FRANC3D & OSM

3D Advanced Tutorial

Automated Crack Growth in a Turbine Disk

Version 2.6

2003

1 Introduction

This manual contains an advanced tutorial example for the programs OSM, FRANC3D and BES. Automated crack growth simulations in a turbine disk illustrate the use and capabilities of the programs. The tutorial is designed for users who are familiar with OSM, FRANC3D, and BES; step by step instructions will not be given. For a more basic tutorial with a complete description of the modeling process, consult the **3D Tutorial**. For details on specific menu commands consult the **FRANC3D Menu & Dialog Reference** or the **OSM Menu & Dialog Reference**.

This tutorial describes the procedure for performing automated crack growth simulations. The model chosen is a disk from a jet engine. We briefly describe how to build such a model using OSM. Once the geometric model is created, the process of attaching material properties and boundary conditions, meshing of the surfaces, and creation of an initial flaw are briefly described. BES (3D BEM code) is used for the stress analysis including rotational inertial velocity boundary conditions. The process of setting up the model for automatic crack growth is described in detail. Computation of stress intensity factor history and fatigue life predictions are then described. In the final section, we describe how one could convert an ANSYS finite element model (FEM) of such a structure into an OSM/FRANC3D model and then use the FEM data during crack growth simulations in FRANC3D

In this tutorial, menu and dialog box options are indicated by bold text, such as **Read FRANC3D File**. Text that is printed to or entered into the program control window is indicated in a typewriter font, such as `Initializing first block in utl_mm`. Information and warning messages that appear in the message box are indicated by italics, such as *Select the vertices that specify the curve*.

The programs are started by typing the program name with the appropriate path. The programs must be started from within the directory that contains the data files. The location of the program can vary; an example session to start FRANC3D from the examples directory with the franc3d executable in the programs directory is:

csh> cd ~/examples
csh> ~/programs/franc3d

where csh> represents the UNIX prompt and the tilde symbol (~) represents the user's home directory.

2 Creating Model Geometry Using OSM

2.1 Introduction

We use OSM (Figure 2.1) to create the disk model. Actually, we will only model a portion of the full disk (Figure 2.2). It is not intended that the user actually build the OSM model at this time, rather the final OSM model is supplied in a file, disk.mod. The purpose of this section is to briefly explain how one would proceed to create such a model from an engineering drawing. Therefore, complete model details are not given and we only give a general description of the modeling steps. The first step is to start the OSM executable and select **Create New Model** from the first menu.
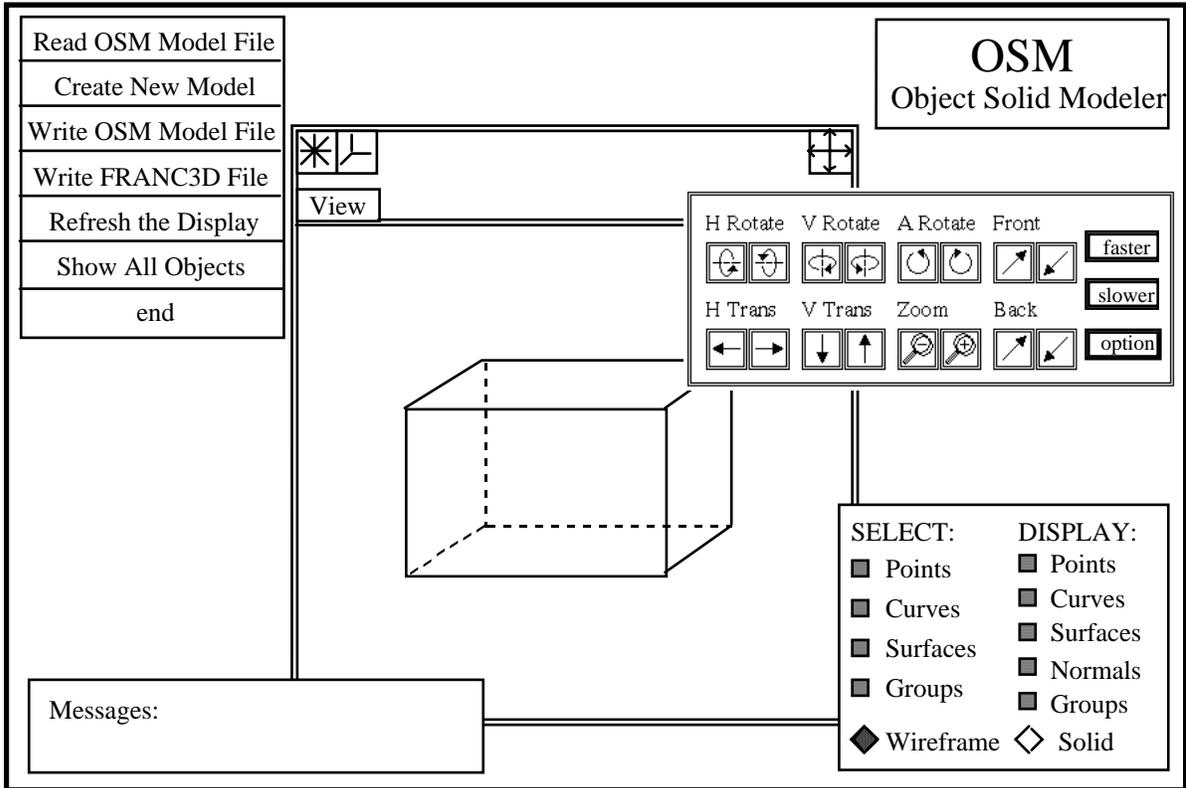

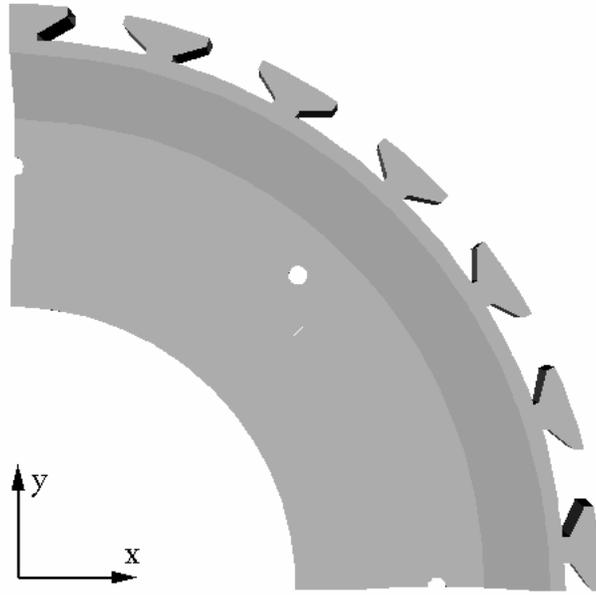Figure 2.1. OSM modeling window and menus.

Figure 2.2. Portion of the disk to be modeled.

The disk model geometry has been simplified significantly for this tutorial, but it still requires some time and effort to build the model. To start, create a cross-section of the disk; for simplicity use x=0.0 and neglect bolt holes for now. Start by selecting the **Type in Points** menu button from OSM and then enter the coordinates of the points that define the cross-section. The point coordinates are provided in Table 2.1. The points are joined with straight lines by selecting the points that define the edges and then selecting the **Create A Curve** menu button from OSM. Figure 2.3 shows the points and edges that define the disk cross-section at x=0 both without and with a bolt hole. Table 2.2 provides the additional point coordinates for the bolt hole.

Table 2.1: Coordinates of disk points for x=0 cross-section without a bolt hole.

X	Y	Z
0	4.5	-0.25
0	4.5	0.25
0	5.25	-0.25
0	5.25	0.25
0	6.0	-0.05
0	6.0	0.05
0	7.5	-0.05
0	7.5	0.05
0	8.5	-0.375
0	8.5	0.375
0	6.75	-0.05
0	6.75	0.05
0	7.1	-0.05

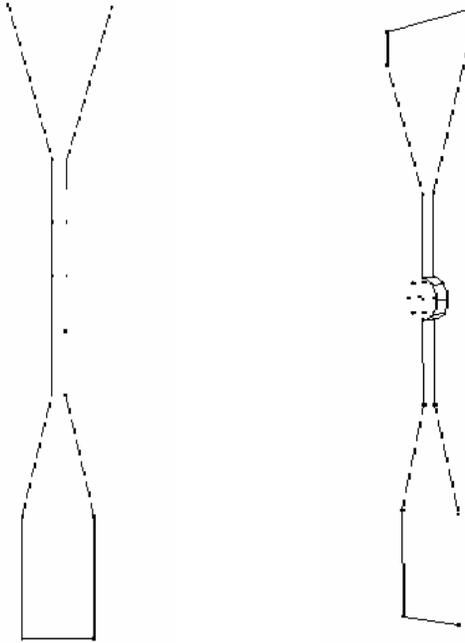| 0	7.1	0.05
0	6.4	-0.05
0	6.4	0.05

Figure 2.3. Cross-section of the disk without and with a bolt hole.

Table 2.2: Coordinates of bolt hole points for x=0 cross-section.

X	Y	Z
-0.106066	6.64393	0.05
-0.106066	6.85607	0.05
0.106066	6.64393	0.05
-0.106066	6.64393	-0.05
0.106066	6.64393	-0.05
-0.106066	6.85607	-0.05
0.106066	6.85607	0.05
0.106066	6.85607	-0.05
0	6.9	-0.05
0	6.9	0.05
0	6.6	-0.05
0	6.6	0.05
-0.15	6.75	-0.05
-0.15	6.75	0.05
0.15	6.75	-0.05
0.15	6.75	0.05

We can either sweep this cross-section about the z-axis by 90 degrees to create a solid and then cut out the holes and other features or build the model in smaller pieces and manually create curves and surfaces that tie the small pieces together. In this case, it is easier to build the model in pieces; remember that OSM and FRANC3D use only three- and four-sided surface patches.

Before we create the bulk of the disk, we will model the hooks (dovetails) as these will determine where we place disk cross sections. Using a separate modeling window, enter the coordinates of the points that define the hook cross-section (Figure 2.4). There are a total of 32 hooks on the disk evenly spaced at 2.8125 degree intervals. We will create the hook cross-sections for the entire quarter model at this stage by copying and pasting the original section at the proper locations (Figure 2.5). The hooks can be tied together by creating arcs between them.
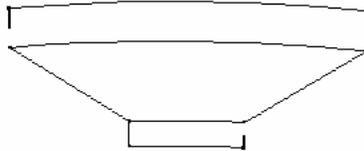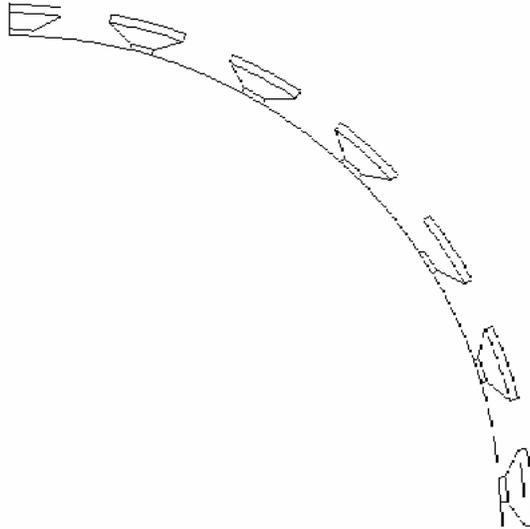

Figure 2.4. Cross-section of the hook.

Figure 2.5. Cross-section of the hook.

Table 2.3: Coordinates of points for the hook section.

X	Y	Z

-0.453636	9.27311	0.375
0.434191	8.842	0.375
-0.446326	9.12354	0.375
0.0	8.85	0.375
0.895716	9.11172	0.375
0.910392	9.26109	0.375
0.0	8.75	0.375
0.429185	8.74531	0.375

Sweep the points and edges in the z direction (-0.75 in) through the disk thickness to complete the hooks. Note that we only sweep the edges and points that lie on the outer boundary of the hook shape.

We now add the disk cross sections to the hooks. There are four sets of two bolt holes evenly spaced about the disk. The holes in each set are separated by 42.1875 degrees. We copy the disk section with the bolt hole into three locations at 0, -42.1875, and -90 degrees from the y-axis (Figure 2.6). We then paste in regular disk cross sections at appropriate multiples of 2.8125 degrees so that the cross sections match up with the hooks (Figure 2.6). These cross-sections are then tied together by creating arcs between them. Finally, surfaces are created from the points and curves (Figure 2.7)
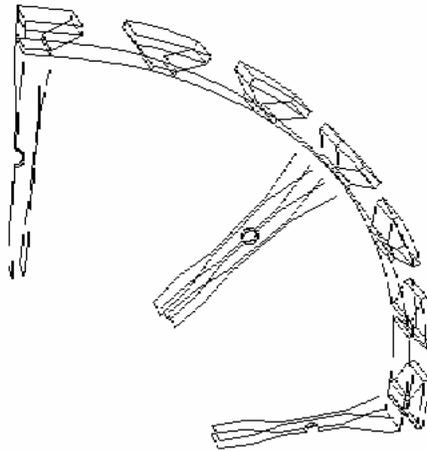

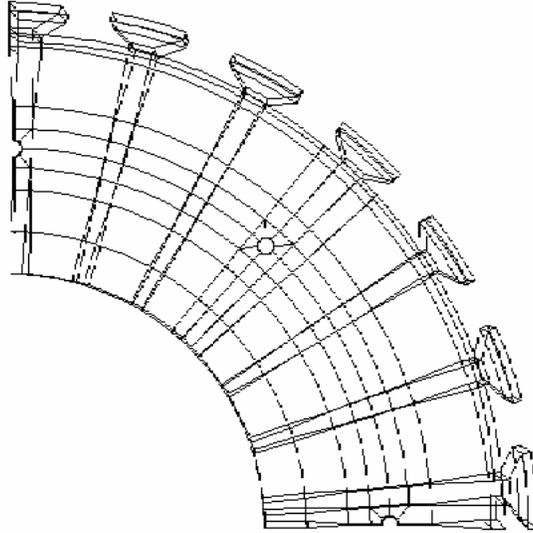Figure 2.6. Disk cross-sections pasted into the model with the hooks.

Figure 2.7. Completed disk model.

The most time consuming task is tying all the hooks, bolt holes, and disk cross-sections together to form a closed solid using only three- or four-sided surface patches (faces). Faces are created by selecting corner points or bounding curves and then selecting the **Create Patch By Corners** or **Create Patch By Curves** menu commands respectively. All the surface normals must point outward from the model. If a surface normal points inward, select that surface and **Reverse Patch Orientations**.

Ensure that the model is a closed manifold solid by selecting the **Create Radial-Edge Database** menu command. A radial-edge database is created only if the model surface normals all point outward and there are no holes (missing surfaces). Problem areas are highlighted if any exist. If the database is created successfully in OSM, FRANC3D will be able to read the model. Both the OSM (.mod) and FRANC3D (.dat) files should be saved by selecting the appropriate OSM menu commands.

3.0 Automated Fracture Analysis Using FRANC3D

3.1 Introduction

The main purpose of this tutorial is to explain how to perform automated crack growth simulations in a real engineering structure and to then analyze the results of the simulation. We use FRANC3D and BES in this tutorial to analyze the stresses and crack growth in a spinning turbine disk. It is assumed that the user is quite familiar with

FRANC3D. We do not provide explicit details and step by step instructions for every step of the modeling process. The first step is to start FRANC3D in the directory where the .dat file is located.

FRANC3D displays a modeling window (Figure 3.1) along with the main menu, the title box, message window, and view control buttons. We will start from the geometry file that we created using OSM by selecting **Read Geometry File**. A file selector box is presented with all the files in the directory with a .dat extension. Select the file called disk.dat, either by double clicking on the file name with the left mouse button, or by single clicking on the name to highlight it and then selecting **Accept**. If the data in the file is okay, it will be read and the model geometry displayed in the modeling window with no messages appearing in the message box.
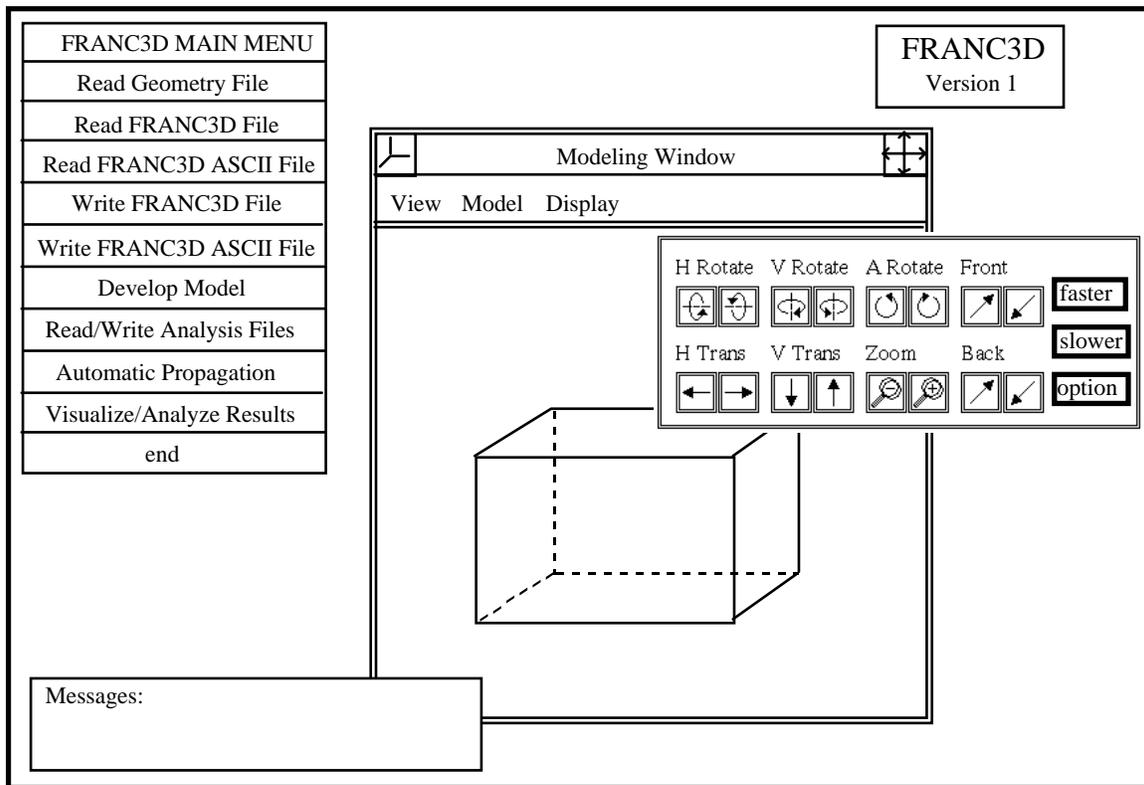

Figure 3.1. FRANC3D modeling window and menus.

3.2 Preparing the Model For Automated Crack Growth Simulations

The model must be prepared for automated crack growth simulation by nucleating the initial crack, setting material properties, defining boundary conditions, and creating the surface mesh.

3.2.1 Initial Crack

We start by nucleating the initial crack. Experimental/field evidence suggests that the cracks start growing from the bolt holes in the disk and propagate radially into the disk. We choose one of the holes in the disk model and zoom in on that location. We will start with a planar corner crack of radius 0.01 inches oriented parallel to the disk radial direction. We can add this flaw using the flaw library.

Select the following menu and sub-menu buttons as the menus pop up: **Develop Model**, **Modify Geometry**, **Nucleate Crack**, **Three-D Cracks**, **Library Flaws** and **show library of flaws**. When the dialog box appears, enter values for the crack size, location and orientation and then select **Calculate** to make sure the crack is correct. If it is correct, select **Accept**; the crack will be added to the model (Figure 3.2). Use 12 points on the crack front, a radius of 0.01 inches, and then provide the following rotations and translations:

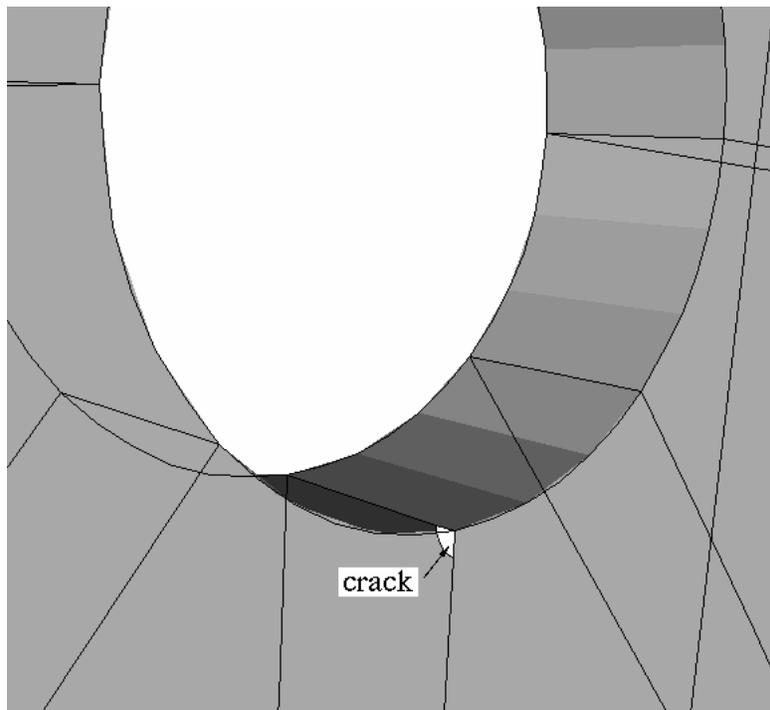| Rotations:		Translations:	
	y=90.0		x=4.43228
	x=-90.0		y=4.89027
	z=-42.1875		z=0.05

Figure 3.2. Initial crack in disk.

3.2.2 Material Properties and Boundary Conditions

Next, we will set the material properties and boundary conditions. Close all the menus except for the **Develop Model**. From the **Develop Model** menu, select **Specify Attributes** and then select **Material Properties**. From the **Material Properties**, select **Edit Material** and select the **Select From List** sub-menu. The default material (mat_1) is the only material in the list; select it. In the dialog box, change the material name to Ti-6-4 and set the elastic modulus to 1.7e7 psi, the Poisson's ratio to 0.33 and the density to 4.17e-4 lbf-s/in^4. Select **Accept** when finished and then close the **Material Properties** menu.

From the **Specify Attributes** menu, select **Boundary Conditions**. Select the **New Boundary Condition** menu button and then select **Face BCs** from the sub-menu. Set the boundary condition name to x_symmetry and select **Accept**. Select the displacement toggle button for the x-direction and then select **Accept** on the dialog box. Next select **Attach Boundary Condtion** and then select **Face BCs** from the sub-menu. From the list of face boundary conditions, choose x_symmetry; then select all the faces of the model that lie on the x=0 symmetry surface. Select **FINISH** when all faces are collected. Repeat this procedure for the cross-section at y=0, setting the y-displacement to 0.

We need to restrain rigid body motion in the z-direction. Select the **New Boundary Condition** menu button and then select **Face BCs** from the sub-menu. Set the boundary condition name to z_constraint and select **Accept**. Select the displacement toggle button for the z-direction and then select **Accept** on the dialog box. Next select **Attach Boundary Condtion** and then select **Face BCs** from the sub-menu. From the list of face boundary conditions, choose z_constraint; then select the faces shown in Figure 3.3.
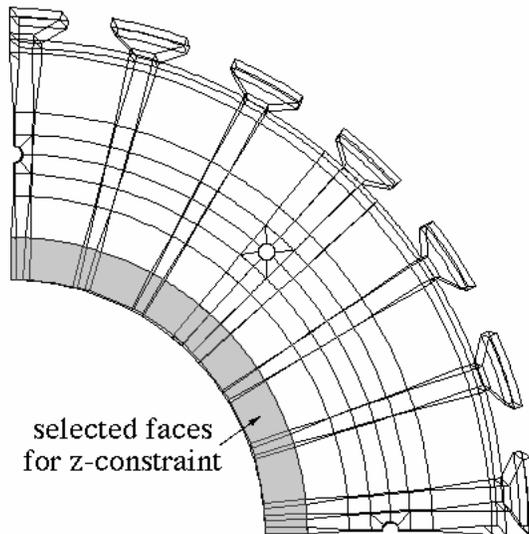

Figure 3.3. Face boundary conditions for z-constraint.

At this stage, we can define the rotation boundary conditions. Select the **New Boundary Condition** menu button and then select **Model BCs** from the sub-menu. When the dialog

box is presented, set the name to rotation and set the magnitude of the rotation vector as x=0, y=0, and z=1047.2. Select **Accept** when finished. The rotation is defined in radians/second (1047.2 is 10,000 rpm).

Note that the blades which are attached to the disk have not been included in the OSM and FRANC3D models. These blades apply an additional force on the disk which is a function of their mass, radial position, and the rotational velocity. Rather than modeling the blades explicitly, the force that is generated by the spinning blades is attached to the hooks of the disk as surface tractions. The radial force due to the blades is computed using the average mass of the blades and the rotation vector defined above. The force is distributed over the contact surface of the hooks in the disk (flat inclined surface in Figure 6) as a surface traction.

The average blade mass is 50.0 grams. The rotational velocity is 1047.2 rad/sec. The radial acceleration a_n is computed as:

$$a_n = r \cdot \omega^2 \tag{3.1}$$

where r is the radius and ω is the rotational velocity. Assuming that we can concentrate the mass of the blades at the base of the hook, the radius is 8.75 in. This gives a radial acceleration of 9.6e6 in/s^2. Converting to m/s^2 and multiplying by the mass (50.0e-3 kg), gives a force of 1.22e4 N. Converting to lbs (1 lb = 4.448 N) gives 2.74e3 lb of force. The area over which this force is applied is computed from the geometry as $(2)(0.532 \cdot 0.75) = 0.798$ in^2 as there are two surfaces for each hook where the blades contact the disk. The surface traction applied to each of the contact areas is given by force/area=3,433 psi and this traction acts in the radial direction. We compute the global x and y traction components for each hook assuming the force is concentrated along the center line of the hook.

We create a face boundary condition for the each of the hooks. Select the **New Boundary Condition** menu button and then select **Face BCs** from the sub-menu. When the dialog box is presented, define a name and select **Accept**. Then in the Face Boundary Conditions dialog box, enter the traction components to be applied to the hook contact surfaces and select **Accept**. Attach this boundary condition to the hook contact surfaces as described previously.

3.2.3 Meshing

The geometry model is very large and requires a significant amount of memory. Therefore, we cannot use a lot of elements in portions of the disk far away from the crack. We start by subdividing all the edges in the model using a unit subdivision. Select **Subdivide Edges** from the **Develop Model** menu. Select **Set Number of Subdivisions** and enter 1 on the key pad. Select **Reset All** and then select **Subdivide All Remaining**. We can now re-subdivide any edges that we think require more points to capture the geometry more closely, specifically, the crack edges and edges near the crack.

The number of subdivisions that you use will depend on the available computer resources, including memory, swap space, and physical disk space. You need enough elements around the crack to give a reasonable approximation to the geometry surface. We suggest that you look at the provided FRANC3D file for guidance. For example, a coarse mesh away from the crack and a more refined mesh on and near the crack could be used; the mesh shown in Figure 3.4 has about 3000 elements.
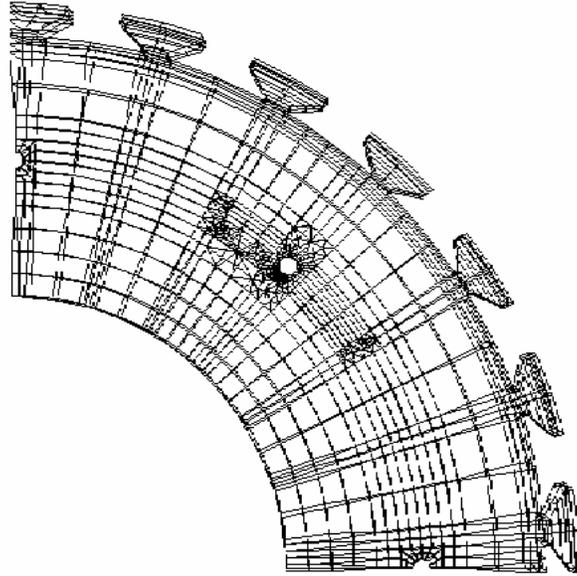

Figure 3.4a. Full mesh for the disk with initial corner crack from the bolt hole.
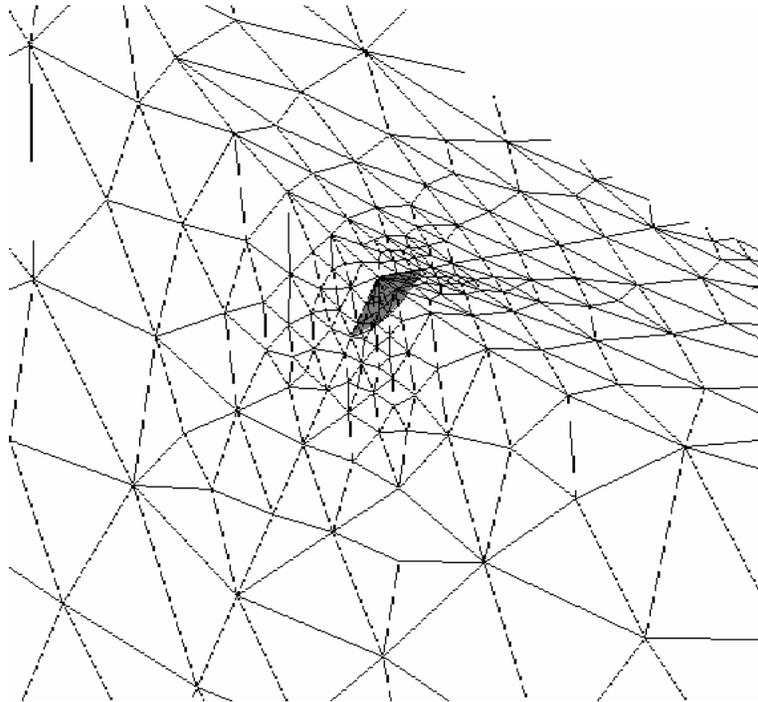
Figure 3.4b. Close-up of the mesh around the initial corner crack in the disk.

The surfaces can be meshed by selecting **Mesh Surfaces** from the **Develop Model** menu and then selecting **Automatic** from the **Mesh Surfaces** menu. Select **Hilight UnMeshed Faces** to make sure all faces are meshed. All faces should be meshed. At this stage you should save the FRANC3D file by selecting **Write FRANC3D File** and entering the file name in the file selector dialog box.

3.3 Setting Crack Growth Control Parameters

The crack growth control parameters must be set before the automated simulations can be started. Select the **Numerical Analysis** menu button from the main menu. Select **Automatic Propagation** and then **Select Crack Growth Model**. The following dialog box is presented (Figure 3.5). Note that the default settings are used if they are not modified below.
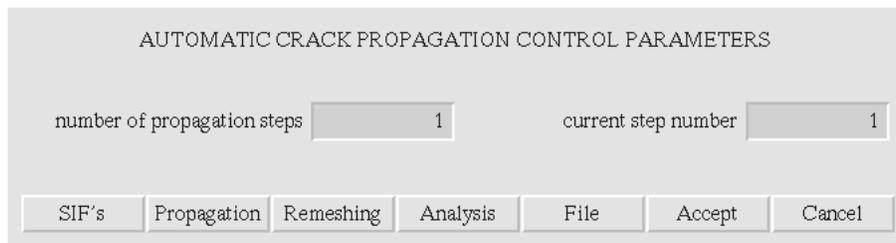

Figure 3.5. The **AUTOMATIC CRACK PROPAGATION CONTROL** dialog box.

First set the **number of propagation steps**, lets use 5 for now. The current step number is 1 because this is the initial crack model. Select the **SIF's** button which causes the dialog box shown in Figure 3.6 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. The default values should be okay so just select **Accept**; we do not want to display the SIF's.
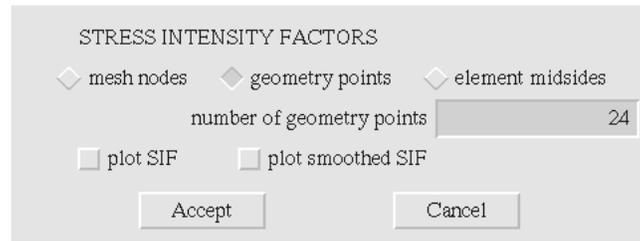

Figure 3.6. The **STRESS INTENSITY FACTORS** dialog box.

Select the **Propagate** button which causes the dialog box shown in Figure 3.7 to be displayed. The method for computing crack growth direction and the maximum crack advance can be defined for each crack growth step. Lets use a constant maximum

extension for all five steps of propagation. Check that the value for constant maximum extension is 0.01 by selecting that radio button. A simple dialog box is displayed with the current value. All other default values and selections can remain. Select **Accept**.
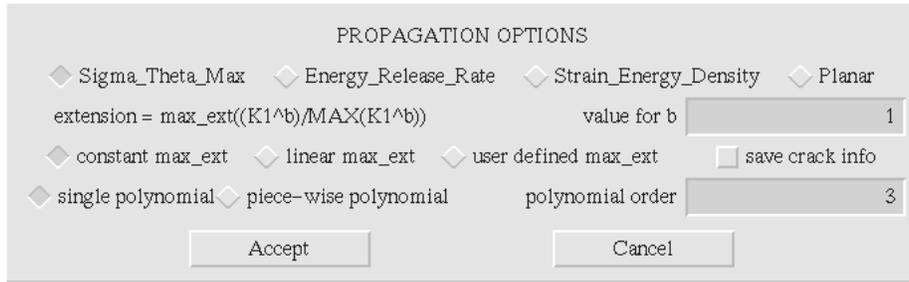
PROPAGATION OPTIONS

◇ Sigma_Theta_Max ◇ Energy_Release_Rate ◇ Strain_Energy_Density ◇ Planar

extension = max_ext((K1^b)/MAX(K1^b))   value for b   [1]

◇ constant max_ext ◇ linear max_ext ◇ user defined max_ext   ☐ save crack info

◇ single polynomial ◇ piece-wise polynomial   polynomial order [3]

[Accept]   [Cancel]

Figure 3.7. The **PROPAGATION OPTIONS** dialog box.

Select the **Remeshing** button which causes the dialog box shown in Figure 3.8 to be displayed. The number of elements along the crack front can be defined along with the transitioning factor. We use the default number of crack front elements with a mesh transition factor of 1.0. Select **Accept** when finished.

CRACK SURFACE REMESHING OPTIONS

set number of crack front elements

◆ default ◇ constant ◇ linear ◇ user defined

set mesh transition (between 0.0 and 1.0)

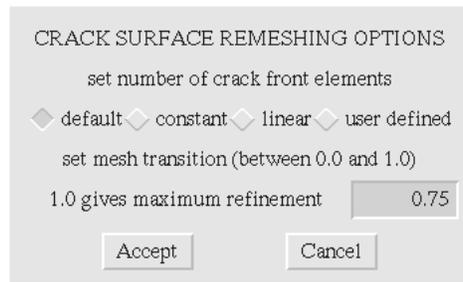1.0 gives maximum refinement   [0.75]

[Accept]   [Cancel]

Figure 3.8. The **REMESHING OPTIONS** dialog box.

Select the **Analysis** button which causes the dialog box shown in Figure 3.9 to be displayed. The parameters to control the stress analysis using the boundary element code, BES, can be set here. Leave all the default settings and select **Accept**.

STRESS ANALYSIS OPTIONS

☐ solve in core ☐ linear elements ☐ flex sets
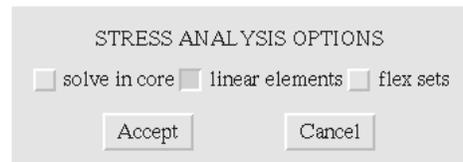
[Accept]   [Cancel]

Figure 3.9. The **STRESS ANALYSIS OPTIONS** dialog box.

Select the **File** button which causes the dialog box shown in Figure 3.10 to be displayed. The crack growth modeling parameters are written to this file which can then be used with FRANC3D running in batch mode to control each step of analysis and crack growth. Select **Accept** to save the parameters into the file.

file name	crack_growth_model

Accept	Cancel

Figure 3.10. The **File name** dialog box.

3.4 Automated Simulations

At this stage, the initial model has been saved as a FRANC3D file and the crack growth control parameters have been saved in a file. You can run the simulations from the interactive FRANC3D session or in a background batch mode. We prefer to run in the background as it uses less computer resources, plus you can run in the background and log out of the workstation session. **End** FRANC3D.

3.4.1 Setting Up BES

You now need to be sure that BES is ready to run from the current directory where the files reside. First, you must have sufficient disk space available to store the coefficient matrix in a file unless you have sufficient memory to run in-core. The file size can be more than 1 GB for this model depending on the number of elements used on the model surface. If the BES executables (cge_bes or cocqr_bes) are not in the current directory, their location should be included in the PATH environment. The cge_bes executable uses a Gauss direct solver and retains the coefficient matrix in-core (in RAM), while the cocqr_bes executable uses a QR solver and stores the coefficient matrix on the local hard disk. The in-core solver is generally much faster as there is neglible hard disk access. The memory requirements can be computed based on the number of elements, but if you attempt to run in-core and receive an error message "mmf error: unable to allocate memory), then you will have to run out-of-core or reduce the number of elements.

See the BES README file on the CFG www site for some additional information on running BES.

3.4.2 Running FRANC3D in Batch Mode

To start the simulations, type:

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

This forces the processes to run in background mode and all standard output is dumped into a file called junk. Note that you may need to specify the path to the franc3d executable if it is not defined in your PATH environment variable.

FRANC3D starts by reading the disk.fys file and then re-saving it as disk_STEP_1.fys. The BES input file disk_STEP_1.bes is then saved. Once this file is written, a separate process to run BES is *forked* from FRANC3D. FRANC3D *sleeps* until this process

finishes. BES produces two results file; disk_STEP_1.l.besout contains the displacements and tractions and disk_STEP_1.l.con contains the stresses. A log file from BES called disk_STEP_1.l.out can be used to monitor BES's progress.

When BES finishes, FRANC3D *wakes up*. The results files from BES are read. The SIF's are computed, the new crack front points are determined, and finally, the crack is propagated (all automatically). Once the crack has propagated, the model is remeshed based on the mesh of the previous crack step and the parameters set above.

When the model has been remeshed, the FRANC3D and BES files are written again as disk_STEP_2.fys and disk_STEP_2.bes (for the second step). BES is then started again, and the process continues until the number of propagation steps reaches the specified number.

3.4.3 Restarting After Propagation Failures

There is a chance that the automated simulations will stop due to FRANC3D's inability to automatically propagate the crack. This happens sometimes when the crack is traversing complicated geometric surface boundaries, especially if the crack step is large in comparison to the geometric complexity of the model near the crack. In this case, the franc3d process will stop. If this happens, you can interactively read in the previous disk_STEP_#.fys file (the .l.besout and .l.con files must exist also), and propagate the crack manually to get the automated simulation running again.

The crack can be propagated manually and then re-meshed manually. The FRANC3D file should be saved as disk_STEP_#+1.fys. You will then need to go back to the **Select Crack Growth Model** menu button and modify the **current step number** and the **number of propagation steps**. Any of the other crack growth control parameters can be adjusted at this time as well. Save the information to a file again, and then restart franc3d in batch mode again giving the disk_STEP_#+1.fys file name after the -f flag.

4.0 Simulation Results

4.1 Introduction

One of the purposes of this tutorial is to show the simulations results can be used. One can look at detailed evaluations of individual crack shapes, displacements, and stress intensity factors for different stages of crack growth and/or stress intensity factor histories and life predictions.

The numerical simulation results examined here consist of crack size or shape and stress intensity factor history. We have completed ten steps of propagation for this model. The corner crack propagates such that it remains in the same radial plane. It transitions from a corner crack to a part-through crack after eight steps of propagation (Figure 4.1).
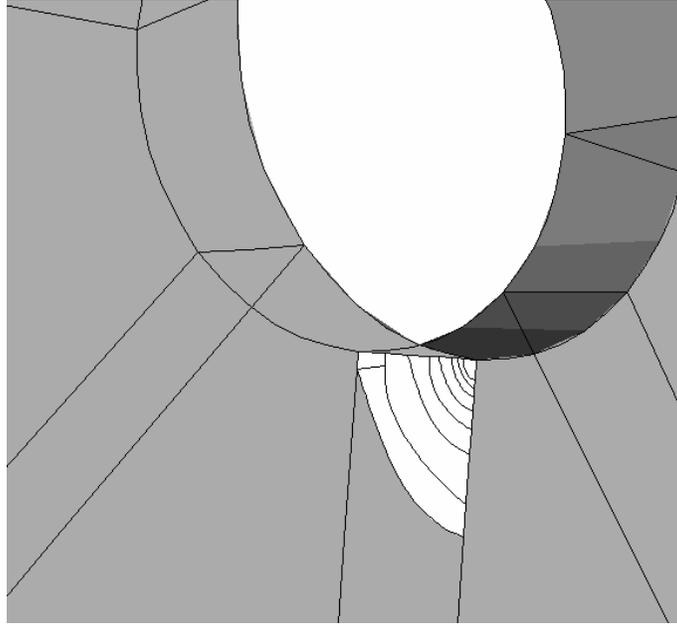

Figure 4.1. The crack extends through the disk thickness and becomes a part-through crack as it continues to propagate radially into the disk.

One can now examine the stress intensity factor history for the propagating crack. Select the **Show Stress Intensity Factor History** command from the **FRACTURE ANALYSIS** menu (Figure 4.2). There are two options: 1) **maximum K vs area** and 2) **along a defined path**. Choosing the second option, the program prompts you to pick the crack front and then define a path on the crack surface. The path is created by picking two points on the screen; the two points are actually two vectors that extend into the modeling window. The two vectors are used to define a plane that will cut through the crack surface. The Mode I SIF is computed where each crack front intersects the plane. The distance along the crack surface between each of these intersection points provides the other axis.

```
FRACTURE ANALYSIS
Stress Intensity Factor
Propagate Crack Front
Save KI History
Show KI History
Predict Fatigue Life
File Crack Info
Crack Description
return
```

Figure 4.2. **FRACTURE ANALYSIS** menu

Figure 4.3 shows the Mode I stress intensity factor history for a single path on the crack surface. Only Mode I SIF is computed to predict the fatigue life, but one can obtain different Mode I SIF histories for different paths along the crack surface. The life predictions for different paths will vary as do the SIFs vary along the crack front.
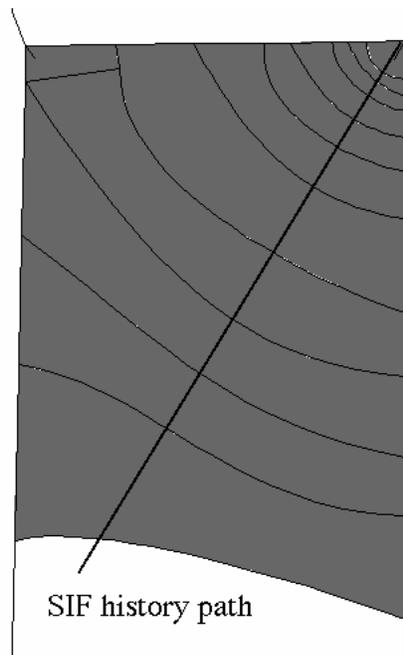
SIF history path

Figure 4.3. The path for computing mode I SIF history.
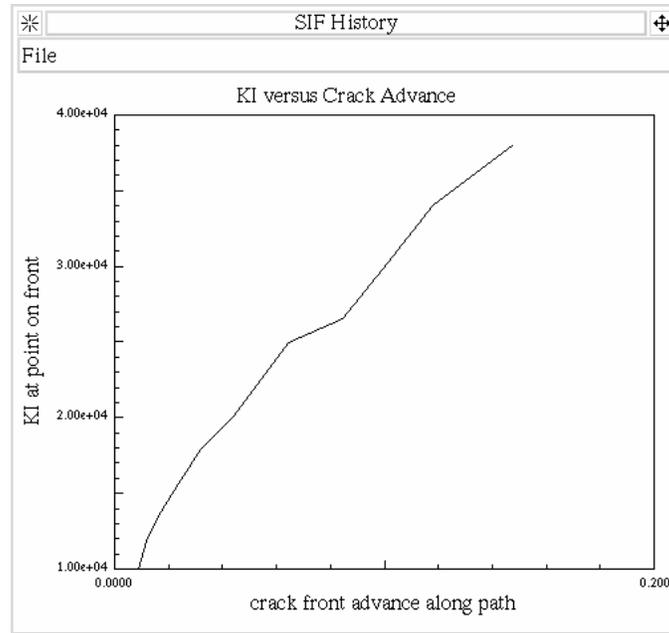
Figure 4.4. The Mode I SIF history for the path on the crack surface shown in Figure 4.3.

The SIF history is used as input to the **Fatigue Life Prediction** dialog box. Select **Predict Fatigue Life** from the **FRACTURE ANALYSIS** menu to obtain the dialog box, Figure 4.5.

Figure 4.5. The **FATIGUE LIFE PREDICTION** dialog box.

Select the **Units Configure** button and select psi-in as the units. Select the **SIF History Configure** button and select **Use Current Model Data**. Ignore the **SIF Transfer Function**. Select the **Fatigue Growth Model Configure** button and select **Forman-Newman-de Koning**. A new dialog box appears, Figure 4.6. In this dialog box, select **TITANIUM ALLOYS**, then select **Ternary alloys**, then select **Ti_6Al_4V;MA(1350F/2h)**, then select **Forg**. This will fill in the fields on the right side of the dialog, Figure 4.5. Ignore the **Retardation Model**. Select the **Loads Model Configure** button and select **Constant Amplitude**. Set the **R** value to 0.0 which corresponds to zero for a minimum rpm loading. Ignore the **Loads Transfer Function**. Select the **Initial Flaw Size Configure** button and select **From History Data**. This defines all the data necessary to do fatigue life predictions.

Figure 4.5. Data level in the hierarchy of the materials database.
The chosen material is a Titanium alloy.

Select the **Fatigue Life** button at the bottom of the dialog shown in Figure 4.5. A plot showing number of cycles versus crack length is displayed. You can save the data or dismiss the plot. The same data can be plotted in a report form. Select the **Reports** button at the bottom of the dialog shown in Figure 4.5. A new dialog box is displayed which allows you to define the data that will appear on the report form, Figure 4.6. See the **FRANC3D Menu/Dialog Reference** for further details. This completes this section of the advanced tutorial, you should now be able to automatically propagate a 3D crack in FRANC3D and then analyze the results of the simulations to predict the fatigue life of the structure.
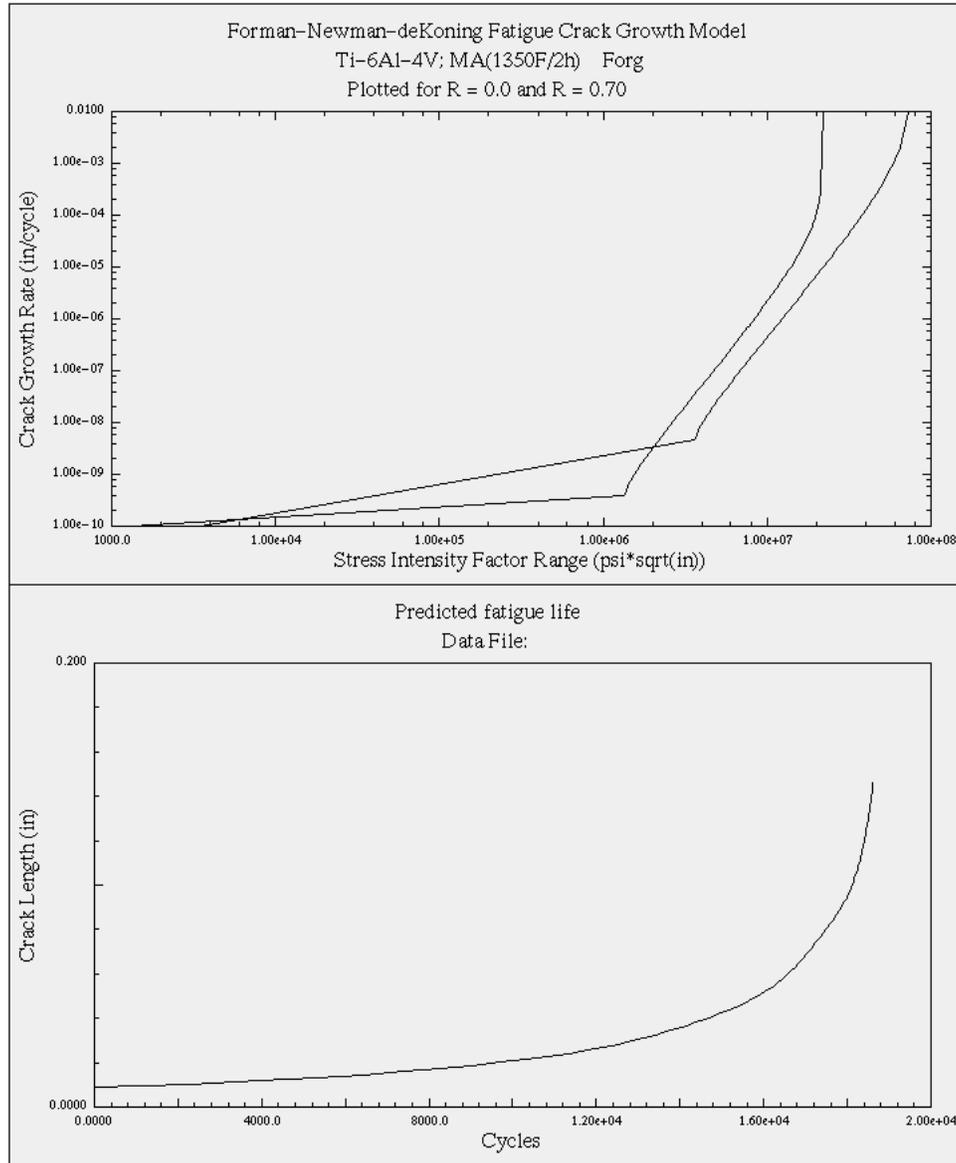
Figure 4.6. Report window showing da/dN data and the predicted fatigue life for this disk for an initial corner crack of radius 0.01 in.

5.0 Crack Growth Simulations Using FEM Stresses as Crack Face Tractions

5.1 Introduction

The final section of this tutorial describes another advanced feature of OSM/FRANC3D. Often, the user has available a finite element model of the structure along with the resulting stresses. Most commercial FEM packages cannot do fully automated crack growth simulations. We allow users to combine the benefits of both their existing FEM capabilities and the automated crack growth capabilities of FRANC3D. We produce an OSM/FRANC3D model directly from the FEM data (see the FEMtoOSM documentation for details on this procedure) and then use FRANC3D in combination with the FEM stresses to model crack growth.

5.2 Converting the FEM Data to OSM/FRANC3D

We will use the same disk model in this section, except that now we simplify the model by removing the hook sections and model a smaller portion of the disk. A model was built in ANSYS, Figure 5.1. The ANSYS model was meshed using 3,327 20-noded brick elements and the boundary conditions and material properties defined previously were applied. The only difference is that now both the blades and the hooks are replaced by an equivalent traction (a negative surface pressure); the is 3,011 psi. The ANSYS analysis was performed and the resulting nodal displacements and stresses as well as the ANSYS .cdb file were written to files. The displacements should be given a .dsp file extension while the stresses should be given a .str file extension. The prefix name should match the .cdb file name.
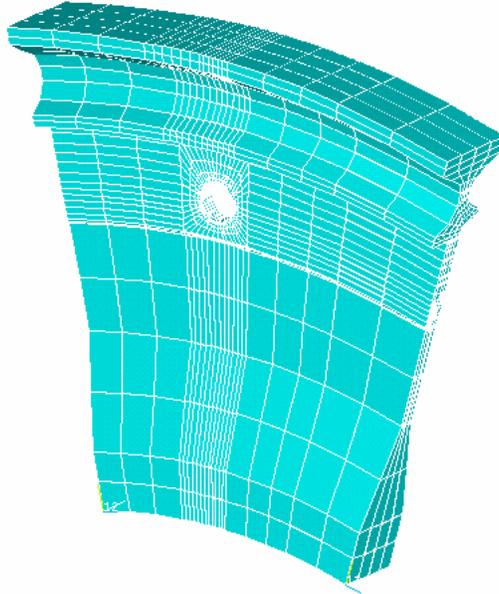

Figure 5.1 ANSYS sub-model of the disk.

The OSM model of this disk sub-model could easily be obtained by direct manipulation of the original model. We would need to cut out a portion of the disk, add some faces to

close the model, and rotate it into position. For demonstration purposes, however, we will actually convert the ANSYS .cdb file into an OSM model. We start by selecting the **Translate ANSYS** menu button in OSM, Figure 5.2.
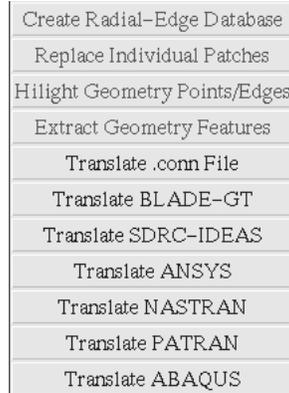
Create Radial–Edge Database
Replace Individual Patches
Hilight Geometry Points/Edges
Extract Geometry Features
Translate .conn File
Translate BLADE–GT
Translate SDRC–IDEAS
Translate ANSYS
Translate NASTRAN
Translate PATRAN
Translate ABAQUS

Figure 5.2 OSM menu that gives access to the FE to OSM translation options.

A file selector box is presented with all the .cdb files listed. Select the disk_submodel.cdb file. The process of translating the ANSYS FEM data to an OSM model begins automatically (see the FEMtoOSM document for details). First, the node and element information is converted to a local format and saved as a disk_submodel.cdb.conn file. Next, the interior FEM data is removed and a disk_submodel.cdb.osm file is saved. The information in the .osm file is then dislayed in an OSM modeling window, Figure 5.3.
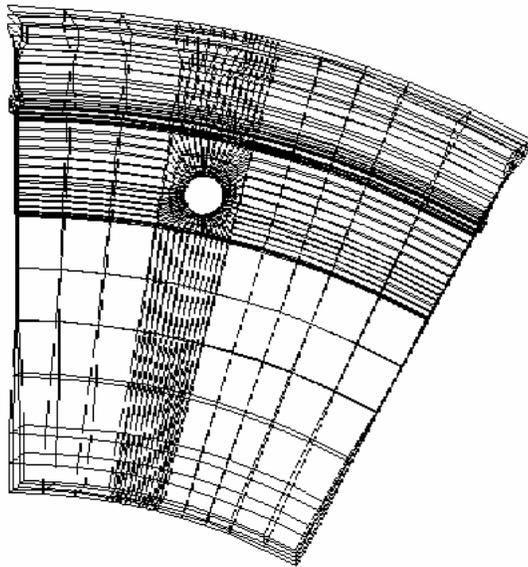
Figure 5.3 OSM modeling window showing the disk_submodel after translating the ANSYS .cdb file to an OSM .mod file.

The next step is to create a radial-edge database for this model. Select the Create Radial-Edge Database menu button, Figure 5.2. If successful, the **Hilight Geometry Points/Edges** and the **Extract Geometry Features** menu buttons will become active. If unsuccessful, a message dialog box is displayed informing you why OSM could not create the database. For this model, the database should be created successfully.

Next select the **Extract Geometry Features** menu button. A dialog for the geometric tolerance is displayed, leave the value at 10 degrees and select **Accept**. The process of hilighting the key geometry edges and points begins. This is followed automatically by the procedure for extracting the key geomtry features, forming faces from the bounding loops of merged edges, and copying the data to a new OSM modeling window. (Again, see the FEMtoOSM document for details.) When finished, a new modeling window containing the extracted geometry is displayed, Figure 5.4.
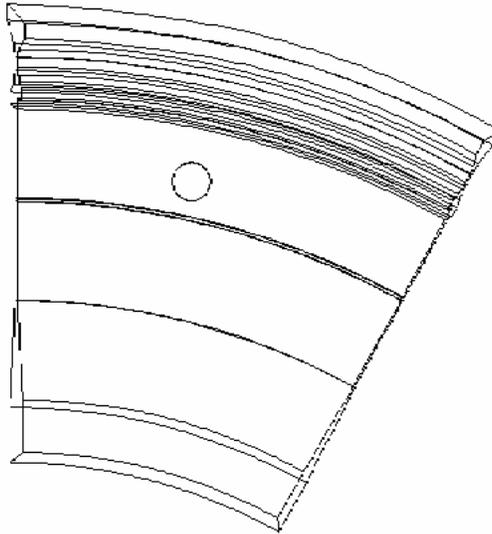

Figure 5.4 OSM modeling window showing the disk_submodel after extracting the key geometric features from the model in Figure 5.3.

The extracted model should be saved to a new .mod file name. Do not save the the original model under the original name as the model contains both original and extracted geometry at this stage. Also save the FRANC3D .dat file using the 3D Solid option. Save it as disk_submodel.dat.

The only other information that is needed from the FEM model at this stage is a file containing the magnitude of the nodal stress components and possibly a file with the nodal displacments. These two files must be in the proper format for FRANC3D. The only information in the file should be a listing of the stress or displacement components at each FE node, in free format, one entry per line. Note that, unless supressed, the listing of ANSYS stresses or displacements when written to a file contain extra information that must be deleted from the file.

5.3 Using the FEM Data in FRANC3D

Read the .dat file into FRANC3D. Insert a crack into the model at the corner of the bolthole-disk surface. Use a 0.03 in corner penny-shaped crack. You can use the library flaw to do this, with rotations set to x=90, y=-90, and z=12 and translations set to x=1.46632, y=6.84393, and z=0.06725. Subdivide the edges and create an initial mesh similar to that of the full model, concentrating elements around the hole and the crack, Figure 5.5. Define the material properties to be the same as for the full model. The boundary conditions will be somewhat different for the sub-model compared to the full model.

We intend to use the ANSYS FEM stresses as crack surface tractions in this model. First, we must constrain the FRANC3D model where displacement boundary conditions are defined in the ANSYS model. This is quite simple, we need to attach roller boundary conditions to the symmetry surfaces first. Do this by defining a new face boundary condition with the coordinate system to local and the x (local n) displacment to 0.0. This boundary condition is then attached to all of the symmetry surfaces. The z-constraint is applied to the forward inner ring of edges on the disk to prevent rigid body motion. Next we attach the FEM stress data to the model.

The first step is to read the .mrp file created by OSM into FRANC3D. OSM reads the .cdb file as well as the .dsp and .str files and creates a .mrp file. See the **Concepts & Users Guide** for additional information on MRPs. Import the .mrp file into FRANC3D as an MRP. We will attach the MRP to a face boundary condition next.

In FRANC3D, we create a new face boundary condition, Figure 5.6. In this dialog box, we select the MRP BC button and then from the list of MRPs that are displayed, we select the disk_submodel_res_str.mrp. Leave all the other options as they are and select Accept. Next, we must attach this face boundary condition to the crack faces.

The FEM data is used only when writing the BES input file. When you select the **Write BES File** menu button, the FEM stresses are retrieved from the MRP and converted to crack face tractions. Every mesh node on the crack surface will have a traction computed for it. The nodal coordinates are first found. These are fed into a routine which determines the FEM element in which the point falls. Given the element, interpolation routines are used to find stresses at the point. The stresses are then combined with the crack surface normal at the node to give crack surface tractions. Equal and opposite tractions are applied to the nodes on each crack face.

Figure 5.6. The **Model Boundary Conditions** dialog box.

You can define the crack growth parameters for automatic propagation and run
FRANC3D/BES as in Section 3. The model boundary conditions do not change. Each
time a new BES input file is written for a step of propagation, all the crack surface
tractions are recomputed. You can compare SIF histories and fatigue life predictions for
both models. The fatigue lives should be very similar if all is correct. Note that the crack
face traction boundary conditions are automatically attached to any new crack surface
generated during propagation. If you grow the crack manually, then you will have to
manually attach the boundary conditions to the new crack surface.

This is the end of the FRANC3D Advanced Tutorial.