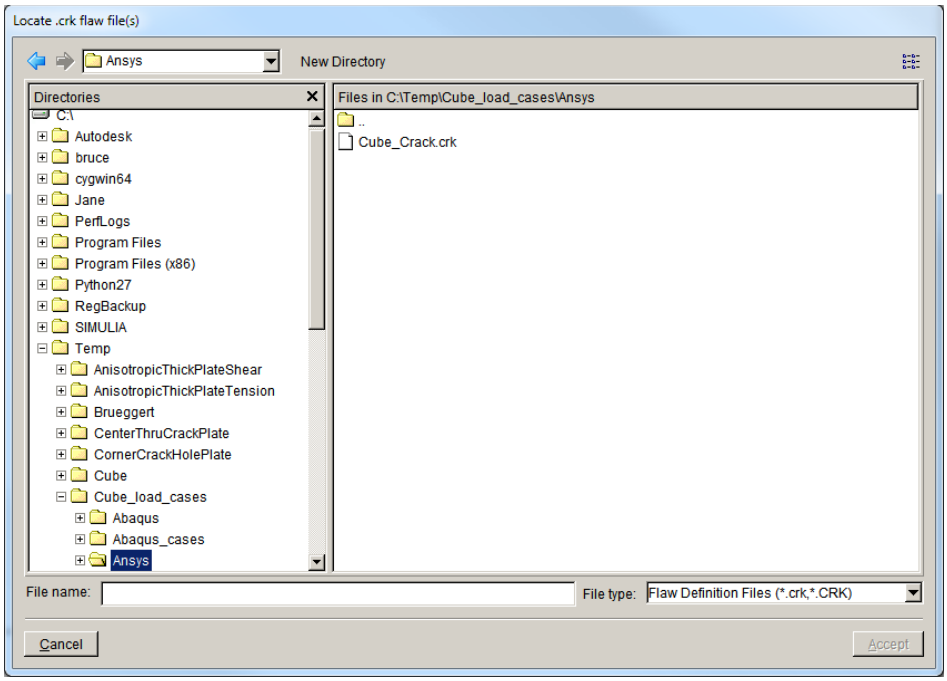


Figure 6.2.1 Locate flaw file dialog.

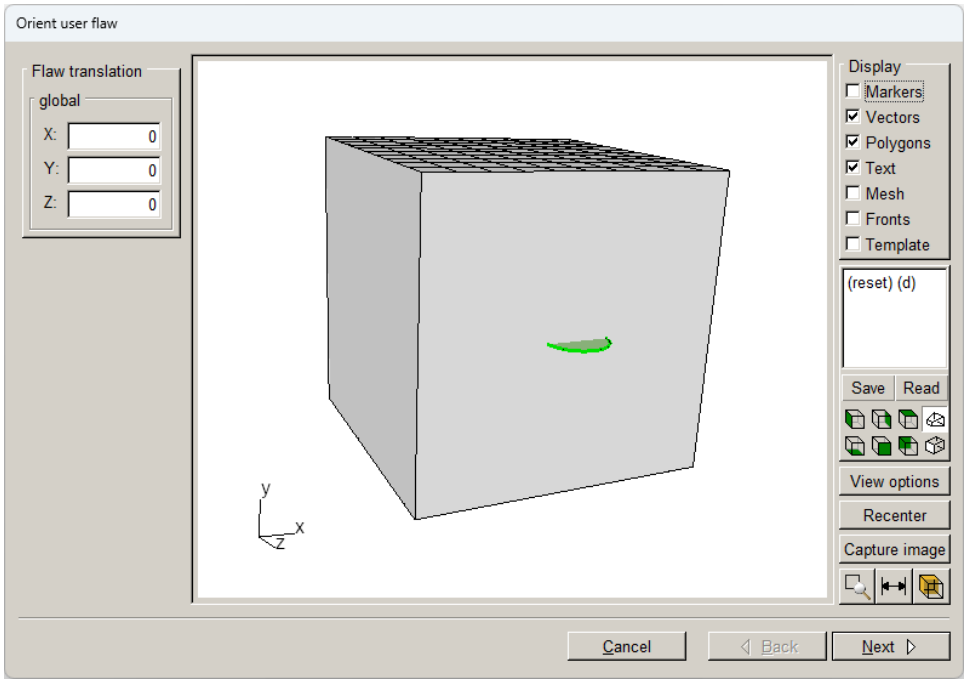


Figure 6.2.2 Orient a flaw-from-file dialog; only translation is allowed.

6.3 Multiple Flaw Insertion

The **Multiple Flaw Insertion** wizard, Fig 6.3.1, allows one to add multiple cracks to a model. The user defines each crack using the **Flaw Insertion** wizard panels, described in Section 6.1. When all cracks have been defined, they can be added to the model and/or saved to a file.

The user starts by selecting the **Add** button in the dialog shown in Fig 6.3.1. This leads to the flaw-insertion wizard panels. Either cracks or voids can be added.

Once a flaw has been added to the list, it can be edited or deleted by selecting the flaw name and then selecting the **Edit** or **Delete** button. All the added flaws can be displayed in the model, Fig 6.3.2, to ensure that they do not overlap or intersect; select the **Display** button to show this dialog. Select the **Finish** button to return to the Multiple Crack Definition dialog.

The set of radio buttons at the bottom of the dialog in Fig 6.3.1 allows one to: 1) add the flaws to the model without saving to a file, 2) add the flaws to the model after prompting for a file name to save a *.crk* file, or 3) save the flaws to a file without adding them to the model.

If the flaws are just being added, select **Accept** to close the dialog and begin the process of crack insertion and re-meshing. The flaw insertion status window as described in Section 6.1.24 is displayed. The Save File dialog is presented before the crack insertion process starts if the user wants to save and add.

Note that mixing of flaw types is not currently supported; for example, you cannot add a symmetry crack and a regular crack at the same time. Multiple symmetry cracks can be inserted if the cracks fall on the same surface; multiple symmetry cracks on multiple surfaces will not have the correct boundary conditions mapped onto the new mesh.

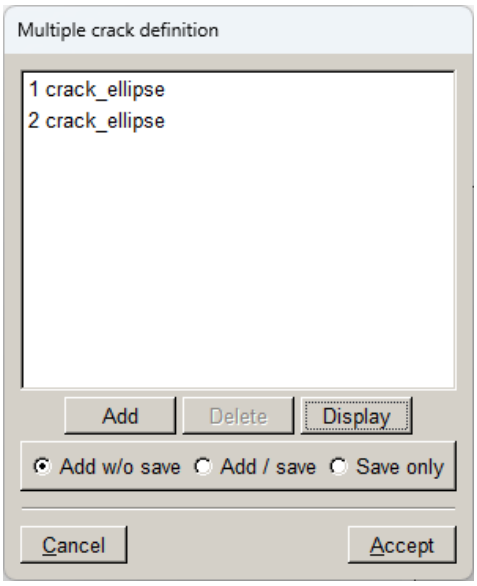


Figure 6.3.1 Multiple flaw insertion - top level dialog.

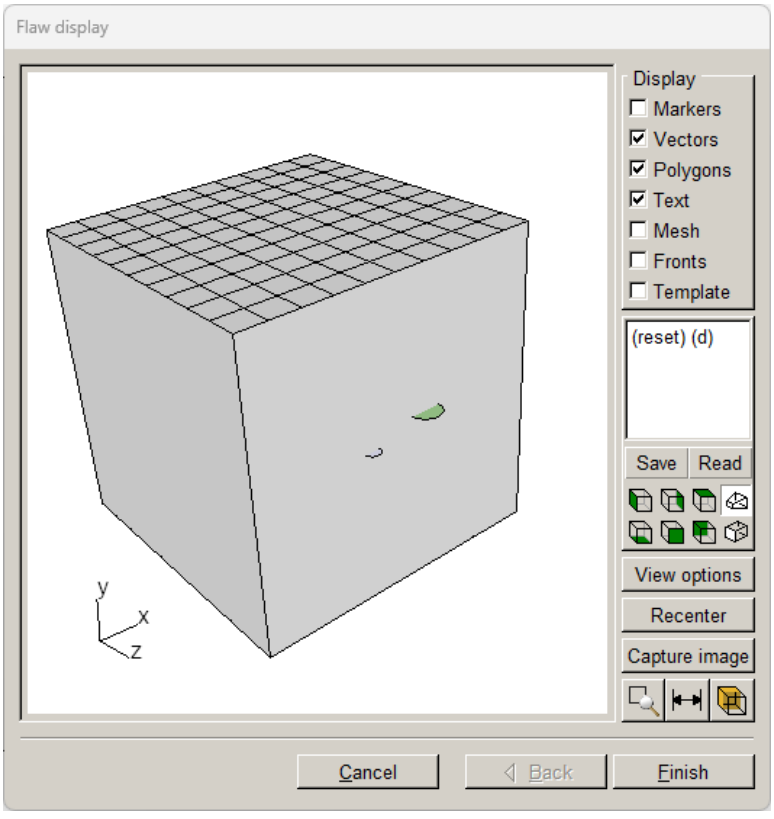


Figure 6.3.2 Multiple flaw insertion – flaw display window.

6.4 Compute SIFs

The user can compute and plot stress intensity factors (SIFs) after performing an analysis of the cracked model; the **Analysis** menu is described in Section 8.

SIFs are computed at mid-side nodes along the crack front.

6.4.1 M-integral, Displacement Correlation or Virtual Crack Closure Panel

The **Compute SIFs** menu option brings up the Compute SIFs dialog box, Fig 6.4.1. Either the M-integral, Displacement Correlation (DC) or Virtual Crack Closure (VCCT) method can be chosen to compute SIFs. The M-integral method[1] is usually the most accurate. The DC option can be used to check the M-integral values. The VCCT method was added primarily for cracks in material interfaces.

Note that if you switch SIF computation method, that method will continue to be used until you switch back. If you use the DC method to check the SIFs, you should recompute SIFs with the M-integral before growing the crack – if you want to use the M-integral SIFs.

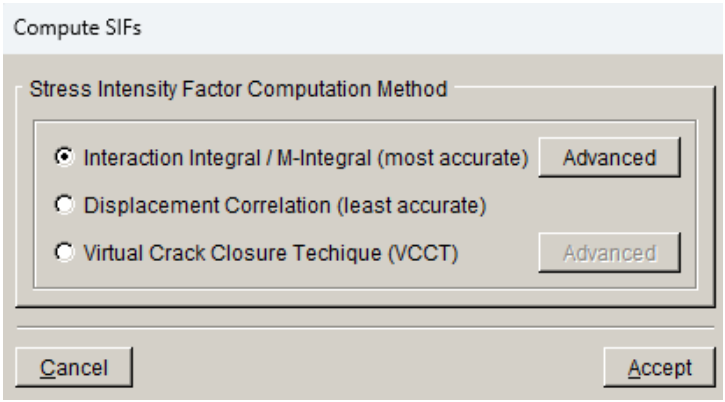


Figure 6.4.1 Compute SIFs panel allows one to choose either M-integral, DC, or VCCT.

The M-integral **Advanced** button displays the dialog in Fig 6.4.2 – left side. The VCCT **Advanced** button displays the dialog shown on the right side of Fig 6.4.2. The fields in the dialog are described below.

[1] Banks-Sills *et al*, EFM 74, p 1293-1307.

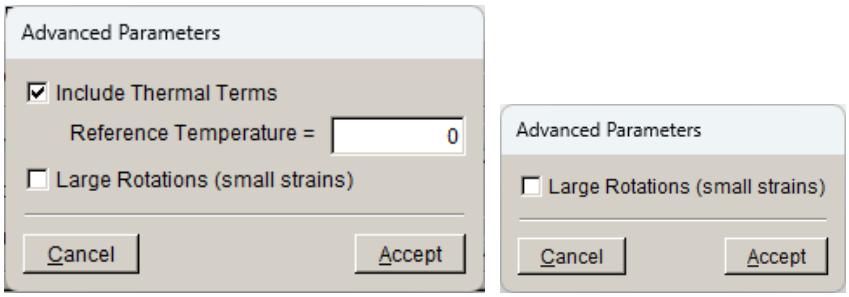


Figure 6.4.2 Compute SIFs M-Integral and VCCT Advanced Parameters dialogs.

Include Thermal Terms: Thermal terms can be included in the M-integral computation. The nodal temperatures should exist in the results database, and the user should supply the reference temperature if not already given with the initial FE input file. Note that these options should be set automatically if there are temperatures in the initial FE model.

Large Rotations: The local crack front coordinate system is defined based on the deformed configuration if this option is checked; this is required when there are large rigid body rotations. This option is not set automatically.

6.4.2 SIF Plot Panel

Stress-intensity factor distributions are displayed in this dialog, Fig 6.4.3. The left side of the dialog is a 3D graphics window that displays the crack in model. The right side of the dialog is a tab box that shows graphs of the computed stress-intensity factor (SIF) distributions along the crack front. The mode I, II, and III SIFs are represented by the KI, KII and KIII tabs, respectively.

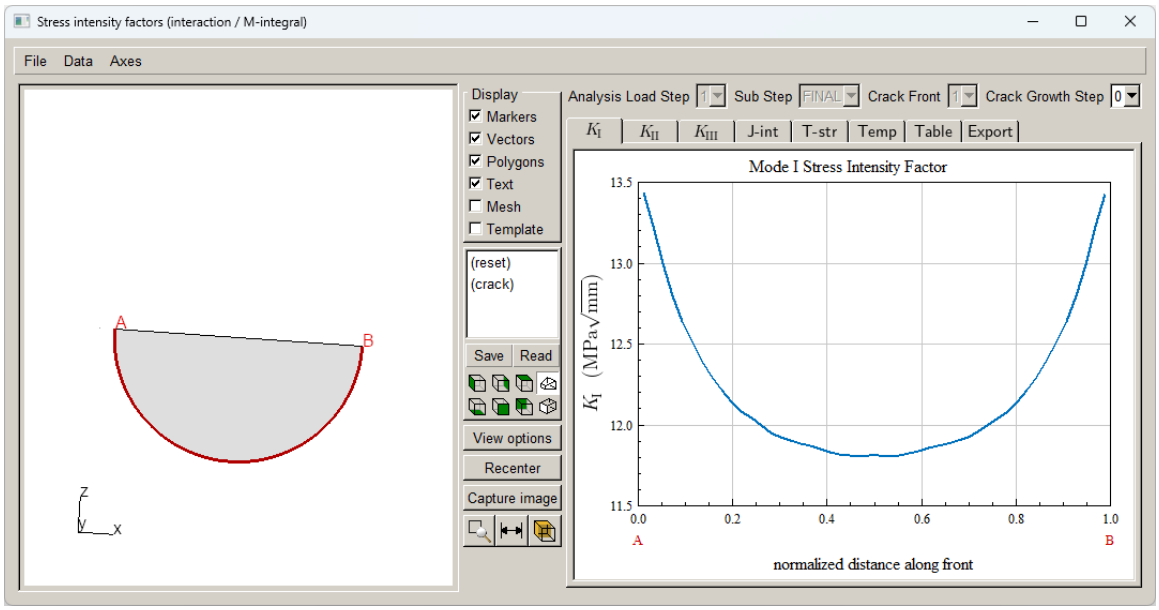


Figure 6.4.3 SIF plot panel.

The J-integral tab displays a plot of the total (elastic) *J*-integral distribution (total energy release rate not segregated into modal components). *J*-integral values from M-integral and Displacement Correlation will be slightly different as the computations are done differently. The T-Str tab displays a plot of the T-stress. The Temp tab shows the temperature. The Table tab shows a table of the SIF values along with the parametric location along the crack front (N Coord); the crack front length is normalized from 0 to 1, going from A to B (shown in the window on the left).

The Export tab allows one to export the SIF data to a file, Fig 6.4.4. The user can specify the type of delimiters to use, the grouping of the data, and order of the data. The program initially assigns one end of the crack front as A and the other end as B. The data is plotted along the crack front, normalized by the crack front length, starting from point A. Use the **Write file** button to set the file name and save the data. A WriteSif command is issued to the session log; see Section 2.1.68 in the Commands & Python reference.

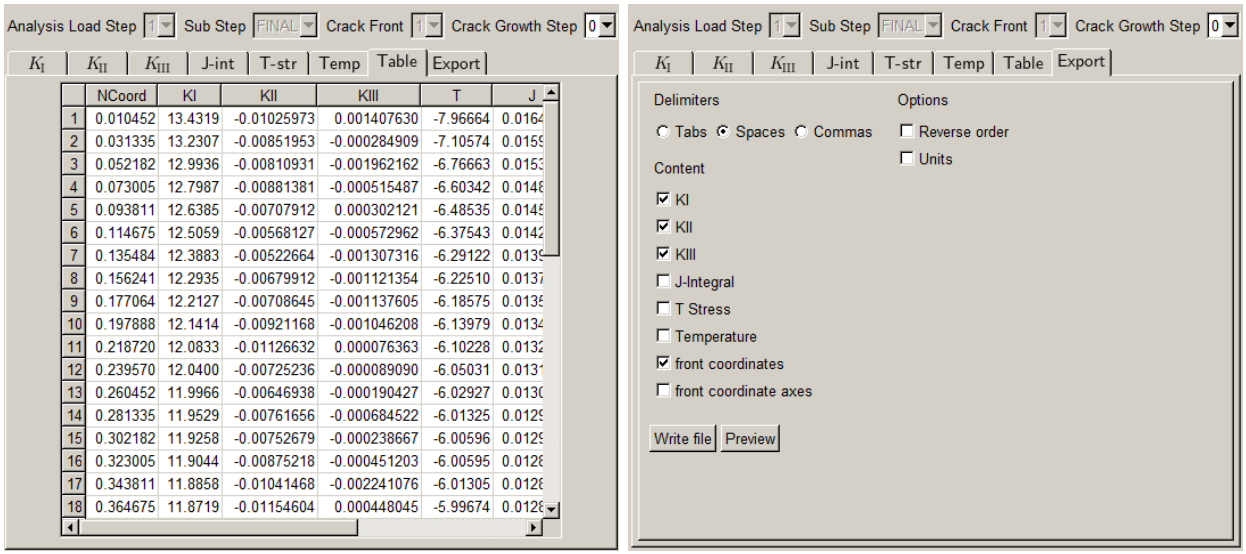


Figure 6.4.4 SIF table tab (left panel) and export tab (right panel).

The **Preview** button displays a table of the data with options to set the precision and delimiters in the file, Fig 6.4.5. Click on a column label to set the format for a single column. If a column is selected, the Default precision option becomes inactive. Columns of data can be copied to the clipboard using the **Copy to clipboard** button (or using the Ctrl-C keys). Use the **Write File** button to set the file name and save the data.

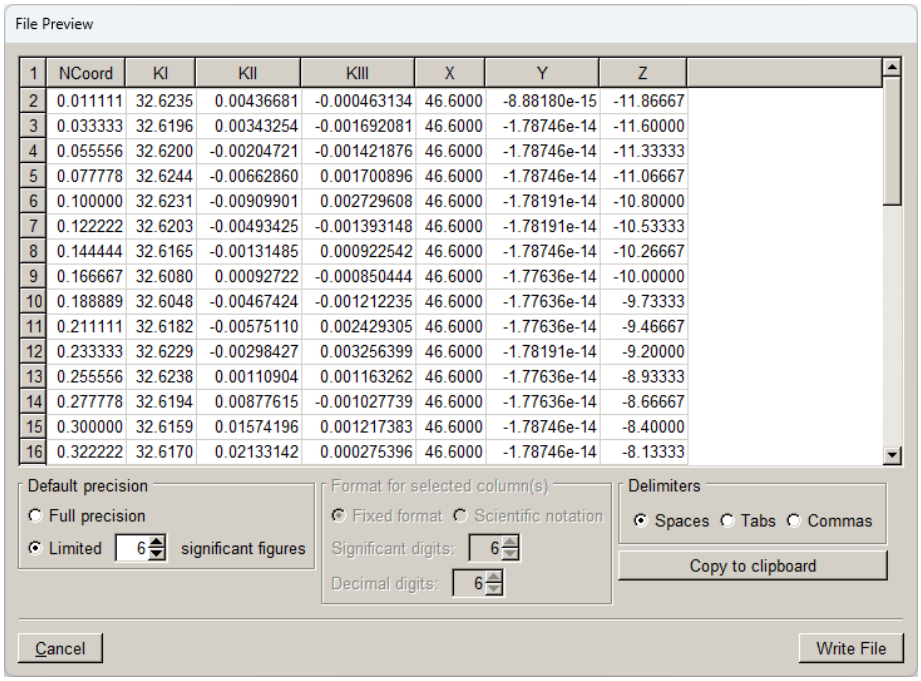


Figure 6.4.5 File preview dialog.

The dialog shown in Fig 6.4.3 includes options to display SIF data for a model with multiple load steps (and substeps), multiple crack fronts, and multiple crack growth steps. The drop-down boxes above the tabs provide these options, Fig 6.4.6.

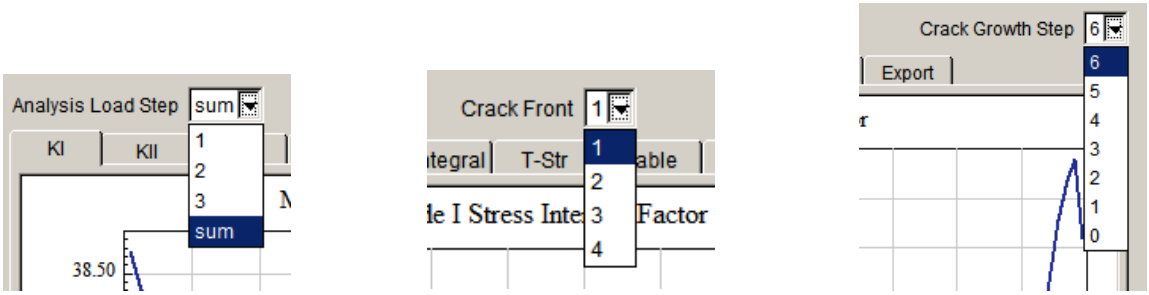


Figure 6.4.6 SIF display drop-down selections for load step, crack front, and crack growth step.

6.4.2.1 SIF Plot menu bar

The menu bar at the top left contains three entries: **File**, **Data** and **Axes**. These are described here. The same menu appears in other dialog panels with XY plots.

The **File** menu is shown in Fig 6.4.7. The **Export as .png…** and **Export as .jpg…** options both present the standard FRANC3D **File Save As** dialog (see Section 4.4). One can save either *.png* or *.jpg* images of the XY plot. The **Close** option closes the SIF display.

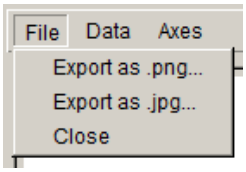


Figure 6.4.7 SIF display panel File menu.

The **Data** menu is shown in Fig 6.4.8. The **Swap A-B** option swaps the X-axis and data in the XY plot, Fig 6.4.9.

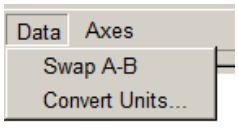


Figure 6.4.8 SIF display panel Data menu.

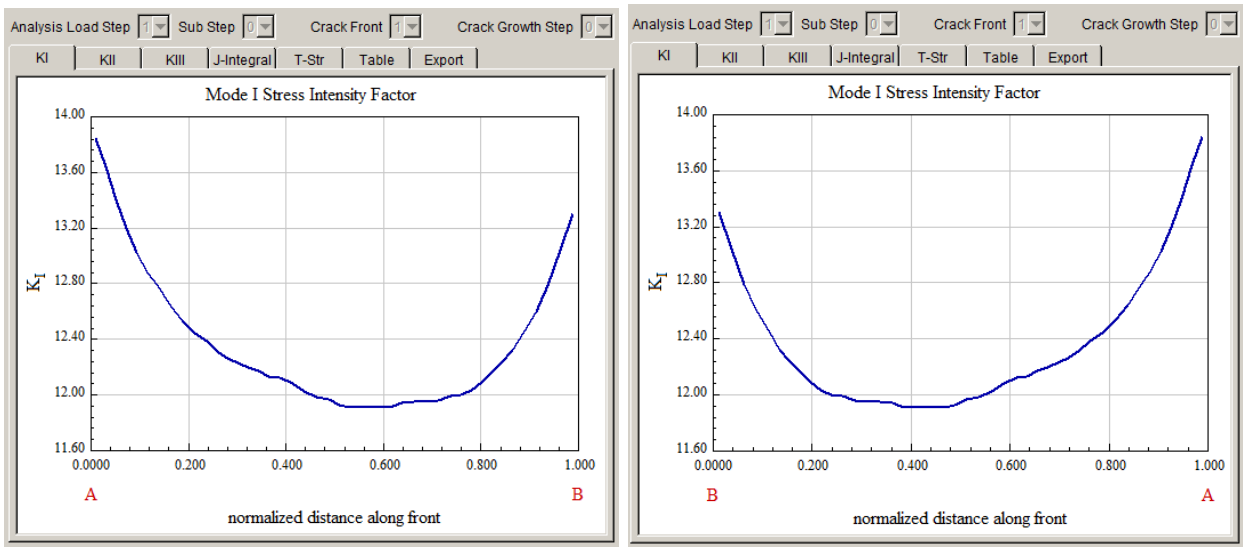


Figure 6.4.9 Effect of the Swap A-B menu option.

Convert Units allows one to change the units for the XY plot. The dialog, Fig 6.4.10, shows the FE units and the current display units; the display units can be changed.

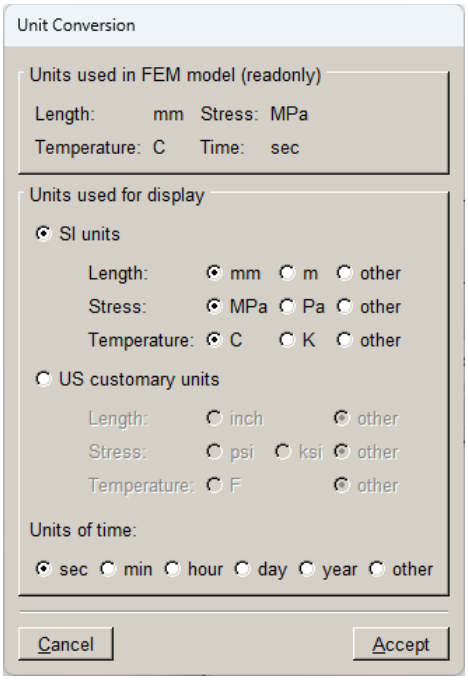


Figure 6.4.10 Units dialog.

The **Axes** menu is shown in Fig 6.4.11. The **Format X Axis** option displays the dialog shown in Fig 6.4.12. One can define the axis limits and grid spacing. The **Format Y Axis** displays the

same dialog with the label changed to Y Axis. The **Labels…** option displays the dialog in Fig 6.4.13. One can change the plot title and axis labels; the syntax for defining the labels is described in Appendix A.

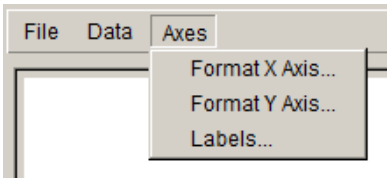


Figure 6.4.11 SIF display panel Axes menu.

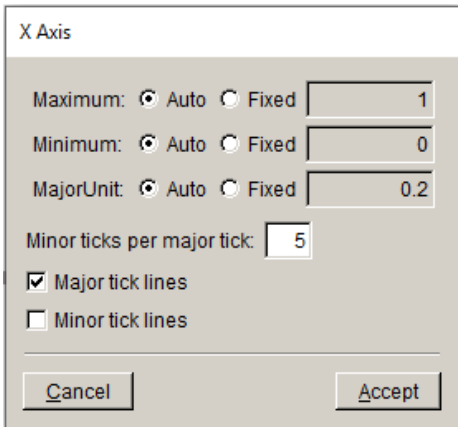


Figure 6.4.12 Format X Axis dialog.

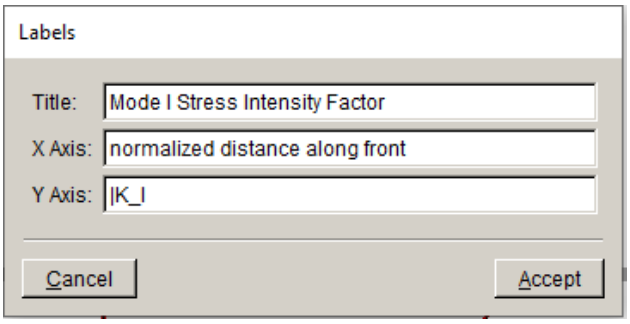
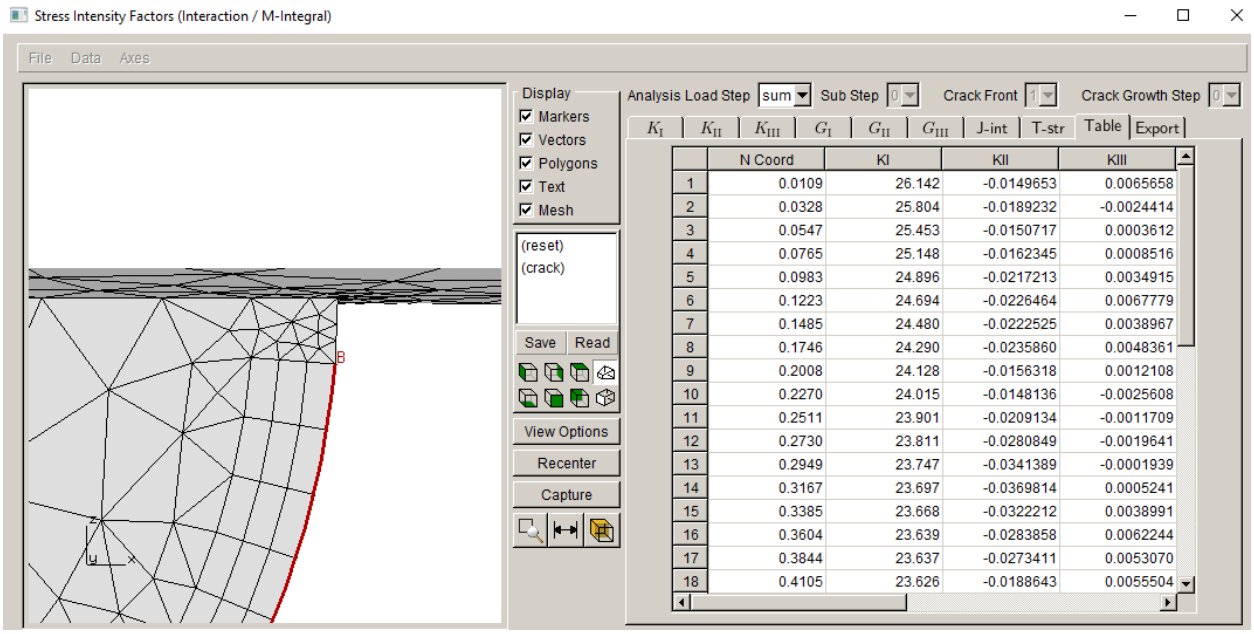


Figure 6.4.13 XY plot Labels dialog.

6.4.2.2 SIF Plot with Simple Template Intersections

If Simple Template Intersections (see Section 6.1.25) is turned on, the template will not extend to the model surface. The SIF plot is based on the template element nodes; thus, A and B represent the ends of the template and not the ends of the geometric crack front, Fig 6.4.14. The

normalized crack front position (N Coord) is also based on the template; A and B always represent N Coord values of 0 and 1. SIFs are computed at the element mid-side nodes. In Fig 6.4.14, the first N Coord position is 0.0109 (and the last is 0.9869). This normalized position is based on the cumulative template length between A and B.



	N Coord	KI	KII	KIII
1	0.0109	26.142	-0.0149653	0.0065658
2	0.0328	25.804	-0.0189232	-0.0024414
3	0.0547	25.453	-0.0150717	0.0003612
4	0.0765	25.148	-0.0162345	0.0008516
5	0.0983	24.896	-0.0217213	0.0034915
6	0.1223	24.694	-0.0226464	0.0067779
7	0.1485	24.480	-0.0222525	0.0038967
8	0.1746	24.290	-0.0235860	0.0048361
9	0.2008	24.128	-0.0156318	0.0012108
10	0.2270	24.015	-0.0148136	-0.0025608
11	0.2511	23.901	-0.0209134	-0.0011709
12	0.2730	23.811	-0.0280849	-0.0019641
13	0.2949	23.747	-0.0341389	-0.0001939
14	0.3167	23.697	-0.0369814	0.0005241
15	0.3385	23.668	-0.0322212	0.0038991
16	0.3604	23.639	-0.0283858	0.0062244
17	0.3844	23.637	-0.0273411	0.0053070
18	0.4105	23.626	-0.0188643	0.0055504

Figure 6.4.14 SIFs when Simple Intersection Template is turned on.

6.5 Grow Crack

When the **Grow Crack** menu option is selected, the **Compute SIF's** wizard is displayed first (see Section 6.4) if the SIFs have not already been computed.

The **Crack Growth** wizard allows the user to specify algorithms and parameters to determine the local direction and relative extension of crack growth.

The first panel in the wizard, Fig 6.5.1, allows one to specify the growth model type. Subcritical crack growth includes fatigue, stress corrosion, and creep models. Quasi-static crack growth is a simplified model that uses a simple power-law to advance the crack front. User-defined crack growth is active if the user has pre-defined Python extensions; Python extensions are described under the **Advanced** menu (see section 14.7). Growth parameters can also be read from a file using the **Browse** button.

Depending on the chosen growth type, different wizard panels will be displayed, which allow the user to choose the rules, models, and parameters for the crack extension and kink angle computations.

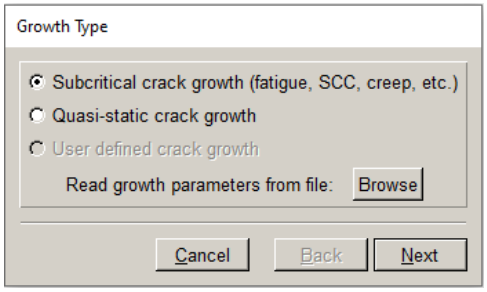


Figure 6.5.1. Growth parameters panel.

6.5.1 Subcritical crack growth

Subcritical crack growth includes computation of both kink angle and extension along the crack front. Cycles or time or a combination of the two are related to the amount of extension. For each step of crack growth, there will be an average number of cycles and/or time stored, which will be used/displayed in the Fatigue Life dialog (see Section 9).

6.5.1.1 Kink Angle Model

The first panel of Subcritical crack growth is shown in Fig 6.5.2. This dialog allows one to specify the method for computing kink angle.

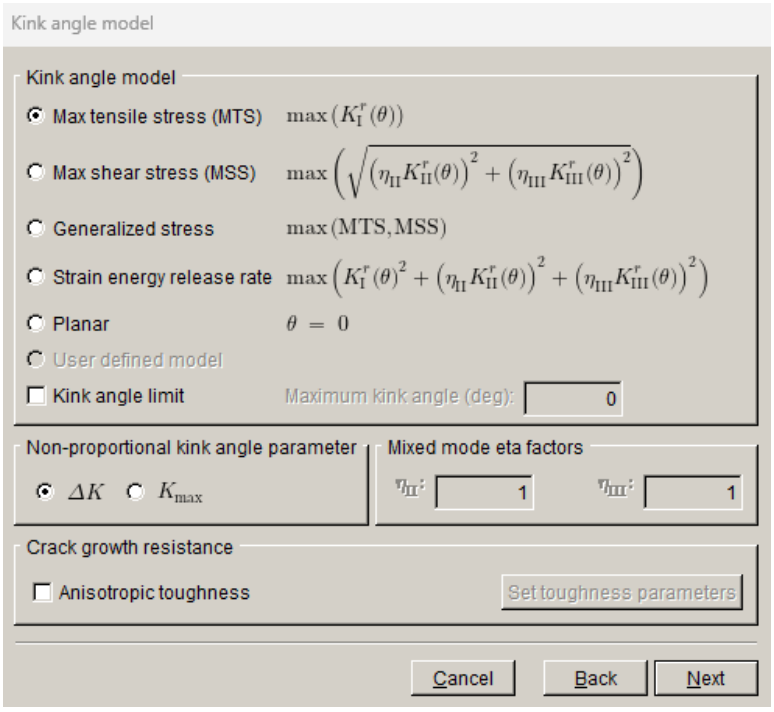
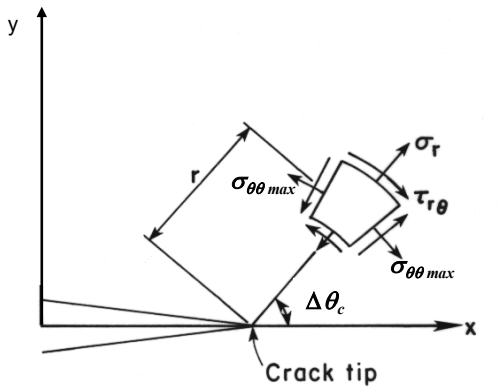


Figure 6.5.2 Kink angle model panel.

6.5.1.1.1 Max Tensile Stress (MTS)

The default for computing the kink angle is the maximum tensile stress (MTS) theory. The crack kinks in the direction where tensile stress ahead of the crack front is maximized, Fig 6.5.3. At each node along the crack front, the computed SIFs are used to determine the kink angle θ:

$$\theta = cos^{-1}\left(\frac{3K_{II}^2 + \sqrt{K_I^4 + 8K_I^2K_{II}^2}}{K_I^2 + 9K_{II}^2}\right)$$



$$\sigma_{\theta\theta} = \frac{1}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left[K_I\cos^2\frac{\theta}{2} - \frac{3}{2}K_{II}\sin\theta\right]$$

Figure 6.5.3 MTS kink angle theory.

6.5.1.1.2 Max Shear Stress (MSS)

The MSS option is used when mode II SIF dominates crack growth and the material allows shear-dominated growth. The user can refer to: Pettit *et al*, Next generation 3D mixed mode fracture propagation theory including HCF–LCF interaction, EFM 102 (2013) 1–14.

6.5.1.1.3 Generalized Stress

The Generalized Stress option determines the kink angle based on the maximum of MTS and MSS. It should be used with care as it can lead to large kinks in the crack front that can be difficult to insert/mesh.

6.5.1.1.4 Strain Energy Release Rate (SERR)

The SERR option computes a kink angle based on the growth direction that maximizes the strain energy release.

6.5.1.1.5 Planar

The Planar option sets the kink angle to zero. If the user knows that the crack growth is planar, this option will negate any numerical noise in the computed SIFs and force the crack to remain planar. This is the only option available for symmetry-cracks.

6.5.1.1.6 User Defined Model

The User-Defined Model will be active for the user-defined Python extensions if the kink angle function is defined and selected; see Section 4 of the Commands & Python document.

6.5.1.1.7 Kink angle limit

The kink angle limit can be used to limit the amount of crack turning for a step of growth. For partial crack front extension and large kink angles, this option can significantly improve the crack growth insertion/remeshing.

6.5.1.1.8 Non-proportional Kink Angle Parameter

The kink angle can be computed based on ΔK or K_{max}. If the loading produces non-proportional cyclic loading and the K_{min} values produce negative K_I, the K_{max} option might be more appropriate.

6.5.1.1.9 Mixed Mode Eta Factors

The η factors apply to the MSS and SERR computations. The user must provide them.

6.5.1.1.10 Crack Growth Resistance

Crack growth resistance can be either isotropic or anisotropic. If the Anisotropic Toughness box is checked, the **Set Toughness Parameters** button is activated, Fig 6.5.4; all the material toughness values should be entered.

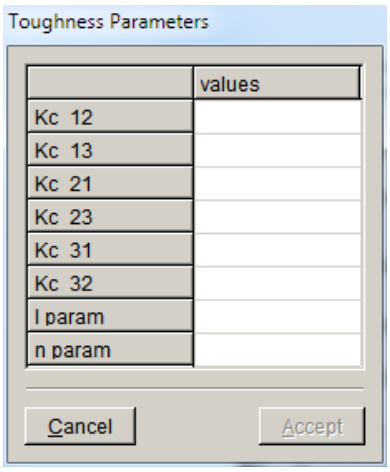


Figure 6.5.4 Anisotropic toughness parameters dialog.

Currently, FRANC3D supports one anisotropic crack growth resistance model. It is an orthotropic model based on six principal material toughness values. These values are illustrated in Fig 6.5.5. The *e*'s are the material property axes, which, in general, will not be aligned with the global Cartesian or the crack front coordinate systems. The first toughness subscript identifies the material axis perpendicular to the crack plane and the second subscript gives the direction of propagation.

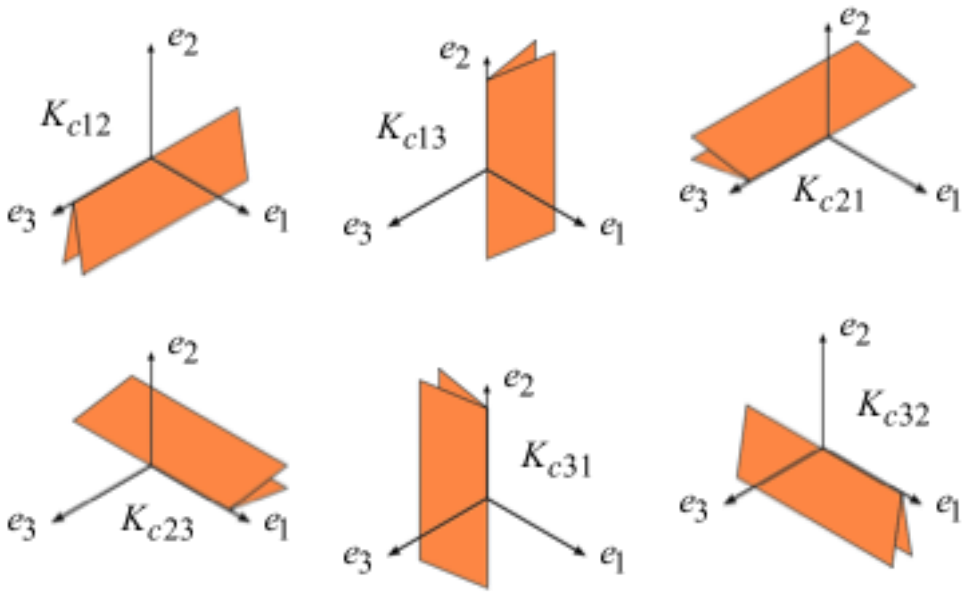


Figure 6.5.5 Orientations for the six principal material toughness values.

Fig 6.5.6 shows the crack-front orientation and defines the **a** and **n** vectors. x, y, and z are local crack-front coordinate axes that, in general, are not aligned with the global Cartesian coordinate axes. θ is the local kink angle.

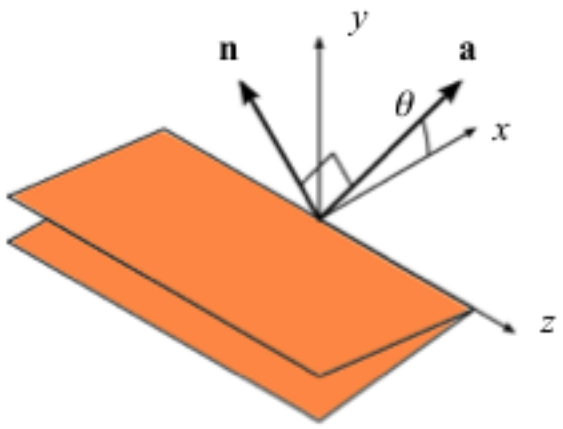


Figure 6.5.6 Definition of the crack-front **a** and **n** vectors.

The crack orientation and kink-angle-sensitive local resistance to crack growth $K_p(\theta)$ is given by

$$K_p(\mathbf{n},\mathbf{a}) = \left[\left(\frac{n_1^2}{K_1^2}\right)^l + \left(\frac{n_2^2}{K_2^2}\right)^l + \left(\frac{n_3^2}{K_3^2}\right)^l\right]^{-1/2l}$$

$$K_i(\mathbf{a}) = \sqrt{1-a_i^2}\left[\left(\frac{a_j^2}{K_{ij}^2}\right)^n + \left(\frac{a_k^2}{K_{ik}^2}\right)^n\right]^{-1/2n}$$

where K_{ij} and K_{ik} are the principal toughness values (no summation on repeated indices), a_i and n_i are the Cartesian components of the **a** and **n** vectors, and l and n (no subscript) are shape parameters. The n shape parameters control the shape of the toughness envelope in the principal planes. For example, Fig 6.5.7 shows the variation in K_3 as function of θ assuming that crack front coordinates are aligned with the Cartesian coordinates. The l shape parameter has a similar effect on the variation of the toughness between the principal planes.

6.5.1.2 Subcritical Growth Units

Selecting **Next** in the dialog seen in Fig 6.5.2 leads to the Subcritical Growth Parameters dialog, Fig 6.5.8. Note that yellow (warning) triangles indicate parameters need to be set before you continue.

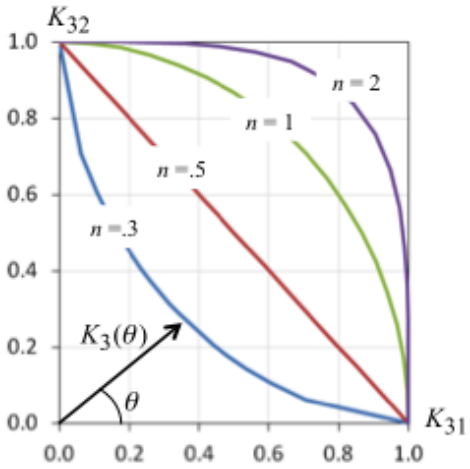


Figure 6.5.7 The variation in K_3 as function of θ assuming that crack front coordinates are aligned with the Cartesian coordinates.

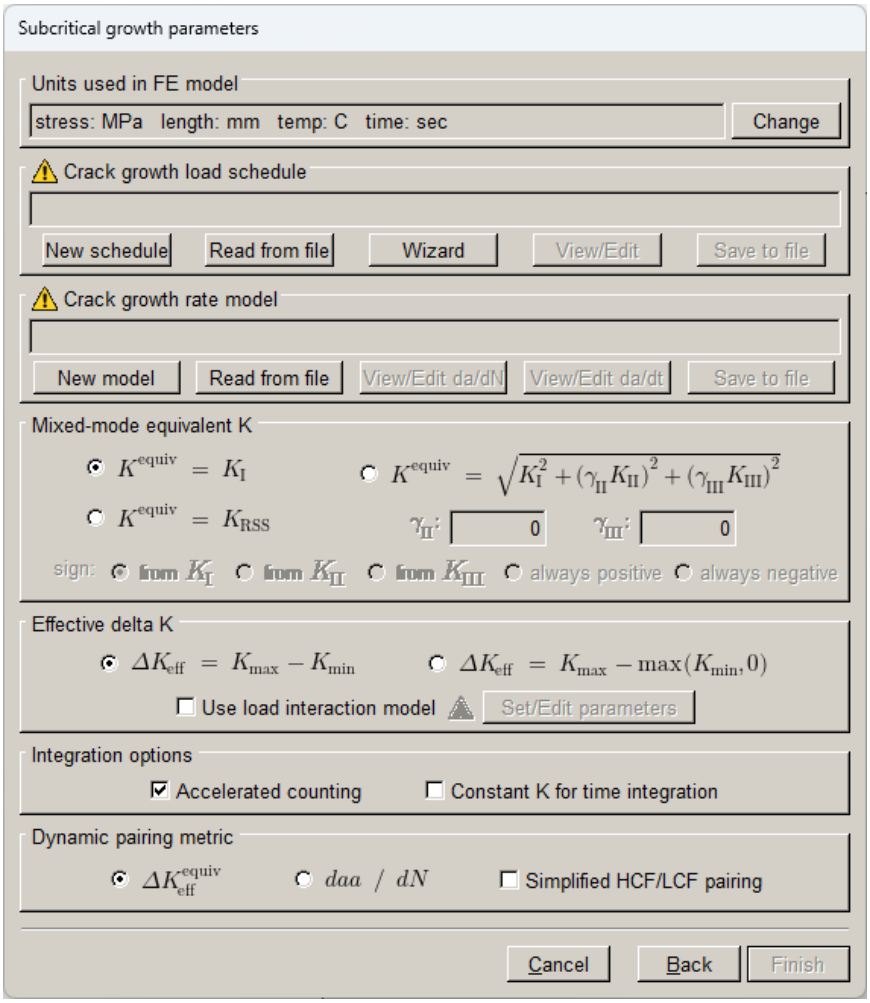


Figure 6.5.8 The Subcritical Growth Parameters dialog.

The Units used in the FE model should have already been set, but this dialog displays the units and allows the user to change them if needed.

Select the **Change** button on the right side of the Units used in the FE model to display the Units dialog box, Fig 6.5.9. Either SI or US units can be chosen; if the FE units do not correspond to the units shown, choose "other". Units of time are independent.

Note that FRANC3D needs to match the FE units with crack growth rate units, which can be different. By explicitly specifying the units used for the FE model and for the growth model, FRANC3D can perform unit conversions automatically.

If other units are used, it is up to the user to make the units consistent between the FE model and the crack growth rate model.

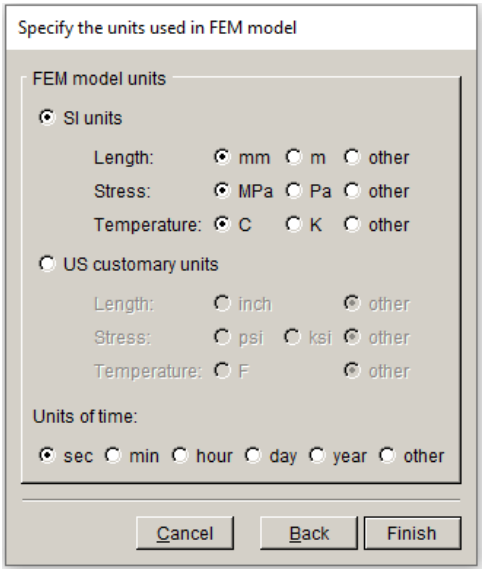


Figure 6.5.9. The Units dialog.

6.5.1.3 Subcritical Load Schedule

A Crack Growth Load Schedule (see Fig 6.5.8) does two main things. First, it defines a mapping from the load steps defined in the FE model to the corresponding SIFs to predict crack growth rates. Second, it defines the sequence these SIFs are applied.

6.5.1.3.1 Load Schedule: New Schedule

Select the **New Schedule** button (see Fig 6.5.8) to display the Load Schedule dialog, Fig 6.5.10. A load schedule is comprised of a collection of load events. Load events are organized in a tree-like structure. The root of the tree is a Schedule event.

Initially, the upper part of the dialog (Fig 6.5.10) shows one Schedule event, which is the main root of the tree. A left click on this event does two things: 1) it makes the **Add** button active, and 2) it raises the Repeat tab in the lower part of the dialog (right panel). In practice, we typically set the top-level schedule to repeat forever.

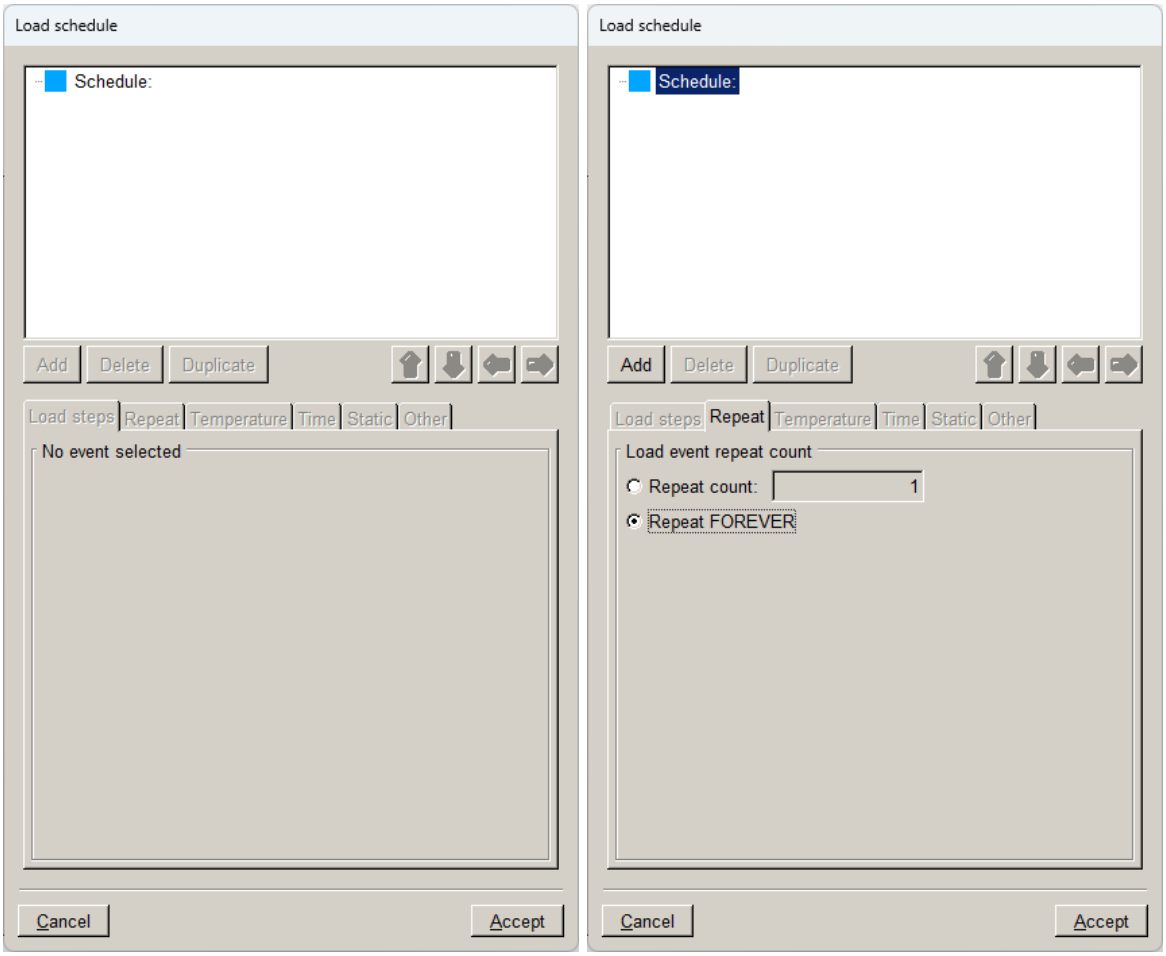


Figure 6.5.10 The Load Schedule dialog.

Select the **Add** button to display the Event Type dialog, Fig 6.5.11. There are seven different types of event:

- Simple Cyclic – a simple $\Delta K = K_{max} - K_{min}$ cycle, where the K_{max} and K_{min} loadings are proportional, so $K_{min} = R^*K_{max}$, and R is given by the analyst.
- Non-Proportional Cyclic – a $\Delta K = K_{max} - K_{min}$ cycle, where the K_{max} and K_{min} loadings are not proportional (they come from different load steps), and $R = K_{min}/ K_{max}$ is computed.

- Transient – For a transient event, a list of analysis load steps, sub-steps (frames), or a combination are specified. FRANC3D will examine all the associated *K*'s and select K_{max} and *Kmin* from among these to define $\Delta K = K_{max} - K_{min}$.
- Spectrum – For a spectrum event, one or more load spectra are specified. A load spectrum is a list of load multipliers used to define a sequence of K_{max} and K_{min} values, which are used to define a sequence of ΔK's. If multiple spectra are specified, they are superimposed.
- Hold Event – The four load events defined above can include a hold (dwell) time. A hold event allows one to define a hold time without any companion load cycling.
- Dynamic Pairing – examines all SIFs and dynamically creates max-min pairs.
- Schedule – A schedule event can contain child events and allow a tree-like structure to be constructed.

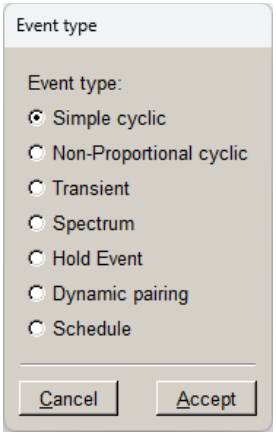


Figure 6.5.11 The event type dialog.

Note that some event types will not be active if there is only one set of SIFs. Non-Proportional Cyclic and Transient types require more than one load step (or set of SIFs).

If the Non-Proportional Cyclic event type is selected, the Load Schedule dialog is updated, Fig 6.5.12; and the Load Steps tab is automatically selected. The yellow triangles indicate that the load steps associated with K_{max} and K_{min} must be specified. The **Select a load step** button brings up the dialog shown to the right in Fig 6.5.12. For both K_{max} and K_{min}, the load step is selected from the "pull-down" list of load steps.

If there is only one substep (frame) for a load step, the Load Sub-Step option is unavailable. If there are multiple substeps, the user can choose the substep as well. A load scaling factor can be defined.

Note that in the Select Load Step dialog, the **Sum Multiple Steps** button allows K_{max} and K_{min} to be defined as the sum of the *K*'s associated with two or more load steps (each with its own load multiplier).

Once the load steps for K_{max} and K_{min} have been selected, the Load Schedule dialog is updated as shown in Fig 6.5.13.

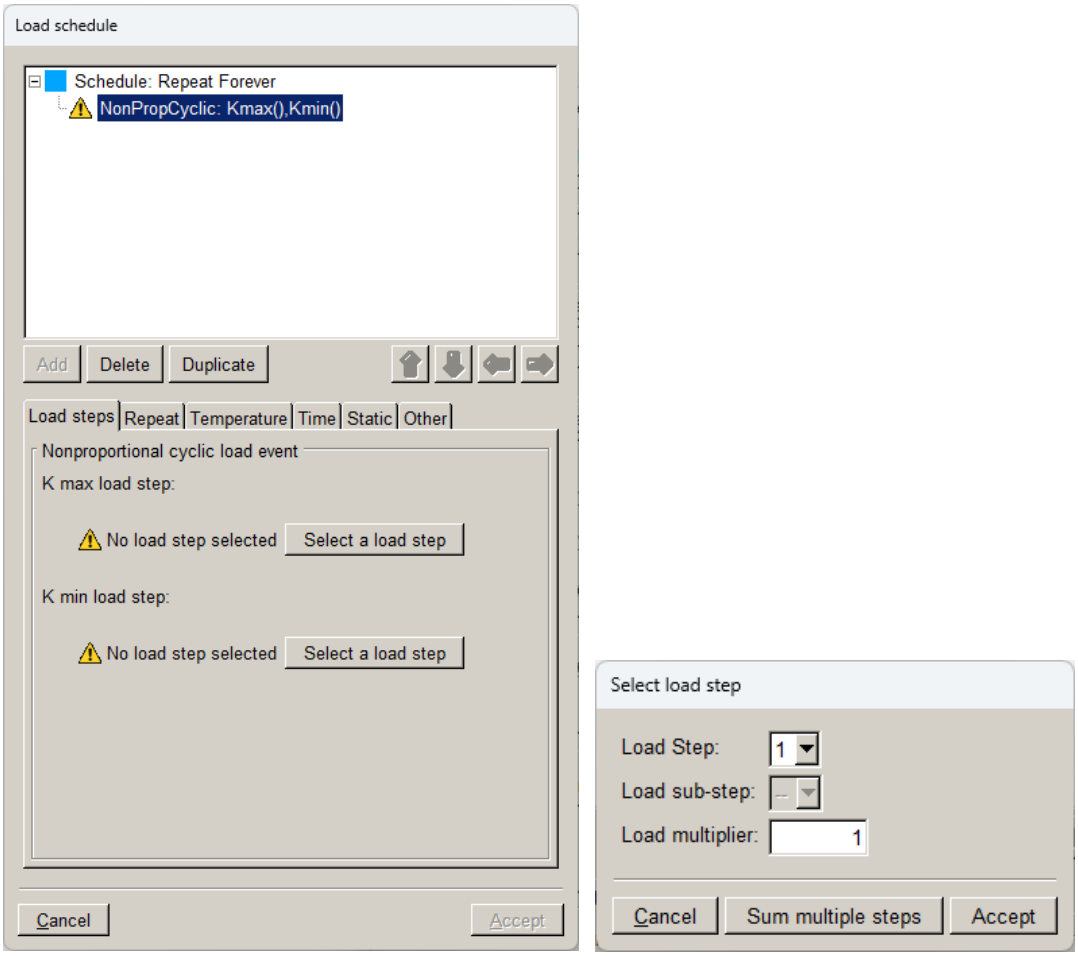


Figure 6.5.12 The Load Schedule dialog with a non-proportional cyclic load event added. The **Select a load step** button displays the dialog on the right.

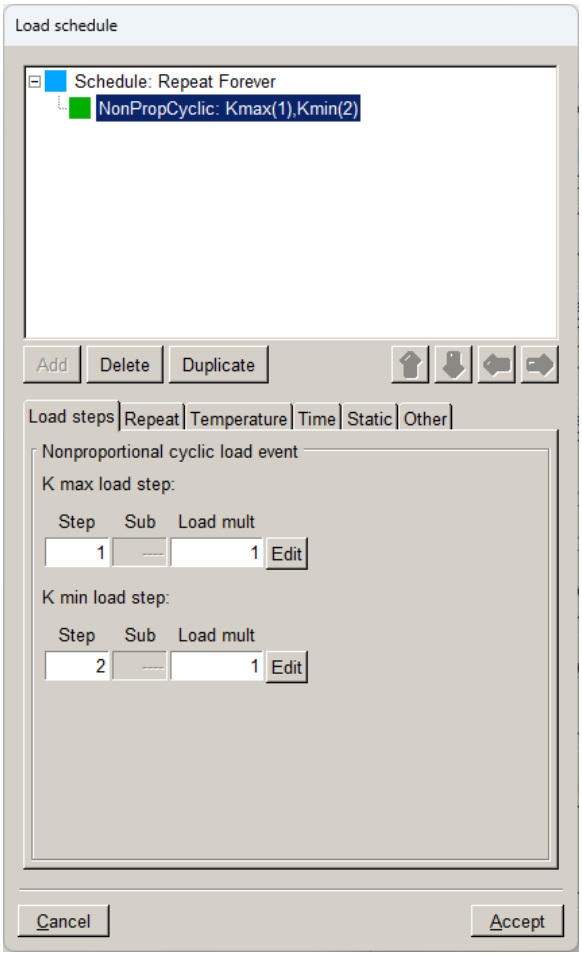


Figure 6.5.13 The Load Schedule dialog after load steps selected for K_{max} and K_{min}.

The Load Schedule dialog with the Temperature tab selected is shown in Fig 6.5.14, with Temperature dependent crack growth checked.

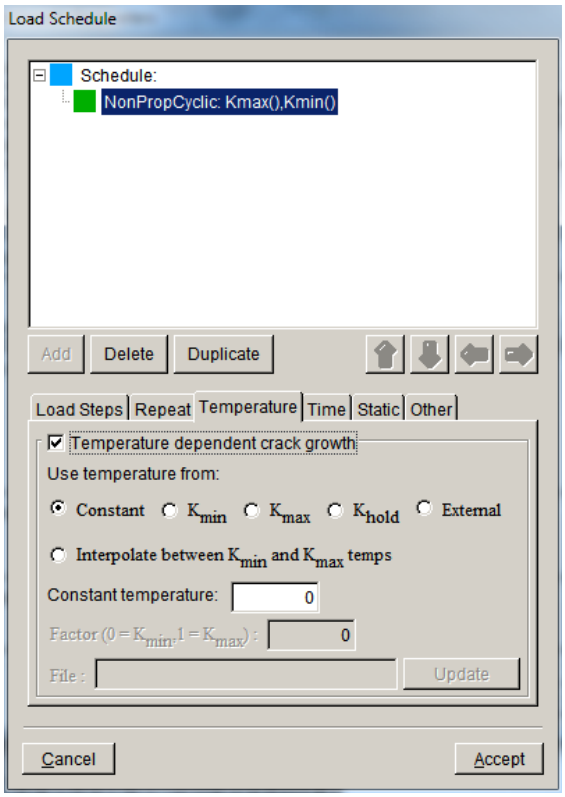


Figure 6.5.14 The Temperature tab of the load schedule dialog.

Because predicting crack growth might involve several different FE load steps, each of which can have associated temperatures, it is not obvious what temperature to use when evaluating a temperature dependent growth model. The following options are available:

- Constant – a constant temperature, provided by the user, is assumed for all crack front nodes, and is used for evaluating temperature dependent growth models.
- K_{min} – the crack-front temperatures associated with the load step that produces K_{min} are used.
- K_{max} – the crack-front temperatures associated with the load step that produces K_{max} are used.
- K_{hold} – the crack-front temperatures associated with the load step that produces K_{hold} are used.
- External – the crack-front temperatures are read from an independent set of analysis results. This will be a mesh file (e.g., *.inp* file) and the associated results file (e.g., an *.odb* or *.dtp* file).
- Interpolate – the crack front temperatures will be linearly interpolated between the K_{min} and K_{max} temperatures using a user-provided parameter that can vary between zero and one, where zero is the K_{min} temperature and one is the K_{max} temperature.

Note that if K_{min}, K_{max}, or K_{hold} are specified to be computed from the sum of multiple load steps, only temperatures from the first load step specified will be used to determine crack-front temperatures.

The Load Schedule dialog with the Time tab selected is shown in Fig 6.5.15, with Time dependent crack growth checked; the hold time is added as part of a *Non-Proportional Cyclic* event.

Simple Cyclic, Non-Proportional Cyclic, Transient, and Spectrum Load event types all allow a hold time. A hold-time can also be specified as an independent Hold event.

Note that one might consider separate cyclic and hold events if, for example, the cyclic growth rate and hold growth rates were to be computed for different temperatures. FRANC3D will compute the crack extension for all the cyclic loading in an event before computing the extension due to a hold-time. If it is desired to have the time dependent growth computed first, that should be done with a separate hold event that appears before the cyclic event in the load schedule.

The Hold FOREVER option makes the hold-time infinitely long. This could be used, for example, if one wanted to simulate time-dependent crack growth (e.g., stress corrosion cracking) and keep the crack growth simulation going until a critical stress intensity factor was reached.

The Static tab is shown in Fig 6.5.16. It allows the analyst to specify that the K from a load step (or sum of load steps) is to be added to both K_{max} and K_{min}. This will not change the ΔK, but it will change the stress ratio, R, for crack growth models sensitive to R. This could represent a situation where there is a steady "fixed" load and an alternating cyclic load.

The static load can be defined as a crack face traction, with stresses read from external analyses results (*e.g.*, *.inp* and *.odb* files).

The Other tab is shown in Fig 6.5.17. It allows one to define how ΔK is treated if the K_{min} becomes greater than K_{max}.

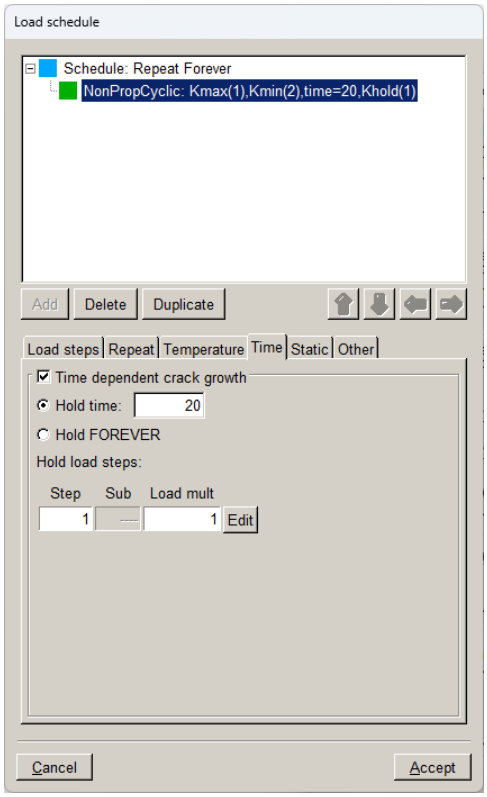


Figure 6.5.15 The load schedule dialog with the Time tab selected.

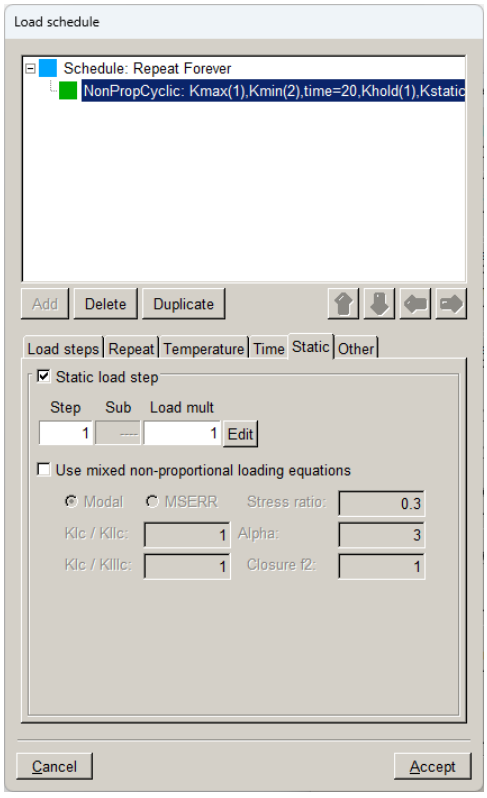


Figure 6.5.16 The load schedule dialog with the Static tab selected.

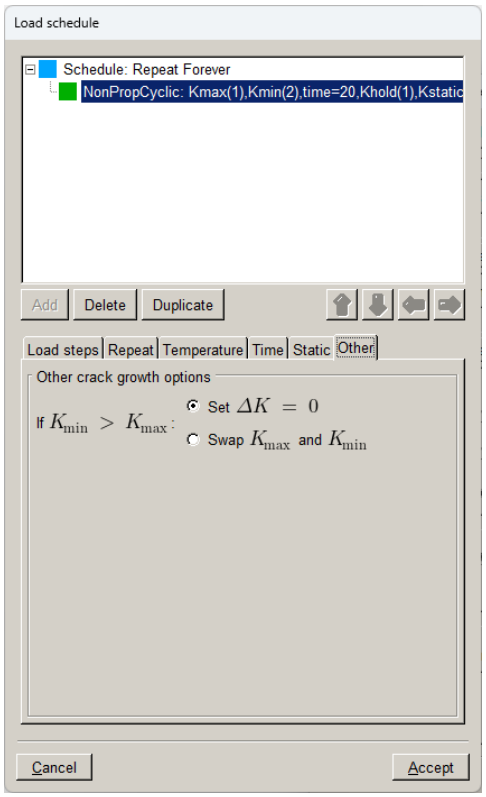


Figure 6.5.17 The load schedule dialog with the Other tab selected.

By default, a load event (load cycle, hold time, or combination) will be applied once. The Repeat tab allows the analyst to specify that the load event should be applied some finite number of times. It also allows one to specify that the event will be repeated **FOREVER**.

All load events are the "child" of a *Schedule*. When the repeat count is set for a *Schedule*, all the child events are repeated for the specified number of times. By organizing the events into a tree-like structure and setting repeat counts for the *Schedule* and loading events, arbitrarily complex load sequences can be defined.

These options, along with options set for the crack growth rate model, will determine when FRANC3D will stop a crack growth simulation. Crack growth will always stop if the K_{max} at any point on a crack front reaches a critical value. It will also stop if the ΔK for all points on the crack front fall below the threshold ΔK. In addition, if none of the load events in the load schedule is set to repeat FOREVER, FRANC3D will make one complete pass through the load schedule and then stop the crack growth. If repeat FOREVER is specified, FRANC3D will continue to apply the corresponding load events (sequence of children events in the case of a *Schedule* event) until one of the stopping criteria from the crack growth rate model is reached.

It is usually best to specify repeat FOREVER for the root schedule event, Fig 6.5.18. This allows FRANC3D to more easily determine if an accelerated counting algorithm can be used when computing cycles.

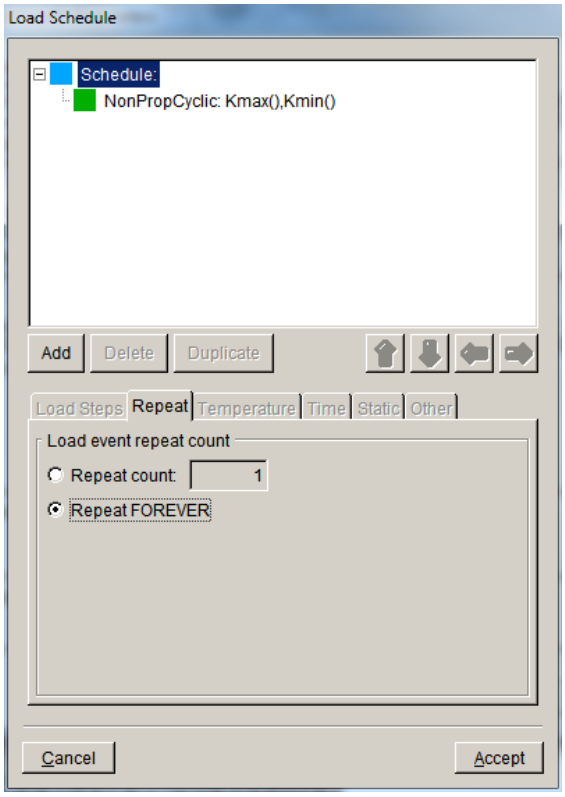


Figure 6.5.18 Load schedule dialog with Repeat FOREVER selected for the schedule.

6.5.1.3.1.1 Simple Cyclic Event

If the user chooses a Simple Cyclic load type (see Fig 6.5.11), the lower portion of the Load Schedule dialog appears as in Fig 6.5.19. The user can specify a given Stress ratio (R) and then specify the load step (and substep if there is more than one) along with a load multiplier. The stress ratio (R) will be used to compute K_{min} and ΔK values.

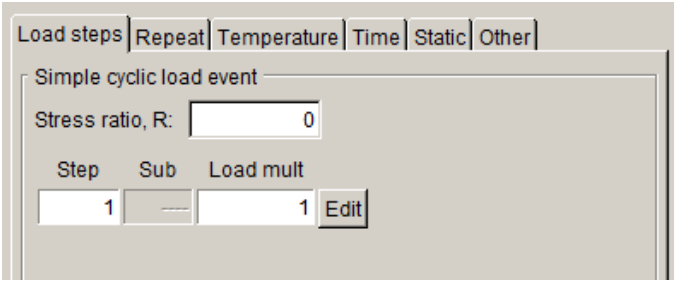


Figure 6.5.19 Simple cyclic load event options.

6.5.1.3.1.2 Non-Proportional Cyclic Event

If the user chooses a Non-Proportional Cyclic load type (see Fig 6.5.11), the lower portion of the Load Schedule dialog appears as in Fig 6.5.20. The user must specify the load step (and substep if there is more than one) along with a load multiplier for both K_{max} and K_{min}; stress ratio and ΔK are computed from these, and kink angle is based on ΔK or K_{max}.

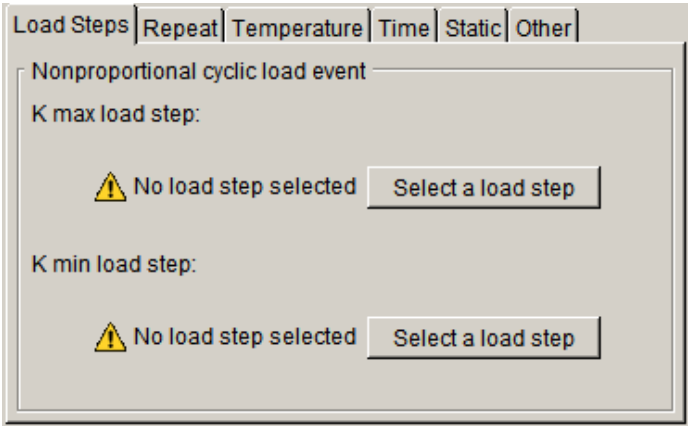


Figure 6.5.20 Non-proportional cyclic load event options.

6.5.1.3.1.3 Transient Event

If the user chooses a Transient load type (see Fig 6.5.11), the lower portion of the Load Schedule dialog appears as in Fig 6.5.21. The user must specify the load steps (and substeps if more than one) along with load multiplier, Fig 6.5.22 – left side. If multiple load steps are to be analyzed, use the **Multiple Steps** button to select the load steps, Fig 6.5.22 – right side. FRANC3D computes the K_{max} and K_{min} values from all the selected load steps; stress ratio and ΔK are computed from these, and kink angle is based on ΔK or K_{max}.

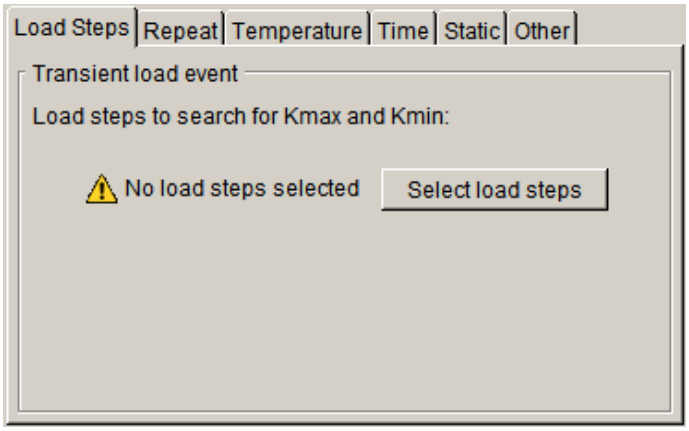


Figure 6.5.21 Transient load event options.

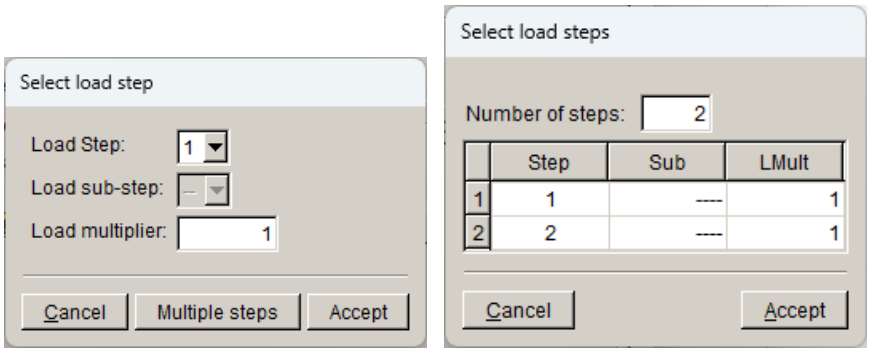


Figure 6.5.22 Transient load event load step selection.

6.5.1.3.1.4 Spectrum Event

If a user chooses a Spectrum load type (see Fig 6.5.11), the lower portion of the Load Schedule dialog appears as in Fig 6.5.23. The user must specify the load steps (and substeps if more than one) along with a load spectrum; multiple spectra can be used.

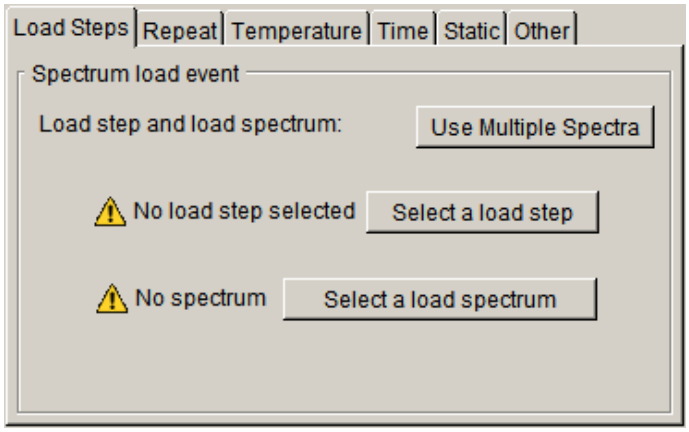


Figure 6.5.23 Spectrum load event options.

A spectrum is defined as a series of load ranges. Each load range can have an optional repeat count (the default value is one). One can also specify a load multiplier that will be applied to all values in the spectrum (e.g., to support the use of a normalized spectrum).

If only one load case is used in the analysis, and load range i has specified values $S^{raw}_{\max,i}$ and $S^{raw}_{\min,i}$ the expressions used for computing the SIF ranges and R-ratio for range i are

$$S_{\max,i} = sS^{raw}_{\max,i} + t\,,\ S_{\min,i} = sS^{raw}_{\min,i} + t$$

$$K_{\max,i} = S_{\max,i}\, K_{comp},\ K_{\min,i} = S_{\min,i}\, K_{comp}$$

$$\Delta K_i = K_{\max,i} - K_{\min,i} \text{ and } R_i = K_{\min,i} / K_{\max,i}$$

where *s* is the spectrum multiplier, and *t* is the spectrum offset. The ΔK_i and R_i are used to compute the crack growth rate for stress range *i*. If multiple load cases are used, there are additional options for setting these values.

For variable amplitude loading, the crack growth rates, which are used for computing the relative amounts of crack growth for crack front points, are the average crack growth rates computed for one pass through the spectrum. That is

$$\frac{da}{dN} = \frac{\sum_{i=1}^{n} \frac{da}{dN_i}(\Delta K_i, R_i, ...)}{n}$$

where da/dN is the crack growth rate computed for any given crack front point, and *n* is the number of load ranges in the spectrum.

The corresponding predicted kink angle is a weighted average kink angle computed as

$$\overline{\theta}_{kink} = \frac{\sum_{i=1}^{n} \frac{da}{dN_i}(\Delta K_i, R_i, ...) \times \theta_{kink,i}(K_{I,i}, K_{II,i}, K_{III,i}, ...)}{\sum_{i=1}^{n} \frac{da}{dN_i}(\Delta K_i, R_i, ...)}$$

where $\theta_{kink,i}$ is the kink angle determined (implicitly) for range *i* using one of the criteria described in Section 6.5.1.1.

The **Select a load step** button (in Fig 6.5.23) displays the dialog for selecting the load step (see Fig 6.5.22).

The **Select a load spectrum** button first displays a dialog for selecting the spectrum file, Fig 6.5.24.

FRANC3D reads the data and displays it in a dialog, Fig 6.5.25 – left side, which allows one to specify the delimiter and to ignore the first line if labels are present. The next dialog, Fig 6.5.25 – right side, allows one to specify the Min, Max, and Count columns.

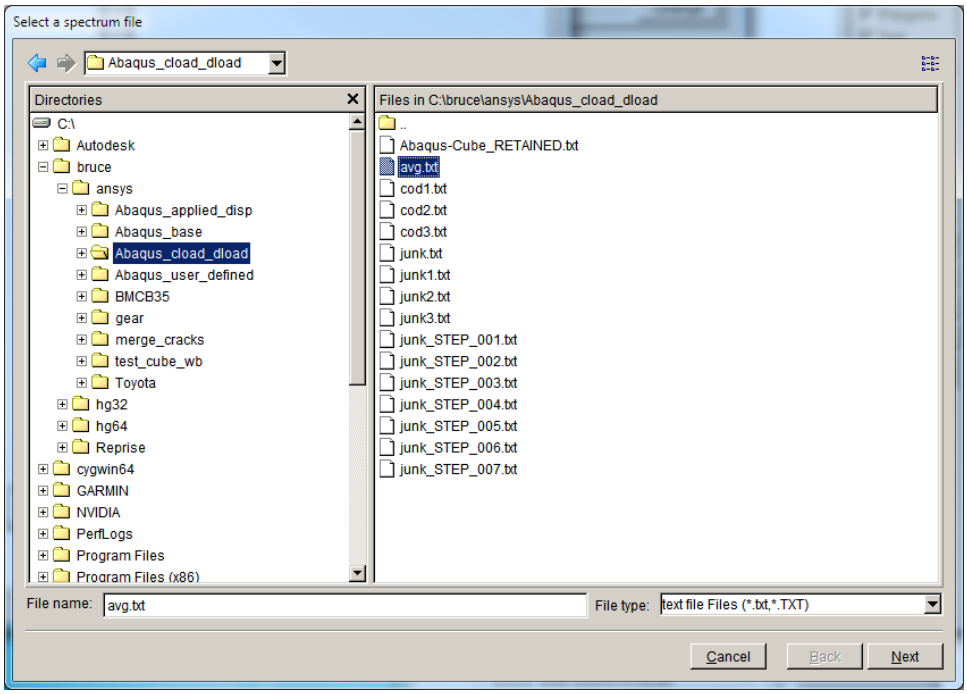


Figure 6.5.24 Select load spectrum file dialog.

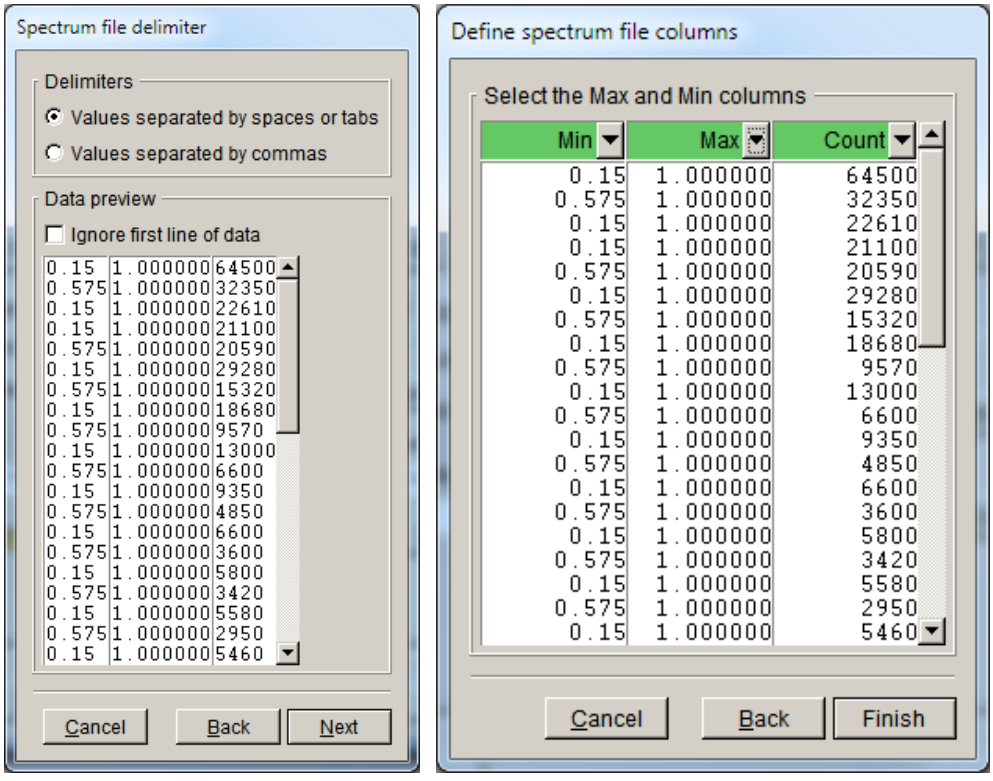


Figure 6.5.25 Select load spectrum file dialog.

Once the spectrum has been read, the Load schedule dialog appears as in Fig 6.5.26. The **Display** button beside the Spectrum label can be used to view the spectrum, Fig 6.5.27.

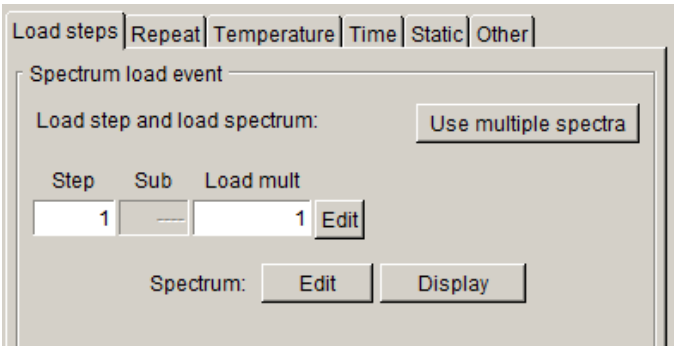


Figure 6.5.26 Spectrum load event with load step and spectrum defined.

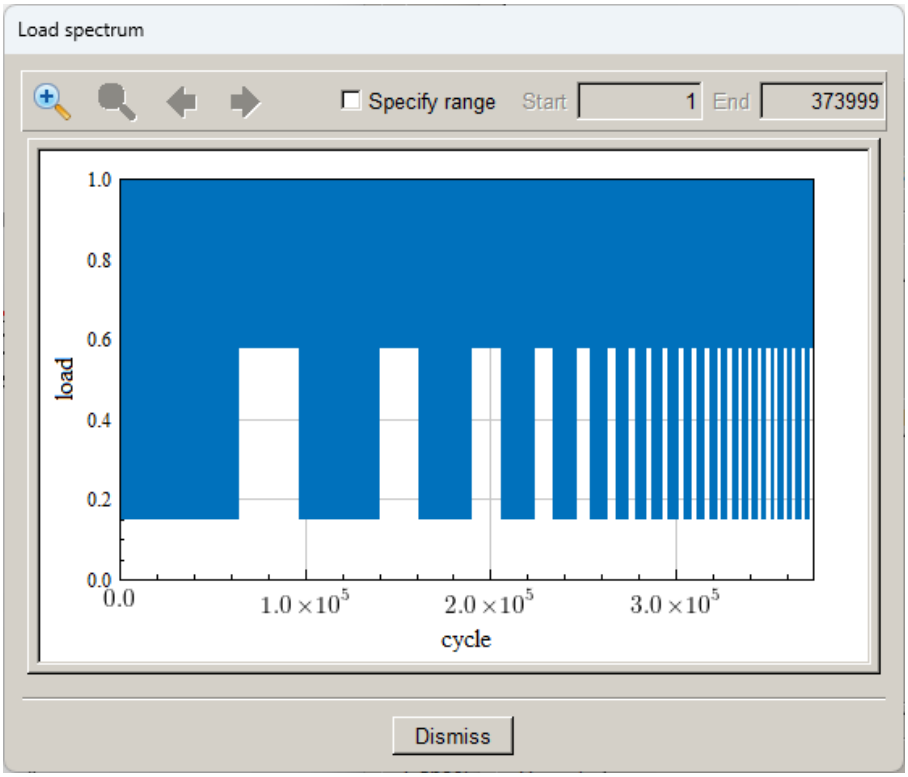


Figure 6.5.27 Spectrum display dialog.

The magnifying glass icons allow one to view a segment of the spectrum and the arrow icons allow one to move forward or backward through the spectrum. A cycle range can be specified as well, which can be faster than multiple levels of zooming.

6.5.1.3.1.5 Hold Event

If the user chooses a Hold load type (see Fig 6.5.11), the lower portion of the Load Schedule dialog appears as in Fig 6.5.28. The user must specify the hold time and the load step. The hold time was described previously as part of the root schedule; see Section 6.5.1.3.

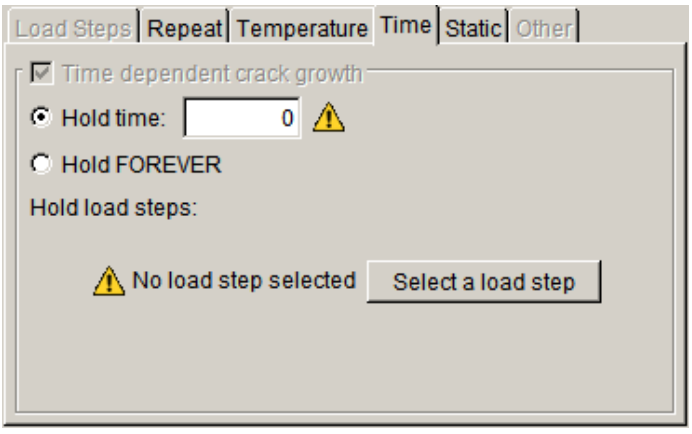


Figure 6.5.28 Hold load event options.

6.5.1.3.1.6 Dynamic Pairing Event

If the user chooses a Dynamic Pairing load type (see Fig 6.5.11), the Load Schedule dialog appears as in Fig 6.5.29. Once the event is added, select this event (with a left click), and click the **Add** button to display the dialog shown in Fig 6.5.30. This dialog allows one to build up a series of events within the Dynamic Pairing event.

Note that the repeat count for Dynamic Pairing events can only be set to one. The top-level schedule repeat count can be set to FOREVER to force the Dynamic Pairing event to be repeated more than one time.

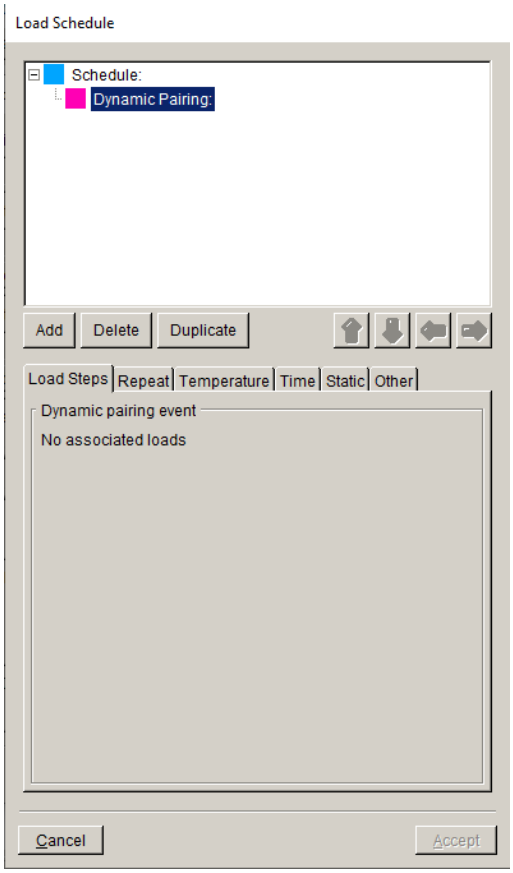


Figure 6.5.29. Dynamic Pairing load event added to the root schedule.

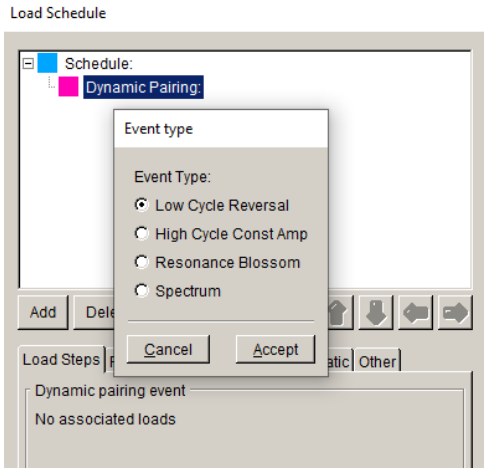


Figure 6.5.30 Dynamic Pairing event types.

There are four event types:

- Low Cycle Reversal – the load step associated with an LCF event is specified.
- High Cycle Const Amp – the load step associated with both a static and a vibratory load step are specified.

- Resonance Blossom – the load step associated with a static load step and a vibratory load step are specified along with several parameters, Fig 6.5.31. One must specify the resonance frequency (in Hz), the radial acceleration or deceleration (in radians/sec^2, this is assumed to be constant throughout the blossom), the engine order (number of forced excitations per rotation), and a critical damping ratio (dimensionless).
- Spectrum – For a spectrum event, one or more load spectra are specified, as described for the Spectrum event in Section 6.5.1.3.1.4.

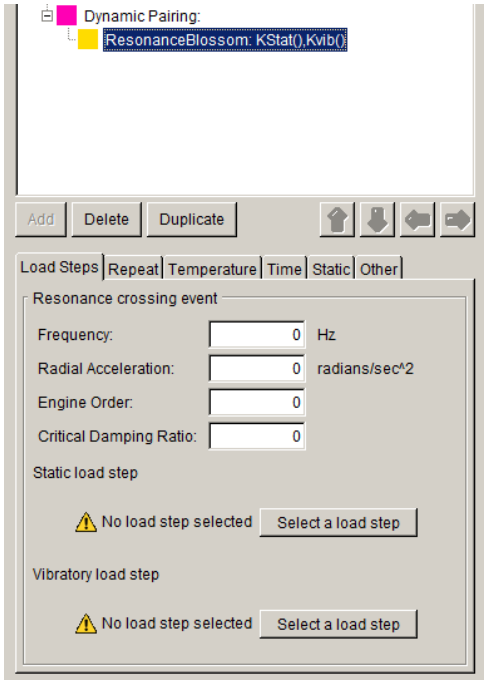


Figure 6.5.31 Resonance blossom parameters.

6.5.1.3.1.7 Schedule Event

If the user chooses a Schedule load type (see Fig 6.5.11), the Load Schedule dialog appears as in Fig 6.5.32. This allows one to build up a sub-schedule with any of the load event types just described.

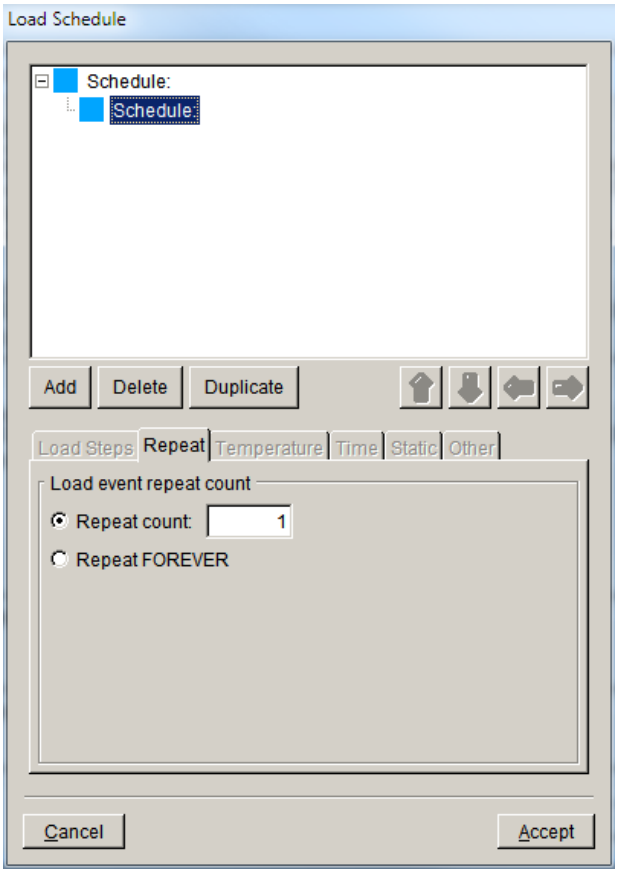


Figure 6.5.32 Schedule load event added to the root schedule.

6.5.1.3.2 Load Schedule: Read From File

The Crack Growth Load Schedule **Read From File** button (see Fig 6.5.8) allows one to read a load schedule that was previously defined and saved to a file. This file contains only the load schedule information. It is used in conjunction with the **Save To File** button in Section 6.5.1.3.5. It is an ASCII text file and should contain this data "block":

```
FATIGUE_LOAD_SCHEDULE
(
  VERSION: 1
  SCHEDULE (
    ....
  )
)
```

6.5.1.3.3 Load Schedule: Wizard

The Crack Growth Load Schedule **Wizard** button (see Fig 6.5.8) presents a series of dialog boxes that leads you through the step-by-step process of creating the load schedule. The first

dialog, Fig 6.5.33, shows the types of load schedules that can be built using this wizard; note that some event types are not included.

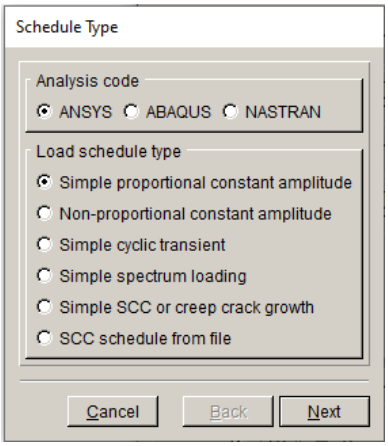


Figure 6.5.33 Load schedule wizard first dialog.

The subsequent dialogs will depend on the selected schedule/event type.

6.5.1.3.4 Load Schedule: View/Edit

The Crack Growth Load Schedule **View/Edit** button (see Fig 6.5.8) displays the previously defined load schedule in the dialog. The load schedule can be completely revised if needed.

6.5.1.3.5 Load Schedule: Save To File

The Crack Growth Load Schedule **Save To File** button (see Fig 6.5.8) allows the user to save the load schedule to a file. This file contains only the load schedule information; it is used in conjunction with the **Read From File** button in Section 6.5.1.3.2.

6.5.1.4 Subcritical Growth Rate Model

The Subcritical Growth Parameters dialog (see Fig 6.5.8) requires that you set a Crack Growth Rate Model. This is material dependent data that describes the crack growth rate based on cyclic loading and/or time.

6.5.1.4.1 Growth Rate Model: New Model – cyclic loading

Select the **New Model** button to display the dialog shown in Fig 6.5.34. Cyclic loading and time dependent growth rate models can be defined. The cyclic loading models are described first.

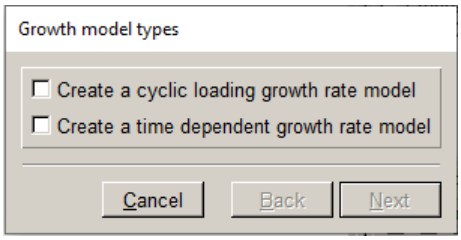


Figure 6.5.34 The crack growth rate model type dialog.

The Cyclic Loading Growth Model dialog, Fig 6.5.35, supports two general types of cyclic growth models, "paired" and "inclusive" models. The difference between them is based on how the stress ratio effects are incorporated. For a paired model, the "shape" of the crack growth rate curve (Paris, Sigmoidal, *etc*.) is selected independently from the stress ratio algorithm (Walker, Newman Closure or Table). The inclusive models (two different versions of the NASGRO equation) have stress ratio effects built into the model equations.

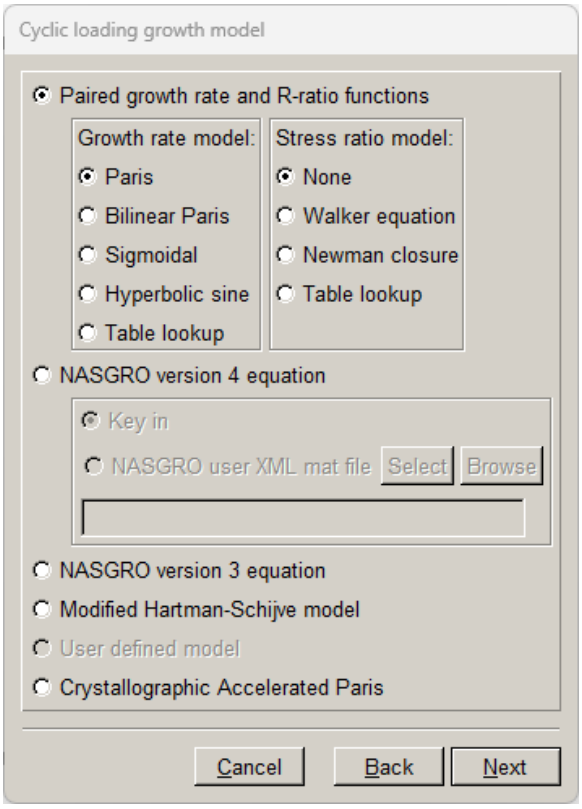


Figure 6.5.35 The cyclic loading growth model.

For the Paris crack growth rate model, for example, with no stress ratio effects, the next dialog allows you specify either a temperature dependent or independent model, Fig 6.5.36. If the growth rate model is temperature independent, this means that only one set of growth rate parameters (C and m for the Paris model) are required.

For temperature dependent models, different sets of parameters are specified for different temperatures, and FRANC3D interpolates among these values to find the appropriate growth rate for the crack-front temperature.

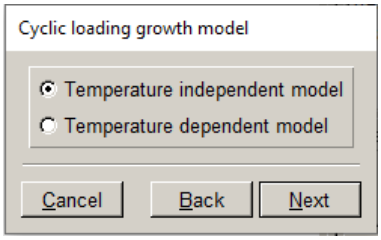


Figure 6.5.36 The temperature independent model selected.

For a temperature independent Paris model, the dialog shown in Fig 6.5.37 is displayed. The Model Label, and Description fields are optional. These might be useful if the model parameters are stored in a file for later reuse.

The units must be specified. The units need not be the same units used in the FE analysis. If the units are different, FRANC3D will automatically perform the necessary unit conversions. However, FRANC3D assumes that the units used within the growth model description (or within the FE analysis) are self-consistent; *i.e.,* C and K_c units must match.

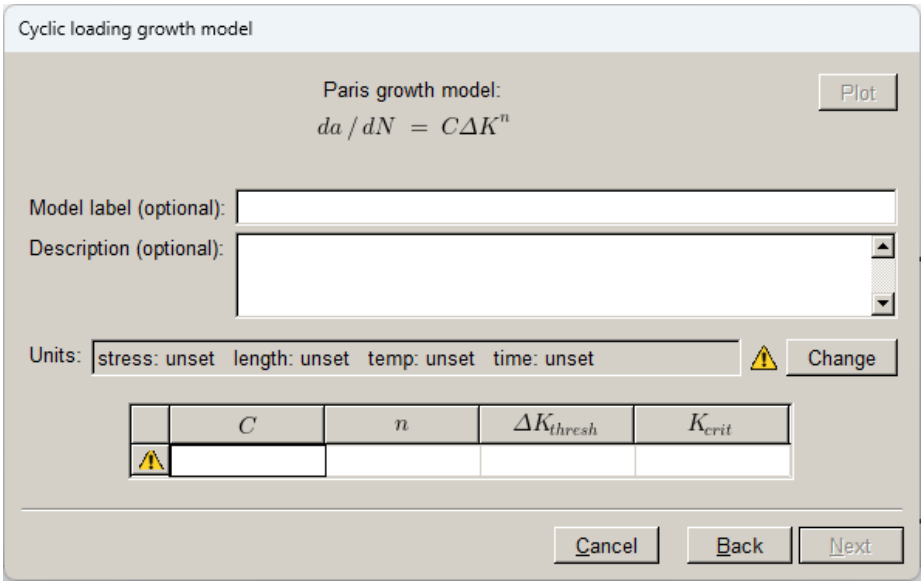


Figure 6.5.37 Dialog for entering temperature independent Paris growth model parameters.

Each growth rate and stress ratio model is described in the following subsections. The dialogs for entering the growth rate model parameters are like that shown in Fig 6.5.37; differences will be noted in each subsection.

6.5.1.4.1.1 Paris model

The Paris growth rate model (see Fig 6.5.37) is a power law model expressed as

$$da/dN = C(\Delta K_{eff})^n$$

The parameters C and n must be specified along with values for $\Delta K_{threshold}$ and $K_{critical}$. If $\Delta K_{eff} \le \Delta K_{threshold}$, da/dN is set to zero. If $\Delta K_{eff} \ge (1-R)K_{critical}$, unstable crack growth is assumed and da/dN is infinite.

Note that FRANC3D labels the exponent for the Paris model as "n". In addition to C and n, values must be entered for $\Delta K_{\text{threshold}}$ and K_c. Specifying a small value for $\Delta K_{\text{threshold}}$ or a large value for K_c will effectively turn off the possibility that the crack growth simulation will stop due to a below-threshold or above-critical condition.

6.5.1.4.1.2 Bilinear Paris model

The Bilinear Paris model, Fig 6.5.38, is an extension of the Paris model; it has two linear portions when plotted on a log/log graph. The equation is:

$$\frac{da}{dN} = \begin{cases} C_1(\Delta K_{eff})^{n_1} & \Delta K_{eff} \le \Delta K^* \\ C_2(\Delta K_{eff})^{n_2} & \Delta K_{eff} > \Delta K^* \end{cases}$$

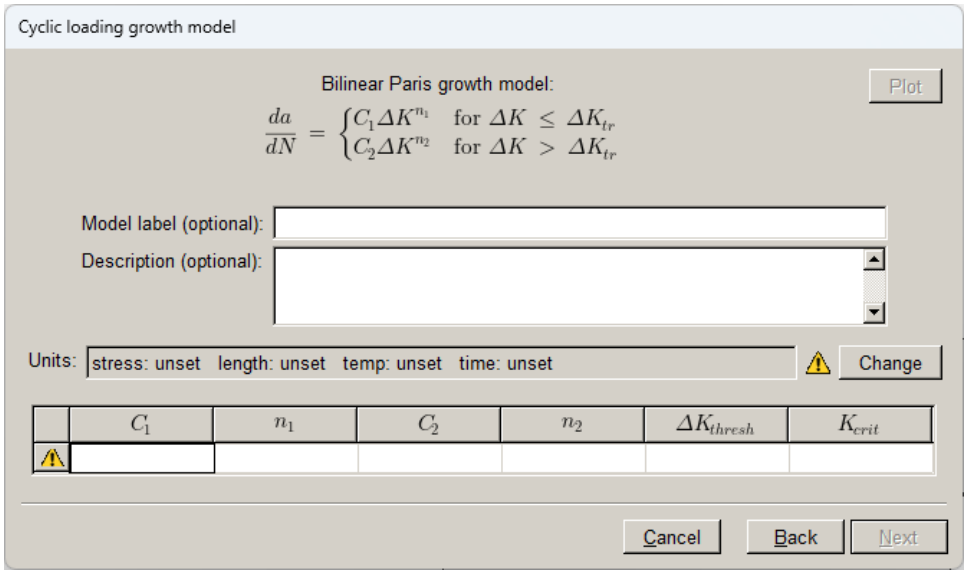


Figure 6.5.38 Dialog for entering temperature independent Bilinear Paris parameters.

The parameters C_1, C_2, n_1, and n_2 must be specified along with values for $\Delta K_{threshold}$ and $K_{critical}$. If $\Delta K_{eff} \le \Delta K_{threshold}$, da/dN is set to zero. If $\Delta K_{eff} \ge (1-R)K_{critical}$, unstable crack growth is assumed and da/dN is infinite.

6.5.1.4.1.3 Sigmoidal cyclic loading growth model

The expression for the sigmoidal model, Fig 6.5.39, is:

$$\frac{da}{dN} = e^{B}\left(\frac{\Delta K_{eff}}{\Delta K_{threshold}}\right)^{P}\left(\ln\left[\frac{\Delta K_{eff}}{\Delta K_{threshold}}\right]\right)^{Q}\left(\ln\left[\frac{K_{critical}}{\Delta K_{eff}}\right]\right)^{D}$$

The parameters B, P, Q, D, $\Delta K_{threshold}$, and K_c must be specified.

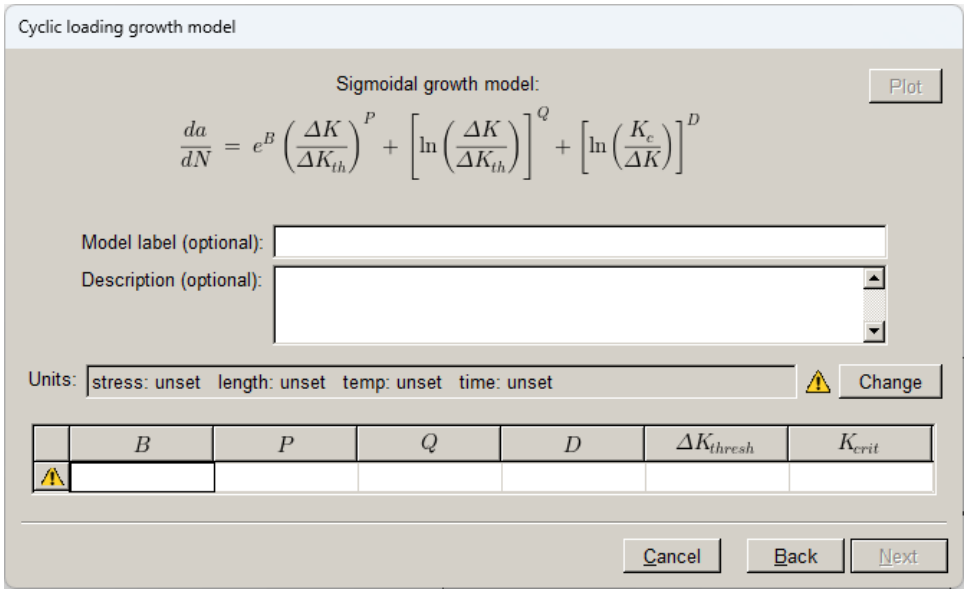


Figure 6.5.39 Dialog for entering temperature independent Sigmoidal growth model parameters.

6.5.1.4.1.4 Hyperbolic Sine model

The expression for the hyperbolic sine model, Fig 6.5.40, is:

$$\log(da/dN) = C_1 \sinh(C_2[\log(\Delta K) + C_3]) + C_4$$

The parameters C_1, C_2, C_3, C_4, $\Delta K_{threshold}$, and K_c must be specified. If $\Delta K_{eff} \le \Delta K_{threshold}$, da/dN is set to zero. If $\Delta K_{eff} \ge (1-R)K_{critical}$, unstable crack growth is assumed and da/dN is infinite.

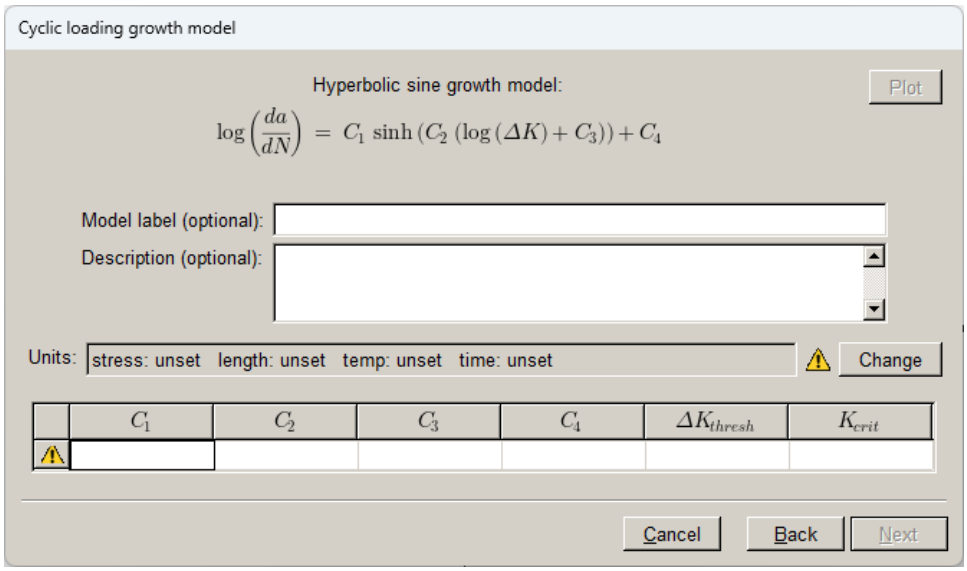


Figure 6.5.40 Dialog for entering temperature independent Hyperbolic sine growth model parameters.

6.5.1.4.1.5 Table Lookup growth model

For a table lookup model, Fig 6.5.41, a list of ΔK_{eff} and corresponding da/dN values are specified. Linear interpolation is performed using log-values. The following equation is used to interpolate between values *i* and *i*+1:

$$\log(da/dN) = \log(da/dN_i) + \frac{\log(\Delta K_{eff}/\Delta K_i)}{\log(\Delta K_{i+1}/\Delta K_i)} \log\left(\frac{da/dN_{i+1}}{da/dN_i}\right)$$

In addition to the $(\Delta K_{eff}, da/dN)$ values, a value must be specified for $K_{critical}$. The smallest specified ΔK_{eff} is assumed to be $\Delta K_{threshold}$. For ΔK_{eff} greater than the largest value specified and $\Delta K_{eff} < (1-R)K_{critical}$, the slope of the curve between the two highest specified points is extrapolated. This is unrealistic for many cases, so it is best to provide $(\Delta K_{eff}, da/dN)$ data that extend past the critical values.

Once data has been read or entered, the **Plot** button (in Fig 6.5.41) can be used to display the data, Fig 6.5.42. Note that data should be supplied in ascending order with respect to ΔK.

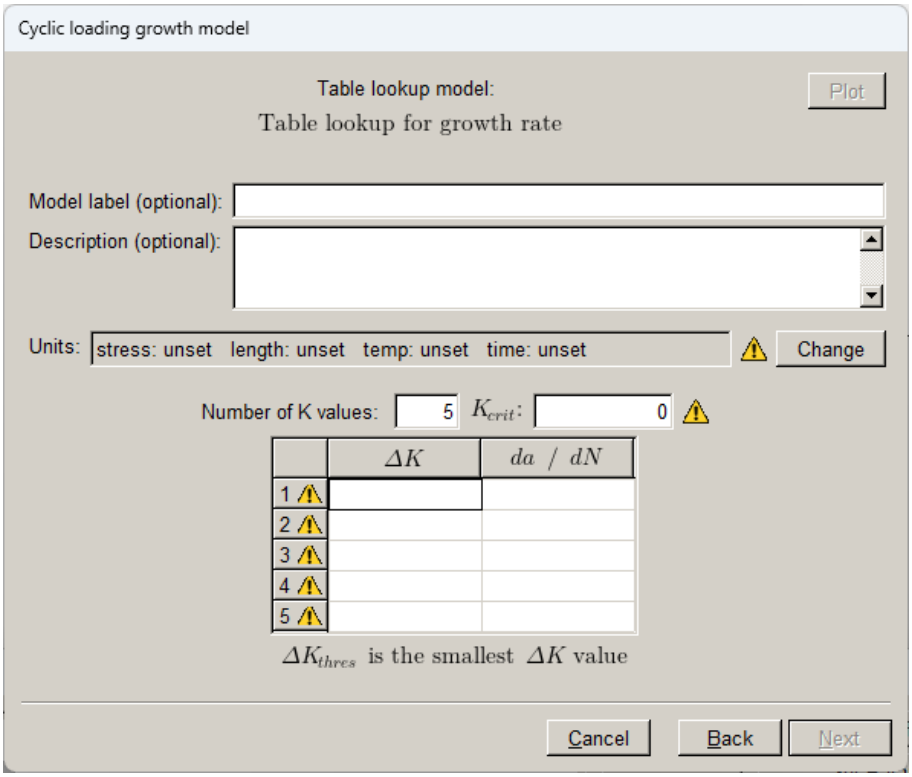


Figure 6.5.41 Dialog for entering temperature independent Table lookup growth model.

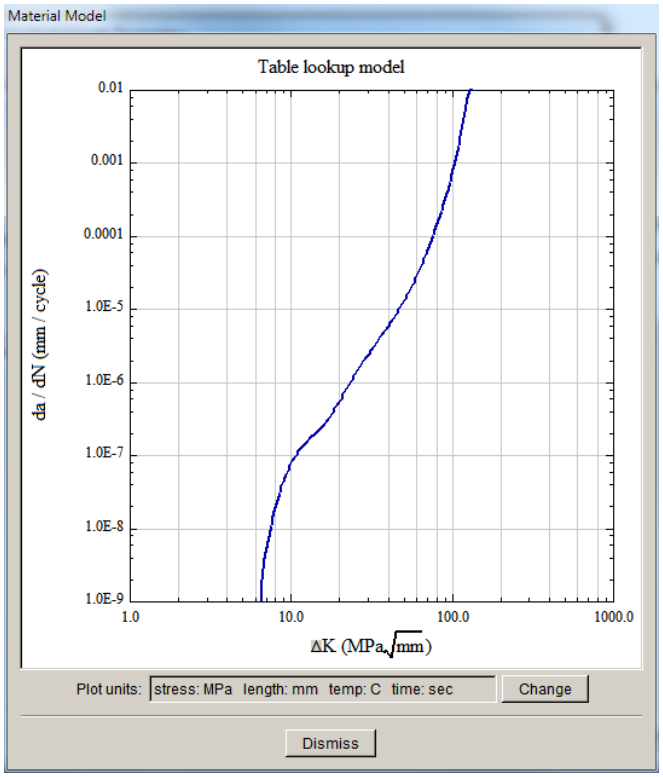


Figure 6.5.42 Plot of Table lookup growth model data.

6.5.1.4.1.6 No Stress Ratio model

This option turns off stress ratio effects. That is

$$\Delta K_{eff} = \Delta K$$

All the above growth rate model dialogs were shown with the stress ratio option set to None.

6.5.1.4.1.7 Walker Stress Ratio model

The expression for the Walker equation is

$$\Delta K_{eff} = (1-R)^{m-1}\Delta K$$

$$m = \begin{cases} m+ & R \geq 0 \\ m- & R < 0 \end{cases}$$

The parameters $m+$ and $m-$ must be specified.

NOTE: the Walker equation is defined relative to $R = 0$. That is, the parameters used for the paired growth rate model (e.g., Paris parameters) should be those appropriate for $R = 0$.

For each of the growth models paired with the Walker model, the dialogs are shown in Figs 6.5.43 – 47.

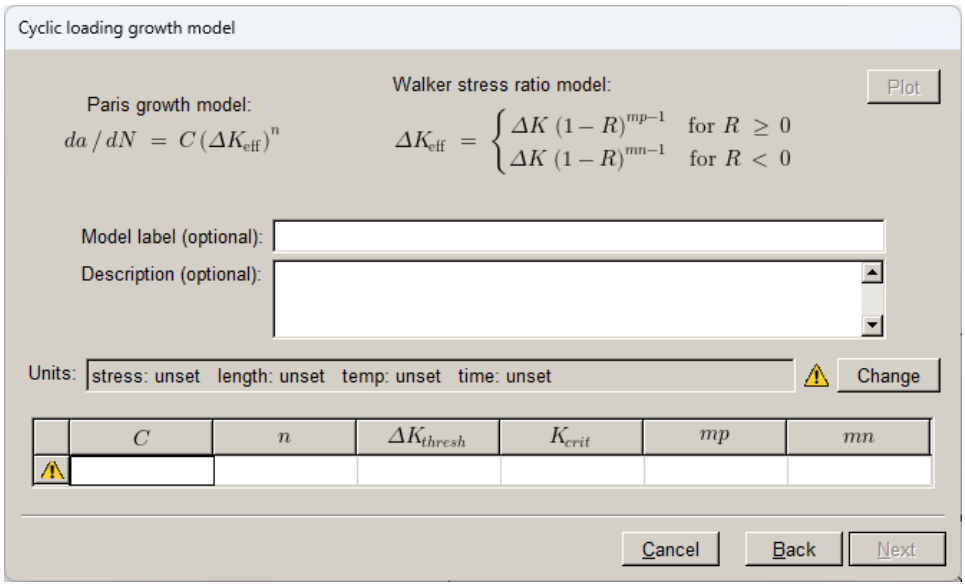


Figure 6.5.43 Dialog for entering temperature independent Paris growth and Walker stress ratio model parameters.

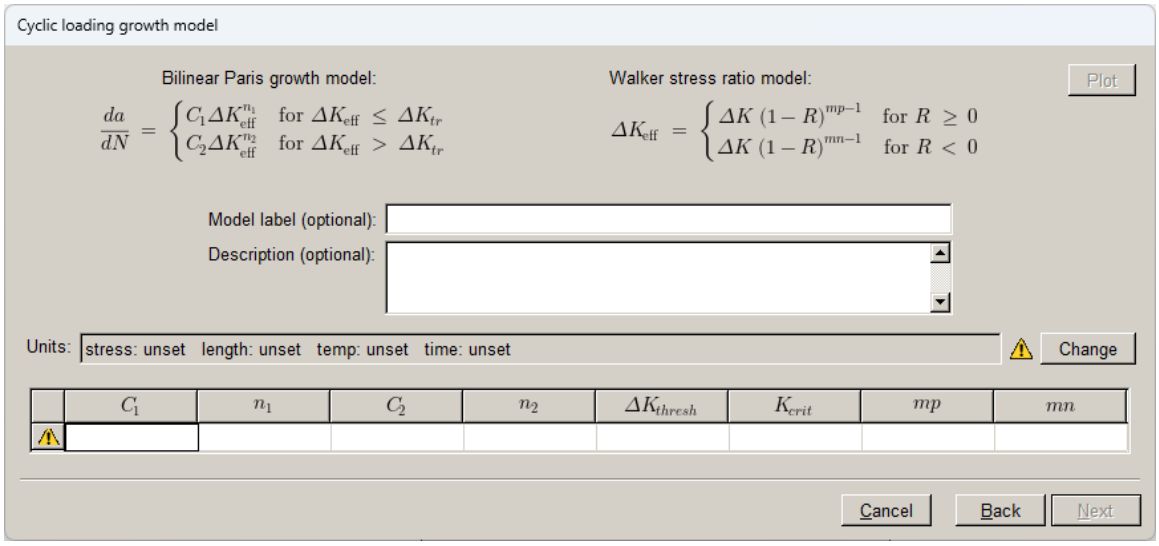


Figure 6.5.44 Dialog for entering temperature independent Bilinear Paris growth and Walker stress ratio model parameters.

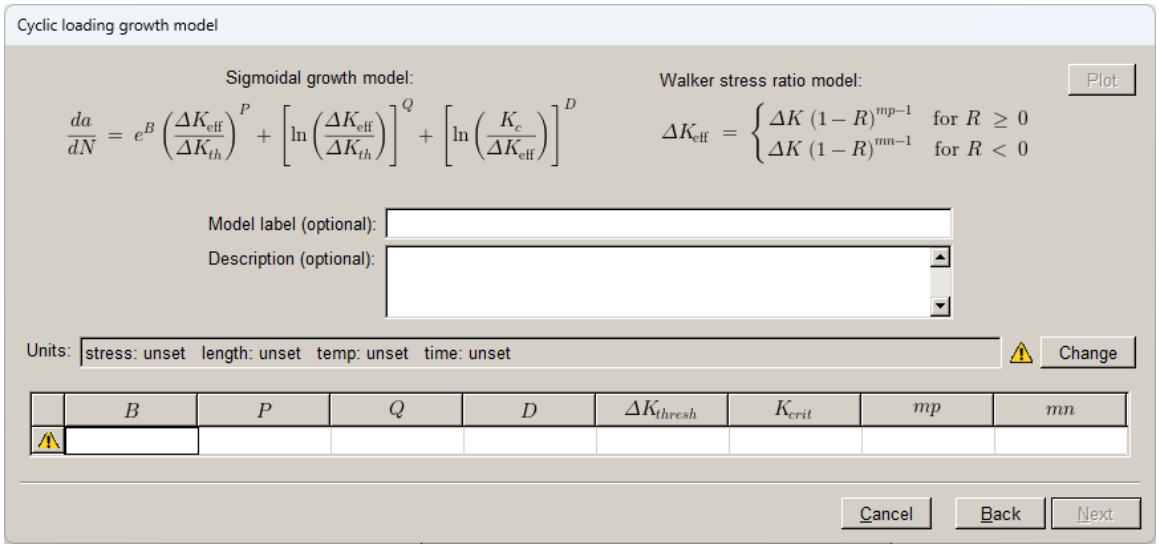


Figure 6.5.45 Dialog for entering temperature independent Sigmoidal growth and Walker stress ratio model parameters.

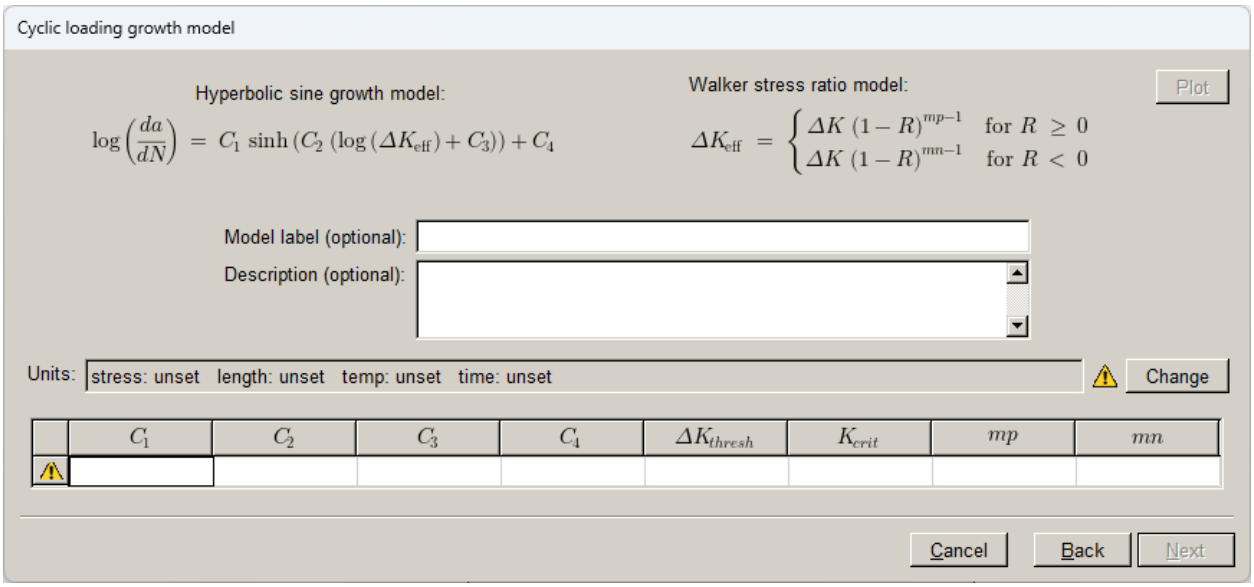


Figure 6.5.46 Dialog for entering temperature independent Hyperbolic sine growth and Walker stress ratio model parameters.

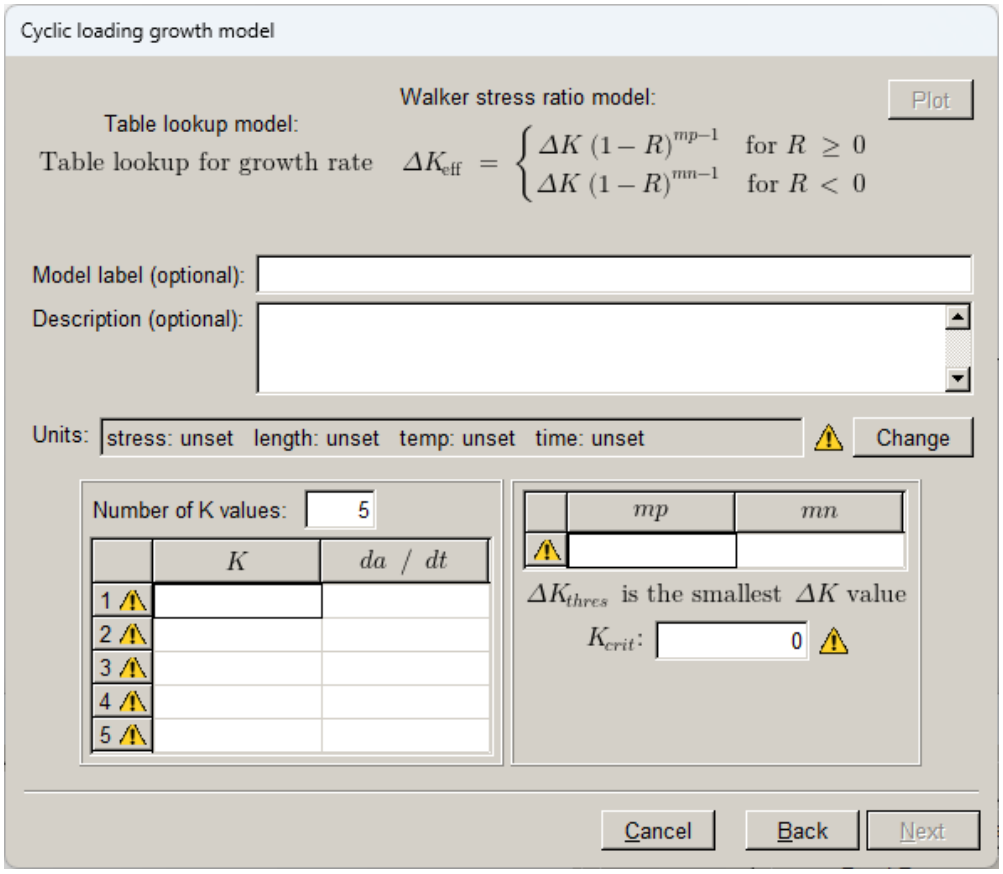


Figure 6.5.47 Dialog for entering temperature independent Table lookup growth and Walker stress ratio model parameters.

6.5.1.4.1.8 Newman Closure Stress Ratio model

The expression for Newman Closure is

$$\Delta K_{eff} = \frac{(1-f)}{(1-R)}\Delta K$$

where *f* is a function that accounts for the crack front being open for only a portion of the load cycle due to plasticity induced crack front closure. It is defined as

$$f = \frac{K_{open}}{K_{\max}} = \begin{cases} \max(R, A_0 + A_1 R + A_2 R^2 + A_3 R^3) & R \geq 0 \\ A_0 + A_1 R & -2 \leq R < 0 \end{cases}$$

where the coefficients are given by:

$$A_0 = (0.825 - 0.34\alpha + 0.05\alpha^2)\left[\cos\left(\frac{\pi}{2}\frac{S_{\max}}{\sigma_0}\right)\right]^{1/\alpha}$$

$$A_1 = \begin{cases} (0.415 - 0.071\alpha)\dfrac{S_{\max}}{\sigma_0} & R \geq 0 \\ 0.415 - 0.071\alpha & R < 0 \end{cases}$$

$$A_2 = 1 - A_0 - A_1 - A_3$$

$$A_3 = 2A_0 + A_1 - 1$$

The values of α and $S_{\max}/\sigma_0$ must be specified, where α is a plane stress/strain constraint factor and $S_{\max}/\sigma_0$ is the ratio of the maximum applied stress to the flow stress.

NOTE: the Newman Closure equation is defined for $R = R_{closure}$. That is, the parameters used for the paired growth rate model (e.g., Paris parameters) should be those appropriate for the case of $R = R_{closure}$, where $R_{closure}$ is the stress ratio where crack closure effects are no longer significant. This value is typically about 0.7 for most materials.

For each of the growth models paired with the Newman Closure model, the dialogs are shown in Figs 6.5.48 – 52.

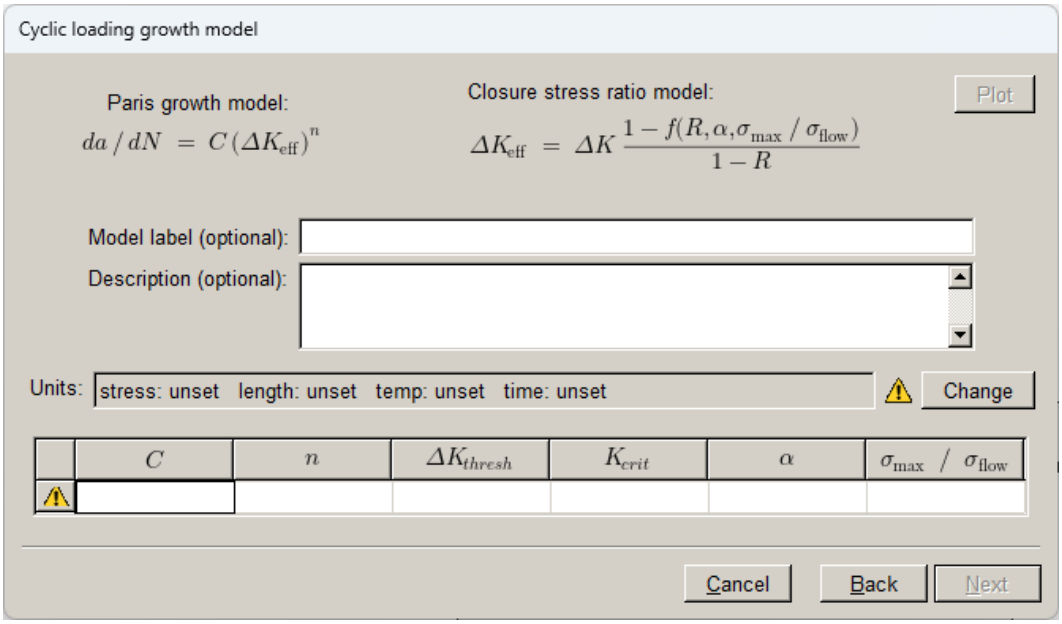


Figure 6.5.48 Dialog for entering temperature independent Paris growth and Newman Closure stress ratio model parameters.

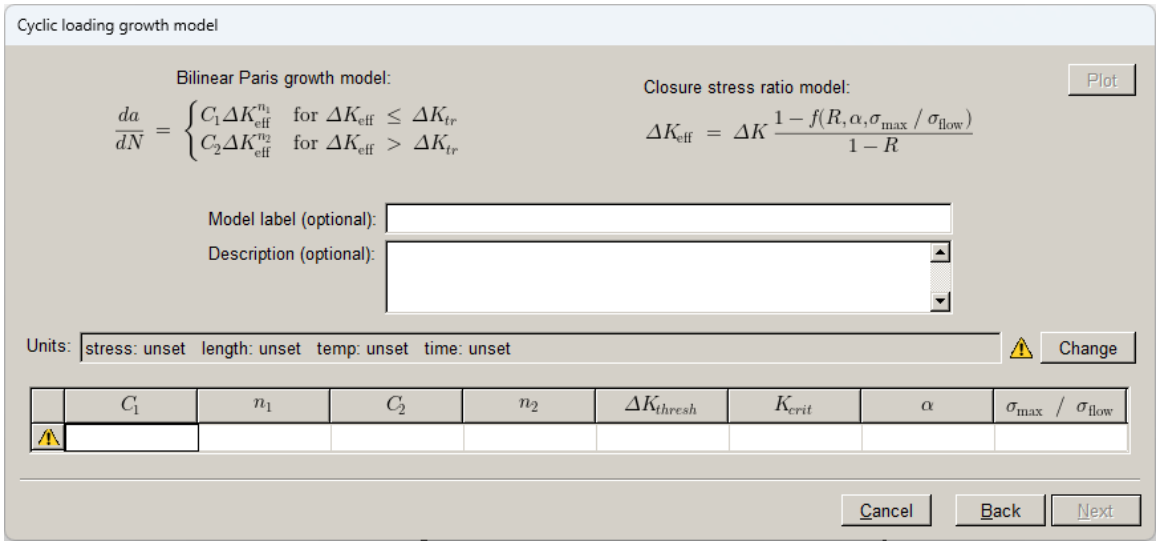


Figure 6.5.49 Dialog for entering temperature independent Bilinear Paris growth and Newman Closure stress ratio model parameters.

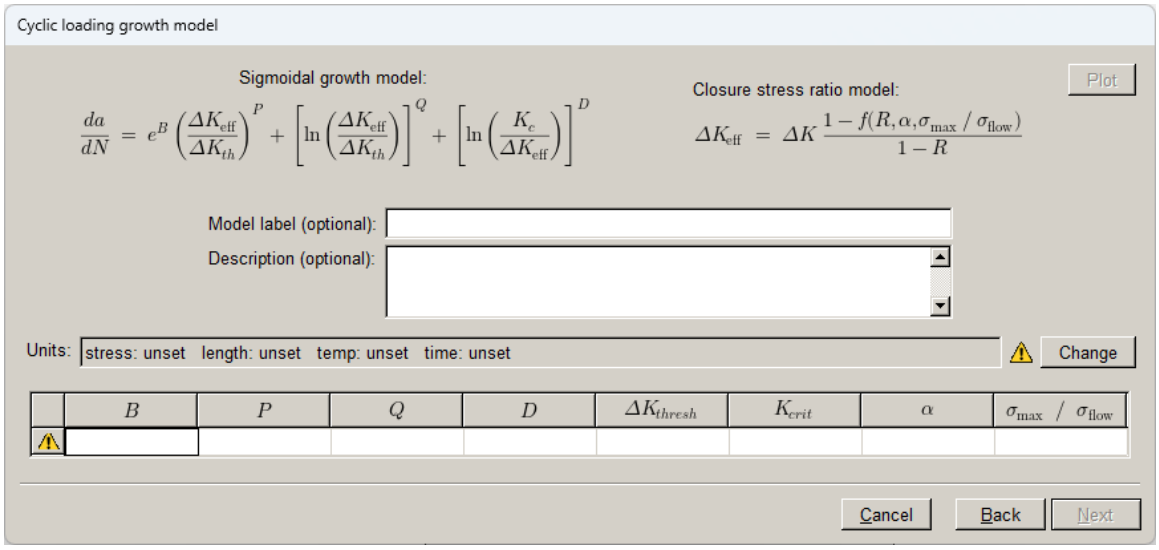


Figure 6.5.50 Dialog for entering temperature independent Sigmoidal growth and Newman Closure stress ratio model parameters.

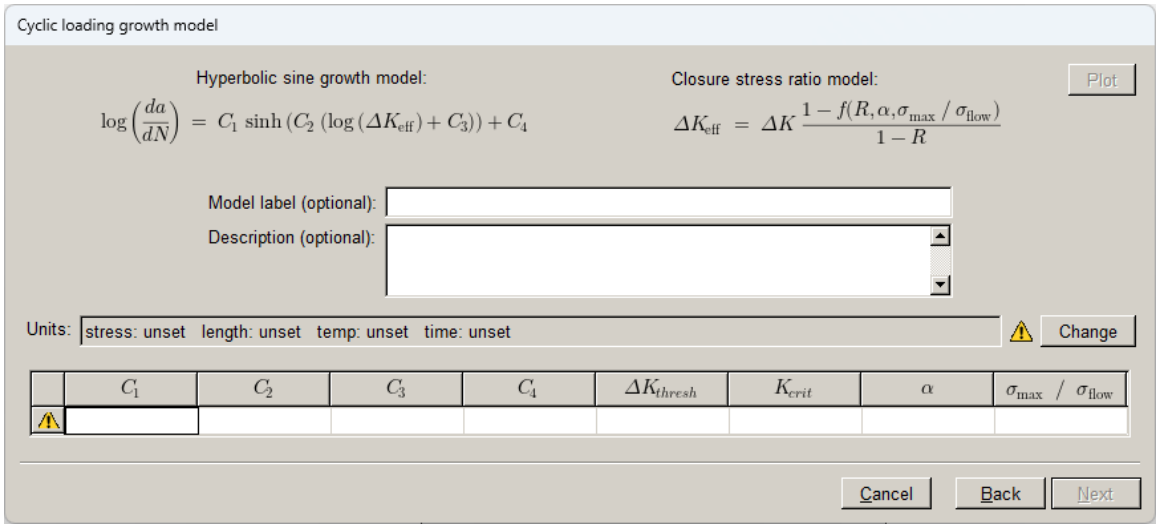


Figure 6.5.51 Dialog for entering temperature independent Hyperbolic sine growth and Newman Closure stress ratio model parameters.

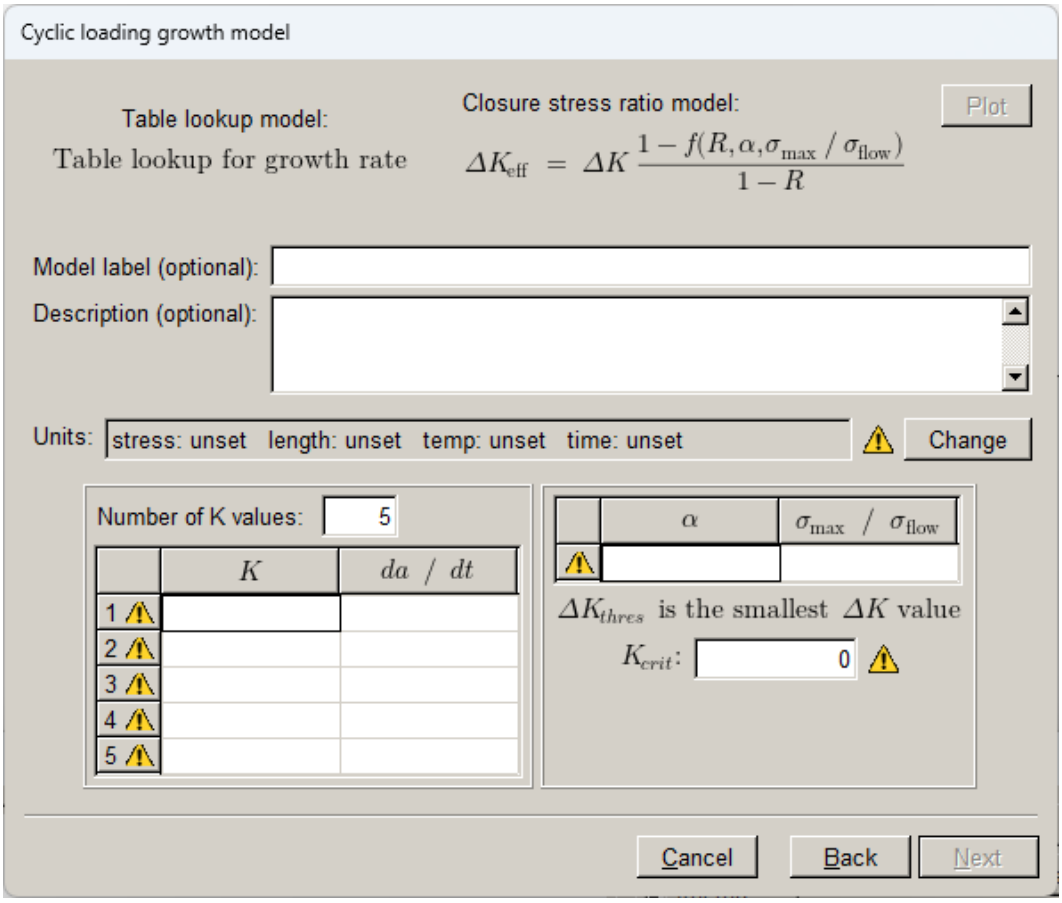


Figure 6.5.52 Dialog for entering temperature independent Table lookup growth and Newman Closure stress ratio model parameters.

6.5.1.4.1.9 Table Lookup Stress Ratio model

For a tabular stress ratio model, parameters for the associated growth rate model are specified for various R values. For example, the specified values for a Paris model with tabular stress ratio model would be

$$R_1, (C, n, \Delta K_{threshold}, K_{critical})_1$$
$$R_2, (C, n, \Delta K_{threshold}, K_{critical})_2$$
$$R_3, (C, n, \Delta K_{threshold}, K_{critical})_3$$
$$\text{M}$$

Linear interpolation is performed to find the crack growth rate for any R value between two specified values R_i and R_{i+1}. That is

$$\frac{da}{dN} = \frac{da}{dN}_i + \frac{R - R_i}{R_{i+1} - R_i}\left(\frac{da}{dN}_{i+1} - \frac{da}{dN}_i\right)$$

If R is smaller than the smallest R specified, the growth rate corresponding to the smallest specified R is used (if the user specifies that the values are capped, otherwise extrapolation is done). Likewise, if R is greater than the greatest R specified, the growth rate corresponding to the largest specified R is used (if the user specifies that the values are capped, otherwise extrapolation is done).

For Table lookup growth rate and Table lookup stress ratio, FRANC3D interpolates using both tables to find a crack growth rate for a given ΔK and R.

Given query values ΔK_{query} and R_{query}, crack growth rates are extracted from the table using the following algorithm:

1. R_l and R_u values are found that bracket the query value, $R_l \le R_{query} \le R_u$.

2. ΔK_{ll}, ΔK_{lu}, ΔK_{ul}, and ΔK_{uu} are found where $\Delta K_{ll} \le \Delta K_{query} \le \Delta K_{lu}$ and $\Delta K_{ul} \le \Delta K_{query} \le \Delta K_{uu}$, with ΔK_{ll} and ΔK_{lu} values associated with R_l and ΔK_{ul} and ΔK_{uu} values associated with R_u.

3. Linear interpolation in log/log space is used to find intermediate growth rate data,

$$\log\left(\frac{da}{dN}_l\right) = \log\left(\frac{da}{dN}(\Delta K_{ll}, R_l)\right) + \log\left(\frac{\frac{da}{dN}(\Delta K_{lu}, R_l)}{\frac{da}{dN}(\Delta K_{ll}, R_l)}\right)\frac{\log\left(\Delta K_{query}/\Delta K_{ll}\right)}{\log\left(\Delta K_{lu}/\Delta K_{ll}\right)}$$

and

$$\log\left(\frac{da}{dN}_u\right) = \log\left(\frac{da}{dN}(\Delta K_{ul}, R_u)\right) + \log\left(\frac{\frac{da}{dN}(\Delta K_{uu}, R_u)}{\frac{da}{dN}(\Delta K_{ul}, R_u)}\right)\frac{\log\left(\Delta K_{query}/\Delta K_{ul}\right)}{\log\left(\Delta K_{uu}/\Delta K_{ul}\right)}$$

4. Finally, normal linear interpolation is used to find the final growth rate

$$\frac{da}{dN} = \frac{da}{dN}_l + \left(\frac{da}{dN}_u - \frac{da}{dN}_l\right)\frac{(R_{queary} - R_l)}{(R_u - R_l)}$$

Note that, as with all interpolation procedures, the accuracy of the results is dependent on the density of the data available for interpolation. Very sparse tables can lead to inaccurate and, in some cases, unrealistic crack growth predictions.

For each of the growth models paired with Table lookup stress ratio selected, the dialogs are shown in Figs 6.5.53 – 57. The data should be supplied in ascending order with respect to R.

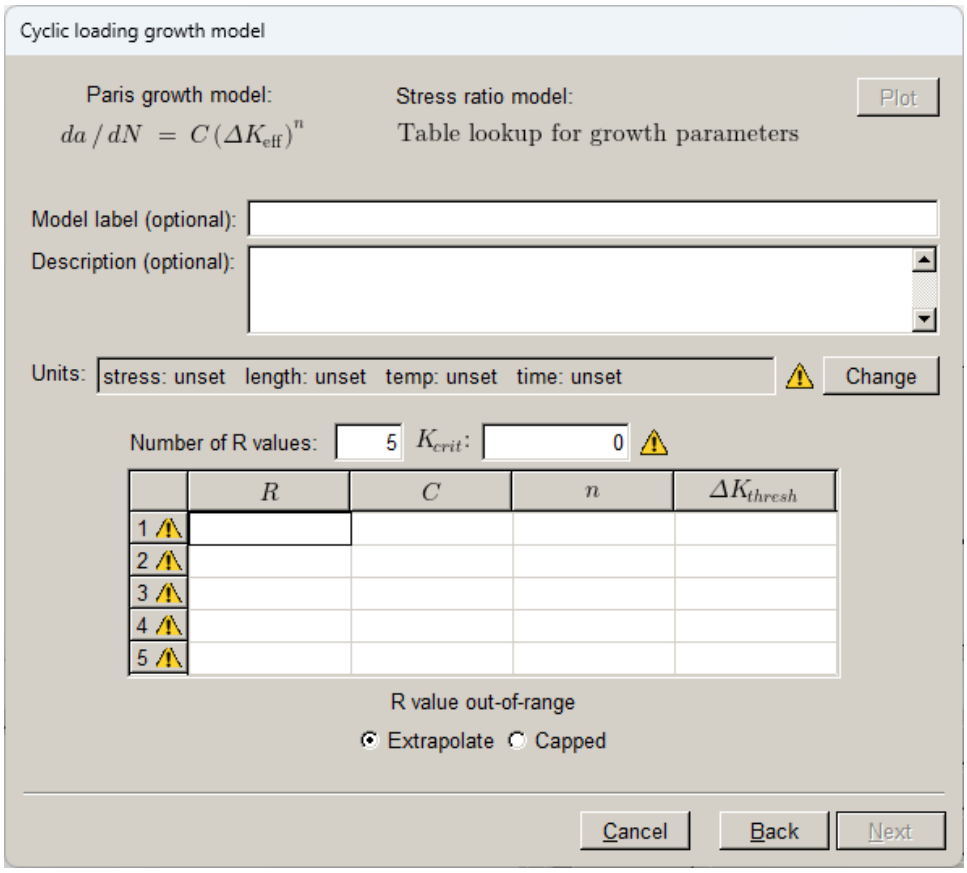


Figure 6.5.53 Dialog for entering temperature independent Paris growth and Table lookup stress ratio model parameters.

$K_{critical}$ is not a function of R; a single value is entered in the dialog (Fig 6.5.53) for all R values. The value is saved in the *.fdb* and session *.log* files for each R value; it should be constant, however.

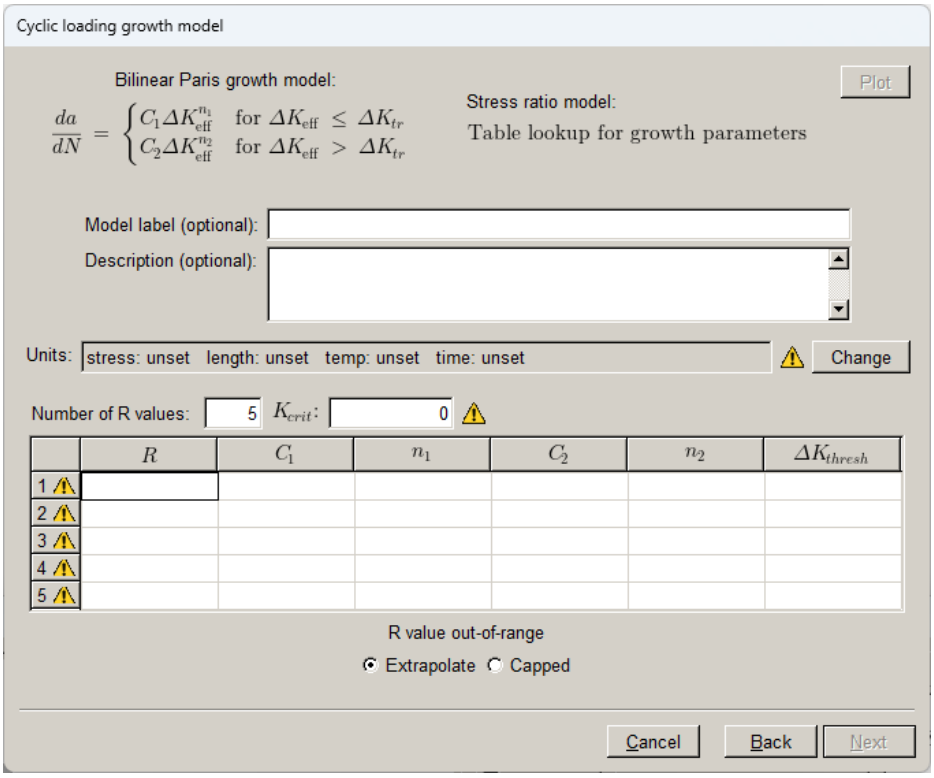


Figure 6.5.54 Dialog for entering temperature independent Bilinear Paris growth and Table lookup stress ratio model parameters.

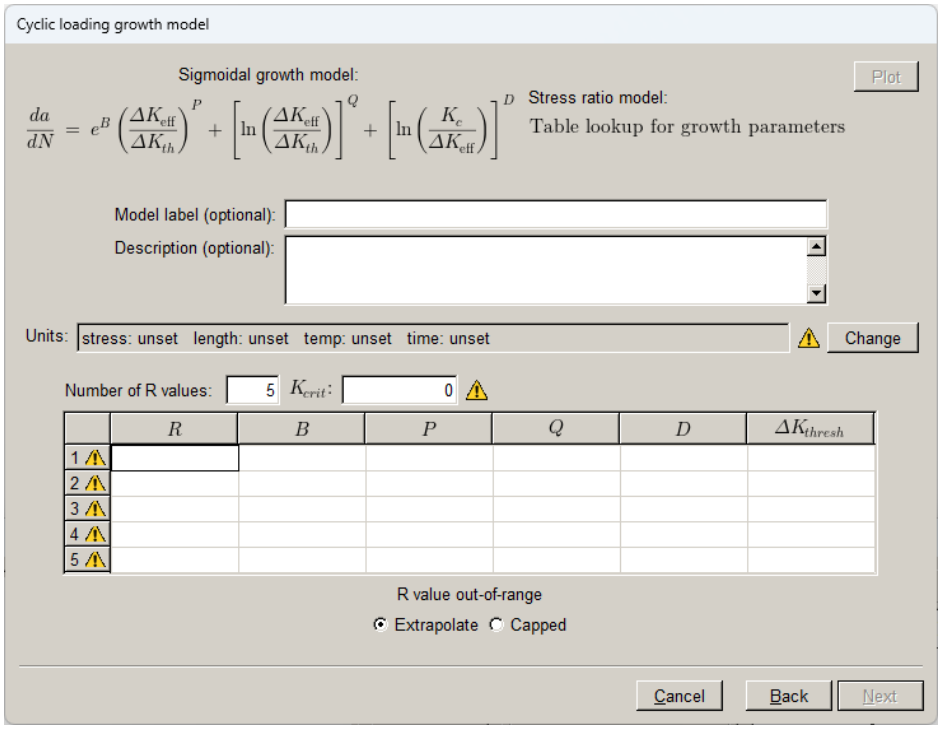


Figure 6.5.55 Dialog for entering temperature independent Sigmoidal growth and Table lookup stress ratio model parameters.

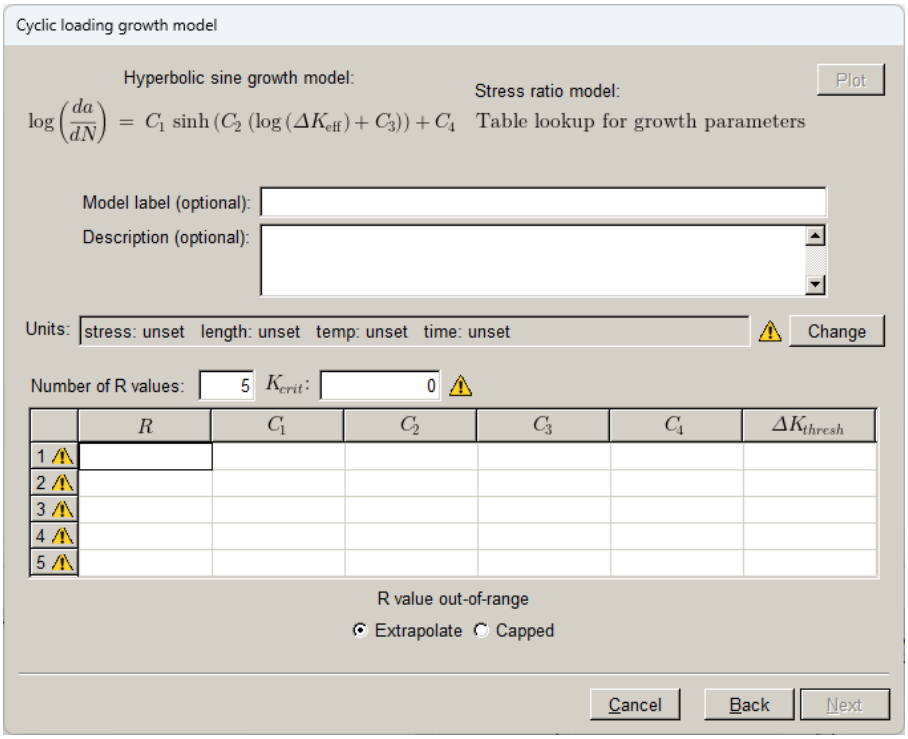


Figure 6.5.56 Dialog for entering temperature independent Hyperbolic sine growth and Table lookup stress ratio model parameters.

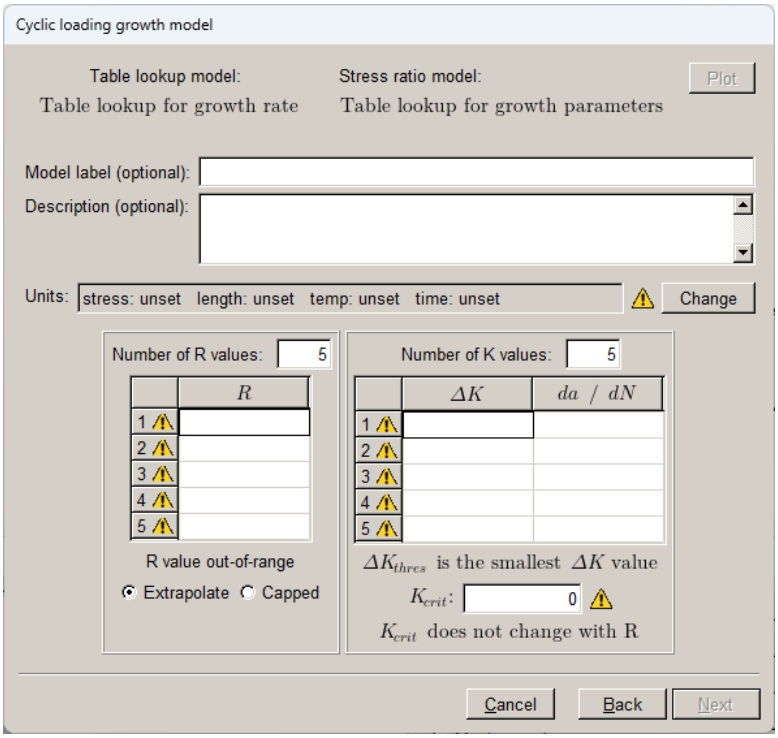


Figure 6.5.57 Dialog for entering temperature independent Table lookup growth and Table lookup stress ratio model parameters.

6.5.1.4.1.10 NASGRO Version 4 equation

NASGRO is a fatigue life prediction program initially developed at the NASA Johnson Space Flight Center and now supported and maintained by Southwest Research Institute. Built into the NASGRO program is an analytical crack growth rate equation that incorporates several features observed in real materials such as sensitivity to near-threshold and near-critical growth, sensitivity to the R-ratio, and small crack sensitivity. The cost for flexibility in the growth model is that there are sixteen fitting parameters. Some are common material properties (e.g., yield stress), but most must be determined by performing regression analyses on empirical data.

The NASGRO package includes a database of pre-computed parameters for a wide range of materials and product forms. However, as of NASGRO version 4, this database is only accessible to members of the NASGRO consortium. As a practical matter, this means that this model will be most useful to users who have access to the NASGRO program and can access the parameters for a material of interest (see NASGRO version 3 below).

The NASGRO version 4 crack growth rate equation, Fig 6.5.58, is:

$$\frac{da}{dN} = C\left[\left(\frac{1-f}{1-R}\right)\right]^{n} \frac{\left(1-\frac{\Delta K_{th}}{\Delta K}\right)^{p}}{\left(1-\frac{\Delta K_{\max}}{K_c}\right)^{q}}$$

where C, n, p, and q are empirical constants. C and n are like the constants in the Paris and Walker equations. However, while C in the Paris model should be selected to correspond to the R of interest, and C in the Walker equation should be for $R = 0$, in the NASGRO equation the C should correspond to a high R where closure effects are no longer significant (usually an R of about 0.7)

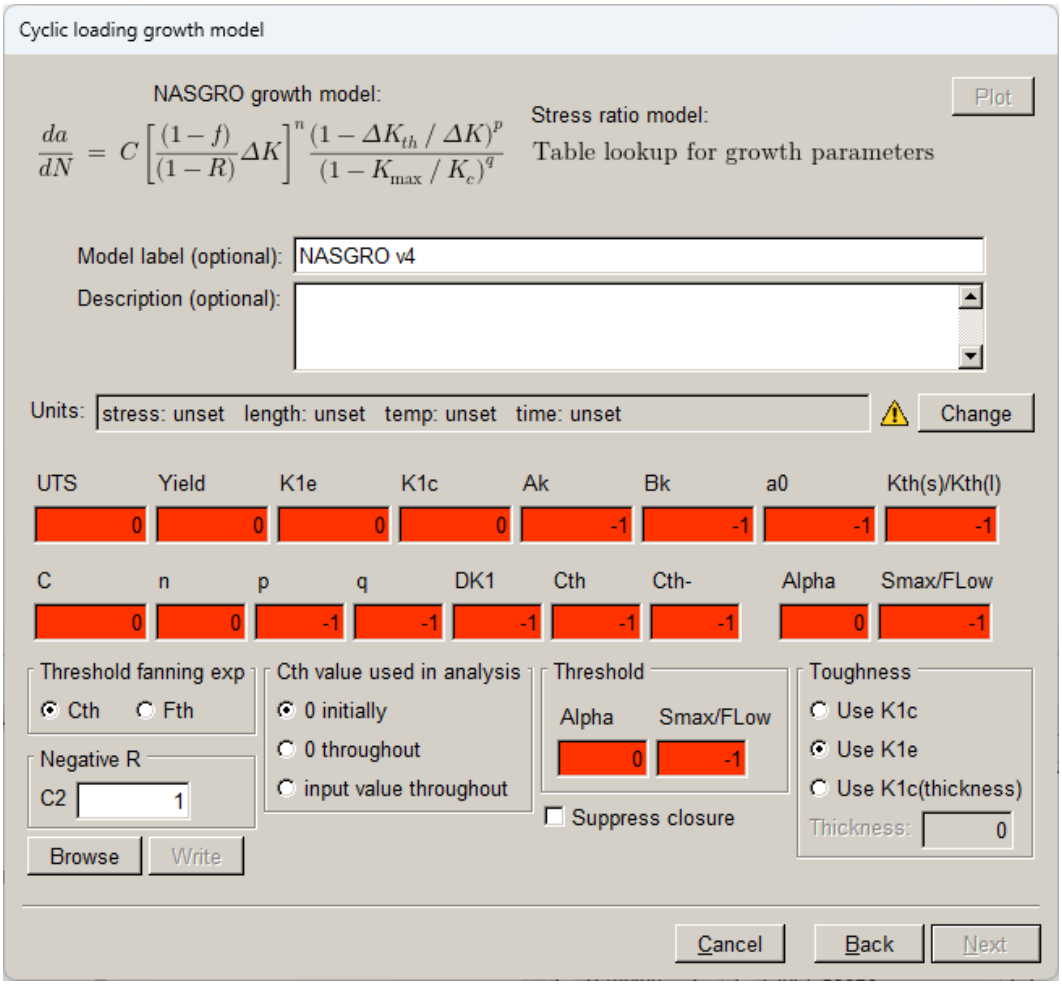


Figure 6.5.58 NASGRO version 4 growth rate model dialog.

f is the function associated with Newman closure. ΔK_{th} is the threshold stress intensity factor range below which there is no crack growth. It is approximated by the following empirical equations:

$$\Delta K_{th} = \begin{cases} \Delta K_1^* \left[\frac{1-R}{1-f}\right]^{(1+RC_{th}^p)} \Big/ (1-A_0)^{(1-R)C_{th}^p} & R \geq 0 \\ \Delta K_1^* \left[\frac{1-R}{1-f}\right]^{(1+RC_{th}^m)} \Big/ (1-A_0)^{(C_{th}^p - RC_{th}^m)} & R < 0 \end{cases}$$

where

$$\Delta K_1^* = \Delta K_1 \left[\frac{a}{a+a_0}\right]^{1/2}$$

and C_{th} is an empirical constant, ΔK_1 is the observed threshold for a high *R*-ratio, and a_0 is a small crack parameter (usually 0.0015in, 0.0381 mm).

K_c is the fracture toughness. In FRANC3D, one has the choice to use the plane strain fracture toughness, K_{Ic}, the part-through toughness, K_{Ie}, or compute it from:

$$K_c = K_{Ic}(1 + B_k e^{-(A_k t/t_0)^2})$$

where

$$t_0 = 2.5\left(K_{Ic}/\sigma_{ys}\right)^2$$

A_k and B_k are empirical constants, and *t*, the "thickness", is specified by the user (in 3D applications the notion of thickness is not well-defined, and engineering judgment must be used to set this value).

More details about this model can be found in the NASGRO theory manual. Note that FRANC3D expects consistent units for the model parameters; in particular, the length units for *K* and C must be the same.

The **Write** button allow one to save the parameters in this dialog to a file. The **Browse** button allows one to read this saved data from a file.

6.5.1.4.1.11 NASGRO Version 3 equation

NASGRO version 3 is an earlier version of the NASGRO crack growth rate equation. The current version of the equation (version 4) does a better job of modeling observed crack growth rates for some conditions. This earlier version is included in FRANC3D because version 3 of NASGRO, the last public domain version, included a database of equation parameters for a wide variety of materials and product forms. That database is accessible from within FRANC3D.

The NASGROv3 crack growth rate equation is slightly different than version 4 equation. The base equation is the same:

$$\frac{da}{dN} = C\left[\left(\frac{1-f}{1-R}\right)\right]^n \frac{\left(1-\frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1-\frac{\Delta K_{\max}}{K_c}\right)^q}$$

The difference has to do with how threshold values are computed. In version 3, the expression is

$$\Delta K_{th} = \Delta K_0 \sqrt{\frac{a}{a + a_0}} \Bigg/ \left(\frac{1 - f}{(1 - A_0)(1 - R)} \right)^{1 + C_{th} R}$$

where a is the crack length and a_0 has a fixed value of 0.0015 in (0.0381 mm).

Fig 6.5.59 shows the dialog for entering the NASGRO model parameters. The NASGRO version 3 material database contains data for a wide variety of materials, and this data is included in FRANC3D and accessible by selecting the **Material Properties Database** button.

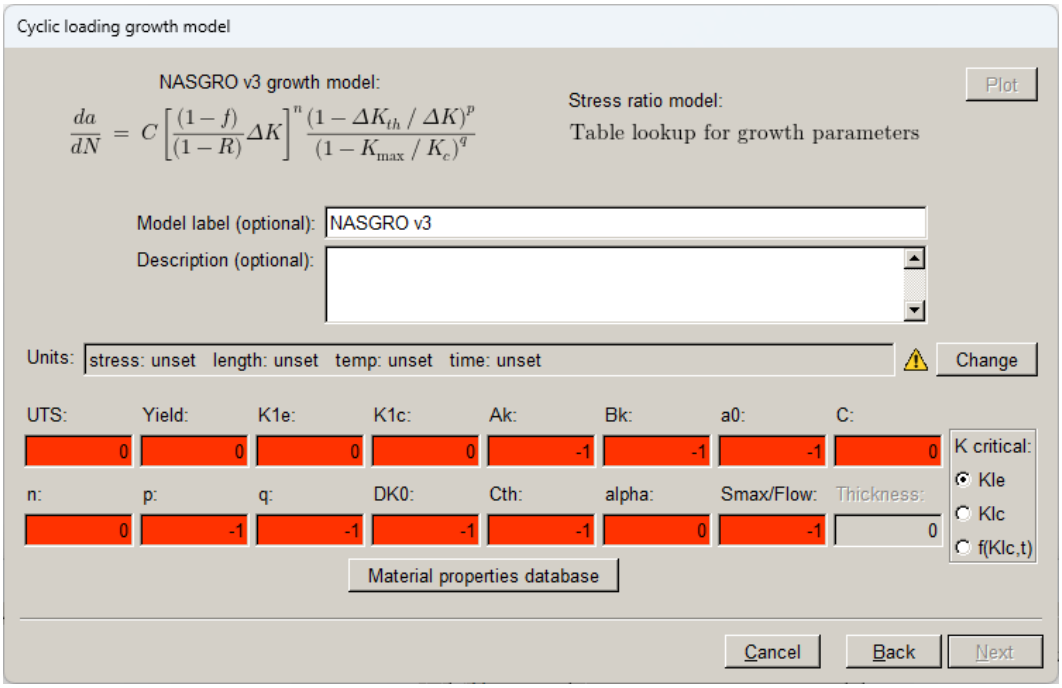
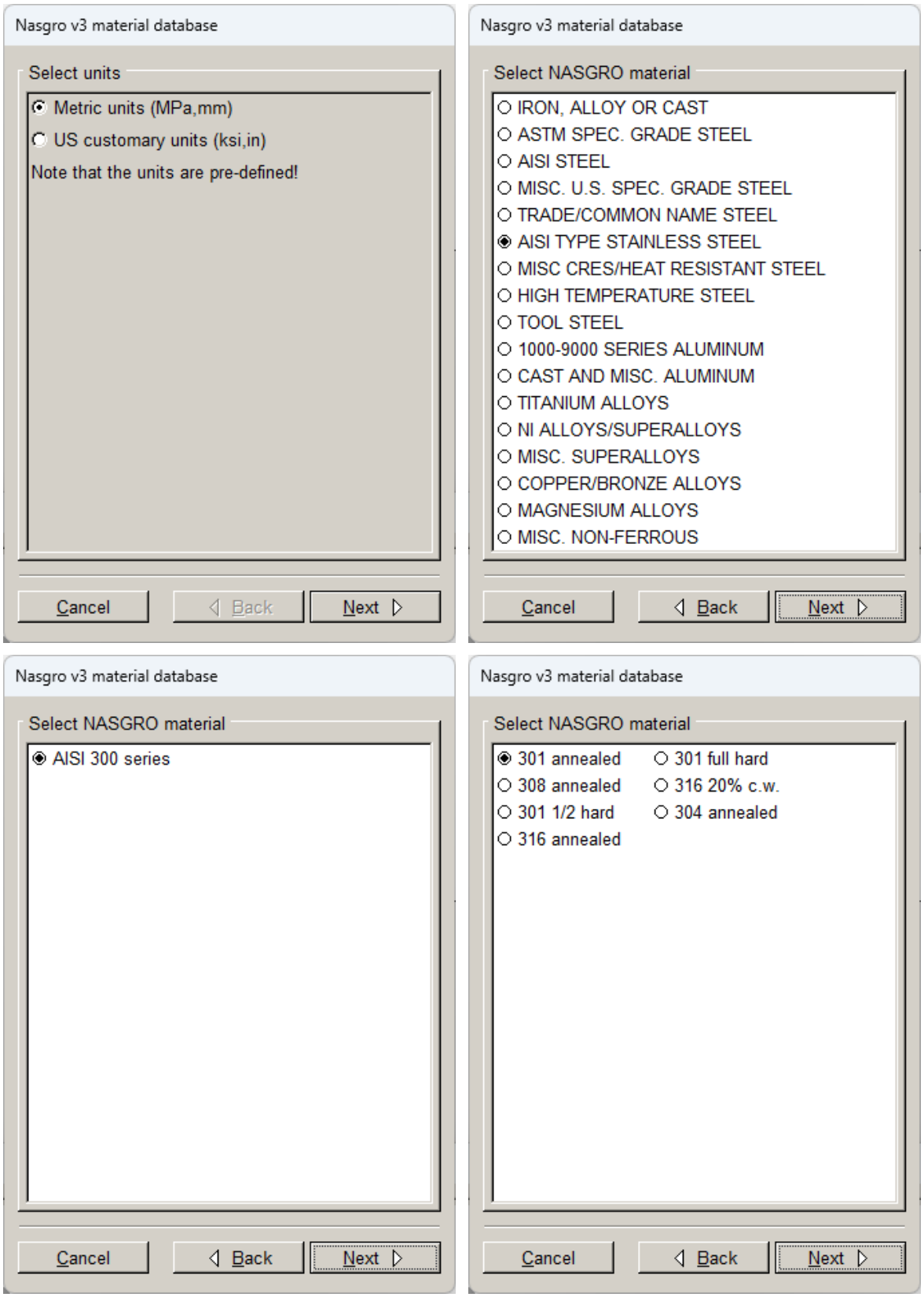


Figure 6.5.59 NASGRO version 3 growth rate model.

The dialog boxes shown in Fig 6.5.60 allow the analyst to automatically fill in the required NASGRO v3 parameter fields from this database.


Nasgro v3 material database
Select units
Metric units (MPa,mm)
US customary units (ksi,in)
Note that the units are pre-defined!
Cancel
Back
Next
Nasgro v3 material database
Select NASGRO material
IRON, ALLOY OR CAST
ASTM SPEC. GRADE STEEL
AISI STEEL
MISC. U.S. SPEC. GRADE STEEL
TRADE/COMMON NAME STEEL
AISI TYPE STAINLESS STEEL
MISC CRES/HEAT RESISTANT STEEL
HIGH TEMPERATURE STEEL
TOOL STEEL
1000-9000 SERIES ALUMINUM
CAST AND MISC. ALUMINUM
TITANIUM ALLOYS
NI ALLOYS/SUPERALLOYS
MISC. SUPERALLOYS
COPPER/BRONZE ALLOYS
MAGNESIUM ALLOYS
MISC. NON-FERROUS
Cancel
Back
Next
Nasgro v3 material database
Select NASGRO material
AISI 300 series
Cancel
Back
Next
Nasgro v3 material database
Select NASGRO material
301 annealed
308 annealed
301 1/2 hard
316 annealed
301 full hard
316 20% c.w.
304 annealed
Cancel
Back
Next

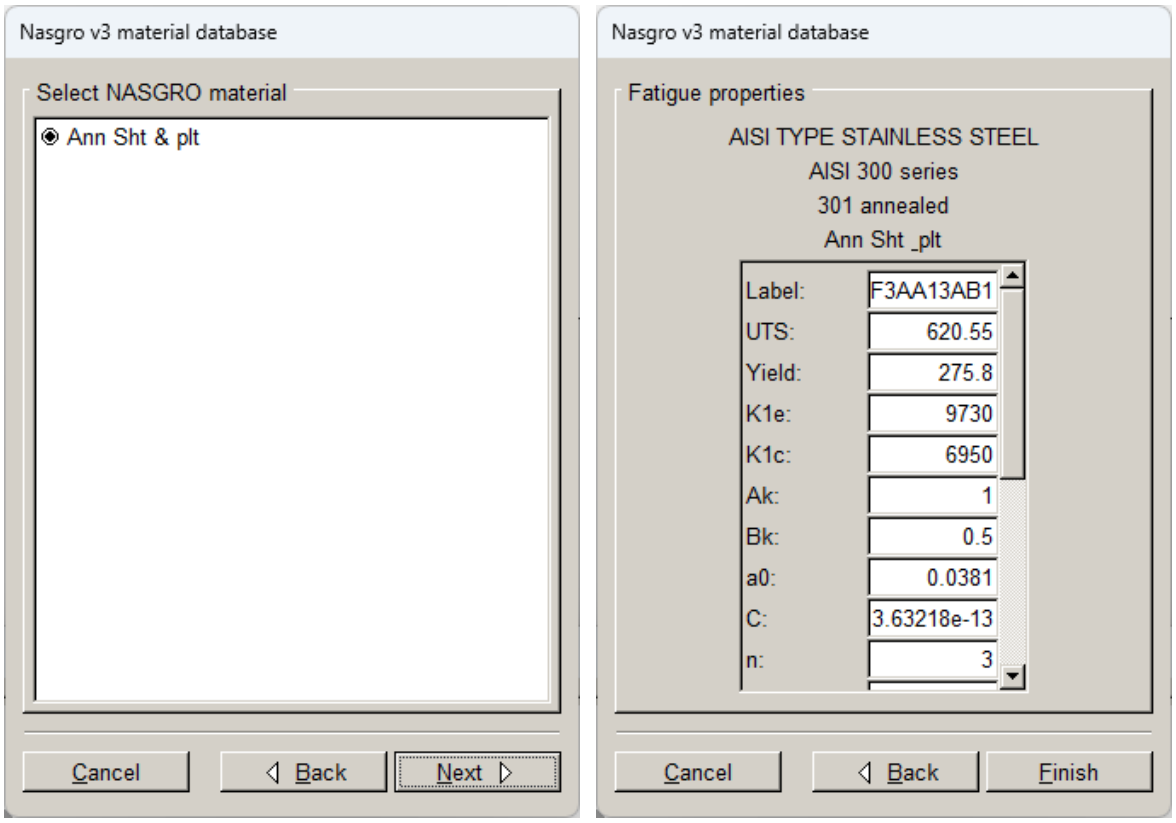


Figure 6.5.60 NASGRO version 3 material database dialog boxes

6.5.1.4.1.12 Modified Hartman-Schijve

The Modified Hartman-Schijve growth model is shown in Fig 6.5.61.

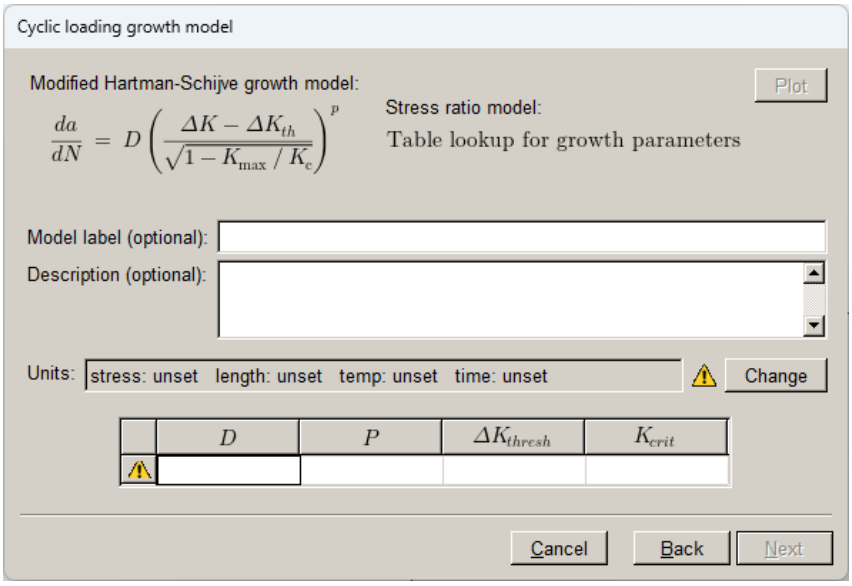


Figure 6.5.61 Modified Hartman-Schijve model.

6.5.1.4.1.13 User-defined model

The user-defined option will be active if the user has included the Python extensions and appropriate Python functions. This is described in Section 14.6.

6.5.1.4.1.14 Temperature dependent growth rate

Temperature dependent growth rate models are implemented in a tabular manner. Growth model parameters are specified for several different temperatures, and interpolation is performed among the computed da/dN values. For example, the specified data for a temperature dependent paired Paris/Walker growth model would be

$$\begin{gathered} Temperature_1, (C, n, \Delta K_{threshold}, K_{critical}, m+, m-)_1 \\ Temperature_2, (C, n, \Delta K_{threshold}, K_{critical}, m+, m-)_2 \\ Temperature_3, (C, n, \Delta K_{threshold}, K_{critical}, m+, m-)_3 \\ \text{M} \end{gathered}$$

Three different interpolation options are available: closest, next highest, and linear interpolation. For closest interpolation, da/dN is determined for the tabular temperature closest to the input value. For next highest interpolation, da/dN is determined for the next tabular temperature higher than the input value. The expression for linear interpolation between two tabular values *i* and *i*+1 is

$$\frac{da}{dN} = \frac{da}{dN}_i + \frac{Temperature - Temperature_i}{Temperature_{i+1} - Temperature_i}\left(\frac{da}{dN}_{i+1} - \frac{da}{dN}_i\right)$$

If the input temperature is smaller than the smallest temperature specified, the growth rate corresponding to the smallest specified temperature is used (if the user specifies that the values are capped, otherwise extrapolation is done).. Likewise, if the input temperature is greater than the greatest temperature specified, the growth rate corresponding to the largest specified temperature is used (if the user specifies that the values are capped, otherwise extrapolation is done).

The temperature dependent dialogs for the Paris growth model are shown in Figs 6.5.62 – 65. Note that data should be supplied in ascending order with respect to Temperature.

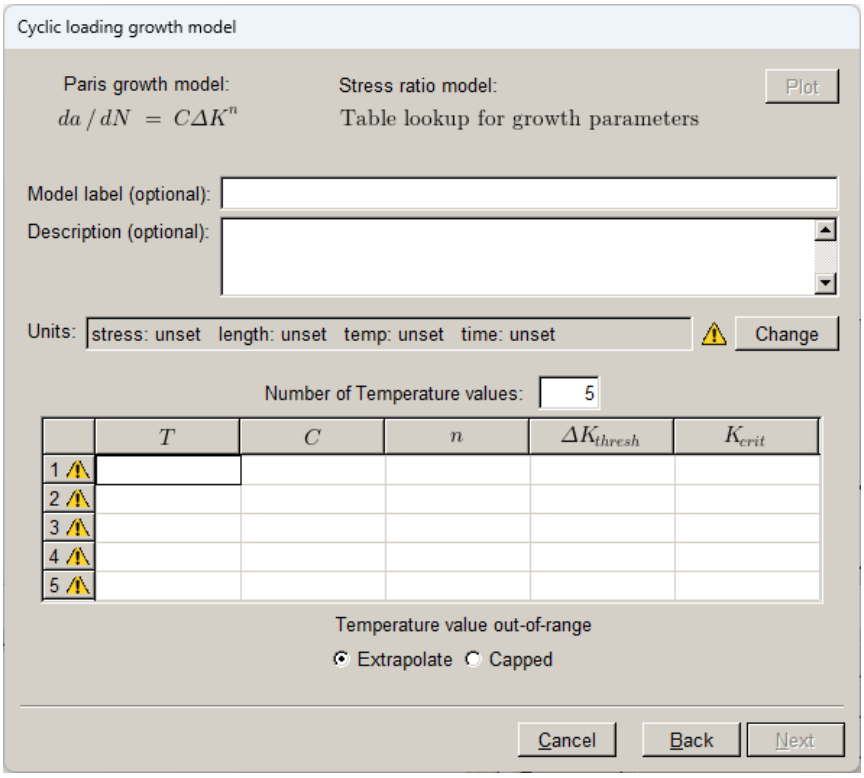


Figure 6.5.62 Paris temperature dependent model with no stress ratio model.

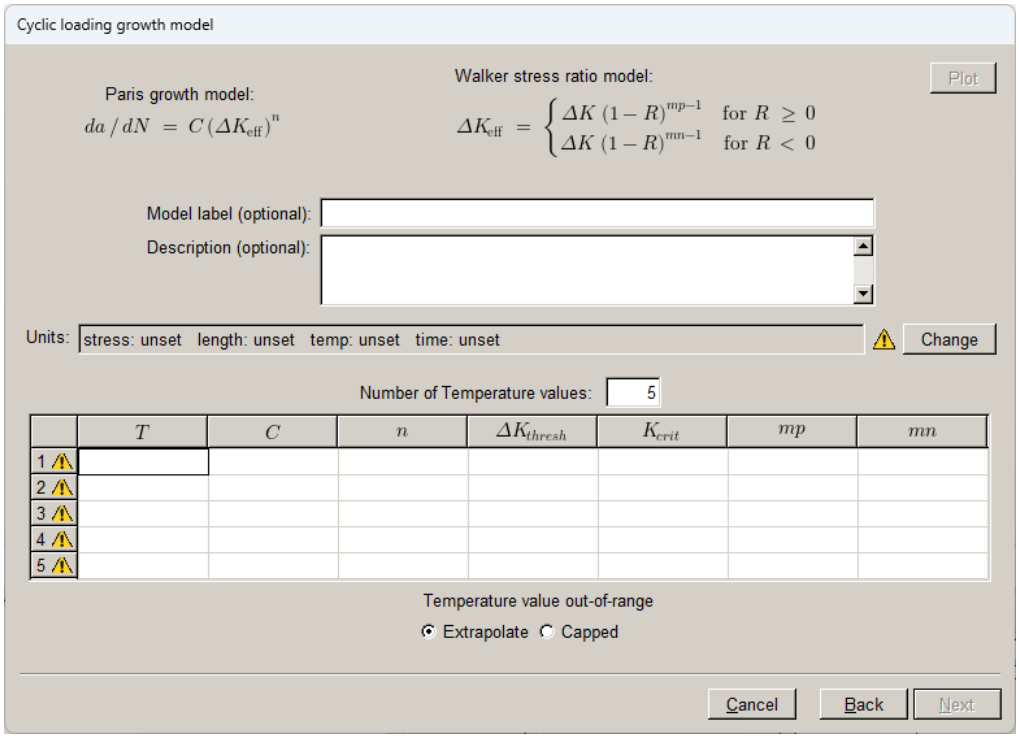


Figure 6.5.63 Paris temperature dependent model with Walker stress ratio model.

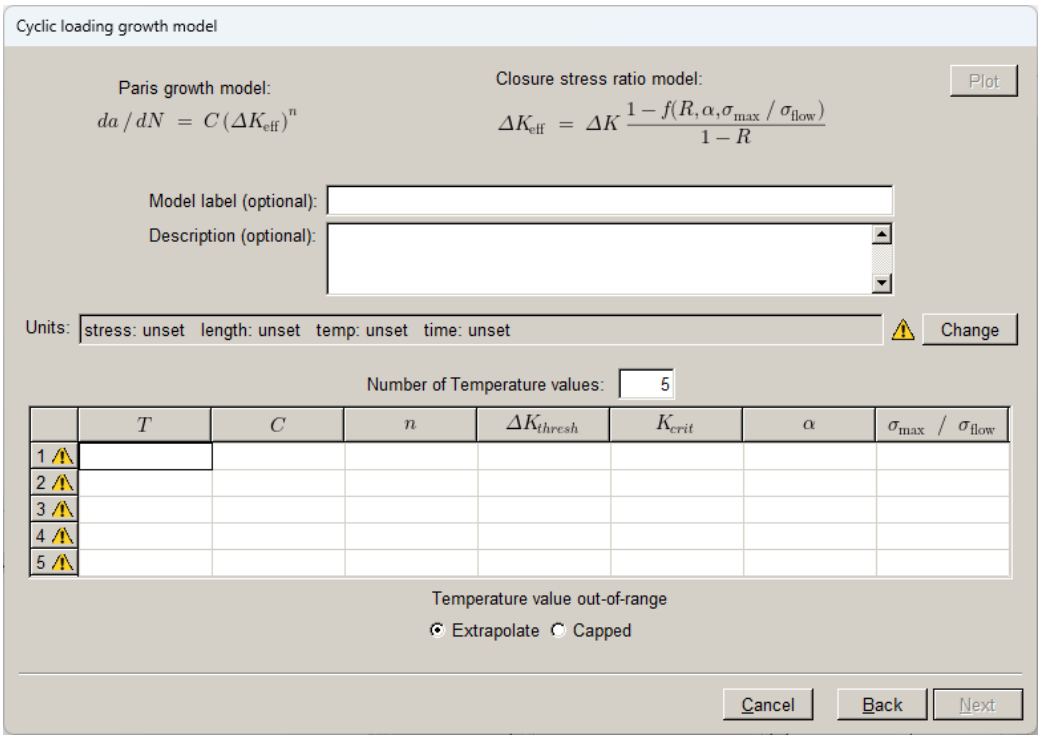


Figure 6.5.64 Paris temperature dependent model with Newman closure stress ratio model.

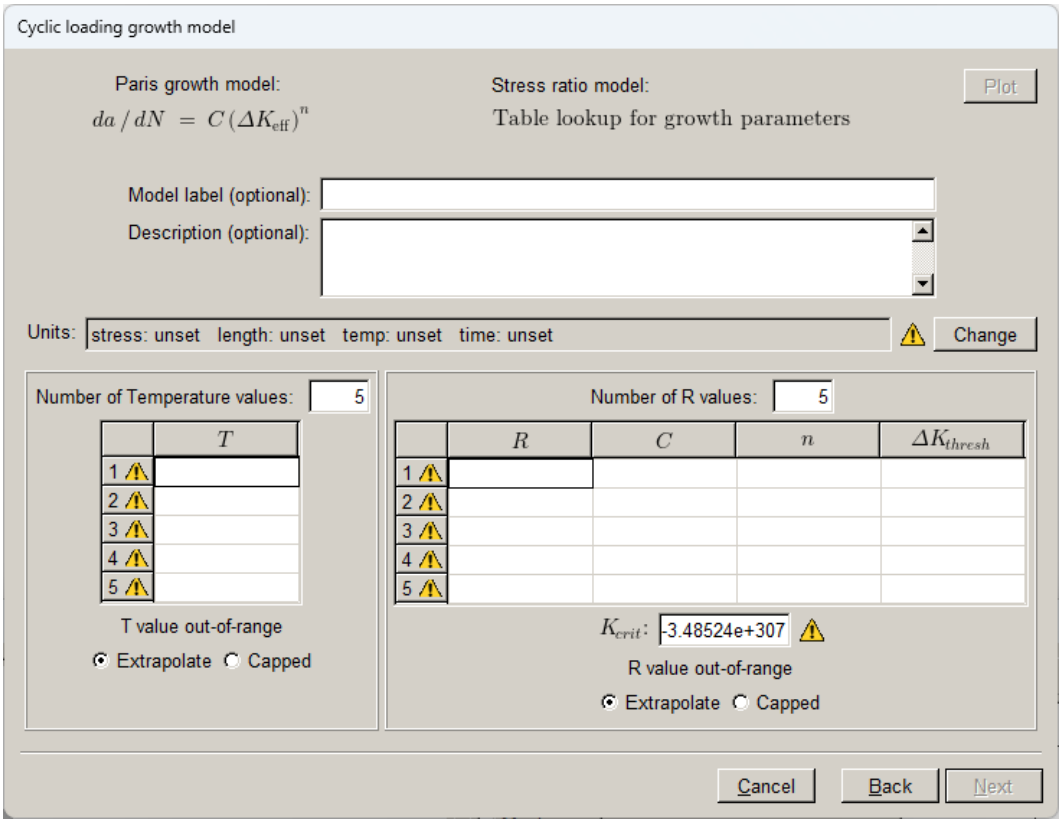


Figure 6.5.65 Paris temperature dependent model with Table lookup stress ratio model.

The temperature dependent dialogs for Bi-linear Paris, Sigmoidal, Hyperbolic sine and Table lookup are all similar.

6.5.1.4.2 Growth Rate Model: New Model – time dependent loading

There are five time-dependent growth models, Fig 6.5.66; these are described in the subsections below. Each of the models can be temperature dependent or independent, Fig 6.5.67, like the cyclic growth rate models.

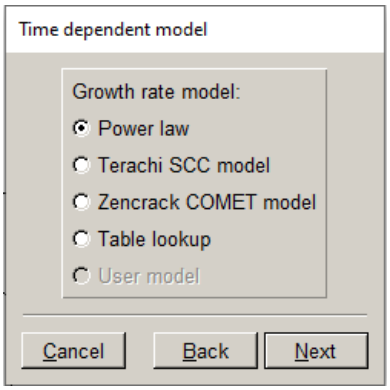


Figure 6.5.66 Dialog to select a time dependent growth model type.

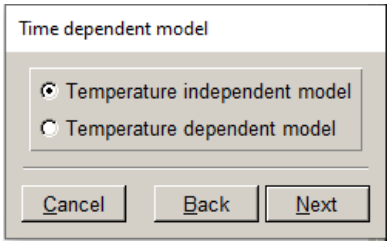


Figure 6.5.67 Dialog to specify temperature dependent or independent for time dependent growth model.

6.5.1.4.2.1 Time Dependent Power Law

The dialog to specify the power law time dependent growth and temperature independent model parameters is shown in Fig 6.5.68. Again, as for cyclic loading, Units must be specified. For time dependent rates, the threshold (K_{th}) is an actual K value rather than a ΔK value. If K_{hold} is below this value for any crack front point, there will be no time dependent crack extension for that point.

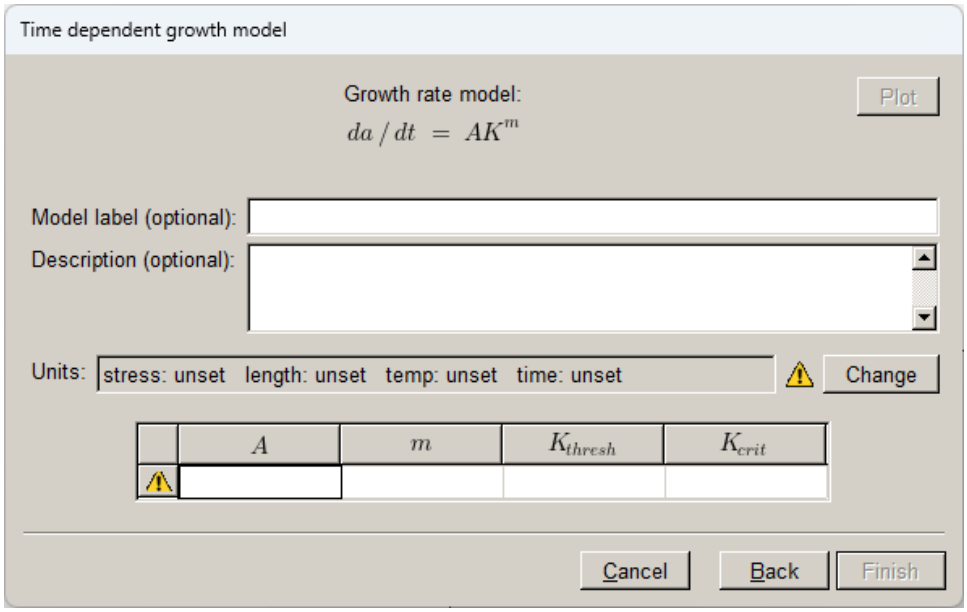


Figure 6.5.68 Dialog box for entering time dependent power law growth model parameters.

6.5.1.4.2.2 Time Dependent Terachi SCC

The dialog to specify the Terachi stress corrosion crack (SCC) time dependent growth model parameters is shown in Fig 6.5.69. The dialog is like that of the Power law model. However, the equation is a function of the material yield stress.

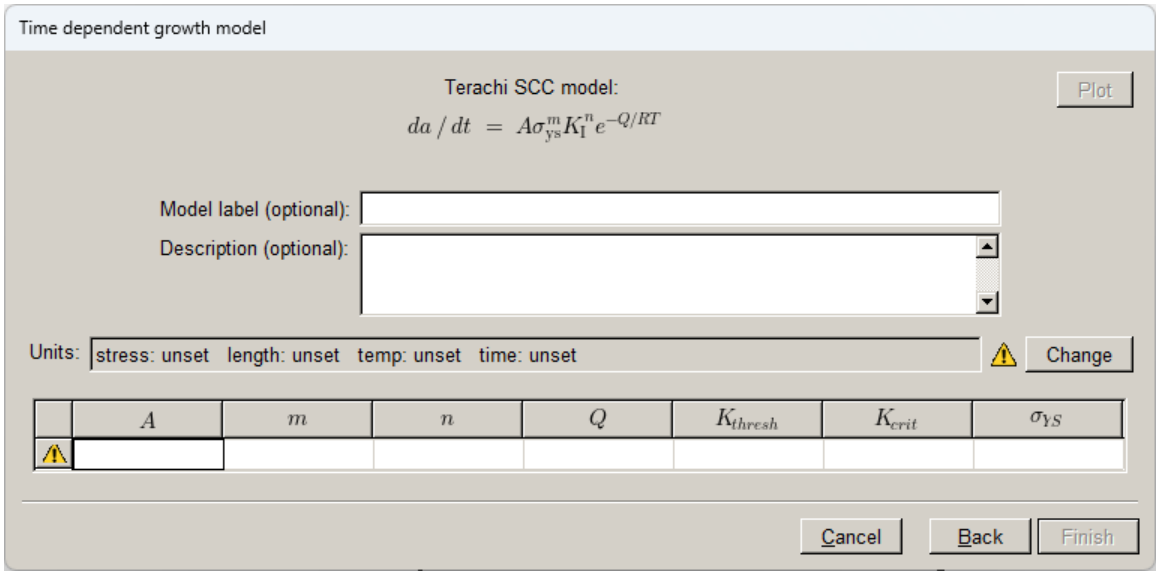


Figure 6.5.69 Dialog box for entering time dependent Terachi SCC growth model parameters.

Yield Stress can be specified as constant or can be given in a table. For the Table option, pairs of numbers are given that specify a depth from the nearest surface and a corresponding yield stress. Linear interpolation is performed within the given values, with the minimum and maximum values assigned to crack front points whose depths fall below or above the given depths, respectively.

6.5.1.4.2.3 Zencrack COMET model

The dialog to specify the Zencrack COMET time dependent growth model parameters is shown in Fig 6.5.70. The exponent is temperature dependent and must be set here.

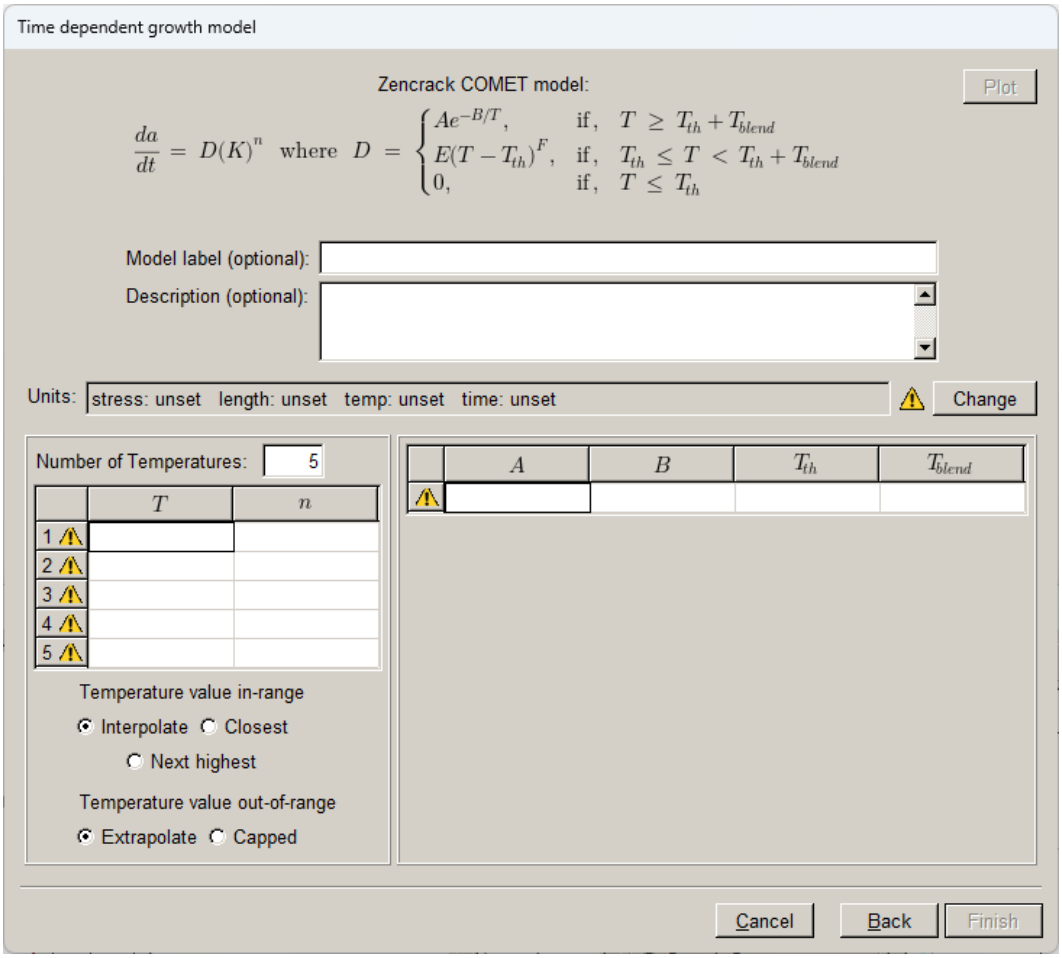


Figure 6.5.70 Dialog box for time dependent Zencrack COMET growth model parameters.

6.5.1.4.2.4 Time Dependent Table Lookup

Time dependent growth can be entered using a Table lookup model, Fig 6.5.71. Interpolation is done based on the K values to obtain the da/dt. Note that data should be supplied in ascending order with respect to K.

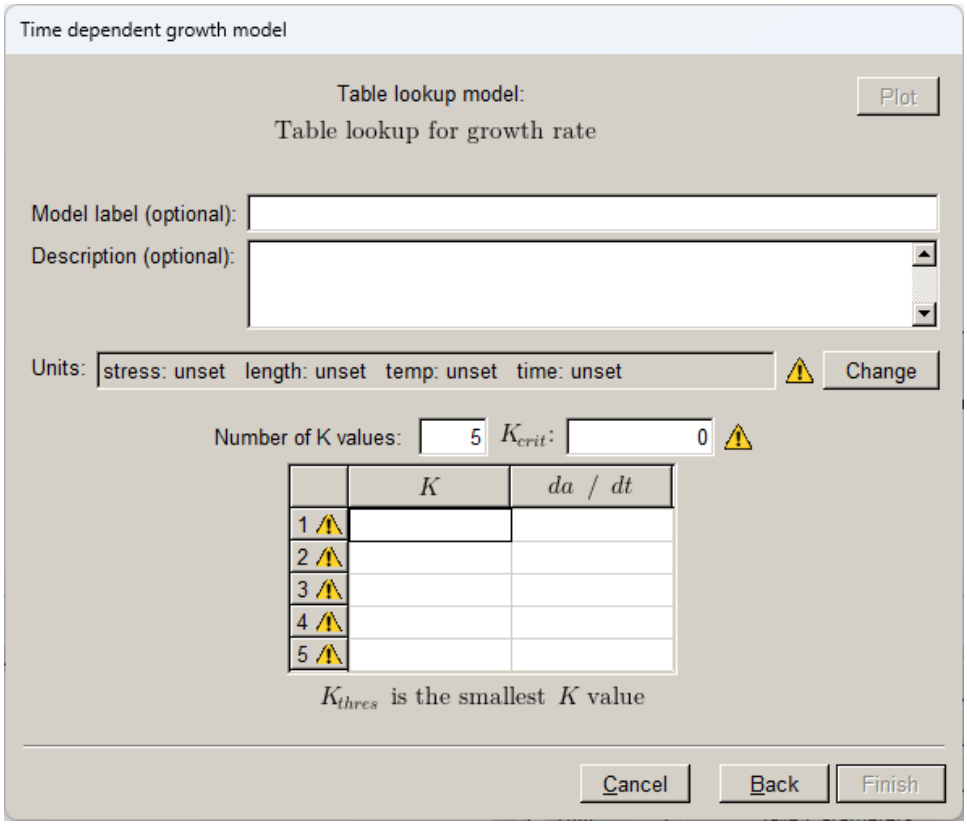


Figure 6.5.71 Dialog box for entering time dependent Table lookup data.

6.5.1.4.2.5 User model

The user-defined option will be active if the user has included the Python extensions and appropriate Python functions. This is described in Section 14.6.

6.5.1.4.2.6 Temperature dependent time dependent growth

As with the cyclic growth rate models, the Power-law time dependent growth model can also be temperature dependent. The dialog shown in Fig 6.5.72 is for the power-law time-dependent model. Fig 6.5.73 shows the dialog for Table lookup.

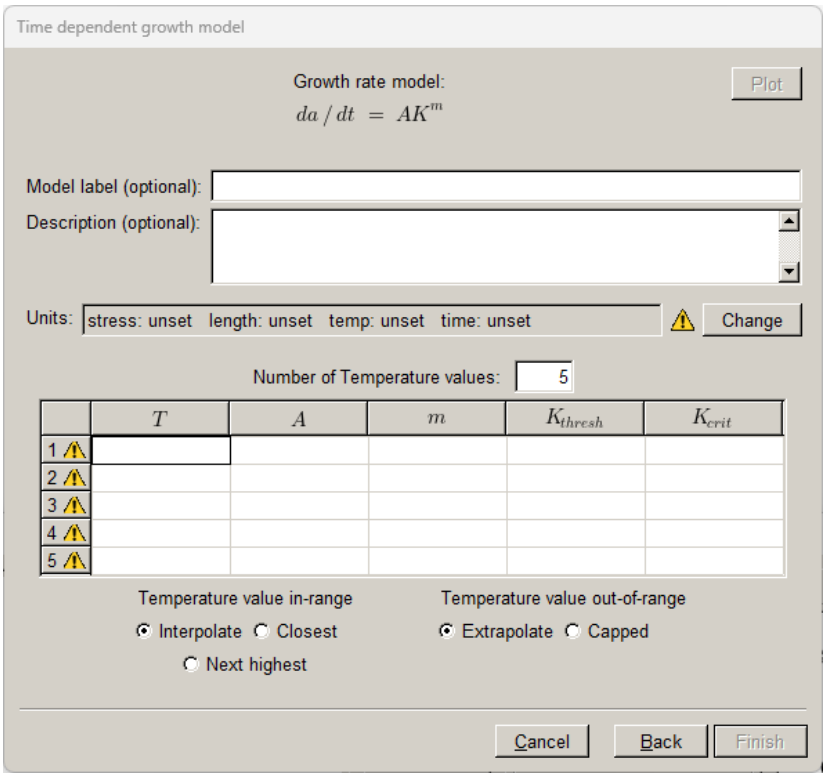


Figure 6.5.72 Dialog for entering temperature dependent time dependent Power law data.

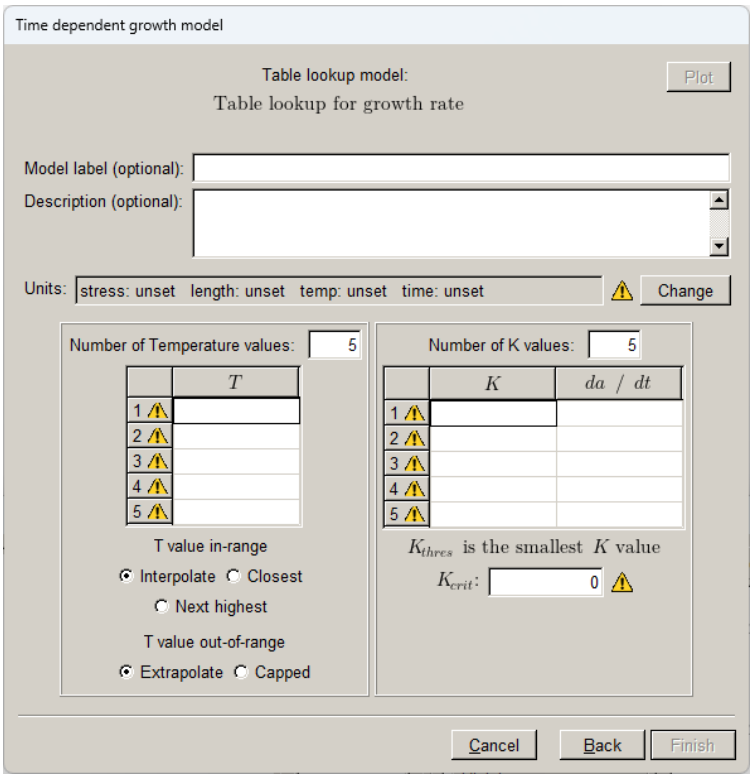


Figure 6.5.73 Dialog for entering temperature dependent time dependent Table lookup data.

6.5.1.4.3 Growth Rate Model: Read From File

The Crack Growth Rate Model **Read From File** button (see Fig 6.5.8) allows one to read a crack growth model that was previously defined and saved to a file. This file contains only crack growth rate information. It is used in conjunction with the **Save To File** button in Section 6.5.1.4.6. It is an ASCII text file and should contain this data "block":

```
CRACK_GROWTH_RATE
(
VERSION: 2
    ....
)
```

6.5.1.4.4 Growth Rate Model: View/Edit da/dN

The Crack Growth Load Schedule **View/Edit da/dN** button (see Fig 6.5.8) displays the previously defined cyclic loading growth rate model in the dialog.

6.5.1.4.5 Growth Rate Model: View/Edit da/dt

The Crack Growth Rate Model **View/Edit da/dt** button (see Fig 6.5.8) displays the previously defined time dependent growth rate model in the dialog.

6.5.1.4.6 Growth Rate Model: Save To File

The Crack Growth Rate Model **Save To File** button (see Fig 6.5.8) allows the user to save the growth model to a file. This file contains only the crack growth rate information; it is used in conjunction with the **Read From File** button in Section 6.5.1.4.3.

6.5.1.5 Subcritical Mixed-mode equivalent K

The equivalent K or ΔK can be defined using only Mode I (KI) or using all three Modes, Fig 6.5.74. If the K^{equiv} is based on all three SIF modes, the sign can be set based on the sign of K_I, K_{II} or K_{III} or it can be set as always positive or always negative.

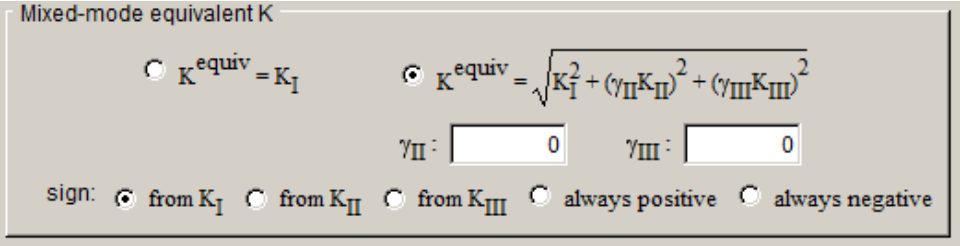


Figure 6.5.74 Equivalent K options.

If the SingleCrystal menu is active, the dialog will include an additional option, Fig 6.5.75, that allows one to use K_{RSS} as K^{equiv}; Section 12 will describe the single crystal options in more detail.

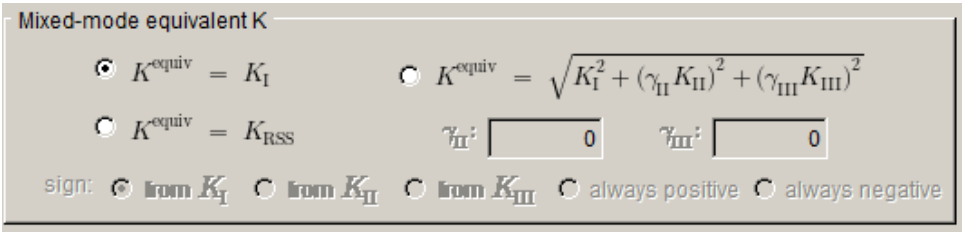


Figure 6.5.75 Single Crystal Equivalent K options.

6.5.1.6 Subcritical Effective dK

There are several options for how ΔK is computed. The default behavior is to simply compute $\Delta K = K_{max} - K_{min}$. In cases where the minimum loading at the crack front is compressive (and contact elements are not explicitly inserted into the crack faces to prevent crack overlap), the computed K_{min} and corresponding stress ratio (R) will be negative. Such crack face overlap is physically unrealistic, but it is customary practice to accept the negative values, and meaningful engineering predictions can be performed with these values provided the appropriate parameters are used with the growth rate and/or stress ratio models.

Optionally, FRANC3D can be instructed to compute a more physically realistic ΔK that only considers portions of a load cycle where the Ks are positive. That is:

$$\Delta K = \begin{cases} K_{max} - K_{min} & \text{if } K_{min} > 0 \\ K_{max} & \text{if } K_{min} \le 0 \end{cases}$$

Effective values of the stress intensity factor range and R account for the situation where the crack is closed during a portion (or all) of a load cycle. The default behavior in FRANC3D is to check for a closed crack for the complete cycle and to set the stress intensity factor range and R to zero:

$$\Delta K_{effective}^{equivalent} = \begin{cases} 0 & K_{\max}^{equivalent} \le 0 \\ \Delta K^{equivalent} & K_{\max}^{equivalent} > 0 \end{cases}$$

and

$$R_{effective}^{equivalent} = \begin{cases} 0 & K_{\max}^{equivalent} \le 0 \\ R^{equivalent} & K_{\max}^{equivalent} > 0 \end{cases}.$$

This default behavior allows for negative *R* values.

FRANC3D allows the option to truncate the stress intensity factor range to eliminate the negative (closed) portion of the cycle. In this case, the expressions are:

$$\Delta K_{effective}^{equivalent} = \begin{cases} 0 & K_{\max}^{equivalent} \le 0 \\ \Delta K^{equivalent} & K_{\min}^{equivalent} \ge 0 \\ K_{\max}^{equivalent} & K_{\min}^{equivalent} < 0 \end{cases}$$

$$R_{effective}^{equivalent} = \begin{cases} 0 & K_{\max}^{equivalent} \le 0 \\ R^{equivalent} & K_{\min}^{equivalent} \ge 0 \\ 0 & K_{\min}^{equivalent} < 0 \end{cases}$$

Note that there are several mechanisms that have been identified as being able to keep the crack closed even when the current "remote" loading is positive. The most significant is plasticity-induced crack closure. This is one proposed explanation for the observed dependency of crack growth rates on the *R*. The effective values computed here do not explicitly account for this (plasticity-induced closure is treated implicitly by some of the available crack growth rate models), they only consider "global" closure where the currently applied "remote" load is negative.

6.5.1.6.1 Load Interaction Model

The Willenborg model can be used to set the effective ΔK, Fig 6.5.76. Once the box is checked, the **Set/Edit Parameters** button is activated; clicking the button leads to the dialog in Fig 6.5.77.

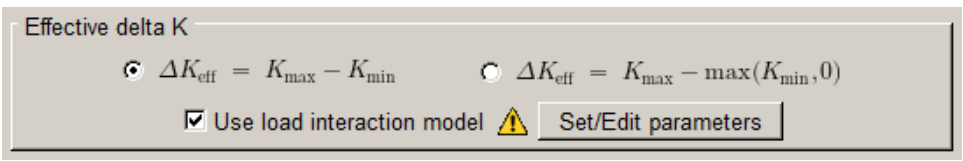


Figure 6.5.76 Load interaction model check box.

6.5.1.7 Subcritical Integration Options

There are two options that affect how the crack growth integrations are performed when computing total life (see Fig 6.5.8). If the Accelerated counting option is selected (the default), FRANC3D will try to use an accelerated cycle counting algorithm based on Runga-Kutta integration. If this option is turned off, FRANC3D will perform cycle-by-cycle counting. The accelerated counting might be slightly less accurate than cycle-by-cycle counting, but usually the difference in the computed fatigue lives for the two methods is less than 1-2% of the total cycle count. On the other hand, for analyses where the per cycle crack extension is small, for some models, the accelerated counting has been shown to give greater than 1000x reductions in the time required to compute total fatigue lives.

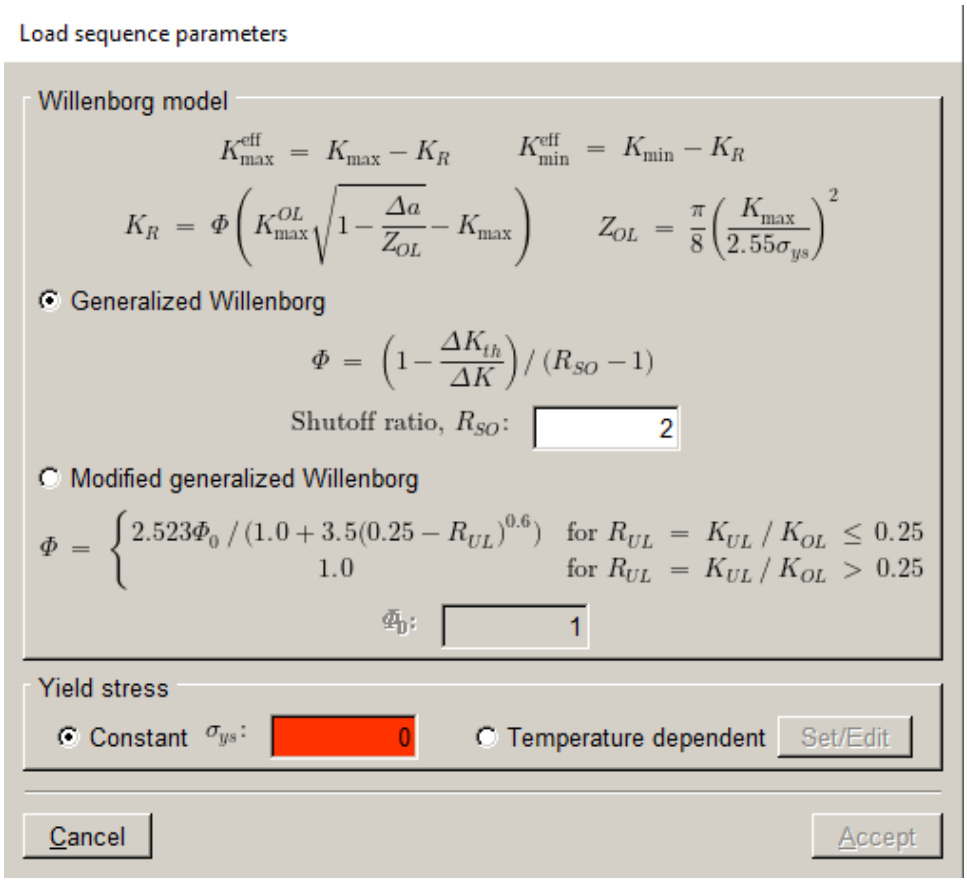


Figure 6.5.77 Willenborg load sequence parameters dialog.

The Constant K for time integration option can be set to speed-up the integration of time dependent growth. In general, the stress intensity factor will be function of time, so FRANC3D evaluates the integral:

$$\Delta a = \int_0^t A \cdot K_{hold}(\tau)^m \, d\tau$$

where t is the hold-time. The integral is evaluated using Runge-Kutta integration.

If, however, the crack growth over a hold interval is small, K can be assumed to be constant during the interval leading to the (much faster to evaluate) expression:

$$\Delta a = A \cdot K_{hold}^{m} \cdot t$$

If this option is selected, FRANC3D will assume that K is constant over all time intervals.

The user can choose between Runge-Kutta (RK) and Cycle-by-cycle (CxC) integration to compute life or cycles. A 2nd order RK approach is implemented, Fig 6.5.78.

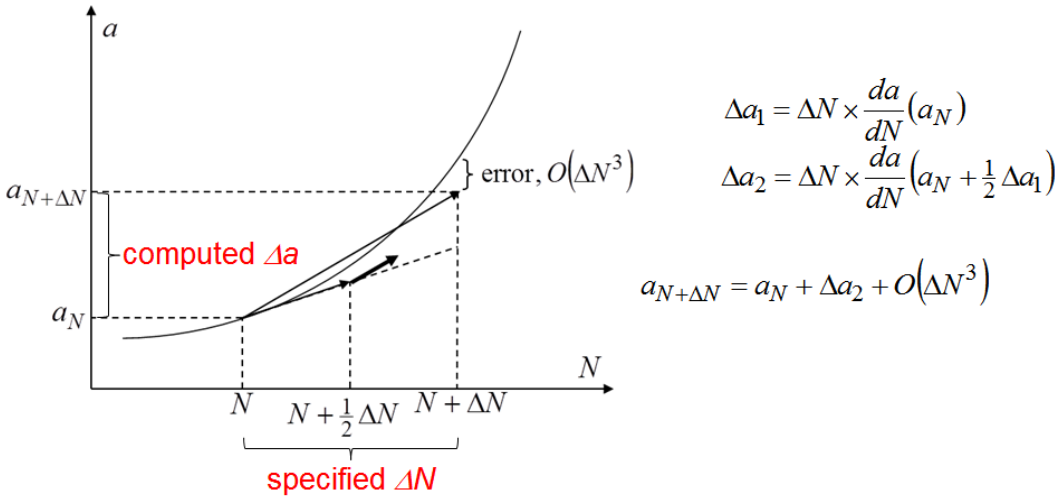


Figure 6.5.78 Second order Runge-Kutta integration.

6.5.1.8 Dynamic Pairing Metric

There are two options, currently active, that determine the dynamic pairing. The ΔK is the standard default, but da/dN can be chosen instead.

6.5.2 Quasi-static crack growth

The first dialog of quasi-static crack growth, Fig 6.5.79, is the same as for subcritical crack growth. The method for computing the kink angle must be specified (see Section 6.5.1.1).

The next dialog, Fig 6.5.80, allows the user to specify the power (n) for the growth model, as well as specifiying the K^{equiv} and the load step(s) to be used. The extension at all crack front points are computed based on the simple power law, Fig 6.5.81. The sign for K^{equiv} can be set (see Section 6.5.1.5).

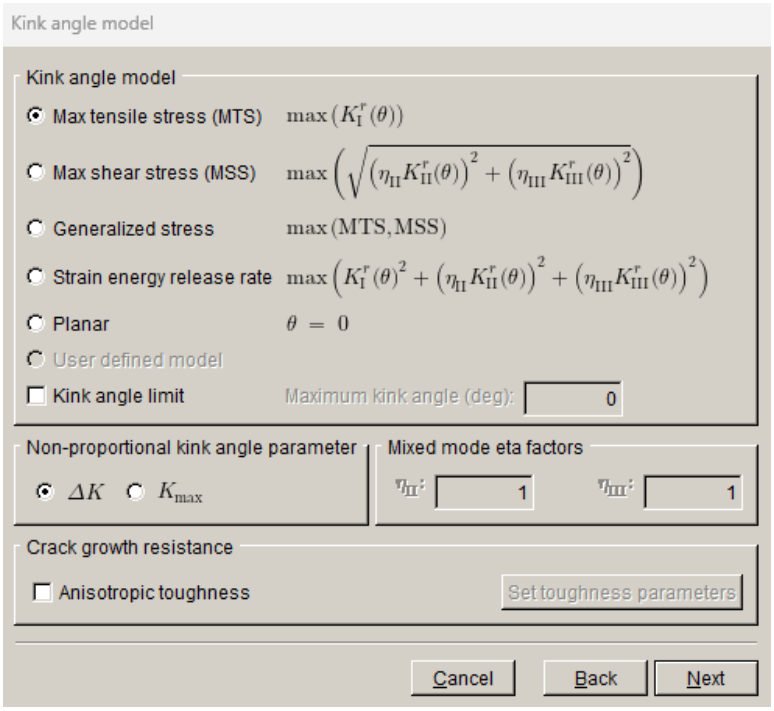


Figure 6.5.79 Kink angle model panel.

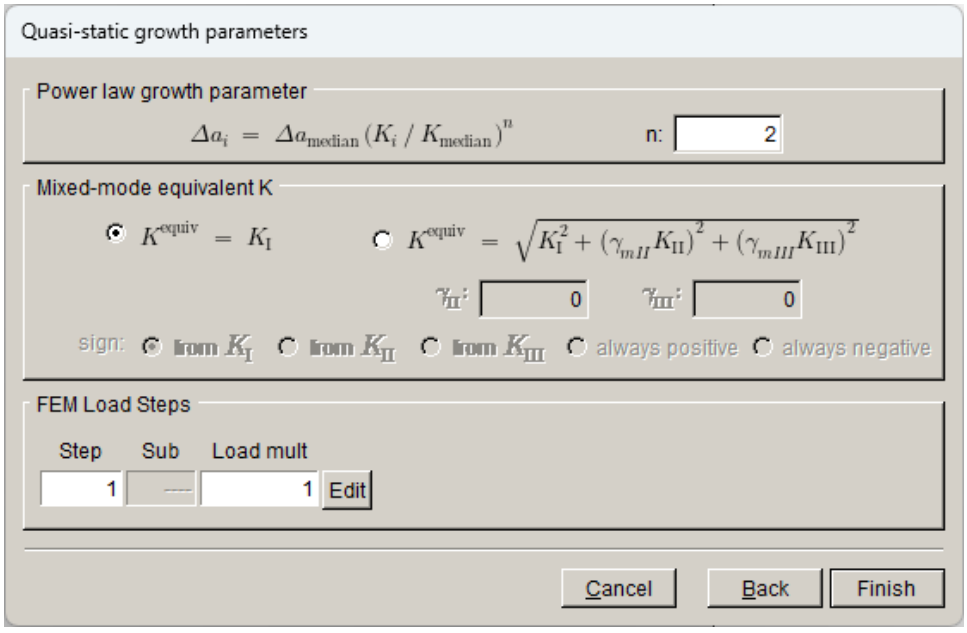


Figure 6.5.80 Quasi-static growth parameters panel.

$$\Delta a_i = \Delta a_m \left(\frac{\Delta K_i}{\Delta K_m} \right)^n$$

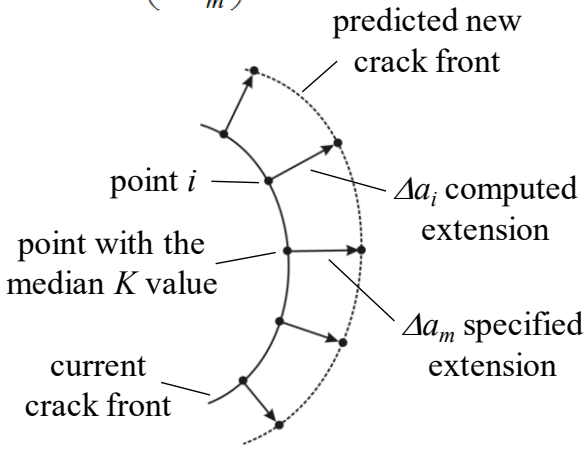


Figure 6.5.81 Quasi-static crack growth.

The default value of n is set to 2. This is based on a simplified analysis of crack front node interactions. Consider a crack front with nodes i and j as shown in Fig 6.5.82. If we ignore interactions among nodes:

$$K_i = K_c = \sigma_i \sqrt{\pi a_i} f(a_i) = (\sigma_i + \Delta\sigma_i) \sqrt{\pi (a_i + \Delta a_i)} f(a_i + \Delta a_i)$$

$$\sqrt{(a_i + \Delta a_i)\pi} = \frac{\sigma_i \sqrt{\pi a_i} f(a_i)}{(\sigma_i + \Delta\sigma_i) f(a_i + \Delta a_i)}$$

$$\Delta a_i = \frac{1}{\pi} \left[\frac{\sigma_i \sqrt{\pi a_i} f(a_i)}{(\sigma_i + \Delta\sigma_i) f(a_i + \Delta a_i)} \right]^2 - a_i$$

$$\Delta a_i = \frac{1}{\pi} \left[\frac{K_i}{(\sigma_i + \Delta\sigma_i) f(a_i + \Delta a_i)} \right]^2 - a_i$$

$$\Delta a_i \sim K_i^2$$

$$\Delta a_i = \Delta a_j \left(\frac{K_i}{K_j} \right)^2$$

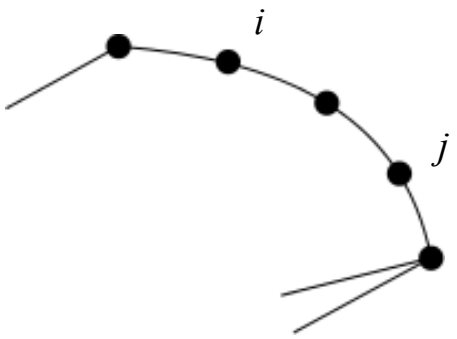


Figure 6.5.82 Nodes *i* and *j* on crack front.

6.5.3 User-Defined crack growth

User-defined crack growth is activated if the user has pre-defined Python extensions; this is described in Section 14.6.

6.5.4 Read growth parameters from a file

The user can save the crack growth model data to a file after defining it, and this file can be read back into FRANC3D. This is especially useful for the subcritical crack growth. This file is an ASCII text file and contains both the load schedule (see Section 6.5.1.3) and the crack growth rate (see Section 6.5.1.4) data.

6.5.5 Crack growth front fitting

Once the crack growth model has been defined, FRANC3D will display the computed crack growth, Fig 6.5.83. The crack extension can be adjusted in this dialog if needed. The computed new-crack-front points are smoothed and extrapolated (for surface flaws); there are several options for this.

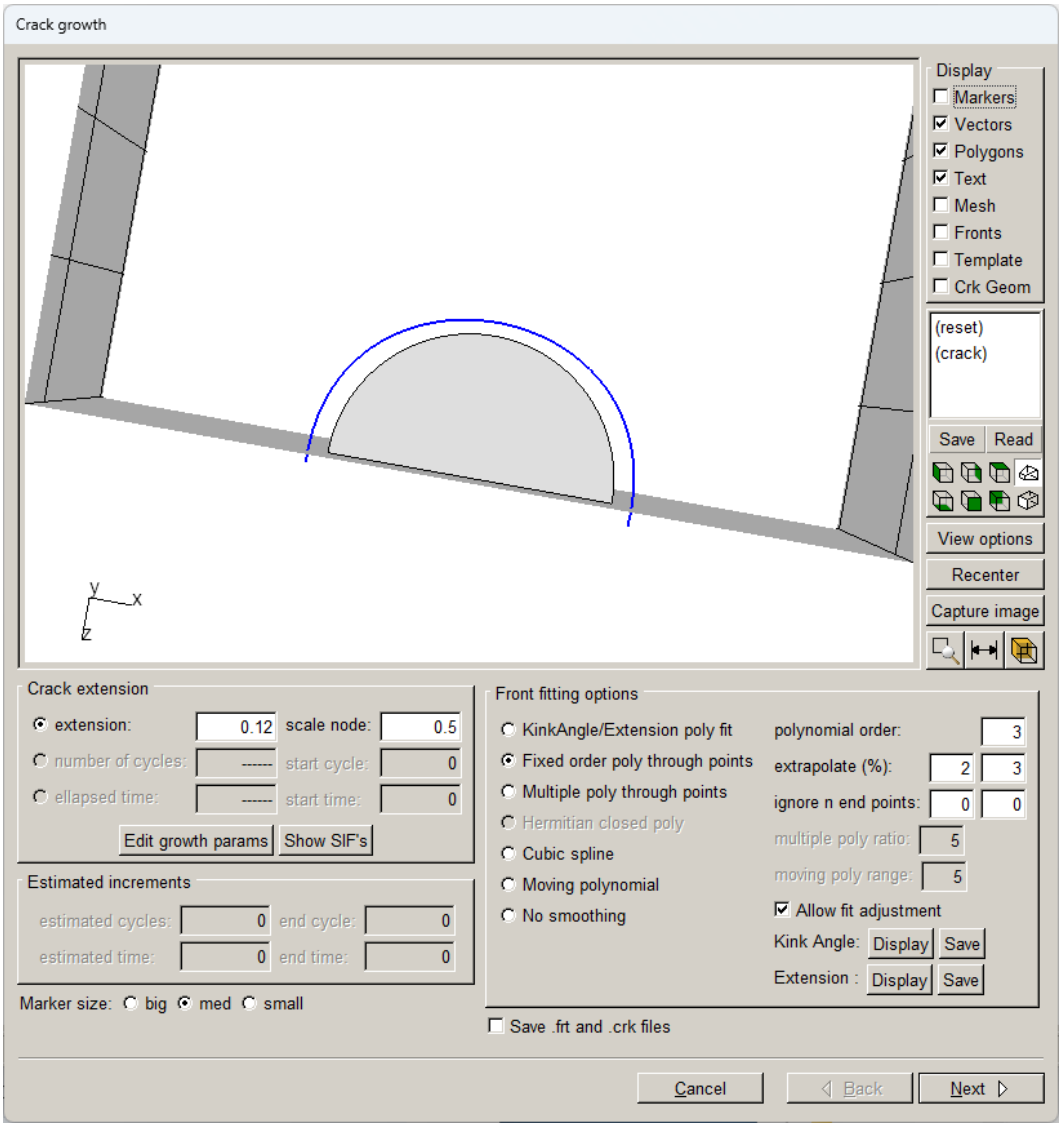


Figure 6.5.83 Crack growth display dialog.

Specified extension or number of cycles or elapsed time can be chosen to adjust the amount of crack extension. The cycles and time options are available if the crack growth model includes such.

Note that a scale node of 0.5 represents the median; extensions are sorted from minimum to maximum, and the scale node varies from 0 to 1. For a specified median-extension, the crack extension at crack front point *i* is computed from the expression:

$$\Delta a_i = \Delta a_{median} \left(\frac{\dfrac{da}{dN}_i (\Delta K_i, R_i, \ldots)}{\dfrac{da}{dN}_{median} (\Delta K_{median}, R_{median}, \ldots)} \right)$$

where Δa_{median} is the crack extension specified for the point with the median stress intensity factor range (Fig 6.5.84), da/dN_i is the crack growth rate computed at point *i* using the specified growth rate model, and da/dN_{median} is the crack growth rate computed at the crack front point with the median stress intensity value. The crack growth rates at crack front points will be functions of the corresponding ΔK, *R*, and other growth model parameters.

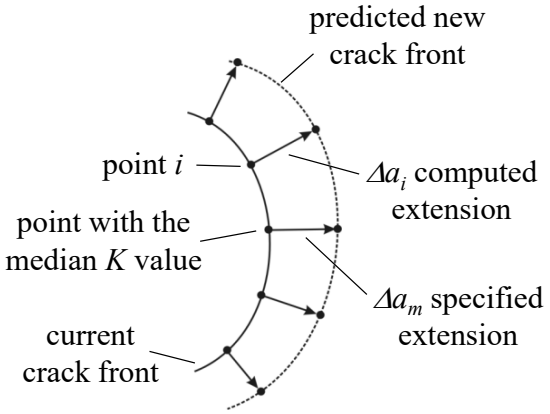


Figure 6.5.84 Computed extension based on specified median extension.

Note that the selection of the reference ΔK value is arbitrary. The maximum, minimum, and average ΔK are reasonable values to consider. The *median* value is used by default in FRANC3D because experience shows that the maximum and minimum values frequently occur where a crack front meets a free surface where the accuracy of the computed stress intensity factors are known to be least accurate.

In some cases, the inaccuracy of these maximum and minimum values can lead to a very unintuitive relationship between the reference specified amount of crack extension and the actual predicted crack front. In extreme cases, inaccurate maximum and minimum values can have a significant impact on the average value leading to unintuitive behavior for this measure also.

The scale node setting allows the user to alter the location of the specified extension. The extensions are sorted from minimum to maximum, and the scale node can vary from 0 to 1. For example, setting the scale node to 1 means that the specified extension will be applied at the crack front location where the maximum extension is computed. The Δa_i are computed relative to this location (and its corresponding values SIF, growth rate, *etc.*).

The expression for crack extension at crack front point *i* using the specified cycles option is

$$\Delta a_i = N \times \frac{da}{dN_i}(\Delta K_i, R_i, \ldots)$$

where N is the specified number of cycles and da/dN_i is the crack growth rate computed at point i using the specified growth rate model.

The specified number of applied-cycles approach seems more intuitive than the specified median crack front extension approach for many analysts. However, the fatigue crack growth models are highly nonlinear functions so trial-and-error might be required when using the specified cycles approach to find the number of cycles that does not give excessively large or small predicted crack extensions. Excessively large values might give inaccurate results, while excessively small crack extensions might require many growth steps or, in some cases, might cause meshing problems.

6.5.5.1 Multiple crack front fitting

For a model with multiple crack fronts, there is an extra panel on the left side at the bottom, Fig 6.5.85. The Grow crack fronts pane allows one to turn off crack growth or to scale the extension for a particular crack front. The growth of multiple crack fronts is treated in the same manner as a single crack front – all the SIFs for all the fronts are examined to find the median value and then all crack front extension is computed as described above.

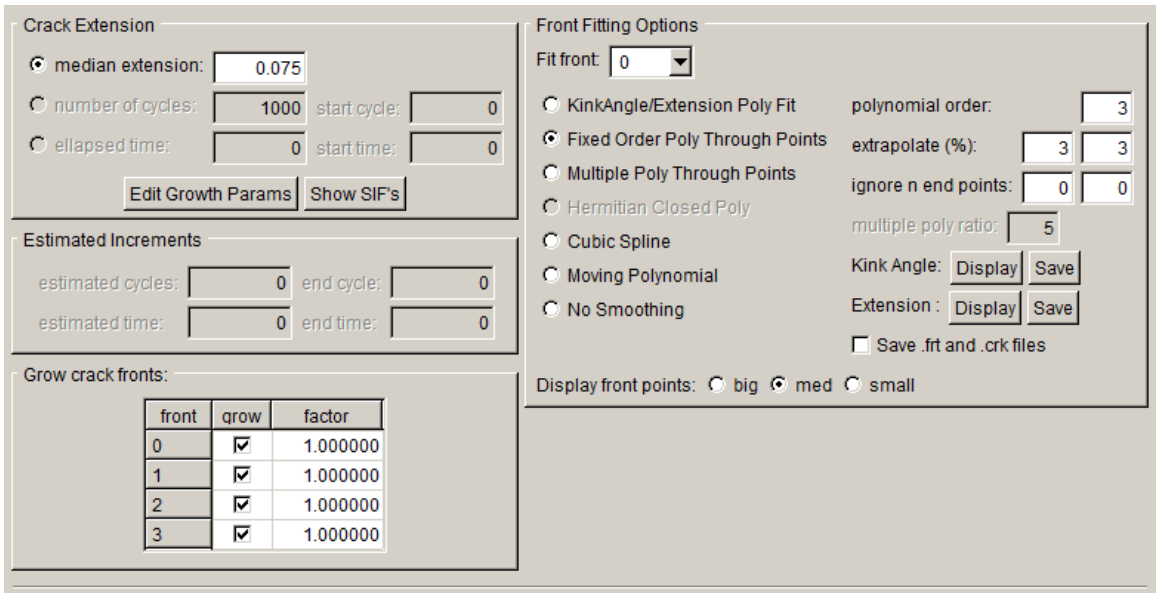


Figure 6.5.85 Crack growth extension and fitting options for multiple crack fronts.

6.5.5.2 Front fitting options

The *Front fitting* pane includes options for curve fitting, fields for extrapolating the ends of the fitted curves, and the option to discard crack front points at either end. The curve-fit options are: 1) kink angle/extension poly, 2) fixed order poly, 3) multiple poly, 4) Hermitian, 5) cubic spline, 6) moving poly, and 7) no smoothing. These are described in the following sub-sections.

The polynomial order for any of the poly-fitting options can be set in the fixed poly order field. The default order is 3.

The ends of the fitted curves can be extrapolated to ensure that the curves intersect the model surface. It is required that the crack surface geometry intersect the model surface. New crack surface geometry is created between the current and new crack fronts. FRANC3D tries to ensure that the curves are extrapolated far enough, but the user also can visually determine whether the extrapolation is sufficient (or too much).

A set of points at either end of the crack front can be ignored. If the end points cause problems in fitting or extrapolating, the user can ignore some of these end points.

The Save .frt and .crk files option allows the user to save the new front points and the new crack geometry to a file before inserting it into the model. The *.frt* file can be read using the **Read Crack Growth** wizard, which is described in Section 6.6. The *.crk* file can be read using the **Flaw From Files** option, see Section 6.2.

The **Kink Angles** and **Extension** buttons bring up plots of the computed kink angle, Fig 6.5.86 – left panel, and crack front extension, Fig 6.5.86 – right panel, along the crack front, respectively.

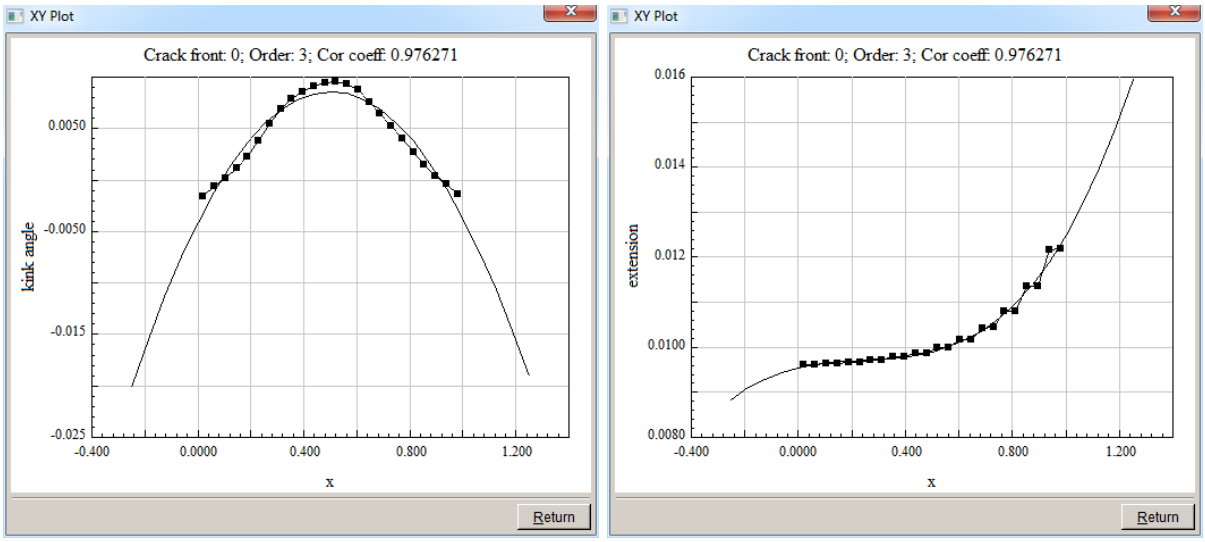


Figure 6.5.86 Plots of kink angle and crack front extension along the crack front.

6.5.5.2.1 Kink Angle/Extension Poly Fit

Using the kink angle and extension data, a best fit polynomial is fit through the kink-angle and extension data, using the polynomial order from the fixed poly order field. The curve-fits can be extrapolated, and then new extrapolated crack front points are defined based on these and the existing crack front geometry.

As an example, consider a penny-surface crack with the kink angle and extension from Fig 6.5.86. The initial crack boundary edges, the actual crack surface, which is shaded grey, the new front points as green dots, and the curve fit shown as a blue line are shown in Fig 6.5.87. The black dots on the original crack front correspond to the fitted front points, where the extension and kink angle are applied to produce the new fitted points. The kink angle and extension curves are extrapolated to compute these new end points that fall outside the model.

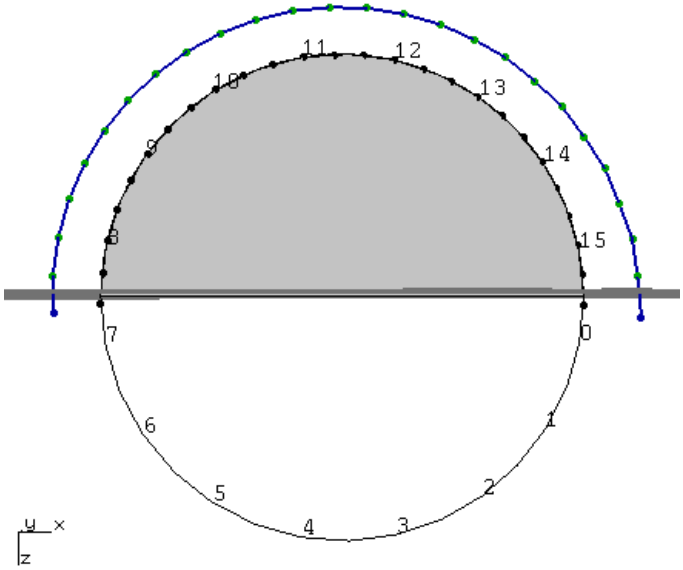


Figure 6.5.87 New front fit based on kink angle and extension curve-fits.

6.5.5.2.2 Fixed order poly through points

The fixed order polynomial option is the simplest and the most used fitting option. A polynomial fit through the new front points in Cartesian space is based on a least-squares fit. The order of the polynomial is provided in the fixed poly order field. This polynomial can be extrapolated a small distance to make sure the new front intersects the model surface. Best results are obtained with low order polynomials and limited extrapolation; if the crack front does not suit a simple polynomial, one of the other fitting options should be chosen.

6.5.5.2.3 Multiple poly through points

For long shallow crack shapes, the multiple polynomial fit might be the best option. This option uses three polynomials. The number of points in the two end polynomials versus the middle polynomial can be adjusted using the multiple poly ratio. The ends of the polynomials are blended, but kinks can be created so this option should be used with care.

6.5.5.2.4 Hermitian closed poly

The Hermitian polynomial is used for fitting closed crack fronts such as interior cracks. It is not active if all cracks are open-ended surface cracks. The Hermitian fit is performed using four equal segments; thus, it works best for circular shapes.

6.5.5.2.5 Cubic spline

A cubic spline fit can be used for crack fronts that do not allow for a simple low order polynomial fit. A cubic spline will follow arbitrary curves. The ends of the crack front are extrapolated using a linear segment through the end points for open-ended surface cracks. This option is comparable to the moving polynomial described next.

6.5.5.2.6 Moving polynomial

A moving polynomial fit uses the polynomial order given in the fixed poly order field. A subset of the crack front points is used for successive curve fits, moving along the full crack front in increments from one end to the other. Like the cubic spline fit, this option is good for arbitrary curves. The ends of the crack front are extrapolated using a linear segment through the end points for open-ended surface cracks.

As an example of crack fronts that do not work well with a simple polynomial fit, consider the crack fronts in Fig 6.5.88. Fig 6.5.89 shows these same crack fronts using a moving polynomial fit; comparable results can be obtained for cubic spline fits.

For the left-side crack in Fig 6.5.88, an 8^{th} order polynomial fits the data, but extrapolating such a high order polynomial does not work well; in this case the extrapolated curve does not intersect the model surface. In the second example, a 12^{th} order polynomial cannot capture the curve.

Fig 6.5.89 shows a moving polynomial (or spline) fit that matches the data and provides extrapolated end points that are outside the model.

For crack fronts with concave segments, new crack front points can overlap depending on the amount of extension. The moving polynomial (and cubic spline) fits look for reversals in the crack front points, and points are discarded if this happens.

6.5.5.2.7 No smoothing

The final fitting option removes all smoothing, but the ends can be extrapolated. The extrapolation of the ends is based on a simple fit through a small set of points at each end.

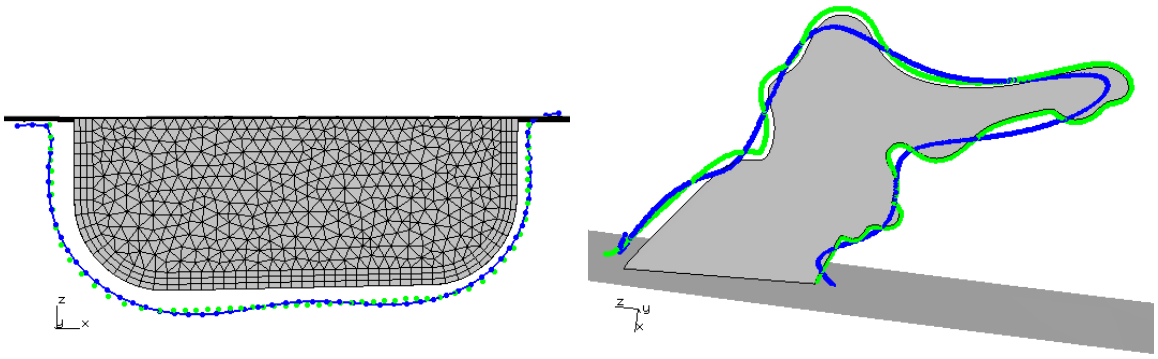


Figure 6.5.88 New front fit based on a simple polynomial fit.

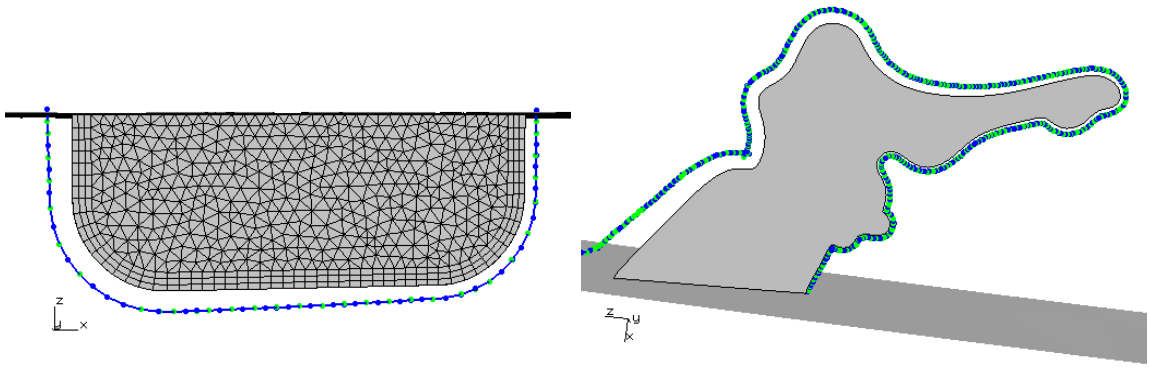


Figure 6.5.89 New front fit based on a moving polynomial fit.

6.5.6 Crack growth front template mesh

The final crack growth display wizard panel, Fig 6.5.90, allows one to set the crack front mesh template parameters. The model is displayed in the upper window, and the user can select a camera position that shows the template. The lower third of the window contains one pane for the template parameters.

Most of the template parameters and buttons are identical to those for inserting a new crack, see Section 6.1.23. The template radius can be set as an absolute value or as a percentage of the crack extension; the choice should depend on the model and expected growth behavior.

Sections 8 and 9 of the Users Guide provide guidance for setting the template radius and the crack growth increment.

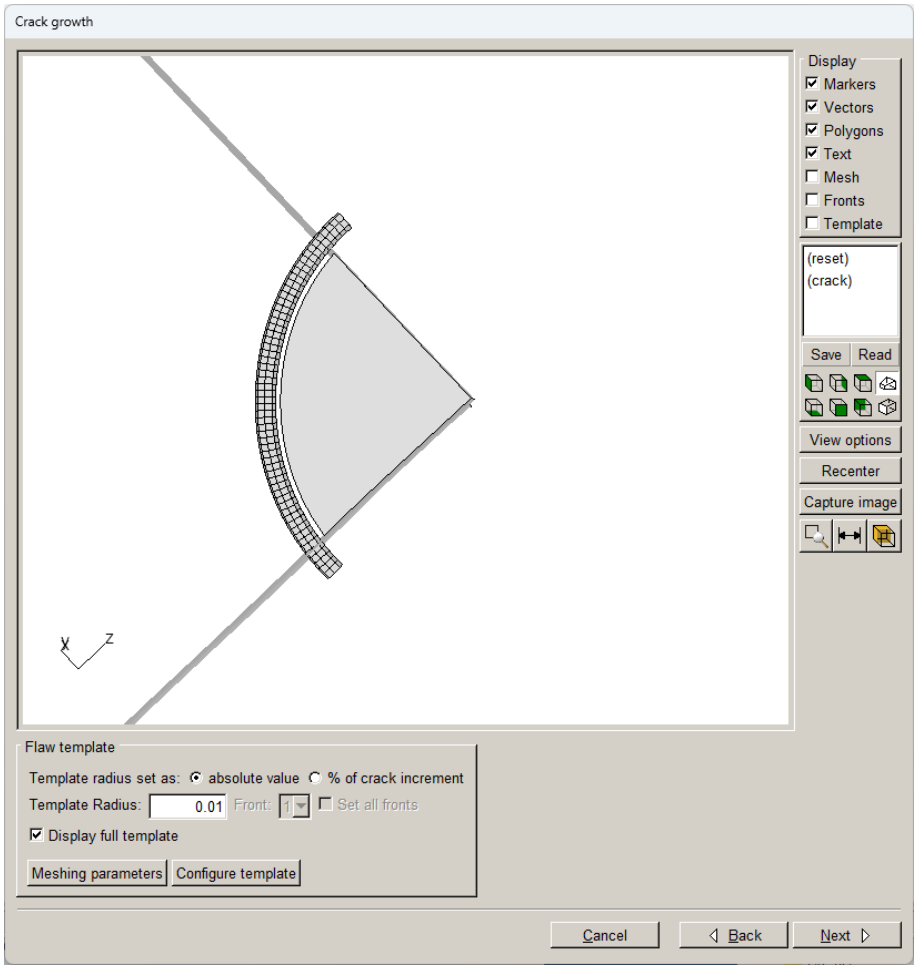


Figure 6.5.90 Crack growth - front mesh template.

6.6 Read Crack Growth Wizard

The Read Crack Growth wizard consists of two panels. The first, Fig 6.6.1, allows the user to specify the file name containing the new crack front points. The points are displayed along with the crack. If there are multiple crack fronts, separate files can be specified for each front if needed.

The format for this file is simply: "x y z" ; one set of coordinates per line.

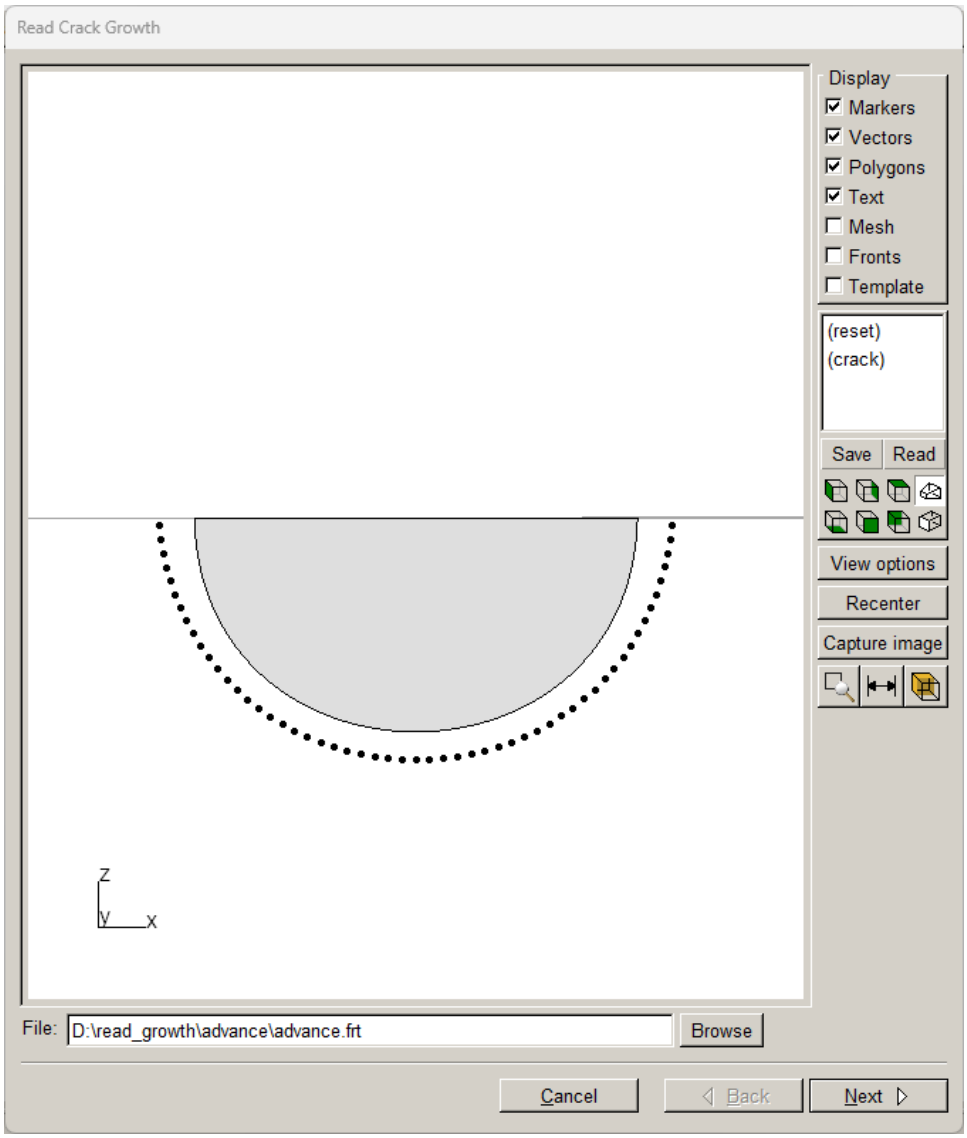


Figure 6.6.1 Read new crack front - file import dialog.

The second panel, Fig 6.6.2, displays the new front points within the model and allows the user to specify the fitting, extrapolation, and template parameters. The *front fitting options* and *flaw template* panes are described in Sections 6.5.5 – 6.5.6.

The template is off in Fig 6.6.2; it can be turned on using the Template check box under Display on the right side

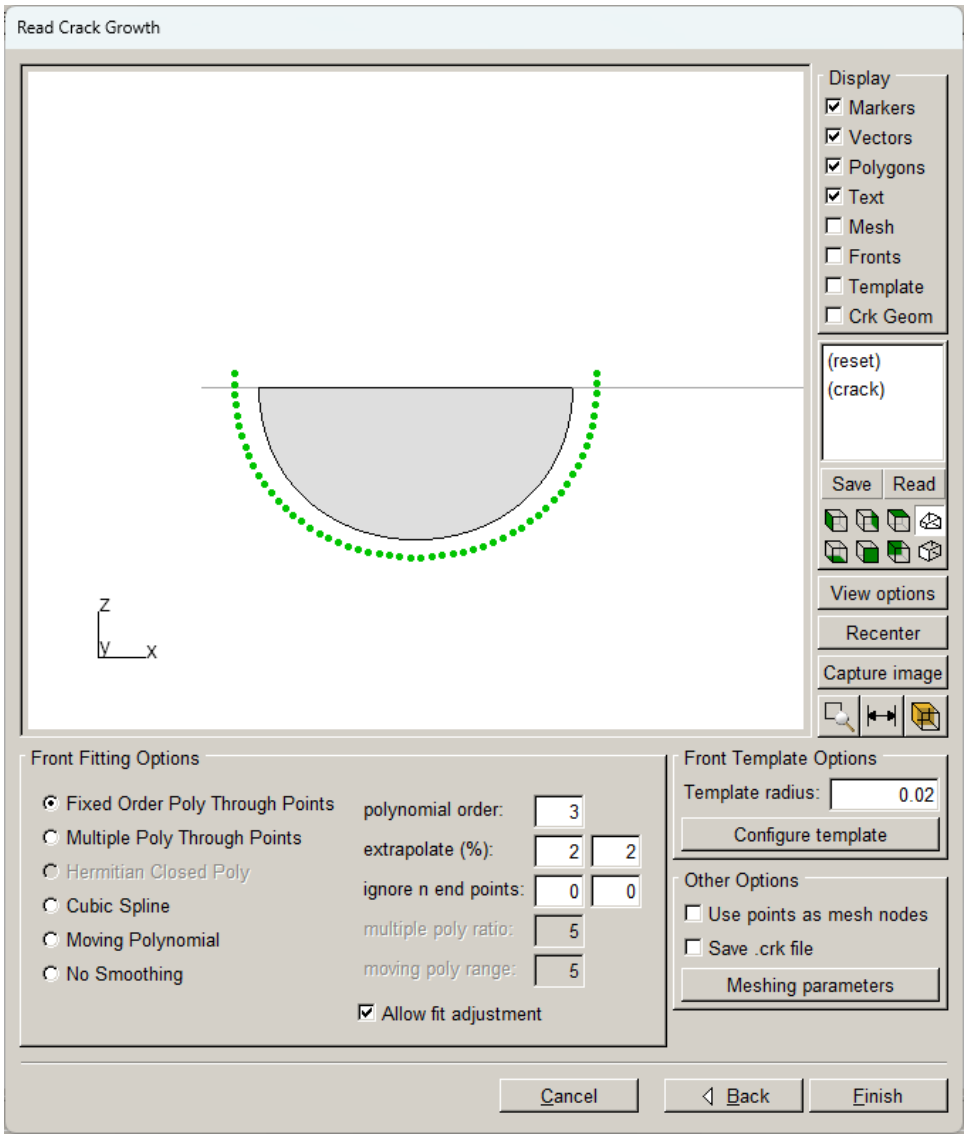


Figure 6.6.2 Read crack growth - display panel.

6.7 Grow/Merge Cracks Wizard

The **Grow/Merge Cracks** allows semi-co-planar cracks, whose advancing crack fronts intersect, to be merged to create a single crack. The dialogs for computing SIFs and setting crack growth parameters are described in Sections 6.4 and 6.5. The difference is seen when the crack growth is displayed, Fig 6.7.1.

The Growth & Merge dialog allows the user to adjust the extension (or cycles) to ensure the fronts intersect at a reasonable location to create a merged front. The fitting options are limited to cubic spline and moving polynomials. The ends of the curve must intersect the model surface, so the user is able to adjust the amount of extrapolation.

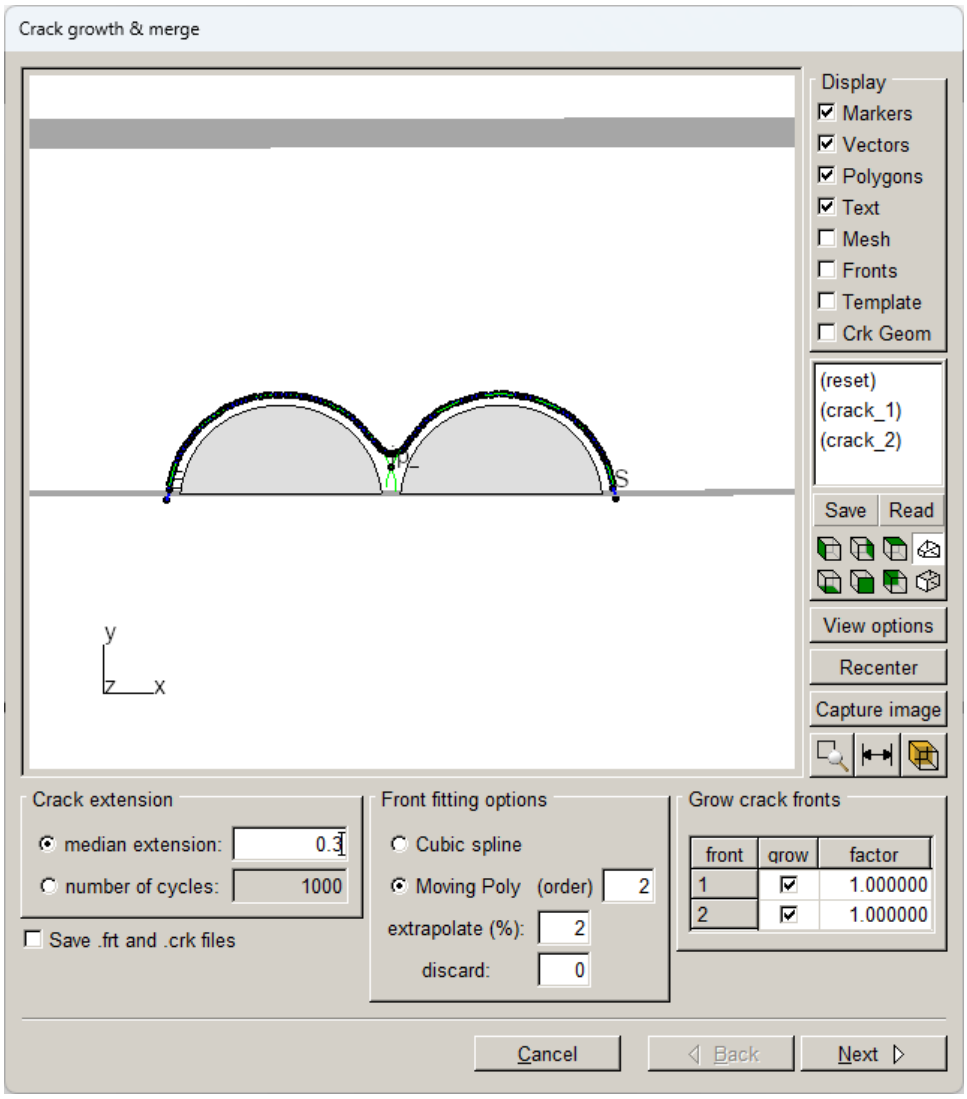


Figure 6.7.1 Grow/Merge Cracks – two co-planar cracks whose advancing fronts intersect.

Select **Next** to define the template mesh for the merged front, Fig 6.7.2. The template mesh radius might need to be adjusted to obtain a reasonable mesh at the point where the fronts merge. Select **Next** to complete the process of merging and subsequent insertion and remeshing of the model. The result should be a merged crack as shown in Fig 6.7.3.

Section 8.4 of the Users Guide describes a current limitation to merging.

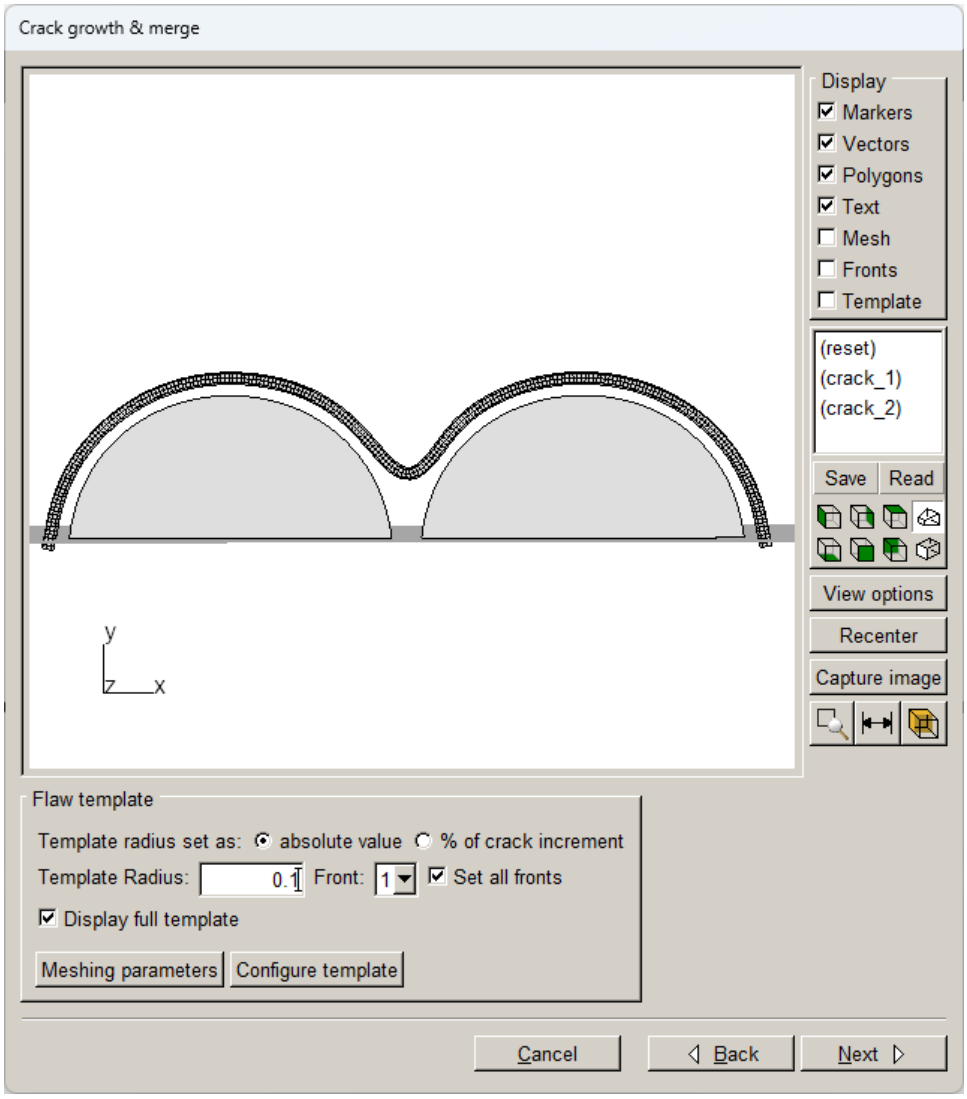


Figure 6.7.2 Grow/Merge Cracks – two co-planar crack fronts merged with template mesh.

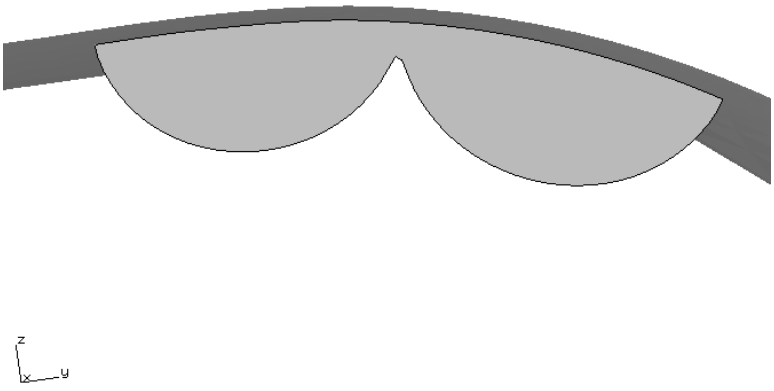


Figure 6.7.3 Two co-planar cracks merged.

6.8 Edit Crack Geometry

The Edit Crack Geometry dialog, Fig 6.8, provides some options to display and modify the crack surface geometry. The crack geometry along with the uncracked FE model is displayed. The left side pane provides options for modifying and displaying the current crack geometry.

The **Replace crack with edited version** button (bottom right in Fig 6.8.1) will replace the crack geometry, currently attached to the model, with the edited geometry. If there are problems when trying to grow the crack, editing the geometry might allow the crack to grow.

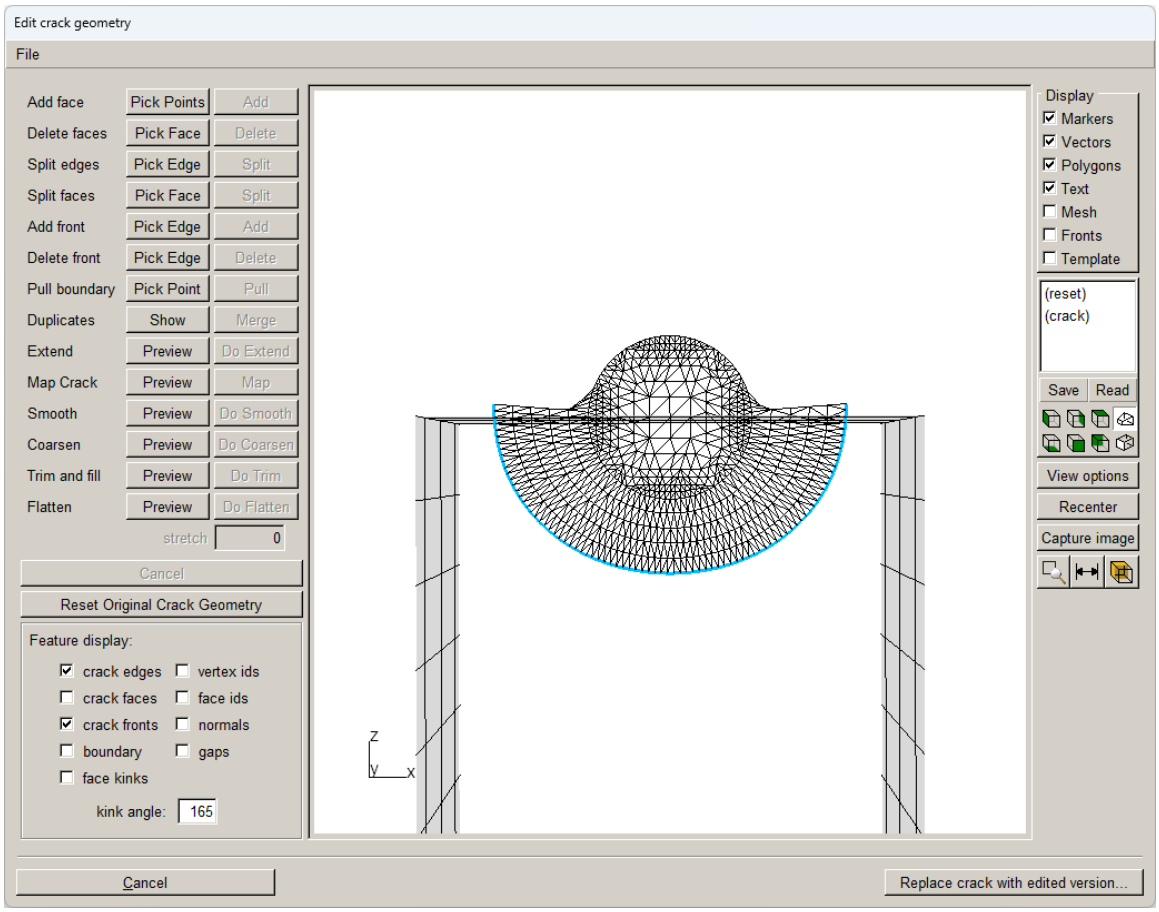


Figure 6.8 Edit Crack Geometry dialog.

6.8.1 File menu

There are three menu items under **File**, Fig 6.8.1. Read .crk file allows the user to read a .crk file in place of the current crack. Save crack to .crk file allows the user to save the current "edited"

crack to a .crk file, and Save crack to .stl file saves the current crack to a .stl file. Note that the *.stl* file will only contain the triangular facets.

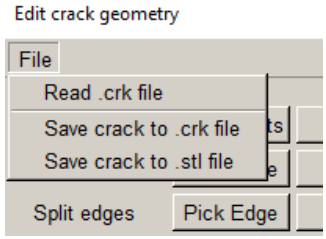


Figure 6.8.1 File menu for Edit Crack Geometry dialog.

6.8.2 Add face

The Add face option allows one to create a new triangular Bezier patch. Use the **Pick Points** button to collect two corner points of a new triangular patch; the points must already exist, Fig 6.8.2. Once the points are collected, use the **Add** button to create the patch.

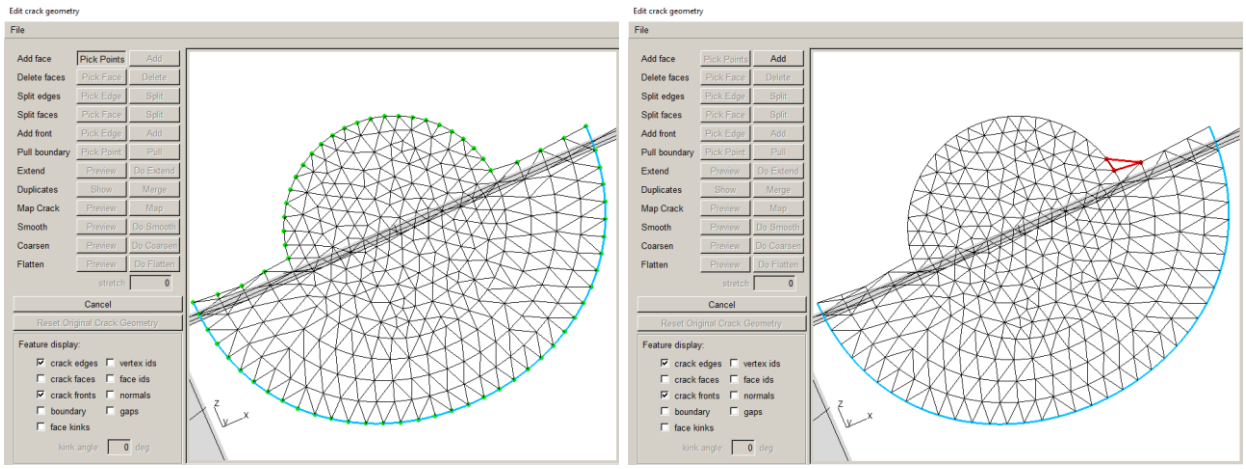


Figure 6.8.2 Add face by picking two existing corner points and then clicking **Add**.

6.8.3 Delete faces

The Delete faces option allows one to remove triangular Bezier patches. Use the **Pick Face** button to collect the patches, Fig 6.8.3, and then use the **Delete** button to remove them.

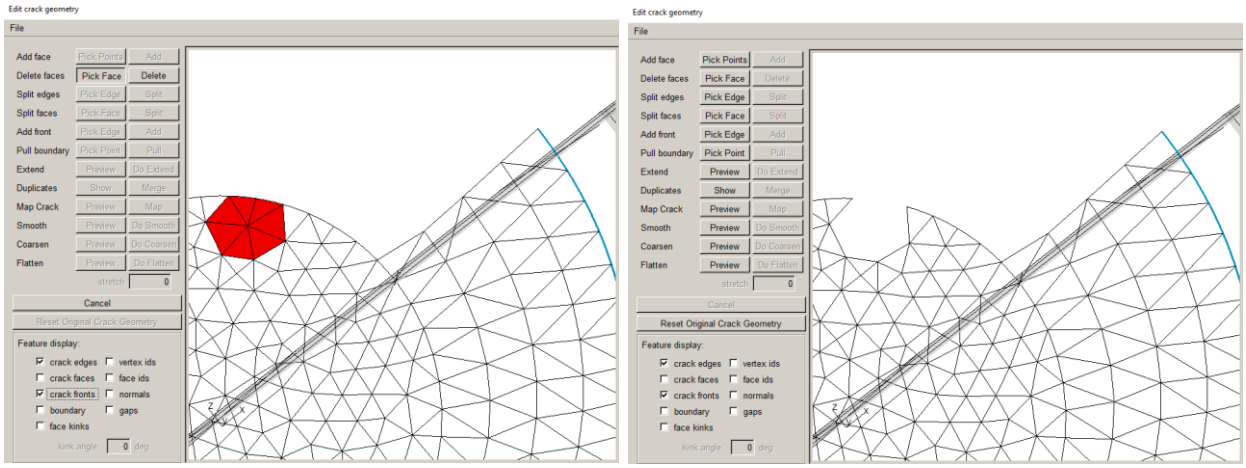


Figure 6.8.3 Delete faces by picking existing faces and then clicking **Delete**.

6.8.4 Split edges

The Split edges option allows one to divide an edge in half. Use the **Pick Edge** button to collect the edge, Fig 6.8.4, and then use the **Split** button to split the edge. This will split the face into two triangular patches.

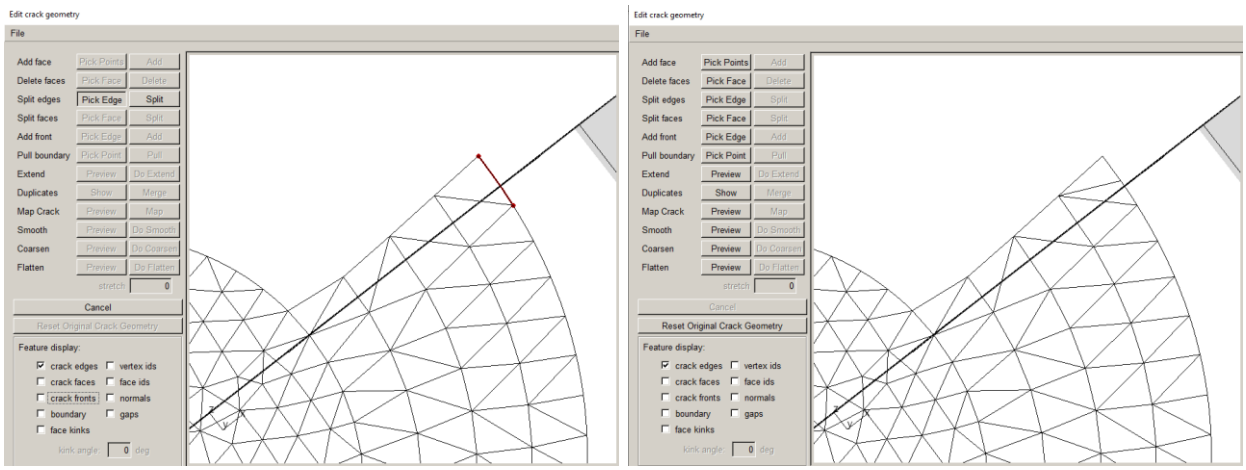


Figure 6.8.4 Split edges by picking existing edges and then clicking **Split**.

6.8.5 Split faces

The Split faces option allows one to divide a triangular Bezier patch into three patches by adding a center point. Use the **Pick Face** button to collect the patches, Fig 6.8.5, and then use the **Split** button to divide the patches.

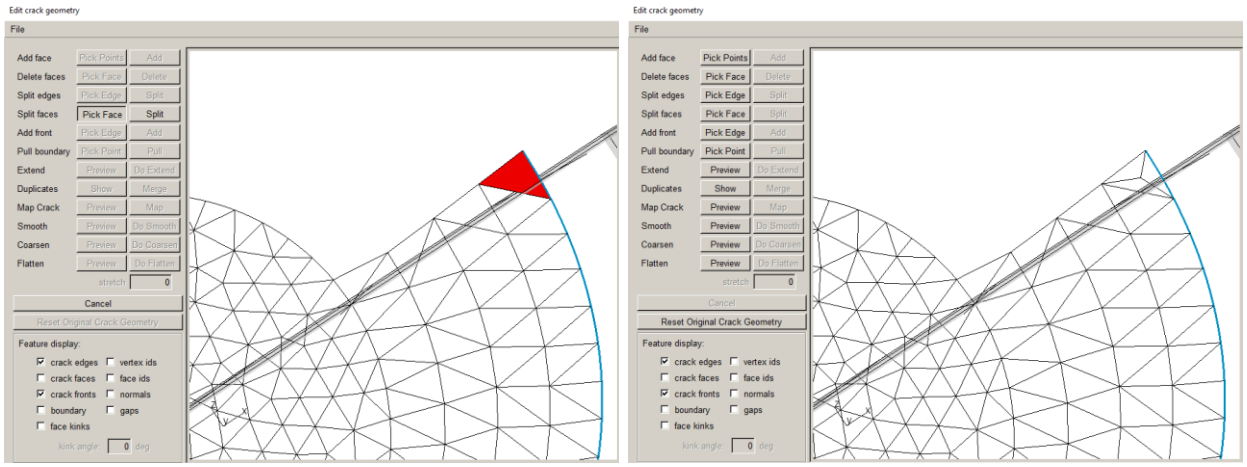


Figure 6.8.5 Split faces by picking existing faces and then clicking **Split**.

6.8.6 Add front

The Add front option allows one to extend the crack front. Use the **Pick Edge** button to collect edges adjacent to the existing front edges, Fig 6.8.6, and then click **Add**.

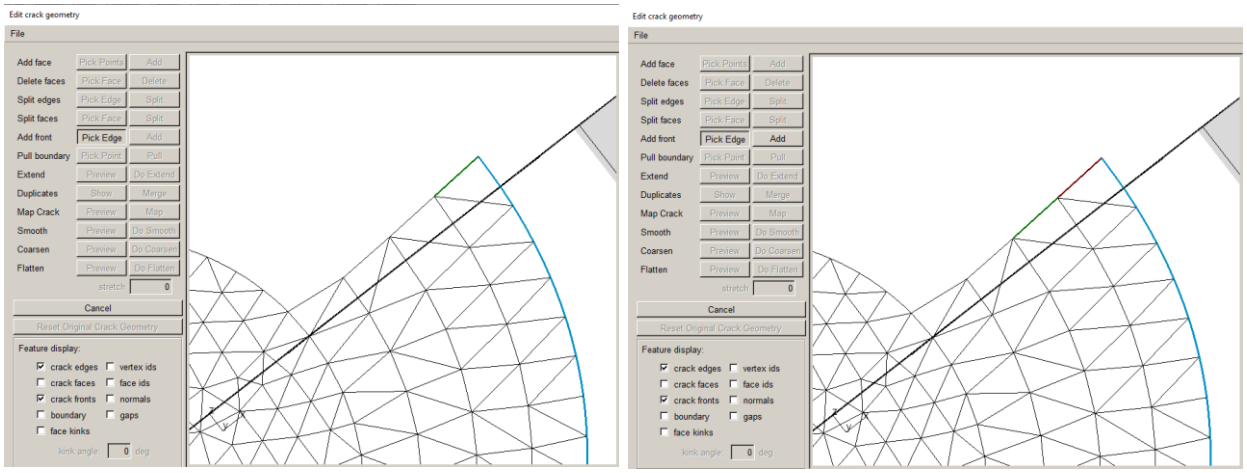


Figure 6.8.6 Add front by picking existing edges adjacent to the front (highlighted in green) and then clicking **Add**.

6.8.7 Delete front

The Delete front option allows one to turn off the crack front flag for an edge. Use the **Pick Edge** button to collect existing front edges, Fig 6.8.7, and then click **Delete**.

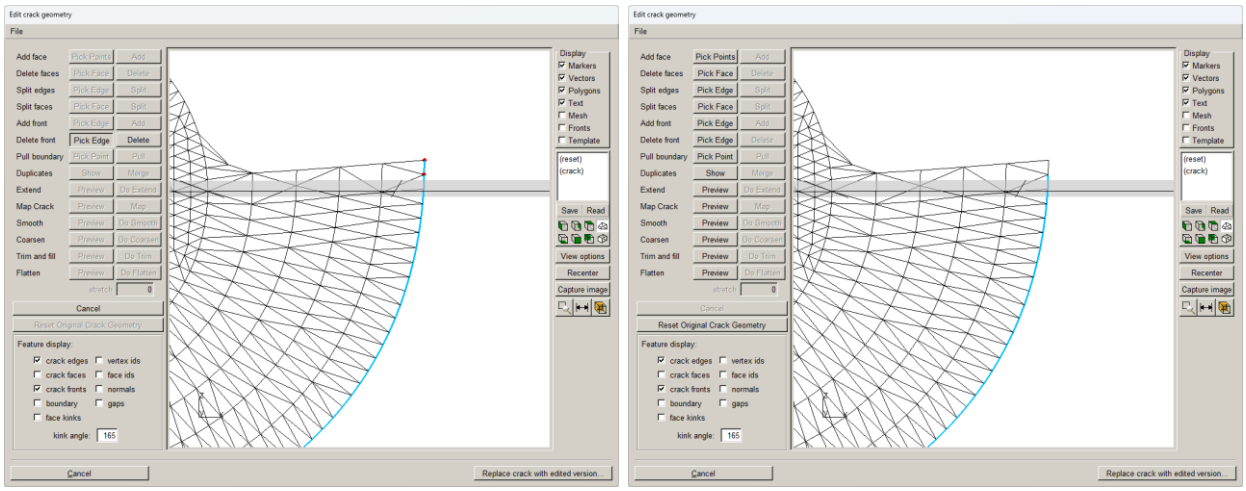


Figure 6.8.7 Delete a front edge by picking existing front edges (highlighted in red) and then click **Delete**.

6.8.8 Pull boundary

The Pull boundary option allows one to pull a boundary point outwards. If there is a gap in the crack surface where it does not extend past the model surface, Fig 6.8.8, one can pull the boundary out using the **Pick Point** button, and then clicking **Pull**. More than one point might need to be pulled to remove the gap.

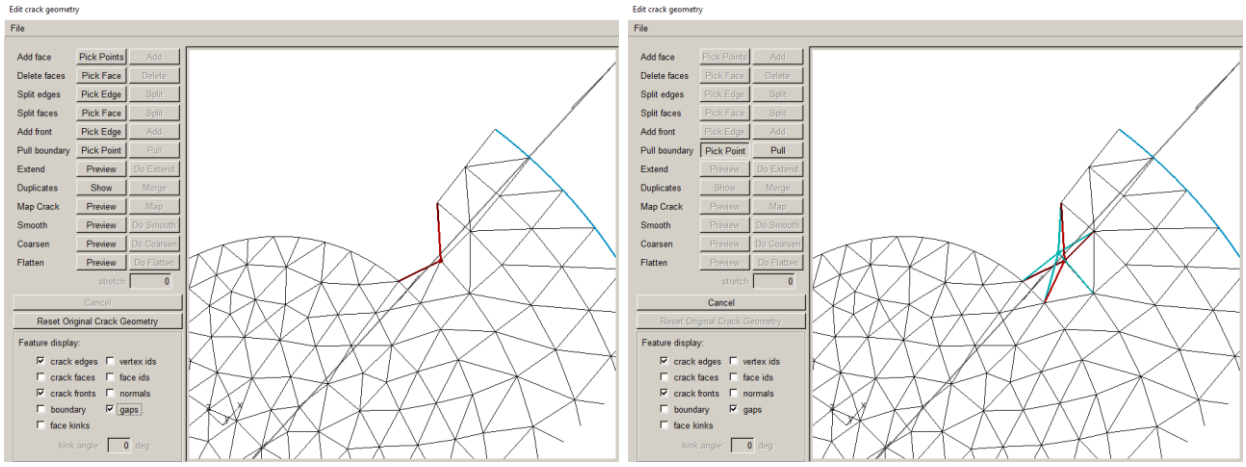


Figure 6.8.8 Pull boundary by picking a boundary point and then clicking **Pull**.

6.8.9 Duplicates

The Duplicates option allows one to merge two geometry points that have identical coordinates. Use the **Show** button to display the points with identical coordinates, Fig 6.8.9, and then use the **Merge** button to condense the two points to a single point.

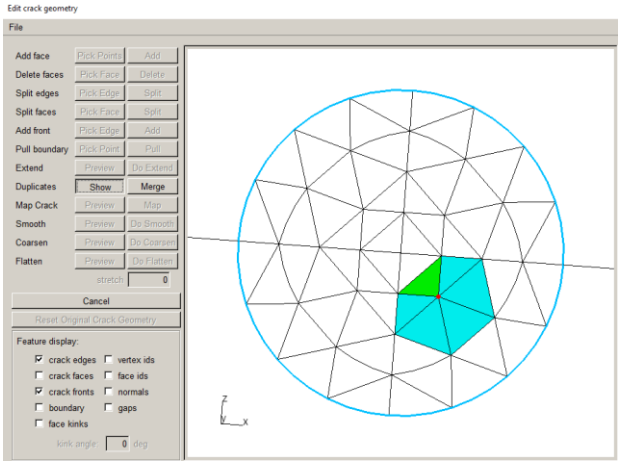


Figure 6.8.9 Merge duplicate points by clicking **Show** and then **Merge**.

6.8.10 Extend

The Extend option allows one to extend the crack geometry by reading a file that contains the xyz coordinates of the extended front. Select the **Preview** button, Fig 6.8.9, and then select the file containing the new crack front points. Typically this will be a .frt file. The new crack surface will be displayed in red; click the **Do Extend** button to extend the crack.

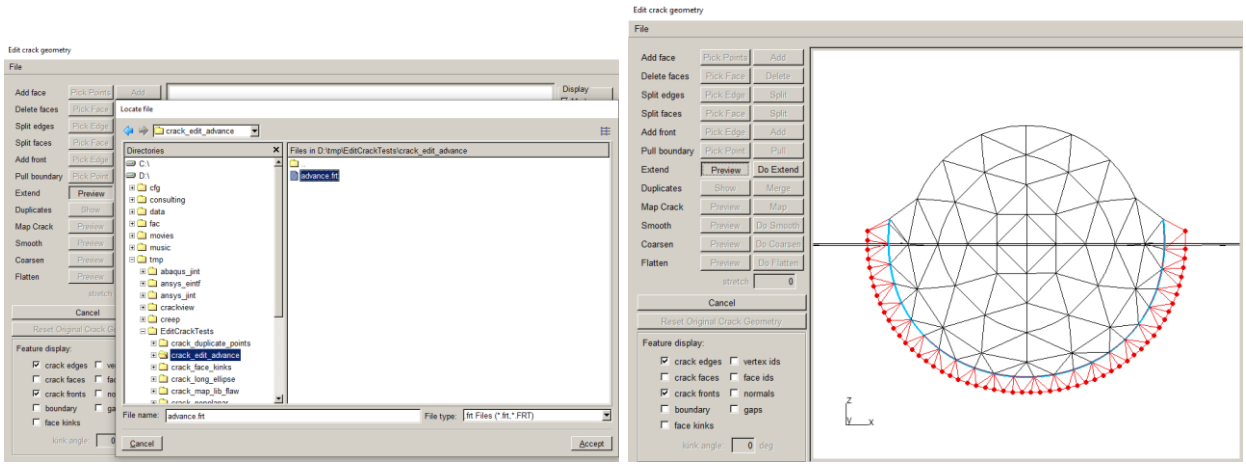


Figure 6.8.10 Extend the crack front by selecting a .frt file and then clicking **Do Extend**.

6.8.11 Map crack

The Map crack option allows one to map a circular penny-shape crack onto the current crack geometry. This allows one to create a non-planar "library" flaw, if the existing crack surface is non-planar. Use the **Preview** button to disply the Circular crack parameters dialog, Fig 6.8.11, and enter the radius and the global Cartesian coordinates of the center of the circle. If the crack maps onto the existing crack successfully, click **Map** to replace the crack with the mapped crack.

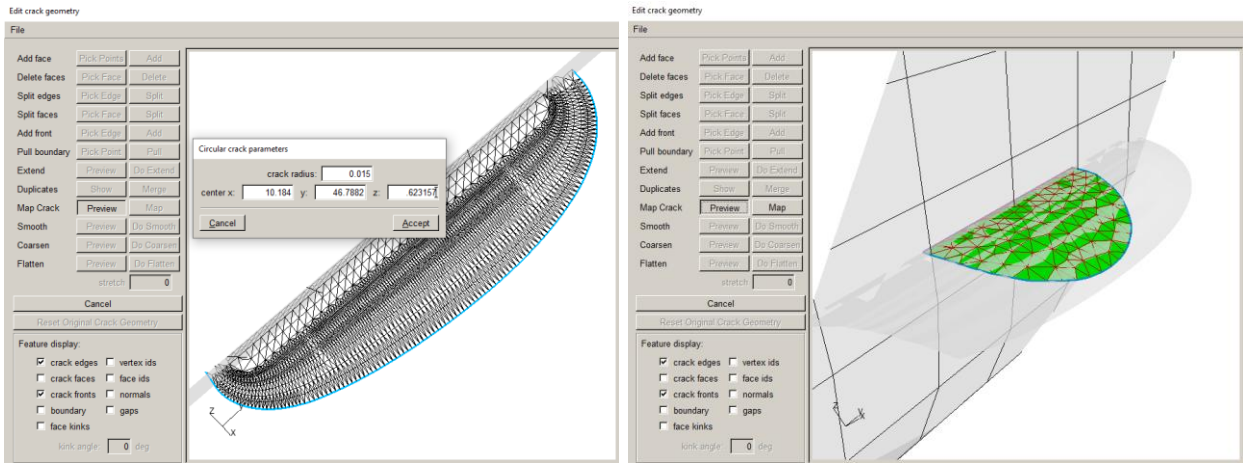


Figure 6.8.11 Map a circular "library" crack onto an existing crack.

6.8.12 Smooth

The Smooth option allows one to improve the Bezier triangle shapes. Click the **Preview** button to display the smoothed facets overlayed on the original geoemtry, Fig 6.8.12, and then click **Do Smooth** to replace the original with the smoothed.

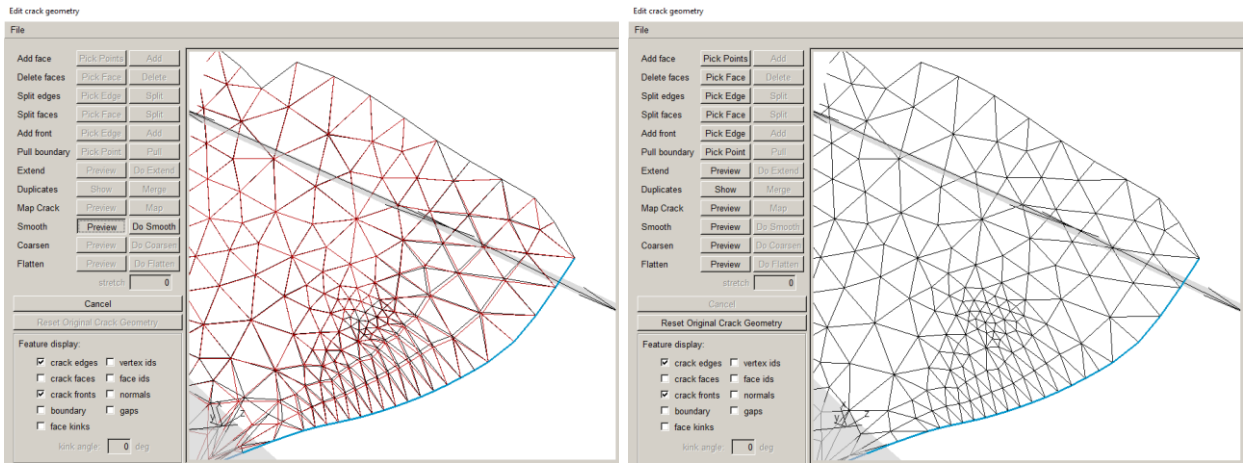


Figure 6.8.12 Smooth crack surface geometry.

6.8.13 Coarsen

The Coarsen option allows one to reduce the number of triangular Bezier patches. Click the **Preview** button to display the coarsened facets overlayed on the original geoemtry, Fig 6.8.13, and then click **Do Coarsen** to replace the original with the coarsened.

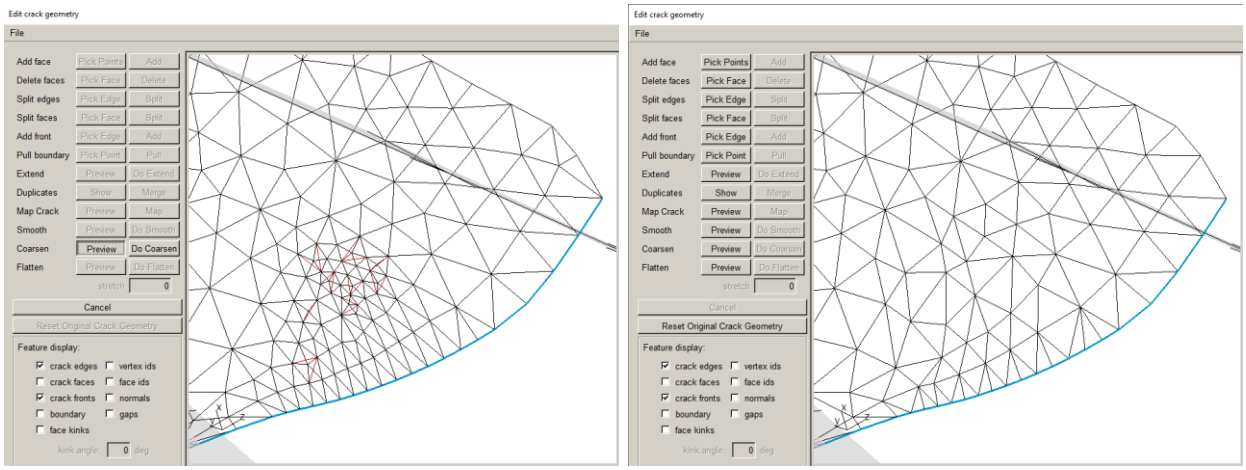


Figure 6.8.13 Coarsen crack surface geometry.

6.8.14 Trim and Fill

The Trim and fill option allows one to clean up the non-front crack boundary edges to produce a more-convex boundary. Click the **Preview** button to display the trimmed/added facets overlayed on the original geoemtry, Fig 6.8.14, and then click **Do Trim** to replace the original with the edited crack.

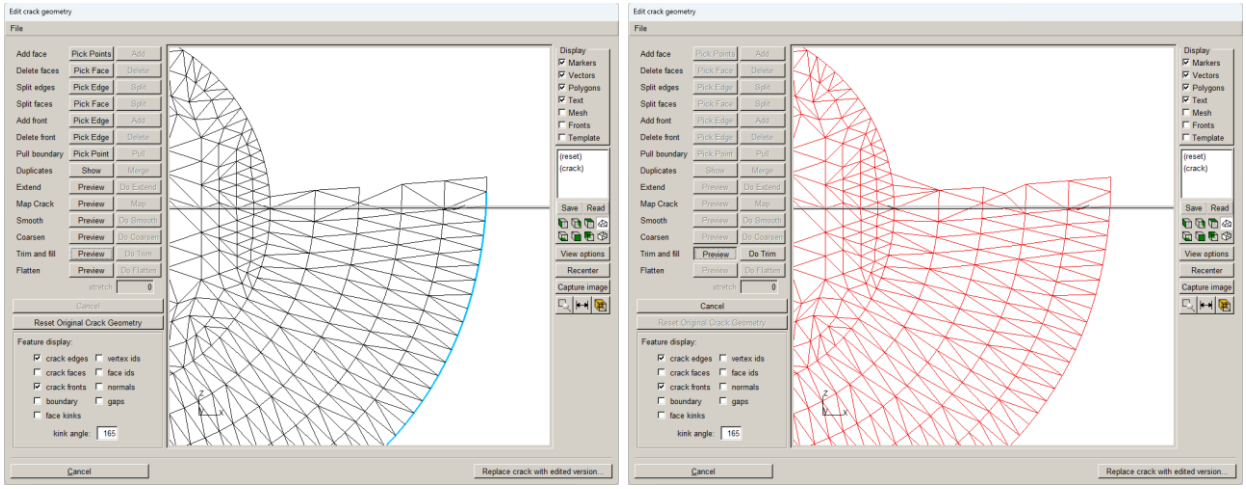


Figure 6.8.14 Trim and fill crack surface geometry.

6.8.15 Flatten

The Flatten option allows one flatten the current geometry to a planar crack. A least-squares planar fit of the crack surface is generated and displayed. Click the **Preview** button to display the flattened crack overlayed on the original geoemtry, Fig 6.8.15, and then click **Do Flatten** to replace the original with the flattened. The boundary of the flattened geometry can be stretched to make sure that it intersects with the model surfaces; the stretch value should be adjusted to prevent gaps along the model surface without overly distorting the geometry.

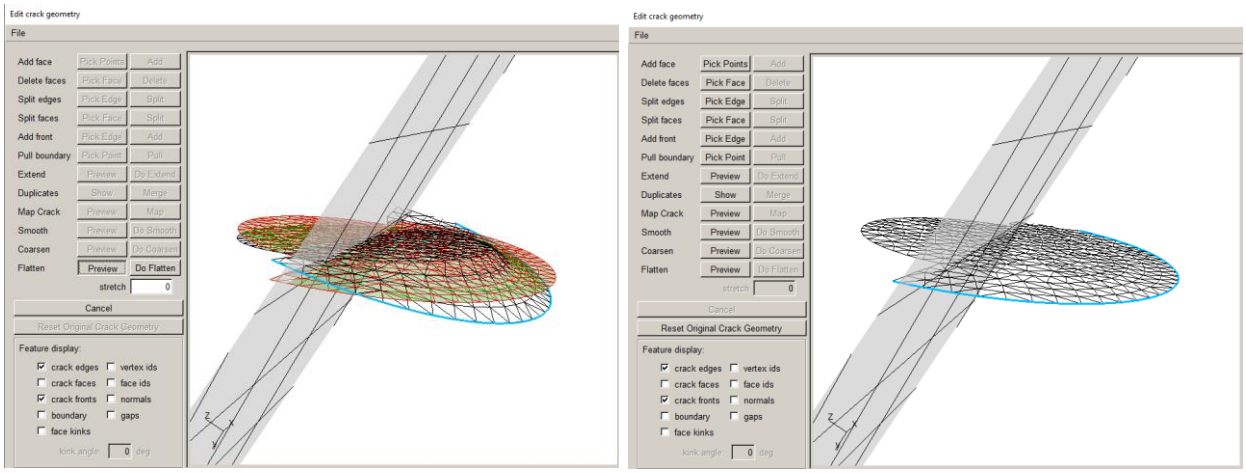


Figure 6.8.15 Flatten crack surface geometry.

6.8.16 Cancel

The **Cancel** button clears any current activity.

6.8.17 Reset Original Crack Geometry

The **Reset Original Crack Geometry** button resets the geometry to the original crack geometry that is first displayed in the dialog.

6.8.18 Feature Display

The Feature display pane has options to turn on/off display of various crack geometry features.

6.8.18.1 crack edges

This turns on/off the crack surface patch edges, Fig 6.8.16.

6.8.18.2 crack faces

This turns on/off the crack surface faces, Fig 6.8.17.

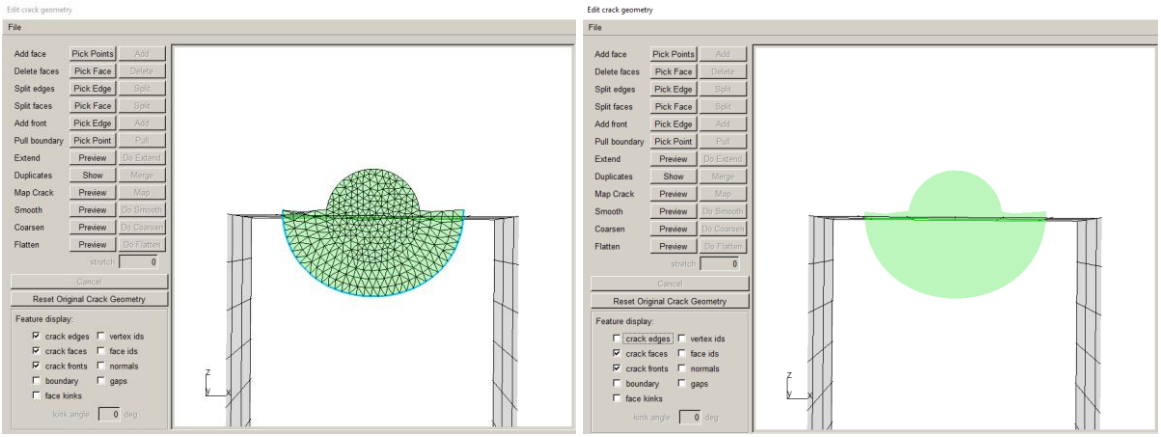


Figure 6.8.16 Crack edges on (left) and off (right).

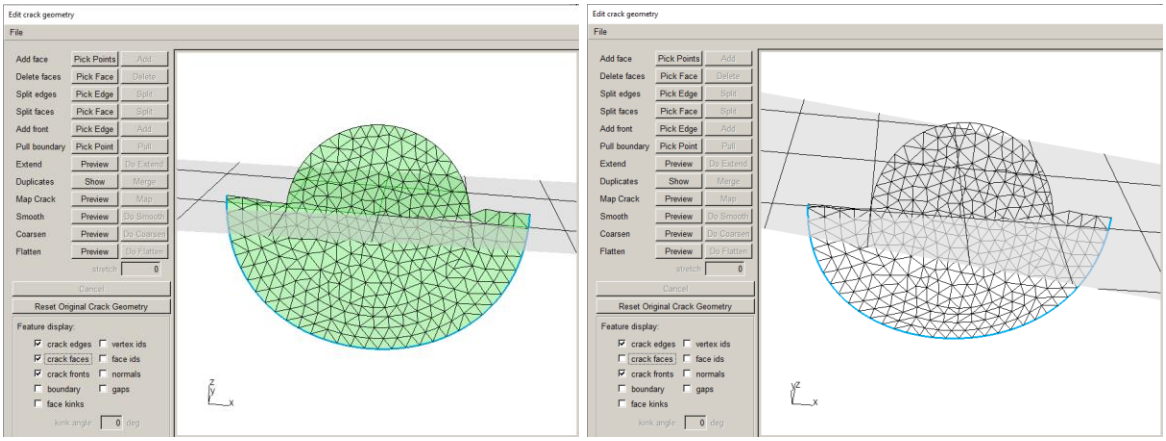


Figure 6.8.17 Crack faces on (left) and off (right).

6.8.18.3 crack fronts

This turns on/off the crack front, Fig 6.8.18.

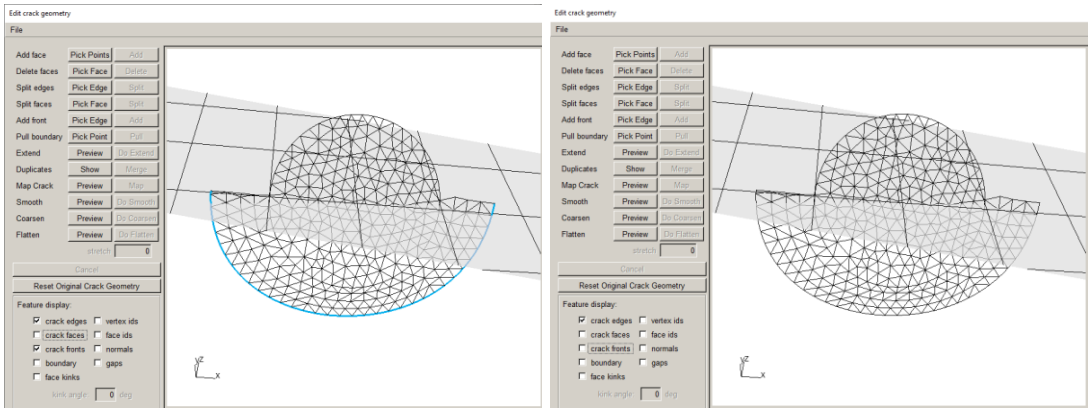


Figure 6.8.18 Crack front on (left) and off (right).

6.8.18.4 boundary

This turns on/off the crack boundary, Fig 6.8.19. This feature is useful for finding holes in the surface. The crack front will be displayed on top of the boundary; the front is off in Fig 6.8.19.

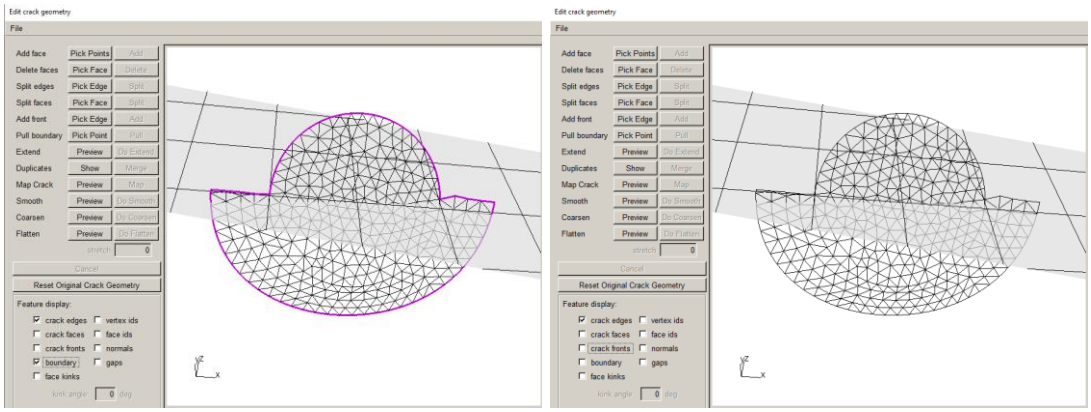


Figure 6.8.19 Crack boundary on (left) and off (right).

6.8.18.5 face kinks and kink angle

This highlights edges where facet normals differ, Fig 6.8.20.

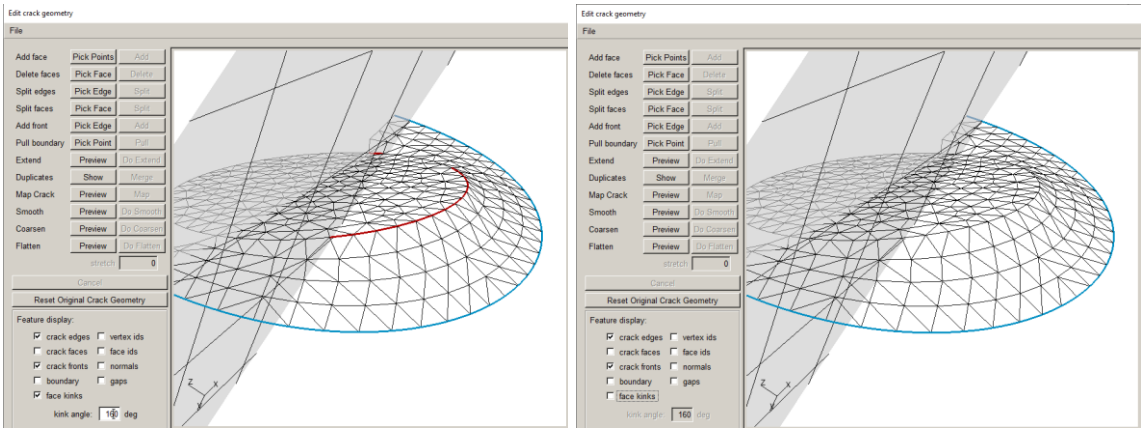


Figure 6.8.20 Face kinks on (left) and off (right).

6.8.18.6 vertex ids

This turns on the vertex IDs, Fig 6.8.21.

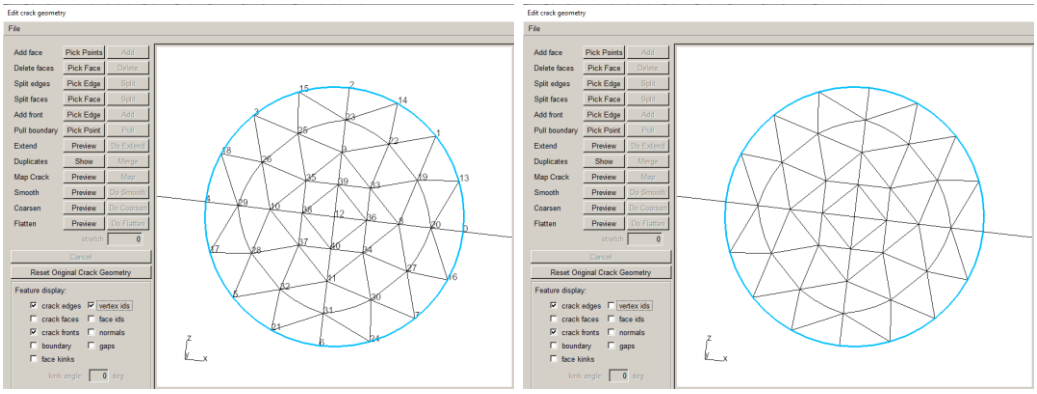


Figure 6.8.21 Vertex ID on (left) and off (right).

6.8.18.7 face ids

This turns on the face IDs, Fig 6.8.22.

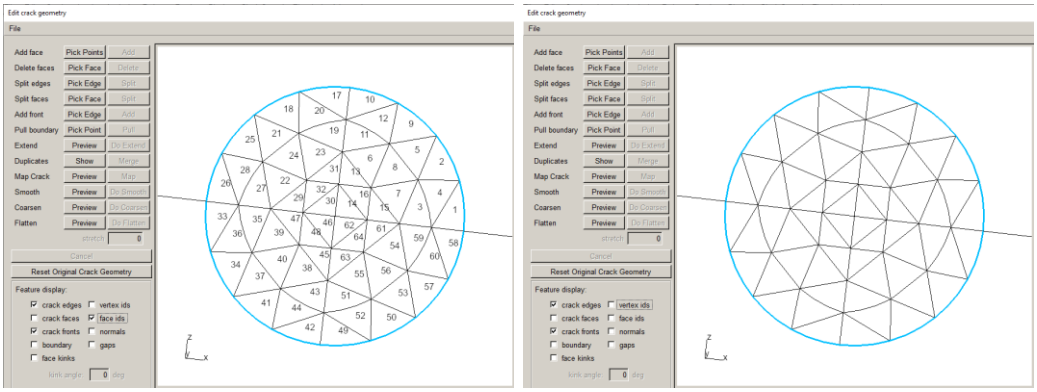


Figure 6.8.22 Face IDs on (left) and off (right).

6.8.18.8 normals

This turns on the face normals, Fig 6.8.23.

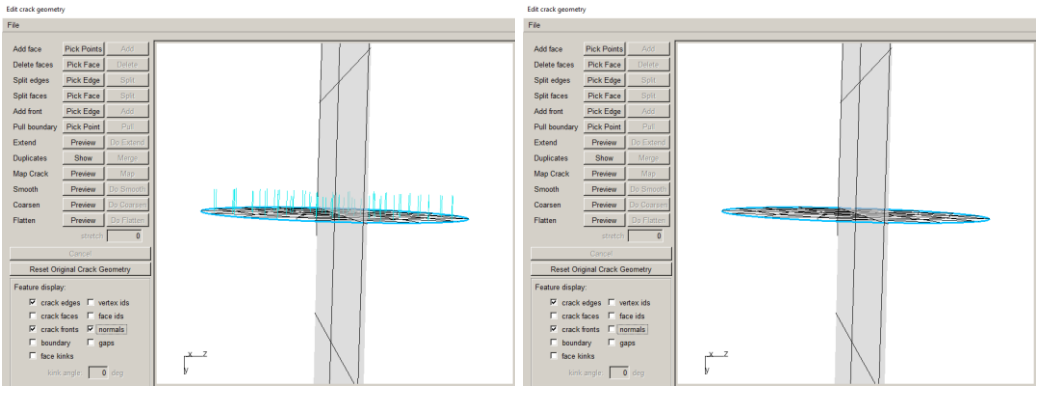


Figure 6.8.23 Face normals on (left) and off (right).

6.8.18.9 gaps

This highlights edges where the crack boundary is inside the model, Fig 6.8.24.

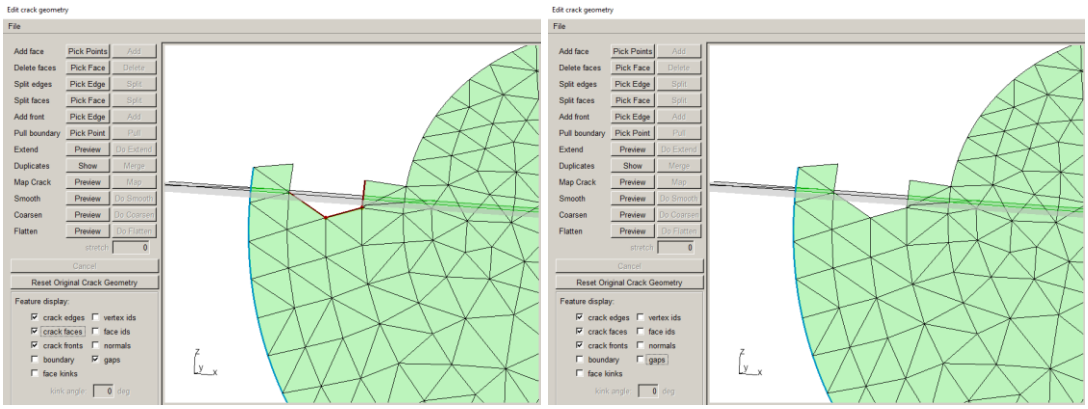


Figure 6.8.24 Gaps on (left) and off (right).

6.9 SIFs Along a Path

The SIFs along a Path dialog, Fig 6.9.1, allows the user to display SIF history data. The crack surface and crack fronts are displayed on the left-side, and the SIF history and path options are displayed on the right-side of the dialog. The tabs along the top of the XY plot allow one to display the three SIF modes (KI, KII and KIII) as well as the J-integral and T-stress values. The Define Path and Export tabs are described in the following sub-sections.

Above the tabs is a drop-down menu that allows the user to specify the load step for which SIF history is plotted. If there is only one load step, the drop-down menu is inactive. This is like the drop-down menus in the SIF plot dialog (see Section 6.4). The substeps dropdown is not available here; only results for the last substep for each load step can be displayed. The crack front is set from the **Path** tab.

6.9.1 Define Path Tab

Traditional fatigue lifing methodologies require a SIF history that consists of a single valued stress intensity factor (K) and a single valued crack size (a) for each crack increment. Developing such a "K-history" is straightforward for a 2-D analysis but is more complicated for 3D analyses. In 3D, one has a distribution of K values along a crack front, and there might not be an obvious crack dimension that uniquely characterizes the crack “length”.

Three different Path Type algorithms are available in the Define Path tab, Fig 6.9.2. The results for the three algorithms are illustrated below using three different crack growth simulations. It should be noted that while the crack growth might appear planar in these three examples, the algorithms work for non-planar growth.

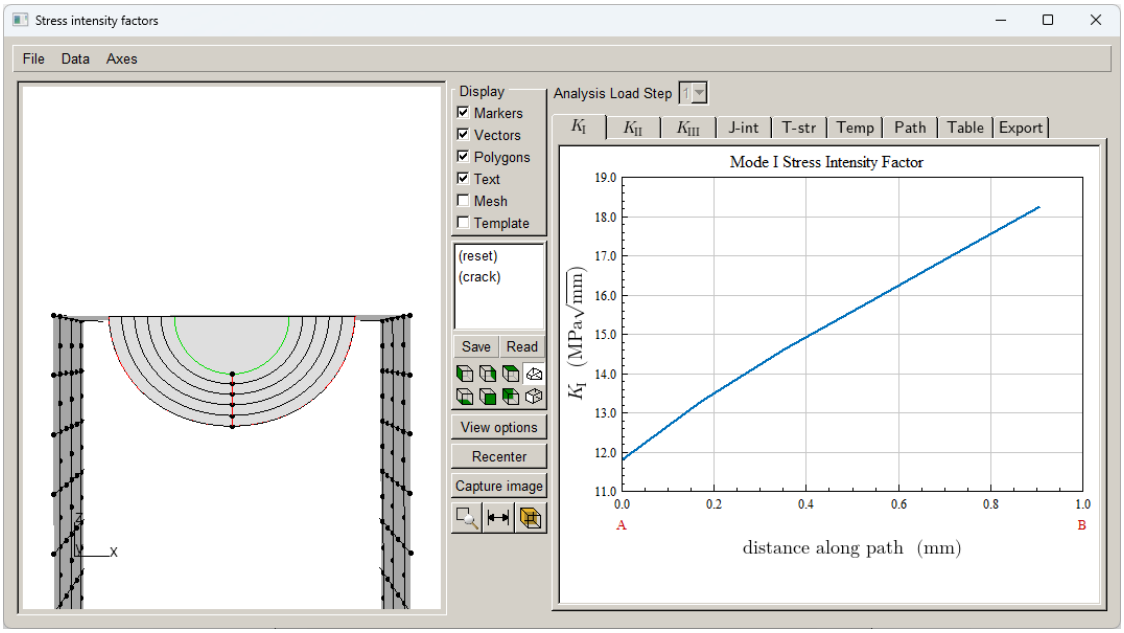


Figure 6.9.1 SIFs along a path dialog.

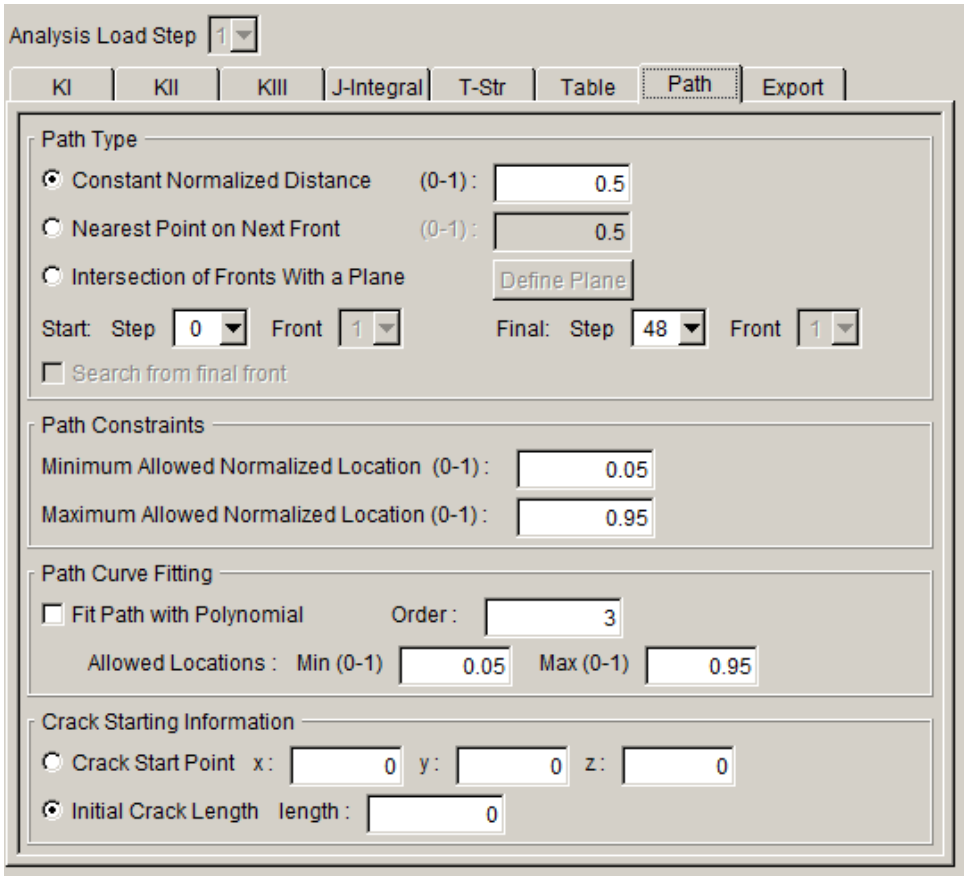


Figure 6.9.2 Define path tab.

6.9.1.1 Constant Normalized Distance

The first algorithm uses a constant, user-specified, normalized distance along the crack fronts. Fig 6.9.3 shows the computed paths for 50% normalized distance. The figure shows that this approach works very well for the first crack and is reasonable for the third crack. For the middle crack, however, the path is sinuous; this can be even more pronounced for crack fronts that transition from corner- to through-cracks.

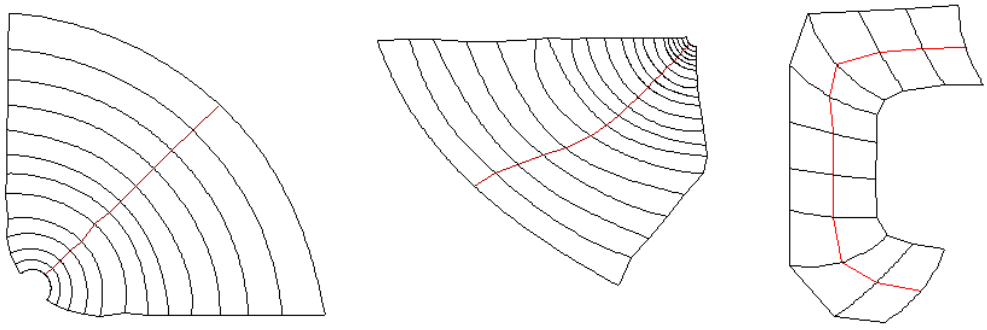


Figure 6.9.3 Computed K paths for a constant normalized distance of 50%.

6.9.1.2 Nearest Point on Next Front

The second algorithm is a modification of the first approach. The analyst selects a point on the initial crack front. The algorithm then searches for the nearest point on the next crack front and makes this the next point in the history. The process is repeated for all remaining crack fronts. The Ks in the history are the interpolated values at the "nearest points", and the crack length is the distance from the crack "origin" to the initial crack front point plus the (straight line) distances between successive "nearest points".

This algorithm is illustrated in Fig 6.9.4. For all three cracks, the path was started at locations of 10%, 20%, ... 90% along the initial crack front. The algorithm performs very well for the first crack. For the middle crack, some of the paths converge into one path at 10% of the distance along the crack front. This is because of the path constraints, which are described in Section 6.9.1.2. For the third crack, most of the paths merge due to these path constraints.

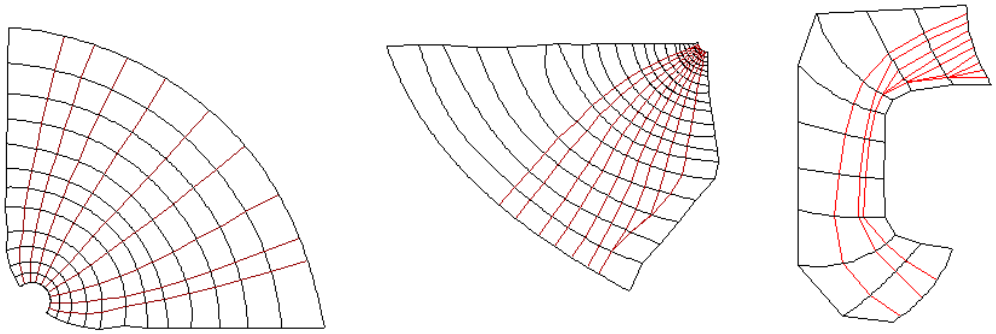


Figure 6.9.4 Computed K paths using nearest next point. Paths are at 10% increments with a 10% and 90% minimum and maximum threshold.

6.9.1.3 Intersection of Fronts with a Plane

The third algorithm defines a plane nominally perpendicular to the crack surface and gives the path as the points where the crack fronts intersect the plane. This algorithm is illustrated for the first of the three cracks above in Fig 6.9.5. It is not clear how one would apply this algorithm to the third crack in Fig 6.9.4.

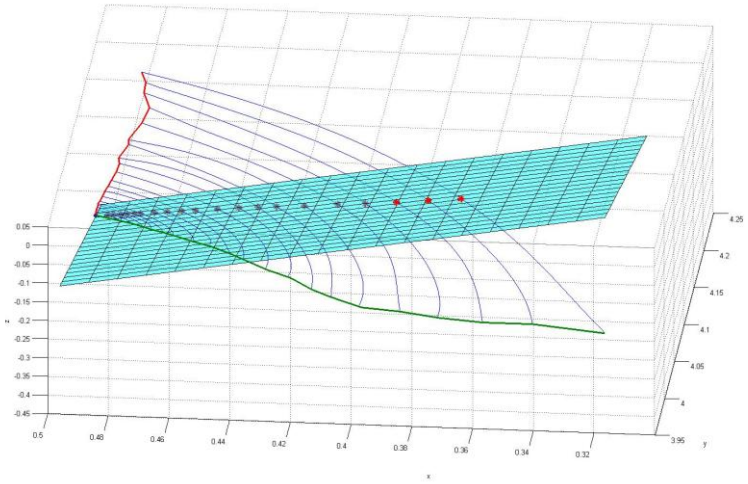


Figure 6.9.5 Computed plane intersection points for a crack.

6.9.1.3.1 Start: Step and Final: Step

For crack fronts that might split, merge, or disappear, there is the option to specify both the starting and ending crack step number and the crack front number, if there is more than one front, Fig 6.9.6.

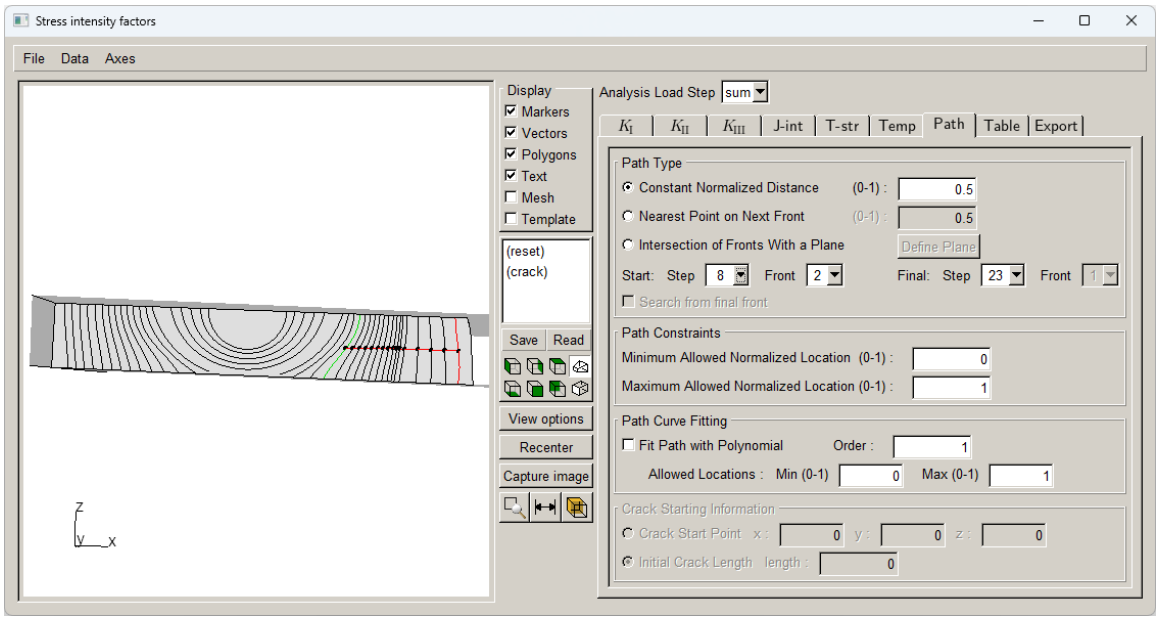


Figure 6.9.6 SIF path starting at step 8 for front 2.

6.9.1.3.2 Search from final front

The Search from final front option is active for the Nearest Point on Next Front. The algorithm starts at the last front and searches 'backwards' for the nearest point on the previous front.

6.9.1.4 Path Constraints

There is a user-settable threshold that prevents the computed paths from getting too close to the free surfaces where the accuracy of the computed stress intensity factors is questionable. The user can set both the minimum and maximum thresholds in the dialog.

6.9.1.5 Path Curve Fitting

Path curve fitting provides a best-fit, in a least-squares sense, through the path computed by one of the three algorithms. Fig 6.9.7 shows the least-squares fit to the paths shown in Fig 6.9.3. A quadratic curve was used for the first crack, and as expected, there is modest improvement in the path for this case. A cubic curve was used for middle crack, which yields a path that is aesthetically more pleasing. A fifth order curve was used for the third crack and improves the computed path by smoothing the kinks.

6.9.1.6 Crack Starting Information

The crack starting information has two options: crack start point and initial crack length. Setting either of these provides a starting point for computing the total crack path length.

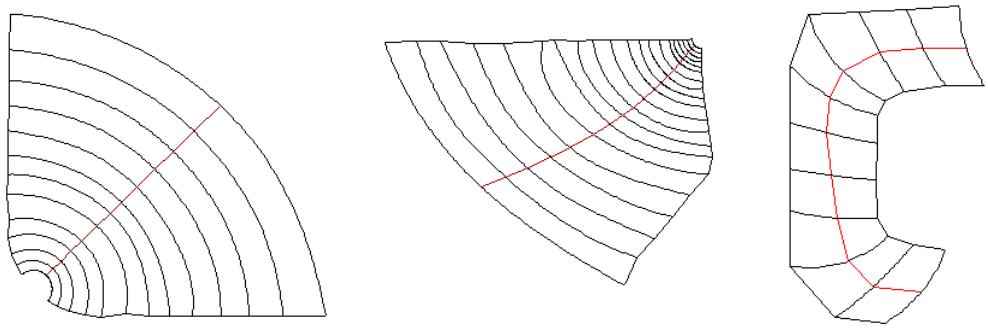


Figure 6.9.7 Computed least-squares fit to the paths of Fig 6.9.3.

6.9.2 Export Tab

The Export tab, Fig 6.9.8, allows one to save the SIF data to a file. The normalized coordinates seen in the XY plot can be exported to regenerate the XY plots. The Cartesian coordinates of the mid-side nodes along the crack front along with the crack front local basis can also be exported. One could use this information to compute crack growth or to plot crack front geometry outside of FRANC3D. A WriteSifPath command is issued to the session log; see Section 2.1.69 in the Commands & Python reference.

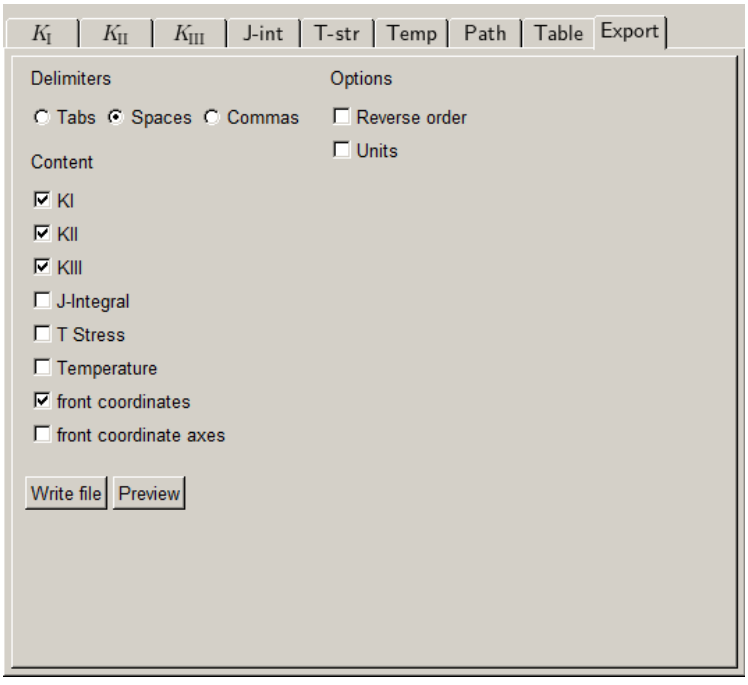


Figure 6.9.8 SIF history dialog – Export tab.

6.9.3 File/Data/Axes Menu

Fig 6.9.1 shows the menu bar with three menu items: File, Data and Axes. These menus are the same as the menus for the SIF Plot dialog (see Section 6.4.2.1).

6.10 SIFs For All Fronts

The SIFs for all Fronts dialog, Fig 6.10.1, allows the user to display SIF data for all fronts in a single plot. The crack front lines are displayed in the left-side model window, and the SIFs for all crack fronts are displayed on the right-side of the dialog. The tabs along the top of the XY plot are the same as the tabs for SIFs Along a Path (see Section 6.9).

There are two drop-down menus that allow the user to specify the load step and crack front for which SIF data is plotted. The substeps dropdown is not available here; only results for the last substep for each load step can be displayed. The Crack Front drop-down is active if there are multiple cracks in the model.

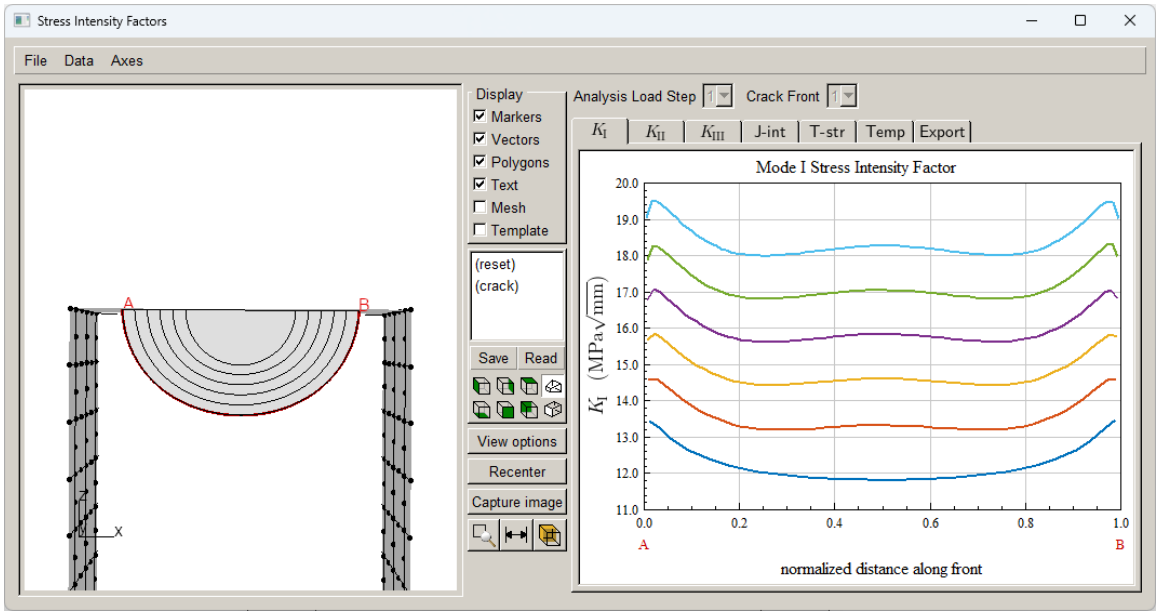


Figure 6.10.1 SIFs for all fronts dialog.

6.10.1 Export Tab

The Export tab displays the dialog in Fig 6.10.2; this is only slightly different from that in Fig 6.9.3. There is an option to reverse the order of the points, so that data is exported going from point B to A.

6.10.2 File/Data/Axes Menu

Fig 6.10.1 shows a menu bar with three menu items: File, Data and Axes. These menus are the same or like the menus for the SIF Plot dialog (see Section 6.4.2.1).

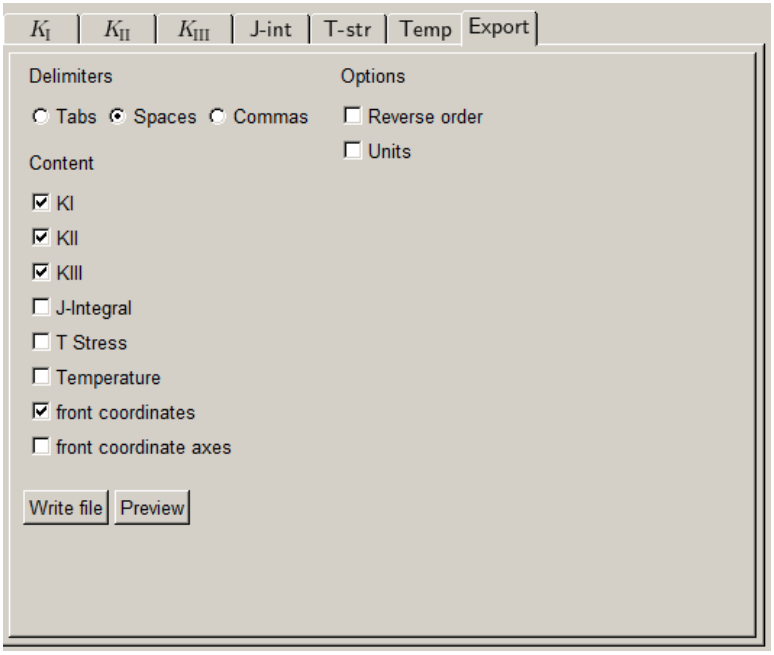


Figure 6.10.2 Export tab for SIFs for All Fronts dialog.

6.11 Js For All Rings

The Js for all Rings dialogs, Fig 6.11.1-2, allows the user to display J-integral data. First, the user can choose to use the displacements to compute elastic stresses, which will be used to compute J. Alternatively, nonlinear stress can be read from a file to compute J. Select Accept in the first dialog to display the J-integral dialog, Fig 6.11.2.

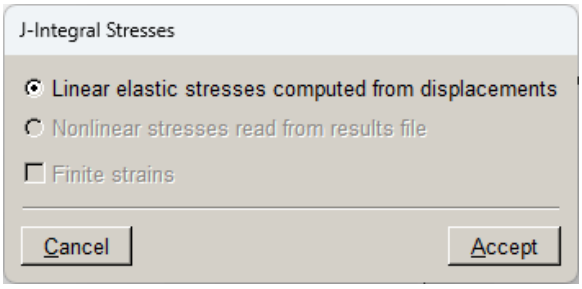


Figure 6.11.1 Js for All Rings dialog.

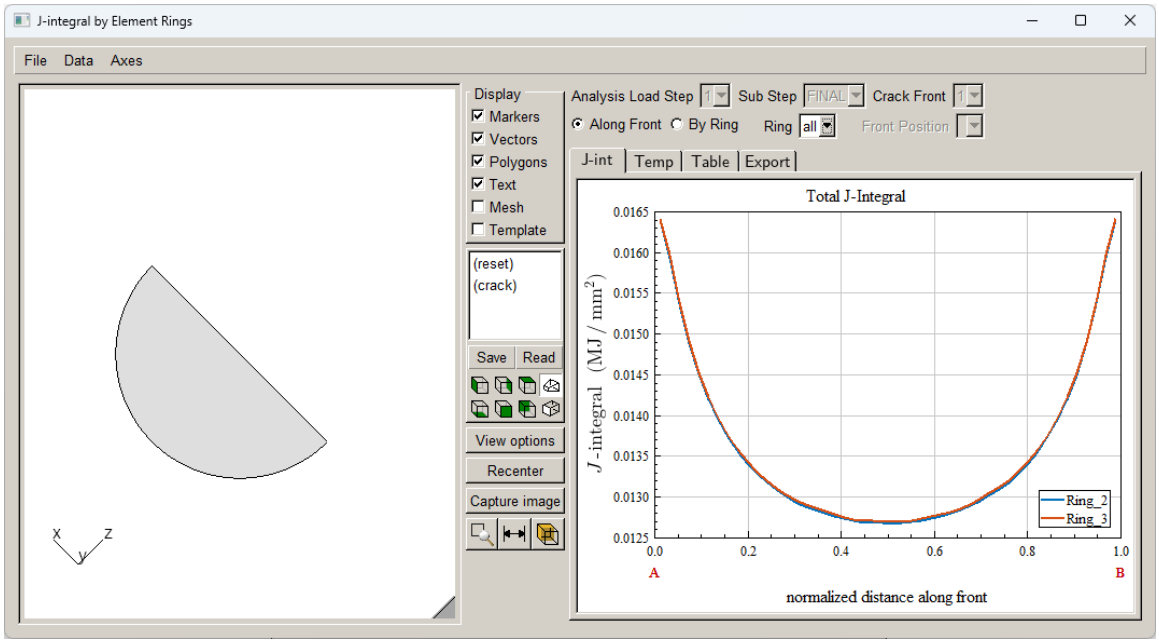


Figure 6.11.2 Js for All Rings dialog.

J-integral values can be plotted along the crack front, using the normalized position. Alternatively, values can be plotted at a single normalized position versus element (or integral) ring. This allows a visual indicator of convergence – if the J reaches a constant value.

The tabs above the plot are similar to the SIF plot (see Fig 6.4.3). Load step, load substep, crack front and crack step can be set if there are multiple values. The Table tab and Export tab allow the user to view the data in a table and to export the data to an ASCII file.

7. Wizards and Dialog Boxes for Loads Menu

The wizards and dialog boxes for the **Loads** menu options are described in this section.

7.1 Crack Face Pressure/Traction

The Loads **Crack Face Pressure/Traction** (CFT) option allows one to define crack face pressures or tractions. This capability is often used to simulate residual stresses. It can also be used in a sub- modeling approach where the submodel containing the crack is analyzed using crack face tractions based on the stresses from an uncracked "global" model.

The first panel of the wizard is shown in Fig 7.1.1. Use the **Add** button to create a new entry in the list. Once an entry has been added, it can be edited or deleted by selecting the name and then the **Edit** or **Delete** button.

The Crack Face Traction/Residual Stress Type panel is displayed when the **Add** button is clicked, Fig 7.1.2; it allows one to choose the type of crack face loading. The choices are: 1) Constant crack face pressure, 2) 1-D Radial residual stress distribution, 3) 2-D Radial residual stress distribution, 4) Surface treatment residual stress distribution, and 5) Residual stress defined on a mesh. These will be described in more detail below.

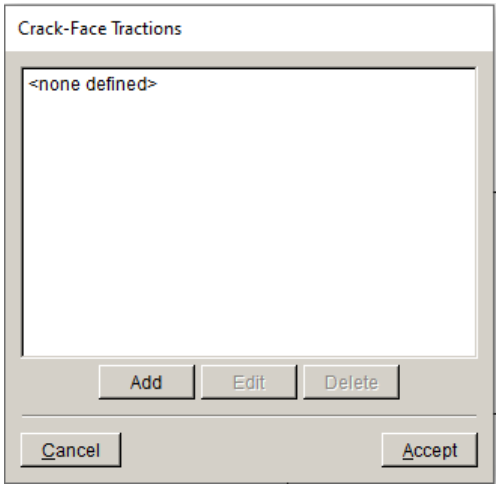


Figure 7.1.1 Crack Face Tractions – top level dialog.

The **Advanced** button at the bottom allows one to specify whether the CFT is applied in a new load step, after all the original FE load steps are applied, or added to one of the existing FE load steps. The dialog is slightly different for ABAQUS compared with ANSYS and NASTRAN; the dialog display is based on the input FE model that is being cracked/analyzed.

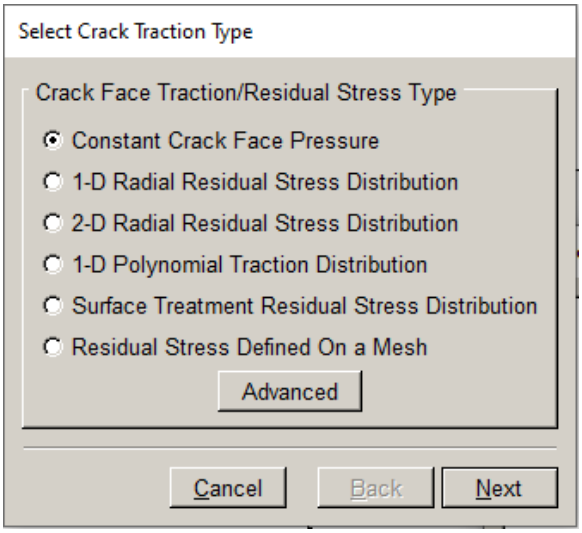


Figure 7.1.2 Crack Face Tractions – type dialog.

7.1.1 Advanced button

The Advanced options dialog is shown in Fig 7.1.3. The left image is for ABAQUS models, and the right image is for ANSYS and NASTRAN models. The default is to create a new load step for the CFT. However, one can change this so that the CFT is added to an existing load step. If the Add to existing load step is chosen, the user can then select the load step that will be used. For ABAQUS, CFT can be applied using an amplitude function so that the CFT is applied the same as the existing loads. However, there are limitations; see Section 12 of the User's Guide.

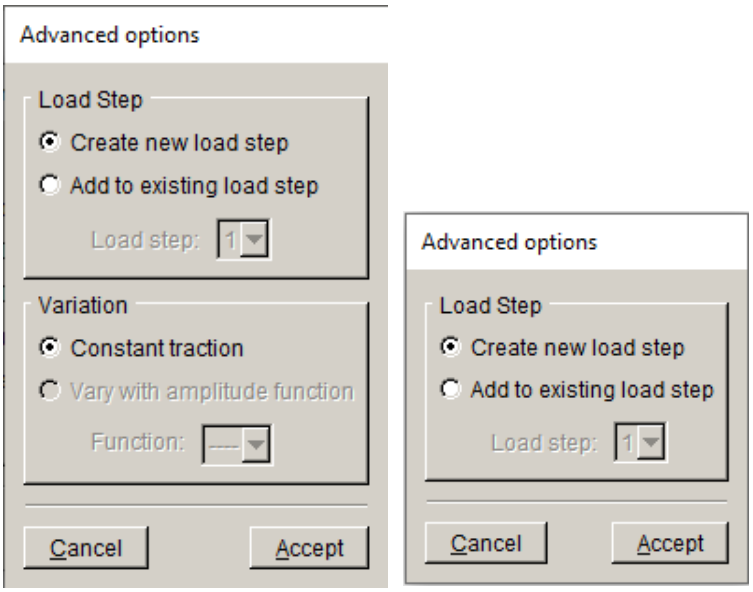


Figure 7.1.3 Advanced options dialog to set the analysis load step (left side is for ABAQUS).

7.1.2 Constant Crack Face Pressure Panel

The constant crack face pressure panel allows one to specify a uniform pressure on the crack face, Fig 7.1.4. A positive pressure value will tend to open the crack. If there are no nodal temperatures, the **Next** button will return the display to the top-level panel (see Fig 7.1.1). If there are nodal temperatures, the **Next** button will display the Set Temperature dialog, Fig 7.1.5.

The temperature for the crack face traction load step can be set to a Constant value, such as the reference temperature, or it can be set based on the last load step, or it can be set from an external source. The External data option allows one to specify an ABAQUS *.inp*, ANSYS *.cdb* or NASTRAN *.bdf* (or *.nas* or *.dat*) file depending on the original model type. The temperature can also be included from the *.dtp* file if that is used for the stress for the Residual Stress Defined on a Mesh option.

The Set Temperature panel is the same for all crack face traction types. The CFT and effects of temperature are described in more detail in Section 12 of the User's Guide.

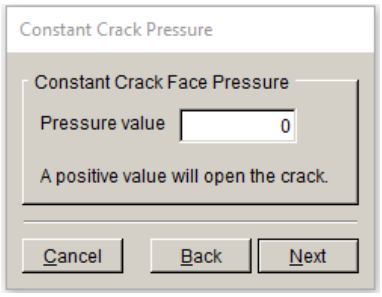


Figure 7.1.4 Constant Crack Face Pressure panel.

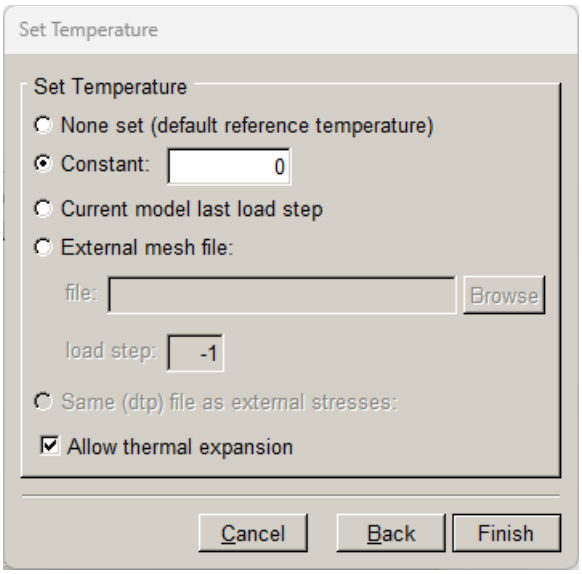


Figure 7.1.5 Set Temperature dialog.

7.1.3 1-D Radial Residual Stress Distribution Panel

The 1-D radial residual stress distribution panel allows one to specify a radial distribution of stress that can be applied to the crack face, Fig 7.1.6. The stress varies along a radius from some origin. The user can specify the distribution axis as well as the axis offset.

The distance vs stress data can be entered into the dialog or read from a file, using the **Read From File** button. The file is a simple ASCII text file with lines of data that matches the dialog. Use the **Save To File** button to save the data from the dialog to a *.txt* file. The Read Only option disables the interactive editing of the data in the table.

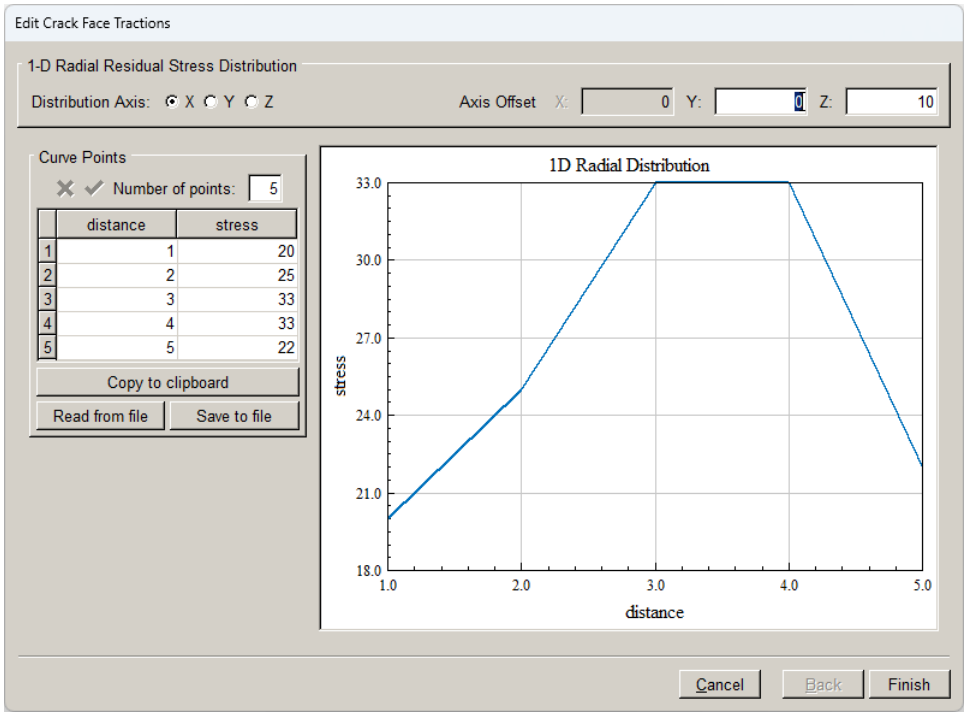


Figure 7.1.6 1-D radial residual stress distribution dialog.

For example, consider a corner crack in a brick, Fig 7.1.7 – left panel. The crack has a radius of 2.0 and the corner is located at x=0, y=5, and z=5. A 1-D radial distribution is defined, Fig 7.1.9 – right panel, with the distribution axis set to y and the axis offset defined as x=0 and z=5. The crack face traction distribution is shown in Fig 7.1.8 – left panel. To visualize the pressure that is applied to the crack, the ANSYS color contours for the stress in the y-direction on one side of the crack are shown in Fig 7.1.8 – right panel. This image shows the radial pattern of the pressures with the origin at the crack corner.

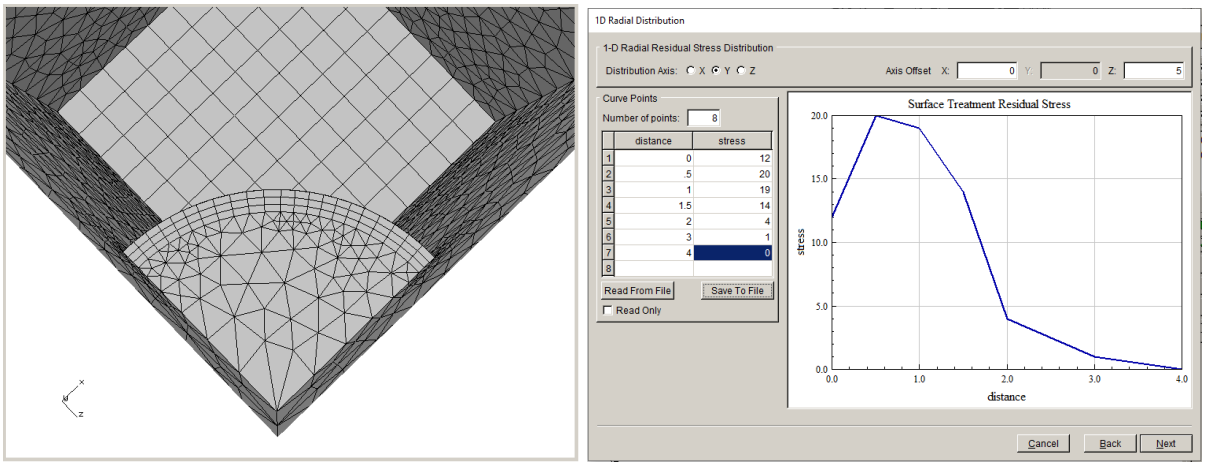


Figure 7.1.7 A corner crack in a brick (left panel) with a 1-D radial residual stress (right panel).

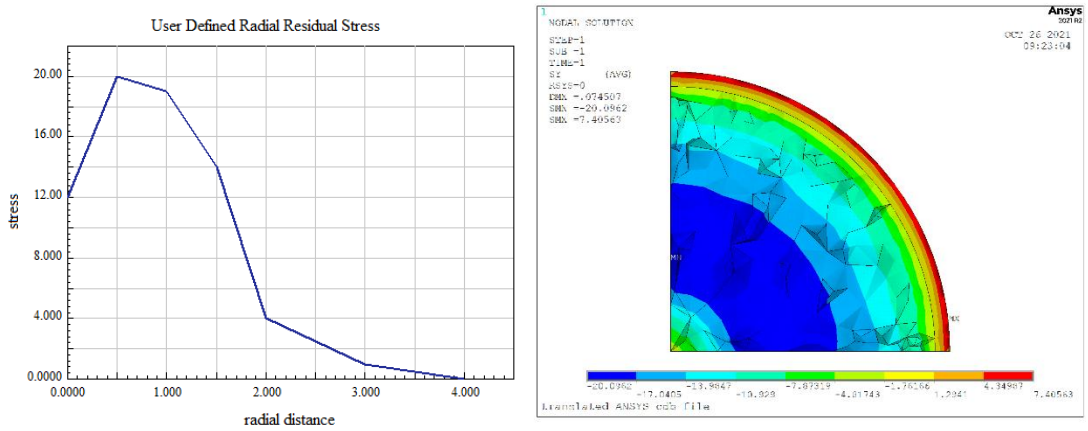


Figure 7.1.8 A 1-D radial residual stress distribution (left panel) applied to the corner crack and corresponding y-stress color contours in ANSYS (right panel).

7.1.4 2-D Radial Residual Stress Distribution Panel

The 2-D radial residual stress distribution panel allows one to specify a stress distribution that varies in two directions: axial and radial, Fig 7.1.9. The stress varies with radius from some origin. Use the **Read Data From File** button to import the data into the dialog.

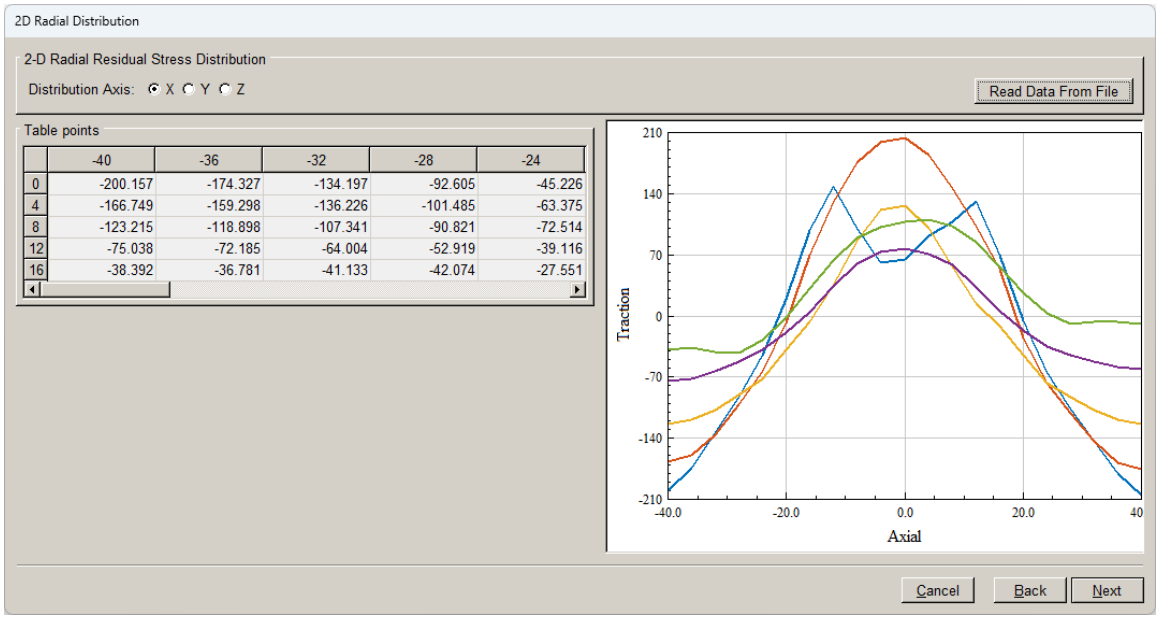


Figure 7.1.9 2-D radial residual stress distribution panel.

The data in the file should be arranged in the same way as it appears in the dialog, Fig 7.1.10. There is a blank space at the top left. The dialog is set as read-only, and there is no option to switch this off; you should edit the file instead.

	4	5	6
8	0.5	0.7	0.9
9	0.8	0.9	1
10	1	1	1

Figure 7.1.10 2-D radial residual stress data from file.

7.1.5 1-D Polynomial Traction Distribution Panel

The 1-D polynomial traction distribution panel, Fig 7.1.11, allows one to specify a shear traction on a planar crack surface.

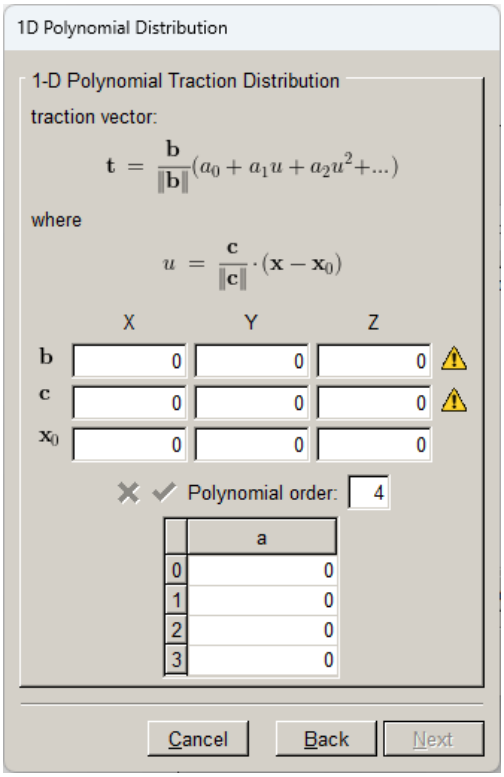


Figure 7.1.11 1-D polynomial traction distribution dialog.

7.1.6 Surface Treatment Residual Stress Distribution Panel

The surface treatment residual stress distribution panel, Fig 7.1.12, allows one to specify a stress distribution that varies in a direction that is normal to a surface that has been retained as a "residual stress surface" when reading the FE file. The stress varies with distance from the surface.

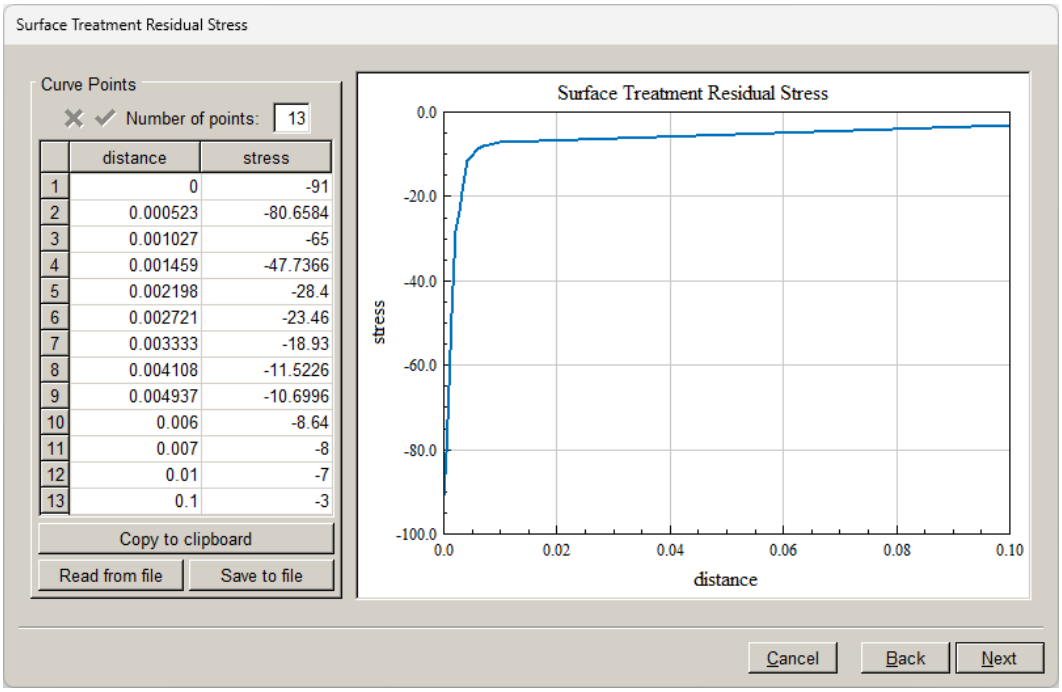


Figure 7.1.12 User defined surface treatment residual stress panel.

Data can be imported using the **Read From File** button; the file is ASCII text and arranged in the same way as shown in the table with each line containing a distance and a stress value. The **Save To File** allows the data to be saved to a *.txt* file. The table can be set to read-only using the Read Only option.

After entering the stress versus distance, the user presses **Next** to select the treated surface, Fig 7.1.13. This dialog is the same as that for selecting surfaces to be retained when importing a FE model (see Section 4.5).

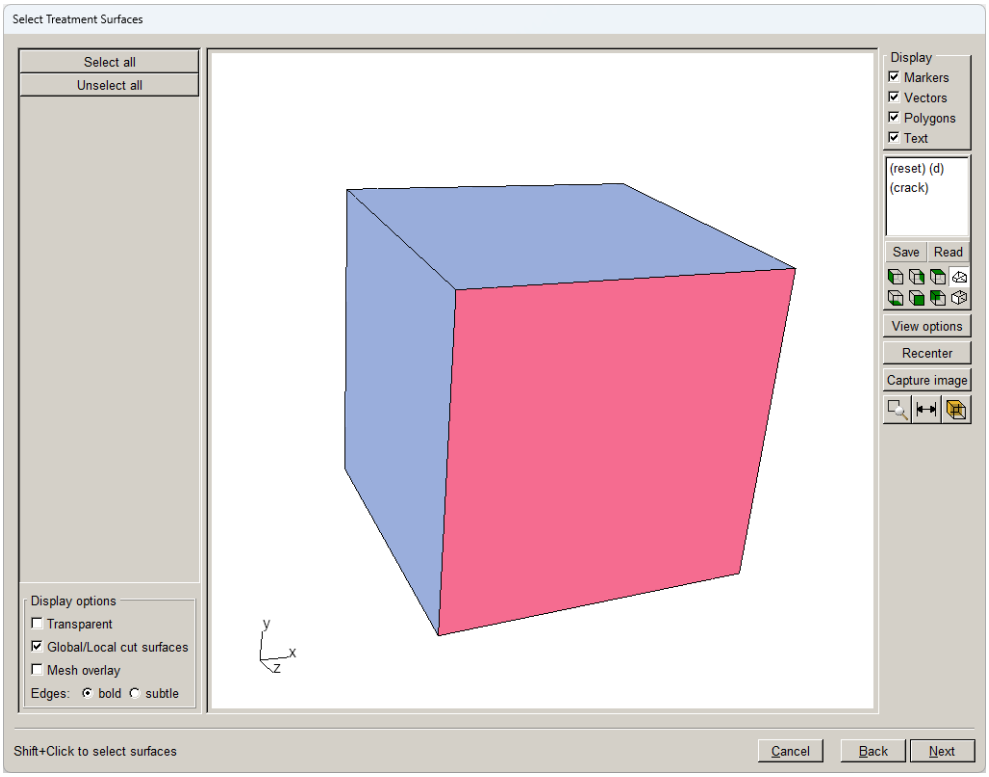


Figure 7.1.13 Surface treatment residual stress – select surface dialog.

As an example, a brick model is defined with the top surface (positive y-axis) defined as a "treated_surface", Fig 7.1.14 – left panel. The stress distribution shown in Fig 7.1.12 is applied as the surface treatment crack face traction. The distribution acts normal to the "treated_surface", and distance will be measured in the -y direction. The resulting crack face traction distribution on the crack shown in Fig 7.1.14 (left panel) can be visualized using ANSYS, Fig 7.1.14 – right panel.

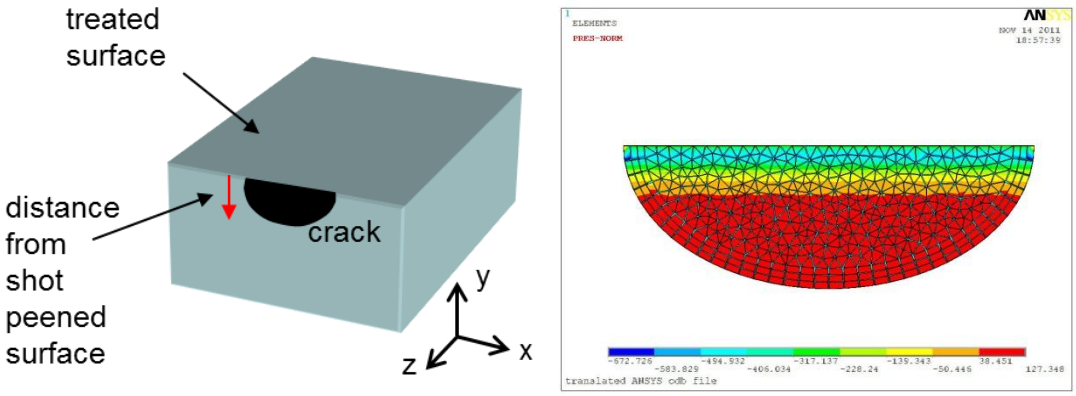


Figure 7.1.14 Example of a surface treatment residual stress crack face traction.

A few things are noted here. Consider the crack shown in the left panel of Fig 7.1.15 and the residual stress distributions shown in the right panel of Fig 7.1.15. First, if the nodes on the crack surface are at a depth that is greater than the residual stress, no crack face tractions are computed. Using the 'a' distribution, the lower third of the crack will not have crack face tractions. Second, if the mesh on the crack surface is coarse compared to the residual stress distribution, the computed crack face traction will not adequately capture the residual stress distribution. Based on the coarse mesh shown in the left panel of Fig 7.1.16, the computed traction value at the node indicated by the red circle, using the 'a' distribution from Fig 7.1.15, would be close to zero. If a pseudo-refined mesh is defined, as in the middle and right panels of Fig 7.1.16, then the 'a' distribution can be captured and appropriately distributed onto the nodes of the actual coarse mesh. Ideally, one would use a suitably refined mesh in the model to avoid this problem.

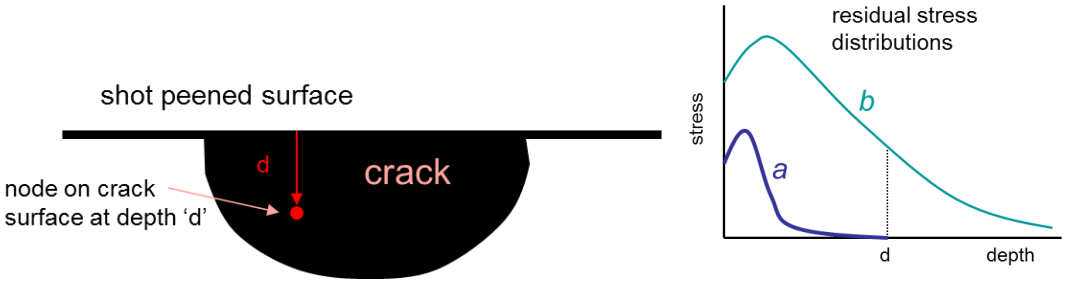


Figure 7.1.15 Surface treatment residual stress applied as crack face traction up to the defined depth of the distribution.

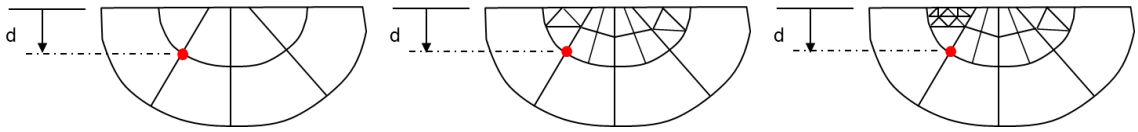


Figure 7.1.16 Surface treatment residual stress applied as crack face traction with automatic pseudo-mesh refinement to capture the distribution.

7.1.7 Mesh-Based Stress Distribution Panel

The mesh-based stress distribution panel allows one to specify a stress distribution based on a FE mesh and stress file, Fig 7.1.17. The user chooses the mesh and associated stress file; normally, this is an uncracked model. FRANC3D queries this data to compute tractions on the crack face.

The mesh and associated stress results do not have to correspond exactly with the model used for crack insertion; however, the mesh and stress must occupy the same physical space as the crack. At each node on the crack surface mesh, FRANC3D queries the stress-mesh to find the element and associated nodes that contains the point and then retrieves the stress for these nodes. The element shape functions are used to interpolate stress at the crack surface node location.

For mesh based CFT, one can apply the results for any of the three analysis codes to the current model. For example, one can apply ANSYS stress to an ABAQUS crack growth simulation.

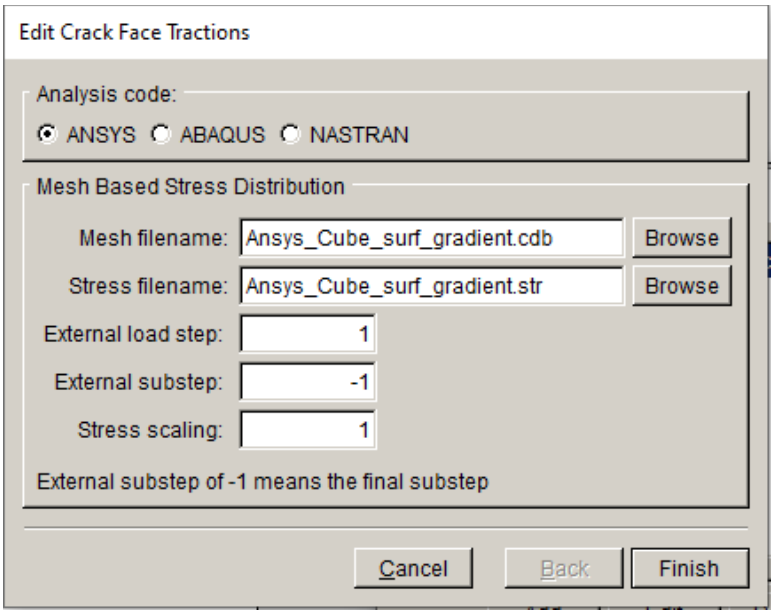


Figure 7.1.17 Mesh based stress distribution panel.

If the chosen FE model (and results) contain multiple load steps, the user should specify the (external) load step (and for the substep if there is more than one per load step). The -1 in the substep field indicates that FRANC3D will use the last substep. If you are using an *.odb* file for the stress and the *.odb* contains many load steps, you should consider extracting the stress for only the load step(s) of interest.

The stress can be scaled; a user might instead choose to scale the SIFs that result from the CFT load step, but if the CFT is added to an existing load step, then the scaling might need to be applied here.

7.2 View Boundary Conditions

The Loads **View Boundary Conditions** option allows one to view the boundary conditions attached to the imported FE model, Fig 7.2.1. The load steps and the boundary conditions attached to each load step can be listed and highlighted on the model.

This dialog allows the user to verify that the FE model has been imported correctly. One should still verify that the initial crack model displacements are consistent with the uncracked model displacements.

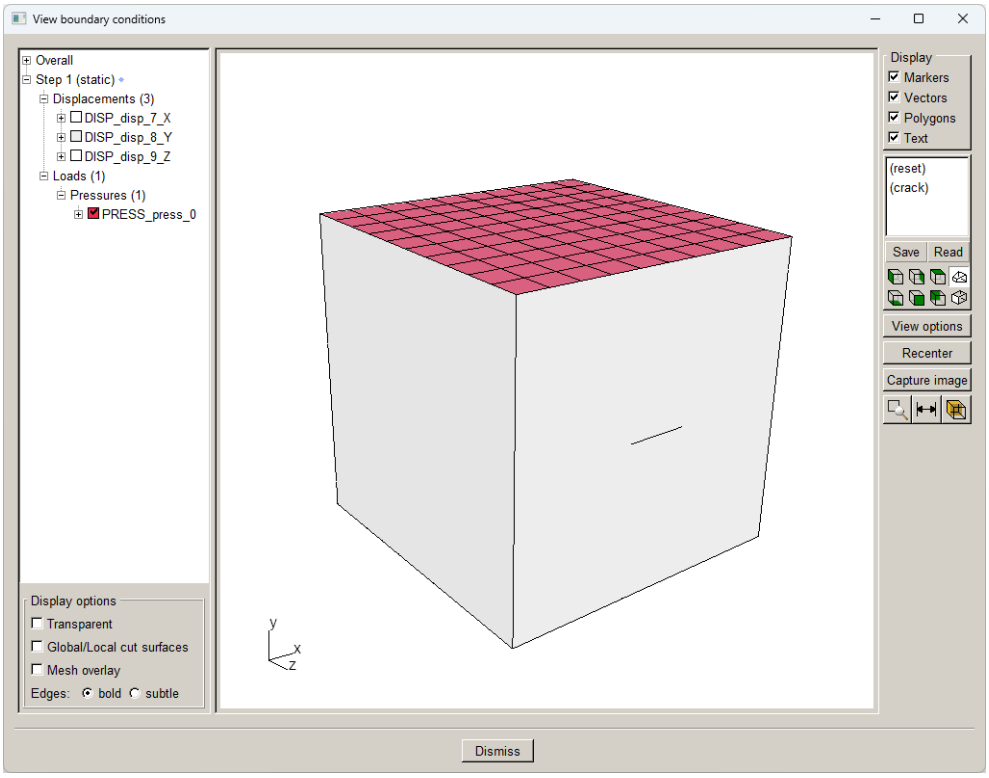


Figure 7.2.1 View Boundary Conditions dialog.

7.3 Plot CFT Stress File

Plot CFT Stress File allows one to contour crack face traction values based on mesh-based stress input. See Section 7.1.7 for a description of the dialog for entering the mesh-based stress input. First, specify the user mesh and results files, Fig 7.3.1.

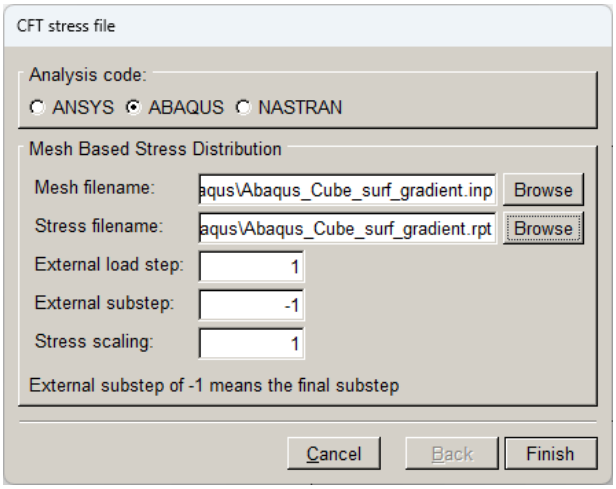


Figure 7.3.1 User-mesh input dialog.

Once the data is imported, the dialog shown in Fig 7.3.2 is displayed. The traction values on the crack surface, computed from the input stress, are contoured. This dialog is intended to be used to verify that the stresses are being extracted and applied as expected.

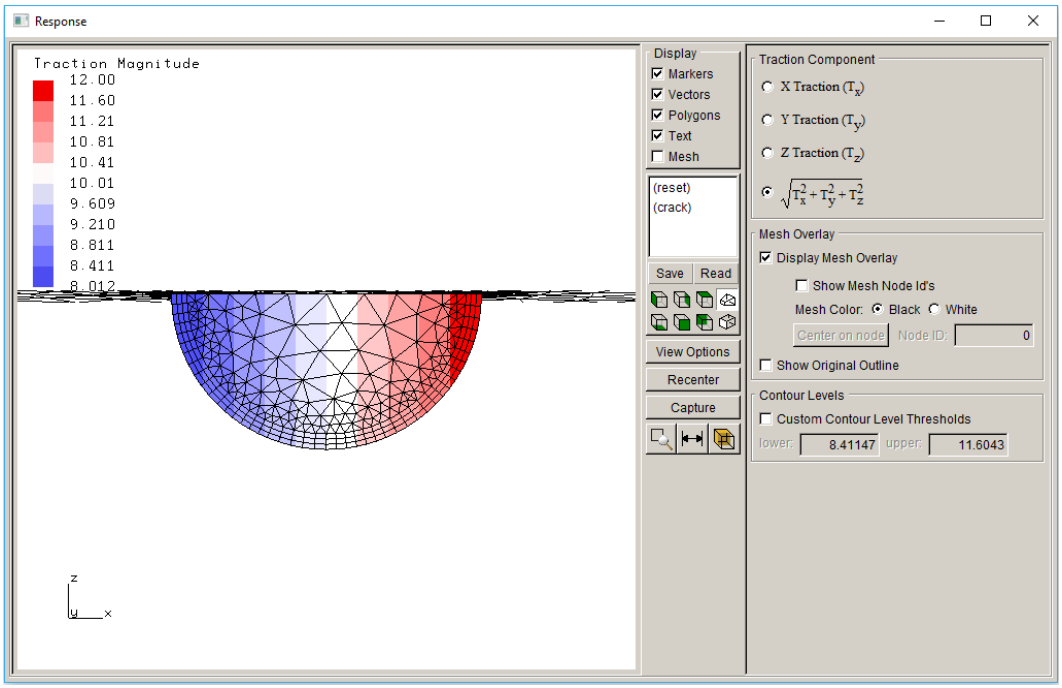


Figure 7.3.2 Plot CFT stress dialog.

8. Wizards and Dialog Boxes for Analysis Menu

The wizards and dialog boxes for the **Analysis** menu options are described in this section.

8.1 Static Crack Analysis

The **Static Crack Analysis** menu item allows one to set up and run a static (current crack configuration with no crack growth) deformation analysis. One would use this, for example, if one had inserted a crack into a body and wanted to compute stress-intensity factors for this crack without performing any crack growth. The ANSYS, ABAQUS and NASTRAN wizard panels are discussed below; the wizard panels are similar for all the analysis codes, but there are some differences as will be noted below. The analysis code is based on the original imported model type.

8.1.1 Fdb File Name Panel

The first panel, Fig 8.1.1, allows one to specify the name of the files. The base file name is used for all files created for this static analysis, with different file extensions applied. The *.fdb* extension will be added if the user does not type it into the File Name field. The *.fdb* file is the FRANC3D restart file and stores information about the imported FE model, the flaw geometry, and the crack growth data.

Note that the uncracked file name must not be re-used here; the uncracked model is needed for each step of crack growth and should not be overwritten.

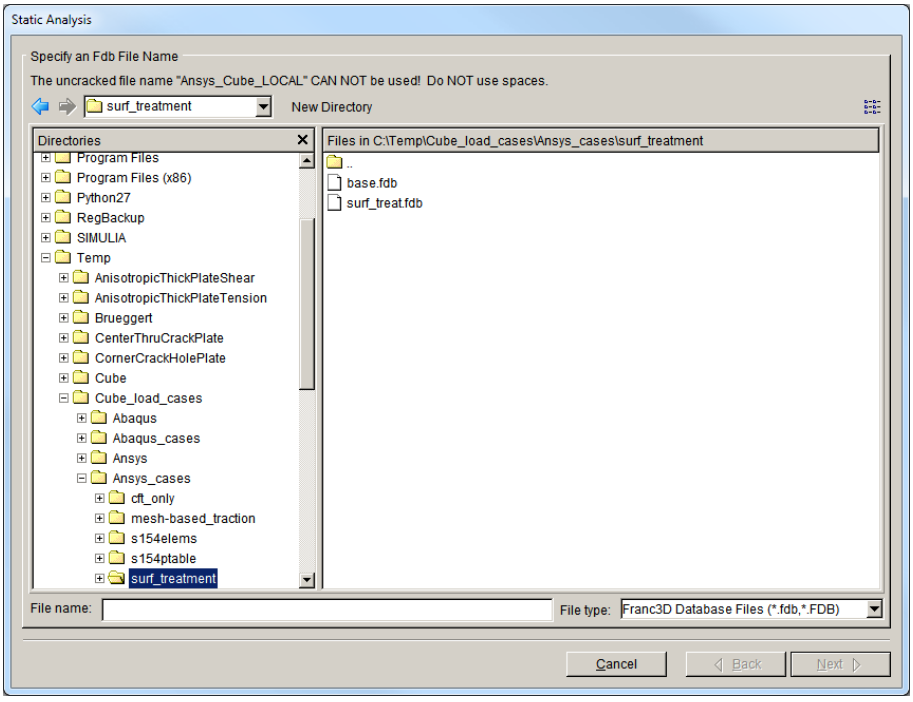


Figure 8.1.1 *.fdb* file name dialog.

8.1.2 ANSYS Options Panel

The dialog in Fig 8.1.2 allows one to set options and analysis parameters for ANSYS.

The top pane contains the ANSYS executable file name and license string. These values should be set in the Preferences (see Section 5.2) but can be modified or set here. A Python script can be executed on the *.cdb* file prior to running ANSYS; the Python executable can be set here also.

The second pane allows one to edit default settings; see Section 8.1.2.1.

The third pane allows the user to select the global model to connect to the local cracked model. If there is no global model, this option should be unchecked. The global model file name should be filled automatically based on the original FE model import.

The fourth pane allows the user to specify whether crack face traction (CFT), crack face contact (CFC), or initial strain conditions are applied. If CFTs are defined (see Section 7.1), the Apply crack face tractions should be checked automatically.

Crack face contact conditions can be applied if the crack is subjected to compressive loads. If the Define crack face contact option is checked, the **Contact** button is activated; see Section 8.1.2.2.

Initial strain can be applied by setting the file names for the node coordinates and the nodal strain, Fig 8.1.3.

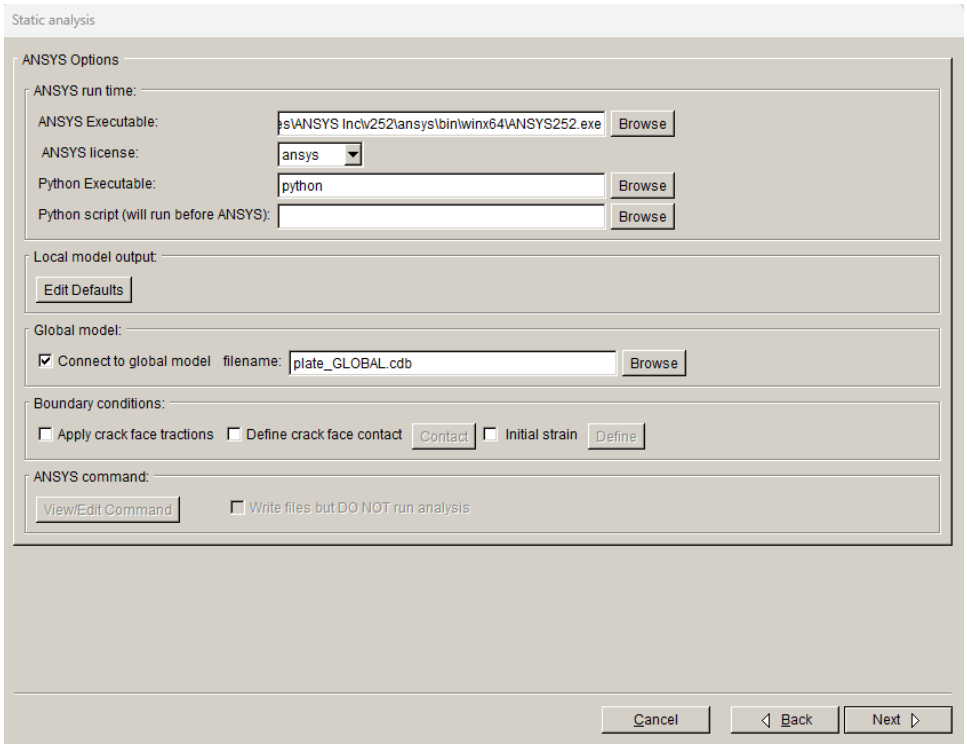


Figure 8.1.2 ANSYS options and analysis parameters.

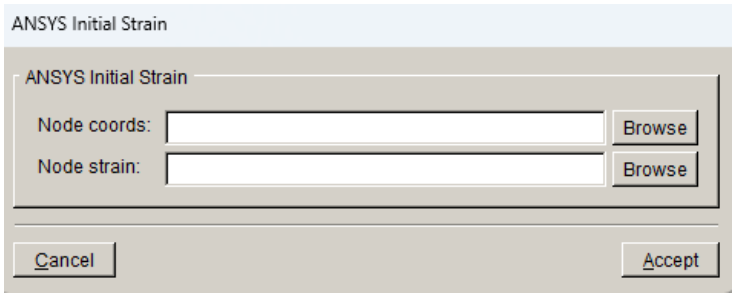


Figure 8.1.3 ANSYS initial strain dialog.

The last pane contains the **View/Edit Command** button, which will be described in Section 8.1.3.4. This option is inactive if the Connect to global model is selected, as it will be active in the subsequent panel.

8.1.2.1 ANSYS Edit Defaults

The ANSYS Local Model Output dialog, Fig 8.1.4, allows the user to adjust some of the default settings.

The Output Results and Extract results options limit the amount of data that is extracted and saved to the *.dtp* file. Results for the crack front mesh template nodes are required for computing SIFs. To display the deformed shape in FRANC3D (see Section 11.1), the results for the local model should be extracted, and this is the default.

Commands to tell ANSYS to compute the contour integral can be included in the *.cdb* file that FRANC3D writes. Either the J-integral or the material-force option can be selected. ANSYS writes this data to the .out file, which can be imported using the FRANC3D Advanced menu.

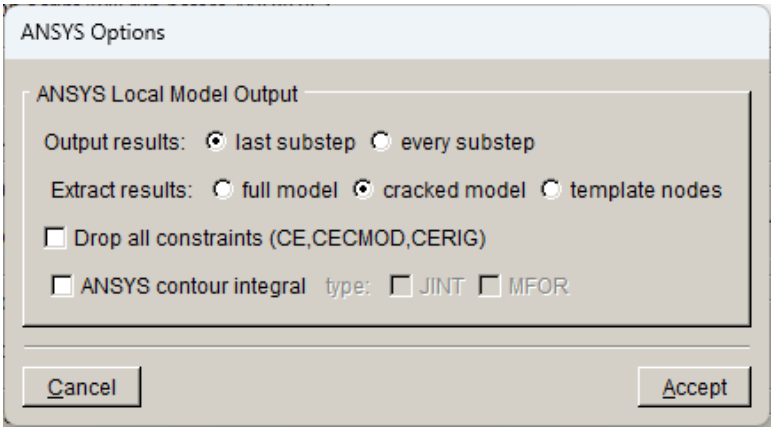


Figure 8.1.4 ANSYS edit defaults dialog.

8.1.2.2 ANSYS Crack Face Contact

The ANSYS Crack Face Contact dialog, Fig 8.1.5, allows the user to set crack face contact options. The material and real constant ids should not conflict with any of the data that exists in the original model. The parameters in the dialog are described in the ANSYS documentation.

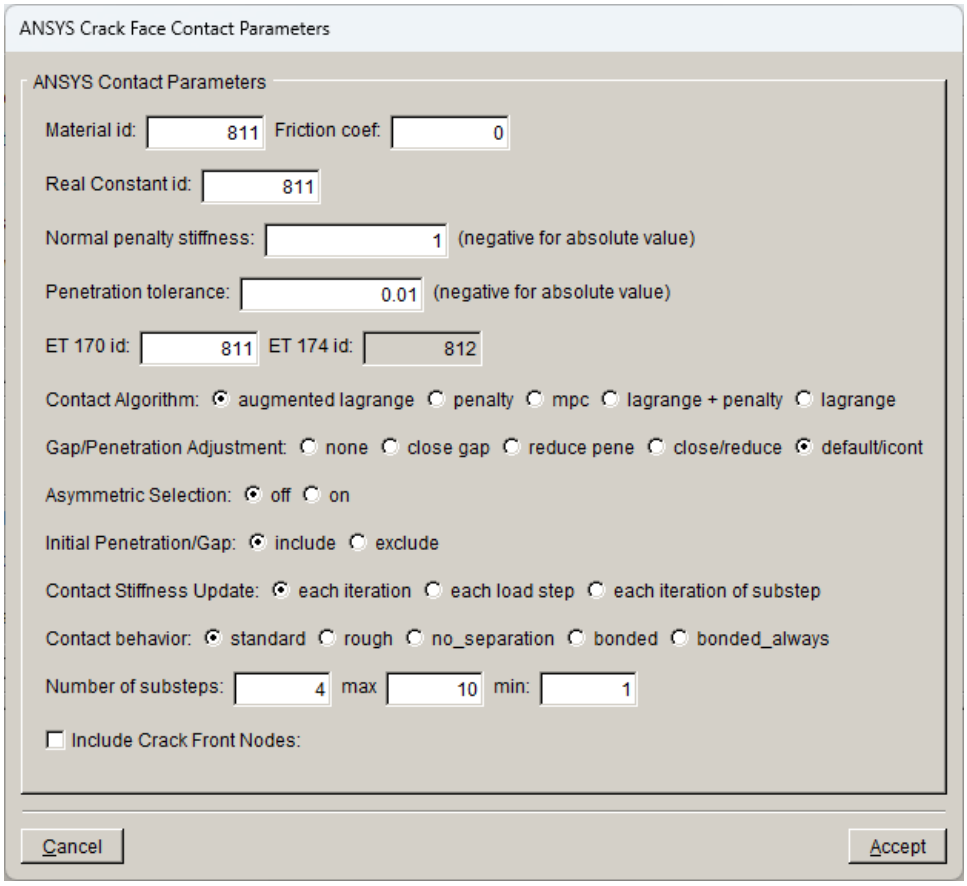


Figure 8.1.5 ANSYS crack face contact dialog.

8.1.3 ANSYS local/global model connection Panel

If the Connect to global model option (see Fig 8.1.2) is selected, the global model file must be set, Fig 8.1.6. The **Browse** button can be used to find the file. The check box and the global model filename should be filled automatically based on the initial FE model import.

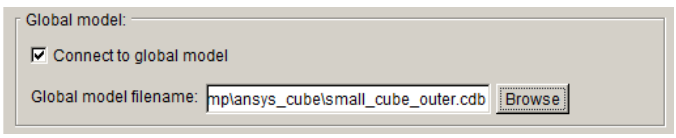


Figure 8.1.6 ANSYS connect to global model box.

If the Connect to global model option is selected, the **Finish** button in Fig 8.1.2, switches to a **Next** button. The next panel, Fig 8.1.7, defines how the local and global models will be combined. Note that the user cannot proceed to the next panel if the Connect to global box is checked and the Global model filename has not been entered.

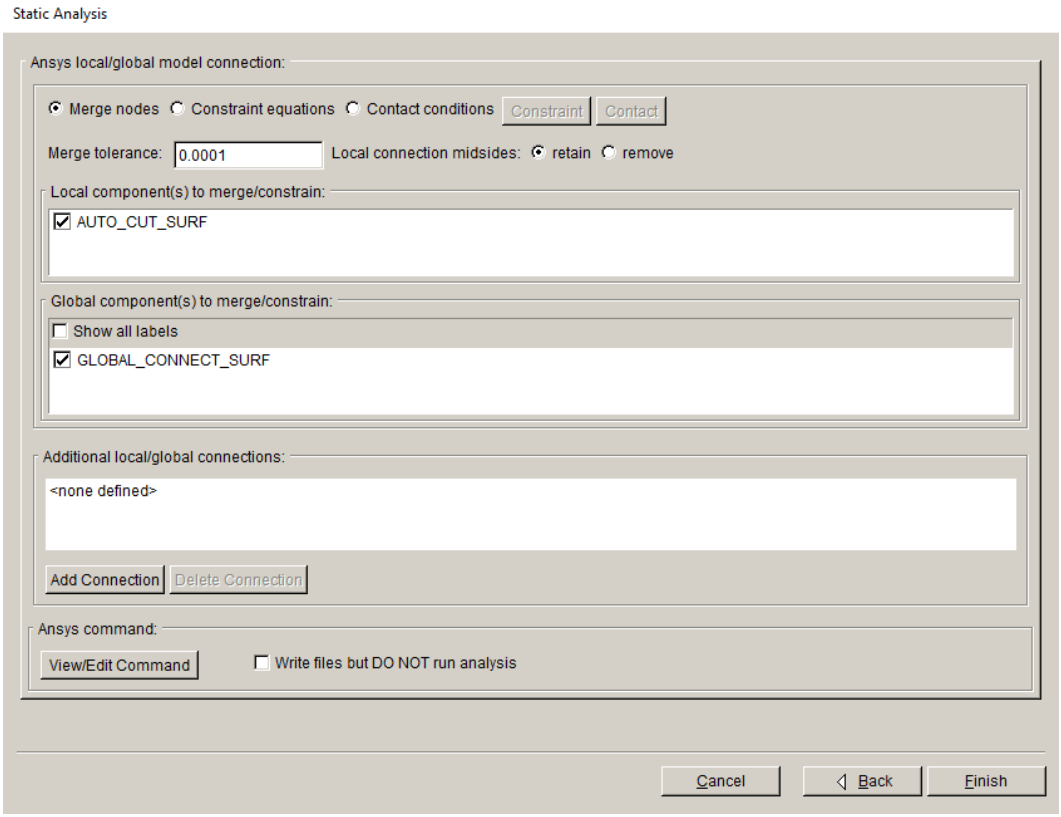


Figure 8.1.7 ANSYS local/global model connection panel.

The simplest method to connect a local crack model to a global model is through node-merging. If the user retained surface mesh facets on the cut-surfaces for the local model, then these surfaces can easily be glued together by merging nodes. Alternatively, constraint equations or contact conditions can be defined between the cut-surfaces of the local and global models. FRANC3D will generate the necessary commands to join the local and global model portions together.

Selecting the Constraint equations option activates the **Constraint** button; see Section 8.1.3.1.

Selecting the Contact conditions option activates the **Contact** button; see Section 8.1.3.2.

Extra local/global contact/constraint can be defined (see lower section of Fig 8.1.7) for surfaces other than the crack faces and the cut-surfaces; see Section 8.1.3.3.

8.1.3.1 ANSYS Constraint

The ANSYS Constraint dialog, Fig 8.1.8, allows the user to set the constraint connection options between the local and global model portions. Typically, all three (displacement) components will be selected, but the user has the option to not select a component.

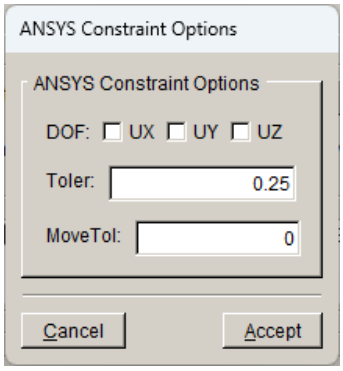


Figure 8.1.8 ANSYS constraint connection dialog.

8.1.3.2 ANSYS Contact

The ANSYS Contact dialog, Fig 8.1.9, allows the user to set contact connection options between the local and global model portions. This dialog is slightly different from the crack face contact dialog; there is no option to include the crack front nodes.

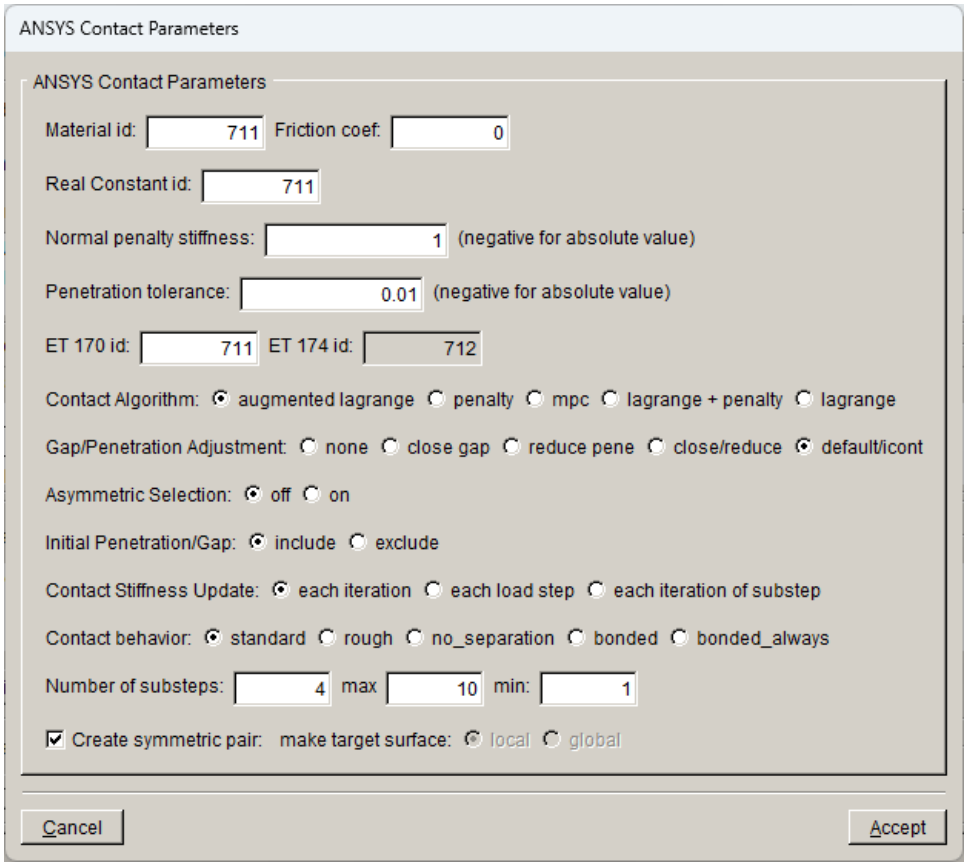


Figure 8.1.9 ANSYS contact connection dialog.

8.1.3.3 ANSYS Extra Connection

Additional local/global connections can be defined (see Fig 8.1.7). Click the **Add Connection** button to display the dialog in Fig 8.1.10. The connection can be made using contact or constraint. The contact and constraint dialogs shown in Sections 8.1.3.1-2 are reused here. The two mate surfaces from the global and local model must be selected by the user.

To delete a connection, highlight the connection name (see Fig 8.1.7) and click the **Delete Connection** button.

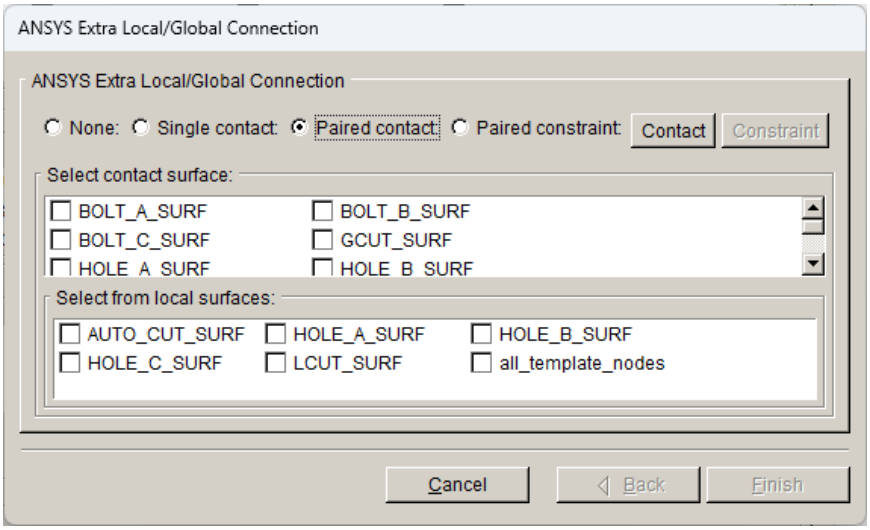


Figure 8.1.10 ANSYS extra connection dialog.

8.1.3.4 ANSYS Command Line Panel

If the user selects the **View/Edit Command** button in Fig 8.1.7, the ANSYS command is displayed, Fig 8.1.11. The command line can be edited if desired. Typically, users will set the ANSYS executable and parameters in the Preferences rather than editing this command. Editing this command line is not "permanent", but the edits should be carried forward for subsequent static analyses of the current model.

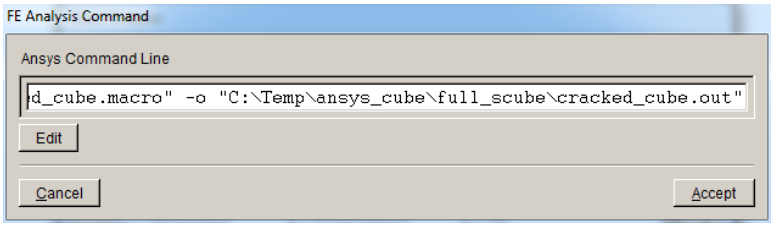


Figure 8.1.11 ANSYS command line panel.

8.1.3.5 ANSYS Write Files but DO NOT run analysis

In some cases, the user might want to write the analysis files from FRANC3D without running ANSYS. The Write files… option (see Fig 8.1.7) will cause FRANC3D to write all the analysis files.

A *.txt* file is written along with the *.cdb* and *.fdb* files; the ANSYS command is written to this *.txt* file for reference. Note that if one sends this file to a different computer for analysis, the *.dtp* file that is extracted must be retrieved so that FRANC3D can compute SIFs (see Section 4.7).

8.1.4 ABAQUS Options Panel

If ABAQUS is the solver, the user can set options and analysis parameters for ABAQUS, Fig 8.1.12. The first pane contains the ABAQUS executable file name. This should be set in the Preferences (see Section 5.2) but can be modified or set here. A Python script can be executed on the *.inp* file prior to running ABAQUS. The Python executable can be set here also.

The second pane allows one to edit default settings; see Section 8.1.4.1.

The third pane allows the user to select the global model to connect to the local cracked model. If there is no global model, this option should be unchecked. The global model file name should be filled automatically based on the original FE model import.

The fourth section allows the user to specify whether CFT and CFC are applied. If CFTs are defined (see Section 7.1), the Apply crack face tractions should be checked automatically.

Crack face contact conditions can be applied also if the crack is subjected to compressive loads. If the Define crack face contact option is checked, the **Contact** button is activated; see Section 8.1.4.2.

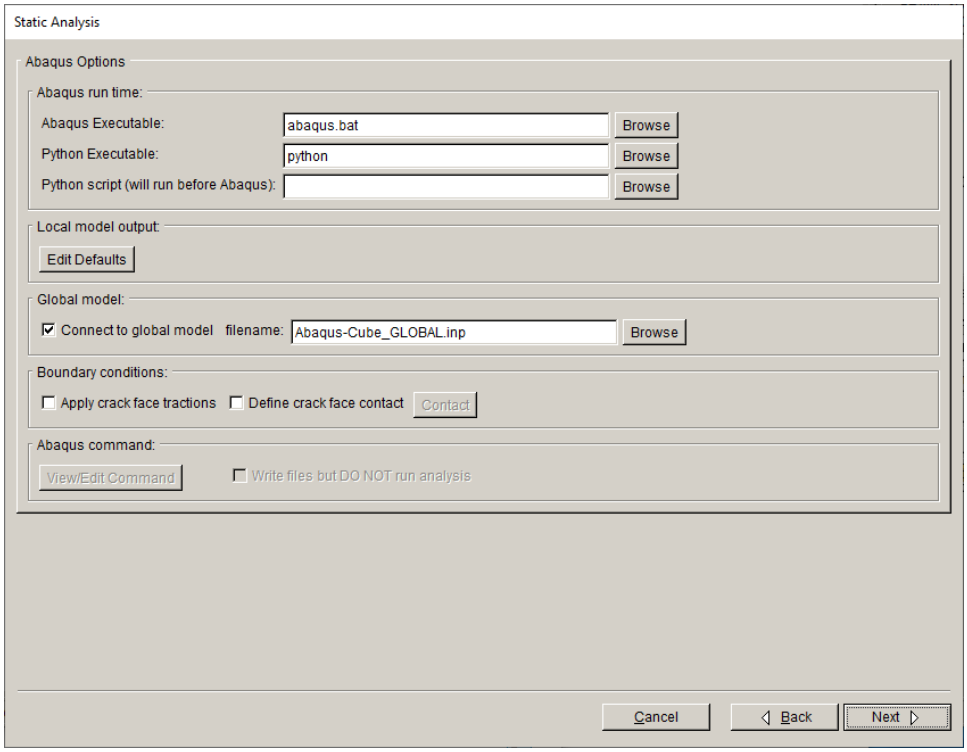


Figure 8.1.12 ABAQUS options panel.

The final pane contains the button **View/Edit Command**, which will be described in Section 8.1.5.4. This option is inactive if the Connect to global model is selected, as it will be active in the subsequent panel.

8.1.4.1 ABAQUS Edit Defaults

The ABAQUS Local Model Output dialog, Fig 8.1.13, allows the user to adjust some of the default settings.

The results for the last frame of each load step are extracted by default, but the user can choose to extract results for every frame (substep). The Extract results option limits the amount of data that is extracted and saved to the *.dtp* file. Results for the crack front mesh template nodes are required for computing SIFs. To display the deformed shape in FRANC3D (see Section 11.1), the results for the local model should be extracted, and this is the default.

Commands to compute contour (J, C, K or T) integrals can be included in the ABAQUS file. Note that ABAQUS uses collapsed bricks rather than singular wedge elements for the contour integral. The contour integral results are saved in the ABAQUS *.dat* file. FRANC3D can import the contour integral results from this file; see the Advanced menu.

FRANC3D is capable of computing J-integral for elastoplastic ABAQUS models.

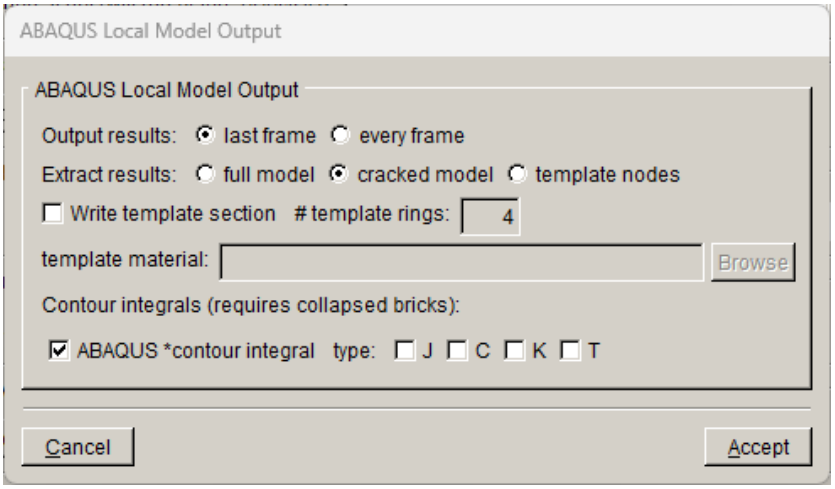


Figure 8.1.13 ABAQUS edit defaults dialog.

8.1.4.2 ABAQUS Crack Face Contact

The ABAQUS Crack Face Contact dialog, Fig 8.1.14, allows the user to set crack face contact options. A new Surface interaction name must be defined, and it should not conflict with any of the data that exists in the original model; the **Accept** button will be activated once a surface interaction name is added. Parameters are described in the ABAQUS documentation.

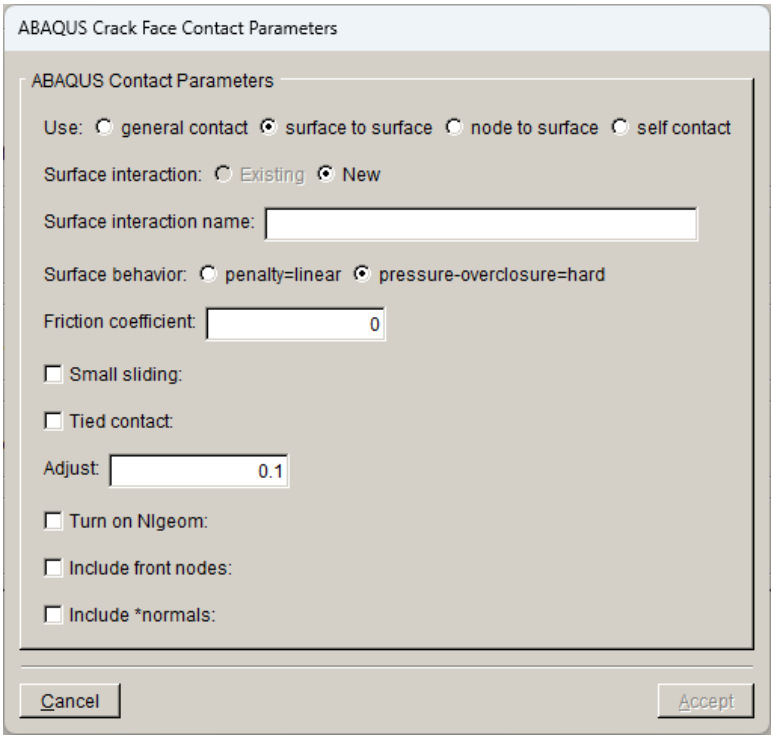


Figure 8.1.14 ABAQUS crack face contact dialog.

8.1.5 ABAQUS local/global model connection Panel

If the Connect to global model option (see Fig 8.1.12) is selected, the global model file must be set, Fig 8.1.15. The **Browse** button can be used to find the file. The check box and the global model filename should be filled automatically based on the initial FE model import.

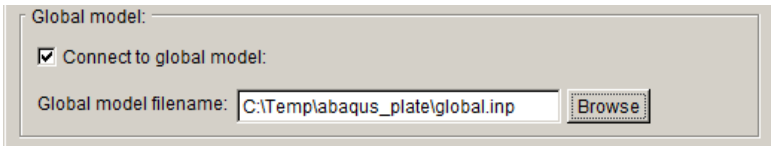


Figure 8.1.15 ABAQUS connect to global model box.

If the Connect to global model option is selected, the **Finish** button in Fig 8.1.12, switches to a **Next** button. The next panel, Fig 8.1.16, defines how the local and global models will be combined. Note that the user cannot proceed to the next panel if the Connect to global box is checked and the Global model filename has not been entered.

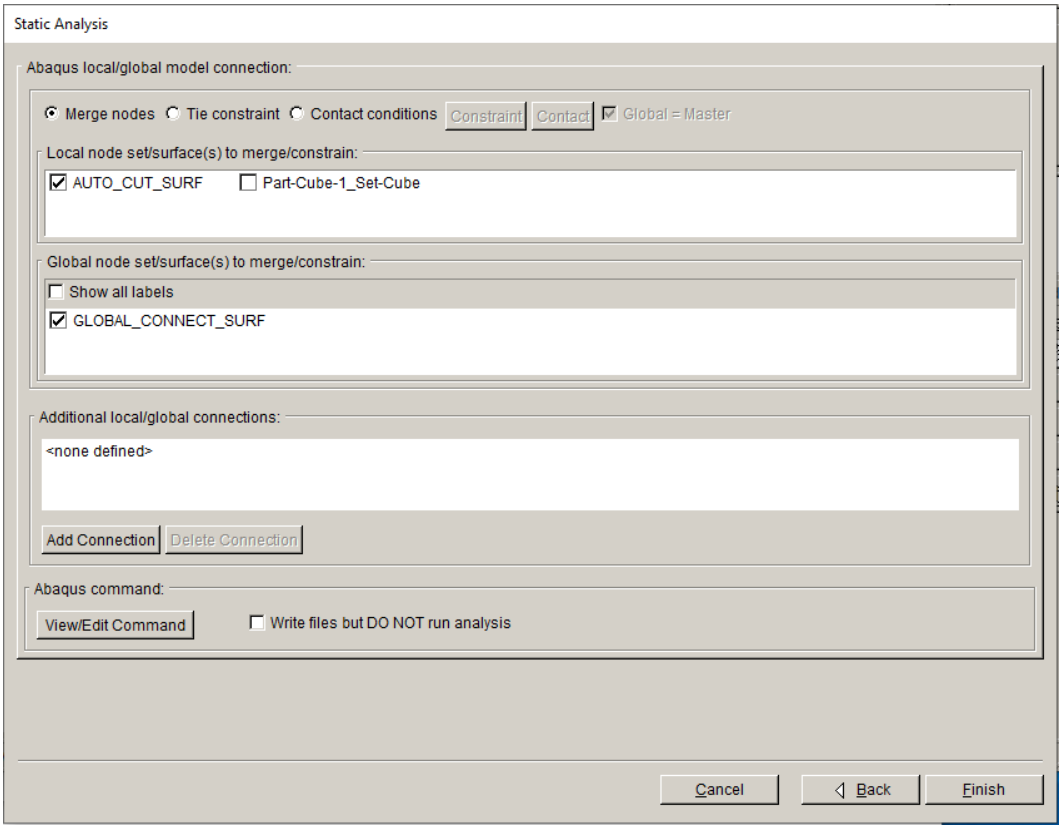


Figure 8.1.16 ABAQUS local/global model connection panel.

The simplest method to connect a local crack model to a global model is through node-merging. If the user retained surface mesh facets on the cut-surfaces for the local model, then these surfaces can easily be glued together by merging nodes. Alternatively, constraint equations or contact conditions can be defined between the cut-surfaces of the local and global models.

Selecting the Constraint equations option activates the **Constraint** button; see Section 8.1.5.1.

Selecting the Contact conditions option activates the **Contact** button; see Section 8.1.5.2.

An extra check box allows the user to switch the master and slave pairing; the default is to use the Global surface as the master.

Extra local/global contact/constraint can be defined (see lower section of Fig 8.1.16) for surfaces other than the crack faces and the cut-surfaces; see Section 8.1.5.3.

8.1.5.1 ABAQUS Constraint

The ABAQUS Constraint dialog, Fig 8.1.17, allows the user to set constraint connection options for the local and global model portions.

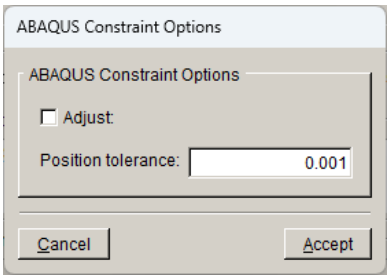


Figure 8.1.17 ABAQUS constraint connection dialog.

8.1.5.2 ABAQUS Contact

The ABAQUS Contact dialog, Fig 8.1.18, allows the user to set contact connection options for the local and global model portions. This dialog is mostly the same as that for CFC, but in this case the user is allowed to select from existing Surface interaction names.

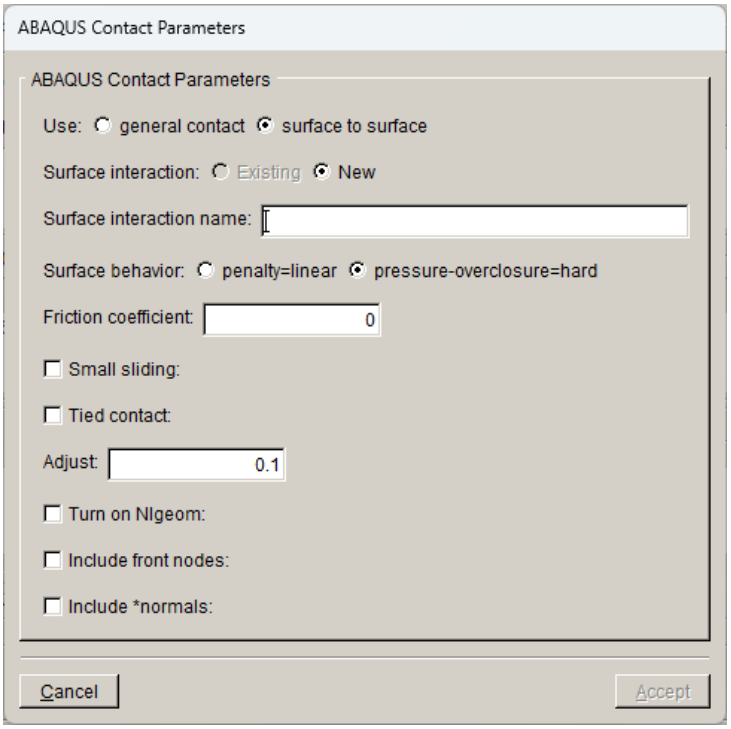


Figure 8.1.18 ABAQUS contact connection dialog.

8.1.5.3 ABAQUS Extra Connection

Additional local/global connections can be defined (see Fig 8.1.16). Click the **Add Connection** button to display the dialog in Fig 8.1.19. The connection can be made using contact or

constraint. The contact and constraint dialogs shown in Sections 8.1.5.1-2 are reused here. The two mate surfaces from the global and local model must be selected by the user.

To delete an extra connection, highlight the connection name and click the **Delete Connection** button (see Fig 8.1.16).

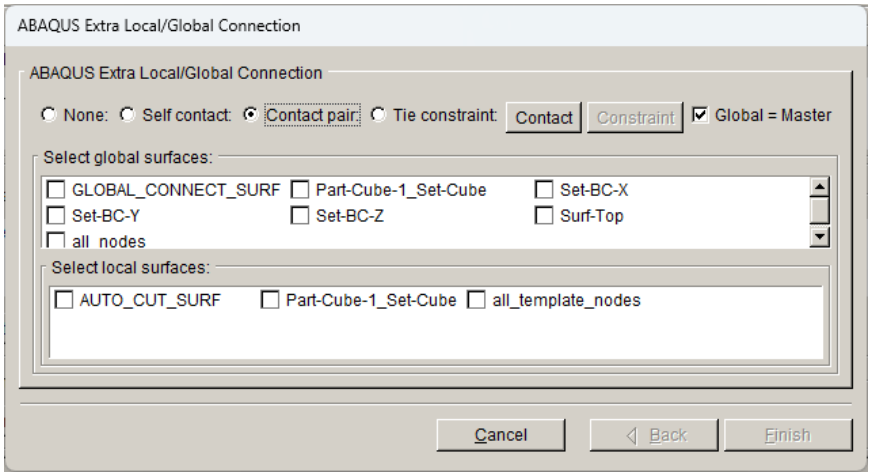


Figure 8.1.19 ABAQUS extra connection dialog.

8.1.5.4 ABAQUS Command Line Panel

If the user selects the **View/Edit Command** button in Fig 8.1.16, the ABAQUS command is displayed, Fig 8.1.20. The command line can be edited if desired. Typically, users will set the ABAQUS executable and parameters in the Preferences rather than editing this command. Editing this command line is not "permanent", but the edits should be carried forward for subsequent static analyses of the current model.

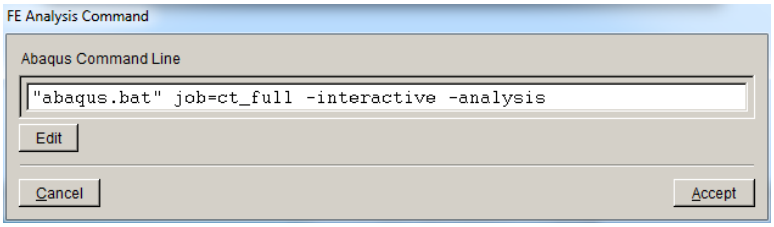


Figure 8.1.20 ABAQUS command line panel.

8.1.5.5 ABAQUS Write Files but DO NOT run analysis

In some cases, the user might want to write the analysis files from FRANC3D without running ABAQUS. The Write files… option (see Fig 8.1.16) will cause FRANC3D to write all the analysis files.

A *.txt* file is written along with the *.inp* and *.fdb* files; the ABAQUS command is written to this *.txt* file for reference. Note that if one sends this file to a different computer for analysis, the *.dtp* file that is extracted must be retrieved so that FRANC3D can compute SIFs (see Section 4.7). The *.dtp* file is extracted from the ABAQUS *.odb* file by running the command:

```
abaqus python writeDtpFile.py
```

The *writeDtpFile.py* is written by FRANC3D along with the other restart/analysis files.

8.1.6 NASTRAN Options Panel

If NASTRAN is the solver, the user can set options and analysis parameters for NASTRAN, Fig 8.1.21. The first pane contains the NASTRAN executable file name. This should be set in the Preferences (see Section 5.2) but can be modified or set here. A Python script can be executed on the *.bdf* file prior to running NASTRAN. The Python executable can be set here also.

The second pane allows one to edit default settings; see Section 8.1.6.1.

The third pane allows the user to select the global model to connect to the local cracked model. If there is no global model, this option should be unchecked. The global model file name should be filled automatically based on the original FE model import.

The fourth section allows the user to specify whether CFT and CFC are applied. If CFTs are defined (see Section 7.1), the Apply crack face tractions should be checked automatically.

Crack face contact conditions can be applied also if the crack is subjected to compressive loads. If the Define crack face contact option is checked, the **Contact** button is activated; see Section 8.1.6.2.

The final pane contains the button **View/Edit Command**, which will be described in Section 8.1.7.3. This option is inactive if the Connect to global model is selected, as it will be active in the subsequent panel.

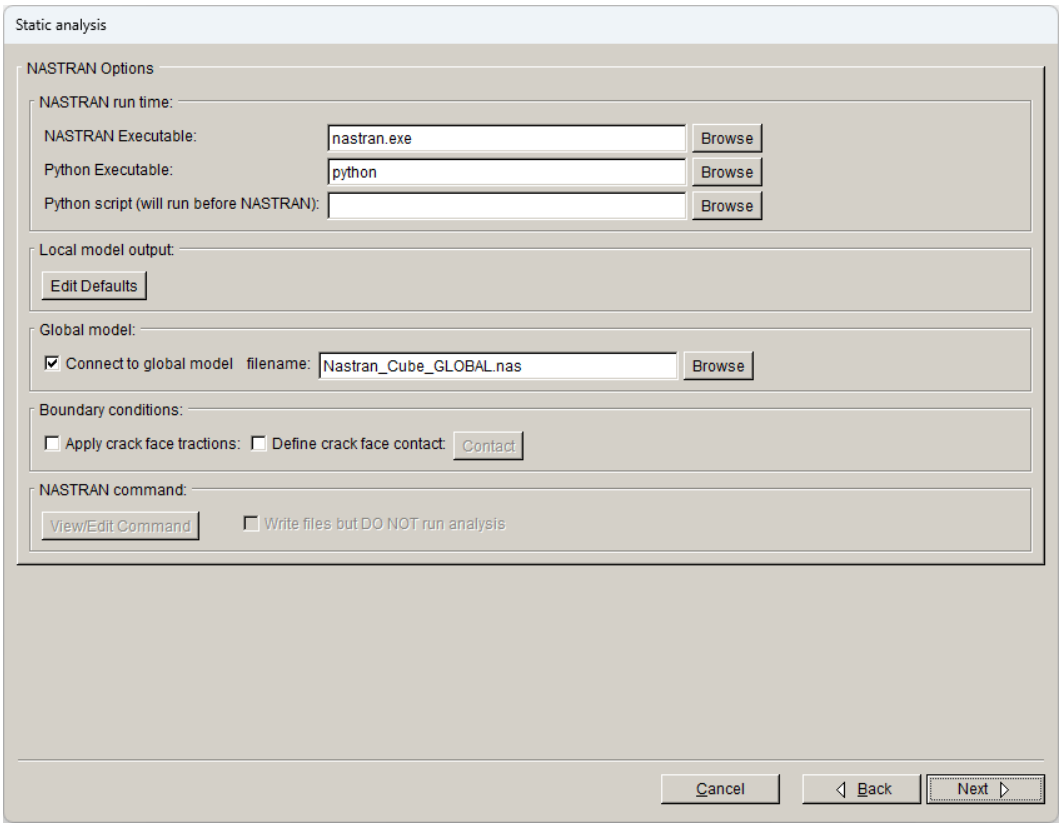


Figure 8.1.21 NASTRAN options panel.

8.1.6.1 NASTRAN Edit Defaults

The NASTRAN Local Model Output dialog, Fig 8.1.22, allows the user to adjust some of the default settings.

Crack front elements typically will be singular quarter-point wedge elements, but these can be converted to regular wedge elements with mid-side nodes. Collapsed brick and blunted brick elements can also be used. NX NASTRAN might generate an error when using quarter point elements, so mid-side nodes might work in that case; there might be a loss of accuracy in the computed SIFs.

Commands to compute contour (J) integrals can be included in the NASTRAN file. Collapsed brick elements are required for this.

NASTRAN checks the element shape by default, but this can be turned off using the GeomCheck option.

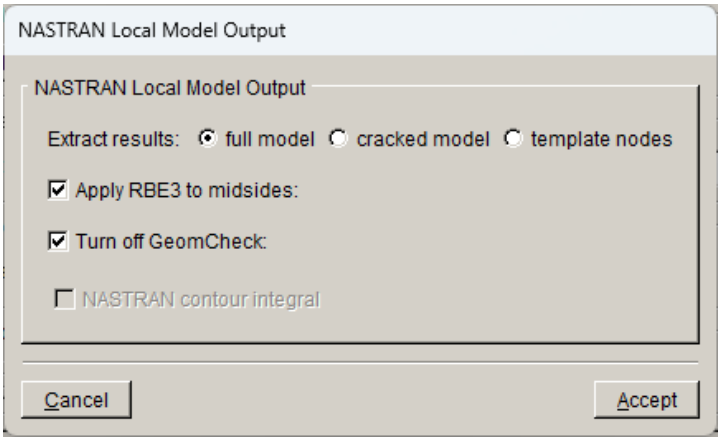


Figure 8.1.22 NASTRAN edit defaults dialog.

8.1.6.2 NASTRAN Crack Face Contact

The NASTRAN Crack Face Contact dialog, Fig 8.1.23, allows the user to set crack face contact options. The parameters in the dialog are described in the NASTRAN documentation and will be different depending on the flavor of NASTRAN that is used. Note that NASTRAN does not output contact pressure to the *.pch* file so one must compute SIFs using displacement correlation if CFC is included.

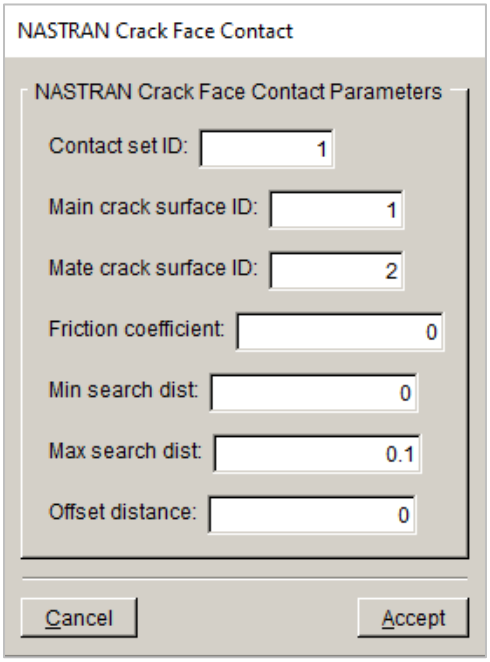


Figure 8.1.23 NASTRAN contact dialog.

8.1.7 NASTRAN local/global model connection Panel

If the Connect to global model option (see Fig 8.1.21) is selected, the global model file must be set, Fig 8.1.24. The **Browse** button can be used to find the file. Note that the check box and the global model filename are filled automatically, based on the initial FE model import.

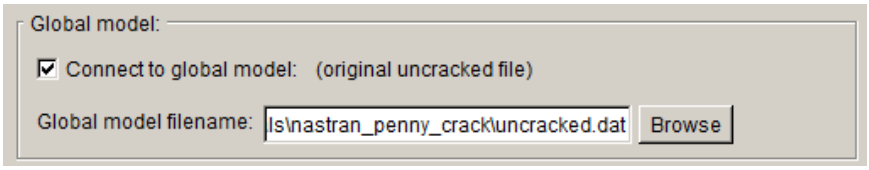


Figure 8.1.24 NASTRAN connect to global model box.

If the Connect to global model option is selected, the **Finish** button in Fig 8.1.21, switches to a **Next** button. The next panel, Fig 8.1.25, defines how the local and global models will be combined. Note that the user cannot proceed to the next panel if the Connect to global box is checked and the Global model filename has not been entered.

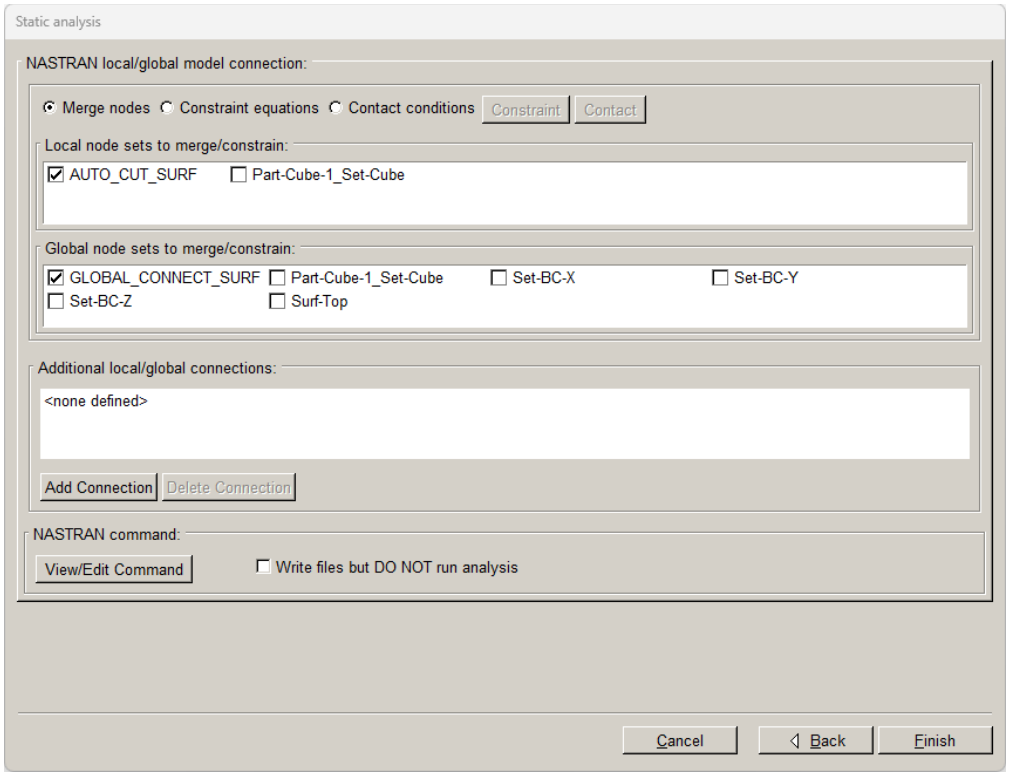


Figure 8.1.25 NASTRAN local/global model connection panel.

The simplest method to connect a local crack model to a global model is through node-merging. If the user retained surface mesh facets on the cut-surfaces for the local model, then these surfaces can easily be glued together by merging nodes. Alternatively, glued contact can be defined between the cut-surfaces of the local and global models by selecting the Constraint surfaces option.

Selecting the Constraint equations option activates the **Constraint** button; see Section 8.1.7.1.

Selecting the Contact conditions option activates the **Contact** button; see Section 8.1.7.2.

Extra local/global contact/constraint can be defined (see lower section of Fig 8.1.25) for surfaces other than the crack faces and the cut-surfaces; see Section 8.1.7.3.

8.1.7.1 NASTRAN Constraint

The NASTRAN Constraint dialog, Fig 8.1.26, allows the user to set constraint connection options for the local and global model portions.

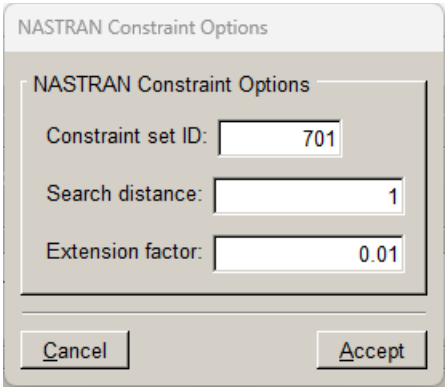


Figure 8.1.26 NASTRAN constraint connection dialog.

8.1.7.2 NASTRAN Contact

The NASTRAN Contact dialog, Fig 8.1.27, allows the user to set contact connection options for the local and global model portions.

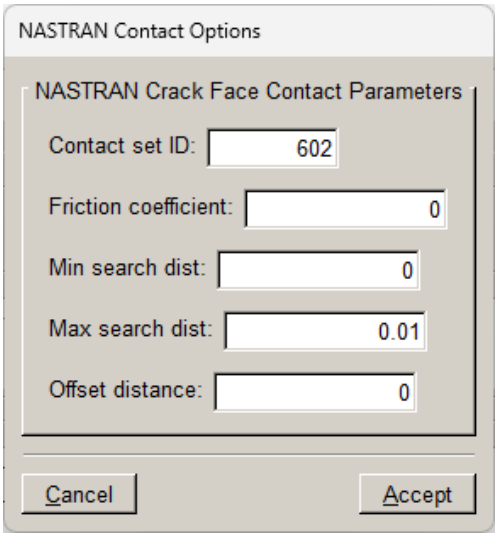


Figure 8.1.27 NASTRAN contact connection dialog.

8.1.7.3 NASTRAN Extra Connection

Additional local/global connections can be defined (see Fig 8.1.25). Click the **Add Connection** button to display the dialog in Fig 8.1.28. The connection can be made using contact or constraint. The contact and constraint dialogs shown in Sections 8.1.7.1-2 are reused here.. Two mate surfaces from the global and local model must be selected by the user.

To delete an extra connection, highlight the connection name and click the **Delete Connection** button.

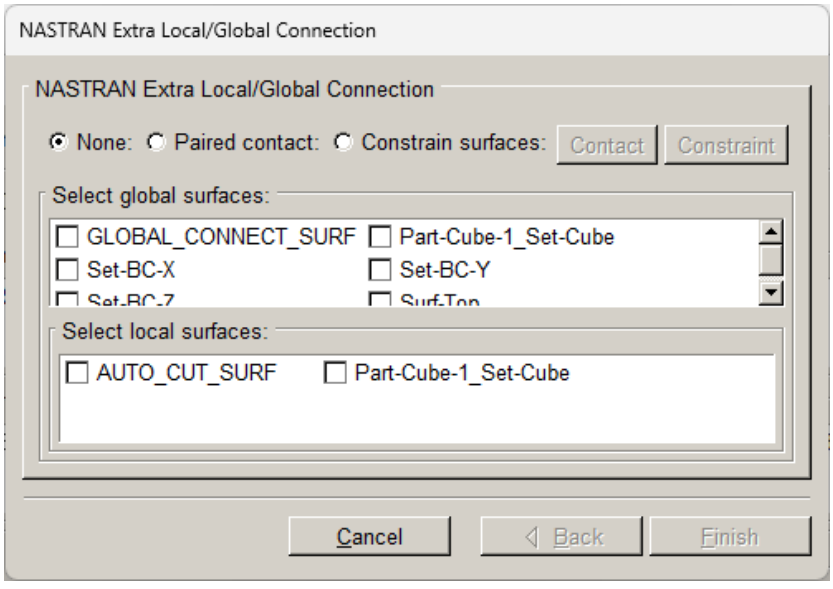


Figure 8.1.28 NASTRAN extra connection dialog.

8.1.7.4 NASTRAN Command Line Panel

If the user selects the **View/Edit Command** button in Fig 8.1.25, the NASTRAN command is displayed, Fig 8.1.29. The command line can be edited if desired. Typically, users will set the NASTRAN executable and parameters in the Preferences rather than editing this command. Editing this command line is not "permanent", but the edits should be carried forward for subsequent static analyses of the current model.

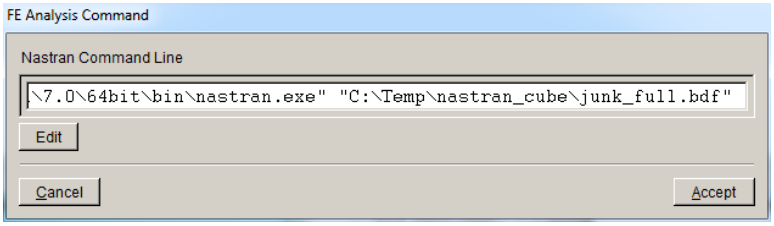


Figure 8.1.29 NASTRAN command line panel.

8.1.7.5 NASTRAN Write Files but DO NOT run analysis

In some cases, the user might want to write the analysis files from FRANC3D without running NASTRAN. The Write files… option (see Fig 8.1.25) will cause FRANC3D to write all the analysis files.

A *.txt* file is written along with the *.bdf* (or *.nas*) and *.fdb* files; the NASTRAN command is written to this *.txt* file for reference. Note that if one sends this file to a different computer for analysis, the *.pch* file that is written by NASTRAN must be retrieved so that FRANC3D can compute SIFs (see Section 4.7).

8.2 Crack Growth Analysis

This wizard allows one to set up and run a series of crack growth analyses. The first set of wizard panels are the same as for the **Grow Crack** wizard described in Section 6.5. The SIF computation method along with the crack growth extension and kink angle models must be defined.

The analysis code panels are the same as those described in Section 8.1.

The wizard panels that are displayed after defining the crack growth model, but before the analysis code panels, are described next.

8.2.1 Crack Front Smoothing Panel

The crack front smoothing panel, Fig 8.2.1, provides crack front fitting and template options. The options here are the same as the front fitting options described in Section 6.5.5, although they are arranged differently. One might do one step of crack growth as described in Section 6.5 before attempting automatic growth to determine the appropriate (best) fitting and extension for the model. The fitting options might need to change as the crack grows.

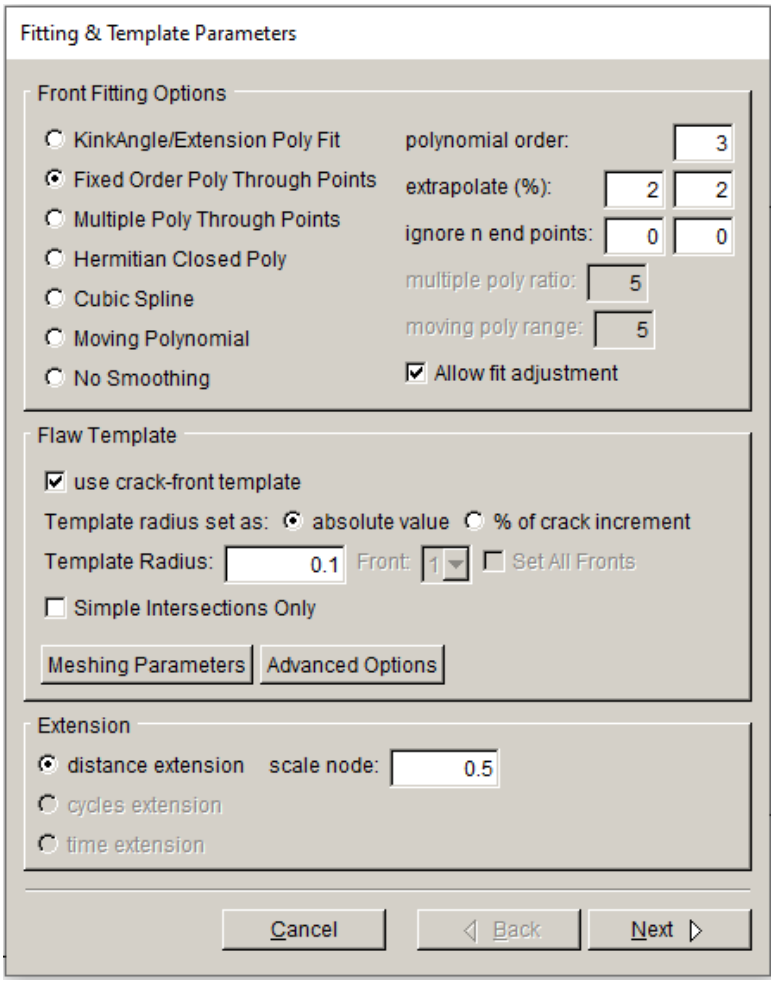


Figure 8.2.1 Crack growth analysis wizard panel for fitting and template parameters.

8.2.2 Crack Growth Increments Panel

The next panel, Fig 8.2.2, allows the user to specify the number of crack growth steps and the median crack growth increment per step. The dialog that is displayed will depend on whether you chose to do quasi-static (left side of Fig 8.2.2) or subcritical (right side of Fig 8.2.2) growth.

There are three choices for entering the extension per step: a constant value for all steps, a linear variation, and user-defined values. The user-defined values can be read from a *.txt* file; the data consists of extension length (or number of cycles) for each step.

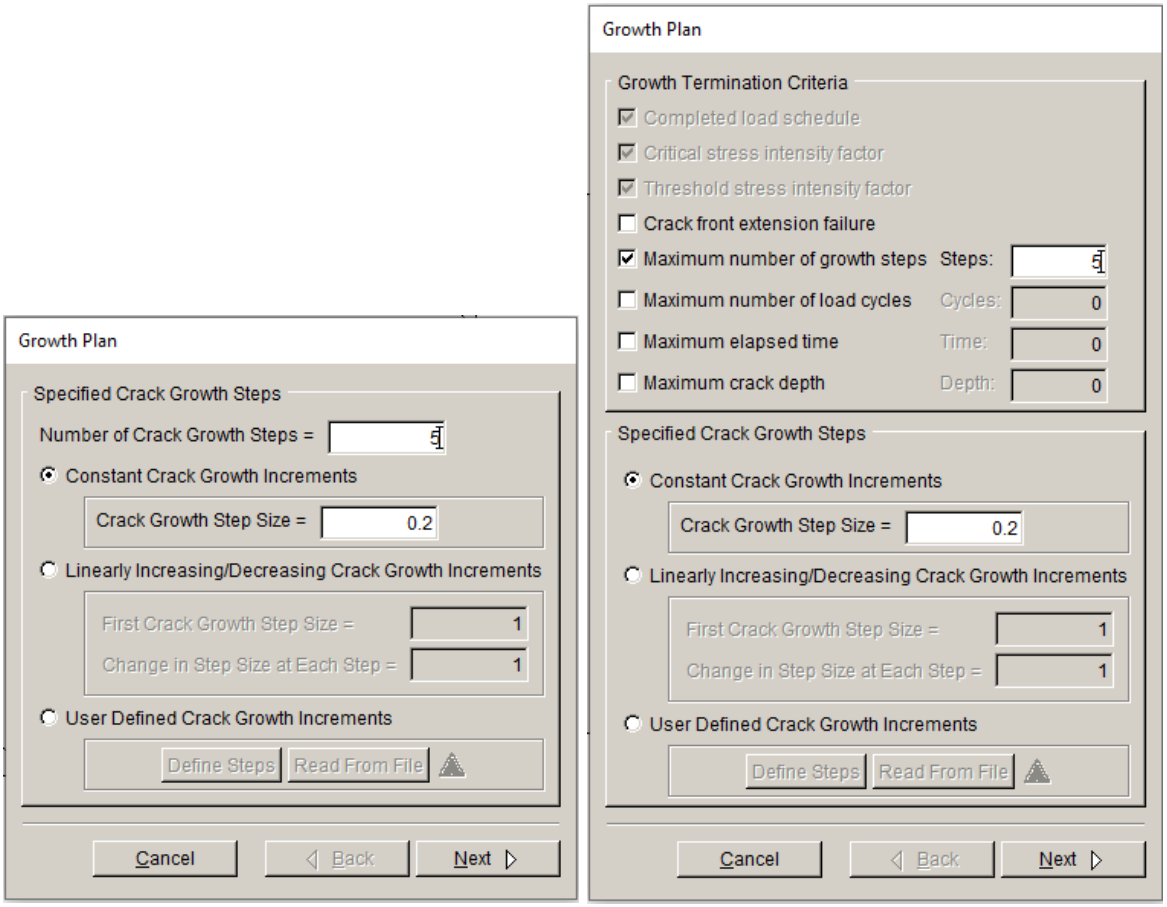


Figure 8.2.2 Crack growth analysis wizard panel for median crack growth increments.

The subcritical growth dialog includes various Growth Termination Criteria. Some of the criteria are turned on by default and cannot be turned off. Other criteria can be turned on as desired. Note that maximum crack depth might be difficult to determine.

8.2.3 Analysis Code Panels

The next panel, Fig 8.2.3, allows one to specify the base file name. All crack growth steps will use this as the base name, and a crack step number will be appended. The current crack growth step number can be set also; this number is used to create the file name (e.g., base_STEP_002). The analysis code is based on the original FE model, and the analysis options are described in Section 8.1.

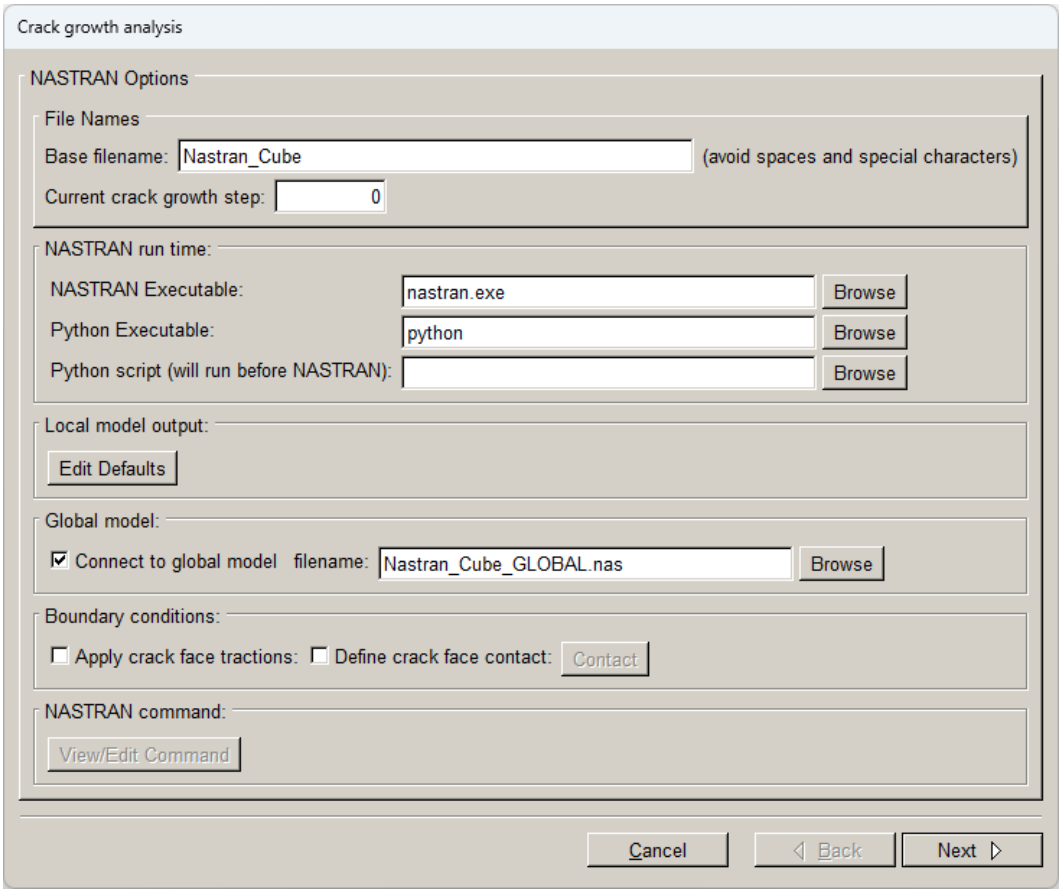


Figure 8.2.3 Crack growth analysis wizard panel for analysis code and base file name.

9. Fatigue Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Fatigue** menu option are described in this section.

9.1 Fatigue Cycle Computation

Earlier versions of FRANC3D, relied on a “K vs a” type of SIF history to compute fatigue cycles. A path through the crack fronts is defined, Fig 9.1.1, and the SIFs are computed at each point where the path crosses a crack front; this gives a SIF versus path length plot, Fig 9.1.2. There are drawbacks to this approach: 1) a single path can be difficult to define for many cracks, and 2) different paths tend to give different numbers of cycles.

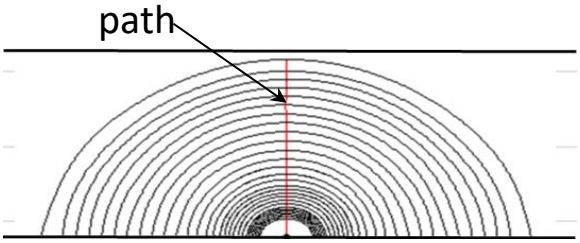


Figure 9.1.1 A path through a series of crack fronts.

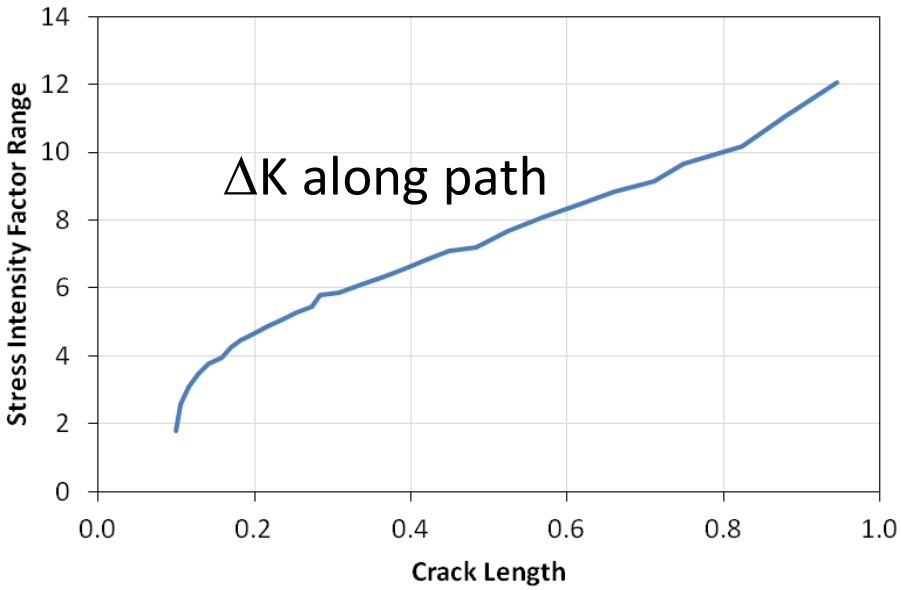


Figure 9.1.2 SIF (range) versus path length.

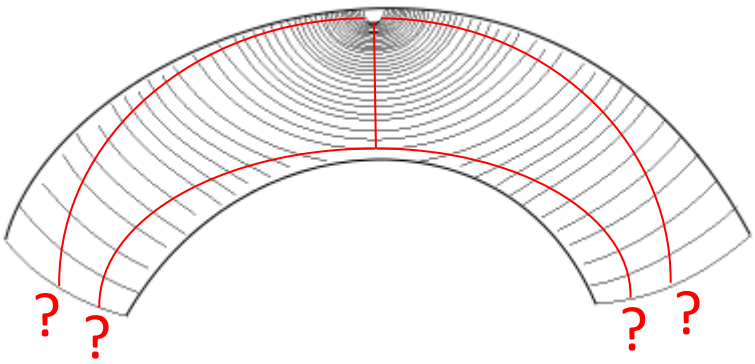


Figure 9.1.3 Ambiguous path selection.

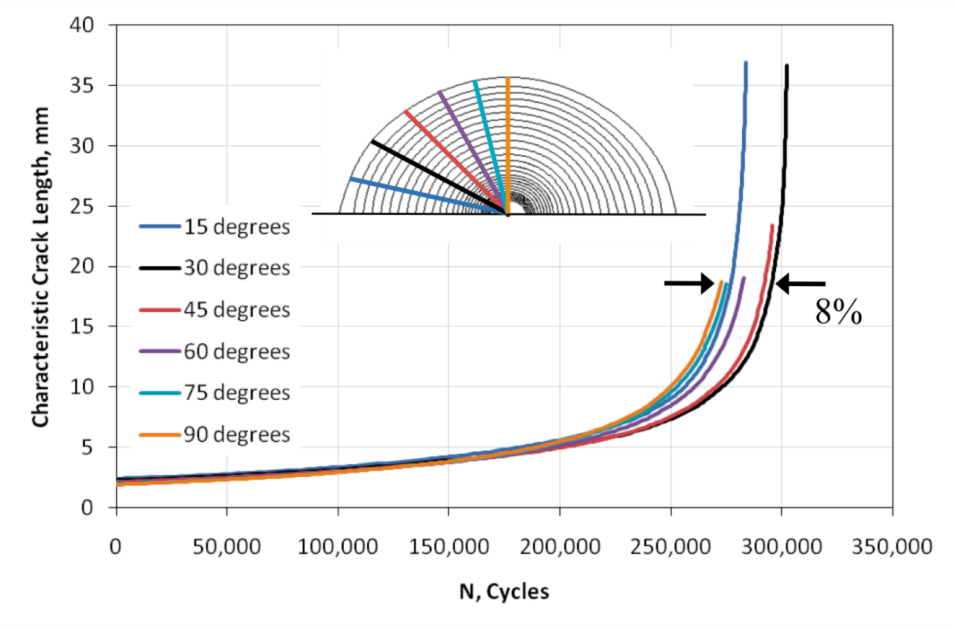


Figure 9.1.4 Different paths give different numbers of cycles.

Cycle counting uses a "multiple variable degree of freedom" approach. For each node *j* on crack front *i*, Fig 9.1.5:

a) project from node *j*, perpendicular to the crack-front *i*, to find the intersection with the next crack front *i*+1,
b) interpolate (if needed) to find ΔK at the intersection point with crack-front *i*+1
c) assume a linear variation in the ΔK's going from step *i* to step *i*+1 and integrate to find the cycles ($N_{i,j}$)
d) average the computed cycles for all crack front nodes to obtain one value for cycles required to grow from crack front *i* to crack front *i*+1.

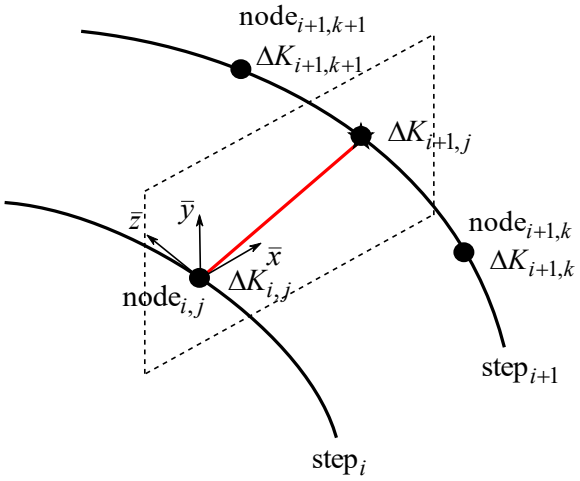


Figure 9.1.5 Cycles computed for each node along the crack front.

This process is repeated for each step of crack growth, Fig 9.1.6, which produces cycles versus crack-growth-step, Fig 9.1.7.

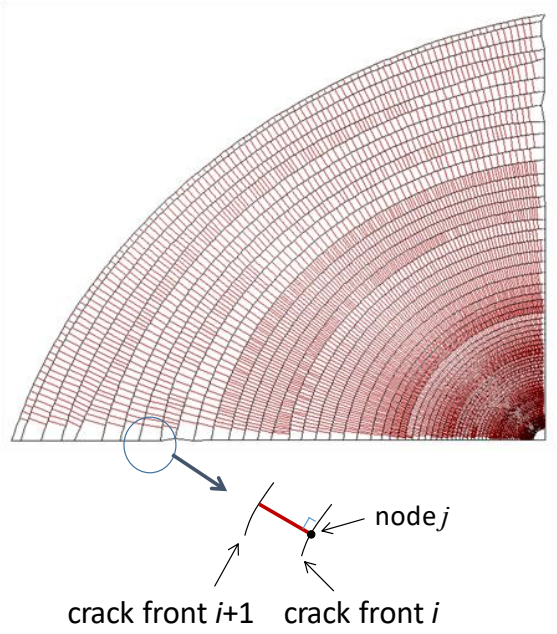


Figure 9.1.6 Cycles computed for each node along each crack front.

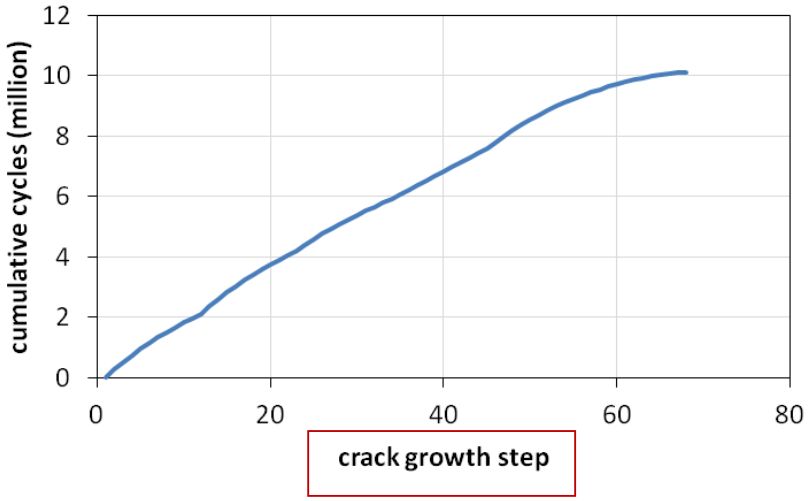


Figure 9.1.7 Cycles versus crack step.

There are several benefits of this approach. The most significant is that all paths give the same number of cycles. Paths along the model surface, where SIFs are least accurate, can be plotted and the cycles will be consistent with a path through the mid-points of the fronts, Fig 9.1.8.

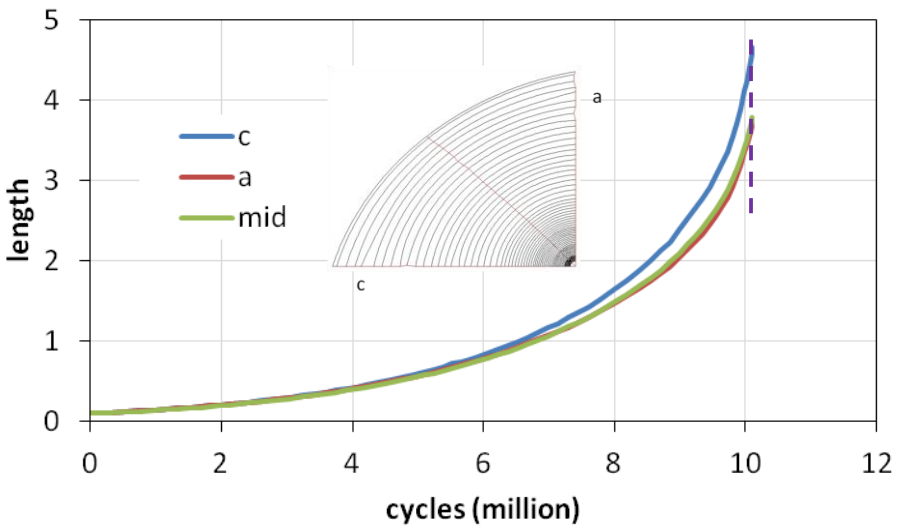


Figure 9.1.8 Different paths give the same number of cycles.

For cracks such as that shown in Figs 9.1.3 and 9.1.9, path selection might be ambiguous, but all paths will give a consistent number of cycles, assuming the paths go through the same number of crack steps.

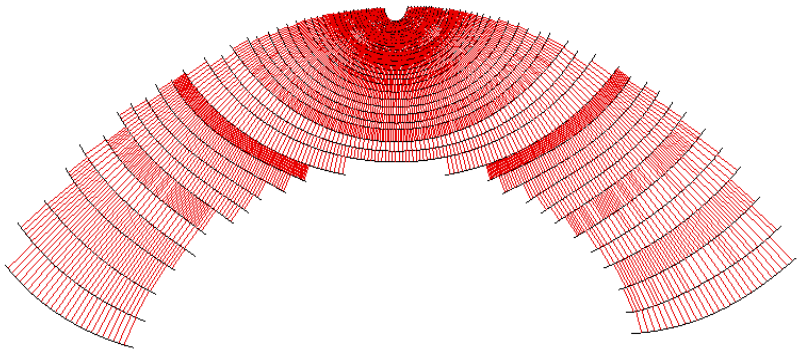


Figure 9.1.9 Different paths give the same number of cycles.

9.2 Fatigue Life Predictions

The Fatigue Life wizard, Fig 9.2.1, allows the user to compute fatigue cycles. The dialog shows the cracked model on the left. The right side will display plots of cycles, SIFs, and other data. The buttons and options in the middle pane are described here.

Note that the **Set Parameters** button leads to the crack growth model selection described in Section 6.5.1. The crack growth model that was used to grow the crack can be re-used here, or one can choose a different model to compute cycles.

If the user grew the crack with a quasi-static growth model, the right-side pane in Fig 9.2.1 will indicate that no lifing parameters have been defined. If crack growth was done using a subcritical growth model, the parameters are defined, and the right-side pane will show a plot of cycles versus crack step.

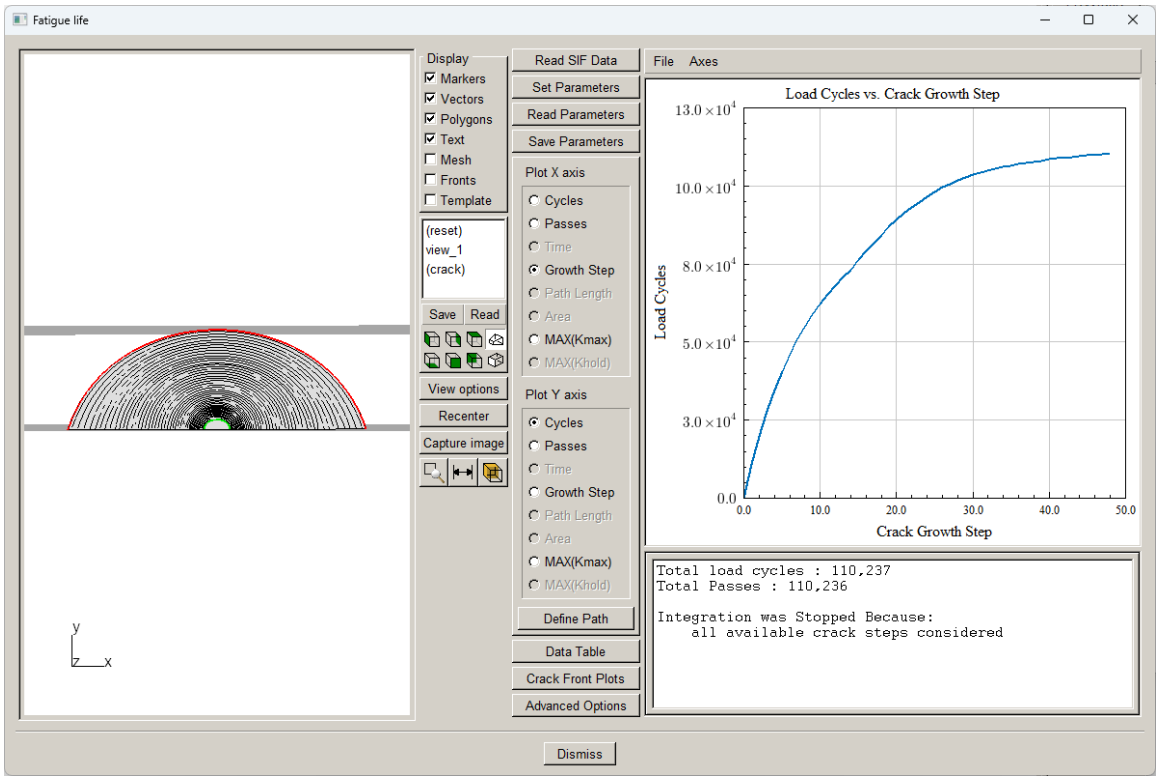


Figure 9.2.1 Fatigue life dialog.

9.2.1 Read SIF Data

The **Read Sif Data** button (Fig 9.2.1) allows one to read the *.fcg* file. The *.fcg* file is FRANC3D's file format for storing fatigue crack growth data, which includes all the SIF history data. The *.fcg* file can be created using the **Advanced** menu option: **View/Write Growth History**, Section 14.4. The file selector dialog, Fig 9.2.2, is displayed.

9.2.2 Set Parameters

The **Set Parameters** button leads to the definition of the crack growth rate model, which was described in Section 6.5.1.

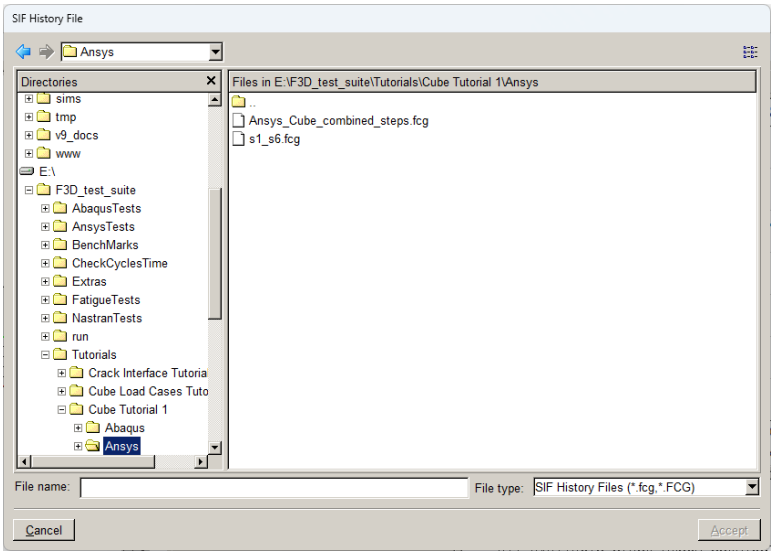


Figure 9.2.2 File selector dialog for *.fcg* files.

9.2.3 Read Parameters

The **Read Parameters** button allows one to read a previously saved crack growth rate model from a file. The file selector dialog, Fig 9.2.3 is displayed with the available *.cgp* files.

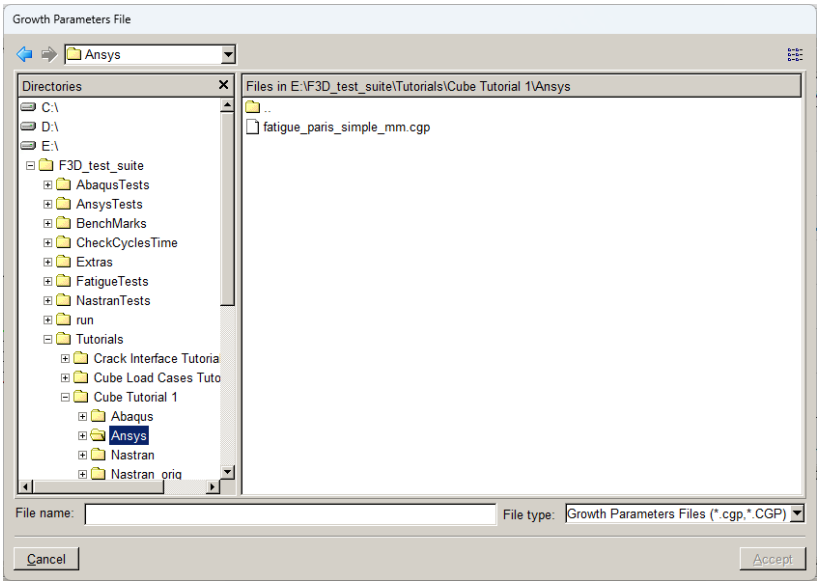


Figure 9.2.3 File selector dialog for *.cgp* files.

9.2.4 Save Parameters

The **Save Parameters** button allows one to save the current crack growth model data to a file. The file selector is displayed; the user can enter a *.cgp* file name. This file can be re-used as in Section 9.2.3.

9.2.5 Plot X axis – Plot Y axis

The right-side pane can be used to plot the available data. By default, the crack growth step number is plotted along the x-axis and the y-axis shows cycles.

9.2.5.1 Plot options

There are eight plot options: Cycles, Passes, Time, Growth Step, Path Length, Area, MAX(Kmax) and MAX(Khold). Options are only active if the data exists.

Growth Step corresponds to the discrete steps of crack growth that the user defines for analysis. The cycle count at each step is the average cycle count based on all mid-side nodes as described in Section 9.1.

Path Length is activated by using the **Define Path** button. Pressing the **Define Path** button, invokes the dialog shown in Fig 9.2.4. This dialog is the same as for the **Path** tab described in Section 6.9.1.

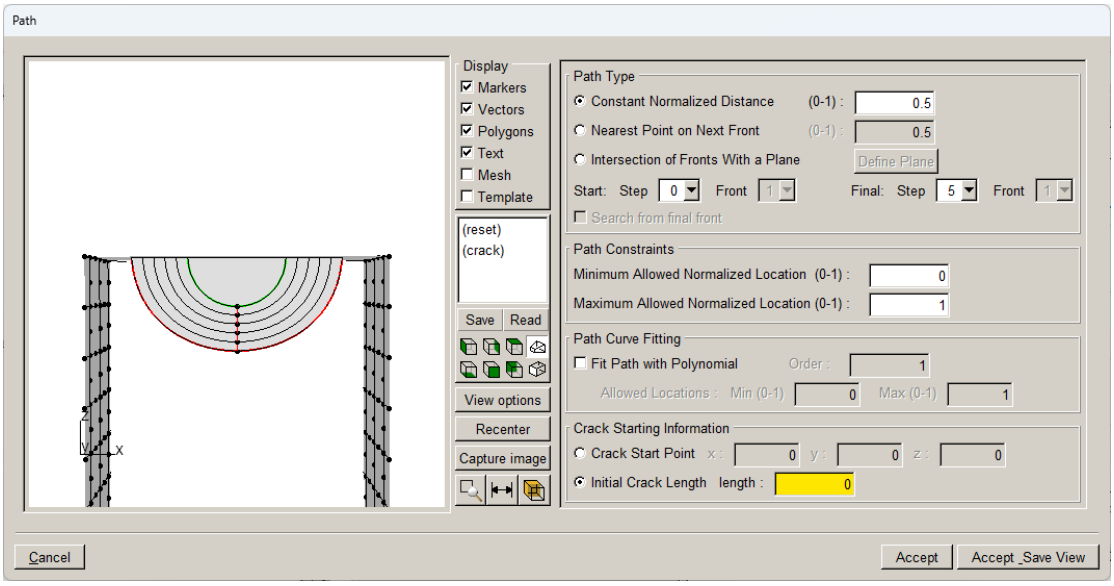


Figure 9.2.4 Define Path dialog for plotting fatigue cycles vs path.

9.2.6 Data Table

The Data Table option displays whatever data is currently active in tabular form, Fig 9.2.5. The dialog has an **Export** button that allows one to save this data to a file. Press the **Write file** button to save the data to a *.txt* file. The WriteFatigueData command is written to the session log; see Section 2.1.57 in the Commands & Python reference.

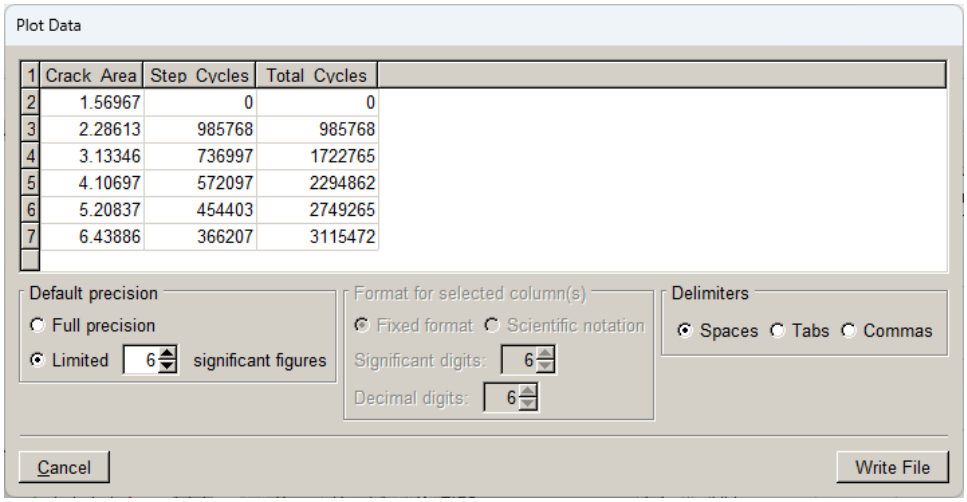


Figure 9.2.5 Data table dialog.

9.2.7 Crack Front Plots

The Crack Front Plots option displays SIFs and other data, Fig 9.2.6. There are tabs along the top right that allow one to display K_{max}, K_{min}, ΔK, K_{eq}, K_{static}, K_{hold}, R, and Temperature. The data can be displayed in a table or exported to a file using the Table and Export tabs. The WriteGrowthData command is written to the session log; see Section 2.1.58 in the Commands & Python reference.

Data can be displayed for each crack front and for each crack growth step.

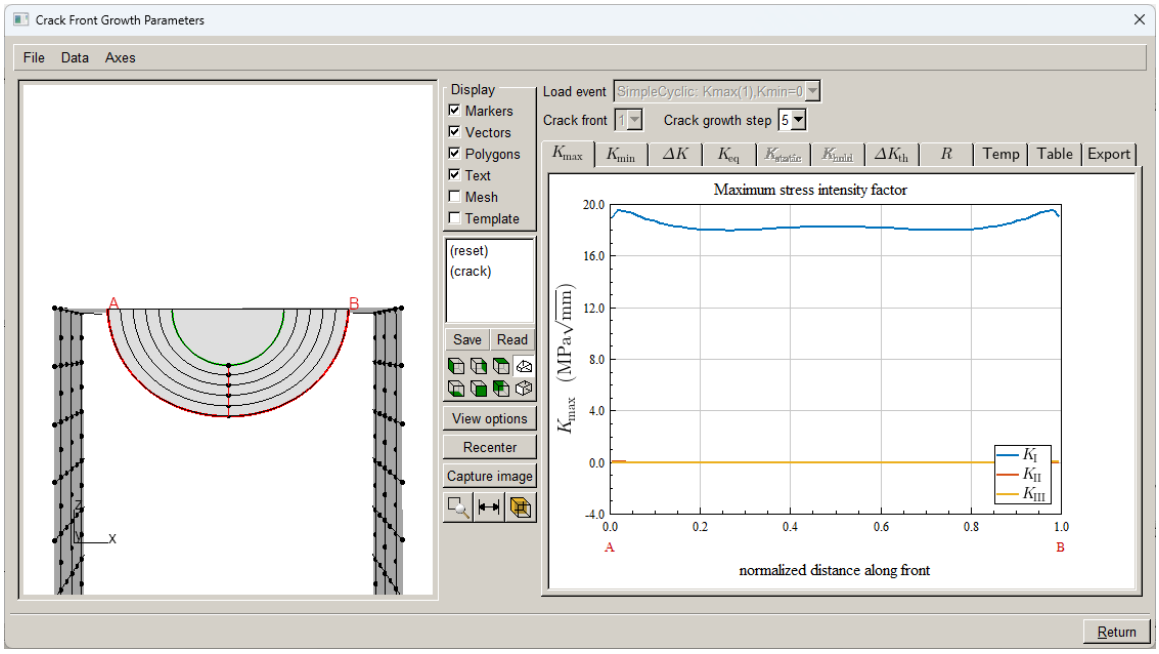


Figure 9.2.6 Crack Front Plots dialog.

If the Path option is selected (see Fig 9.2.4), the Crack Front Plots dialog displays the data along the defined path, Fig 9.2.7. There is an extra Path tab that can be used to change the path. The data can be displayed in a table or exported to a file using the Table and Export tabs. The WriteGrowthPathData command is written to the session log; see Section 2.1.59 in the Commands & Python reference.

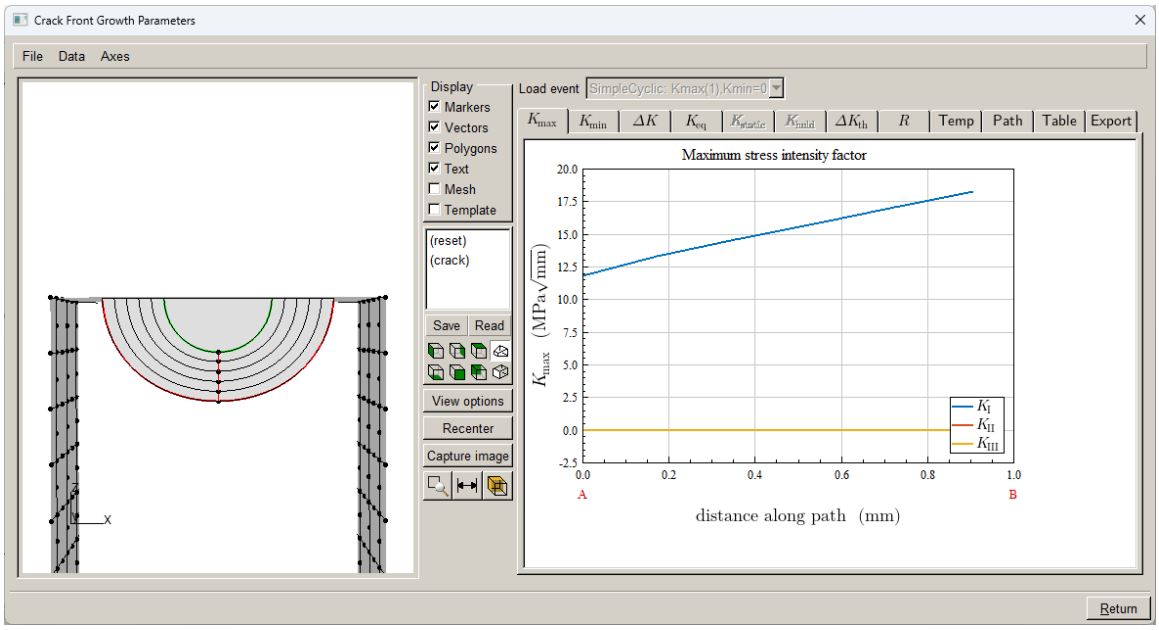


Figure 9.2.7 Crack Front Plots along a path dialog.

9.2.8 Advanced Options

The Advanced Options button is used to set the starting crack step and to limit the amount of crack front, Fig 9.2.8. The default uses the initial crack front (# 0) as the starting point for fatigue cycle counting; the user can change this to a later crack growth step. The default Lower and Upper bounds are set so that the cycle counts from the two ends of the crack front are not included in the final average for the front; the user can change these limits here.

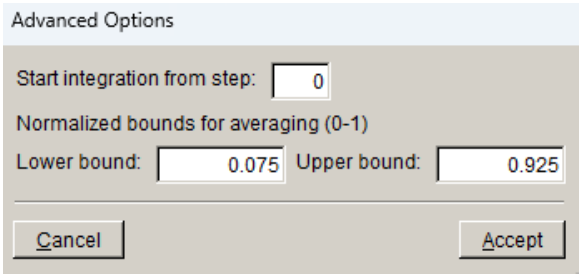


Figure 9.2.8 Advanced Options dialog.

9.3 View/Edit Growth Parameters

The View/Edit Growth Parameters menu option allows the user to view and edit all the current crack growth parameters. The dialogs described in Section 6.5 are presented with the current settings and values; these can be edited here.

9.4 Crack Front Fatigue Values

The Crack Front Fatigue Values menu option displays the dialog shown in Fig 9.2.6.

9.5 Fatigue Values Along a Path

The Fatigue Values Along a Path menu option displays the dialog shown in Fig 9.2.7.

9.6 Growth Rate File

The Growth Rate File menu option generates a file giving the crack extensions for the events in the schedule. The file format is:

```
1.11774e-05     7.70014e-06     3.47724e-06
1.07664e-05     7.41722e-06     3.34917e-06
1.01888e-05     7.01946e-06     3.16935e-06
9.74347e-06     6.7128e-06      3.03067e-06
…
```

There is one row for each crack front point. The first column gives the total crack extension for each point for one pass through the load schedule. The second and third columns give the extensions for one pass through events 1 and 2, respectively. The WriteGrowthRateFile command is issued to the session log; see Section 2.1.61.

10. Fretting Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Fatigue** menu option are described in this section.

10.1 Read Model and Results

FRANC3D can read ANSYS and ABAQUS files for fretting analysis. The FE information is contained in *.cdb* and *.inp* files, respectively, Fig 10.1.1. FE information consists of nodes, elements, and a definition of the contact surfaces.

Once the FE model is imported, analysis results must be imported. A typical fretting test is conducted by applying a normal load (P) and then applying a maximum and minimum tangential (shear) load (Q_{max} and Q_{min}). Simulating this can produce a pair of results files for the Q_{max} and Q_{min} conditions. In most cases, the Q load is applied in smaller increments, which provides results at multiple intermediate substeps, as well as the Q_{max} and Q_{min} maximum load steps.

Select the Fretting **Read model and results** menu item, to display the dialog in Fig 10.1.1, and choose the fretting model. The File type can be changed to Abaqus to display *.inp* files.

Once the FE model is chosen, the results file must be selected, Fig 10.1.2. One can choose to read a pair of analysis results, corresponding to the Q_{max} and Q_{min} load steps, or read a *.dtp* file with multiple load step results.

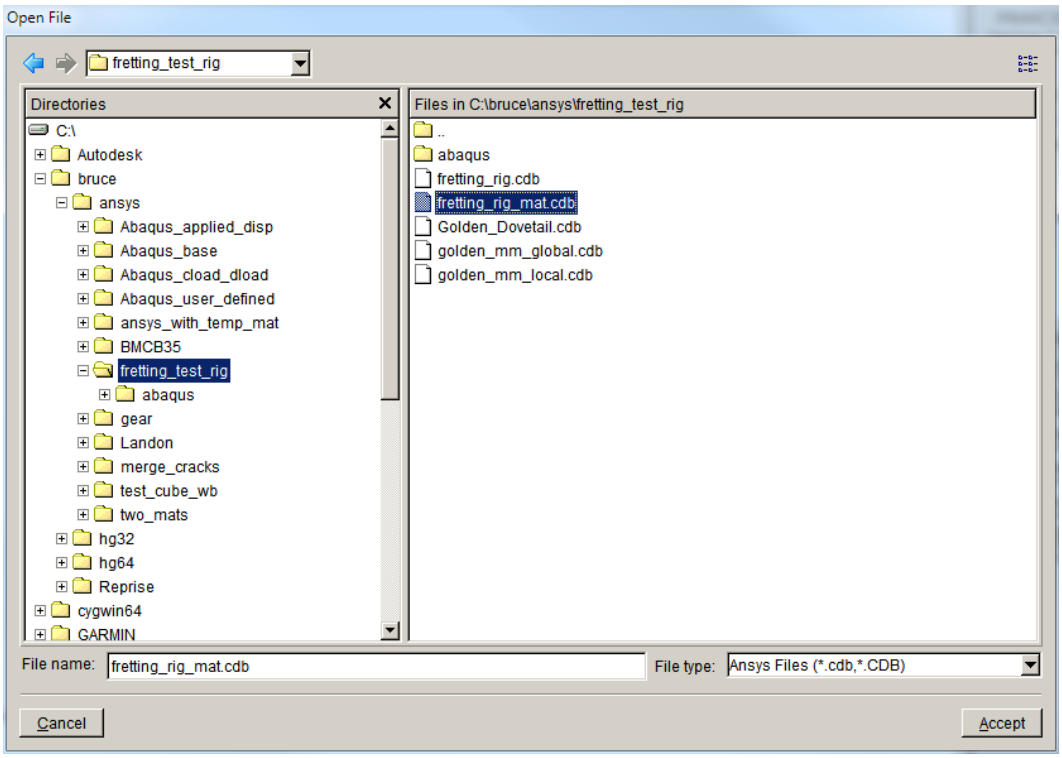


Figure 10.1.1 Fretting dialog for importing FE model data.

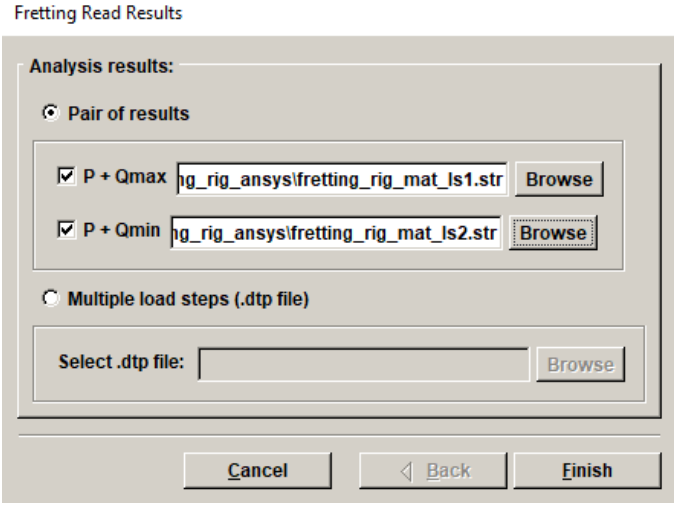


Figure 10.1.2 Fretting Read Results dialog.

For the Pair of results option, select the **Browse** button for Qmax or Qmin to display the dialog shown in Fig 10.1.3. If an ANSYS *.cdb* file was imported, ANSYS results files are expected. The ANSYS *.str* files are simply the nodal stress listings saved to a file. Corresponding strain (*.stn*), displacement (*.dsp*) and contact status (*.con*) listings should also exist with the same name prefix.

ABAQUS results are read from *.fil* files.

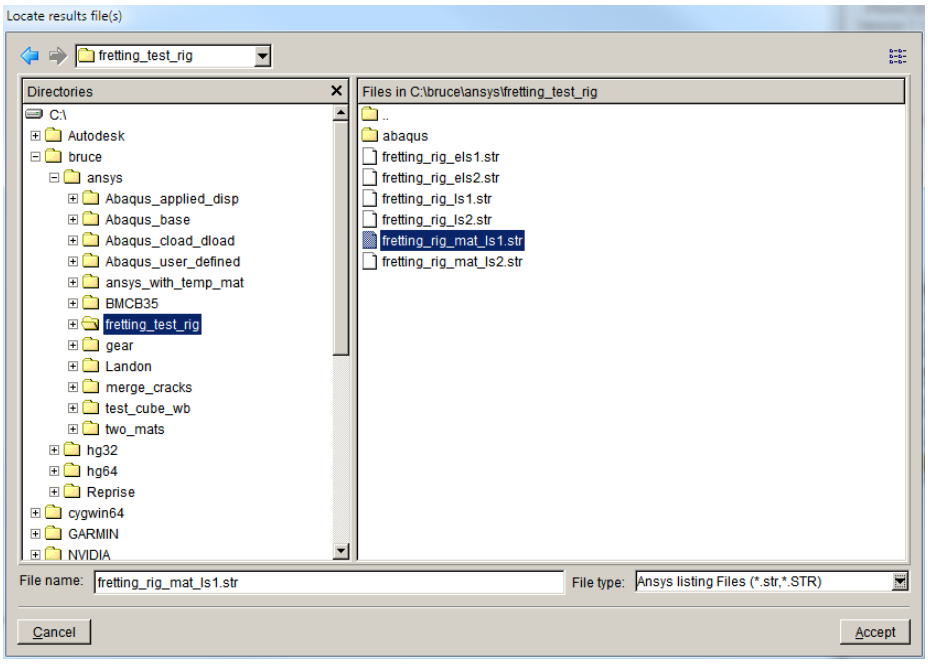


Figure 10.1.3: Fretting dialog for locating results files.

Selecting the Multiple load steps option activates the Select .dtp file field, Fig 10.1.4, and allows one to **Browse** for the *.dtp* file. The *.dtp* files can be created from ANSYS using APDL commands and from ABAQUS using Python commands.

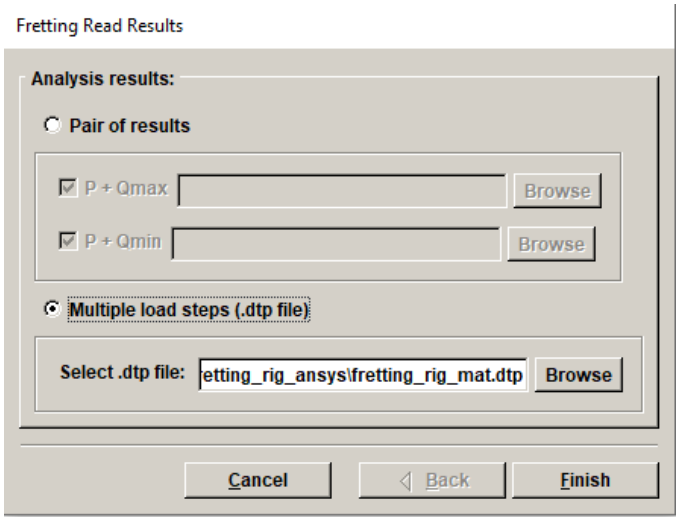


Figure 10.1.4: Fretting dialog for locating results files.

FRANC3D reads the mesh information and results and determines the contact surfaces. The user can choose the contact pair to be analyzed using the dialog shown in Fig 10.1.5. The contact

pairs are listed on the left side and colored blue in the model. The blue surface is colored red when it is selected. Note that only one contact pair is analyzed at a time.

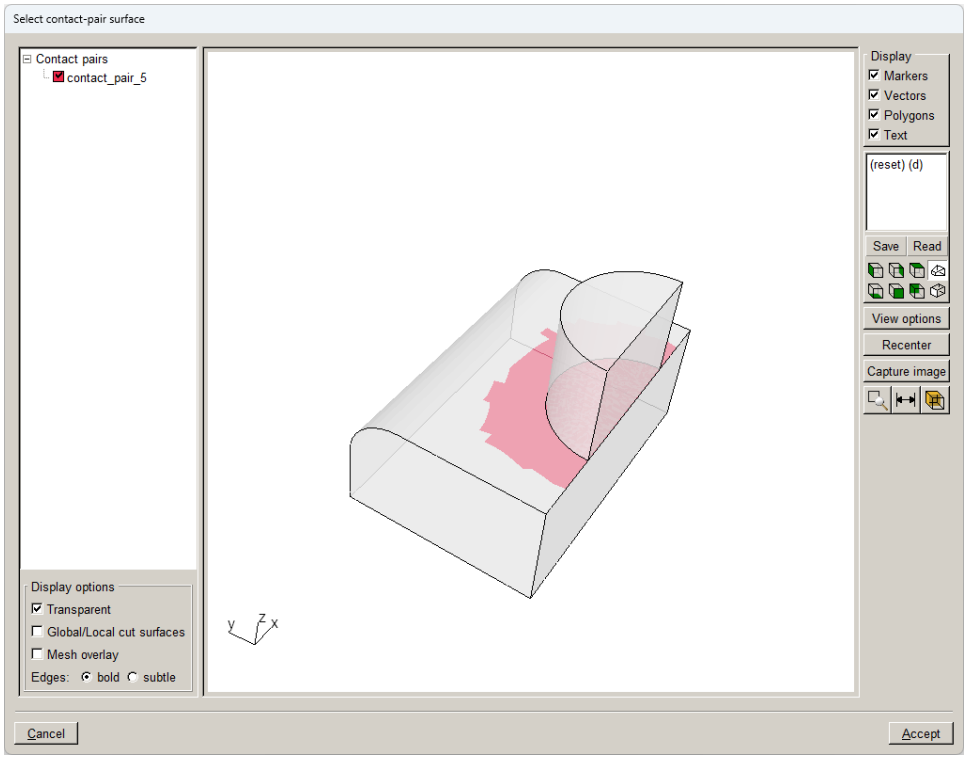


Figure 10.1.5 Select contact-pair surfaces dialog.

10.2 Import Nucleation Data

FRANC3D can import experimental fretting fatigue test data and perform curve fit operations on the data. Select the **Import Nucleation Data** menu item to display the dialog shown in Fig 10.2.1. The user can select a file containing the experimental data using the **Read from file** button. The format for this text file is:

sigma_axial	cycles
675.3	2.23e4
655.7	2.63e4
533.8	4.45e4

The fretting data is shown in tabular and graph form, Fig 10.2.2. The user can perform curve fitting on the data using the **Curve Fit** drop-down menu. The menu provides seven different equations for comparison, but the standard fretting model corresponds to the Power Law 2 equation.

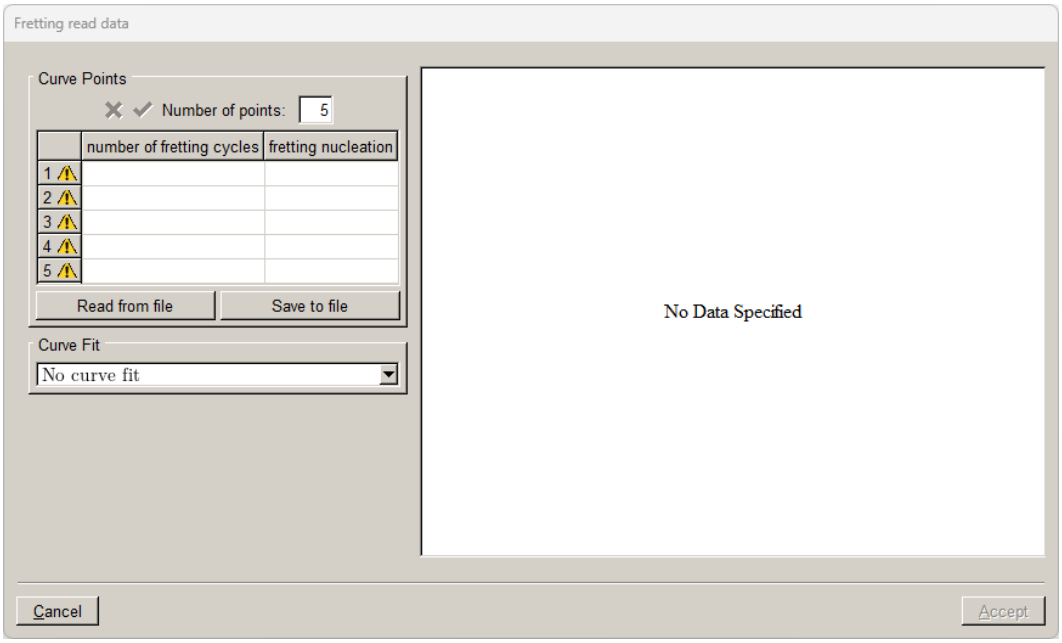


Figure 10.2.1 Fretting dialog for importing experimental fretting nucleation data.

The usual fretting nucleation model uses a power law relationship with four coefficients to relate the fretting nucleation cycles on the horizontal axis with the fretting parameter on the vertical axis. This power-law is shown in Fig 10.2.2, which also shows the fit through the data. The goodness-of-fit is displayed on the top of the plot along with the equation coefficients.

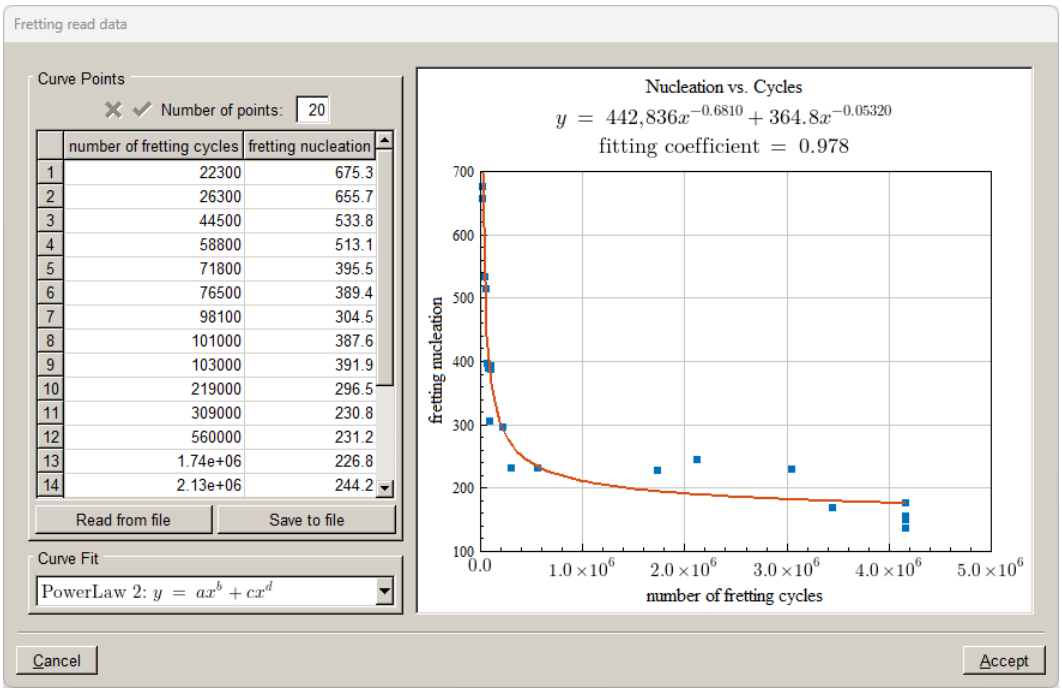


Figure 10.2.2 Fretting nucleation data with power law curve fit through the data.

10.3 Fretting Crack Nucleation

FRANC3D computes fretting fatigue crack nucleation cycles and initial crack location (and possibly the orientation) based on one or more of the fretting nucleation models that have been implemented. These include: the equivalent stress, the critical plane shear stress amplitude, the critical plane Smith-Watson-Topper, the Ruiz-Chen model, and a modified Smith-Watson-Topper model. The model equations and parameters are briefly summarized here.

For more details, refer to: *Three-dimensional Simulation of Fretting Crack Nucleation and Growth, B.J. Carter, E.C. Schenck, P.A. Wawrzynek, A.R. Ingraffea, and K.W. Barlow, EFM, 96 (2012), p.447-460.*

The equivalent stress model is defined by:

$$\sigma_{eq} = 0.5\ (\Delta\sigma_{psu})^{w}\ (\sigma_{max})^{1-w} \tag{1}$$

where $\Delta\sigma_{psu}$ and σ_{max} are functions of the nodal stress components and w is a material dependent fitting parameter. Based on curve fitting of experimental data, the equation in the form of the power law:

$$\sigma_{eq} = aN_i^{b} + cN_i^{d} \tag{2}$$

relates σ_{eq} to N_i, where coefficients a – d are material dependent fitting parameters. For example, for Ti-6Al-4V, a = 52476, b = -0.6471, c = 450.85 and d = -0.03582.

The critical plane shear stress amplitude model is defined by:

$$\gamma_{crit} = \tau_{max}/G\ (1-R_{\tau})^{m} \tag{3}$$

where τ_{max} is the maximum shear stress on the critical plane, R_{τ} is the shear stress ratio, G is the shear modulus, and m is a material dependent fitting parameter. Substituting γ_{crit} for σ_{eq} in equation (3), one can compute N_i given the appropriate values for the coefficients a – d. For Ti-6Al-4V, a = 6.46, b = -0.686, c = 4.65x10-3 and d = 1.22x10-2.

The critical-plane Smith-Watson-Topper model is defined by:

$$\Gamma_{crit} = \sigma_{max}\ (\Delta\varepsilon/2) = (\sigma_f^2 / E)\ (2N_i)^{2b} + \sigma_f\ \varepsilon_f\ (2N_i)^{b+c} \tag{4}$$

where σ_{max} is the maximum principal stress and $\Delta\varepsilon$ represents the normal strain amplitude on the critical plane; this equation is already in the form of equation (3). E is the elastic modulus, σ_f and ε_f are material strength parameters, and b and c are fitting parameters. For Ti-6Al-4V, σ_f = 376.6 MPa, b = 1.78x10-4, c = -0.767 and ε_f = 32.4.

The Ruiz-Chen model fretting parameter, F_2, is equal to the product of the shear stress (τ), relative slip (δ), and bulk normal stress (σ):

$$F_2 = \sigma \tau \delta \tag{8}$$

The number of cycles is computed by replacing σ_{eq} in equation (2) with F_2. The parameters a, b, c, and d will be material dependent.

Recent work by Hirsch[2] (2013) following work by Ding *et al*[3] (2007) indicate that a fretting model that combines the Ruiz-Chen and SWT (or Fatemi-Socie) models can more effectively predict fretting crack nucleation location and cycles, in addition to accounting for the transition from fretting to wear. Hirsch (2013) used the Fatemi-Socie model for his material and test specimens and proposed the following:

$$FS \cdot \mathrm{D} = f(N_f) \tag{9}$$

where

$$D = \left[(1-a)e^{bRW_{eff}} + a\right]\left[(1-c)e^{dRW_{eff}} + c\right]$$

$$W_{eff} = \frac{k_{dam}}{1+\alpha_{cp}}(\tau\delta)$$

and a, b, c, and d are constants, FS is the Fatemi-Socie parameter, R is the wear rate, W_{eff} is the effective work, k_{dam} is the fraction of the total frictional energy that goes toward fatigue damage, α_{cp} is the ratio of the frictional energy, and ($\tau\delta$) is the frictional work.

The first dialog box that is displayed after selecting the **Fretting Crack Nucleation** menu item, if there are more than two load steps, is shown in Fig 10.3.1. The user can specify the pair of load steps of interest or have FRANC3D process all load step pairs to find the pair that maximizes the fretting parameter.

[2] Hirsch, M. (2013) Temperature Dependent Fretting Damage Modeling of AISI 301 Stainless Steel. PhD Thesis, Georgia Institute of Technology.
[3] Ding, J. *et al*., (2007) Fretting fatigue predictions in a complex coupling. International Journal of Fatigue, 29, 1229-44.

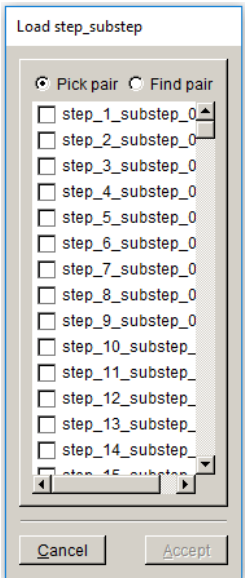


Figure 10.3.1 FRANC3D **Fretting** dialog to select load step results.

The next dialog box, Fig 10.3.2, allows the user to select one of the fretting nucleation models. Selecting **Next** leads to the dialog shown in Fig 10.3.3, which lets the user define the fretting model parameters. Note that the user can select the fitting coefficients based on a curve fit through the experimental data (see previous section). If these coefficients are not available, the option for curve fit values is disabled. All fretting models have similar dialogs, Figs 10.3.4 - 10.3.7.

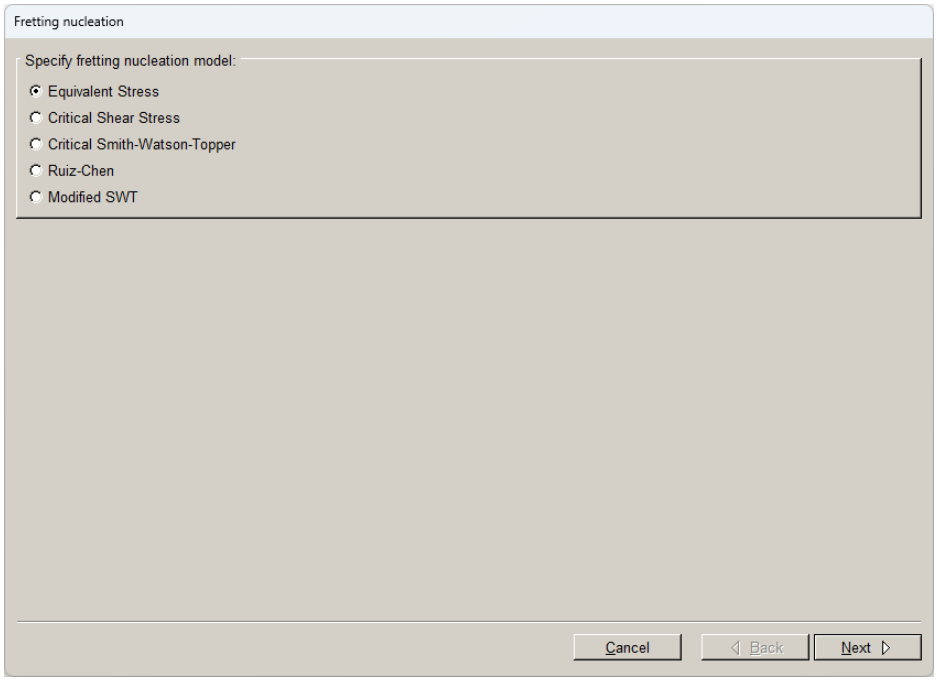


Figure 10.3.2 Dialog to select fretting nucleation model.

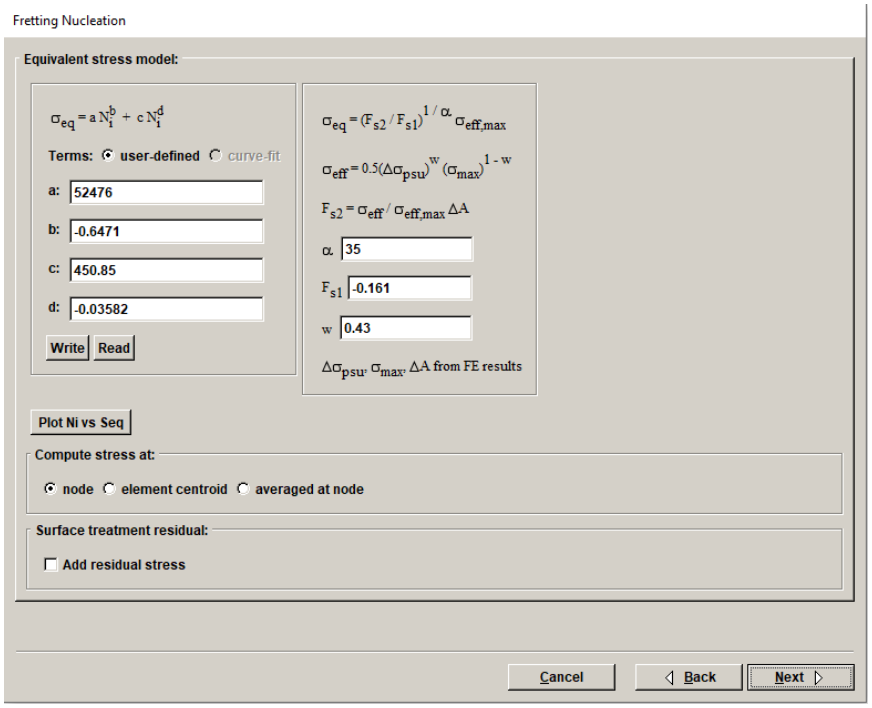


Figure 10.3.3 Fretting dialog to set equivalent stress fretting model parameters.

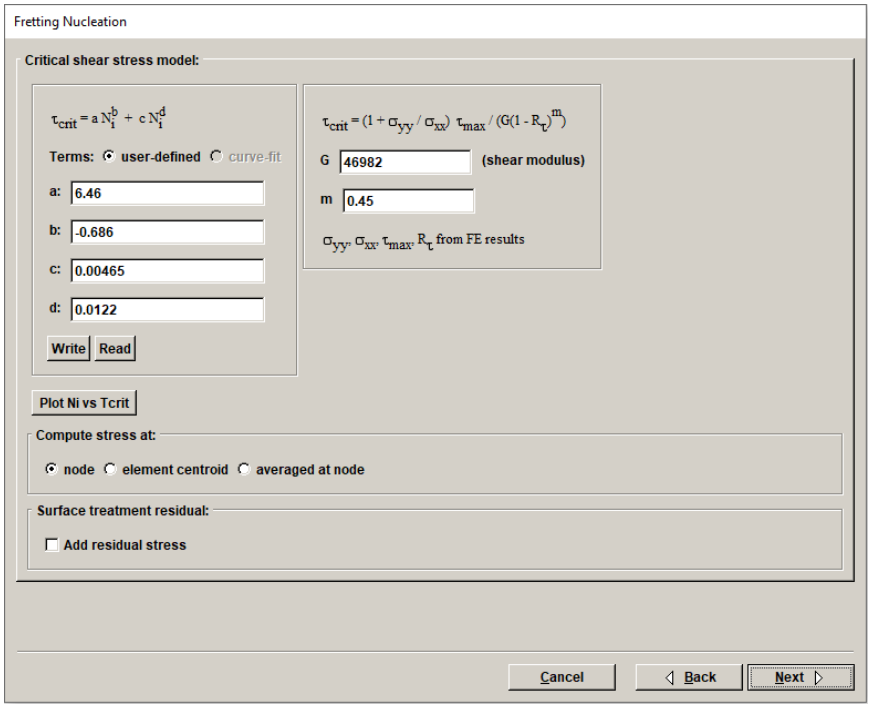


Figure 10.3.4 Fretting dialog to set critical shear stress model parameters.

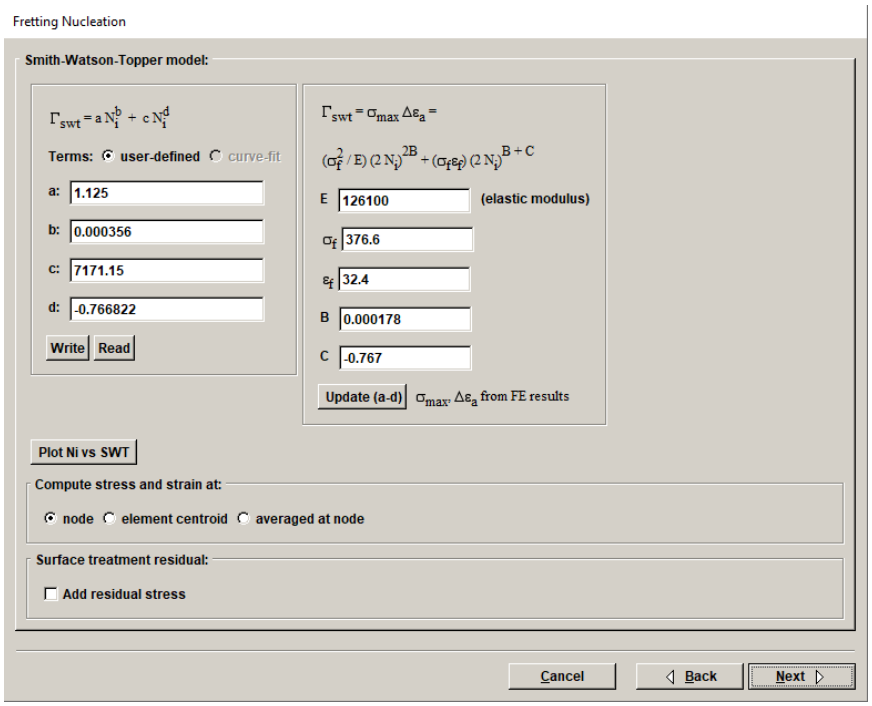


Figure 10.3.5 Fretting dialog to set Smith-Watson-Topper (SWT) model parameters.

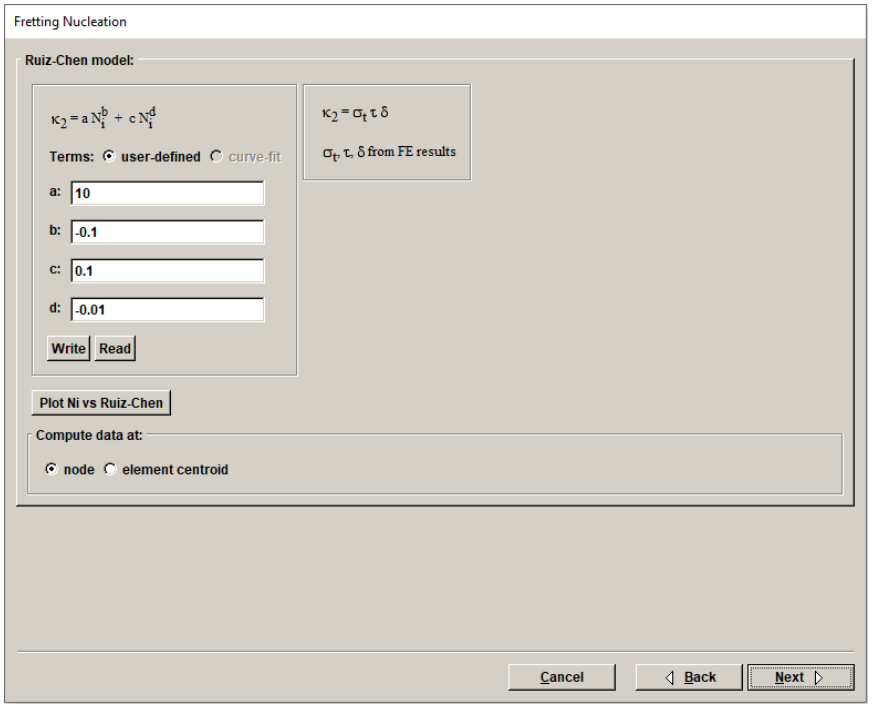


Figure 10.3.6 Fretting dialog to set Ruiz-Chen model parameters.

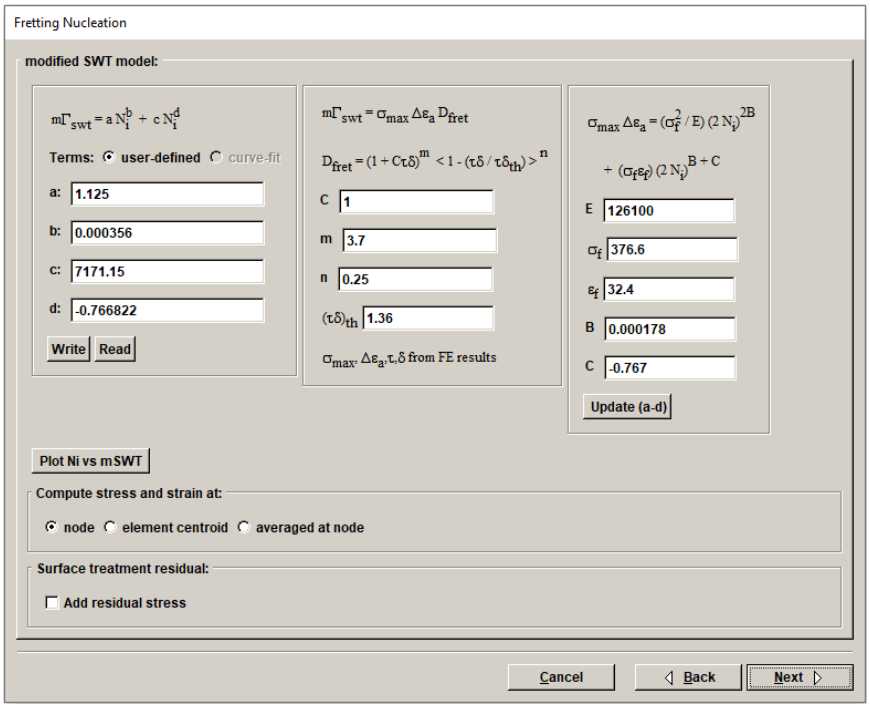


Figure 10.3.7 Fretting dialog to set modified SWT (FS) model parameters.

The dialog boxes shown in Figs 10.3.3 - 10.3.7 have an option for averaging stress and strain. Rather than computing the fretting parameters based on surface node values, the user can choose the element centroid or an averaged value. If the averaged at node option is chosen, the next dialog box is shown in Fig 10.3.8. The user can adjust the size of the displayed sphere by changing the Averaging length value. Note that stress values are not averaged across material region boundaries. Typical length values used in the literature are based on material grain size.

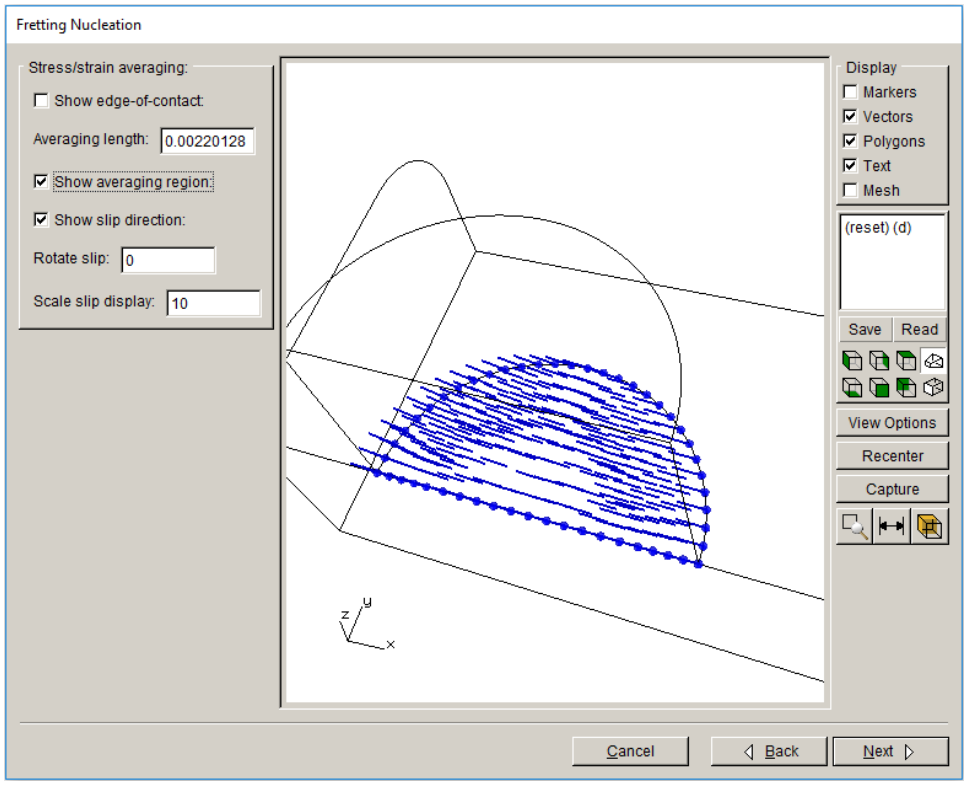


Figure 10.3.8 Fretting dialog for volume averaging of stress (and strain).

Selecting **Next** on the dialog shown in Fig 10.3.8 leads to the dialog shown in Fig 10.3.9. This dialog shows the model and allows the user to view the contact surfaces, the edge of contact, the fretting nucleation cycles, and the fretting parameter values. The latter two items can be displayed as text or using color contours.

Selecting the **Save to File** button on the dialog shown in Fig 10.3.9 leads to the file save dialog shown in Fig 10.3.10. The fretting nucleation data is saved to a text file with the following format:

primary surface

node	x	y	z	parameter	cycles
19222	0.49648972	0.91291108	0.15	25	19283
20917	0.52414545	0.94056681	0.12	29	18453
20881	0.51283174	0.92925310	0.12	24	19898

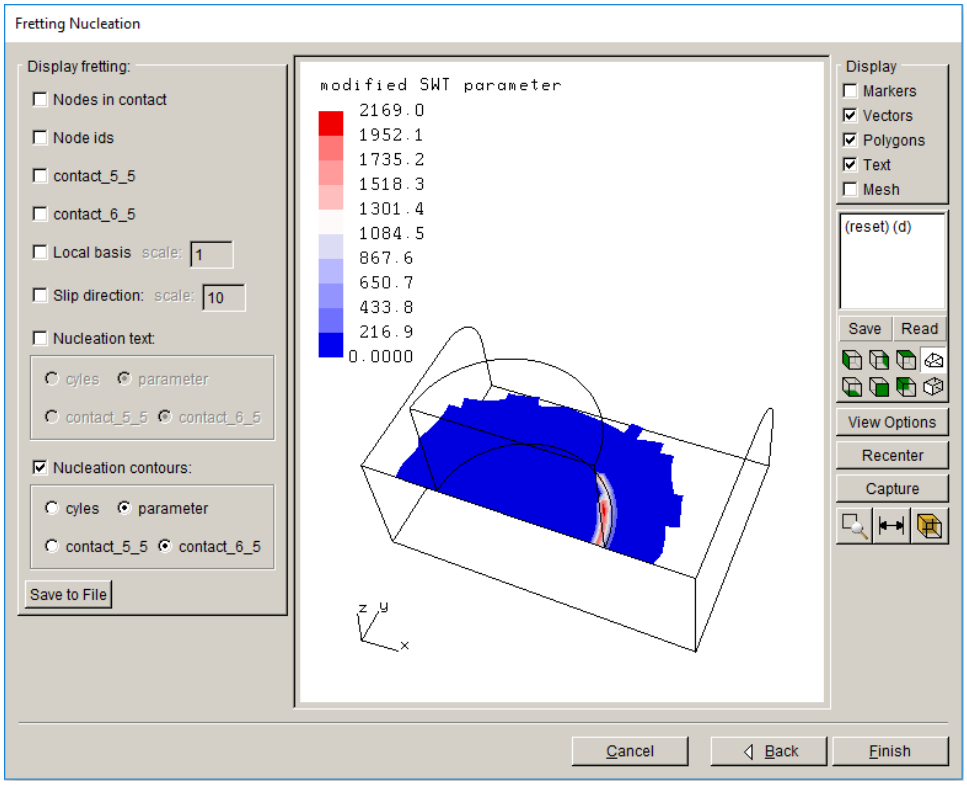


Figure 10.3.9 Fretting dialog for displaying fretting nucleation cycles and parameter values and contours.

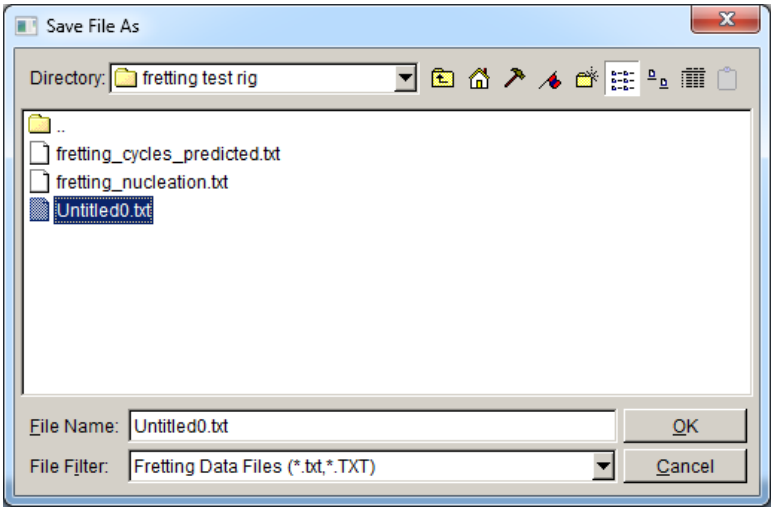


Figure 10.3.10 Fretting dialog for selecting file to save fretting nucleation predictions.

The user can go **Back** and choose a different fretting nucleation model to compare predictions.

10.3.1 Surface treatment residual

An additional option for each of the fretting models (except for the Ruiz-Chen model) is the ability to add a surface treatment residual stress. Checking the Add residual stress box leads to the dialog in Fig 10.3.11. The residual stress treatment is the same as described in Section 7.2.4. The stress versus distance from the surface is specified or read from a file, and then the treated surface is identified. The residual stress is added to (or subtracted from) the bulk stress when computing the fretting parameters.

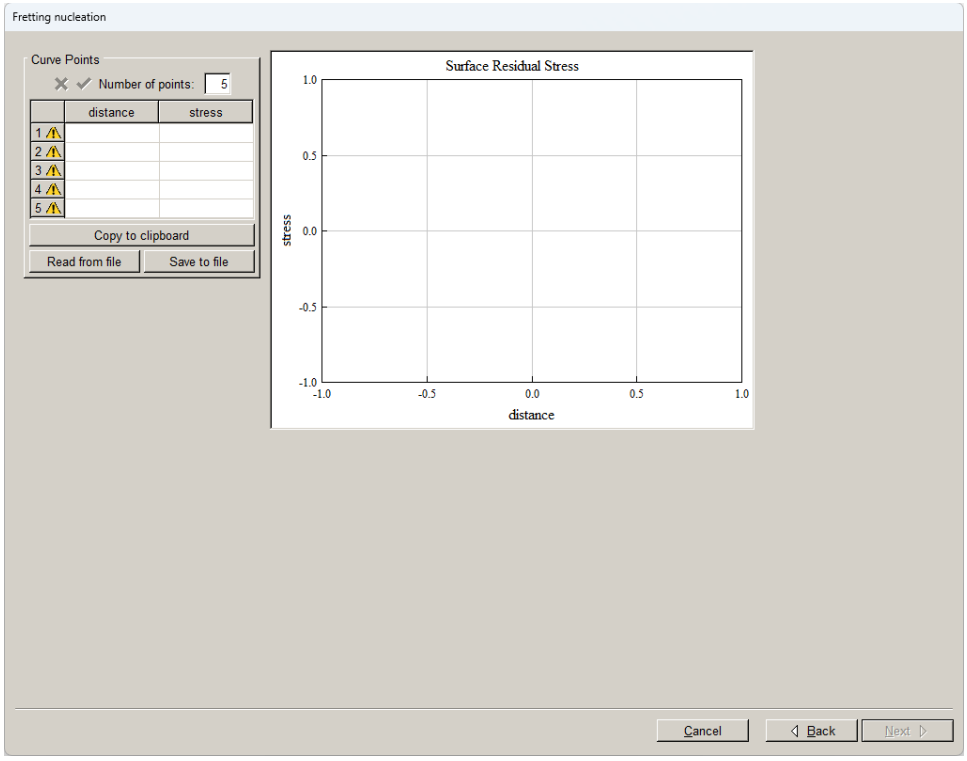


Figure 10.3.11 Fretting dialog for selecting file to save fretting nucleation predictions.

10.4 Region Color

FRANC3D will display material regions with distinct colors. The **Region Color** menu item allows the user to turn on or off the region coloring, Fig 10.4.1. This option is purely for display purposes, and regions can be differentiated in complex multi-region models.

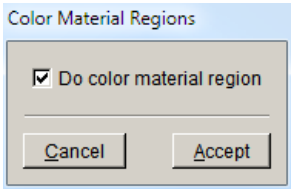


Figure 10.4.1 Fretting dialog for turning off/on region coloring.

11. Display Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Display** menu options are described in this section.

11.1 View Response Dialog

The **View Response** menu item allows one to view the deformed shape of the current model. Analysis results are required to activate this menu item. The dialog shown in Fig 11.1.1 is displayed.

The Display Mesh Overlay can be turned off to remove the surface mesh facets from the model view.

The deformed shape can be displayed by turning on the Display on Deformed Geometry option and entering a non-zero magnification factor, Fig 11.1.2. If there are multiple load steps, the deformed shape for a specific load step or for the sum of all load steps can be selected.

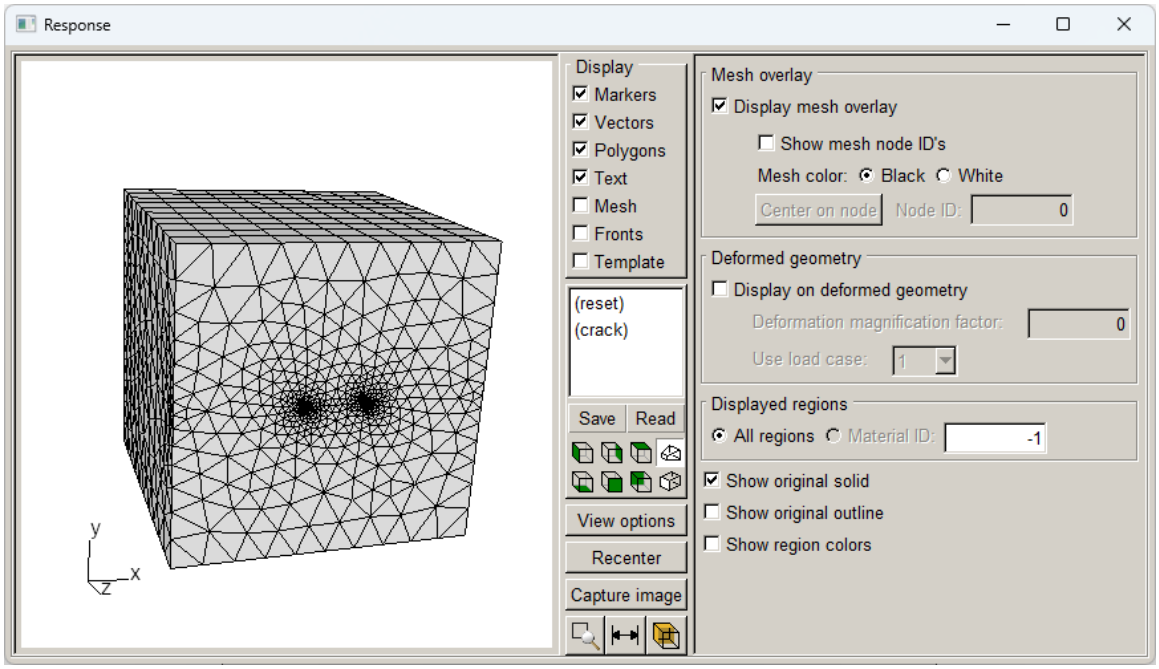


Figure 11.1.1 View Response dialog showing undeformed mesh.

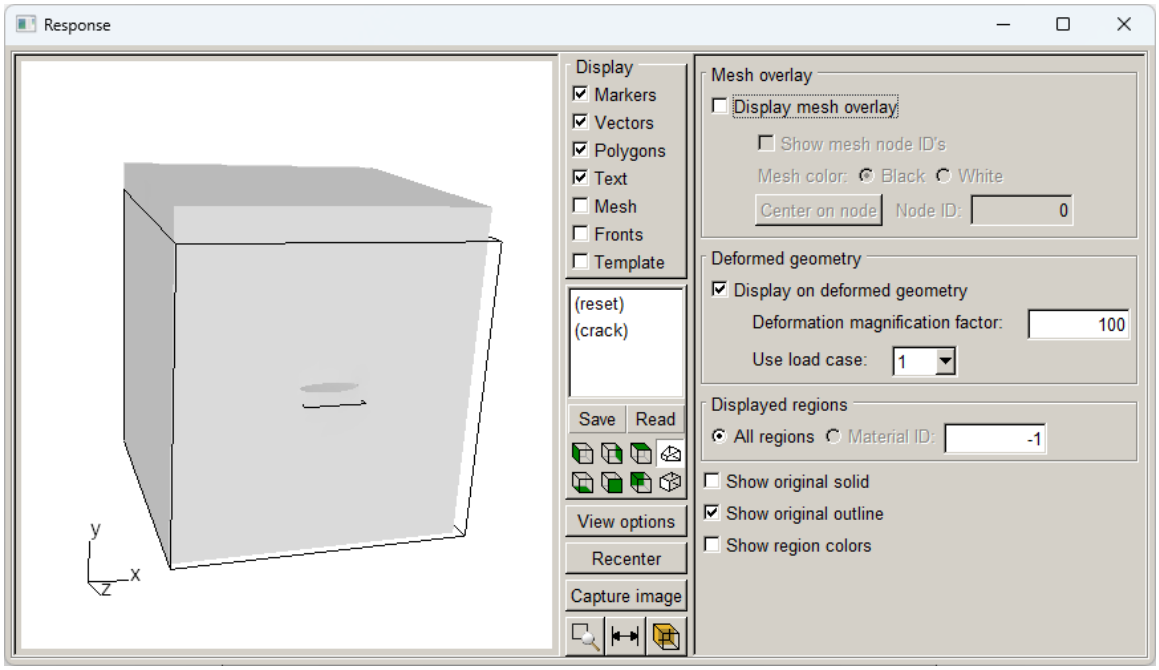


Figure 11.1.2 View Response dialog showing the deformed shaded solid and the original wireframe outline.

11.2 Create Animation Wizard

The **Create Animation** menu item allows one to create and save crack growth animation files. The dialog, Fig 11.2.1, allows one to select a set of FRANC3D restart (*.fdb*) files to create an animation of the crack growth.

The **File** – **Read Files** menu option displays the file selector dialog, Fig11.2.2. One can select the set of *.fdb* files for the crack growth animation. Use the Shift or Ctrl key to select multiple files.

Select **Accept** when ready, and FRANC3D will read each of the *.fdb* files along with the associated FE files. This process can take some time depending on the number of files and the size of the model; a status bar is displayed as the files are processed.

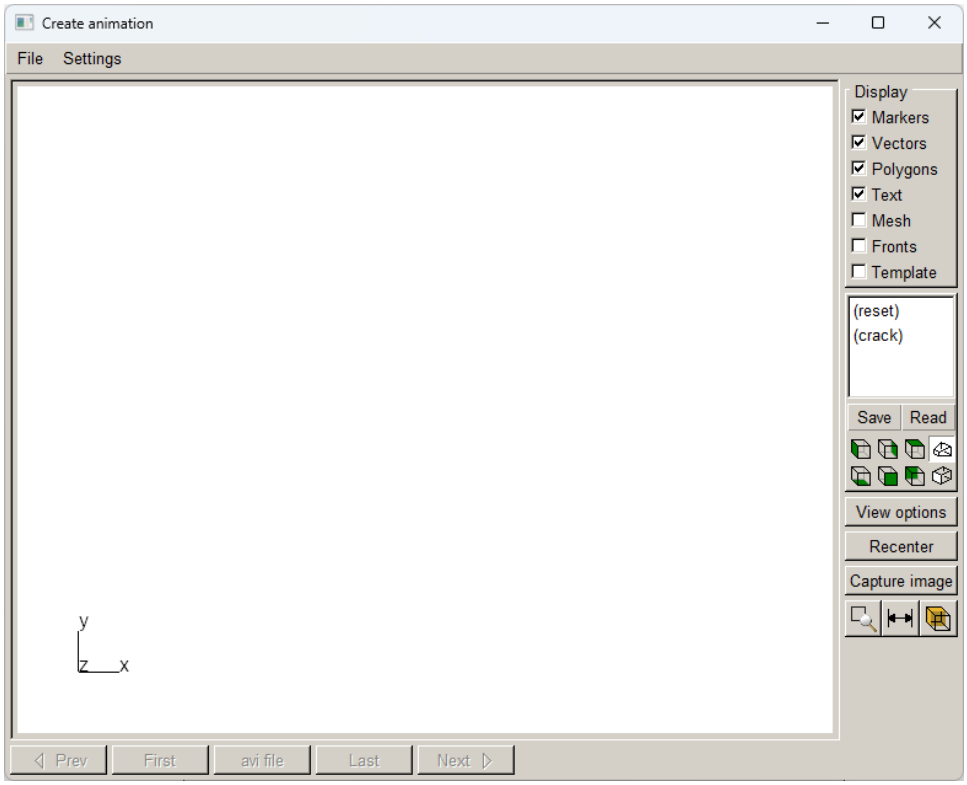


Figure 11.2.1 Create Animation dialog.

Once all files have been processed, the first crack model is displayed, Fig 11.2.3. The ordering of the crack models is based on the file name.

The **Last** and **Next** buttons at the bottom are activated; the **First** and **Prev** buttons will be activated if one steps to the next or last crack model.

The **avi file** button is also activated. This allows one to save an *.avi* file of the crack growth steps. The file consists of a series of images (one for each step of crack growth) based on the current model view (camera position). You should orient the model into a camera position that shows all of the crack growth steps before clicking the **avi file** button.

The **Settings** menu has two options: 1) Frame Rate and 2) Angle Threshold. The frame rate of the *.avi* file can be adjusted to speed up or slow down the animation. The angle threshold controls the display of the geometric lines and surfaces (see Section 14.1).

Note: if the last step of crack growth is not displayed correctly when playing, make a copy of the last step, and include it at the end of the list of files that are selected. For example, in Fig 11.2.2, the extra copy of the '_step_006' might be called "_step_006b'.

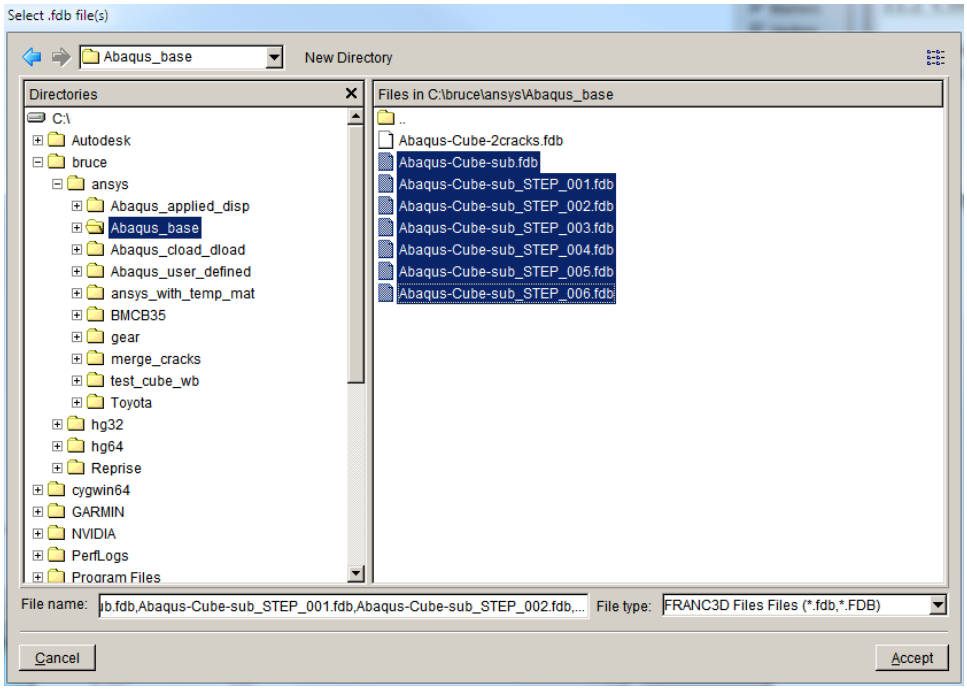


Figure 11.2.2 Create Animation select *.fdb* files.

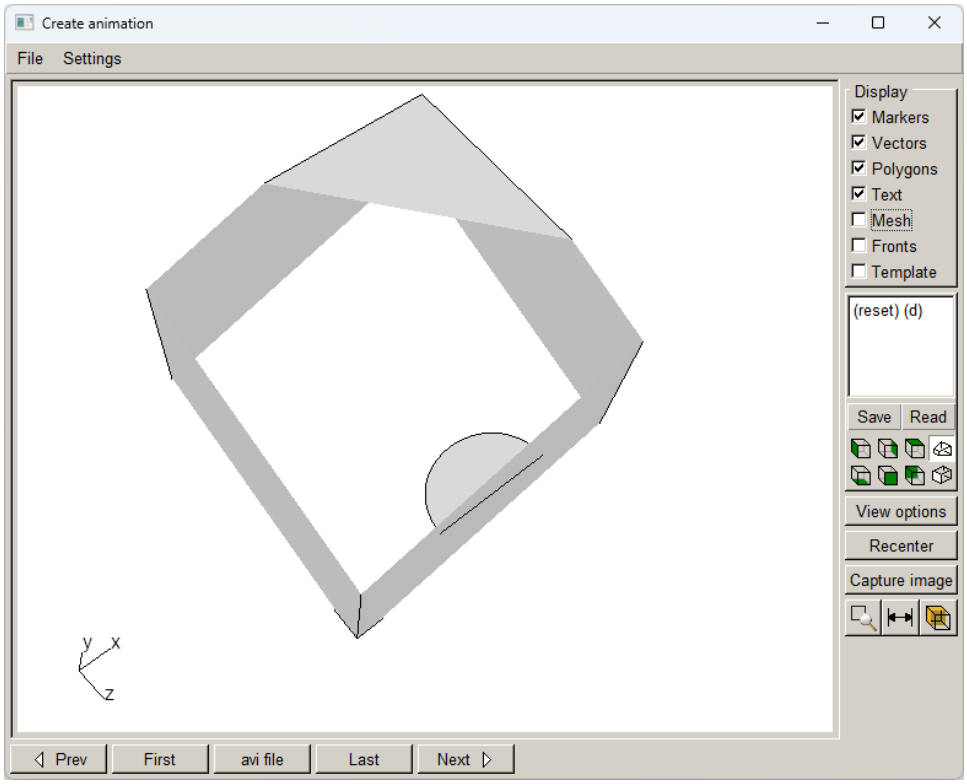


Figure 11.2.3 Create Animation dialog with cracked model imported.

12. Single Crystal Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Single Crystal** menu options are described in this section.

12.1 Resolved Slip System Ks

The **Resolved Slip System Ks** menu item allows one to view the SIFs along the crack front resolved onto slip planes. The first dialog, Fig 12.1.1, allows you to select the SIF computation method; this is the same as in Section 6.4. The SIFs Display dialog, Fig 12.1.2, is different from the standard SIF display (see Section 6.4); it includes options to display the SIFs resolved into the various slip planes (e.g., 111). Rather than the standard three SIF modes (K_I, K_{II} and K_{III}), one can plot K_{RSS}, K_{RNS} and K_{RSS}/K_{RNS}.

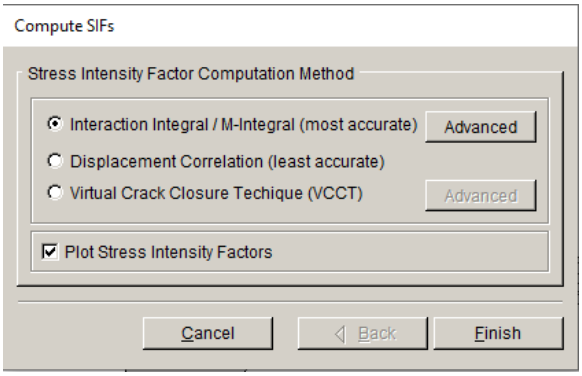


Figure 12.1.1 Compute SIFs computation method dialog.

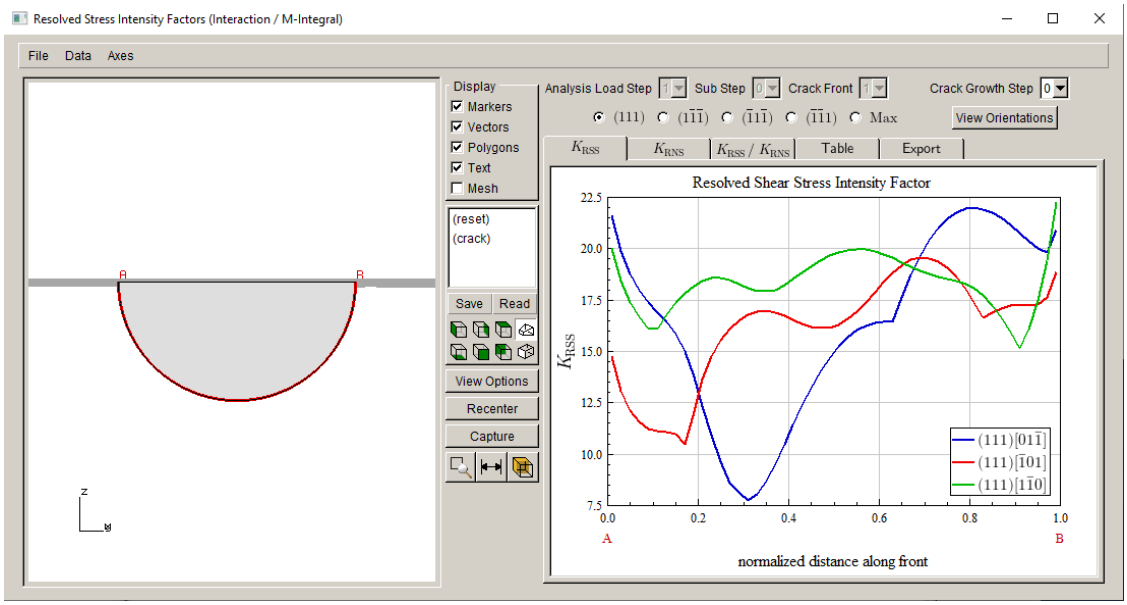


Figure 12.1.2 Resolved SIFs display dialog.

The **View Orientations** button displays the dialog in Fig 12.1.3. This gives a visual display of the slip plane orientation relative to the crack orientation.

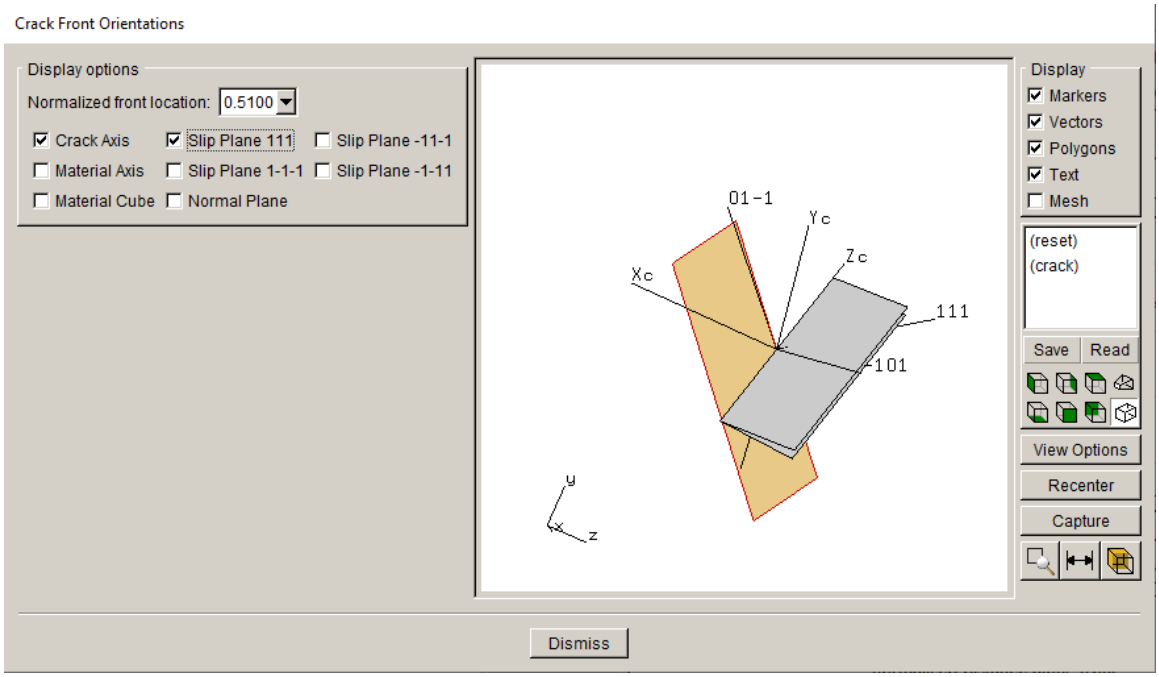


Figure 12.1.3 View Orientations dialog.

12.2 Resolved Slip System Ks Along a Path

The **Resolved Slip System Ks Along a Path** menu item allows one to view the SIFs along a path through the crack fronts. The dialog, Fig 12.2.1, is slightly different from the standard dialog in Section 6.9. As for the Resolved SIFs dialog (see Fig 12.1.2), the user can select the slip plane and plot K_{RSS}, K_{RNS} and K_{RSS}/K_{RNS} along the path.

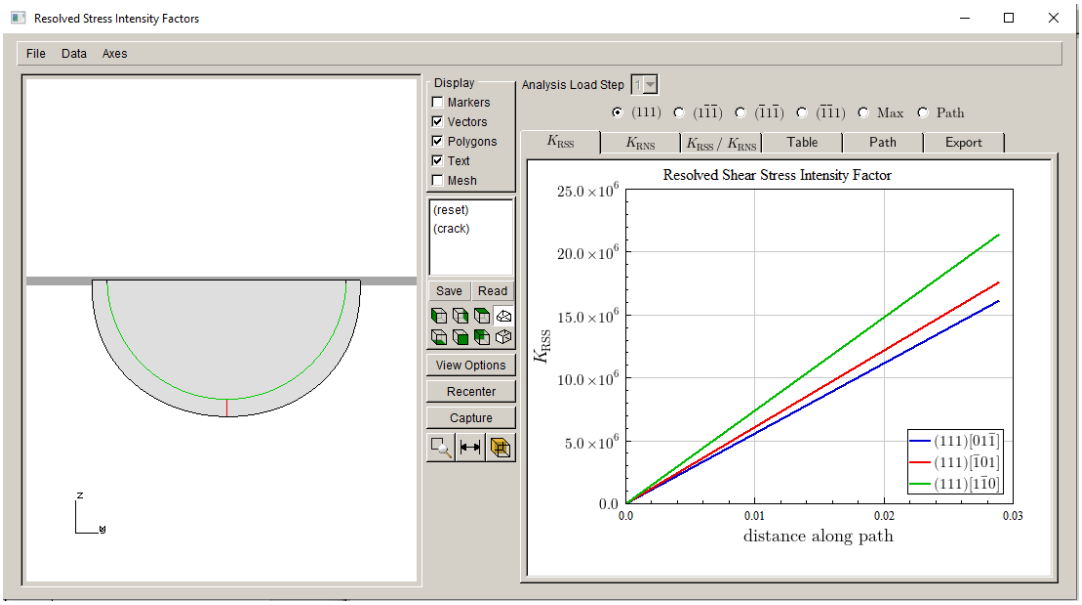


Figure 12.2.1 Resolved SIFs along a path dialog.

12.3 Resolved Max Shear Direction Ks

To be added….

12.4 Resolved Max Shear Direction Ks Along a Path

To be added….

12.5 View Crystal Orientations

The **View Crystal Orientations** menu item allows one to visualize the crystal orientation, Fig 12.5.1, relative to the model and the crack. The slip planes can be displayed and moved around in the model using the options on the left side.

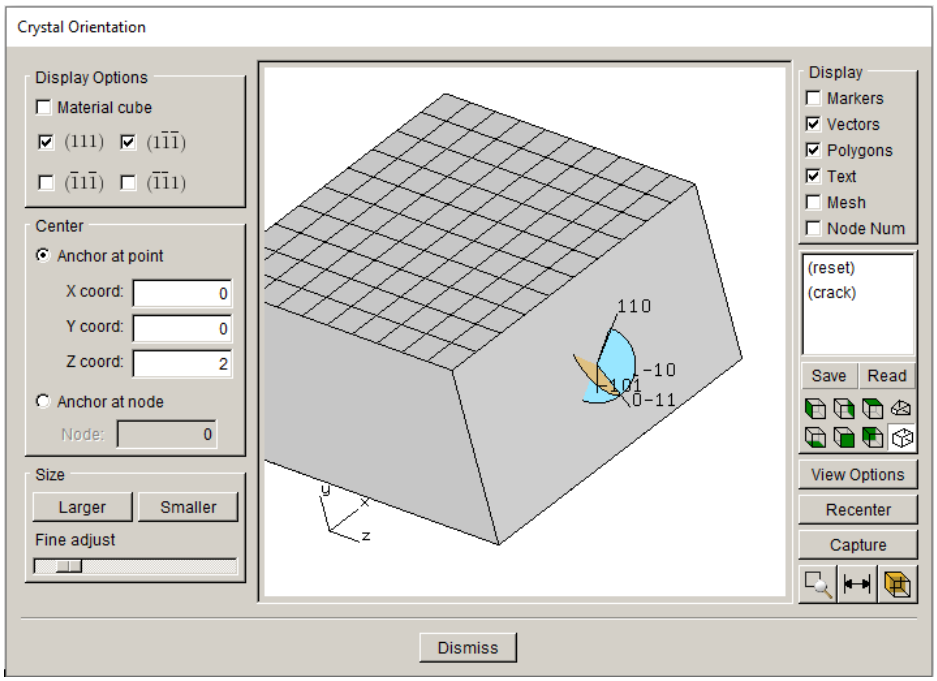


Figure 12.5.1 Crystal Orientation dialog.

13. Electrical Menu Wizards and Dialog Boxes

13.1 Electrostatic Energy Release Rates

To be added….

14. Advanced Menu Wizards and Dialog Boxes

The wizards and dialog boxes for the **Advanced** menu options are described in this section.

14.1 Edges Wizard

The **Edge Extraction** dialog, Fig 14.1.1, allows the user to control the geometry that is extracted from the finite element facets. The Angle Threshold can be adjusted to increase or decrease the number of surfaces by blending facets together.

The left side image in Fig 14.1.1 shows a curved surface that is treated as a single surface based on the default angle threshold. If the angle is increased from 151 to 166, the surface appears as

shown on the right. In most cases, the default angle is good. However, for coarse meshes and doubly curved surfaces, it is sometimes helpful to break the surface into more patches.

Section 4.4 of the User's Guide describes a case where the edge wizard is used to improve the geometry of an airfoil model.

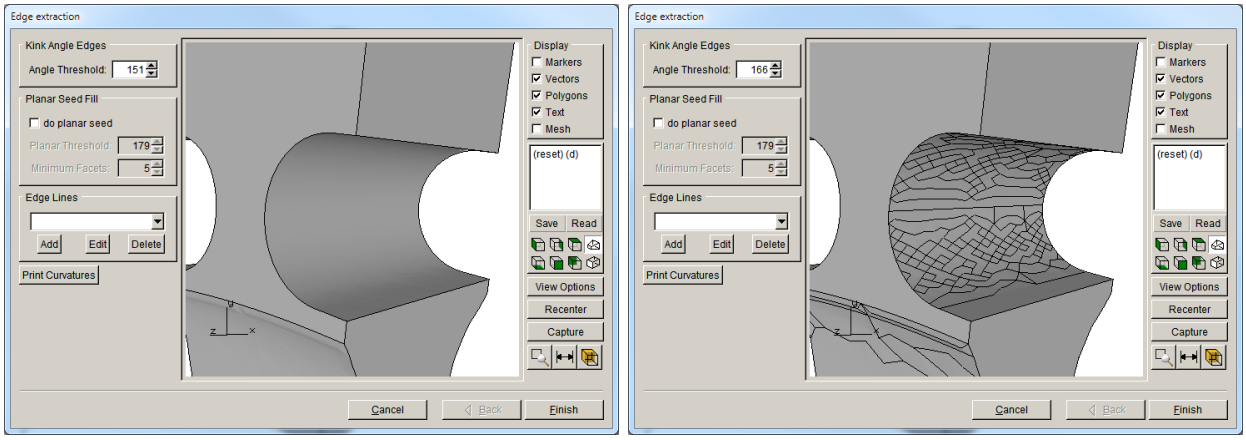


Figure 14.1.1 Edge extraction dialog; angle threshold increased.

The Planar Seed Fill option uses a different algorithm to break the surface patches, Fig 14.1.2. The Planar Threshold and Minimum Facets can be used to adjust the algorithm results. The results will vary depending on the underlying model mesh.

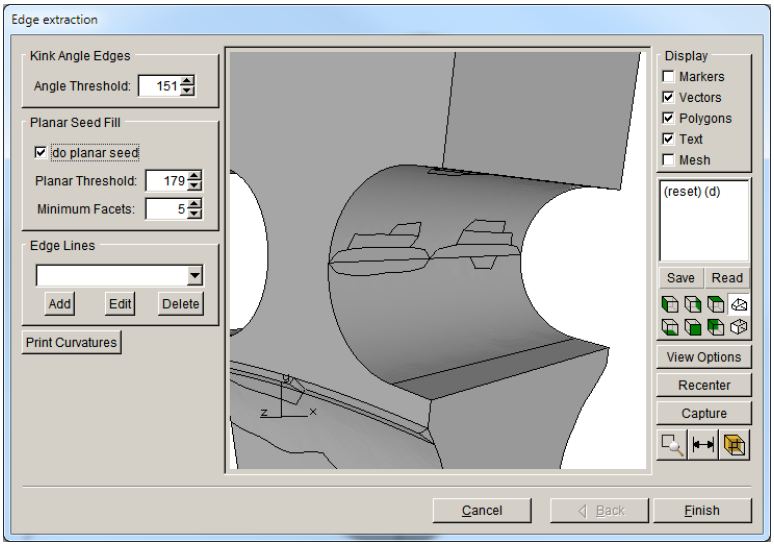


Figure 14.1.2 Edge extraction dialog with do planar seed.

For cylindrical surfaces such as a hole in a plate, the user can add edges to break up the surface. By selecting the **Add** button for **Edge Lines** in Fig 14.1.1, the dialog in Fig 14.1.3 is displayed to allow the user to define edges, which are used to divide the surface.

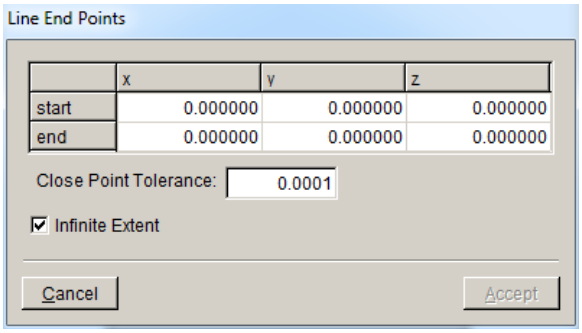


Figure 14.1.3 Add edge dialog.

14.2 Display COD Data

The **Display COD Data** menu item allows the user to view and save crack displacement data, Fig 14.2.2. This dialog displays the crack opening (COD), sliding (CSD) and tearing (CTD) displacements along with the corresponding SIFs (K_I, K_{II} and K_{III}).

The three menu items: **File**, **Data** and **Axes** are the same as for the SIF plot dialog (see Section 6.4). As with the SIF plot dialog, the data can be exported to a file, Fig 14.2.2. The crack displacements, along with the locations where displacements are computed, can be exported.

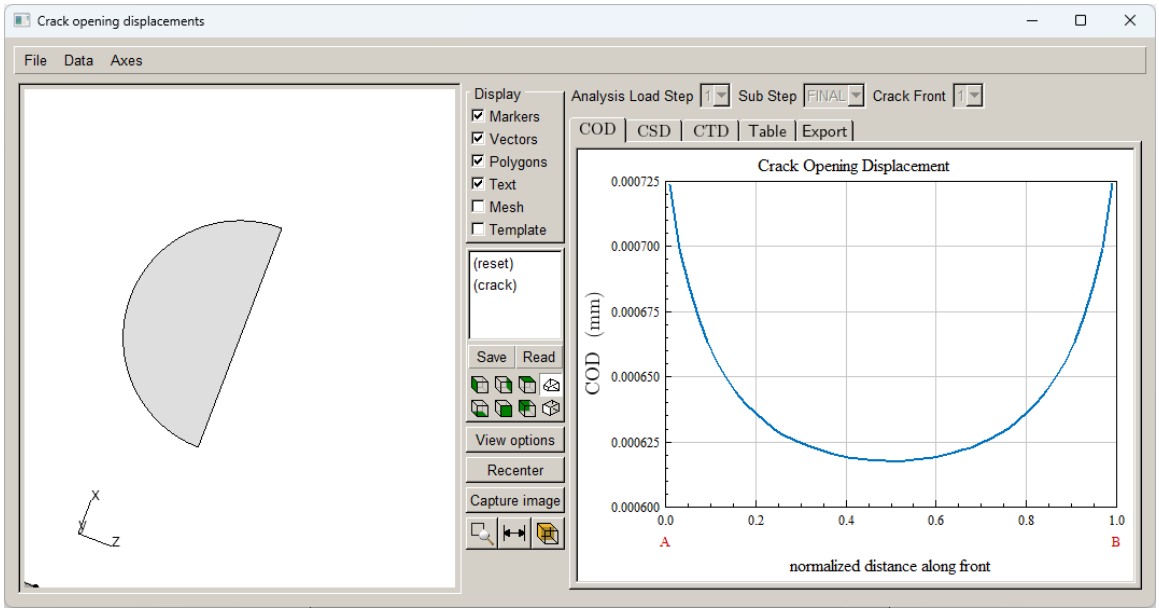


Figure 14.2.1 Write COD data – create file dialog.

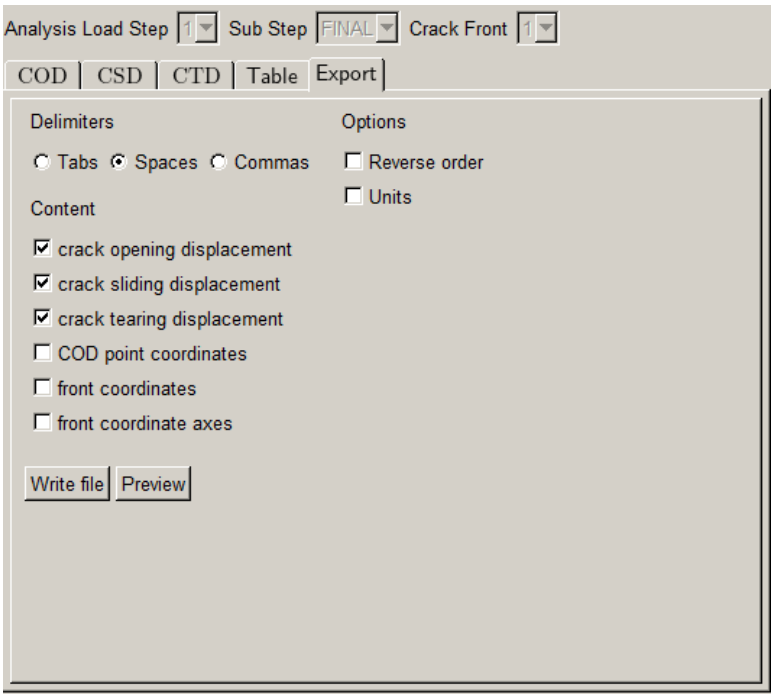


Figure 14.2.2 Write COD data – export tab.

14.3 Write Template Data

The **Save File As** dialog, Fig 14.3.1, is presented so that template data can be saved to a file. The data is saved to an ASCII text file with *.std* extension.

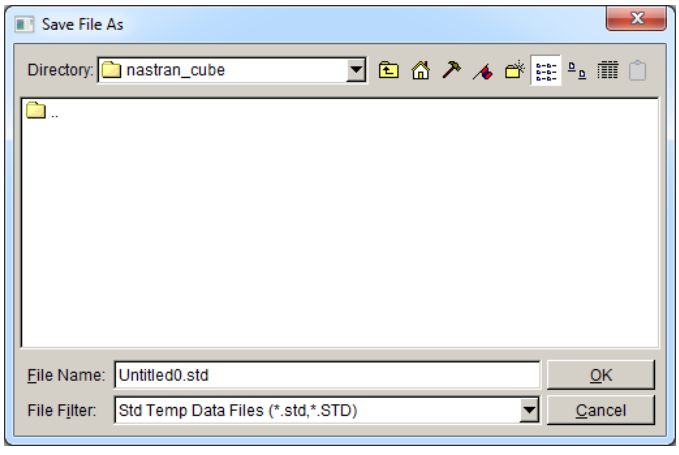


Figure 14.3.1 Write template data - Save File As dialog.

The file format for this file is briefly described here.

```
STD_TEMPLATE_DATA
{
VERSION: 1
NUM_FRONTS: 1
CRACK_FRONT: 0 49 25 OPEN
 CORNER_SET: 0              # corner node ids of template element ring
   11978 12907 11421 12440 11888 6756 7686 7222 8147 5762
   6227 10350 10816 973 1436 4673 5142 8373 8835
 ....
 SIDE_SET: 0                # side node ids
   6938 3386 6470 14975 6003 2013 616 1081 3384
   2479 2916 6383 2450 7784 6385 6854 9277 8246
   8815 4877 8353 3061 1198 1662 7643 3527 7180
   13551 6713 14015 14362 6338 13898 4288
 .....
NODE_COORDS: 1781       # list of all node ids and coordinates
 7321          -0.52397588          -0.01178516           9.9308077
 11252          -0.47625961       1.0479856e-014           9.7249057
....
NODE_DISP: 1781            # list of all node ids and displacements
 7321  2.34113830186791e-005   0.000939063262674639  -0.000290174323814175
 11252  2.24602369565477e-005   0.000940419013229567    -0.0002799458448084
....
}
```

14.4 View/Write Growth History

The **View/Write Growth History** dialog, Fig 14.4.1, allows a user to view and write the SIF history data. A list of the crack growth steps is provided, and for each growth step, the data for each crack front can be displayed. Use the **Write file** button to save a *.fcg* file.

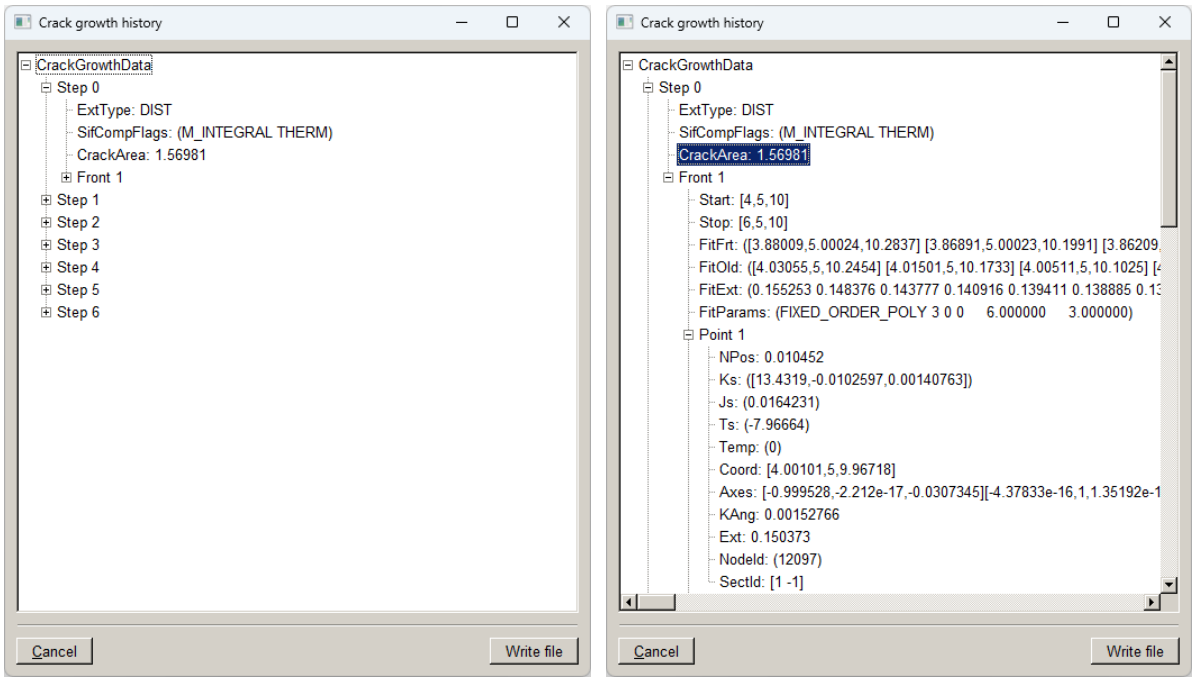


Figure 14.4.1 Create Growth History dialog.

14.5 Merge Growth History Files

The **Merge Growth History Files** dialog**, Fig 14.5.1,** allows one to combine multiple .fcg files into a single *.fcg* file.

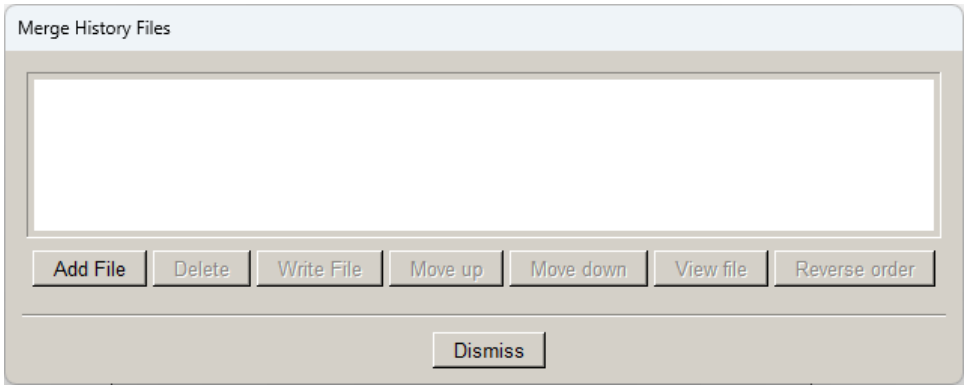


Figure 14.5.1 Merge Growth History Files dialog.

Use the **Add File** button to select a *.fcg* file. Select a file name and use the **View File** button to display the SIF history data (see Section 14.4).

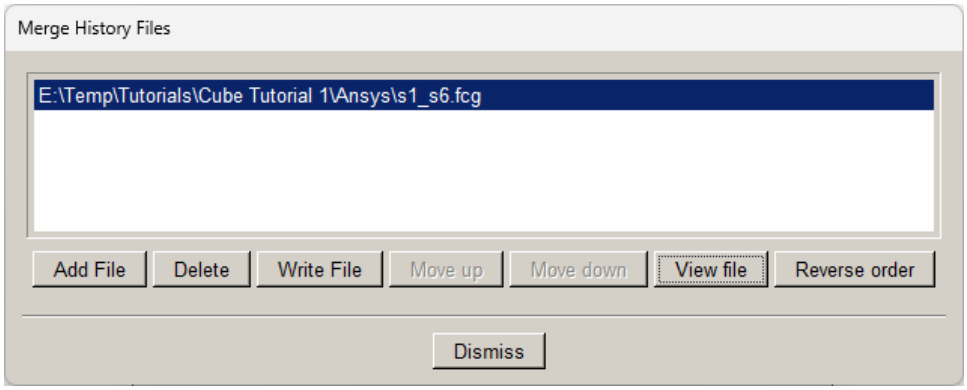


Figure 14.5.2 Merge Growth History Files dialog – with *.fcg* file selected.

14.6 Read User Extensions

The Read User Extensions menu item allows one to import Python functions into FRANC3D for use during crack growth. A user can define their own rules for growing a crack with Python functions. This is more fully described in the Command Language & Python Extensions reference.

Note that the PATH and PYTHONPATH environment variables will need to be set to have all the Python capabilities. Python 3.6 and later versions should work.

A dialog is displayed allowing one to choose the Python (*.py*) file containing the functions, Fig 14.6.1. After selecting the file and clicking **Accept**, FRANC3D reads the *.py* file and displays a list of the "known" functions that were read, Fig 14.6.2.

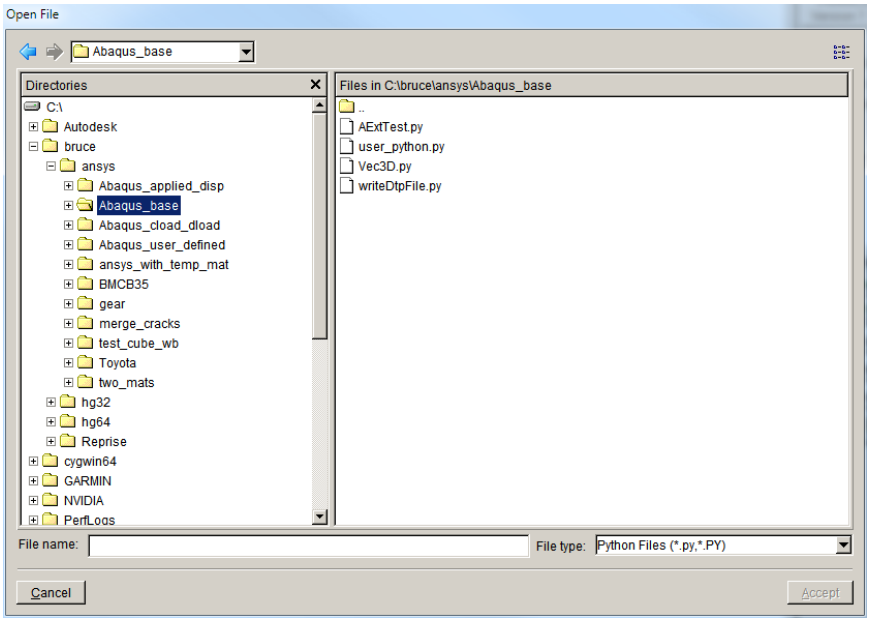


Figure 14.6.1 User extensions open file dialog.

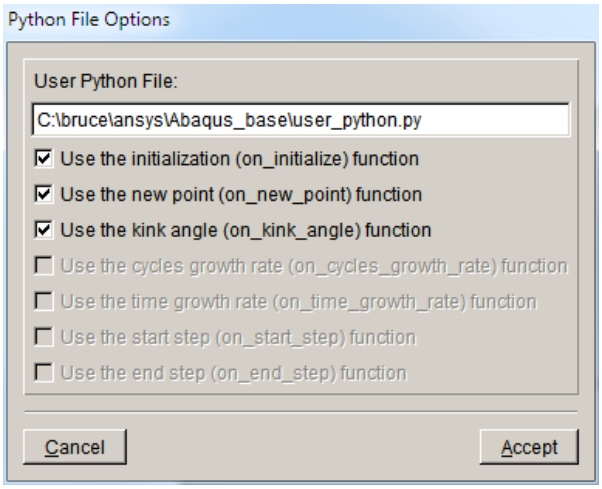


Figure 14.6.2 User Python extension functions.

See the Command & Python reference for a more complete description of the Python extension.

14.7 Edit Retained Nodes

The Edit Retained Nodes menu item allows one to revise the retained nodes and facets if these are interfering with crack growth. A crack cannot be inserted/extended into a retained surface. Use of this dialog, Fig 14.7.1, is described in Section 5.6 of the User's Guide.

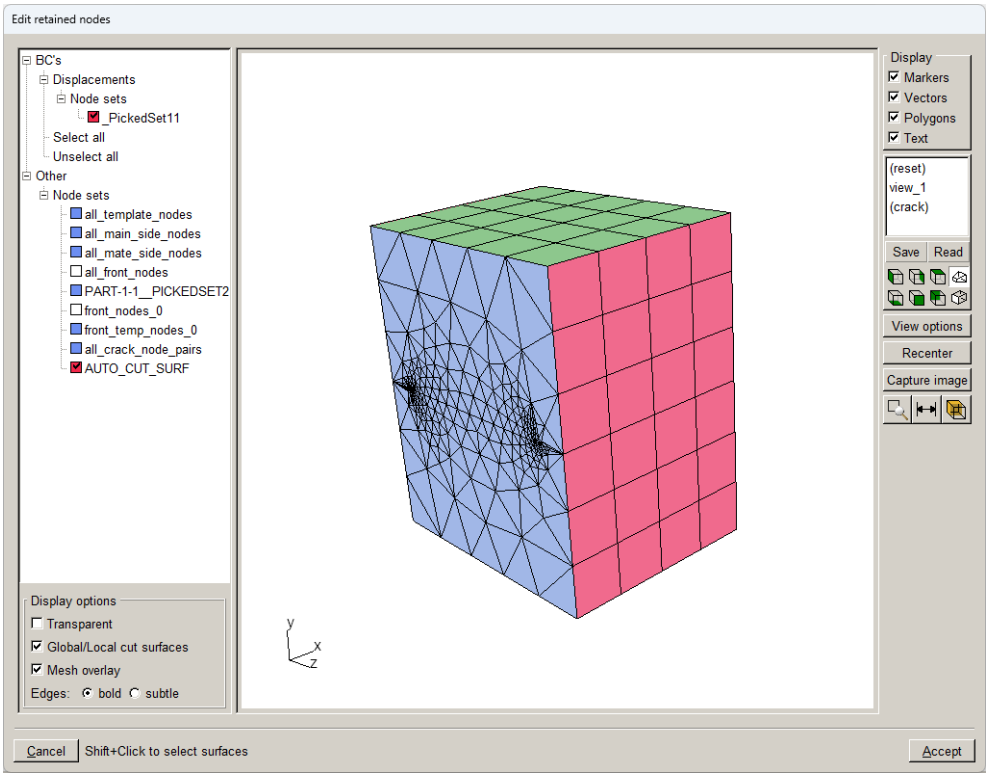


Figure 14.7.1 Edit Retained Nodes dialog.

14.8 Contour Integral Data

The Contour Integral Data menu item allows one to import and plot the analysis contour integral (CI) data. ABAQUS and ANSYS contour integral data is supported. The ABAQUS *.dat* file or the ANSYS *.out* file can be selected; these files contain the CI data in the following format for ABAQUS:

J-INTEGRAL ESTIMATES

CRACK NUMBER	CRACKFRONT NODE SET	CONTOURS 1	2	3	4	5	6
1	-10-	1.4192E-02	1.5211E-02	1.5659E-02	1.6162E-02	1.6702E-02	1.7303E-02
1	-11-	1.5466E-02	1.6031E-02	1.6078E-02	1.6093E-02	1.6102E-02	1.6115E-02

and for ANSYS:

```
***** POST1 J-INTEGRAL   RESULT LISTING *****

 CrackID =   1
  Crack Front Node =    5656
   Contour Values =   0.15010E-01  0.15471E-01  0.15495E-01  0.15504E-01
   Contour Values =   0.15512E-01  0.15521E-01
  Crack Front Node =   28626
   Contour Values =   0.15631E-01  0.16093E-01  0.16144E-01  0.16167E-01
   Contour Values =   0.16181E-01  0.16195E-01
```

The CI data is displayed in a dialog, Fig 14.8.1. The data can be displayed either Along the Front or By Ring. The number of contour rings is based on the crack front template mesh. The template elements should be collapsed bricks, and more than three rings of elements are usually required to get convergence.

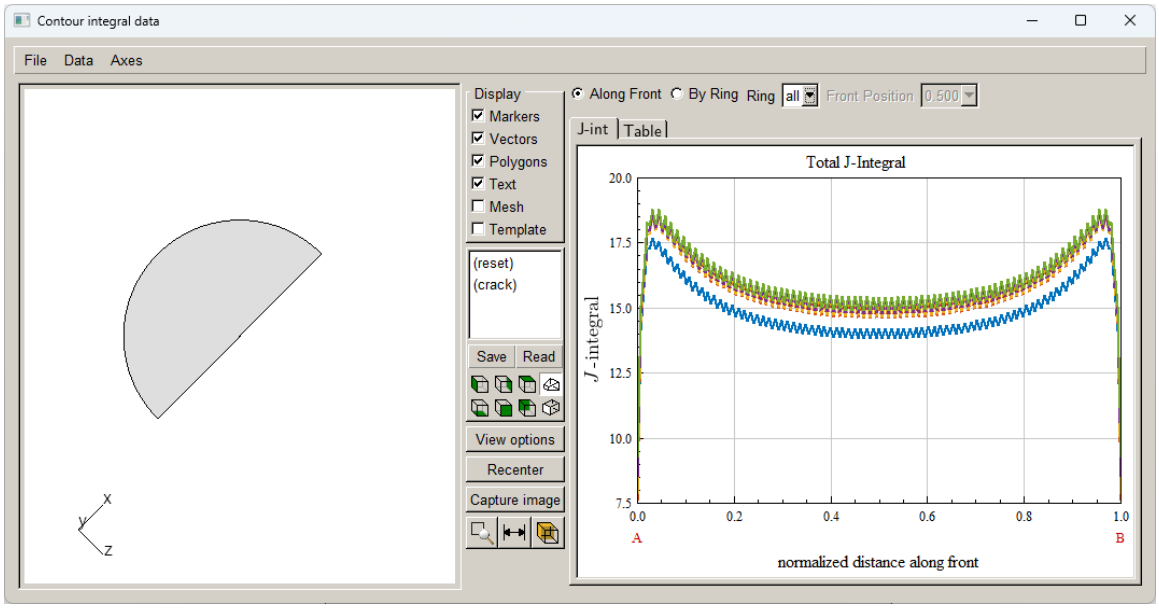


Figure 14.8.1 Contour Integral Data dialog.

Appendix A: XY Plot Label Syntax

```
// Layout a label like an equation
//
// Items:
//    |Seq(<item1>,<item2>, ... ,<itemn>)   - sequence of items
//    |Sub(<base>,<subscript>)              - item with a subscript
//    |Sup(<base>,<superscript>)            - item with a superscript
//    |Ss(<base>,<superscript>,<subscript>) - item with a both sub and superscript
//    |Sqrt(<item>)                        - square root
//
// Symbols and Operators:
//    $Eq      - equals
//    $Gt      - greater than
//    $Lt      - less than
//    $Ge      - equal or greater than
//    $Le      - equal or less then
//    $Plus    - plus sign
//    $Minus   - minus sign
//    $Mult    - multiplication sign (cross)
//    $Div     - division sign (bar and two dots)
//    $Del     - partial derivative
//    $About   - two wavy lines
//    $alpha   - lower case alpha
//    $Alpha   - upper case Alpha
//    $beta    - lower case beta
//    $Beta    - upper case beta
//    $delta   - lower case delta
//    $Delta   - upper case delta
//    $(       - left paren
//    $)       - right paren


//
// Render a math equation using a subset of the Latex equation grammar
//
// Native symbols
//     + - = ! / ( ) [ ] < > | ' :
//
// Escaped symbols
//     \Alpha   \alpha
//     \Beta    \beta
//     \Delta   \delta
//     \Eta     \eta
//     \Gamma   \gamma
```

```
//    \Sigma   \sigma
//    \Theta   \theta
//    \geq
//    \leq
//    \times
//    \div
//    \partial
//    \sim
//
// Items:
//    \sum{}
//    \sqrt{}
//    \frac{}{}
//    \lines{}{}{}
//
// Other:
//    "_" denotes subscript
//    "^" denotes superscript
//
// Approximate Grammar:
//
// <equation> := <sequence>
//
// <sequence> := <seq_item> | <sequence> <seq_item>
//
// <seq_item> := <atom> | <group> | <sub_item>
//               <sup_item> | <both_item> | <sum_item>
//               <sqrt_item> | <frac_item> | <lines_item>
//
// <atom> ::= <character> | <symbol> | <digit>
//
// <group> ::= {<sequence>}
//
// <modifier> ::= <atom> | <group>
//
// <sub_part>  ::= _<modifier>
// <sup_part>  ::= ^<modifier>
// <both_part> ::= _<modifier>^<modifier>
//
// <sub_item>  ::= <eqn_item><sub_part>
// <sup_item>  ::= <eqn_item><sup_part>
// <both_item> ::= <eqn_item><both_part>
//
// <sum_item> ::= \sum <eqn_item> |
//                \sum<sub_part> <eqn_item> |
//                \sum<sup_part> <eqn_item> |
```

```
//                \sum<both_part> <eqn_item>
//
// <sqrt_item> ::= \sqrt<eqn_item>
//
// <frac_item> ::= \frac<group><group>
//
// <lines_item> ::= \lines {<group> <group> ... }
```