

FRANC3D

Menu & Dialog Reference

Version 2.6

2003

1 Introduction

This manual is a reference for the FRANC3D menus and associated dialog boxes. This document contains concise descriptions of each menu entry, in a form that is useful to someone already familiar with the concepts behind FRANC3D, such as solid modeling, the model hierarchy, or arbitrary region meshing, but needs to look up specifics about various menu commands. For introductory or tutorial information, please consult:

- (a) the FRANC3D Concepts & Users Guide,
- (b) the FRANC3D & OSM 3D Tutorial, or
- (c) the PhD Theses and technical papers referenced in the Concepts & Users Guide.

The first section of this document describes the menu system, and the interaction between the menu buttons and the FRANC3D system for interactively collecting data from the user. The second section describes windows and other features that remain on the FRANC3D screen throughout the FRANC3D interactive session. There are a variety of command boxes, text boxes, radio buttons, dialog boxes, and visualization windows that comprise the interactive FRANC3D system. Collectively, these items are referred to as widgets. The subsequent sections describe each menu cluster, giving specific information for each menu button.

2 Menu Format Description

Each section heading that corresponds to a menu is indicated by bold underlined text, as in **Example Menu**. The section heading corresponds to the menu label button that is shown in capital letters in each menu figure. Menu buttons are indicated in the text by bold letters while the menu labels are indicated by bold capital letters. Some menus have a section that describes concepts and keywords or phrases that are associated with the menu button actions. The format that describes each menu is as follows:

Example Menu

[short paragraph describing the menu]

concepts for this menu

keyword or phrase [description of important concepts for this menu]

specific commands on this menu

text on menu button [description of command]

2.1 Menu System Commands and Prompts

Each menu button is unique and the action that follows the selection of any menu button varies. The types of responses are:

- invoke a collector to gather data interactively from the user (e.g. **Part-Thru Crack** asks the user to select the edges that form the crack)
- create a child menu (e.g. **Develop Model**)
- pop up a sub-list or sub-menu (e.g. **Linear/Quadratic/Cancel** from **Write BES File**)
- invoke a dialog box directly (e.g. **Active Subdivision Data**)
- direct action

The result of each menu command should be apparent from the information displayed on the screen. Informative or prompt messages often are written to the information text box that is initially located in the lower left corner of the screen.

The most common result of pressing a menu button is the creation of a child menu or the invocation of a collector. Collectors are used to gather information by making the user interactively select entities in the main visualization window. The process of collecting data will be described along with the menu description when appropriate. A collector is active whenever the following menu is on the screen, Figure 2.1a.

This menu appears in the lower right quadrant of the screen. Other buttons can also be present during a collector operation, for example Figure 2.1b, and they will be located close to these three buttons.



Figure 2.1. (a) buttons common to all collectors, and (b) specialized collector buttons.

RUBOUT

Rejects all data just collected and restarts the collection process.

QUIT

Rejects all data just collected and cancels the collection process and the command.

HELP

Currently not active; is intended to provide a text description of the collector.

Some menu buttons invoke child menus. Depending on the state of the current model, some of the child menu buttons may be inactive. If a menu button is inactive, it is displayed in a different color and will not produce the indicated action when pressed. All child menus are removed when the parent is removed (by selecting **return**).

Some menu buttons have sub-lists or sub-menus attached to them rather than separate child menus. Each entry in a sub-menu performs a unique action. Sub-lists are used to display a list of available items that can be selected to control other commands. All of the sub-lists and sub-menus have a cancel option that simply exits the sub-menu or sub-list without causing any changes or actions.

Dialog boxes provide a way of collecting arbitrary data from the user. For example, the dialog box in Figure 2.2 collects text, a floating-point value, and optional data based on toggle boxes. Dialog boxes generally are blocking, meaning that you must either select **Accept** or **Cancel** in the dialog box before you can continue with any other commands. These boxes can be moved around the desktop by pressing the right mouse button while the cursor is anywhere on the box and then dragging it. Dialog boxes are also described in the FRANC3D Concepts & Users Guide.

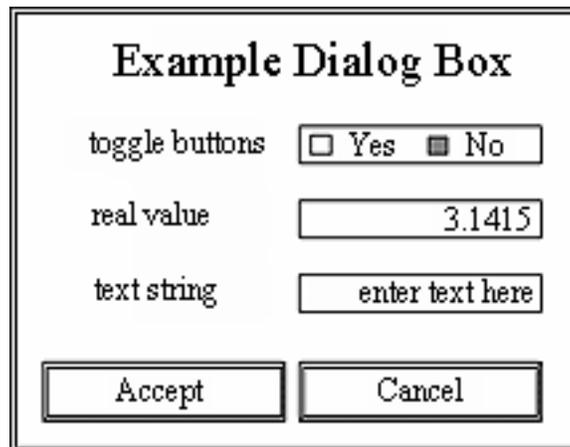


Figure 2.2. Example generic dialog box.

Dialog boxes are described as they are encountered within the menu commands. The entry fields, toggle options, and buttons are described immediately after the figure, i.e.:

toggle buttons

This toggle box allows the user to specify Yes or No.

3 Permanent Widgets

There are five sets of widgets that are always part of the FRANC3D screen, Figure 3.1:

- The main modeling window is placed near the middle or to the left side of the FRANC3D desktop. The model is displayed here and all interaction with the model is conducted through this window. This window can be resized and moved with the left and right mouse button respectively. The window has additional pull-down menus under its title bar that will be discussed in Section 3.1.
- The FRANC3D main menu, with commands to (a) read and write geometry files and (b) activate lower level menus is initially placed at the top left corner of the desktop. It can be dragged elsewhere using the right mouse button. This menu is discussed in Section 4.
- The view specification panel with buttons to rotate, zoom, pan and clip the current geometry are placed as a group at the upper right of the screen. They can be dragged elsewhere using the right mouse button. The function of all these buttons is discussed in Section 3.2.
- A title box is placed in the top right corner. Selecting the logo presents an acknowledgement along with copyright and version information
- The text box in the lower left corner is an information window. Warning and error messages as well as prompt messages are printed here. Additional information is often printed to the terminal window.

3.1 Main Modeling Window

The main modeling and visualization window is where the model is displayed and where interactive collection of the entities, which comprise the model, is performed. The window is shown in the Figure 3.2 with one of its pull-down menus exposed. The window has two ‘grip’ boxes at either end of the title bar at the top of the window. These are accessed by selecting them with the left mouse button. The one on the left displays the orientation of the global coordinate system. Click in this box to get a picture of how

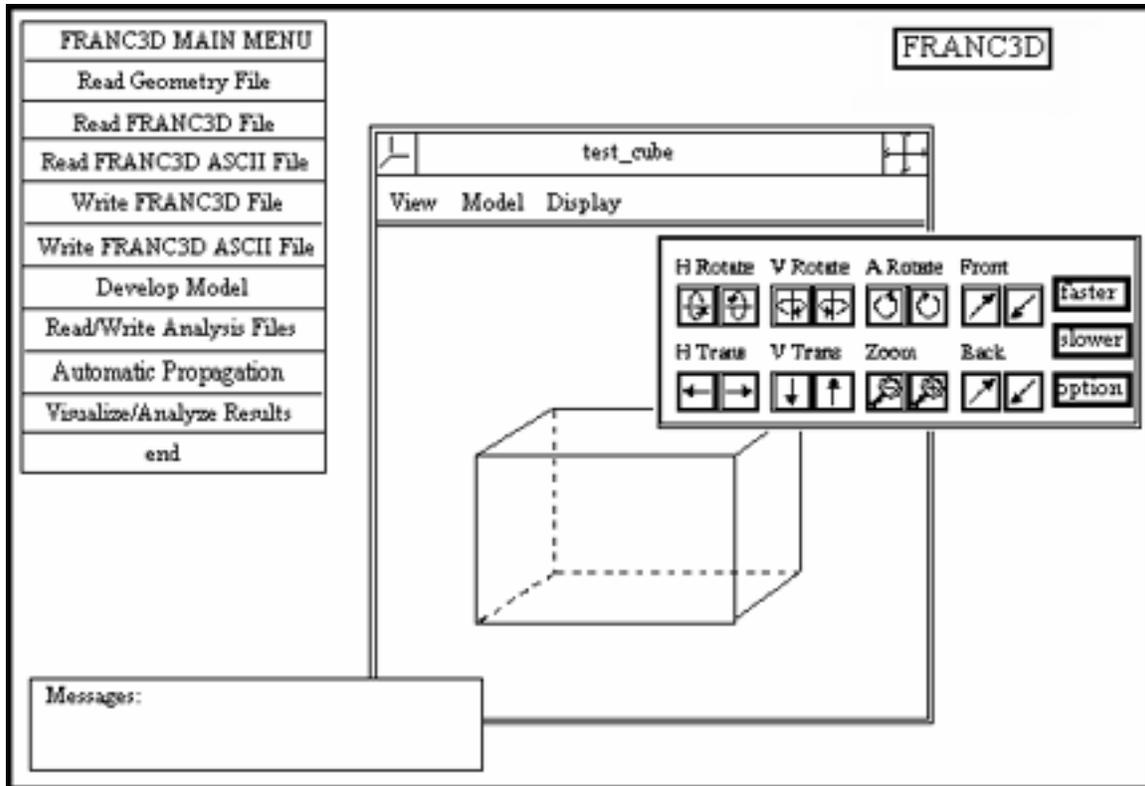


Figure 3.1 FRANC3D modeling environment.

the axes are oriented. The box on the right is used for resizing the window. To resize, select the box with the left mouse button, and, while keeping the mouse button depressed, drag the mouse until the window is of the desired size. The entire window can be dragged by selecting the title bar with the right mouse button, and, while keeping the button depressed, dragging the window. Selecting the title bar with the middle mouse button brings the window forward to the front of the screen. The operations for dragging and bringing the window to the foreground are the same for all the widgets. The label bar contains the current model name; Modeling Window is the default name.

3.1.1 Main Modeling Window Pull-Down Menus

There are three pull-down menus in the main modeling window accessed by buttons on the left side of the pull-down menu bar. These menus provide control of the display of the model in the modeling window and the ability to store the current camera position (view of the model in the window) to a file.

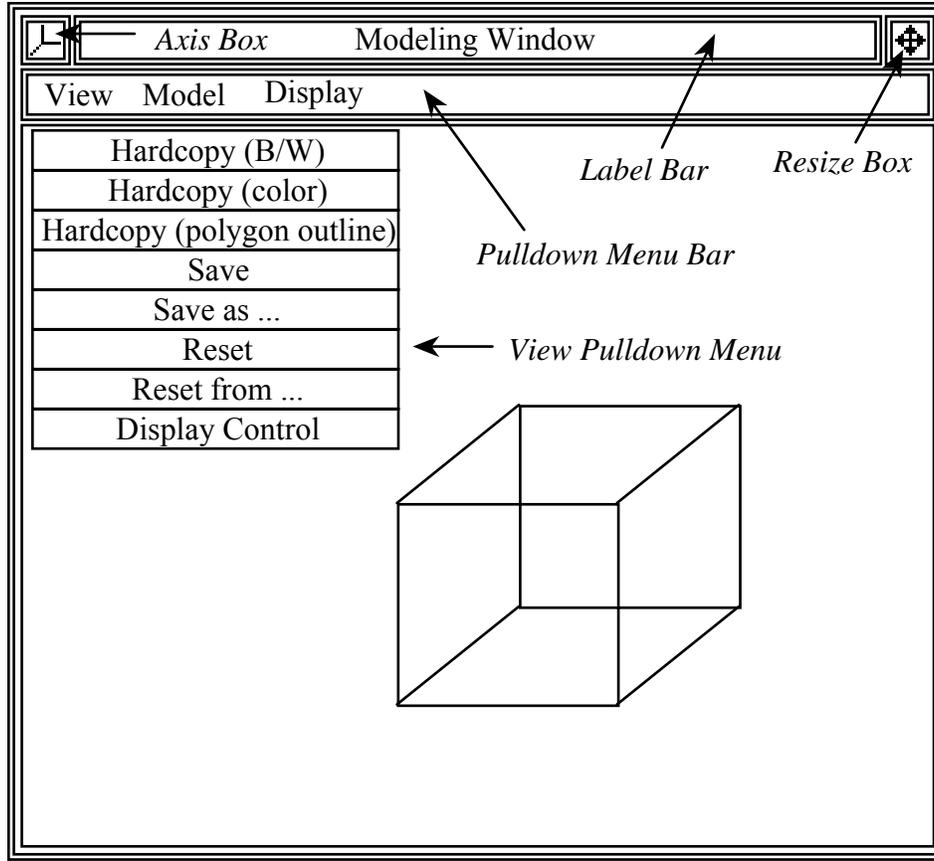


Figure 3.2. FRANC3D main modeling window.

3.1.1.1 View Pull-Down Menu

The **View** pull-down menu is shown in Figure 3.3.

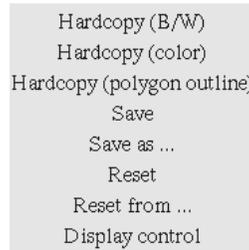


Figure 3.3. The **View** pull-down menu.

This menu presents choices for dumping the current display to a postscript file, for saving the current camera position, and for selecting the type of geometric information to be displayed.

Specific Commands on this menu are:

Hardcopy (B/W)

Generates a black and white postscript file of the current view of the model. A file selector box is presented with a list of all the .ps files. You can overwrite an existing file or create a new file by entering the name in the Filename text field. The .ps extension will be added for you if you do not type it. These files may be printed directly to a postscript printer.

Hardcopy (color)

Generates a color postscript file of the current view. See **Hardcopy (B/W)** for details.

Hardcopy (polygon)

Generates a postscript file of the current view with surfaces shaded and hidden lines behind the shaded surfaces not shown. See **Hardcopy (B/W)** for details.

Save

Save the current camera position as the default.

Save as...

Save the current camera position to a file. A file selector box is presented with a list of all the .cam files. You can overwrite an existing file or create a new file by entering the name in the Filename text field. The .cam extension will be added for you if you do not type it.

Reset

Reset the view based on the default camera position stored by **Save**.

Reset from...

Reset the view based on the camera position from a file. A file selector box is presented with a list of all the .cam files. You can select a file by double clicking on the file name or by highlighting the file name and then selecting **Accept**.

Display Control

Allows the user to turn on and off the display of vectors and polygons.

3.1.1.2 Model Pull-Down Menu

In the **Model** pull-down menu, the user is presented with a dialog box, Figure 3.4, with choices for showing different hierarchical models in the current display. Only one of the layers of the model hierarchy can be displayed at any time. Hierarchy levels consist of geometry, volume decomposition, face decomposition, edge decomposition, and mesh. See the FRANC3D Concepts & Users Guide for a description of the five model levels.

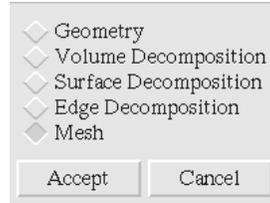


Figure 3.4. The **Model** pull-down menu.

3.1.1.3 Display Pull-down Menu

In the **Display** pull-down menu, the user is presented with choices for showing different topological features in the current display, Figure 3.5.



Figure 3.5. The **Display** pull-down menu.

Specific Commands on this menu:

Features

A dialog box is presented, Figure 3.6, which allows the user to turn on and off the display of surfaces, edges, and vertices, including highlighting the crack surfaces, edges, and vertices. The volume mesh (edges of the volume elements) can be turned on or off via the last option of the dialog box.



Figure 3.6. The **Features** dialog box.

Redraw Screen

Redraws the current display and scales the drawing based on the current camera position.

Bigger Vertices

Increases the factor for redrawing the boxes that define points or vertices.

Smaller Vertices

Decreases the factor for redrawing the boxes that define points or vertices.

3.2 View Specification Buttons

The view of the object or the camera position in the main modeling window is controlled with the use of the buttons in the view control panel, Figure 3.7. These buttons allow the object to be rotated and translated. The camera can be zoomed in on a portion of the object. Front and back clipping planes can be used to eliminate portions of the object from the current view. The FRANC3D Concepts & Users manual describes the view specification process in more detail. A short summary is given here; the function behind each button is as follows:

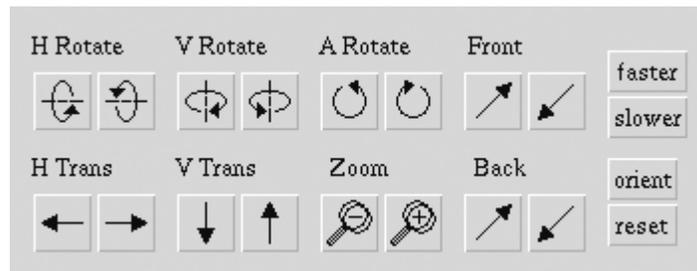


Figure 3.7. The view control panel.

H Rotate

Rotates the model about a horizontal axis. The left button rotates the model upwards and the right button rotates the model downwards.

V Rotate

Rotates the model about a vertical axis. The left button rotates the model to the left and the right button rotates the model to the right.

A Rotate

Rotates the model in a clockwise/counterclockwise direction (about an axis that extends out of the screen). The left button rotates the model counterclockwise and the right button rotates the model clockwise.

Front

Moves the front clipping (cutting) plane. The left button pushes the front plane towards the model; the right button pulls the plane away from the model.

H Trans

Translates the model in the horizontal direction. The left button moves the model to the left and the right button moves the model to the right.

V Trans

Translates the model in the vertical direction. The left button moves the model downward and the right button moves the model upward.

Zoom

The left button zooms out (pushes the model away making it appear smaller) and the right button zooms in.

Back

Moves the back clipping (cutting) plane. The left button pushes the back plane away from the model and the right button pulls the back plane towards the model.

faster/slower

These buttons control the speed of the actions commanded by the above buttons. The speed is changed each time one of these buttons is pressed.

option

This button presents a dialog box that allows the user to snap the model into a view along one of the axes, Figure 3.8.



Figure 3.8. The view option dialog box.

reset

This button resets the view of the model to the initial default.

The view can also be controlled with the mouse along with the Shift and Ctrl buttons. Holding the right mouse button down and dragging it allows the user to translate the model in the drag direction. Holding the Shift button down while holding the right mouse button down and dragging the mouse allows the user to rotate the model. Holding the Ctrl button down while holding down the right mouse button and dragging the mouse allows the user to zoom in and out. Holding the Shift button down and holding the middle

mouse button down moves the front cutting plane forward and backward. Holding the Ctrl button down and holding the middle mouse button down moves the back cutting plane forward and backward.

4 Highest Level Menus

The **FRANC3D Main Menu**, Figure 4.1, and its immediate children are the highest level menus, and these are described in this section. The **Develop Model**, **Read/Write Analysis Files**, **Automatic Propagation**, and **Visualize/Analyze Results** menu options have many child menus. Thus, each of these menus is described in a separate section, Sections 5-8.

4.1 FRANC3D Main Menu

This **FRANC3D MAIN MENU**, Figure 4.1, is displayed in the upper left corner of the screen when FRANC3D is started, and remains there throughout the session. The commands on this menu support reading and writing of files, and activation of other menus for manipulating the model.



Figure 4.1. The **FRANC3D MAIN MENU**.

Concepts for this menu:

Geometry file formats - FRANC3D can read model data from three types of file formats: ascii geometry data in .dat files, binary data in .fys files, and ascii versions of the binary format in .afys files. FRANC3D writes files only in .fys and .afys formats. The .dat file format is used for entering data from external sources; FRANC3D cannot output this format. Additional details about these files are found in the Geometry File Format section of the FRANC3D Concepts & Users Guide.

Specific Commands on this menu:**Read Geometry File**

Reads geometry stored as an ascii .dat file. See the Geometry File Format section of the FRANC3D Concepts & Users manual for detailed information of the .dat file format.

- FRANC3D displays a list of all .dat files found in the current directory inside of a file selector box.
- The user selects the .dat file to be read by either double clicking on the file name with the left mouse button or by highlighting the file name (single click with the left mouse button) and selecting **Accept**.
- The current model is deleted.
- The new geometry is read into the database and displayed in the modeling window.

Read FRANC3D File

Reads geometry stored in a binary .fys file.

- FRANC3D displays a list of all .fys files found in the current directory in a file selector box.
- The user selects the .fys file to be read by either double clicking on the file name with the left mouse button or by highlighting the file name (single click with the left mouse button) and selecting **Accept**.
- The current model is removed.
- The file is read, the database filled, and the model displayed.

Read FRANC3D Ascii File

Identical to **Read FRANC3D File**, but uses ascii .afys files instead of .fys files.

Write FRANC3D File

Writes geometry in the .fys binary format.

- FRANC3D prompts for the name of the file to be written by presenting a file selector box. Overwrite a file by selecting an existing file name, or enter a new file name in the text entry field. The .fys extension will be added if you do not type it.
- All geometry and attributes are written to the named file.

Write FRANC3D Ascii File

Identical to "**Write FRANC3D File**", but writes an ascii .afys file instead of the binary .fys file.

Develop Model

Activates the **DEVELOP MODEL** menu, which displays commands to modify the geometry, discretize the model, and attach simulation attributes.

Read/Write Analysis Files

Activates the **READ/WRITE ANALYSIS FILE** menu, which displays commands to read and write files for various analysis programs.

Automatic Propagation

Activates the **AUTOMATIC PROPAGATION** menu, which displays commands to perform automatic propagation.

Visualize/Analyze Results

Activates the **VISUALIZE/ANALYZE RESULTS** menu, which displays commands to visualize and post-process results, including fracture analysis and propagation.

end

Exits this menu and terminates FRANC3D.

A flag is set whenever a change is made to the topology, geometry, or attributes. This flag is checked when the user selects **end**. If any changes have been made to the database since the file was last saved, a dialog box appears, Figure 4.2, asking if the user would like to save the file before ending the program. The user can choose which type of file to save – either ascii or binary. Selecting **Cancel** prevents the program from ending. The default is to exit without saving by selecting **Accept**.

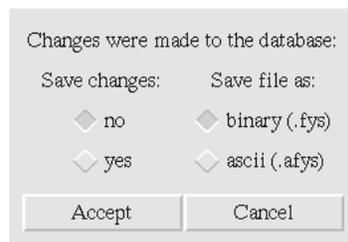


Figure 4.2. The **end** FRANC3D dialog box.

Save Changes

Allows the user the choice of saving the file before ending the program.

Save File as

Allows the user the choice of saving the file as a binary (.fys) or ascii (.afys) restart file if the previous option is set to yes.

4.2 Develop Model Menu

This **DEVELOP MODEL** menu, Figure 4.3, initiates many choices for modifying the geometry, discretizing the model to create a mesh, and attaching simulation attributes such as material properties. Detailed description of all of these choices is presented in Section 5.

Specific Commands on this menu:

Modify Geometry

Activates the **MODIFY GEOMETRY** menu, which displays commands to (a) modify geometry and topology at the geometry level and (b) nucleate cracks; see Section 5.1.

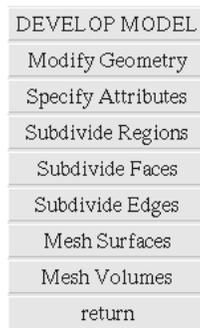


Figure 4.3. The **DEVELOP MODEL** menu.

Specify Attributes

Activates the **SPECIFY ATTRIBUTES** menu, which displays commands to attach attributes to geometric features; see Section 5.2.

Subdivide Regions

Activates the **SUBDIVIDE REGIONS** menu, which displays commands to insert surfaces that subdivide volumes; see Section 5.3.

Subdivide Faces

Activates the **SUBDIVIDE FACES** menu, which displays commands to insert edges that subdivide faces; see Section 5.4.

Subdivide Edges

Activates the **SUBDIVIDE EDGES** menu, which displays commands to subdivide edges into line segments prior to meshing; see Section 5.5.

Mesh Surfaces

Activates the **MESH SURFACES** menu, which displays commands to construct surface meshes; see Section 5.6.

Mesh Volumes

Activates the **MESH VOLUMES** menu, which displays commands to construct volume meshes; see Section 5.7.

return

Exits the menu.

4.3 Read/Write Analysis Files Menu

Selection of the **Read/Write Analysis Files** button from the main menu pops up the **READ/WRITE ANALYSIS FILE** menu, Figure 4.4. This menu contains commands for reading and writing data files formatted for or by finite and boundary element programs.

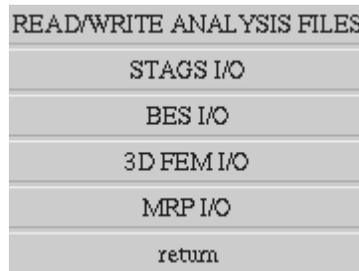


Figure 4.4. The **READ/WRITE ANALYSIS FILE** menu.

Specific Commands on this menu:**STAGS I/O**

Activates the **STAGS I/O** menu, which displays commands to read and write STAGS input and results files; see Section 6.1.

BES I/O

Activates the **BES I/O** menu, which displays commands to read and write BES input and results files; see Section 6.2.

3D FEM I/O

Activates the **3D FEM I/O** menu, which displays commands to read and write input and results files for the ANSYS finite element programs; see Section 6.3.

MRP I/O

Activates the **MRP I/O** menu, which displays commands to read and write MRP files; see Section 6.4.

return

Exits the menu.

4.4 Automatic Propagation Menu

The **AUTOMATIC PROPAGATION** menu, Figure 4.5, contains commands to perform automatic crack propagation.

Specific Commands on this menu:

Select Crack Growth Model

Presents a dialog box for defining the crack growth parameters for automatic crack growth simulations, see Section 7.1.

Automated Analyses Using BES

Starts the automatic crack growth simulations using BES as the analysis program, see Section 7.2.

Propagate All Cracks

Propagates automatically all the cracks in the model, see Section 7.3.

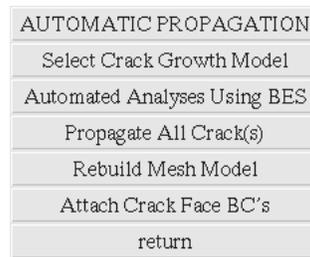


Figure 4.5. The **AUTOMATIC PROPAGATION** menu.

Rebuild Mesh Model

Remeshes the model after automatic propagation, see Section 7.4.

Attach Crack Face BC's

Attaches boundary conditions to the new crack surfaces created during propagation, see Section 7.5.

return

Exits the menu.

4.5 Visualize/Analyze Results Menu

The **VISUALIZE/ANALYZE RESULTS** menu, Figure 4.6, contains commands to post-process analysis results including displaying the deformed structure, displaying color contours of stresses, displaying line plots, and analyzing and propagating fractures.



Figure 4.6. The **VISUALIZE/ANALYZE RESULTS** menu.

Specific Commands on this menu:

Deformation & Contour

Display deformed shapes and color contours of analysis results, see Section 8.1.

Surface Line Plot

Display line plots of analysis results, see Section 8.2.

Point Information

Display point information on the terminal window, see Section 8.3

3D Fracture Analysis

Invoke the **FRACTURE ANALYSIS** menu for determining stress intensity factors, propagating fractures, and predicting fatigue life, see Sections 8.4 and 8.5.

Fracture Initiation

Compute the orientation and location for an initial crack, see Section 8.6.

Crack Front Node Position

Determine crack front nodes.

T-STRESS

Compute T-stress.

Write Data Explorer File

Writes the DataExplorer file.

return

Exits the menu.

5 Develop Model

As mentioned in Section 4.2, selecting the **Develop Model** button from the FRANC3D main menu opens many levels of actions. These are all described in this section.

5.1 The Modify Geometry Menu

The first button on the **DEVELOP MODEL** menu, Figure 4.3, pops up the **MODIFY GEOMETRY** menu, Figure 5.1. This menu contains commands to modify the primary geometry model, including crack nucleation.



Figure 5.1. The **MODIFY GEOMETRY** menu.

Specific Commands on this menu:

Add Face

Adds a face at the geometry level. A sub-menu (select Vertices for interior face / select Edges for interior face / select Edges for exterior face / cancel) is presented allowing the user to select either pre-defined edges or vertices. Three face types can be created: planar polygons, quadrilateral B-splines, or tri-cubic bezier triangles.

Select Vertices for interior face:

The user is prompted to collect the vertices along the boundary of the proposed face. The following conditions are applied to the selected vertices:

- If two vertices on a single face are not joined by an edge, an edge is created between the two points. This excludes wireframe edges. If two vertices are connected by a wireframe edge that is not on a face, then use the *Select Edges for interior face* option.
- The command will create either a planar polygon, quadrilateral bi-cubic B-spline, or tri-cubic bezier triangle depending on the geometry of the edges joining the boundary vertices and the number of edges bounding the surface.
- No face will be created if the proposed face violates topological conditions of the existing solid, for example if the face will cross another face.

Select Edges for interior face:

The user is prompted to select the edges that will form the boundary of the face. The following conditions apply:

- The edges must form a valid closed loop within a single region.
- The user has the option of specifying the type of face and the corner vertices of the face. Press the **Specify Corner Vertices** button before collecting the edges and a dialog box will appear, Figure 5.2, which will allow the user to specify the type of face and number of corner vertices. Consistency checks are done after the edges are collected to ensure that the specified face type is valid.
- The user has the option of specifying the region to which the face will be added before collecting edges. Press the **Specify Region** button before doing any collecting or the button will disappear. NOTE: a region can be specified by picking the region based on a unique integer identifier or by picking a point in the region. To pick a point in the region, first pick a point in the region, then rotate the model so that the pick line appears, and then pick another point along the first pick line so that the intersection point is in the desired region, or enter the coordinates of a point inside the region after pressing the **Key-In-Coordinates** button. Also, if there is only a single region, this region is automatically selected when the **Specify Region** button is pressed.
- By default, if more than four edges are specified, a b-spline surface is created with four corner vertices located at the four smallest angles between adjacent edges.
- Bezier surfaces are created if only three edges are chosen.
- Planar surfaces can be created if there are three or four straight-line edges selected that all lie in a single plane.

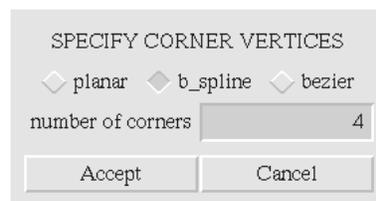


Figure 5.2. The **SPECIFY CORNER VERTICES** dialog box.

planar/b_spline/bezier

This option defines the type of face geometry.

number of corners

This parameter defines the number of corner vertices for the face geometry. It should be set to 4 for b-spline and 3 for bezier, but can be either 3 or 4 for planar faces.

Select Edges for exterior face:

The user is prompted to select the edges that will form the boundary of the face. This option is identical to Select Edges for interior face, except that the resulting face will be assigned as an exterior face of the model. The database maintains 'codes' that define the location of the faces in relation to the model. Exterior faces must have surface normals that point outward from the model interior. The edges should be collected in a clock-wise order when looking from the exterior of the model towards a point in the interior. *The user should verify that the surface normal points outwards*; this is done from the **Specify Attributes** menu using the **Coordinate Systems** menu.

Delete Face

Deletes a face from the geometry model. A sub-menu is presented (face and edges / face only / merge two faces / cancel) allowing the user to delete the face and all its edges, or just the face. Two faces can be merged into a single face by deleting one of the faces and reforming the remaining face. Both internal and external faces can be deleted; however, crack faces cannot be deleted directly.

face and edges:

The user is prompted to touch the faces to be deleted. Select **Finish** when all faces are collected. The conditions are:

- The face will not be deleted if there are dangling geometry edges on the face, or if the face has been split into two or more faces.
- Boundary edges will only be deleted if they are not used by any other face.
- All wireframe boundary edges will be deleted where wireframe edges exist after deleting the face.

face only:

The user is prompted to touch the faces to be deleted. Select **Finish** when all faces are collected. The conditions are:

- The face will not be deleted if there are dangling geometry edges on the face.
- If the face has been split into two or more faces, all faces that use the geometric descriptor are deleted. Bounding edges between these faces remain intact.

- All bounding edges and vertices remain.

merge two faces:

The user is prompted to touch the edge between the two faces that are to be merged. The conditions that apply are:

- The faces will be merged if there are only four bounding edges on both faces.
- The faces must both be B-spline faces.
- The edge between the faces is deleted and the vertices defining the end points of this edge are removed, leaving a single face with four edges.

Add Edge

Adds an edge to the geometry model. A sub-menu is presented (to Face / to Region / to Face/Region / cancel) allowing the user to add the edge to either a face or a region. The latter option of *to Face/Region* is used when the edge falls in a region, but intersects a face.

to Face:

The user is prompted to identify the face and end points of the line.

- The face can be chosen before adding the edge by pressing the **Select Face** button first and then selecting the face.
- The exact coordinates of the end points of the edge can be entered by pressing the **Key-In-Coordinates** button before selecting the point on the face.
- Select the end points of the edge using the mouse. Make sure both points lie on the same face.
- The type of edge depends on the surface geometry. If the surface is planar, the edge is a straight-line edge. If the surface is a b-spline or bezier, the edge will also be a b-spline.

to Region:

The user is prompted to identify the region and end points of the line.

- The region can be chosen before adding the edge by pressing the **Specify Region** button first and then selecting the region.
- The exact coordinates of the end points of the edge can be entered by pressing the **Key-In-Coordinates** button before selecting the points.
- Select the end points of the edge using the mouse. The end points can lie on existing vertices, edges, or faces, but are not required to do so. The **Pick Vertex as Point** button can be selected before picking the end point to force the picked point to be the nearest vertex.
- Edges can be created by reading points from a file by pressing the **Points From File** button. A list of all files in the directory is presented in a file selector box. Select the file that contains the points that define the edge.

The points can be arranged in free format with the x-y-z coordinates for each point on a separate line.

- The type of edge depends on the number of points. If only two points are used to define the edge, a straight-line edge is created. If more than two points are used, the edge will be a b-spline edge.

to Face/Region:

The user is prompted to identify the region and end points of the line. This option is identical to the *to Region* option, except that the edge geometry is extrapolated in order to intersect the edge with the model faces. The *to Region* option will add a vertex to a face if the end of the edge lies exactly on the face, but extrapolation is not performed.

Delete Edge

Removes an edge previously defined by the **Add Edge** command of this menu. A sub-menu is presented (from Face / from Region / cancel) allowing the user to delete the edge from either a face or a region. If the edge was added to a face, then the edge should be deleted from a face. If the edge was added to a region, then the edge should be deleted from a region. The user is prompted to touch the edges to be deleted. Select **Finish** when all the edges are collected. The conditions that apply are:

- An edge on a face will be deleted if there are only two faces adjacent to the edge and the faces use the same geometric descriptor. Any self-loop vertices that would be left on the face are deleted.
- If a wireframe edge is deleted from a region, the end vertices will remain in the region; these should be deleted also if they are not reused.

Add Vertex

Adds a vertex at the geometry level. A sub-menu is presented (to an Edge / to a Face / N vertex to a face / to a Region / cancel) allowing the user to add the vertex to either an edge, face, or region.

to an Edge:

The user is prompted to enter the position of the new vertex on an edge of the model. The vertex coordinates can be entered in a dialog box by pressing the **Key-In-Coordinates** button before selecting the point in the modeling window. After picking the point, a dialog box (Figure 5.3) is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.

to a Face:

The user is prompted to enter the position of the new vertex on a face of the model. The vertex coordinates can be entered in a dialog box by pressing the **Key-In-Coordinates** button before selecting the point in the modeling window. After picking the point, a dialog box (Figure 5.3) is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.



Figure 5.3. The **Key-In-Coordinates** dialog box.

N vertex to a Face:

The user is prompted to enter the position of a number of new vertices on a face of the model. The user is prompted to pick the face first. The user can then select the **Points From File** button and select a file containing the x y z coordinates of multiple points or simply pick the points on the face using the mouse.

to a Region:

The user is prompted to enter the position of a set of new vertices in a region of the model. The vertex coordinates can be entered in a dialog box by pressing the **Key-In-Coordinates** button. A dialog box (Figure 5.4) is displayed allowing the user to enter the set of point coordinates. The vertex coordinates can be entered from a file as well by pressing the **Points From File** button. Vertices are created for each valid point entry in the file; the x-y-z data is entered in free format with one point per line.

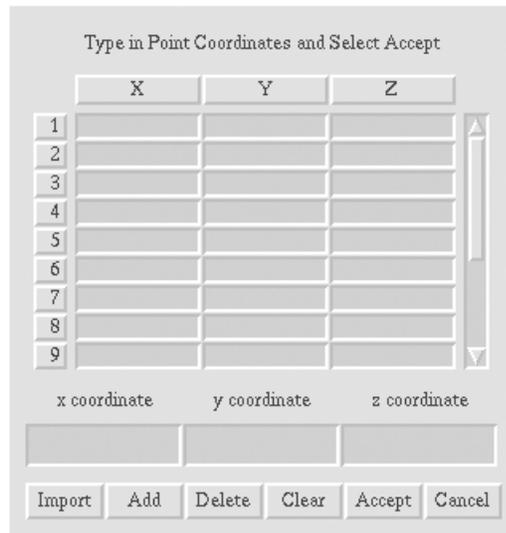


Figure 5.4. The **Type-In-Points** dialog box.

Import:

The **Import** button allows the user to read and import into the dialog points from a file. A file selector dialog box is presented and any valid xyz data will be imported.

Add:

The spreadsheet area shows the coordinates of points. Coordinates can be entered directly in the spreadsheet. Use the mouse or the Tab key to move from entry to entry. The coordinates can be entered in the three lower entry fields and then inserted into the spreadsheet by selecting the **Add** button. This is useful when one or two of the coordinates stay constant. Using these boxes, only the changing coordinates need to be entered before pressing the **Add** button.

Delete:

The **Delete** button deletes the spreadsheet entry that is active, i.e., where the cursor is located.

Clear:

The **Clear** button erases all data from the spreadsheet.

Accept:

The **Accept** button sends all the data currently in the spreadsheet (excluding data in the lower entry fields) to the calling routine. Consistency checks are made for complete x-y-z entries; any row that is not completely filled is discarded.

Cancel:

The **Cancel** button closes the dialog and returns no data to the calling routine.

Delete Vertex

Removes a vertex previously defined by the **Add Vertex** command of this menu. A sub-menu is presented (from an Edge / from a Face / from a Region / cancel) allowing the user to delete the vertex from either an edge, face, or region. If a vertex is added to an edge, it should be deleted from an edge, and the same holds true for vertices added to faces and regions. The user is prompted to pick the vertices to be deleted. Select **Finish** when all vertices are collected. Conditions that apply are:

- A vertex is deleted if there are two and only two edges using the vertex or if there are no edges using the vertex.
- If a vertex was added to an edge previously, the original edge is recovered when the vertex is deleted. If the two edges are distinct geometry edges, then they are merged into a single edge when the vertex is deleted. This may alter the geometry of both edges slightly.

Nucleate Crack

Activates the **NUCLEATE CRACK** menu which displays the crack types that can be created in the geometry; see Section 5.1.1.

Delete Crack

Deletes a crack from the geometry model. The user is prompted to touch a crack front or an open boundary (on the model surface) edge of the crack to be deleted. Shell cracks cannot be deleted. Note that deleting a crack can take several minutes for large models.

Geometric Info

Activates the **GEOMETRIC INFO** menu, which allows the user to obtain point coordinates and Cartesian distances between two points on the model surface; see Section 5.1.2.

return

Exits the menu.

5.1.1 Nucleate Crack Menu

The **NUCLEATE CRACK** menu, Figure 5.5, presents the choices for creating new cracks in the geometry.

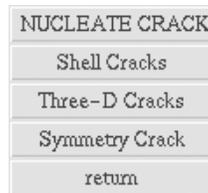


Figure 5.5. The **NUCLEATE CRACK** menu.

Specific Commands on this menu:**Shell Cracks**

Activates the **SHELL CRACK** menu, which displays commands for creating cracks in shell or plate structures; see Section 5.1.1.1.

Three-D Cracks

Activates the **THREE-D CRACK** menu, which displays commands for creating various types of cracks in solid model structures; see Section 5.1.1.2.

Symmetry Crack

Creates a symmetry crack using an existing geometry face and set of geometry edges on the solid model surface.

- Prompts for a face that is to be treated as the symmetry crack face.
- Prompts for a chain of edges that are treated as the free surface edges of the symmetry crack.
- Prompts for a chain of edges that are treated as the front of the symmetry crack.

5.1.1.1 The Shell Cracks Menu

The **SHELL CRACKS** menu, Figure 5.6, presents the choices for creating shell cracks in the geometry.

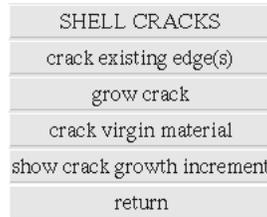


Figure 5.6. The **SHELL CRACKS** menu.

Specific Commands on this menu:

crack existing edge(s)

Creates a new shell crack by tearing existing geometry edges apart. The user is prompted to click on the edges that are to be turned into cracks.

grow crack

Propagates a shell crack along an existing geometry edge. The user is prompted to click on the edges adjacent to the crack tip to be propagated.

crack virgin material

Propagates a shell crack by adding an edge to a face and then tearing the edge. The user is prompted to select the crack tip vertex that is to be extended. A dialog box is presented to enter the propagation angle and the amount of extension.

show crack growth increment

Shows the direction of crack growth for a shell crack by drawing a red line from the crack tip. The user is prompted to select a crack tip. The crack tip coordinate system is shown along with a (yellow) line indicating the crack extension. Select **Finish** to remove these from the display.

5.1.1.2 The Three-D Cracks Menu

The **THREE-D CRACKS** menu, Figure 5.7, presents the commands for creating new three-dimensional cracks in the geometry, or propagating old cracks by tearing faces adjacent to existing crack fronts.

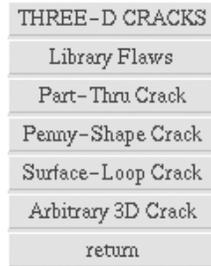


Figure 5.7. The **THREE-D CRACKS** menu.

Specific Commands on this menu:

Library Flaws

Presents a library of flaw shapes that can be sized, rotated, and translated, and then added to the model geometry. A sub-menu is presented, Figure 5.8:

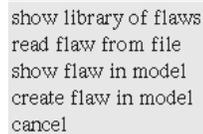


Figure 5.8. The **Library Flaw** sub-menu.

show library of flaws:

A dialog box is presented, Figure 5.9, with a library of flaw shapes and parameters for controlling the size, location and orientation of the flaw in the model. Select the flaw shape and assign values to the parameters. Select the **Calculate** button. The points that define the crack are determined and displayed as red boxes in the main modeling window. If the location is correct, select **Accept** to add the flaw to the model. Select **Cancel** to quit at any time. The data can be saved to a file as well. Select **Write File** to display the file selector box. Note that selecting the appropriate button on the top of the file selector box can save the crack front data only or all the crack data.

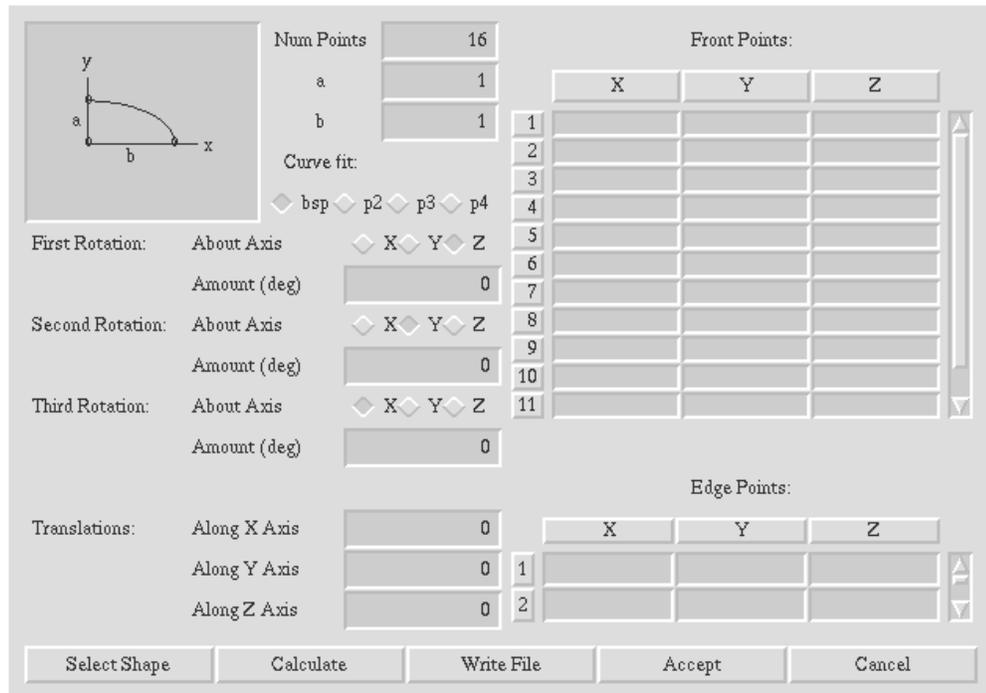


Figure 5.9. The flaw library dialog box.

Num Points

The crack front geometry available in the flaw library has a simple algebraic form (circles, ellipses, and straight lines). FRANC3D modeling software, however, uses splines as a fundamental geometry modeling form. To represent the flaws in the geometry, the algebraic form is evaluated at a number of points. An interpolating spline curve is then fit through the points and used for the actual stored geometry. This option specifies the number of points for evaluating and fitting the spline curve. Note that for some flaws, the number of points is fixed at certain values.

a,b

These correspond to the parameterized dimensions that define the geometry of the flaw. Values specified for these dimensions define the actual flaw size that will be placed in an object. They correspond to the diagram to the left of the option.

Curve fit: bsl/p2/p3/p4

This option controls how intersections are performed if the flaw geometry does not match the objects surface geometry exactly. If the fit polynomial option is selected, the crack front curve is first represented as a polynomial curve. Polynomials of 2nd, 3rd, or 4th order are available. Extrapolation and interpolation is performed along this polynomial to determine where the crack front curve will intersect free surfaces. When the fit B-spline option is selected, only limited interpolation is performed. If the crack front end points lie on or very close to free surfaces, the crack ends will be placed on the surfaces. Otherwise the crack

ends will be placed in the interior of a region rather than on an existing surface; in most cases, this is not what one wants.

Rotations

There are three rotations that can be applied to a flaw, in sequence, to set its orientation. The amount of the rotation can be specified. The sign of the rotation is determined from a "right-hand" rule. That is, if one's thumb points in the positive direction along an axis, their fingers will curl in the direction of a positive rotation. Initially, all flaws are positioned in the X, Y plane, with all Z coordinates equal zero. This orientation is illustrated in the diagram in the upper left corner of the dialog box.

Translations

These options specify how the origin of the flaw is to be translated to place it in a structure. The flaw origin is illustrated in the diagram in the upper left corner of the dialog box.

Front Points/Edge Points

These are not input fields; they are used to display output information. Once the **Calculate** button has been selected, these fields display the computed coordinates of points on the crack front and at the ends of the crack front.

Select Shape Button

Selecting this button displays a dialog box showing the library of currently available flaw shapes. This dialog box is shown in Figure 5.10. Double clicking on one of the flaw diagrams makes it the currently active flaw shape. The flaw shapes that are available are shown in Figure 5.11.

Calculate Button

Selecting this button will cause the program to calculate flaw geometry using the currently specified flaw parameters, rotations, and translations. The coordinate of the crack front and edge points will be displayed in the fields on the right side of the dialog box, and graphically in the main modeling window.

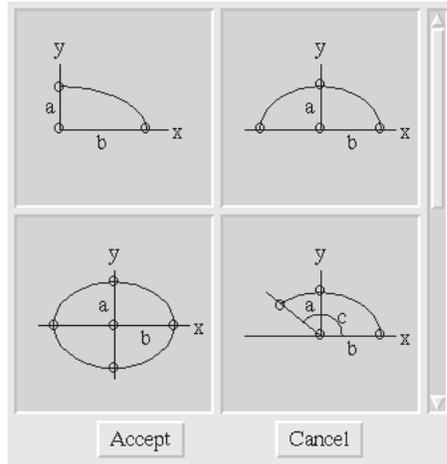


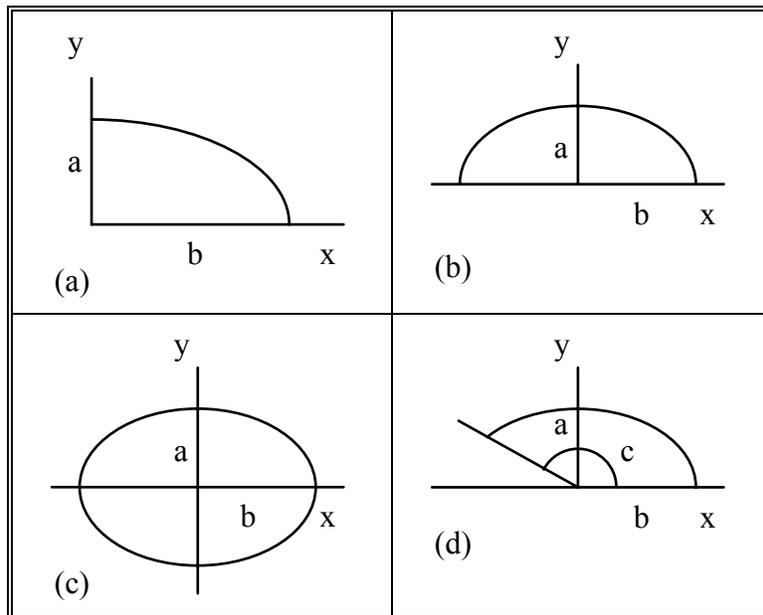
Figure 5.10. Dialog box for selecting the library flaw shape.

Write File

Selecting this button allows the analyst to save the current flaw specification data in a file, to be reused at another time.

Accept Button

Selecting this button will insert the currently defined flaw into an object.



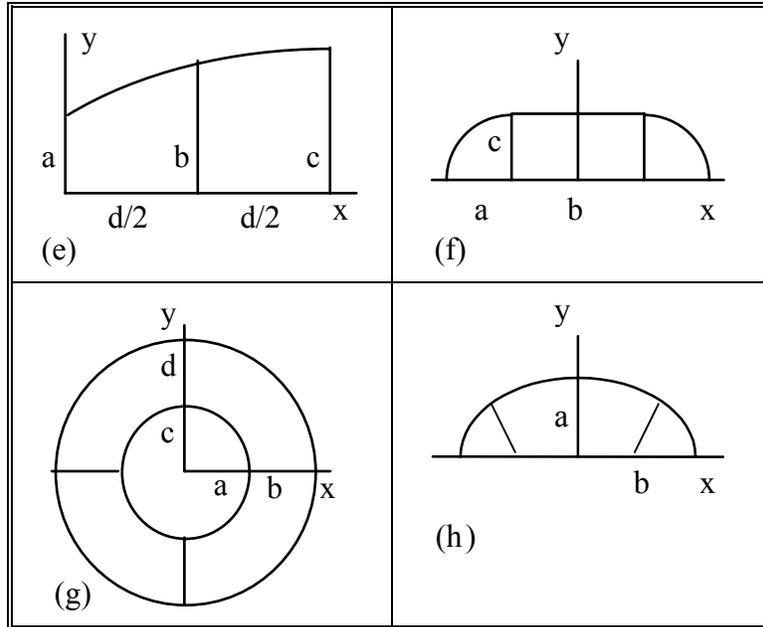


Figure 5.11. Dialog box for selecting the library flaw shape.

read flaw from file:

A file selector box is presented. Select a file that contains the complete description of the flaw and select **Accept**. The complete description of the flaw consists of all the information contained in the dialog box, Figure 5.9

show flaw in model:

The flaw is displayed in the main modeling window with the points that define the crack shown as red boxes.

create flaw in model:

The flaw is added to the model.

Part-Thru Crack

Creates a part-through surface crack whose face is bounded by 3 or more (previously existing) surface edges and one (newly created) crack front edge. The crack face is topologically treated as a 4-sided face with 1 edge as the crack front and the 3 or more edges along the exterior surface. The geometry is either a planar or b-spline patch whose corner points are (a) the two crack tips, and (b) two additional points on the free surface chain of edges. The collector for the creation of a part-through crack proceeds as follows:

- Prompts for a chain of 3 or more edges where the crack face is to intersect the free surface. The ends of this chain are the new crack tips.

- If more than 3 edges were entered on the chain, the user is prompted to identify two points on the exterior edge chain to be used as the other corner points of the bilinear mapping in addition to the tip vertices.
- Draws 2 red lines on the screen to indicate a trial surface for the crack. The first line is a trial straight-line crack front between the two crack tips. The second line is normal to this tentative crack front.
- Prompts for a point either on the crack front line or on the line normal to the front.
- If the crack front line is touched, the crack is created using that line as the front.
- If the line normal to the front is touched, the crack front is recreated as a curved line passing through the two tips and the point just touched. That is, clicking on a point ahead of the trial crack front creates a (convex) curved crack front.
- Prompting and redisplay of the trial crack continues until (a) the input point is entered on the crack front to accept the crack, or (b) the command is canceled.
- The crack front edge can be created by entering the points along the edge from a file. Pressing the **Points From File** button causes a list of all files in the directory to be displayed in a file selector box. Select the file name containing the points that will define the new crack front. The file can be free format with x-y-z coordinates of points on separate lines. Reading the points from a file allows the front to be defined explicitly.

Penny-Shaped Crack

A sub-menu is presented (arbitrary penny / quarter penny) allowing the user to add a penny-shaped crack to the model. The second option is turned off in Version 2.6.

Creates a surface crack much like the Part-Thru Crack, but topologically treated as if it were a 90 or 180 degree portion of a penny-shaped crack. The crack topology is built as (a) a set of three faces, two bezier faces and a b-spline face with the bounding edge of the b-spline as the crack front, or (b) two faces, one bezier and one b-spline. If the set of edges form an angle less than about 120 degrees, then only a single bezier face is created. For larger angles, two bezier faces are used. The collector performs the following:

- Prompts for a chain of 4 or more edges where the crack face is to intersect the free surface. The ends of this chain are the new crack tips. If more than 4 edges are entered, FRANC3D heuristically chooses the point for the center of the penny shaped crack.
- Draws two red lines on the screen to indicate a trial surface for the crack. The first line is either an arc or a straight line between the crack tips, and represents the trial crack front. The second line is normal to this crack front.
- Prompts for a point touch along either (a) the crack front line, or (b) the line normal to the front.
- If the crack front line is touched, the crack is created using that line as the front.
- If the line normal to the front is touched, the crack front is recreated as a curved line passing through the two tips and the point just touched.

- Prompting and redisplaying of the trial crack continues until (a) the input point is entered on the crack front to accept the crack, or (b) the command is canceled.
- The crack front edge can be created by entering the points along the edge from a file. Pressing the **Points From File** button causes a list of all files in the directory to be displayed in a file selector box. Select the file name containing the points that will define the new crack front. The file can be free format with x-y-z coordinates of points on separate lines. Reading the points from a file allows the front to be defined explicitly.

Surface Loop Crack

Creates a surface crack that can consist of an open or closed loop of edges, although it was designed specifically for creating circumferential cracks in cylinders.

- Prompts for a set of edges where the crack face is to intersect the free surface.
- Draws 2 red lines on the screen to indicate a trial surface for the crack. The first line consists of a series of straight-line edges matching the number of collected edges. The other line is drawn from one of the vertices along the collected set of edges in a direction normal to the outside surface.
- Vertices are created to match the vertices along the collected edges.
- The location of these vertices can be changed by selecting the corresponding vertex on the collected edges, and keying in the coordinates using the dialog box.
- The trial crack front can be moved by touching the normal line.
- Prompting and redisplaying of the trial crack continues until (a) the input point is entered on the crack front to accept the crack, or (b) the command is canceled.
- The crack front edge can be created by entering the points along the edge from a file. Pressing the **Points From File** button causes a list of all files in the directory to be displayed in a file selector box. Select the file name containing the points that will define the new crack front. The file can be free format with x-y-z coordinates of points on separate lines. Reading the points from a file allows the front to be defined explicitly.

Arbitrary 3D Crack

Nucleate a new or propagate an existing crack by tearing existing geometry faces. The user is prompted to collect the faces that will be turned into crack surfaces. The program automatically detects whether the collected faces are adjacent to an existing crack. If not, a new crack is created. If any of the faces is adjacent to an existing crack, the program then determines whether the crack will simply be propagated or whether a branch crack will be formed. The program handles all cases automatically.

return

Exits the menu.

5.1.2 Geometric Info Menu

The **GEOMETRIC INFO** menu, Figure 5.12, allows the user to obtain coordinates and Cartesian distances between two points on the model surface.

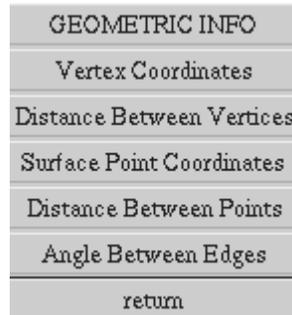


Figure 5.12. The **GEOMETRIC INFO** menu.

Vertex Coordinates

Display the vertex coordinates.

- The user is prompted to pick a vertex on the screen.
- The coordinates are presented in an Acknowledge dialog box; select **Acknowledge** to remove the dialog box from the screen.

Distance Between Vertices

Display the coordinates for two vertices and the Cartesian distance between them.

- The user is prompted to pick two vertices on the screen.
- The coordinates and the Cartesian distance between the two vertices are presented in an Acknowledge dialog box; select **Acknowledge** to remove the dialog box from the screen.

Surface Point Coordinates

Display the surface point coordinates.

- The user is prompted to pick a point on the model surface on the screen.
- The coordinates are presented in an Acknowledge dialog box; select **Acknowledge** to remove the dialog box from the screen.

Distance Between Points

Display the coordinates for two surface points and the Cartesian distance between them.

- The user is prompted to pick two points on the model surface on the screen.

- The coordinates and the Cartesian distance between the two points are presented in an Acknowledge dialog box; select **Acknowledge** to remove the dialog box from the screen.

Angle Between Edges

Display the angle between two edges that have a common end point.

- The user is prompted to pick two edges of the model on the screen.
- If the edges have a common end point, the angle between the edges is presented in an Acknowledge dialog box; select **Acknowledge** to remove the dialog box from the screen.

return

Exits the menu.

5.2 The Specify Attributes Menu

Selecting the Specify Attributes button from the **DEVELOP MODEL** menu presents the **SPECIFY ATTRIBUTES** menu, Figure 5.13. This menu presents the choices for attaching simulation attributes such as material properties and boundary conditions to the geometry.

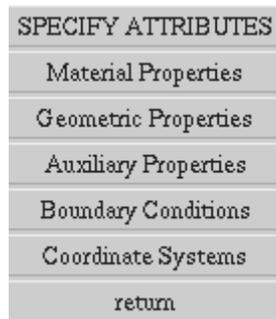


Figure 5.13. The **SPECIFY ATTRIBUTES** menu.

Specific Commands on this menu:

Material Properties

Activates the **Material Properties** menu, which displays commands to add and attach material properties to the model; see Section 5.2.1.

Geometric Properties

Activates the **Geometric Properties** menu, which displays commands to add and attach geometric properties to the model. This is used for shells only; see Section 5.2.2.

Auxiliary Properties

Activates the **Auxiliary Properties** menu, which displays commands for adding auxiliary properties to the model; see Section 5.2.3.

Boundary Conditions

Activates the **Boundary Conditions** menu, which displays commands for attaching kinematic boundary conditions and tractions; see Section 5.2.4.

Coordinate Systems

Activates the **Coordinate Systems** menu, which displays commands for displaying local face, edge, and shell-crack coordinate systems; see section 5.2.5.

return

Exits the menu.

5.2.1 The Material Properties Menu

The **MATERIAL PROPERTIES** menu, Figure 5.14, contains commands to attach material properties to regions at the geometry level.



Figure 5.14. The **MATERIAL PROPERTIES** menu.

Specific Commands on this menu:**Add New Material**

Add a new material to the list of materials. The default material is displayed in the material properties dialog box, Figure 5.15. Modify the name and select the material type button to create a new material.

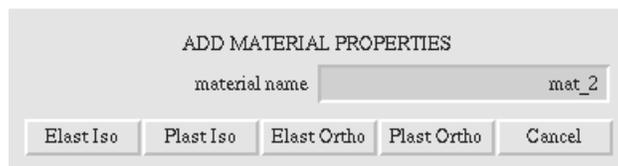


Figure 5.15. The **MATERIAL PROPERTIES** dialog box.

material name

This is the user-defined name of the material.

elast-iso/plast-iso/elast-ortho/plast-ortho

Select the material type in order to access the dialog box (Figure 5.16) that contains the relevant properties.

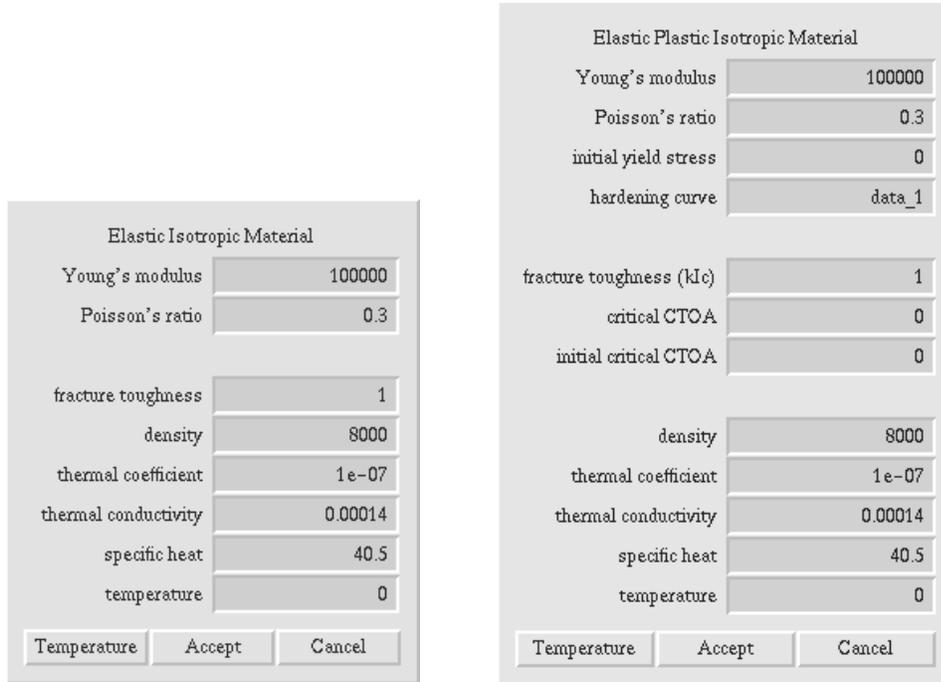


Figure 5.16a,b. The **Elastic** and **Elasto-plastic Isotropic Material** dialog boxes.

Young's modulus/Poisson's ratio/fracture toughness/density/thermal coefficient/thermal conductivity/specific heat/temperature.

These are the standard elastic isotropic properties.

initial yield stress/hardening curve

These are the standard elasto-plastic parameters. The hardening curve is defined by entering a table name in the entry field. The data for the table is entered via the **Auxiliary Properties** menu.

critical CTOA/initial critical CTOA

These are the critical crack-tip-opening-angle parameters for elasto-plastic crack growth.

For orthotropic materials, three values for Young's modulus, shear modulus, and Poisson's ratio are required, one for each coordinate direction. Two values are required for the critical crack tip-opening angle.

The *material coordinates* button defines the coordinate system for the material axes. For example, the ‘Young’s modulus 1’ corresponds to the modulus for the first axis direction in the coordinate system. A dialog of available coordinate systems is presented and the user is prompted to select a system.

The figure shows two dialog boxes side-by-side. The left dialog is titled 'Elastic Orthotropic Material' and the right is titled 'Elastic Plastic Orthotropic Material'. Both have a similar layout of input fields for material properties.

Property	Value
Young's modulus 1	200000
Young's modulus 2	0
Young's modulus 3	0
shear modulus 12	0
shear modulus 23	0
shear modulus 31	0
major Poisson's ratio 12	0.29
major Poisson's ratio 23	0
major Poisson's ratio 13	0
fracture toughness kIc1	100
fracture toughness kIc2	0
density	7800
thermal coefficient	1.3e-07
thermal conductivity	0.00014
specific heat	40.5
temperature	20
material coordinates	default global sys

The right dialog, 'Elastic Plastic Orthotropic Material', includes additional fields for plasticity:

Property	Value
Young's modulus 1	200000
Young's modulus 2	0
Young's modulus 3	0
shear modulus 12	0
shear modulus 23	0
shear modulus 31	0
major Poisson's ratio 12	0.29
major Poisson's ratio 23	0
major Poisson's ratio 13	0
initial yield stress	0
hardening curve	data_1
fracture toughness kIc1	100
fracture toughness kIc2	0
critical CTOA (CTOAc1)	0
critical CTOA (CTOAc2)	0
density	7800
thermal coefficient	1.3e-07
thermal conductivity	0.00014
specific heat	40.5
temperature	20
material coordinates	default global sys

Both dialogs have buttons for 'Temperature', 'Accept', and 'Cancel' at the bottom.

Figure 5.16c,d. The **Elastic** and **Elasto-plastic Orthotropic Material** dialog boxes.

Note that the Temperature button is currently inactive for all materials.

Edit Material

Edit the properties of a material. A sub-menu is presented (Select a Region / Select From List) which allows the user to select a material from the list of available materials or select a region of the model and thereby select the material attached to that region.

Select a Region:

The user is prompted to pick a region in the modeling window. The material properties attached to the picked region are displayed in the material properties dialog box and can be edited.

Select From List:

A list of the available materials is displayed. The user is prompted to select a material from the list. The properties are displayed in the dialog box and can be edited.

Attach Material To Region

Attach the active material to a geometry region. The user is prompted to select the region by picking the region id or by picking a point inside the region.

Delete Material

A list of the available materials is displayed if there is more than one material. The user is prompted to select one of these materials to delete.

Show Regions With Material

A list of the available materials is displayed. The user is prompted to select one of these materials. All regions that use the selected material are highlighted; the bounding edges are displayed in white. Select **FINISH** to remove the highlighting.

return

Exits the menu.

5.2.2 The Geometric Properties Menu

The **GEOMETRIC PROPERTIES** menu, Figure 5.17, contains commands to attach geometric properties to faces at the geometry level.

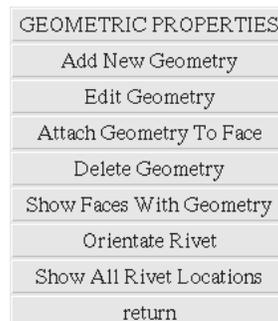


Figure 5.17. The **GEOMETRIC PROPERTIES** menu.

Specific Commands on this menu:

Add New Geometry

Add a new set of geometric properties to the list, Figure 5.18. Modify the name and number of layers and select **Accept** to create a new entry in the list. Another dialog is presented (Figure 5.19), which allows the user to define all the geometric properties for each of the layers.

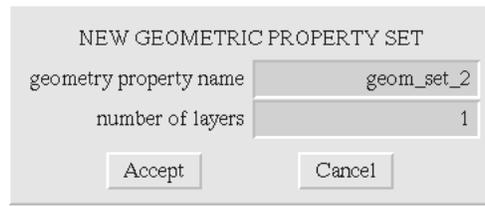


Figure 5.18. The **GEOMETRIC PROPERTIES** dialog box.

geometry property name

This is the user-defined name of the geometric property.

number of layers

The user is prompted to enter the number of layers for this geometric property.

GEOMETRIC PROPERTIES			
geometry property name	geom_set_2		
number of layers	1	layer number	1
Single Layer Information			
<input checked="" type="checkbox"/> plane stress	<input type="checkbox"/> plane strain		
layer thickness	0.036		
eccentricity	0		
material property name	mat_1		
(CONNECT TO LOWER LAYER BASED ON ECCETRICITY)			
<input type="checkbox"/> share same node	<input type="checkbox"/> rivet	<input type="checkbox"/> rivet plus adhesive	
	rivet property name	rivet_1	
	adhesive property name (unit area)	rivet_1	
	distance from orientated parallel edge to first rivet row	0	
	distance from orientated perpendicular edge to first rivet row	0	
number of rivet rows	0	row spacing	0
rivets in each row	0	column spacing	0
Accept	Prev Layer	Next Layer	Cancel

Figure 5.19. The **Geometric Properties** dialog box.

geometry property name

This field cannot be edited; it is the name entered in the previous dialog box.

number of layers

This field cannot be edited; it is the number of layers entered in the previous dialog box.

layer number

This is the number of the current layer. Geometric properties are entered for each of the layers. This number is updated by selecting the PrevLayer or NextLayer buttons at the bottom of the dialog box.

plane stress/plane strain

Choose either plane stress or plane strain by selecting the appropriate toggle button.

layer thickness/eccentricity

This defines the thickness and position of the current layer with respect to other layers. Note that FRANC3D uses a single topological face to represent all of the layers.

material property name

This is the material name defined in the Material Properties dialog.

share same node/rivet/rivet plus adhesive

This option defines how layers are bonded together. Sharing the same node between layers creates a perfect bond. Rivets and rivets-plus-adhesive use distinct nodes for each layer.

rivet property name

This is the rivet name defined in the Spring Properties in Auxiliary Properties dialog.

adhesive property name

This is the adhesive name defined in the Spring Properties in Auxiliary Properties dialog.

distance/ number/ spacing

These are parameters to define the rivet distribution pattern.

Edit Geometry

Edit the geometry properties. A sub-menu is presented (Select a Face / Select From List), which allows the user to select a geometry property from the list of available geometry property entries or select a face of the model and thereby select the geometry property attached to that face.

Select a Face:

The user is prompted to pick a face in the modeling window. The geometry properties attached to the picked face are displayed in the geometry properties dialog box and can be edited.

Select From List:

A list of the available geometry properties is displayed. The user is prompted to select a geometric property from the list. The properties are displayed in the dialog box and can be edited.

Attach Geometry To Face

Attach the active geometry property to a geometry face. The user is prompted to select the face by picking a point on the face.

Delete Geometry

A list of the available geometry properties is displayed if there is more than one. The user is prompted to select one of these geometry properties to delete.

Show Faces With Geometry

A list of the available geometry properties is displayed. The user is prompted to select one of these. Displaying the bounding edges in blue highlights all faces with those geometric properties. Select **FINISH** to remove the highlighting.

Orientate Rivet

Orient the rivets on the face of the model.

Show All Rivet Locations

Displays the rivet locations on the model surfaces. A sub-menu with layer number is presented. Rivets are shown as red highlighted boxes.

return

Exits the menu.

5.2.3 The Auxiliary Properties Menu

The **AUXILIARY PROPERTIES** menu, Figure 5.20, contains commands to define auxiliary properties which are attached to the material properties or geometric properties dialog boxes.

Specific Commands on this menu:**Add New Data Table**

Add a new data table. The user is prompted to select an entry from the submenu (Type In Data / Read From File). Selecting *Type In Data* causes a new dialog box to be displayed, Figure 5.21, and the user can define the table name and the stress/strain data. If the user selects *Read From File*, the file selector box is displayed from which the user can select the file that contains the relevant data.



Figure 5.20. The **AUXILIARY PROPERTIES** menu.

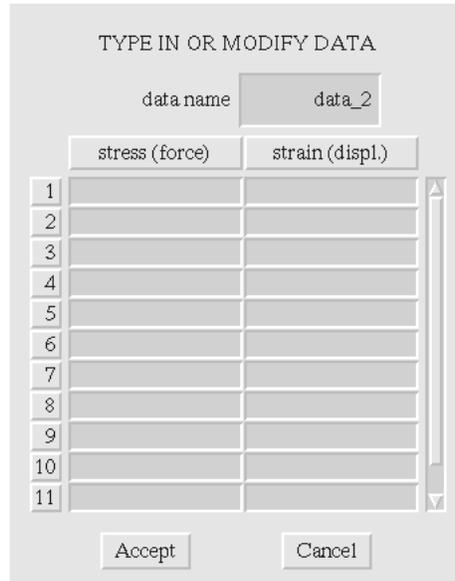


Figure 5.21. The **Type In Data** dialog box.

data name

This is the user defined name of the table.

stress (force) / strain (displ)

The spreadsheet is used to define a piecewise linear stress/strain or force/displacement curve for springs and elasto-plastic materials.

Edit Data Table

Edit the data in one of the tables. The user is prompted to select a table from the list of available tables. A dialog box, Figure 5.21, displays the data in the table. The data can be edited.

Delete Data Table

Delete one of the data tables. The user is prompted to select a table from the list of available tables. This table is deleted from the list.

Add New Spring Property

Add a new spring property. A dialog box, Figure 5.22, is presented to enter the spring property name and type of spring; either a rigid link or a generalized spring can be defined. Selecting the rigid link toggle button causes a new dialog box, Figure 5.23, to be displayed within which the user can define the element scale factor. Selecting the generalized spring toggle button causes a new dialog box, Figure 5.24, to be displayed within which the user can define the spring properties by entering the data table names that contain the spring force/displacement data.

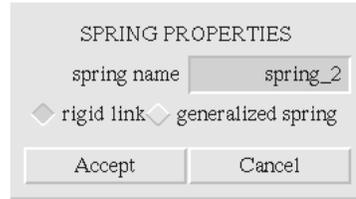


Figure 5.22. The **Spring Properties** dialog box.

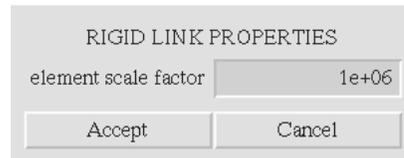


Figure 5.23. The **Rigid Link Properties** dialog box.

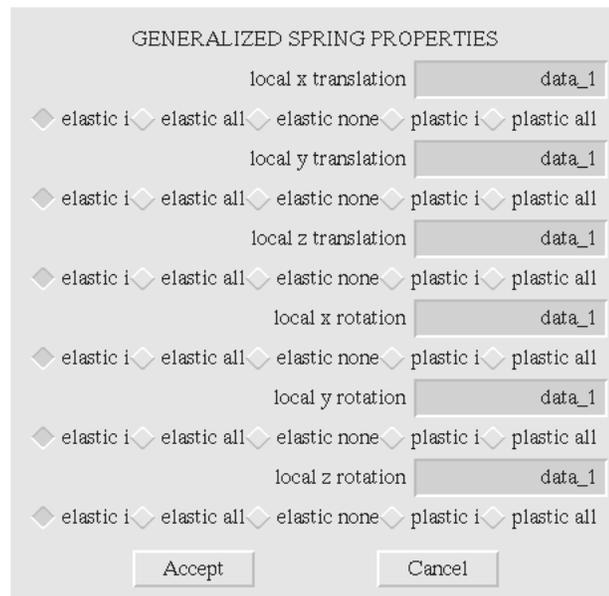


Figure 5.24. The **Generalized Spring Properties** dialog box.

Edit Spring Property

Edit the spring properties. The user is prompted to select a spring property from the list of available named spring properties.

Delete Spring Property

Delete one of the spring property entries. The user is prompted to select a spring property entry from the list of available entries. This spring property is deleted from the list.

return

Exits the menu.

5.2.4 The Boundary Conditions Menu

The **BOUNDARY CONDITIONS** menu, Figure 5.25, contains commands to attach boundary conditions to geometric faces, edges or vertices.



Figure 5.25. The **BOUNDARY CONDITIONS** menu.

Concepts for this menu:

Local versus global coordinate systems - Boundary conditions can be added in a local or in the global, or in a user defined coordinate system. The local coordinate system varies for each geometric entity. The local coordinate system can be displayed in the modeling window (see the **Face Coordinate System Menu**, Section 5.2.4). Traction applied in the global coordinate system are positive if they point in a positive axis direction. Traction applied in the local coordinate system are positive if they point in the same direction as the local normal. The user defined coordinate system is defined in Section 5.2.5.

Uniform values or general distributions - Boundary conditions applied to faces and edges can be either uniform values or arbitrary distributions. Arbitrary distributed loads are applied by creating a mesh representation (mrp) file for the geometric entity (see the FRANC3D Concepts & Users manual for details).

Model boundary conditions - Model boundary conditions can be applied by interpolating boundary conditions from existing finite element results or by applying acceleration (either linear as in gravity or rotational). Finite element results, such as nodal temperatures, can be stored in a MRP and attached to the model.

Specific Commands on this menu:**Define MRP BC's**

Activates the **Define MRP** menu, which displays commands for defining mesh representations (MRPs); see section 5.2.4.1.

New Boundary Condition

Create a new boundary condition set. The user is prompted to select an entry from the sub-menu (Model BCs / Face BCs / Edge BCs / Vertex BCs / Cancel). The user is then presented with the appropriate dialog boxes for defining the boundary condition.

Model BCs:

The **New Boundary Condition Set** dialog is presented (Figure 5.26a). A boundary condition name must be defined to differentiate between different model boundary conditions. The number of load cases must be defined. Model boundary conditions can then be set for each load case. Select **Accept** and the **Model Boundary Conditions** dialog is displayed (Figure 5.26b). This dialog allows the user to specify the linear or rotational acceleration. Also the user can attach an MRP by pressing the MRP BC button and selecting the MRP from the list. The MRP list is discussed in section 5.2.4.2.

NEW MODEL BOUNDARY CONDITION SET

boundary condition name: model_bc_set_1

number of load cases: 1

Accept Cancel

Figure 5.26a. The **New Boundary Condition Set** dialog box for model boundary conditions.

MODEL BOUNDARY CONDITIONS

boundary condition name: model_bc_set_1

load case: default load case

none rotation linear acceleration

Magnitude of Rotation Vector (Omega)

x = 0 y = 0 z = 0

Magnitude of Linear Acceleration Vector (Gravity)

x = 0 y = 0 z = 0

MRP BC: no MRP

Accept Prev Load Case Next Load Case Cancel

Figure 5.26b. The **Model Boundary Conditions** dialog box.*boundary condition name*

This is the user defined name of the model boundary condition.

load case

The load case button displays a list of the load cases (see section 5.2.4.1). The model boundary conditions are added to the chosen load case. NOTE: all boundary conditions belong to the default load case.

MRP BC:

The MPR BC button displays a list of the MRPs (see section 5.2.4.2). The selected MRP becomes part of the model boundary conditions.

Note that crack face tractions are applied as face boundary conditions in Version 2.6 rather than using the .conn and .fstr files as model boundary conditions in Version 1.15.

Face BCs:

The **New Boundary Condition Set** dialog box is presented (Figure 5.27a). A boundary condition name must be defined to differentiate between different face boundary conditions. The number-of-layers entry allows the user to define different boundary conditions on the layers defined in the **Geometric Properties** dialog. Select **Accept** and the **Face Boundary Conditions** dialog box (Figure 5.27b) is displayed. This dialog allows the user to define kinematic constraints and traction or surface loads. The boundary conditions can be applied in either global Cartesian or surface local coordinate systems. A negative surface traction in the local n-direction is a positive pressure on the surface. Both uniform and arbitrary general distributions can be applied. General distributions are defined through the use of MRP's; MRP stands for mesh representation. Outside of FRANC3D, the user must create a pseudo-mesh for the geometry face. Displacement or traction values are defined at the nodes of this pseudo-mesh and this information along with the node coordinates and element connectivity is written to a file with a .mrp extension. The pseudo-mesh does not have to match the actual mesh on the model. Values at the nodes of the actual mesh are computed from the information in the MRP, i.e., values are interpolated using the pseudo-mesh element shape functions to obtain displacement/traction boundary condition values at the nodes of the actual mesh.

NEW FACE BOUNDARY CONDITION SET

boundary condition name	face_bc_set_1
number of layers	1
number of load cases	1

Accept Cancel

Figure 5.27a. The **New Boundary Condition Set** dialog box for face boundary conditions.

BOUNDARY CONDITIONS FOR A FACE

BC Name: face_bc_set_1 Layer: 1

Coord Sys: default global sys global Cartesian surface local user defined

Load Case: default load case MRP BC: no MRP

x (global or user Cartesian [green]) or n (surface local [red])

displacement traction 0 rotation moment 0

y (global or user Cartesian [blue]) or u (surface local [green])

displacement traction 0 rotation moment 0

z (global or user Cartesian [red]) or v (surface local [blue])

displacement traction 0 rotation moment 0

Accept Prev Layer Next Layer Prev Load Case Next Load Case Cancel

Figure 5.27b. The **Face Boundary Conditions** dialog box.

BC name

This is the user defined name of the face boundary condition.

Coord Sys:

The user can select the coordinate system for the boundary conditions.

Load Case

The load case button displays a list of the load cases (see section 5.2.4.1). The model boundary conditions are added to the chosen load case. NOTE: all boundary conditions belong to the default load case.

MRP BC:

The MRP BC button displays a list of the MRPs (see section 5.2.4.2). The selected MRP becomes part of the model boundary conditions. Crack face tractions are applied using MRP's in the global coordinate system assuming that the stresses in the MRP are defined in the global Cartesian system.

Edge BCs:

The **New Edge Boundary Condition Set** dialog box is presented (similar to Figure 5.27a). After editing the edge boundary condition name, the number of layers, and load cases, select **Accept**. A dialog box similar to that for faces is displayed (similar to Figure 5.27b).

Vertex BCs:

The **New Vertex Boundary Condition Set** dialog box is presented (similar to Figure 5.27a). After editing the vertex boundary condition name, the number of layers, and number of load cases, select **Accept**. A dialog box similar to that for faces is displayed (Figure 5.28). A general distribution boundary condition cannot be applied to a vertex, so the MRP BC button is missing.

BOUNDARY CONDITIONS FOR A VERTEX

BC Name: vert_bc_set_1 Layer: 1

Coord Sys: default global sys global Cartesian point local user defined

Load Case: default load case

x (global or user or local Cartesian [green])

displacement traction 0 rotation moment 0

y (global or user or local Cartesian [blue])

displacement traction 0 rotation moment 0

z (global or user or local Cartesian [red])

displacement traction 0 rotation moment 0

Accept Prev Layer Next Layer Prev Load Case Next Load Case Cancel

Figure 5.28. The **Vertex Boundary Conditions** dialog box.

Edit Boundary Condition

The user is prompted to select an entry from the sub-menu (Model BCs / Face BCs / Edge BCs / Vertex BCs / Cancel). The user is then presented with a scrollable list of the appropriate boundary conditions (Figure 5.29). The boundary conditions are listed by their name. Figure 5.29 shows a list of available face boundary conditions. Select an entry in the scroll box by double clicking the left mouse button on it or a single-click and select **Accept**. The appropriate dialog box is displayed with the current boundary conditions associated with this name. The user can edit the boundary conditions, but cannot edit the name or number of layers. To do that, create a new boundary condition.



Figure 5.29. The scrollable list of boundary conditions.

Delete Boundary Condition

The user is prompted to select an entry from the sub-menu (Model BCs / Face BCs / Edge BCs / Vertex BCs / Cancel). The user is then presented with a scrollable list of the appropriate boundary conditions. The boundary conditions are listed by their name. Select an entry in the scroll box by double clicking the left mouse button on it or a single-click and select **Accept**. This entry will be deleted from the list.

Attach Boundary Condition

The user is prompted to select an entry from the sub-menu (Model BCs / Face BCs / Edge BCs / Vertex BCs / Cancel). The user is then presented with a scrollable list of the appropriate boundary conditions. The boundary conditions are listed by their name. Select an entry in the scroll box by double clicking the left mouse button on it or a single-click and select **Accept**. The user is then prompted to collect the appropriate entities in the modeling window to which this boundary condition set will be attached.

Detach Boundary Condition

The user is prompted to select an entry from the sub-menu (Model BCs / Face BCs / Edge BCs / Vertex BCs / Cancel). The user is then prompted to collect the appropriate entities in the modeling window from which the boundary conditions are to be detached.

Show Boundary Condition

The user is prompted to select an entry from the sub-menu (List All BCs / BCs on Model / BCs on Face / BCs on Edge / BCs on Vertex / Cancel). If the user selects List All BCs, a set of scrollable lists of all boundary conditions is presented (Figure 5.30). The boundary conditions are listed by their name. Select an entry in one of the scroll boxes by double-clicking the left mouse button on it. *NOTE: if double-clicking with the left mouse button does not work, try a left click to select the name followed by a middle click.* The entities that have this boundary condition are highlighted in the modeling window. In addition, the boundary condition dialog box is displayed so that the user can immediately see the

boundary conditions (the boundary conditions cannot be edited here). Selecting the **FINISH** button turns off the highlighting. The boundary condition dialog is dismissed by pressing the **Cancel** button; the boundary conditions cannot be edited here. The list of available boundary conditions remains on the screen until the user selects the **Cancel** button.

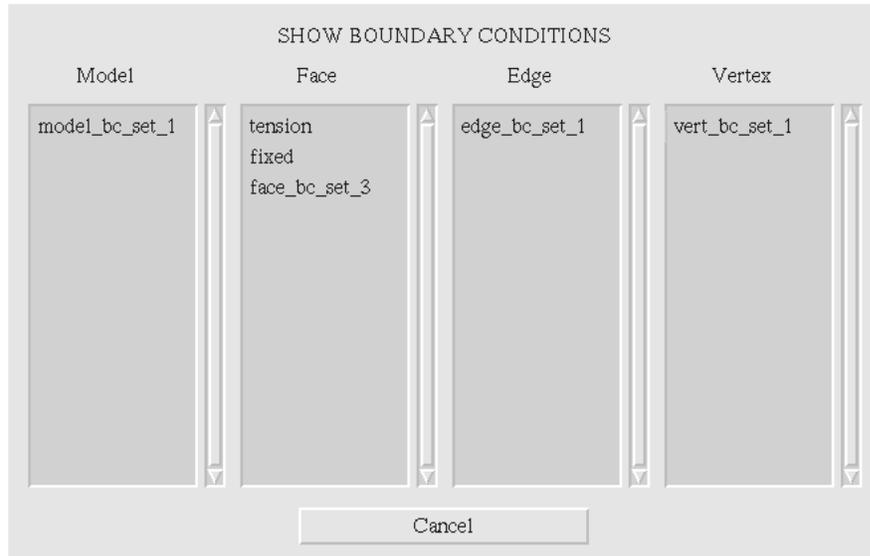


Figure 5.30. The set of scrollable lists showing all boundary condition sets.

If the user selects BCs on Model, any model boundary condition attached to the model is displayed in the appropriate dialog. If a model boundary condition is not attached, an acknowledge message is displayed instead. If the user selects BCs on Face or BCs on Edge or BCs on Vertex, the user is then prompted to select the appropriate entity on the model and the attached boundary conditions are displayed in the appropriate dialog box. If the entity does not have a boundary condition attached, an acknowledge message is displayed instead.

Hilight Non-default BC's

Hilight the entities with non-default boundary conditions. The user is prompted to select an entry from the sub-menu (Face BCs / Edge BCs / Vertex BCs / Cancel). All entities with non-default boundary conditions are highlighted in the modeling window.

Reset BC's to default values

Resets the boundary condition data to its default value. The user can choose to reset either Model, All Faces, All Edges, All Vertex, or All Of The Above boundary conditions. A dialog box is presented verifying the requested operation allowing the user to cancel if a menu button was selected by accident. The default boundary conditions are zero tractions and zero moments in the global coordinate system and no model boundary conditions.

return

Exits the menu.

5.2.4.1 The Define MRP BC's Menu

The **DEFINE MRP** menu, Figure 5.31, contains commands to add, edit and delete MRPs that are to be used as boundary conditions.



Figure 5.31. The **DEFINE MRP** menu.

Concepts for this menu:

Mesh representations – general distributions of boundary conditions are defined through the use of MRP's; MRP stands for mesh representation. The user must create a pseudo-mesh for the geometry volume, face, or edge. The pseudo-mesh does not have to match the geometry completely, but it must match that portion where the boundary conditions are to be applied. In addition, the pseudo-mesh does not have to match the actual mesh on the model. MRP's are described further in the FRANC3D Concepts & Users manual. Displacement or traction values are defined at the nodes of this pseudo-mesh. Values at the nodes of the actual mesh are computed from the information in the MRP; values are interpolated using the pseudo-mesh element shape functions to obtain displacement or traction boundary condition values. The MRP supports both interpolation and extrapolation. In the case of a 3D finite element volume MRP, where only some surface nodes have real boundary condition values, extrapolation to the surface of the MRP is desired.

Specific Commands on this menu:**Add MRP**

Display the **MRP** dialog box, Figure 5.32. The user enters a unique name for the MRP. Then the nodes, elements, and boundary condition values are defined. The MRP can be imported from a file or defined manually. The MRP can be exported to a file.

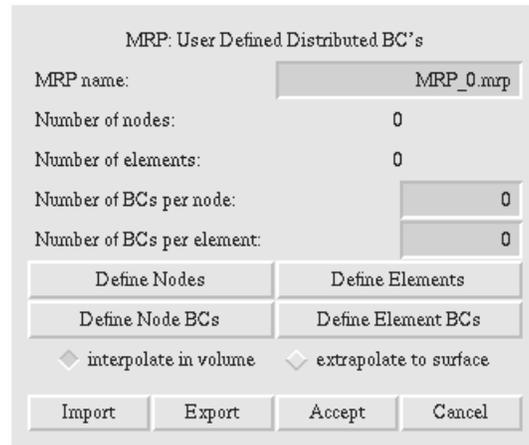


Figure 5.32. The **MRP** dialog box.

MRP name

This is the user defined name of the MRP.

Number of nodes:

The number of nodes is updated as the nodes are defined. A spreadsheet is presented, Figure 5.33, to define the nodes with their global Cartesian coordinates.

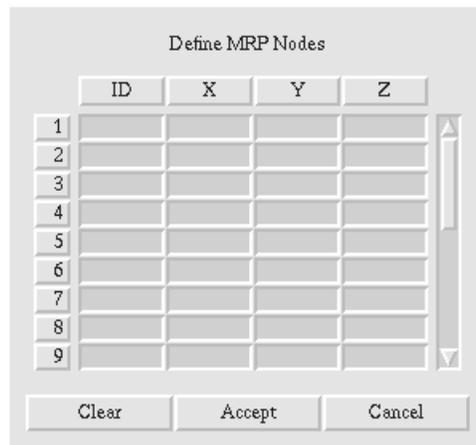


Figure 5.33. The **MRP Nodes** dialog box.

Number of elements:

The number of elements is updated as the elements are defined. A spreadsheet is presented, Figure 5.34, to define the elements with the element type and node connectivity. The element type is an integer value that can be obtained by selecting the **TYPE** button in the dialog box, Figure 5.35. The MAT entry is for a material identifier and can be set to 0 if not needed.

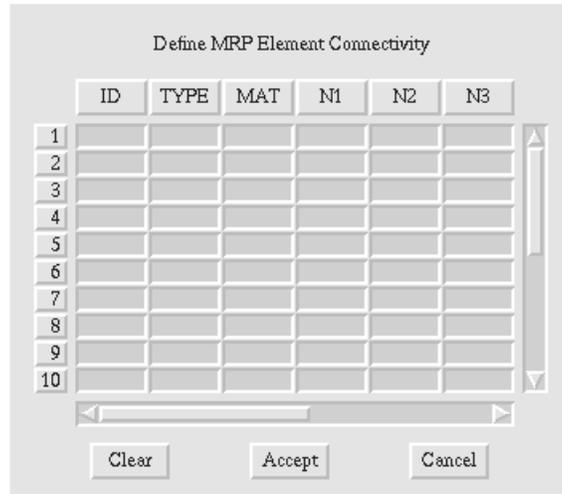


Figure 5.34. The **MRP Element** dialog box.

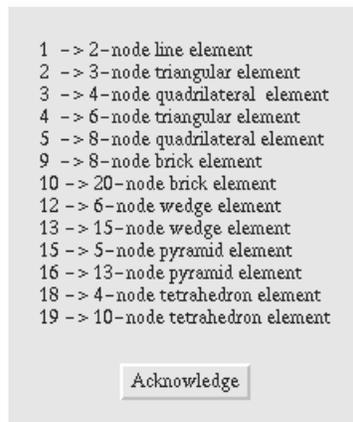


Figure 5.35. The **MRP Element Type** dialog box.

Number of BCs per node

The user specifies the number of boundary condition values to be defined at each node of the MRP. A spreadsheet is presented, Figure 5.36, to define the node boundary conditions. The boundary condition type is specified by pressing the PICK_BC button and selecting the boundary condition type from the list, Figure 5.37.

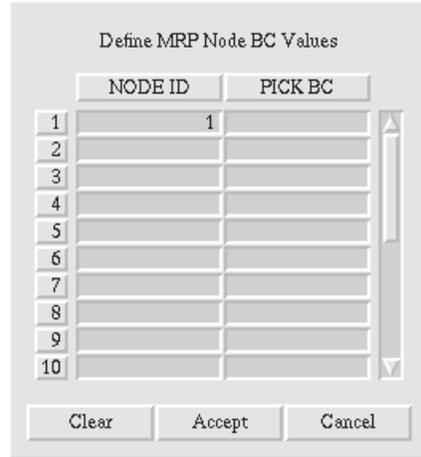


Figure 5.36. The **MRP Node BC Values** dialog box.

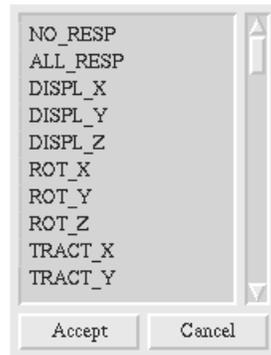


Figure 5.37. The **MRP PICK_BC Type** dialog box.

Number of BCs per element

The user specifies the number of boundary condition values to be defined at each node of each element of the MRP. The values at common nodes can be different for adjacent elements. A spreadsheet is presented, similar to that for nodes, Figure 5.36, to define the element boundary condition values.

interpolate in volume / extrapolate to surface

The default is interpolate in volume, but in the case of an MRP that is being used to apply surface boundary conditions, the user can switch to extrapolate to surface so that zero values at internal nodes are not factored into computing the surface boundary condition values.

Edit MRP

Display the list of MRP names in a selector box. The user can select a MRP and then edit the MRP information. If the MRP is imported from a file and contains more than 50 lines of information, the interactive dialog boxes truncate the

information and do not allow the user to edit the information. Editing can be done in the file.

Delete MRP

Display the list of MRP names in a selector box. The user can select a MRP to delete.

return

Exits the menu.

5.2.5 The Coordinate System Menu

The **COORDINATE SYSTEMS** menu, Figure 5.38, contains commands to display additional menus for displaying face, edge, and crack tip coordinate systems. It also has options for adding user defined coordinate systems for use with the boundary conditions.

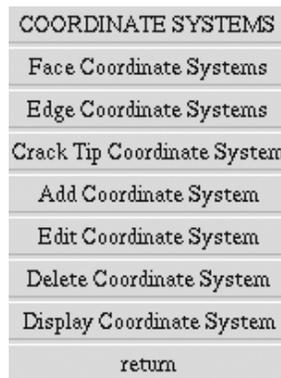


Figure 5.38. The **COORDINATE SYSTEMS** menu.

Specific Commands on this menu:

Face Coordinate Systems

Display the **Face Coordinate Systems** menu, see Section 5.2.5.1.

Edge Coordinate Systems

Display the **Edge Coordinate Systems** menu, see Section 5.2.5.2.

Crack Tip Coordinate System

Display the local coordinate system at the crack tip. This is used for shell cracks to determine the orientation of the crack growth vector. The user is prompted to select the crack tip.

Add Coordinate System

Display the dialog box, Figure 5.39, to define a coordinate system,.

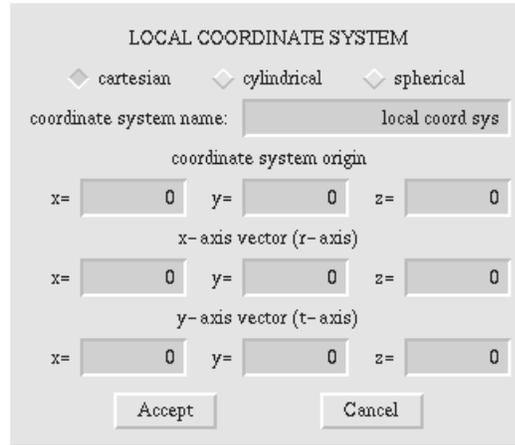


Figure 5.39. The **COORDINATE SYSTEMS** dialog box.

Cartesian/cylindrical/spherical

The user selects the type of coordinate system to define.

Coordinate system name

Unique name for the coordinate system.

Origin/axis

The user defines the origin and the first two axes.

Edit Coordinate System

Display the list of user defined coordinate systems. The user selects the coordinate system to edit and the dialog box, Figure 5.39, is displayed.

Delete Coordinate System

Display the list of user defined coordinate systems. The user selects the coordinate system to delete.

Display Coordinate System

Display the list of user defined coordinate systems. The user selects the coordinate system to draw in the modeling window. Green is the x, blue is the y, and red is the z axis.

return

Exits the menu and turns off the coordinate systems.

5.2.5.1 The Face Coordinate System Menu

The **FACE COORDINATE SYSTEMS** menu, Figure 5.40, contains commands to modify the local face coordinate systems. The normal to the face is shown in red, and the in-plane coordinate axes are shown in blue and green.

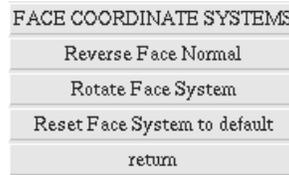


Figure 5.40. The **FACE COORDINATE SYSTEMS** menu.

Specific Commands on this menu:

Reverse Face Normal

Reverse the local normal coordinate direction of a face. The user is prompted to select the face.

Rotate Face System

Rotate the local in-plane coordinate axes. The user is prompted to select the face and then enter the amount of rotation in the dialog box, Figure 5.41.

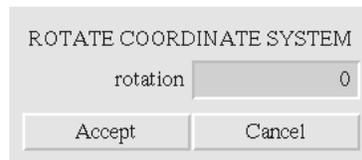


Figure 5.41. The **Rotate Coordinate System** dialog box.

Reset Face System to default

Reset the face coordinate system to the default by removing the rotation and reverse normal flag. A dialog box is displayed asking the user if it is okay to reset all face coordinate systems; press **Ok** or **Cancel**.

return

Exits the menu and turns of the coordinate systems.

5.2.5.2 The Edge Coordinate System Menu

The **EDGE COORDINATE SYSTEMS** menu, Figure 5.42, contains commands to reverse local tangents of geometric edges. The edge tangent direction is shown in green; the blue and red show two orthogonal directions.

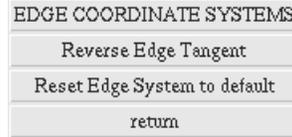


Figure 5.42. The **EDGE COORDINATE SYSTEMS** menu.

Specific Commands on this menu:

Reverse Edge Tangent

Reverse the local tangent direction for a selected edge. The user is prompted to select the edge.

Reset Edge System to default

Reset the edge coordinate system to the default by removing the reverse normal flag. A dialog box is displayed asking the user if it is okay to reset all edge coordinate systems; press **Ok** or **Cancel**.

return

Exits the menu and turns off the coordinate systems.

5.3 The Subdivide Regions Menu

The **SUBDIVIDE REGIONS** menu, Figure 5.43, has commands for subdividing volume regions by the addition of interior faces. These faces are defined at the subdomain level of FRANC3D, and do not alter the actual model geometry. The changes apply only to this hierarchy level and to the levels below.



Figure 5.43. The **SUBDIVIDE REGIONS** menu.**Specific Commands on this menu:****Add Vertex**

Adds a vertex at the subdomain level. A sub-menu is presented (to a Face / N vertex to a Face / cancel) allowing the user to add one or multiple vertices to a face.

to a Face:

The user is prompted to enter the position of the new vertex on a face of the model. The vertex coordinates can be entered in a dialog box by pressing the **Key-In-Coordinates** button before selecting the point in the modeling window. After picking the point, a dialog box (see Figure 5.3) is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.

N vertex to a Face:

The user is prompted to enter the position of a number of new vertices on a face of the model. The user is prompted to pick the face first. The user can then select the **Points From File** button and select a file containing the x y z coordinates of multiple points or simply pick the points on the face using the mouse.

Add Edge

Adds an edge to a face of the subdomain model. The user is prompted to identify the face and end points of the edge. The process is similar to that at the geometry level.

- The face can be chosen before adding the edge by pressing the **Indicate Face Before** button first and then selecting the face.
- The exact coordinates of the end points of the edge can be entered by pressing the **Key-In-Coordinates** button before selecting the point on the face.
- Select the end points of the edge using the mouse. Make sure both points lie on the same face. The **Pick Vertex as Point** button can be selected before picking the end point to force the picked point to be the nearest vertex.

Delete Edge

Removes an edge previously defined by the **Add Edge** command of this menu. The user is prompted to touch the edges to be deleted and then select **FINISH** when done.

Split Edge

Adds a vertex to an edge at the subdomain level. The user is prompted to enter the position of the new vertex. The exact coordinates can be entered by pressing the **Key-In-Coordinates** button before selecting a point along the edge. After

picking the point on the edge, a dialog box is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.

Unsplit Edge

Removes a vertex previously defined by the **Split Edge** command of this menu. The user is prompted to touch the vertex to be deleted. Note that if the edge was split more than once, all 'split' vertices are deleted to recover the original edge. This is true as long as the 'split' vertex only has two incident edges.

Add Face

Adds an interior surface at the subdomain level. A sub-menu (select Vertices / select Edges / cancel) is presented allowing the user to select either pre-defined edges or vertices. Three face types can be created: planar polygons, quadrilateral B-splines, or tri-cubic bezier triangles.

select Vertices:

The user is prompted to collect the vertices along the boundary of the proposed face. The following conditions are applied to the selected vertices:

- If two vertices on a single face are not joined by an edge, an edge is created between the two points. This excludes wireframe edges. If two vertices are connected by a wireframe edge that is not on a face, then use the *Select Edges for interior face* option.
- The command will create either a planar polygon, quadrilateral bi-cubic B-spline, or tri-cubic bezier triangle depending on the geometry of the edges joining the boundary vertices and the number of edges bounding the surface.
- No face will be created if the proposed face violates topological conditions of the existing solid, for example if the face will cross another face.

select Edges:

The user is prompted to select the edges that will form the boundary of the face. The following conditions apply:

- The edges must form a valid closed loop within a single region.
- The user has the option of specifying the type of face and the corner vertices of the face. Press the **Specify Corner Vertices** button before collecting the edges and a dialog box will appear, (see Figure 5.2), which will allow the user to specify the type of face and number of corner vertices. Consistency checks are done after the edges are collected to ensure that the specified face type is valid.
- The user has the option of specifying the region to which the face will be added before collecting edges. Press the **Specify Region** button before doing any collecting or the button will disappear. NOTE: a region can be specified by picking the region based on a unique integer identifier or by picking a point in the region. To pick a point in the region, first pick a point in the region, then rotate the model so that the pick line appears, and then pick another point along the first pick line so that the intersection point is in

the desired region, or enter the coordinates of a point inside the region after pressing the **Key-In-Coordinates** button. Also, if there is only a single region, this region is automatically selected when the **Specify Region** button is pressed.

- By default, if more than four edges are specified, a b-spline surface is created with four corner vertices located at the four smallest angles between adjacent edges.
- Bezier surfaces are created if only three edges are chosen.
- Planar surfaces can be created if there are three or four straight-line edges selected that all lie in a single plane.

Delete Face

Removes a face previously defined by the **Add Face** command on this menu. The user is prompted to touch the face to be deleted. The command will not delete a face that has interior edges added by the Add Line command of this menu (i.e. dangling edges).

return

Exits the menu.

5.4 The Subdivide Faces Menu

The **SUBDIVIDE FACES** menu, Figure 5.44, provides commands to add and remove edges from faces of the FRANC3D subregion model.



Figure 5.44. The **SUBDIVIDE FACES** menu.

Specific Commands on this menu:

Add Vertex

Adds a vertex at the subdomain level. A sub-menu is presented (to a Face / N vertex to a Face / cancel) allowing the user to add one or multiple vertices to a face.

to a Face:

The user is prompted to enter the position of the new vertex on a face of the model. The vertex coordinates can be entered in a dialog box by pressing the **Key-In-Coordinates** button before selecting the point in the modeling window. After picking the point, a dialog box (see Figure 5.3) is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.

N vertex to a Face:

The user is prompted to enter the position of a number of new vertices on a face of the model. The user is prompted to pick the face first. The user can then select the **Points From File** button and select a file containing the x y z coordinates of multiple points or simply pick the points on the face using the mouse.

Delete Vertex

Delete a vertex added by the **Add Vertex** command of this menu. The user is prompted to select the vertex to be deleted.

Add Edge

Add an edge at the subregion level leaving the geometry and subdomain levels of the hierarchy unchanged. The user is prompted to select the two end points of the edge. The process is similar to that of the geometry and subdomain levels.

- The face can be chosen before adding the edge by pressing the **Indicate Face Before** button first and then selecting the face.
- The exact coordinates of the end points of the edge can be entered by pressing the **Key-In-Coordinates** button before selecting the point on the face.
- Select the end points of the edge using the mouse. Make sure both points lie on the same face. The **Pick Vertex as Point** button can be selected before picking the end point to force the picked point to be the nearest vertex.

Delete Edge

Delete an edge added by the **Add Edge** command of this menu. The user is prompted to select the edges to be deleted and then select **FINISH** when done.

Split Edge

Adds a vertex to an edge at the subregion level. The user is prompted to enter the position of the new vertex. The exact coordinates can be entered by pressing the **Key-In-Coordinates** button before selecting a point along the edge. After picking the point on the edge, a dialog box is displayed with the point coordinates. Enter the exact coordinates and select **Accept**.

UnSplit Edge

Removes a vertex from an edge that was added by the **Split Edge** command of this menu. This removes the vertex and merges the two incident edges. Note that if the edge was split more than once, all 'split' vertices are deleted to recover the original edge. This is true as long as the 'split' vertex only has two incident edges.

return

Exits the menu.

5.5 The Subdivide Edges Menu

Commands in the SUBDIVIDE EDGES menu, Figure 5.45, control the subdivision of edges into straight-line segments. These segments are used for later finite or boundary element meshing. Changes at this hierarchy level do not affect the levels above.

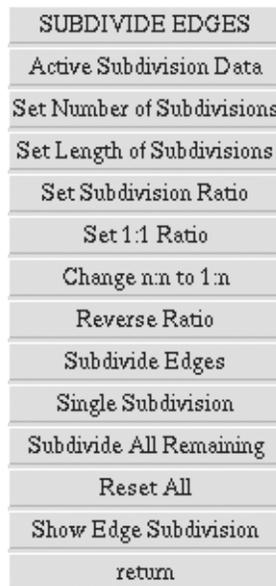


Figure 5.45. The **SUBDIVIDE EDGES** menu.

Concepts for this menu:

Display of current subdivision points - When the Subdivide Edges menu is active, current subdivision points are displayed along each edge.

Edge Subdivision - Subdivision of edges into straight lines is controlled by three integer parameters:

- **N** - the number of segments that are placed along the line.
- **L1:L2** - integers that determine the size ratio between the first and last segment.

These are modal quantities; once they are set, their values remain in effect for all subsequent uses of the **Subdivide Lines** and **Subdivide All Remaining** commands.

Example:

- When N=3, L1=1, L2=1, **Subdivide Lines** will subdivide geometry edges into 3 segments of equal length.

- When $N=3$, $L1=1$, $L2=4$ **Subdivide Lines** will subdivide geometry edges into 3 segments, with the last segment 4 times the length of the first.
- When $N=3$, $L1=3$, $L2=2$, **Subdivide Lines** will subdivide geometry edges into 3 segments, with the last segment $2/3$ the length of the first.

Specific Commands on this menu:

Active Subdivision Data

Displays a dialog box, Figure 5.46, containing the three parameters N (number of subdivisions), $L1$ (first segment length), and $L2$ (last segment length). The user is able to set the number of subdivisions as well as the ratio between the size of the first and last segments. If the user selects the **Accept** button, the changes are displayed in the **Active Subdivision Data** information box. This dialog box can be displayed during the process of collecting edges to be subdivided and any changes will affect only those edges that are collected after selecting **Accept**.

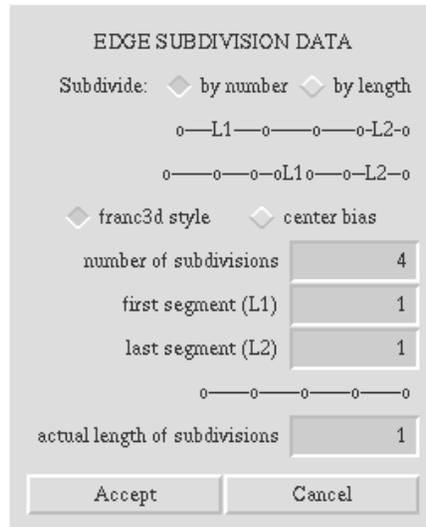


Figure 5.46. The **EDGE SUBDIVISION DATA** dialog box.

by number/by length

The user can subdivide edges based on number or approximately based on a defined length.

franc3d style subdivisions

The subdivision ratio defaults to the franc3d style.

center bias

The subdivision ratio can be biased towards or away from the center of the edge rather than from one end of the edge.

number of subdivisions / first segment length / last segment length

Set these fields when subdividing by number.

actual length of subdivisions

Set this field if subdividing based on length.

Number of Subdivisions

Displays a keypad for entry of the number of segments. Select the number on the keypad and press ENT(er). The data in the **Active Subdivision Data** information box will be updated immediately.

Subdivision Ratio

Displays the keypad twice for entry of the L1 and L2 parameters.

1:1 Ratio

Sets both L1 and L2 to the larger of the two values of L1 and L2. Subsequent subdivisions will produce segments of equal length.

Change n:n to 1:n

Resets the ratio of L1 to L2.

- Sets L1 to 1.
- Sets L2 to the larger of the prior values of L1 and L2.

Reverse Ratio

Reverses (swaps) the L1 and L2 values. Any edges that are already selected that need to be subdivided in the reverse order must be re-selected.

Subdivide Edges

Subdivides selected edges based on the current subdivision data. The user is prompted to select the edges to be subdivided.

- Each edge that is selected is subdivided according to N, L1, and L2 at the time of the selection.
- Selected edges are displayed in green with boxes at the subdivision points.
- Subdivision does not take effect until **FINISH** is selected.

Single Subdivisions

The user is prompted to select edges. The edges are subdivided using only one segment.

Subdivide All Remaining

Applies the current N, L1 and L2 subdivision data to all edges that still have their default subdivision.

Reset All

Resets all edges to their default subdivision. A dialog box is presented stating that all edges will be reset to their default subdivisions. The user can select **Cancel** if he/she hit this menu button by accident. The default subdivision varies

for each edge depending on the original geometry. For straight-line edges, the default is one segment. For curved edges, the default is usually between two and four segments with a one-to-one ratio.

Show Edge Subdivision

The user is prompted to touch an edge. The subdivision ratio and number of segments is displayed in a message box. Select **Acknowledge** to continue.

return

Exits the menu.

5.6 The Mesh Surfaces Menu

The **MESH SURFACES** menu, Figure 5.47, contains commands for creating and deleting surface meshes.

Concepts for this menu:

Edge Subdivision - Before surfaces can be meshed, edges must be subdivided into line segments. This is done with the **Subdivide Edges** menu, Section 5.5. The nodes of the edge subdivision subsequently become nodes of the mesh. Midside nodes of the elements are not explicitly defined in the MSH model. Rather, midside nodes can be generated when writing the analysis files.



Figure 5.47. The **MESH SURFACES** menu.

Mapped Meshing - A mapped mesh is one in which a specific regular mesh - e.g. an m by n grid over a 4 sided region - is constructed on a surface. Mapping can only be

applied if (a) the region can be treated as if it has exactly 3 or 4 sides (some of which may contain several actual edges chained together), and (b) the number of nodes along the various sides agrees with a mapping method. When these conditions apply, mapping produces smooth meshes of well-shaped elements. Mapping may not be applicable in transition regions.

Triangulation - Triangulation methods can produce meshes over surfaces of arbitrary shape and with edge subdivisions that prevent use of regular mappings. Triangulation methods in FRANC3D will often produce excellent transition meshes with well-shaped triangles in all portions of a surface.

Automatic Meshing - For a given surface with edge subdivisions, FRANC3D automatic meshing commands will first attempt to apply a mapping method. If no mapping is applicable, triangulation is applied.

Single-face Meshing - Mapped meshing requires that key nodes on a face be identified as corner points of the mapping. Each mapped meshing command begins by prompting for a choice between (a) user-specified corner points, or (b) automatic (no) corner points. If the user-specified corner points option is chosen, FRANC3D issues one prompt for a single face, followed immediately by (depending on the mapping) 3 or 4 prompts to identify key corner points. If the **No Corner Points** option is chosen, FRANC3D prompts only for a face touch. FRANC3D examines the face and selects the key points. Note that multiple faces can be selected for meshing when using the No Corner Points option. Pick all the faces and then select **FINISH**. The meshing algorithm will be applied to each face.

Specific Commands on this menu:

Element type

Allows the user to select either triangular or quadrilateral elements as the preferred element type; a sub-list is presented (Triangle / Quadrilateral / cancel). The active mesh information window is updated as the element type is selected. If **Quadrilateral** is chosen, but the surface cannot be meshed entirely with quadrilateral shaped elements, triangles are used in the mapped meshing. However, if **Triangle** is chosen, all elements will be triangular.

Diagonal Option

Allows the user to specify the pattern that is to be used for the tri-linear mapping algorithm. This applies if element type is set to triangles. The sub-list (Right / Left / Union Jack / Optimum / cancel) is presented and the information box is updated upon selection of one of the items in the list.

Bi-linear Mapping

Mesh a face using a bi-linear transfinite mapping algorithm. The face to be meshed must be decomposable into four super-edges such that opposing super-edges have the same number of subdivisions. The user can use the **Given Corner**

Points option to specify the four corners in the case of more than four edges on the face.

- Select either **No Corner Points** or **Given Corner Points** from the sub-menu.
- If **No Corner Points** is selected, pick the faces to be meshed and select **FINISH**.
- If **Given Corner Points** is selected, pick the face and the four corner vertices.

Bi-linear Collapsed

Mesh a face using a collapsed bi-linear transfinite mapping algorithm. The face to be meshed must be decomposable into three super-edges such that two of the super-edges have the same number of vertices. If two super-edges have the same number of vertices and the other super-edge has a different number of vertices, the triangular elements will emanate from the vertex at which the two super-edges with the same number of vertices connect.

- Select either **No Corner Points** or **Given Corner Points** from the sub-menu.
- If **No Corner Points** is selected, pick the faces to be meshed and select **FINISH**.
- If **Given Corner Points** is selected, pick the face and the three corner vertices.

Transition Mapping

Mesh a face using a bi-linear mapping with an element size transition using five-noded elements. Two small elements can transition to a single larger element that has a mid-side node on the edge where the two small elements join. Two of the bounding edges must have equal number of segments, while the other two edges must have n and $2n$ segments.

- Select either **No Corner Points** or **Given Corner Points** from the sub-menu.
- If **No Corner Points** is selected, pick the faces to be meshed and select **FINISH**.
- If **Given Corner Points** is selected, pick the face and the four corner vertices.

Tri-linear Mapping

Mesh a face using a tri-linear transfinite mapping algorithm. The face to be meshed must be decomposable into three super-edges.

- Select either **No Corner Points** or **Given Corner Points** from the sub-menu.
- If **No Corner Points** is selected, pick the faces to be meshed and select **FINISH**.
- If **Given Corner Points** is selected, pick the face and the three corner vertices.

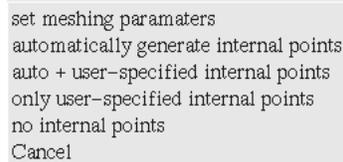
Arbitrary Region

Mesh a face using a general algorithm that can create either all triangular or all quadrilateral shape elements. A complete description of the meshing algorithm is available in Potyondy (1993) and the FRANC3D Concepts & Users manual

contains a summary of this information. The user can define points to be used during meshing or allow the algorithm to generate the interior points. From this sub-menu, Figure 5.48, the user can choose to set the parameters that control the arbitrary region-meshing algorithm, or just proceed to mesh faces with the existing parameters. Reasonable default values exist for generating a relatively coarse mesh. If **set meshing parameters** is chosen, a dialog box is presented, Figure 5.49.

The user is able to set the type of elements for meshing the region—either all triangles or all quadrilaterals. The quadtree parameters allow the user to set the refinement factor and boundary factor. These values describe how much the mesh is refined from the boundary to the inside of the region. The next toggle button—points at quad corners—allows the user to put nodes at the quadtree corner points rather than at the centers. The element merging parameters allow the user to set the merging and smoothing operations that control the final mesh appearance. The mixed mesh cutoff and polygon merge cutoff control the merging of the triangles into quadrilaterals or the splitting of polygons into triangles. The angles between edges of elements can be calculated in parametric or Cartesian space. Templates of quadrilateral crack tip elements can be generated at crack tips in shell models.

The user is prompted to select the face(s) to be meshed. If points are generated automatically, multiple faces can be collected for meshing. Pick the faces and then select **FINISH**. If user-specified points are to be specified, the user must pick a single face and can then pick any number of points on that face. Select **FINISH** after picking the points on the face.



```
set meshing parameters
automatically generate internal points
auto + user-specified internal points
only user-specified internal points
no internal points
Cancel
```

Figure 5.48. The **Arbitrary Region** meshing sub-menu.

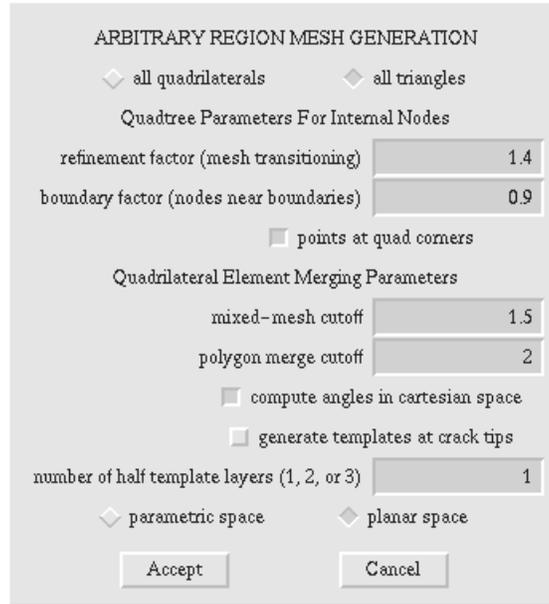


Figure 5.49. The **ARBITRARY REGION MESH GENERATION** dialog box.

all quadrilateral/all triangles

This option controls whether the face will be meshed using all quadrilateral shaped or all triangular shaped elements. Mixed meshes are not supported.

refinement factor

This parameter controls how quickly the mesh transitions from small to large elements.

boundary factor

This parameter controls the mesh density near the boundaries of the face.

points at quad corners

This parameter controls the location of internal points of the mesh. A quad-tree algorithm is used to generate internal points; the mesh points can be chosen to lie at the center of the quad-shapes or at the corners.

mixed-mesh cutoff

This parameter controls the shape and size of the quadrilateral elements.

polygon merge cutoff

This parameter controls the shape and size of the quadrilateral elements.

compute angles in Cartesian space

This parameter controls the angles of the elements. During element formation and smoothing, the internal angles of the element are checked. The angle can be computed in Cartesian space or in the surface parametric space.

generate templates at crack tips

This option controls the formation of a template of uniform shape elements at the crack tip for shell cracks.

number of template layers

This parameter controls the number of uniform elements that are created by the template at the crack tip. Only two or three layers of elements are possible.

Parametric space / planar space

This parameter controls the meshing space. Either a rectangular parametric space or a least squares tangent planar space can be used. The tangent planar space is often better for complex geometric shapes as the bounding geometry is preserved.

Automatic

All unmeshed faces of the sub-region model are meshed. For each face, the following mesh generation algorithms are attempted (in the given order): bi-linear transfinite mapping, collapsed bi-linear transfinite mapping, general triangulation algorithm with automatic interior point generation.

Construction

Activates the **Mesh Construction** menu, Section 5.6.1, which displays commands to edit a surface mesh by adding and deleting individual edges of the mesh.

Clear Mesh

The sub-list (Clear Some Faces / Clear All Faces / cancel) is presented. **Clear Some Faces** deletes the mesh from the faces specified by the user. Select **FINISH** to delete the mesh on these faces. **Clear All Faces** presents a dialog box stating that all meshed surfaces will be cleared. The user can select **Cancel** if he/she hit this menu button by accident. Otherwise, all surface meshing is cleared.

Hilight Unmeshed Faces

Hilight all the unmeshed faces. The geometric surface is hilighted and then all unmeshed faces on the surface are indicated by hilighting the edges; a surface can be subdivided into several faces.

return

Exits the menu.

5.6.1 Mesh Construction Menu

The **MESH CONSTRUCTION** menu, Figure 5.50, has commands for editing surface meshes at the level of individual elements. This provides tedious, but precise control over the mesh.

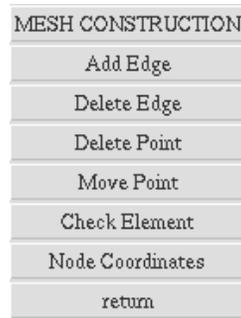


Figure 5.50. The **MESH CONSTRUCTION** menu.

Concepts for this menu:

Mesh editing - Finite and boundary element programs require 3- and 4- sided elements. However, FRANC3D treats the element mesh using the more general concept of faces, edges, and vertices that can be edited in the same fashion as the faces, edges, and vertices of the geometry model. However, the final discretization of the surface must include only three and four sided mesh faces.

Specific Commands on this menu:

Add Edge

Adds a edge to the surface.

- The user is prompted to select two points in the mesh model.
- Adds an edge between the points.

Delete Edge

Deletes mesh level edge that does not have a parent edge at a higher hierarchical level.

- The user is prompted to select an edge in the mesh.
- Deletes the edge and merges the adjacent elements into a single element.

Delete Point

Deletes a point or mesh node and all adjacent edges. The point is deleted if it does not have a parent vertex at a higher hierarchical level.

- The user is prompted to select a node in the mesh.

Move Point

Moves a point or mesh node and all adjacent edges. The point is moved if it does not have a parent vertex at a higher hierarchical level.

- The user is prompted to select a node in the mesh.
- Select a new point on the surface where the node is to be placed.

- The exact coordinates of the new point can be specified by selecting the **Key In Coordinates** button before selecting the second point and entering the coordinates in the dialog box.

Check Element

Allows the user to verify that a mesh level face is a valid three- or four-sided element.

- The user is prompted to select an element or mesh level face.
- The element information is printed in the message box and on the terminal window.

Node Coordinates

Allows the user to print the global Cartesian coordinates of a surface mesh node.

- The user is prompted to select a node of the surface mesh.
- The node coordinates are printed on the terminal window.

return

Exits the menu.

5.7 The Mesh Volumes Menu

The **MESH VOLUMES** menu, Figure 5.51, contains commands for creating and deleting volume meshes.

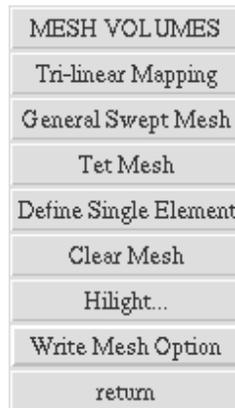


Figure 5.51. The **MESH VOLUMES** menu.

Specific Commands on this menu

Tri-linear Mapping

Mesh a region of the Subdomain Model using a tri-linear, transfinite mapping algorithm. The user is prompted to select the region to be meshed. The region to be meshed must be decomposable into six super-faces such that opposing super-

faces have topologically identical meshes. The meshes on each super-face must be comprised of quadrilaterals only. The resulting mesh is composed of brick elements. The corner points refer to the corners that define the six super-faces. (The region must be a topological cube.)

General Swept Mesh

Creates a volumetric mesh by sweeping a surface mesh on a given face to a topologically identical surface mesh on another face. Each surface mesh can be comprised of a general collection of triangles and quadrilaterals. The intervening surface mesh must be composed of consistent quadrilaterals. The resulting mesh is comprised of bricks and wedges. The user is prompted to select the region to be meshed followed by four vertices that define the sweep direction.

Tet Mesh

Mesh a region of the Subdomain Model using an arbitrary region advancing front tetrahedral meshing algorithm. The user is prompted to select the region to be meshed. A volume mesh is created using tetrahedral elements based on an advancing front meshing algorithm. The user is prompted to select the region to be meshed if there is more than one region. The user can control the refinement factor, Figure 5.52, where a smaller number means fewer elements and a more rapid transition from small to larger elements.

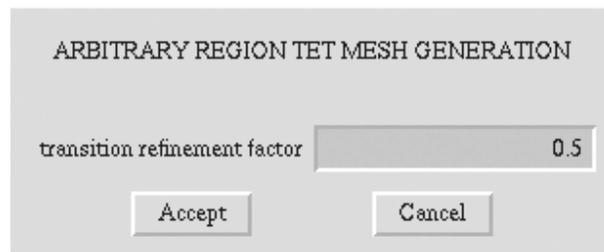


Figure 5.52. The **Advancing Front Parameters** dialog box.

transition refinement factor

Controls the number of interior points that are added by controlling the refinement of the octree. A larger number means that more elements are generated.

Define Single Element

Attempts to create a single element in the model.

Clear Mesh

The sub-list (One Region / All Regions / cancel) is presented. Selecting **One Region** prompts the user to select the region to clear. If the user selects **All Regions**, a dialog is presented asking the user if he/she is sure that all regions are to be cleared – select **Ok** to delete all the volume elements from all regions.

Hiligh...

The sub-list (unmeshed Regions / non-triangle Faces / poor triangle Faces / cancel) is presented. Selecting **unmeshed Regions**, causes all regions that do not have a volume mesh to be highlighted. The edges of the region are highlighted in white. Select **FINISH** to remove the highlighting. Selecting **non-triangle Faces** or **poor triangle Faces** causes faces of the surface mesh to be highlighted.

Write Mesh

Activates the **Write Mesh** menu, Section 5.7.1, which displays commands to write the surface mesh for and read the volume mesh from the stand-alone version of JMESH.

return

Exits the menu.

5.7.1 Write Mesh Menu

The **WRITE MESH** menu, Figure 5.53, has commands for writing surface mesh information and reading volume mesh information to and from the stand-alone version of JMESH (the tet meshing program).

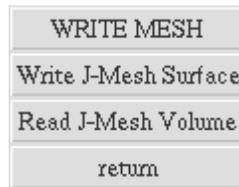


Figure 5.3. The **WRITE MESH** menu.

*Specific Commands on this menu:***Write J-Mesh Surface**

Write the surface mesh information as input to JMESH.

Read J-Mesh Volume

Read the volume mesh information from JMESH.

return

Exits the menu.

6 Read/Write Analysis Files Menu

Selection of the **Read/Write Analysis Files** button from the main menu pops up the **READ/WRITE ANALYSIS FILE** menu, Figure 4.4. This menu contains commands

for reading and writing data files formatted for or by finite and boundary element programs. Each of the options is described in this section.

6.1 STAGS I/O

Selecting the first menu button pops up the **STAGS I/O** menu, Figure 6.1. This menu contains commands for reading and writing STAGS input and output files as well as defining special STAGS input. STAGS is a general purpose shell finite element program developed by Lockheed.

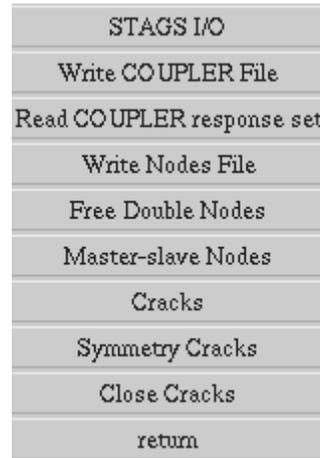


Figure 6.1. The **STAGS I/O** menu.

Specific Commands on this menu

Write COUPLER File

Write a STAGS coupler file for performing shell analyses with the STAGS program. A dialog box is presented, Figure 6.2 allows the user to define specific commands in performing STAGS analysis.

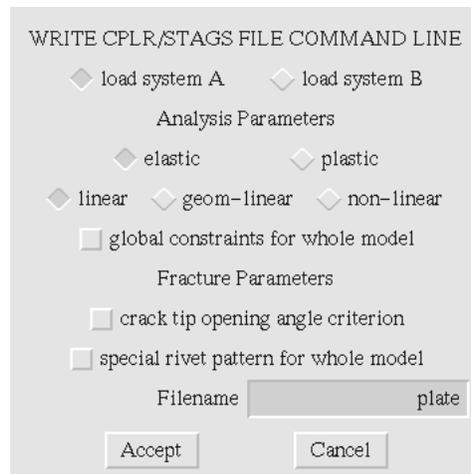


Figure 6.2. The **Write CPLR/STAGS File Command Line** menu.

Read COUPLER response set

Read the STAGS response information into FRANC3D. A list of all response sets is presented in a list, from which the user can choose one. If the response set and the FRANC3D model do not match, the user is asked if he wants to continue, otherwise the data is read and can be viewed through the **Visualize Analysis Results** menu.

Write Nodes File

This file is used for mapping responses during stages of shell crack growth simulation using STAGS. Writes a file consisting of node numbers and their global coordinates. This file also contains additional information for nodes along the crack, defining which side the node is on and whether a node is at a crack tip.

Free Double Nodes

Free double nodes.

Master-slave Nodes

Define master-slave nodes.

Cracks

Define cracks.

Symmetry Cracks

Define symmetry cracks.

Close Cracks

Close cracks.

return

Exits the menu.

6.2 BES I/O

Selecting the first menu button pops up the **BES I/O** menu, Figure 6.3. This menu contains commands for reading and writing BES input and output files as well as running BES from FRANC3D.

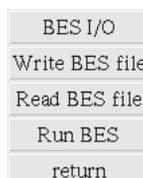


Figure 6.3. The **BES I/O** menu.**Specific Commands on this menu****Write BES File**

Writes geometry and boundary conditions formatted for the BES boundary element program. The user has the option of writing out the file with either linear or quadratic elements. Select either linear or quadratic from the sub-menu and then enter the .bes file name in the file selector box. The default file name is the current FRANC3D model (file) name.

Read BES File

Reads results from the BES boundary element analysis program.

- Displays a list of all .besout files found on the current directory.
- Prompts for selection of the file.
- Reads displacement and tractions from the .besout file.
- Searches for a .con (contour data) file of the same name. If one is found the stress data is read from that file.

Run BES

Presents a dialog box, Figure 6.4, for setting the parameters to run BES. Selecting **Accept** with BES defined as local will invoke BES on the local workstation with the supplied parameters. If BES is to be run on a remote machine, the command line arguments that should be used are dumped to a file.

RUN BES COMMAND LINE PARAMETERS

.bes file name	<input type="text" value="cube.bes"/>	
solver memory	<input type="radio"/> in_core	<input type="radio"/> out_of_core
create flex matrix	<input type="radio"/> NO flex sets	<input type="radio"/> flex sets
element shape functions	<input type="radio"/> linear	<input type="radio"/> quadratic
element integration	<input type="radio"/> conforming if possible	<input type="radio"/> all non-conforming
analysis results	<input type="radio"/> NO stress output	<input type="radio"/> stress output
e-mail when BES is finished	<input type="radio"/> NO mail	<input type="radio"/> send mail
e-mail address	<input type="text" value="none"/>	
run BES on computer	<input type="radio"/> local	<input type="radio"/> remote
file name (if remote)	<input type="text" value="runbes"/>	

Figure 6.4. The **Run BES** dialog box.*bes file name*

The default file name for the bes file is supplied based on the name of the model.

solver memory

This option determines whether BES runs using the in-core Gauss elimination solver or the out-of-core QR solver. This option depends on the size of the model and available computer memory resources.

create flex matrix

This option determines whether BES creates the influence matrix for unit tractions on the crack surface, meaning multiple back-substitutions. No flex sets is the usual choice unless one is doing hydraulic fracture simulations.

element shape functions

This option determines whether BES uses linear or quadratic shape functions in the elements. Linear is the usual choice.

element integration

This option determines whether BES uses conforming element integration (at a node) or non-conforming integration (inside an element near a node).

analysis results

This option controls whether BES writes the stresses to a file. By default, displacements and tractions are written to a .l.besout file. The stresses can be suppressed or written to a .l.con file.

e-mail when BES is finished

This option determines whether BES sends e-mail to the address entered in the next entry field when BES has finished the analysis.

run BES on computer

This option determines whether BES runs on the local workstation from within the interactive FRANC3D session or on a remote workstation in which case a file is created with the appropriate “runbes” command including the command line options based on the dialog box settings. The file name is entered in the next entry field.

return

Exits the menu.

6.3 3D FEM I/O

Selecting the second menu button pops up the **3D FEM I/O** menu, Figure 6.5. This menu contains commands for reading and writing FEM input and output for several finite element programs.

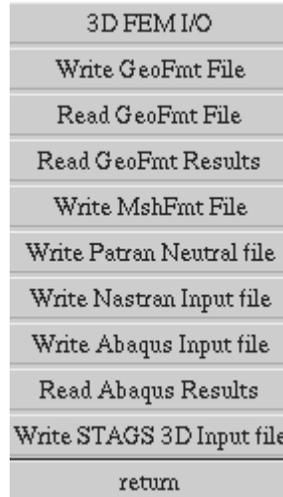


Figure 6.5. The **3D FEM I/O** menu.

Specific Commands on this menu

Write GeoFmt File

Write a geometry file for an in-house FEM program.

Read GeoFmt File

Read the geometry from an in-house FEM program.

Read GeoFmt Results File

Read the results from an in-house FEM program.

Write MshFmt File

Write a mesh file for an in-house FEM program.

Write Patran Neutral File

Write a Patran neutral file consisting of the nodes and their coordinates followed by the elements and their connectivity. No simulation attributes (boundary conditions, etc.) are included.

Write Nastran Input File

Write a Nastran ascii file consisting of the nodes and their coordinates followed by the elements and their connectivity. No simulation attributes (boundary conditions, etc.) are included.

Write Abaqus Input File

Write a Abaqus ascii input file consisting of the nodes and their coordinates followed by the elements and their connectivity.

Read Abaqus Results

Read a Abaqus ascii results file consisting of the nodal displacements.

Write STAGS3D Input File

Write a STAGS ascii input file consisting of the nodes and their coordinates followed by the elements and their connectivity.

return

Exits the menu.

6.4 MRP I/O

Selecting the fourth menu button pops up the **MRP I/O** menu, Figure 6.6. This menu contains commands for reading and writing MRP files. These MRP files contain the mesh information and displacements for the entire model. The MRP file is used to define initial values.

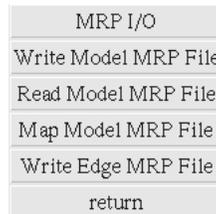


Figure 6.6. The **MRP I/O** menu.

Specific Commands on this menu**Write Model MRP File**

Write a MRP file for the model. The user is prompted to enter the file name in the file selector box.

Read Model MRP File

Read a MRP file. The user is prompted to select the MRP file name from the file selector box.

Map Model MRP File

Maps a MRP to the current model.

Write Edge MRP File

Write a MRP file for an edge.

return

Exits the menu.

7 Automatic Propagation

Selection of the **Automatic Propagation** button from the main menu pops up the **AUTOMATIC PROPAGATION** menu, Figure 4.5. This menu contains commands for performing automatic crack growth simulations. The menu options are described in this section.

7.1 Select Crack Growth Model

A dialog box, Figure 7.1, is presented which allows the user to set parameters that are used for saving the files for each step of analysis, determining stress intensity factors, propagating cracks, and running the boundary element program. From this top-level dialog box, selecting the appropriate buttons on the bottom can access sub-dialog boxes. The number of crack growth steps and the current step number can be entered in this dialog box. The following options are available:

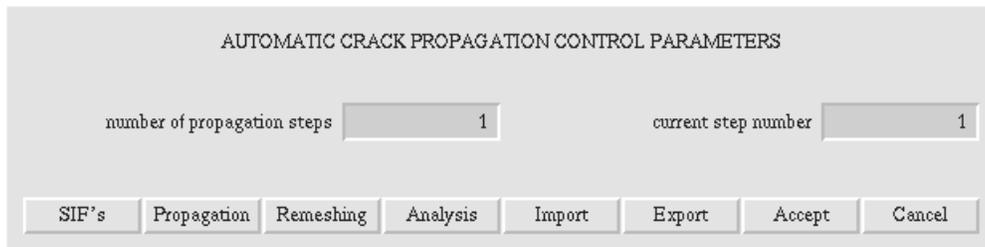


Figure 7.1. The **AUTOMATIC CRACK PROPAGATION MODEL** dialog box.

number of propagation steps

Allows the analyst to specify the number of crack growth steps to take in an automated analysis.

current step number

Assigns a step number for the current crack growth step. Normally the program manages this value. This option allows the analyst to override the normal sequential numbering of analyses steps. Note, however, if this number is set lower than the current step number, information may be overwritten.

SIF's button

Selecting the SIF's button causes the dialog box shown in Figure 7.2 to be displayed. The number of points along the crack front can be set and the plot of the stress intensity factors can be turned on or off.

Propagate button

Selecting the Propagate button causes the dialog box shown in Figure 7.3 to be displayed. The method for computing crack growth direction and the maximum crack advance can be defined for each crack growth step.

Remeshing button

Selecting the Remeshing button causes the dialog box shown in Figure 7.5 to be displayed. The number of elements along the crack front can be defined along with the transitioning factor.

Analysis button

Selecting the Analysis button causes the dialog box shown in Figure 7.7 to be displayed. The parameters to control the stress analysis using the boundary element code, BES, can be set here.

Import button

Selecting the Import button brings up the file selector box. The user can import a previously defined crack growth model data.

Export button

Selecting the Export button brings up the file selector box. The user can export a the crack growth model data for later import.

7.1.1 The Stress Intensity Factors Dialog Box

The stress intensity factors dialog box, shown in Figure 7.2, allows the analyst to set the parameters used when computing stress intensity factors. The following options are available:

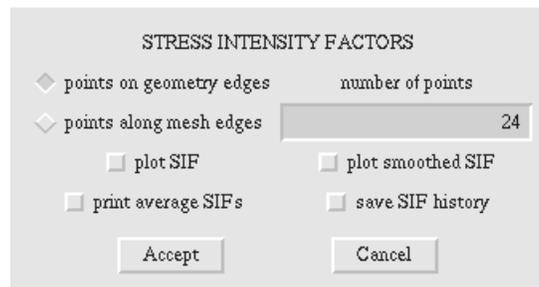


Figure 7.2. The **STRESS INTENSITY FACTORS** dialog box.

points on geometry edges / points along mesh edges

This option selects the technique used for determining the locations along the crack front where stress intensity factors will be computed. Points on geometry edges, the default technique, divides the crack front into a number of equal sized segments, and evaluates the stress intensity factors at the ends of these segments. Points along mesh edges evaluates the stress intensity factors at the nodes of the elements adjacent to the crack front.

number of geometry points

If the geometry points option is selected for evaluating stress intensity factors, this option sets the number of points where they will be computed. The default value is 24.

plot SIF/plot smoothed SIF

These options determine if plots of stress intensity factor distributions will be displayed for each crack growth step. The plot SIF option plots "raw" SIF values as computed. The plot smoothed SIF option plots this data after it has been "smoothed" with a running average algorithm.

print average SIFs

Prints the average SIF along the crack front in an acknowledge box.

save SIF history

If this option is chosen the SIF values are saved for this crack front. The SIF history is required to compute fatigue life later.

7.1.2 The Propagation Options Dialog Box

The propagation options dialog box, shown in Figure 7.3, allows the analyst to set the method used for computing crack growth direction and the maximum crack advance for each crack growth step. The following options are available:

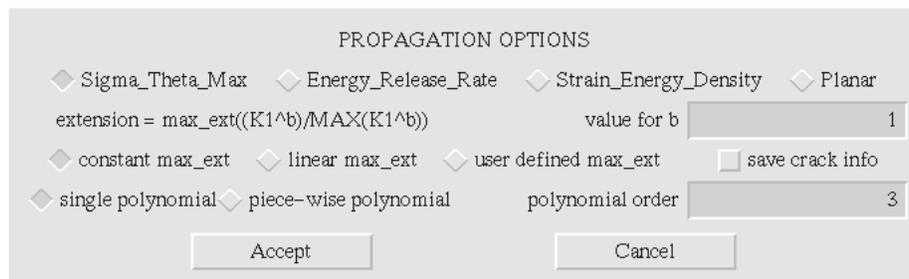


Figure 7.3. The **PROPAGATION OPTIONS** dialog box.

Sigma Theta Max/Energy Release Rate/Strain Energy Density/Planer

Specifies the technique used to determine the direction of crack propagation for all points along the crack front. For the first three options, the mode I and II stress intensity factors are determined for a plane perpendicular to the crack front passing through the point where the angle of crack growth is being determined. The sigma theta max option, which is recommended, grows the crack towards the direction of maximum normal stress. The energy release rate option grows the crack in the direction where the energy release rate is maximized. The strain energy density option grows the crack in the direction where the strain energy density is minimized. The planer option forces the crack to remain planer.

value for b

Specifies the value to use for b in the crack increment equation. The amount of crack growth for any one point is the ratio between the stress intensity factor at this point and the maximum value raised to a power. b is the specified power.

constant max_ext/linear max_ext/user defined max_ext

Specifies the technique used to determine the maximum crack extension at each crack step. The constant maximum extension option grows the crack a constant amount at each crack step (Figure 7.4a). The linear maximum extension option grows the crack with a linearly increasing crack extension at each step (Figure 7.4b). The first step of propagation uses the initial value as the maximum extension; subsequent steps use a value that is a function of the initial value, the linear slope and the propagation step number. The user defined maximum extension option grows the crack with extension amounts specified by the user for each step. A simple spreadsheet listing the maximum extension per step of propagation is provided (Figure 7.4c).

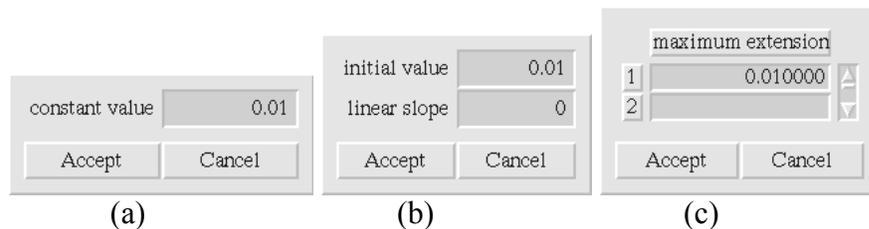


Figure 7.4. Constant, linear, and user-defined maximum extension dialog boxes.

save crack info

This option determines if crack front data will be stored in a separate file for each crack step.

single polynomial/piece-wise polynomial

Once crack growth points have been determined, they are fit with a polynomial curve in order to smooth out numerical "noise" and to have an algebraic crack front description that can be extrapolated or interpolated to find the intersection of the crack front with free surfaces. This option specifies if the entire crack front will be fit with one polynomial, or if the fit will be done piecewise for each of the curves that are used to define the geometry of the crack front.

polynomial order

Specifies the order of the polynomial used to fit the predicted crack growth points.

7.1.3 The Crack Surface Remeshing Dialog Box

The crack surface remeshing options dialog box, shown in Figure 7.5, specifies the number of elements to be placed along the crack front for each crack step. It also specifies a factor that controls the crack face mesh density. The following options are available:

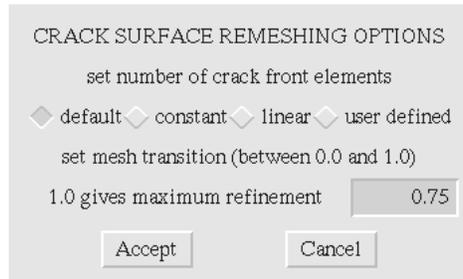


Figure 7.5. The **REMESHING OPTIONS** dialog box.

default/constant/linear/user defined

Specifies the technique used to determine the number of element placed along the crack front for each crack growth steps. The default option specifies that a built in heuristic algorithm is to be used, which determines the number of crack front elements based on the current element sizes and the crack growth increment. The constant option keeps the number of crack front elements constant (Figure 7.6a). The linear option specifies that the number of crack front elements will increase at a linear rate for each crack step (Figure 7.6b). The first step of propagation uses the initial value for the number of crack front elements; subsequent steps use a value that is a function of the initial value, the linear slope and the propagation step number. The user defined option uses a number of crack front elements specified by the analyst at each step. A simple spreadsheet listing the maximum extension per step of propagation is provided (Figure 7.6c).

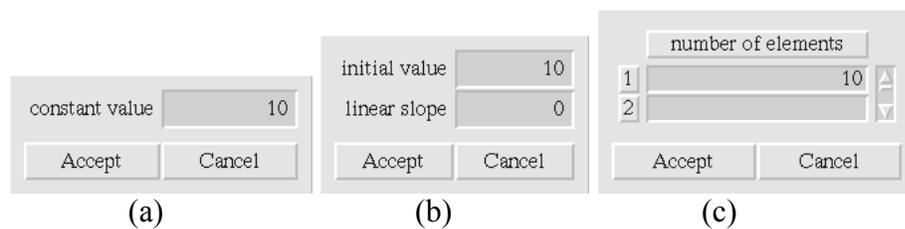


Figure 7.6. Constant, linear, and user-defined crack front elements dialog box.

mesh transition

Specifies a factor used to determine the mesh refinement on the crack faces.

7.1.4 The Stress Analysis Options Dialog Box

The stress analysis options dialog box, shown in Figure 7.7, sets parameters used to control the stress analysis using the boundary element code, BES. The following options are available:

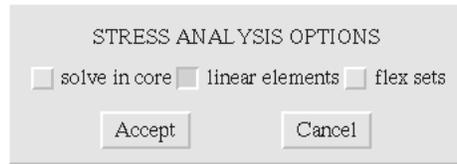


Figure 7.7. The **STRESS ANALYSIS OPTIONS** dialog box.

solve in core

Determines if the analysis will be performed using virtual memory (in core) or with explicitly managed coefficient matrix files. The in core option will be faster for very small problems, but will work only for very small problems.

linear elements

Determines if linear or quadratic displacement variation elements will be used in the boundary element analysis. Linear elements are recommended. Quadratic elements require much more time and computational resources for a solution.

flex sets

Determines if a normal stress analysis will be performed or if the program will generate an influence function solution used for fluid driven crack analyses.

7.1.5 The File Name Dialog Box

The file name dialog box, shown in Figure 7.8, allows the analyst to specify the name of the file that contains the crack growth options for a batch mode automatic propagation.

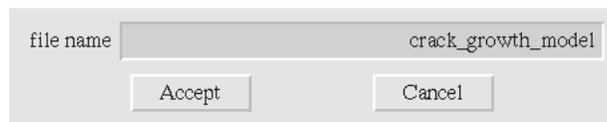


Figure 7.8. The **File name** dialog box.

To run in batch mode, the initial model must be prepared for analysis. This means defining and creating the initial crack, meshing the model, and defining and attaching the boundary conditions. Once this is completed, the restart file must be saved and the crack growth model defined. The analyst can then exit the interactive FRANC3D session and start the automated analyses from the terminal window. Assume that the restart file is named `geometry_model.fys` and the crack growth model information was saved to a file called `crack_growth_model`. The command to start up `franc3d` in batch mode with these files is:

```
prompt> franc3d -b -f geometry_model.fys -c crack_growth_model &
```

NOTE: It is important that BES executables exist in the PATH or current directory.

7.2 Automated Analyses Using BES

Start the automatic analysis process from FRANC3D using the crack growth model data defined above. Note that the BES executables (cge_bes or cocqr_bes) should be in the current directory or the PATH environment variable should include their location.

7.3 Propagate All Cracks

Propagate all cracks based on the crack growth model defined above. This is an interactive method for automatic propagation of all cracks in the model. The amount of extension for all cracks is scaled based on the ratio of their mode I stress intensity factor and the maximum K_I for all cracks.

7.4 Rebuild Mesh Model

Remesh the crack and surrounding faces. This is an interactive method for rebuilding the mesh on and around the cracks that were propagated in the previous step, **Propagate All Cracks**; it is completely automatic. The crack faces are meshed first. The non-crack faces adjacent to the crack are meshed last. The number of elements along the crack front as well as the rate of mesh transitioning is controlled by the parameters set in the remeshing options dialog box.

7.5 Attach Crack Face BCs

Attach boundary conditions to the new crack faces after propagation based on the boundary conditions applied to the previous crack faces. This only is applicable for constant uniform traction or displacement.

return

Exits the menu

8 Visualize/Analyze Results

The **VISUALIZE/ANALYZE RESULTS** menu, described in Section 4.5, contains commands to post-process analysis results including displaying the deformed structure, displaying color contours of stresses, displaying line plots, and analyzing fracture stability. These options are described in this section.

8.1 Deformation & Contour

The first command on the menu is **Deformation & Contour**. This option allows the user to display the deformation and color contours of the analysis results. The results are displayed in a separate window that is similar to the modeling window. It has different pull-down menus, however, and supports color contour labels, see Figure 8.1 and 8.2.

The user is required to enter an initial magnification factor on the presented keypad. Selecting **Enter** on the keypad invokes the deformation/contour window with the deformed mesh and undeformed boundary edges shown by default. The deformed shape is shown in red while the undeformed shape is shown in the default gold-orange color.

The deformation/contour window is capable of displaying both deformed shapes and color contours. The edges and faces can be turned on and off as desired and contour results changed as desired by simply clicking on the pull-down menus in the top left corner of the window. Note that if a pull-down menu is not needed, click on the pull-down menu button and the menu will disappear.

There are three pull-down menus. The first is **View** and is the same as that of the main modeling window. The second is **Deformation**. Under **Deformation** is **Alter magnification** and **Display control**. **Alter magnification** pops up a dialog box and allows the magnification factor to be changed. The magnification factor affects the display of the deformed shape and depends on the displacement as well as the view. The current view is redrawn upon selecting **Accept**. **Display control** provides for different views of the deformed and undeformed shape. Under **Display control**, any or all of the options can be checked, such as Undeformed EDGES with Deformed boundary EDGES, depending on what the user wishes to see.

The third pull-down menu is **Contour**. Under **Contour** is **Response value**, **Alter Range**, **Display Control**, and **Clear Contours**. Selecting **Response Value** gives a list of responses that can be contoured. Only one response value can be checked at any one time. In order to view the contours, click on the **Display Control** button and check the desired option. The contours can be displayed on the undeformed or deformed model in unsmoothed or smoothed formats. The **Alter Range** menu entry allows the color contours to be restricted to a certain range of values. Selecting **Alter Range** causes a dialog box to be displayed. The minimum and maximum values for contouring can then be entered. The current color contours are redrawn using the new range.

8.2 Surface Line Plot

Display line plots of analysis results. The results are displayed in separate windows as follows:

- Collect multiple lines on the surfaces of the model. Each line requires a starting and ending point. The lines do not have to be connected or on the same face.
- Invokes the line plot window; the model is shown with the collected lines. Note that the analysis results will be plotted along the normalized length of the total line.

The line plot window, Figure 8.3, contains three pull-down menus, **View**, **File** and **Data**. The **View** pull-down menu is the same as for the main modeling window. Under **Data** is **Response Value**. Selecting **Response Value** causes a list of all the response data types to be displayed. The user can select an item from this list to be plotted. The data is plotted in an adjacent x-y plot window. The response value is plotted against normalized distance along the line. The normalized distance starts from the first point selected when picking the set of lines.

The other pull-down menu is **File**. Under **File** is **ASCII** and **PostScript**. The x-y data can be written to a file in ascii or postscript format.

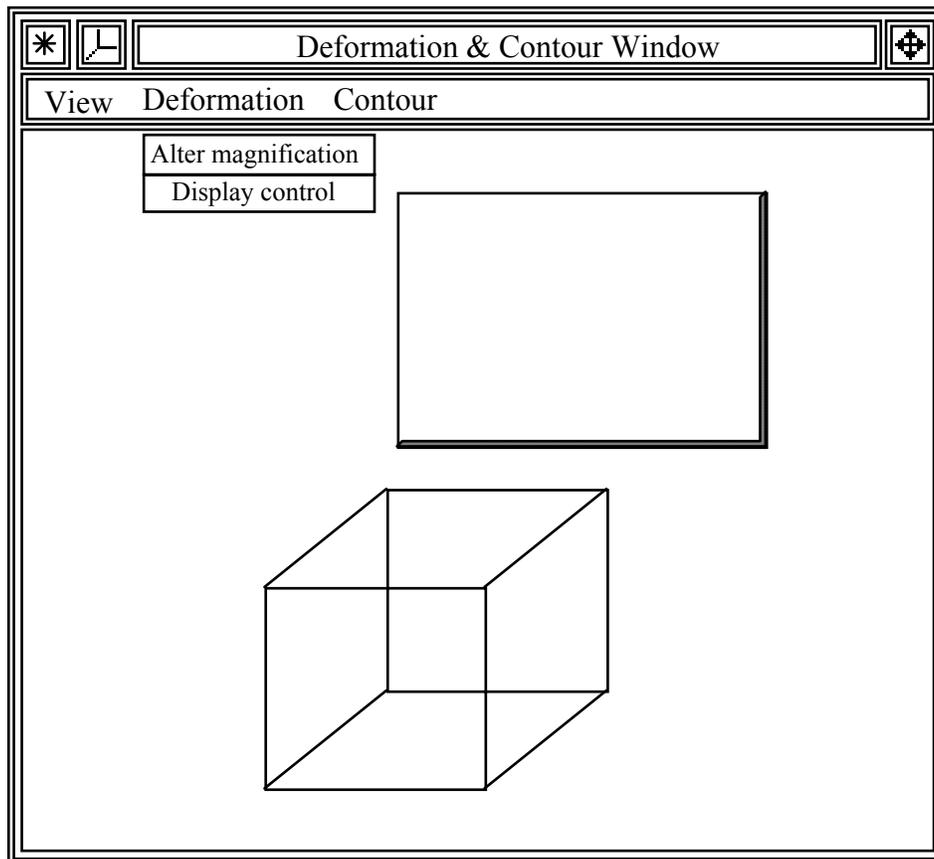


Figure 8.1. The **DEFORMATION/CONTOUR** window showing the **Deformation** pull-down menu and **Display Control** dialog box.

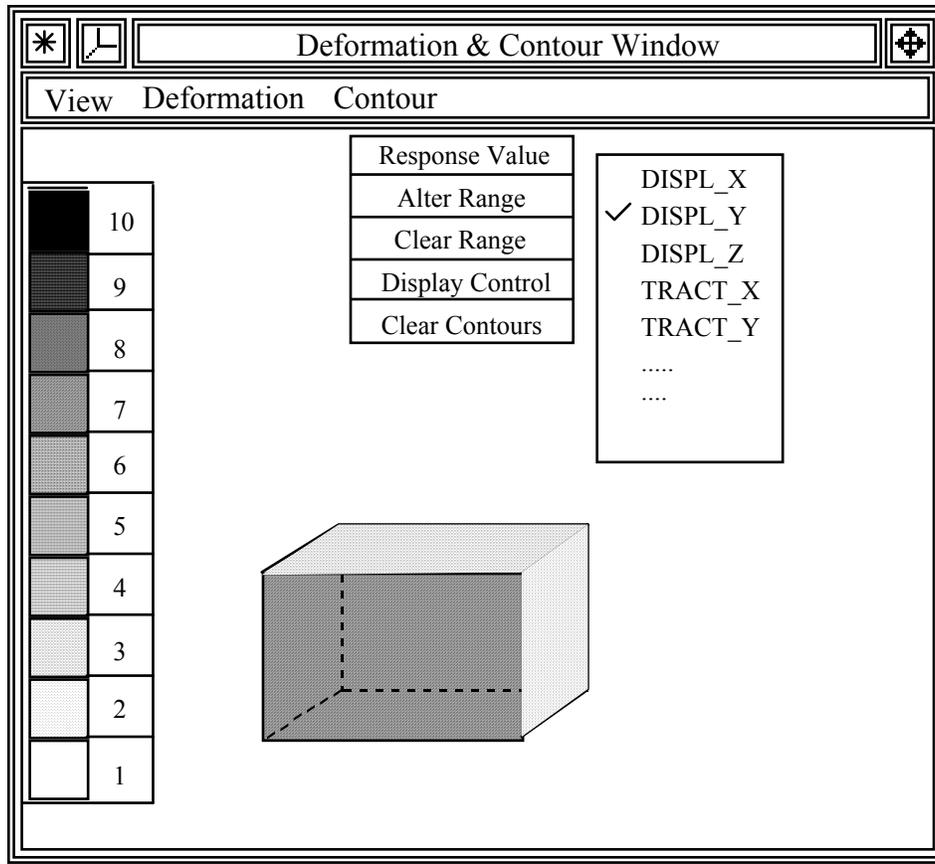


Figure 8.2. The **DEFORMATION/CONTOUR** window showing contours.

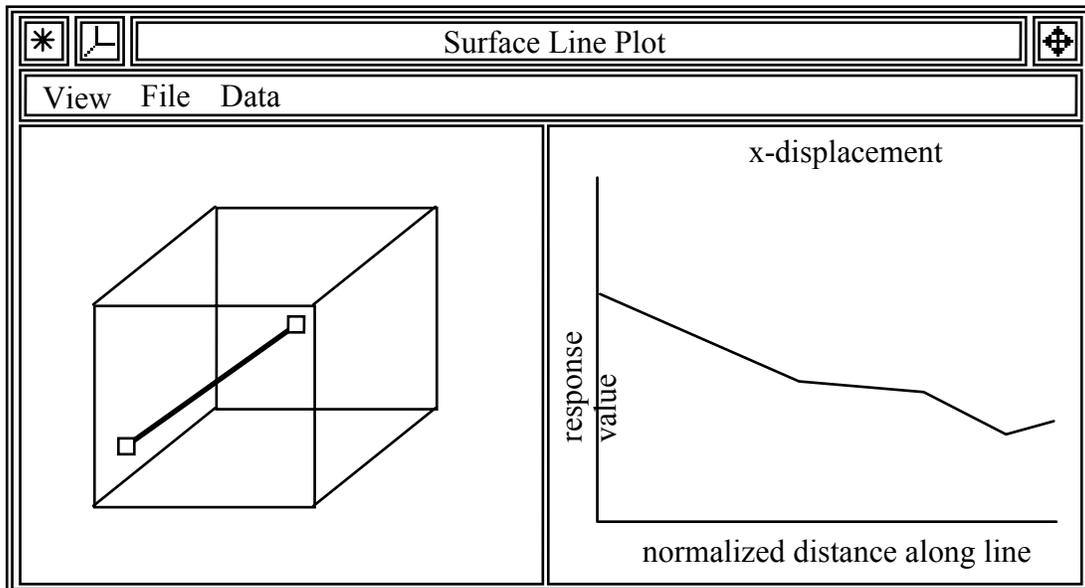


Figure 8.3. The **LINE PLOT** window.

8.3 Point Information

Display point information on the terminal window.

- The point may be an arbitrary surface point or a vertex. The vertex information can be either with respect to a given element or the average from all surrounding elements. Results are interpolated for points not corresponding to vertices.
- The data is printed on the terminal window.

8.4 3D Fracture Analysis Menu

The **3D FRACTURE ANALYSIS** menu, Figure 8.4, has commands for computing stress intensity, extending the geometry of cracks, and computing stress intensity factor history and fatigue life.



Figure 8.4. The **FRACTURE ANALYSIS** menu.

Specific Commands on this menu:

LEFM SIFs

Computes stress intensity factors for a selected crack front. The SIF's are computed using the displacement correlation technique. SIF's are computed at a requested number of points along the front, and may be displayed both in raw form and as a smoothed curve, where smoothing is done using a moving point linear regression.

The sub-list (compute SIFs for all cracks / compute SIFs for selected crack / display SIFs for selected crack / cancel) is presented. If the first option is chosen, stress intensity factors for all crack fronts are computed. If the second option is chosen, the user is prompted to select the crack front edge and then the crack tip

vertex that will be the origin of the plotted SIFs versus crack front length. The third option displays previously computed SIFs once the user selects a crack front edge.

For the first two options, a dialog box is presented, Figure 8.5. The user is able to calculate stress intensity factors (SIFs) at either the mesh nodes along the row of elements at the crack front or at arbitrary subdivision points along the crack front geometry. The geometry points should probably be used in most cases. When geometry points are used, the number of subdivision points along the crack front can be set. The default is 24, but for long curved crack fronts, more points may be needed. The user has the option of plotting the raw data as well as the smoothed data. Separate x-y plots are displayed for each of the three stress intensity factor modes. The displayed curves are plotted against the normalized length of the crack front with the zero point corresponding to the two red boxes at one end of the crack front. Finally, the user can display the average SIFs and save the SIF history for later use in computing fatigue life. See Section 7.1.1 for a better description of the dialog box options.

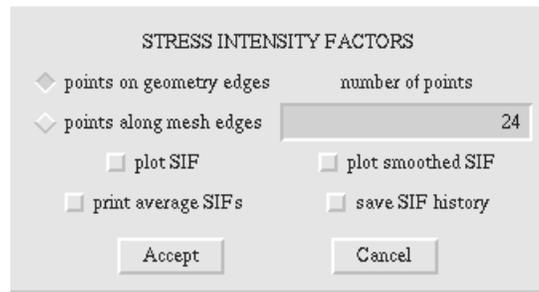


Figure 8.5 The **STRESS INTENSITY FACTORS** dialog box.

Propagate Selected Crack

Propagates a crack front of a three dimensional crack. This option should not be used for shell crack propagation. Shell cracks can be propagated through the **Shell Crack** menu entries, Section 5.1.1.1. A sub-menu is presented, Figure 8.6, which allows the user to calculate the new crack front points, view the calculated and fitted new crack front points, add the new crack front edges, or add and tear the edges and faces to complete the propagation. After selecting one of the sub-menu entries, the user is prompted to select the crack front edge to be propagated. Note that if there are multiple cracks and multiple crack fronts, it is up to the user to determine the proper amount of extension for each crack front assuming that there is some interaction.

```

determine new front points
read crack increments from file
read front points from file
display new/fitted front points
add and tear edges and faces
cancel

```

Figure 8.6. The **Propagate Crack Front** sub-menu.

- **determine the new front points** requires that the user select the model for crack propagation direction and extension. This is done by selecting the model from the dialog box shown in Figure 8.7. The direction is calculated using the standard equations from two dimensional crack growth theories. These include the maximum tangential stress, the maximum energy release rate, and the minimum strain energy density. A planar model is also available for those cases when the user wishes to force the propagating crack to remain in the same plane. Based on the model of extension chosen, another dialog box is presented to retrieve the extension parameters, Figure 8.8.

There are two extension models available, but only one of the corresponding dialog boxes is shown here. The extension is based on the ratio of the mode 1 stress intensity factor (K_I) at a point on the crack front and the maximum value of K_I along the crack front. The maximum extension is used to scale the amount of extension to reasonable amount based on the geometry, the number of steps that the user wants to take, and the accuracy of the solution. In general, many small steps should give more accurate results than a few large steps. The user can define a different crack extension model outside of FRANC3D, predict the crack growth increments along the crack, store them in a file, and then read these values using the next sub-menu entry. See Section 7.1.2 for a description of the dialog box options.

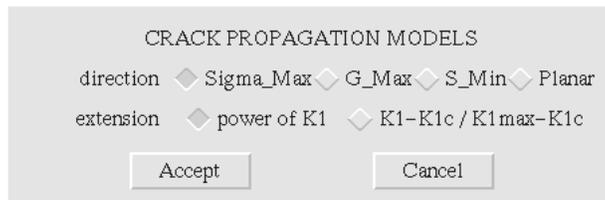


Figure 8.7. The **CRACK PROPAGATION MODELS** dialog box.

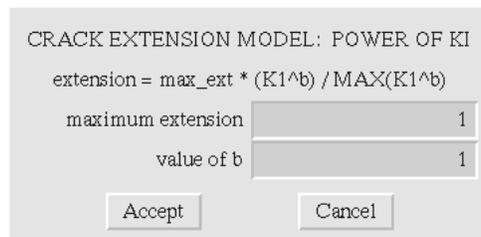


Figure 8.8. The **CRACK EXTENSION MODEL** dialog box.

- **read crack increments from file** allows the user to read in the crack growth increments for the subdivision points along the crack front. This means that the user can predict crack growth extension using a crack growth model that is not included in FRANC3D. The direction of propagation is still based on the previously selected model for crack growth direction.
- **read front points from file** allows the user to read in the new crack front points. This means that the user can predict crack growth direction and extension using a crack growth model that is not included in FRANC3D.
- **display new/fitted front points** allows the user to redisplay the determined crack front points or the fitted crack front points. A dialog box, Figure 8.9, presents the option of showing either set of points. If the fitted points are to be shown and they have not been calculated or are to be re-calculated, another dialog is presented. This dialog box is not presented for internal crack fronts; in that case, a set of Hermitian polynomials of third order is used automatically. For surface crack fronts, polynomial fitting using a least squares approach is performed. A second dialog box, Figure 8.10, is presented to obtain the polynomial order and to allow the user the option of fitting the points with a single curve or with a set of curves. Note that a value of zero for the polynomial order, simply uses the determined points without any fitting. See Section 7.1.2 for a description of the dialog box options.

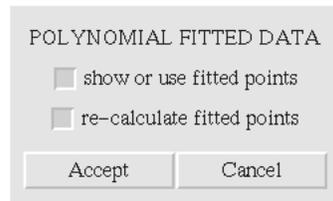


Figure 8.9. The **POLYNOMIAL FITTED DATA** dialog box.

- **add and tear edges and faces** adds any necessary edges from the old crack tips to the new crack tips and the new crack front edges, creates faces from the new and previous edges, and then tears the faces and necessary edges to produce an new crack geometry.

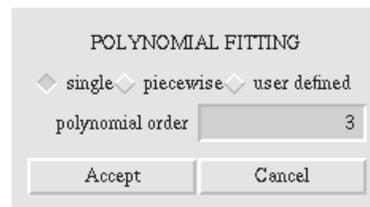


Figure 8.10. The **POLYNOMIAL FITTING** dialog box.

Propagate All Cracks

Propagates all crack fronts if the new fitted front points have been computed for all the crack fronts.

Show SIF History

Presents a sub-menu allowing the user to choose between two methods of determining the stress intensity factor history as well as an option for presenting an evaluation of the mode II based error. If the mode II SIF increases from the previous to the current step, this might indicate an error in the computations.

maximum KI vs area:

The user is prompted to select the crack front edge. A line plot showing the history of maximum mode I stress intensity factors versus the square root of the crack area for all the steps of crack propagation up to the current crack configuration is displayed, Figure 8.11.

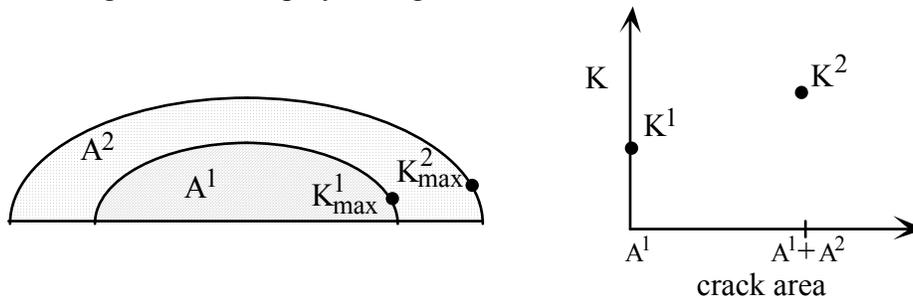


Figure 8.11 Crack surface area (A) versus maximum mode I stress intensity factor (K) for successive steps of crack growth.

along a defined path:

The user is prompted to select the crack front edge and then select two points on the screen; these two points project into the window. A plane is generated using the two vectors. The crack front edges are intersected with this plane to define the path along the crack surface. A line plot showing the history of mode I stress intensity factors versus the crack front advance for all the steps of crack propagation up to the current crack configuration is displayed. The distance between successive crack front intersection points defines crack front advance. The stress intensity factor is computed at these points also, Figure 8.12.

Predict Fatigue Life

The user can compute fatigue life based on the stress intensity factor history and material parameters as described in Section 8.5.

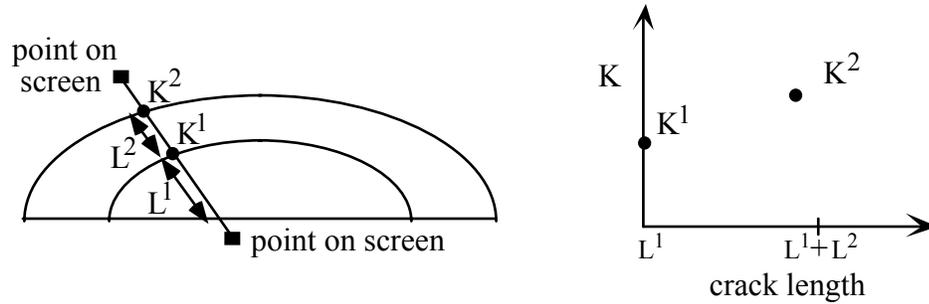


Figure 8.12 Crack length (L) versus mode I stress intensity factor (K) for successive steps of crack growth along the path defined by the intersection of the successive crack fronts with a plane defined by selected points on the modeling window.

File Crack Info

Invokes a dialog box, Figure 8.13, prompting the user to enter a file name and select the data to be added to the file. The data includes the location of the subdivision points along the crack front and the local coordinate system at these points. The local coordinate system is composed of the local tangent to the crack front, the normal to the crack surface, and an orthogonal vector on the crack surface. In addition, the crack displacements and the corresponding three modes of stress intensity factor can be saved. The crack growth increment and direction and the resulting new crack front points can be saved once they have been calculated. Finally, the fitted points can be saved once a polynomial fit has been obtained.

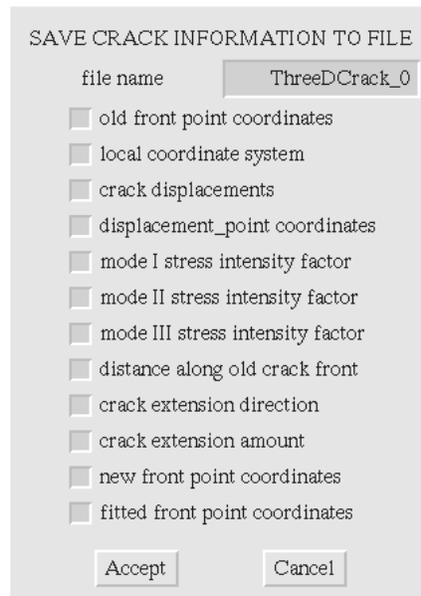


Figure 8.13. The **SAVE CRACK INFO TO FILE** dialog box.

Crack Description

Invokes a sub-menu, Figure 8.14, which allows the crack name, area, volume or front length to be calculated and the data printed on the terminal window.

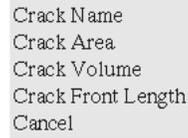


Figure 8.14. The **Crack Description** sub-menu.

return

Exits the menu

8.5 Fatigue Life Prediction

The **FATIGUE LIFE PREDICTION** dialog box, Figure 8.15, has 9 configurable options and two buttons. Each of these are described in this section

8.5.1 Units

The Units parameter specifies the units that the program will use to interpret the stress intensity factor history, the growth rate model parameters, and the initial flaw size. There are four types of units available: ksi-in, psi-in, MPa-mm, and MPa-m. Table 8.1 shows how the stress intensity factor history file will be interpreted for each type of units.

Table 8.1. Available Units

units	crack length	SIF
ksi-in	in	ksi $\sqrt{\text{in}}$
psi-in	in	psi $\sqrt{\text{in}}$
MPa-mm	mm	MPa $\sqrt{\text{mm}}$
MPa-m	m	MPa $\sqrt{\text{m}}$

Units:	Configure
ksi - in	
SIF History:	Configure
History File: d4_full_cent.hst	
SIF Transfer Function:	Configure
effective K = 1 * K + 0	
Fatigue Growth Model:	Configure
FNK: Ti-6Al-4V; MA(1350F/2h)	
Retardation Model:	Configure
no retardation	
Loads Model:	Configure
Constant Amplitude: R=0.002	
Loads Transfer Function:	Configure
effective S = 1 * S + 0	
Initial Flaw Size:	Configure
from history data	
Thickness:	Configure
thickness: 1	
<input type="button" value="Fatigue Life"/> <input type="button" value="Reports"/> <input type="button" value="Cancel"/>	

Figure 8.15. The **FATIGUE LIFE PREDICTION** dialog box.

8.5.2 SIF History

The SIF History parameter specifies the stress intensity factor history. Selecting the **Configure** button presents a menu that allows you to specify that the history should be read from a file, or come from the currently active model. If you select **Read From File**, a file selection box will be displayed that contains a list of the files with a *.hst* extension in the current working directory.

SIF History files contain point-wise values of crack length (*a*) and corresponding stress intensity factor values (*K*). There should be one *a*-*K* pair per line. The *K* values are assumed to be those computed when the maximum load within a load cycle is applied (see the SIF Transfer Function, Loads Mode, and Loads Transfer Function, below).

Optionally, values for all other fatigue life parameters can be specified in the file. These must appear at the beginning of the file, and are delimited at the beginning and end by two '%' characters at the beginning of the line. The parameters themselves are identified

by a name, followed by a colon, followed by one or more parameter values. The format of this file is described in the FRANC3D Concepts & Users Guide and an example file is given here.

```
%%
hist_trans: 10 0
paris: 9.39e-12 2.7
constant_amp: -1.0
%%
0.125      5.0898
0.246      7.29339
0.429      8.97501
0.616      10.4485
0.795      12.0904
1.086      14.3392
1.341      18.2897
1.751      24.1566
1.966      31.4183
2.503      50.3131
2.801      79.1282
3.25       172.62
```

8.5.3 SIF Transfer Function

The third parameter is the SIF Transfer Function. This parameter allows one to perform linear scaling and translations of the K values in the SIF history. Two values, a factor and an offset specify a transformation. The effective K value used for life predictions is:

$$K_{eff} = factor \times K_{\text{from SIF history}} + offset . \quad (8.1)$$

The SIF Transfer Function is used for a number of reasons. It is used most commonly when the K 's in the SIF history file were computed for load levels other than the maximum, as assumed by the life prediction computations.

For example, assume that crack growth was modeled in a component subjected to a load of 2 kips. The corresponding SIF history has been stored in a file. We now want to perform a life prediction of the component subjected to in-service constant amplitude cyclic loading with a mean applied load of 4 kips, and an R of 1/3. The relationships among K_{max} , K_{min} , K_{mean} , and R , are:

$$R = \frac{K_{min}}{K_{max}} \quad \text{and} \quad K_{mean} = \frac{K_{min} + K_{max}}{2} ,$$

giving

$$K_{max} = \frac{2K_{mean}}{R + 1} .$$

which, in the present case, gives $K_{\max} = 1.5 * K_{\text{mean}}$. Therefore, a multiplication factor of 2 is required to transform from an applied load of 2 kips to an applied load of 4 kips, times a multiplication factor of 1.5 to go from the mean K 's to the maximum K 's for this R . This gives a total factor of 3 with an offset of 0 for the desired SIF history transfer function.

8.5.4 Fatigue Growth Model

The Fatigue Growth Model is the relationship that is used to correlate crack growth rate (da/dN) with the stress intensity factor range (ΔK). There are three crack growth rate models available: Paris, Forman-Newman-de Koning (FNK), and a hyperbolic sine function-based model, Figure 8.16. The Lookup-Table is functional in Version 2.6, but the data that is entered into the table cannot be used to compute life – this is still being developed.

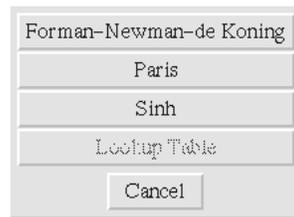


Figure 8.16. The **Fatigue Growth Model** dialog box.

The Paris fatigue growth model, Figure 8.17, is the simplest widely used crack growth rate model. It assumes a power law relationship between the stress intensity factor range and the crack growth rate. The expression for the Paris model is

$$\frac{da}{dN} = C(\Delta K)^n \quad (8.2)$$

In this equation, C and n are constants that are found by a curve fit to experimental data.

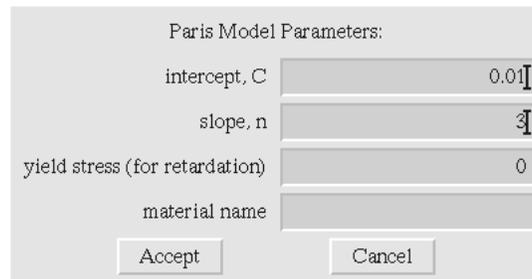


Figure 8.17. The **Paris Model** dialog box.

The hyperbolic sine model, Figure 8.18, has four empirical constants (A , B , C , and D) with the form:

$$\log\left(\frac{da}{dN}\right) = A \sinh(B(\log(\Delta K) + C)) + D. \quad (8.3)$$

The Forman-Newman-de Koning equation starts with the basic Paris relationship, and adds modifications to account for retardation near threshold, acceleration near fast fracture, and the effect of R . The expression for this model is:

$$\frac{da}{dN} = \frac{C'(1-f)^n \Delta K^n \left(1 - \frac{\Delta K_{th}}{\Delta K}\right)^p}{(1-R)^n \left(1 - \frac{\Delta K}{(1-R)K_c}\right)^q} \quad (8.4)$$

where C , n , p , and q are empirical constants (note that C' is not the same as the C used in the Paris model; the n 's, however, are the same). See the FRANC3D: Fatigue Life Prediction documentation for further details. The empirical constants for a wide variety of materials and environments have been incorporated into FRANC3D from the NASGRO material database.

Sinh Model Parameters:

$\log(da/dN) = A \sinh(B(\log(DK) + C)) + D$

A: 0

B: 0

C: 0

D: 0

Kc: 0

yield stress (for retardation): 0

material name:

Accept Cancel

Figure 8.18. The **Sinh Model** dialog box.

The Material Database dialog allows one to browse through the material database and to select a material to be used with the FNK crack growth model. The dialog box is accessed through the Fatigue Growth Model on the main Life Prediction dialog box.

The material database is organized in a hierarchical manner, with four levels in the hierarchy. One moves down a level in the hierarchy by double clicking on the desired option in the scrolling list. To move up a level, double click on the '*** go back ***' option, always the first item in the list (except for the top level). The '**user defined**' option allows the user to enter their own FNK data either interactively in a dialog box (Figure 8.19a) or by importing from a file.

The top level in the hierarchy contains broad material groups, such as stainless steels and aluminum alloys, Figure 8.19b. The next lower level contains more refined groupings of materials, such as 300 series stainless steels or 2000 series aluminum, Figure 8.19c. The third level gives specific materials and heat treatments, such as 304 annealed or 2024-T3, Figure 8.19d. The lowest level gives specific form and environment information, such as "sheet and plate, 550F air", Figure 8.19e. Once an actual material and environment is selected, the values for the FNK parameters stored in the database are displayed in the appropriate fields in the dialog, Figure 8.19f.

Use Data

This option tells the program to close the dialog box and use the currently selected data for the FNK fatigue crack growth model.

Plot da/dN

This option creates a graph showing the crack growth rate versus a ΔK range for the currently selected material data.

Abbreviations

This option creates a popup scrolling box containing definitions for the abbreviations used in the material database.

Discard

This option tells the program to close the dialog box without using the currently selected data for the FNK fatigue crack growth model.

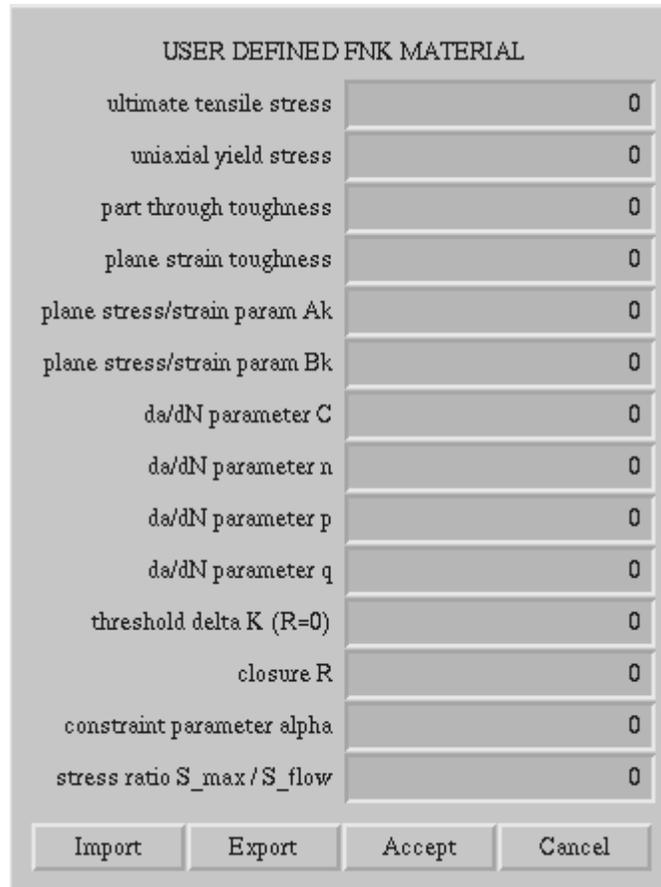


Figure 8.19. (a) User defined FNK material dialog box.

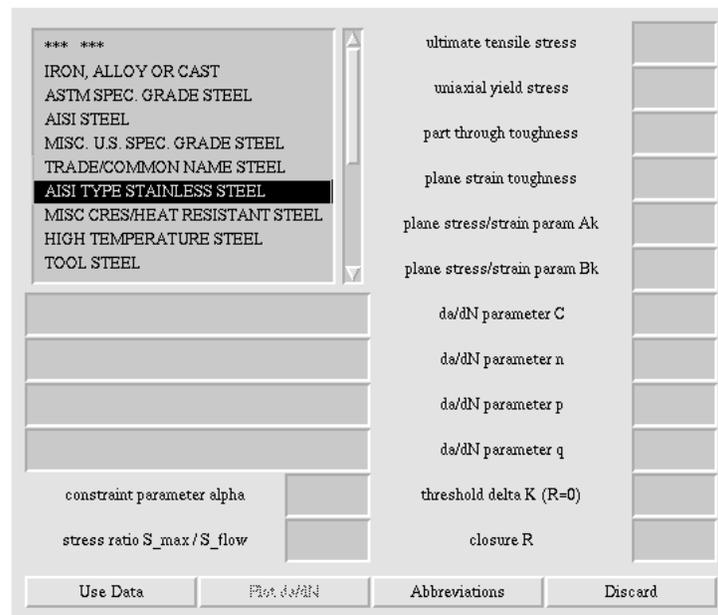


Figure 8.19. (b) Top level in the hierarchy of the materials database.



Figure 8.19. (c) Alloy type level in the hierarchy of the materials database.



Figure 8.19. (d) Heat-treat level in the hierarchy of the materials database.



Figure 8.19. (e) Shape level in the hierarchy of the materials database.



Figure 8.19. (f) Data level in the hierarchy of the materials database.

8.5.5 Retardation Model

The Retardation Model incorporates the effects of load sequencing. In variable amplitude loading situations, overloads can effectively retard crack growth. This is because the overload creates a larger plastic region near the crack tip. When the overload is removed, residual compressive forces tend to keep the crack closed for a larger portion of the load cycle, leading to retardation in the rate of growth. The only retardation model available currently is the Willenborg model.

8.5.6 Load Model

The Load Model parameter specifies the type of loading to be used for the life prediction. The options are "Constant Amplitude" or "Spectrum Loading". If the constant amplitude option is selected, an R must be supplied, Figure 8.20. This is used with the effective maximum K (specified by the SIF History and the SIF Transfer Function) and the Load Transfer Function (see below) to obtain the stress intensity factor range, ΔK , used in the Fatigue Growth Model to determine the crack growth rate.

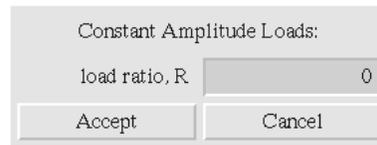


Figure 8.20. The **Constant Amplitude Loads** dialog box.

If the Spectrum Loading option is selected, the user is presented with another dialog, Figure 8.21, in which a load spectrum can be specified (or read from a file). Briefly, a spectrum describes a series of minimum/maximum load pairs. A fatigue life prediction is performed by a cycle-by-cycle integration of the spectrum. This is done by repeating the following steps:

- the next min/max load pair is determined from the spectrum;
- effective min and max load values are obtained by applying the Load Transfer Function to the spectrum load values;
- a current R is computed as the ratio of the min load to the max load;
- a K corresponding to the current crack length is interpolated from the SIF history;
- an effective K is determined by applying the SIF Transfer Function to the interpolated K ;
- min and max effective K 's are computed by multiplying the effective K times the effective min and max loads;
- the stress intensity factor range, ΔK , is computed as the difference between the min and max effective K 's;
- ΔK is used in the Fatigue Growth Model to determine the crack growth rate da/dN ;
- since we are integrating cycle-by-cycle, ΔN is 1 and Δa is the crack growth rate. The current crack length is incremented by this amount and the process is repeated.

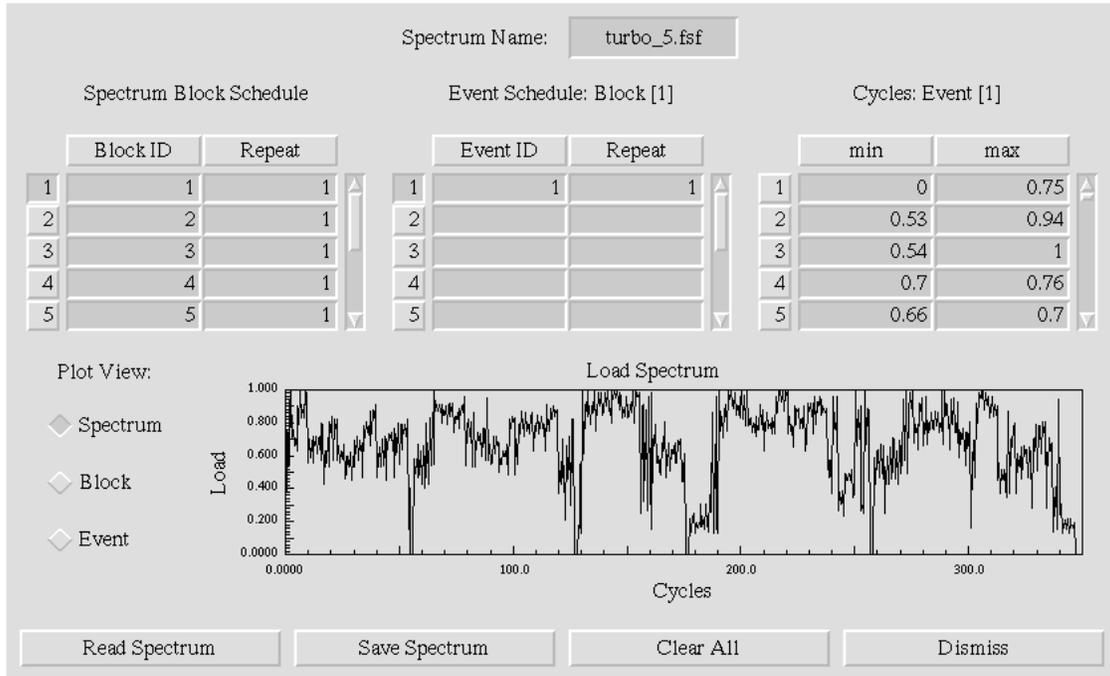


Figure 8.21. An image of the load spectrum definition dialog box.

Spectrum Name

Specifies the name of the spectrum. This is the file name used when saving the spectrum to a file, or when reading it back from a file.

Spectrum Block Schedule

The "spectrum block schedule" specifies the order in which load blocks appear in the spectrum and the number of times each block is repeated.

Event Schedule

The block "event schedule" specifies the order in which events appear within a block and the number of times each event is repeated.

Cycles

The "cycles" schedule specifies the sequence of minimum and maximum loads within an event.

Note: events and blocks are identified by a number, and these numbers are unique throughout the spectrum. That is, event n in block one is the same as event n in block two.

Plot View: Spectrum/Block/Event

Specifies what information is plotted in the dialog box. Options are the full spectrum, the current block, or the current event.

Read Spectrum Button

Allows the analyst to load a previously defined spectrum from a file. The spectrum file should be in the current working directory and have a ".fsf" extension.

Save Spectrum Button

Allows analyst to save the currently defined spectrum in a file. The file name is taken from the current *Spectrum Name* value and will have a ".fsf". The file is stored in the current working directory.

Clear All Button

Clears all the spectrum description information.

Dismiss Button

closes the dialog and asks the analyst if the current information should be used as the active load spectrum for life prediction.

8.5.7 Loads Transfer Functions

The Loads Transfer Function is similar to the SIF Transfer Function, except that it is applied to the loads rather than the SIF's. It specifies a linear scaling and translation of the loads. A transformation is specified by two values, a *factor* and an *offset*. The effective load values used for life predictions is:

$$S_{eff} = factor \times S_{from\ loadmodel} + offset . \quad (8.5)$$

Load transfer functions are used for a number of different reasons. For example, a load spectrum might be specified as normalized load values. The loads transfer function is then used to scale the loads (and effectively the *K*'s) to the proper in-service load.

Note, that *K*-scaling is slightly different between constant amplitude and spectrum loading. In the constant amplitude case, an effective *K* is computed by first applying the SIF Transfer Function to a *K* interpolated from the SIF History. This is further transformed by the Load Transfer Function to find the *K*, which together with *R*, is used to find ΔK .

In the case of spectrum loading, an effective *K* is determined by applying the SIF Transfer Function to the *K* interpolated from the SIF History. The max and min loads obtained from the load spectrum are transformed using the Loads Transfer Function. The effective *K* times the effective max and min loads give max and min *K* values, the difference of which gives ΔK .

8.5.8 Initial Flaw Size

The eighth parameter is the Initial Flaw Size. The default is to use the smallest crack length in the SIF History. However, the SIF history is extrapolated to a zero SIF at a zero

crack length. By setting the Initial Flaw Size to values either smaller or larger than the smallest value specified in the SIF History, one can study the effect of the initial flaw size on the predicted fatigue life.

Also, when using the FNK fatigue crack growth model, it is possible to find situations where the K range is below a threshold value for a material, and no crack growth is possible. In this case one can vary the Initial Flaw Size to find a critical crack size from which fatigue crack growth is possible.

8.5.9 Thickness

The final parameter is the thickness. The default value is unity. This value is used to determine the toughness of material, which is a function of the thickness of the structure at the crack location.

8.5.10 Fatigue Life

There are three control buttons at the bottom of Life Prediction dialog: "Fatigue Life", "Reports", and "Cancel". The Fatigue Life button initiates the cycle integration computations. The Reports button allows one to view and create hardcopies of the results. The Cancel button returns from the Life Prediction dialog.

The Fatigue Life button starts the computations necessary for performing a fatigue life prediction. In the case of constant amplitude loading, this process is very fast, and the resulting fatigue life plot will be displayed almost instantaneously. Spectrum loading, however, requires a cycle-by-cycle integration of the load history, and can take considerably longer. In this case the time required for the integration will be proportional to the actual predicted fatigue life (In most cases this will be on the order of minutes, but may be over an hour for components with long predicted lives).

Before a fatigue life computation is performed, a curve is fit through the SIF history, which is specified pointwise. This is done by fitting piecewise cubic segments through adjacent points. The slopes of adjacent segments are set to be the same, and equal to the mean slope of the converging line segments generated by fitting straight lines through the points. The SIF history is extrapolated to a zero K at a zero crack length. This point is given an infinite (vertical) slope. The fitted SIF history provides a continuous relationship between the crack length and K.

The first three steps in life prediction are the same for constant amplitude and spectrum loading. These are:

1. The initial flaw size is determined. This is either specified or is the smallest crack length given in the SIF history.

2. If the FKN model is being used, a threshold ΔK is computed. If the ΔK corresponding to the initial flaw size is less than this threshold, the computations stop.
3. The maximum crack length is determined. If the FKN model is being used, the critical K is computed, and the maximum crack length is the corresponding crack length in the SIF history. If the Paris model is being used, the maximum crack length in the SIF history is used.

For constant amplitude loading, the life prediction continues with these steps:

4. The total crack length increment, from the initial flaw to the maximum crack length, is divided into 1000 intervals.
5. For each step, the crack length is determined at the beginning and end of the interval. From the SIF history, the corresponding K 's are computed. These are transformed by the SIF and Loads Transfer Functions. The specified R and the transformed K 's are then used to determine the K ranges as the beginning and end of the interval.
6. The K ranges are used with the selected growth model to determine the crack growth rate at the beginning and end of the interval. The mean of these is assumed to be the crack growth rate over the interval.
7. From the interval length and the mean growth rate, the increment of cycles for this interval is computed and added to the total cumulative cycles.
8. Steps 5 through 7 are repeated for all 1000 crack growth intervals, yielding a relationship between crack length and the number of load cycles.
9. The number of intervals is doubled, and steps 4 through 8 are repeated. The resulting total number of cycles is compared to that computed previously. If the difference is more than 0.5%, the number of intervals is doubled again, and the process is repeated until the difference between successive steps is less than 0.5%.

For spectrum loading, the life prediction continues with these steps:

4. A spectrum cycle (minimum and maximum load pair) is obtained from the load spectrum and transformed as described above.
5. From the transformed loads, the K range and R are determined. These are used with the selected Fatigue Growth Model to determine the crack growth rate, which is the amount of growth for one cycle.
6. The crack length is incremented, and if the current crack length is less than the maximum length, steps 4 and 5 are repeated.

8.5.11 Reports

The Reports button gives one access to the report generation portion of the program. Reports are laid out to fit on an 8.5 x 11-inch page. A report page is divided into four different areas. There are two panels that contain graphs or data, a small title block that gives the program name and data, and a larger data block that gives the current program parameters.

When the Reports button is selected the program displays another dialog box, Figure 8.22, showing a schematic of a report layout, and a description of the information that will be displayed in the two panels (initially, *unspecified*). Selecting the Configure button in this dialog pops up another dialog box, Figure 8.23, which lists the available options that can be displayed on the report form. An example report is shown in Figure 8.24.

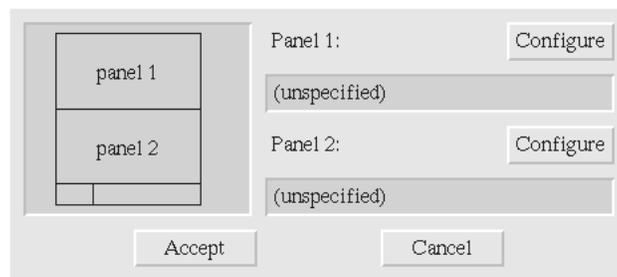


Figure 8.22. The **Reports** dialog box for fatigue life prediction.

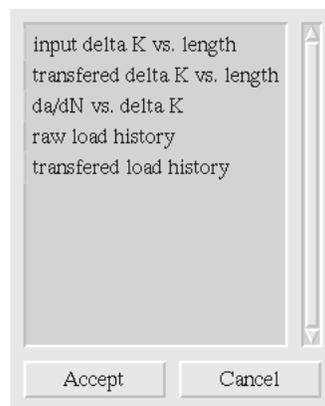


Figure 8.23. The Reports **Configure** dialog.

input K vs. length - a graph of the input stress intensity factors as a function of the crack length.

transferred K vs. length - a graph of the stress intensity factor versus crack length after the specified SIF Transfer Function has been applied to the data.

da/dN vs. delta K - a log-log graph of the fatigue crack growth rate as a function of the stress intensity factor range.

raw load history - a graph of the untransformed load history. In the case of constant amplitude loading, this shows two representative cycles. In the case of spectrum loading, this shows the load spectrum.

transferred load history - similar to above, but with the Load Transfer function applied to the load history.

predicted fatigue life - a graph of the predicted crack length as a function of the number of load cycles.

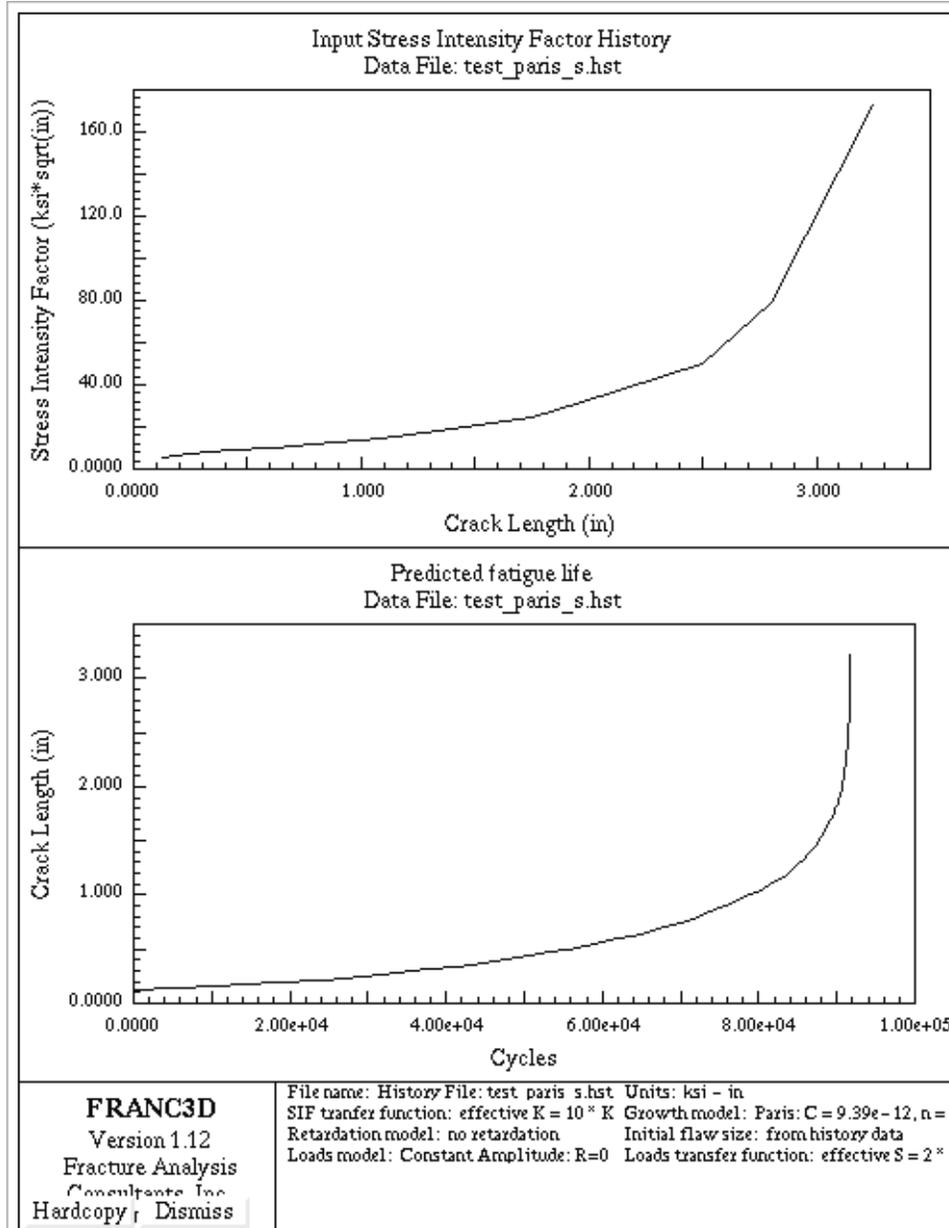


Figure 8.24. Example Report window.

In the lower right corner of the report are two fields in which one can type additional information that will appear on the printed report. In the lower left corner of the report are two buttons, Hardcopy and Dismiss. The Hardcopy creates a Postscript file of the report page. The Dismiss button closes the report preview window.

8.6 Fracture Initiation

The **Fracture Initiation** option allows the user to compute the location of maximum tensile stress in the model and to compute the orientation of the most critical flaw at that location. The information is printed on the terminal window and the triad representing the three principal stress directions at the point of maximum tension is displayed on the screen. Select **Finish** to turn off the highlighting.