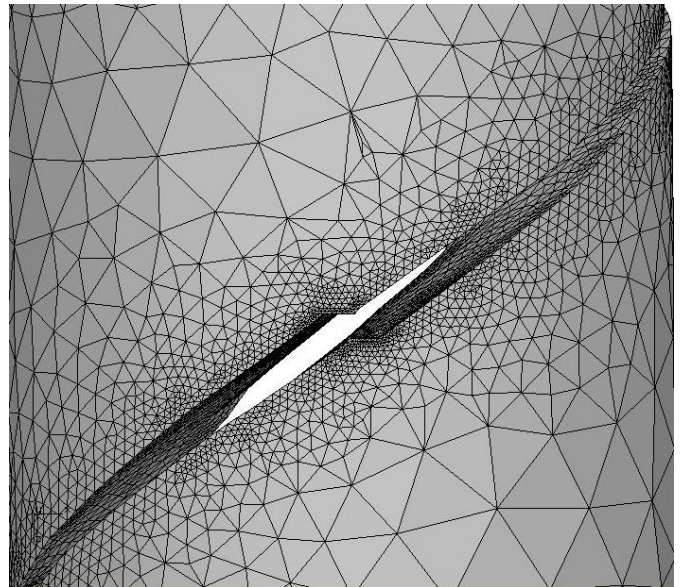


FRANC3D

Tutorials #2-15

Version 9.2



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

Revised: Aug 2026

Table of Contents:

Table of Contents:	2
1.0 Introduction	7
2.0 Tutorial 2: Multiple Load Cases and Crack Face Traction (CFTs)	7
2.0.1 ABAQUS Load Cases	8
2.0.2 ANSYS Load Cases	8
2.0.3 NASTRAN Load Cases	8
2.1 Step 1: Reading FE Model into FRANC3D	9
Step 1.1: Importing FE Model	9
2.1.1 ANSYS .s## files	10
Step 1.2: Select the Retained Items in the FE Model	11
Step 1.3: Displaying the FE Model	13
2.2 Step 2: Insert Crack From File	13
2.3 Step 3: Apply Crack Surface Traction	17
2.4 Step 4: Static Crack Analysis	18
2.5 Step 5: Compute SIFs	22
2.6 Step 6: Re-run static crack analysis with Crack Face Contact	24
2.7 Step 7: Compute SIFs with Crack Face Contact	29
2.8 Step 8: Apply Surface Treatment Residual Stress	32
2.9 Step 9: Apply Traction from Mesh-Based Stress	38
2.9.1 Uncracked Model Stress	38
2.9.2 Apply Mesh-Based Stress as Crack Face Traction	42
2.10 Step 10: Apply Traction and Temperature from Mesh	46
2.10.1 Uncracked ABAQUS Cube Stress and Temperature	46
2.10.2 Apply Mesh-Based Temperature and Stress as Crack Face Traction	51
3.0 Tutorial 3: Two Cubes Glued Together	56
3.1 Step 1: Create the FE Model	56
3.1.1 ABAQUS Model	56
3.1.2 ANSYS Model	57
3.1.3 NASTRAN Model	59
	2

3.2	Step 2: Import FE Model into FRANC3D	60
	Step 2.1: Importing FE Model	60
3.3	Step 3: Insert a Crack	61
3.4	Step 4: Static Crack Analysis	63
	3.4.1 ABAQUS	63
	3.4.2 ANSYS	63
	3.4.3 NASTRAN.....	63
3.5	Step 5: Compute SIFs.....	63
3.6	Plot Deformed Shape	64
3.7	Contact surface mesh not Retained	65
3.8	Local / Global Cases	67
	3.8.1 Local model has part of the contact surface.....	68
	3.8.2 Local model has one side of the contact surface.....	74
	3.8.3 Local model has one side of the contact surface plus a cut-surface	79
4.0	Tutorial 4: Disk with Rotation and Temperature	86
4.1	Step 1: Create the Disk Model	86
	4.1.1 ABAQUS Disk.....	86
	4.1.2 ANSYS Disk.....	91
	4.1.3 NASTRAN Disk	94
4.2	Step 2: Import Disk into FRANC3D.....	97
4.3	Step 3: Insert Initial Crack	99
4.4	Step 4: Static Crack Analysis	101
	4.4.1 ABAQUS Solver.....	101
	4.4.2 ANSYS Solver	103
	4.4.3 NASTRAN Solver	104
4.5	Step 5: Compute SIFs.....	106
4.6	Step 6: Crack Growth.....	107
4.7	Step 7: Automatic Crack Growth.....	111
4.8	Step 8: Analysis Results.....	112
	4.8.1 ABAQUS crack model results	114
	4.8.2 ANSYS crack model.....	116

4.8.3	NASTRAN crack model	117
4.9	Step 9: Fatigue Cycles.....	118
4.10	Apply Crack Face Traction (CFT).....	125
4.10.1	Unloaded FE Model.....	125
4.10.2	Original FE Model	128
5.0	Tutorial 5: Multiple Cracks in a Plate.....	131
5.1	Step 1: Create the Plate Model.....	131
5.1.1	ABAQUS Plate	131
5.1.2	ANSYS Plate	133
5.1.3	NASTRAN Plate.....	134
5.2	Step 2: Import Plate Model into FRANC3D.....	136
5.3	Step 3: Insert Multiple Cracks.....	138
5.4	Step 4: Static Crack Analysis	150
5.5	Step 5: Compute SIFs for Multiple Crack Fronts	154
5.6	Step 6: Grow Multiple Cracks.....	154
5.7	SIFs History for Multiple Cracks.....	157
5.8	Crack Growth Transitions	160
5.8.1	Grow a Crack around a Corner	160
5.8.2	Grow Crack through a Back Surface	165
5.9	Grow Multiple Cracks in Multiple Submodel Portions	169
5.9.1	Step 1: Import Plate Model into FRANC3D.....	169
5.9.2	Step 2: Insert Multiple Cracks	173
5.9.3	Step 3: Crack Analysis.....	178
6.0	Tutorial 6: Computing Fatigue Life Cycles	181
6.1	Step 1: Perform Crack Growth.....	183
6.2	Step 2: Compute Fatigue Crack Growth Cycles	191
6.3	Crack Growth and Fatigue Options.....	193
6.3.1	Constant Amplitude Loading.....	194
6.3.2	Variable Amplitude Loading	195
6.4	Spectrum Loading for Manu Model.....	197
7.0	Tutorial 7: Fretting Fatigue	202

7.1	FE Model.....	202
7.1.1	ABAQUS Model.....	202
7.1.2	ANSYS Model.....	204
7.1.3	NASTRAN Model	206
7.1.4	Results in .dtp File	206
7.2	Step 2: Fretting Model Import.....	206
7.2.1	Import ABAQUS Results	207
7.2.2	Import ANSYS Results.....	208
7.3	Step 3: Specify Fretting Contact Surface	209
7.4	Step 4: Fretting Crack Nucleation.....	211
7.5	Step 5: Discrete Crack Nucleation	214
7.6	Step 6: Static Crack Analysis.....	219
7.6.1	ABAQUS Static Crack Analysis	219
7.6.2	ANSYS Static Crack Analysis.....	223
8.0	Tutorial 8: Session Log Playback	228
8.1	GUI Playback	230
8.2	Command Line Playback	230
9.0	Tutorial 9: Python Interface	231
9.1	Python PyF3D Module.....	231
9.2	Python Extension for User-defined Crack Growth	233
9.2.1	Step 1: Read Cube Crack Model.....	233
9.2.2	Step 2: Read Python Extension.....	233
9.2.3	Step 3: Grow Crack.....	235
9.3	Python Extension File	237
10.0	Tutorial 10: Fatigue with Time Dependent Growth with a User-Mesh Crack.....	241
10.1	SubModel and Crack Insertion	242
10.2	Define Crack Growth Parameters.....	246
10.2.1	Load Schedule.....	250
10.2.2	Crack Growth Rate Model	258
10.2.3	Effective dK and Integration.....	262
10.3	Crack Growth Analysis	265

10.3.1	ABAQUS Analysis	268
10.3.2	ANSYS Analysis	269
10.3.3	NASTRAN Analysis.....	271
10.4	Life Prediction	272
11.0	Tutorial 11: Stress Corrosion Crack.....	274
11.1	Import Uncracked Model.....	274
11.2	Insert a Crack	275
11.3	SCC Analysis.....	277
12.0	Tutorial 12: Symmetry Crack.....	290
12.1	Import Uncracked Model.....	290
12.2	Insert Symmetry Crack	291
12.3	Symmetry Crack SIFs	295
12.4	Symmetry Crack Growth	296
13.0	Tutorial 13: Interface Crack	298
13.1	Import Uncracked Model.....	298
13.2	Embedded Interface Crack.....	299
13.3	Crack Crossing an Interface.....	301
14.0	Tutorial 14: LCF + HCF with Dynamic Pairing	304
14.1	Import Uncracked Model.....	304
14.2	Step 2: Insert Initial Crack.....	306
14.3	Step 3: Static Crack Analysis	308
14.4	Step 4: Compute SIFs	311
14.5	Step 5: Crack Growth	312
14.6	Step 6: Fatigue Cycles	318
14.7	Summary.....	319
15.0	Tutorial 15: Building SIF Histories.....	322
15.1	Resuming Growth with a Larger Submodel	322
15.1.1	Extract and Save Crack Geometry	323
15.1.2	Restart from Saved Crack Geometry	324
15.1.3	Combine SIF Histories.....	329
15.2	Building a SIF History from Multiple Crack Insertions	333

1.0 Introduction

These tutorials describe many of the fracture simulation capabilities of FRANC3D. The tutorials are applicable to ABAQUS, ANSYS and NASTRAN; where there are differences between the analysis codes, extra sub-sections are added. The tutorials start at #2 because Tutorial #1 is the base tutorial, which is provided for each of the analysis codes in separate documents.

Note that all tutorial model files are available from the FAC download site. We do not describe how to build the FE models that we use in these tutorials, although there should be enough information included so that one can build their own models.

2.0 Tutorial 2: Multiple Load Cases and Crack Face Tractions (CFTs)

In the first part of Tutorial 2, we describe the process of importing a model with multiple load cases. Once a crack is inserted, additional load cases are applied as a crack face pressure or traction (CFT). We apply constant pressure in Section 2.3, a surface treatment residual stress in Section 2.8, and a mesh-based stress in Section 2.9. Section 12 of the User's Guide provides additional information on CFTs.

A simple cube model with three load cases is used; this is the same finite element (FE) model used in the base tutorial (see either of the FRANC3D/ABAQUS, FRANC3D/ANSYS or FRANC3D/NASTRAN Tutorials), but with two extra load cases (or load steps). The first extra load includes positive (far-field) surface traction, equal in magnitude to the negative traction of the first load case. The second extra load includes negative (far-field) surface traction that is double the magnitude of the first load.

Note that the first extra load will cause the cube to compress, and we describe how to apply crack face contact (CFC) to prevent negative Mode I SIFs in Section 2.6.

2.0.1 ABAQUS Load Cases

For ABAQUS, the load cases are defined in the *.inp* file and are differentiated by “*STEP” commands. **Note that FRANC3D does not support *LOAD CASE data; you should use multiple *STEPS instead.**

2.0.2 ANSYS Load Cases

For ANSYS, the load cases can be defined in the *.cdb* file and differentiated by “lswrite”, or “lsnum=”, or they can be included with “/input, file, s##”, or they can be defined in separate *.s##* files. If *.s##* files are used, make sure the same load case(s) are not already contained in the *.cdb* file. FRANC3D will look for boundary condition data in the *.cdb* file, and if data exists, appropriate load cases will be created.

If one “imports and divides” a model with extra load steps, FRANC3D will incorporate the “divided” boundary conditions/loads into the appropriate local and global portions. This data will be written to the corresponding *.cdb* files, rather than writing extra load step files. Consequently, when importing an “already divided” model, the option to include extra load step files is not active. This is important for users that wish to divide the model themselves (not using FRANC3D), as the global and local portions must include all the appropriate boundary conditions for all load steps.

2.0.3 NASTRAN Load Cases

For NASTRAN, the load cases are defined in the *.bdf* (or *.nas* or *.dat*) file and are differentiated by “SUBCASE”.

2.1 Step 1: Reading FE Model into FRANC3D

Step 1.1: Importing FE Model

Start the FRANC3D graphical user interface (GUI) and set the work directory. Select **File** → **Import**, Fig 2.1, and in the dialog shown in Fig 2.2, choose Complete Model. Switch the Mesh File Type radio button in the Select Import Mesh File dialog, Fig 2.3, to the appropriate analysis code. Select the file name for the model, and then select **Next**.

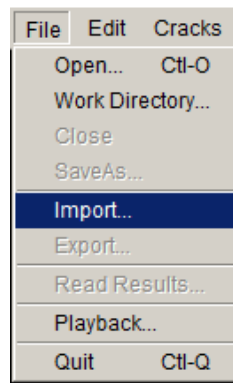


Figure 2.1 FRANC3D graphical user interface

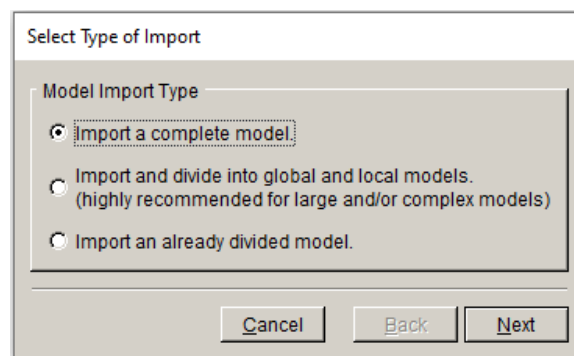


Figure 2.2 Import type

Although the cube could be subdivided, we import the full model for simplicity. For small models, the time required to remesh the volume is short. However, for most models, we want to subdivide to avoid remeshing the full model at each step of crack growth.

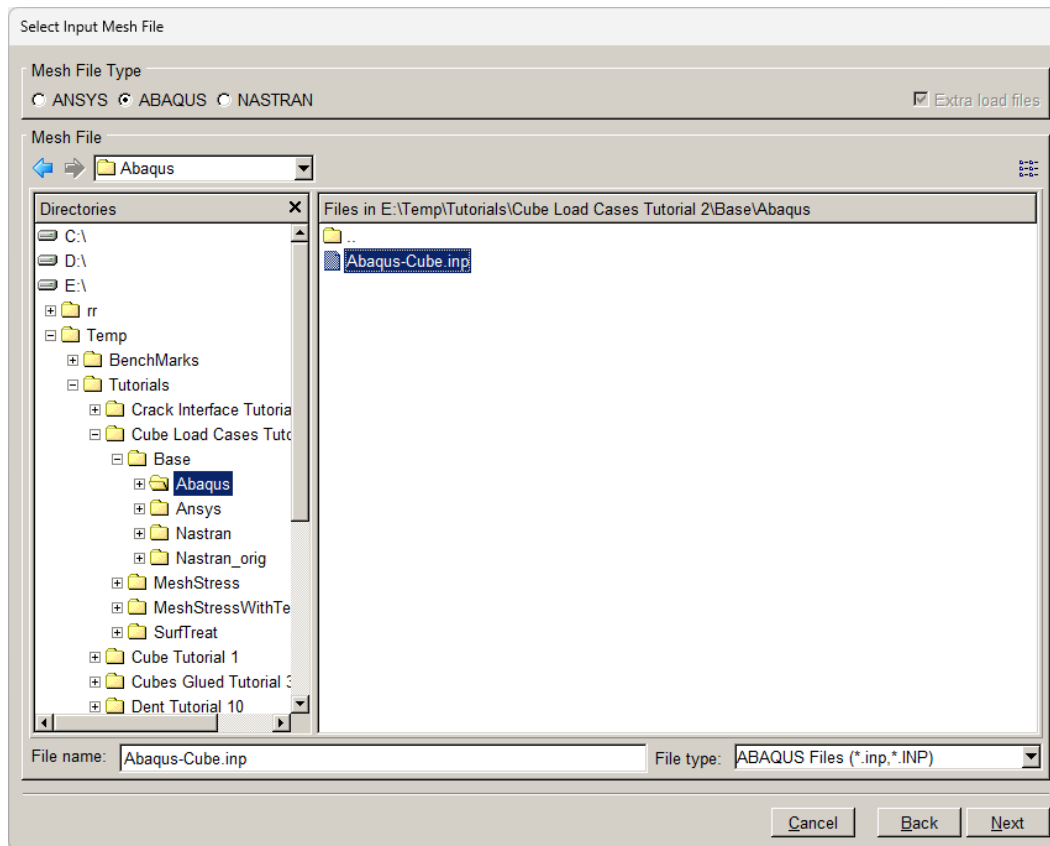


Figure 2.3 Select Input Mesh File dialog box – ABAQUS mesh type selected.

2.1.1 ANSYS .s## files

For ANSYS, if extra load steps are contained in .s## files, the Extra load files box (top right of Fig 2.3) must be checked. An extra dialog with all .s## files will be presented after choosing the base .cdb file. The file selection dialog allows you to choose multiple files by holding the Shift or Ctrl keys.

The tutorial files on the FAC download site include the base .cdb file and two extra load step files with extensions .s02 and .s03. The .cdb file contains the first load step.

Step 1.2: Select the Retained Items in the FE Model

The next panel, Fig 2.4, allows you to choose the mesh surface facets that are retained. Surfaces with boundary conditions appear blue, Fig 2.5, and turn red when selected, Fig 2.6. We retain the surfaces with boundary conditions (top and bottom of the cube) by choosing Select all. The boundary conditions are transferred automatically to the new mesh when the crack is inserted. Select **Next** to set the FE model units. SI units of mm and MPa can be used for length and stress; there are no temperatures in this model, so those units are not relevant.

Note that the boundary conditions for all load steps are applied to the same surfaces in this model.

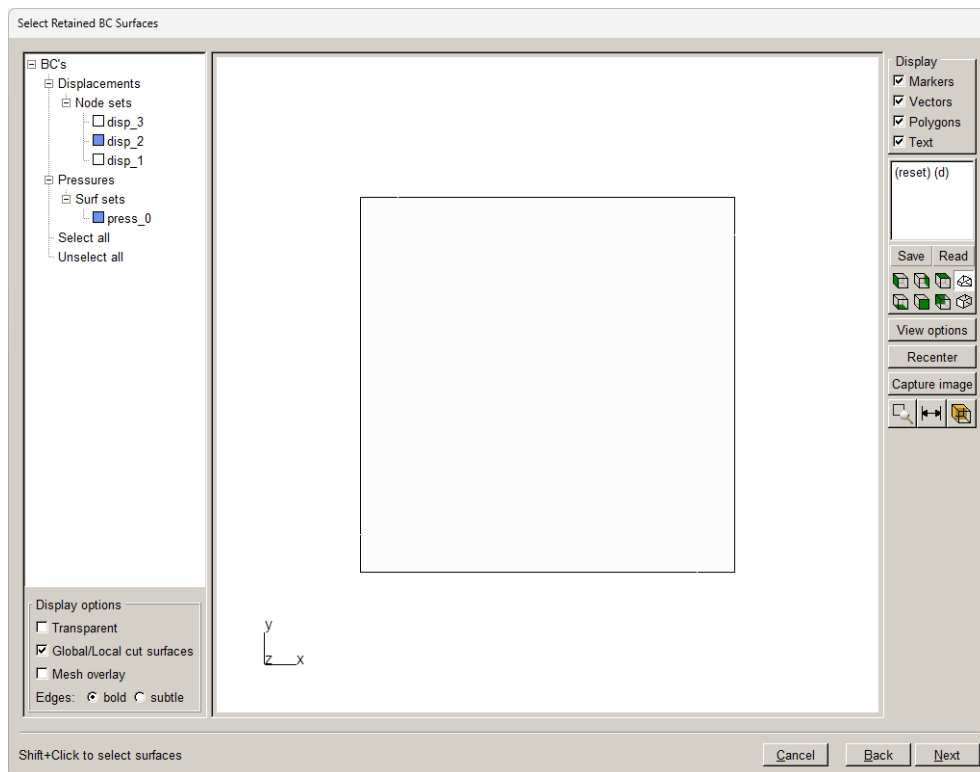


Figure 2.4 Select Retained BC Surfaces wizard panel.

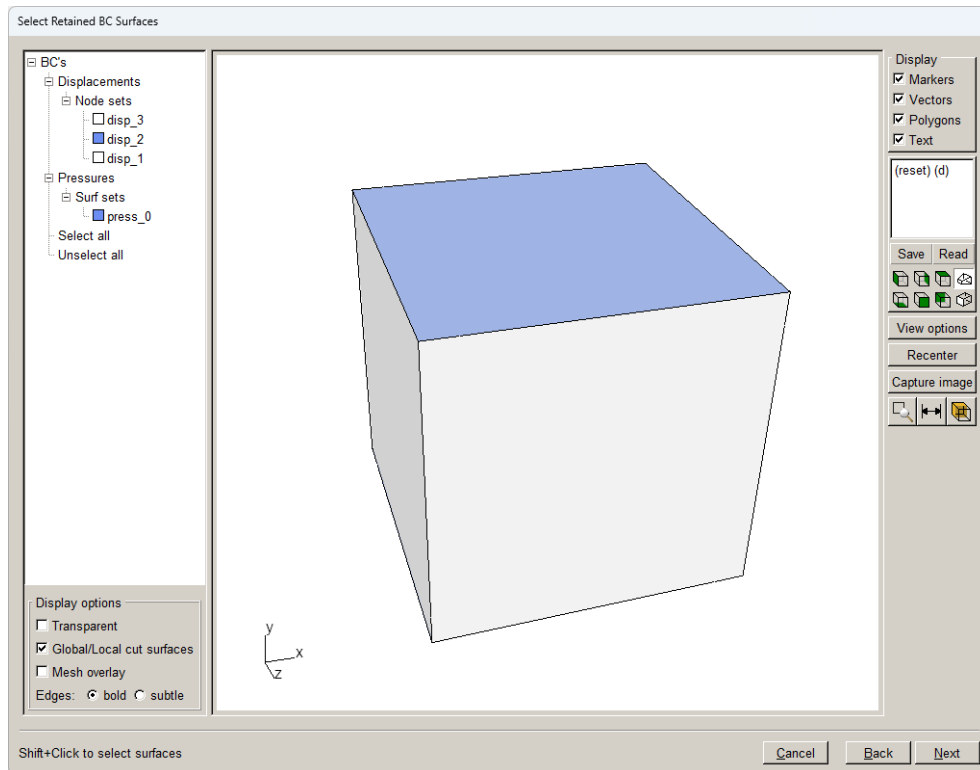


Figure 2.5 Select Retained BC Surfaces dialog – model is rotated.

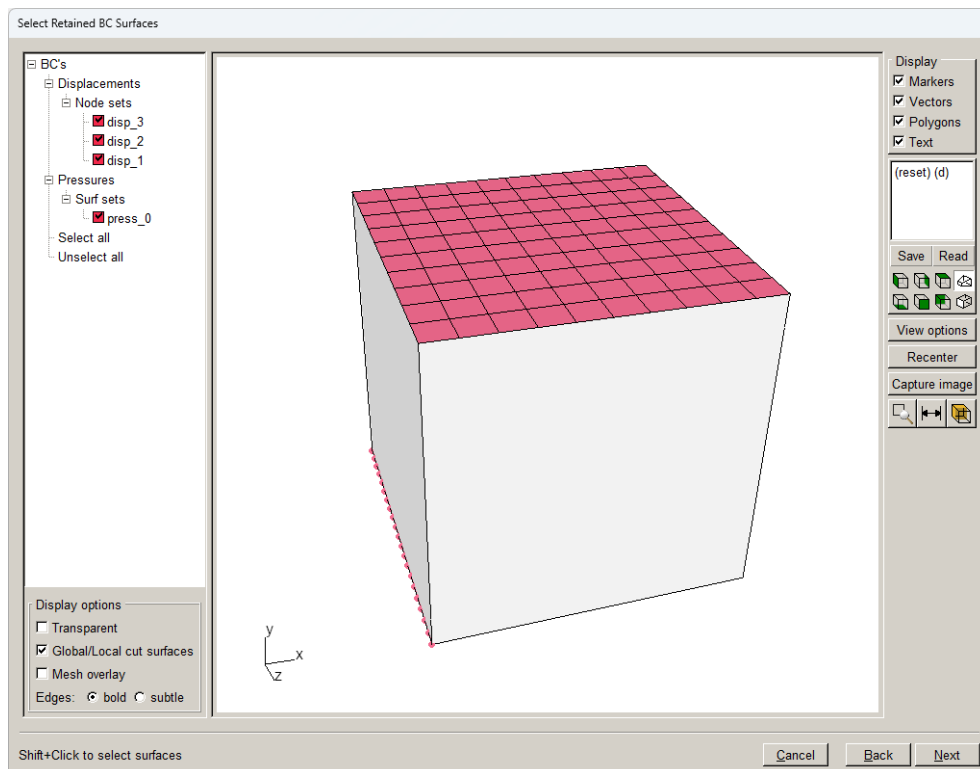


Figure 2.6 Select Retained BC Surfaces dialog after clicking Select all.

Step 1.3: Displaying the FE Model

The model is displayed in the main window. You can turn on the surface mesh and manipulate the view. The model should appear as in Fig 2.7, which shows that the mesh is retained on the top and bottom (not shown) surfaces where the boundary conditions are applied.

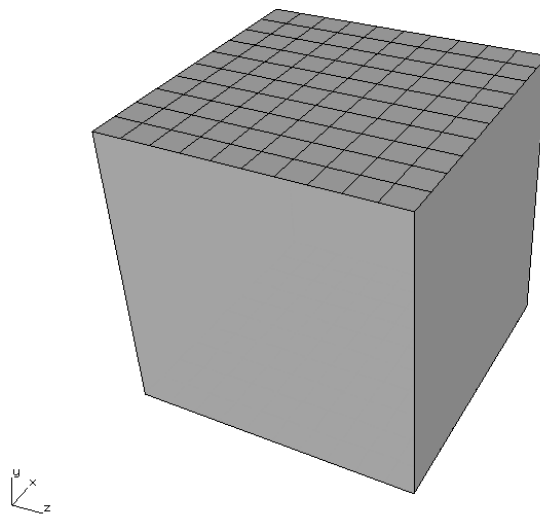


Figure 2.7 Model imported into FRANC3D, showing retained facets on the top.

2.2 Step 2: Insert Crack From File

From the FRANC3D menu, select **Cracks** → **Flaw From Files**, Fig 2.8. The first panel of the wizard, Fig 2.9, lets us choose *Cube_Crack.crk*; this file was created in Tutorial #1. Select **Accept** once the file has been selected.

The flaw being added is a circle with radius=1, centered on the front face and parallel to the xz-plane. This crack was created previously using the New Flaw Wizard with the save to file option selected; you can use the New Flaw Wizard if you do not have the .*crk* file.

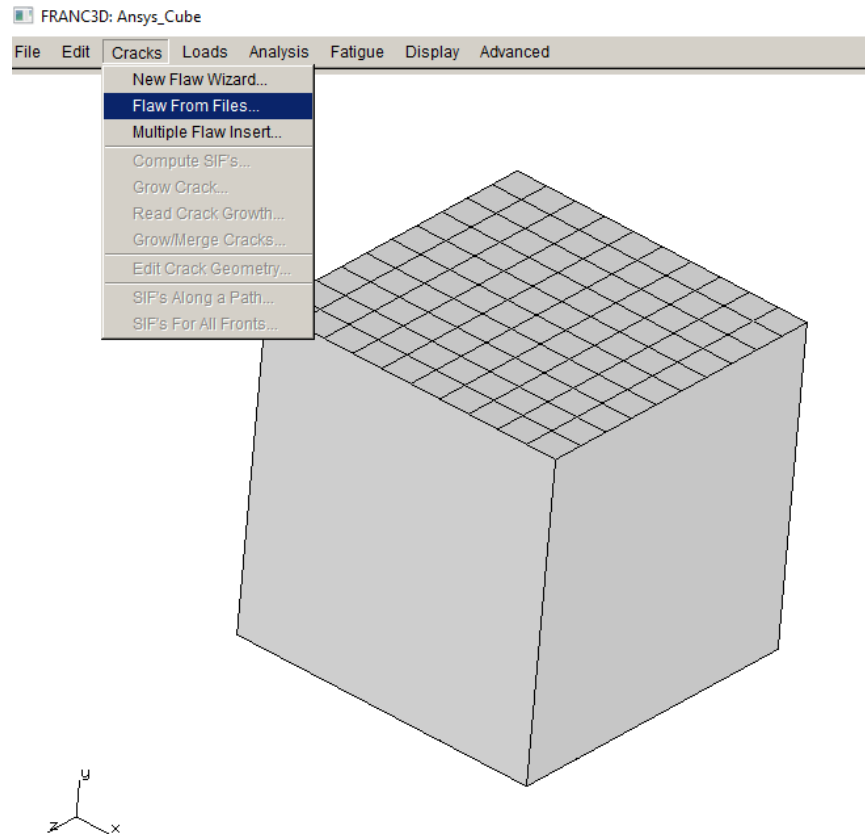


Figure 2.8 Flaw from Files option in Crack menu.

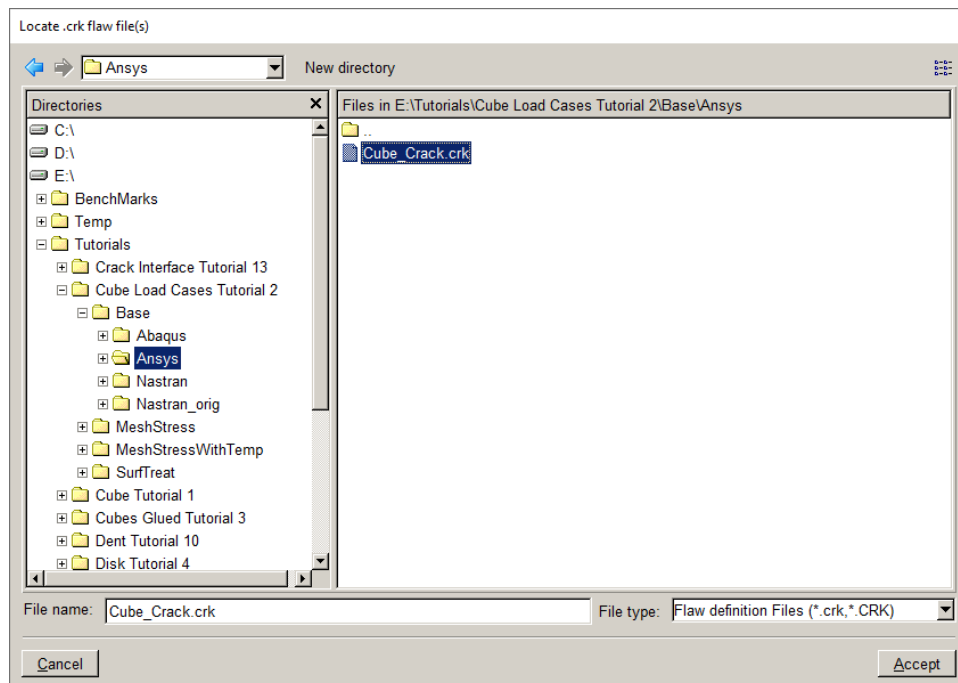


Figure 2.9 Flaw from file dialog to select .crk file.

The next panel of the wizard, Fig 2.10, allows you to adjust the location of the crack; a flaw from a file cannot be rotated. Leave the crack at its location and select **Next**. The final panel, Fig 2.11, allows you to define the template. The template radius is set to 0.05; leave all the other defaults and select **Finish**. Fig 2.12 shows the resulting remeshed cracked model. Note that the template radius is reduced from the default to improve the solution when we include CFC later.

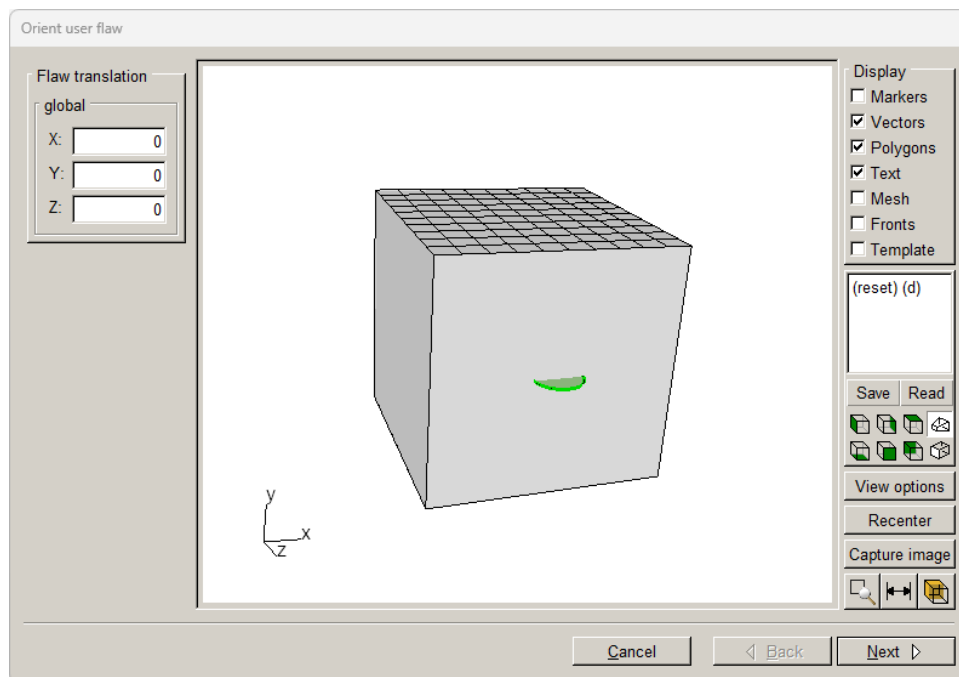


Figure 2.10 Flaw wizard panel to set location.

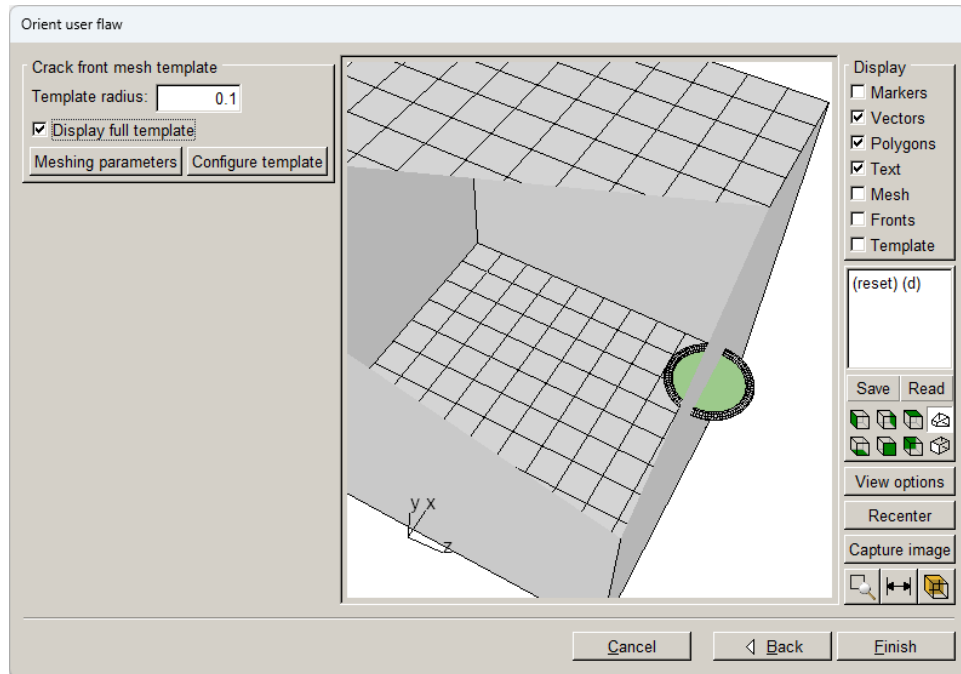


Figure 2.11 Flaw wizard panel to set the crack template mesh.

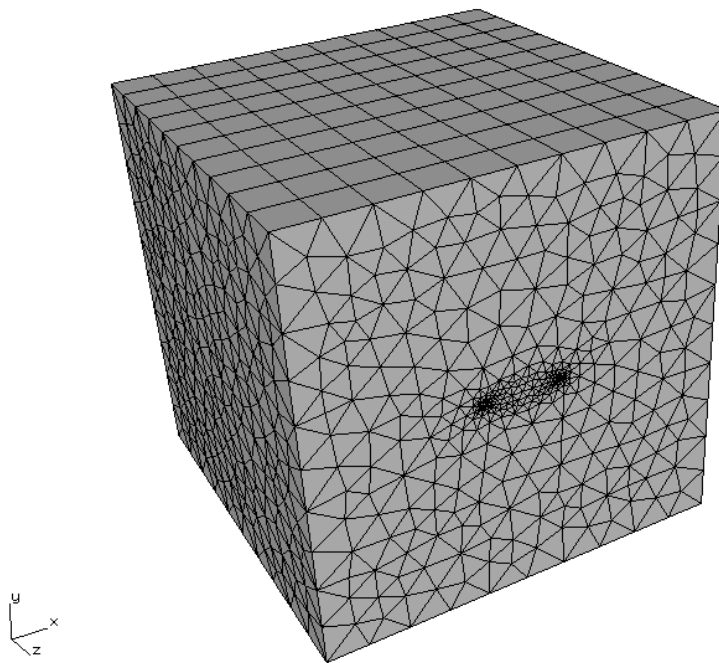


Figure 2.12 Remeshed cracked model.

2.3 Step 3: Apply Crack Surface Traction

In addition to the three applied load steps, we will add a CFT load step also. From the FRANC3D main menu, select **Loads** → **Crack Face Pressure/Traction**, Fig 2.13. Click **Add** in the dialog box shown in the top left panel of Fig 2.14. Choose Constant Crack Face Pressure, as shown in the top right panel of Fig 2.14; it is the default. The **Advanced** button is not used here; the default is to apply the CFT as a separate load step. Select **Next**, set the Pressure value to 1 in the next dialog, Fig 2.14 – bottom left, and select **Next**. Note that a positive value of pressure will tend to open the crack. The original dialog now contains a single entry, Fig 2.14 – bottom right; click **Accept**.

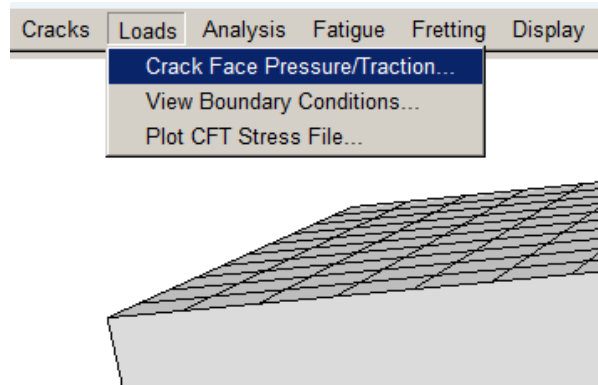


Figure 2.13 Crack Face Pressure/Traction menu

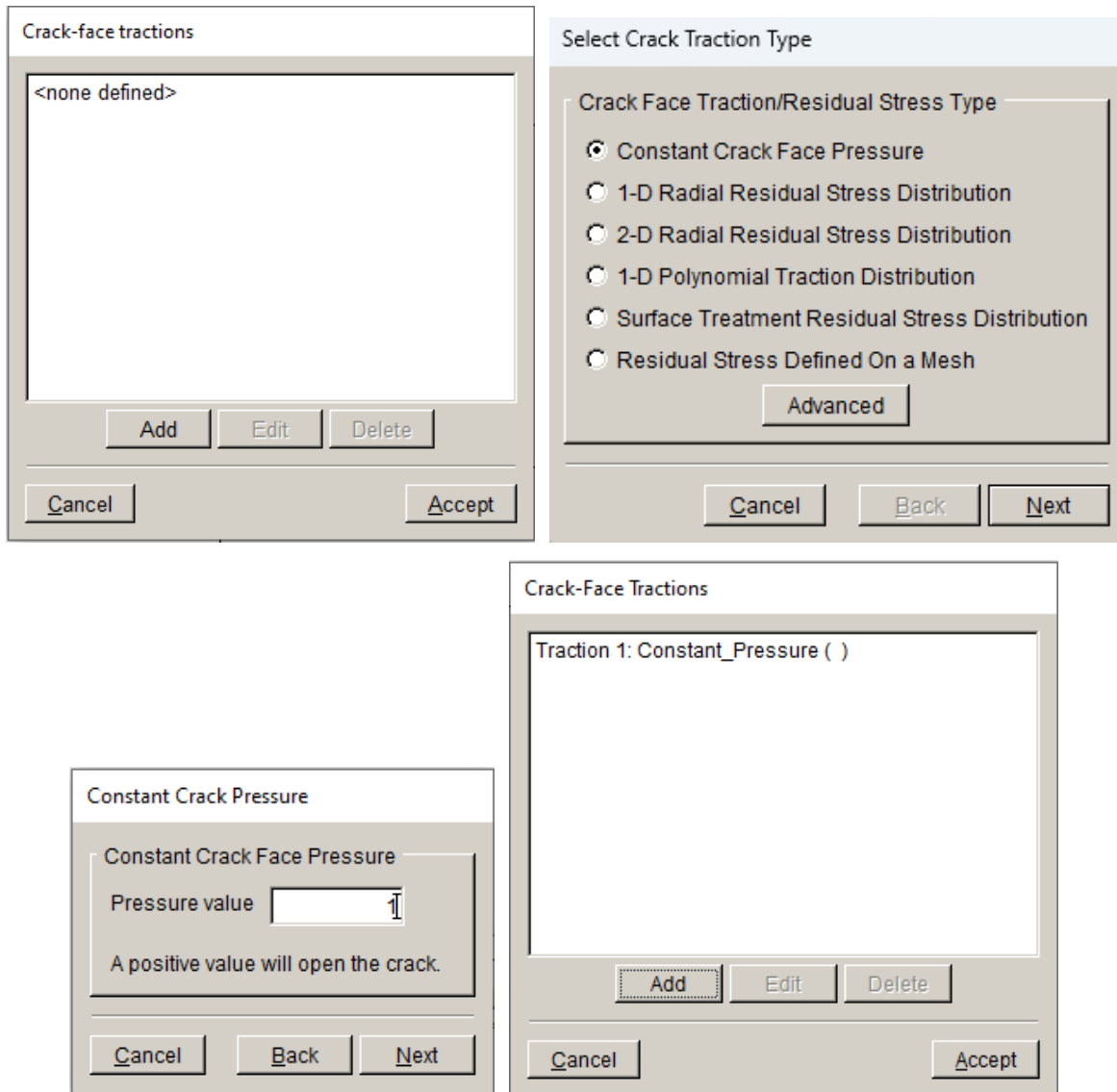


Figure 2.14 Crack face traction dialogs.

2.4 Step 4: Static Crack Analysis

From the FRANC3D menu, select **Analysis** → **Static Crack Analysis**. The first panel of the wizard, Fig 2.15, requests the file name for the FRANC3D database; call it *Cube_LoadCases.fdb*. Select **Next** after entering the File Name. Depending on the analysis code, proceed to the appropriate subsection.

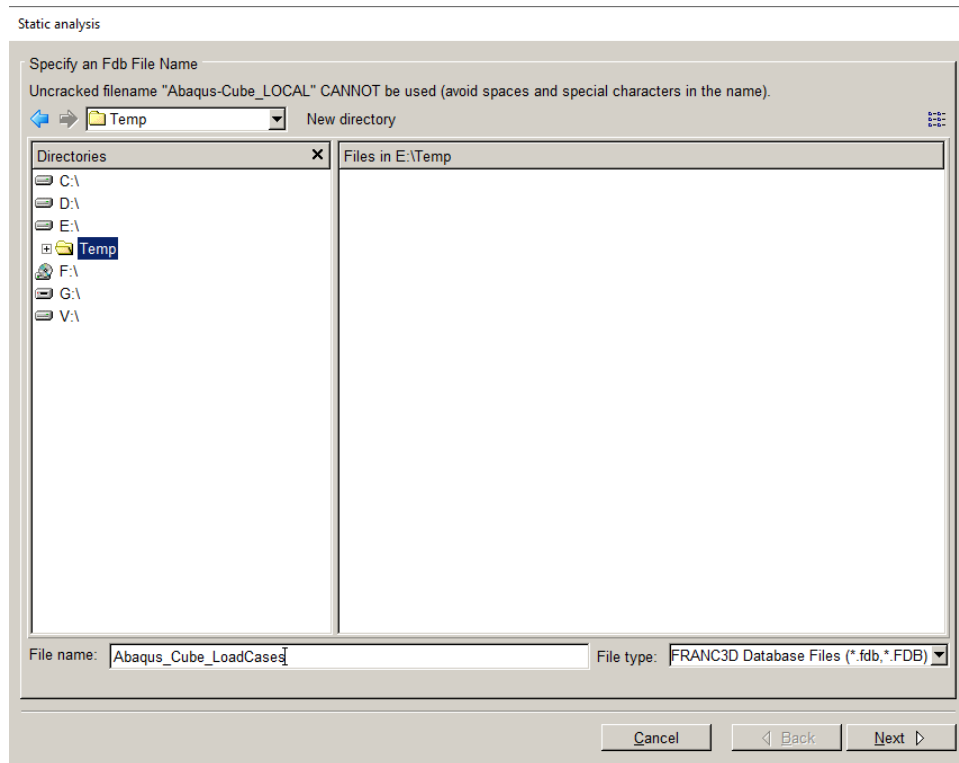


Figure 2.15 Static Analysis wizard first panel – File Name.

2.4.1 ABAQUS Analysis

If you are using ABAQUS, the next panel of the wizard, Fig 2.16a, allows you to specify the ABAQUS analysis options. The Apply crack face tractions box is checked automatically. There is no global model. Click **Finish** when ready to proceed with the ABAQUS analysis. When ABAQUS is finished running, proceed to Step 5. You can monitor the FRANC3D CMD window as well as the ABAQUS *.dat* and *.msg* files.

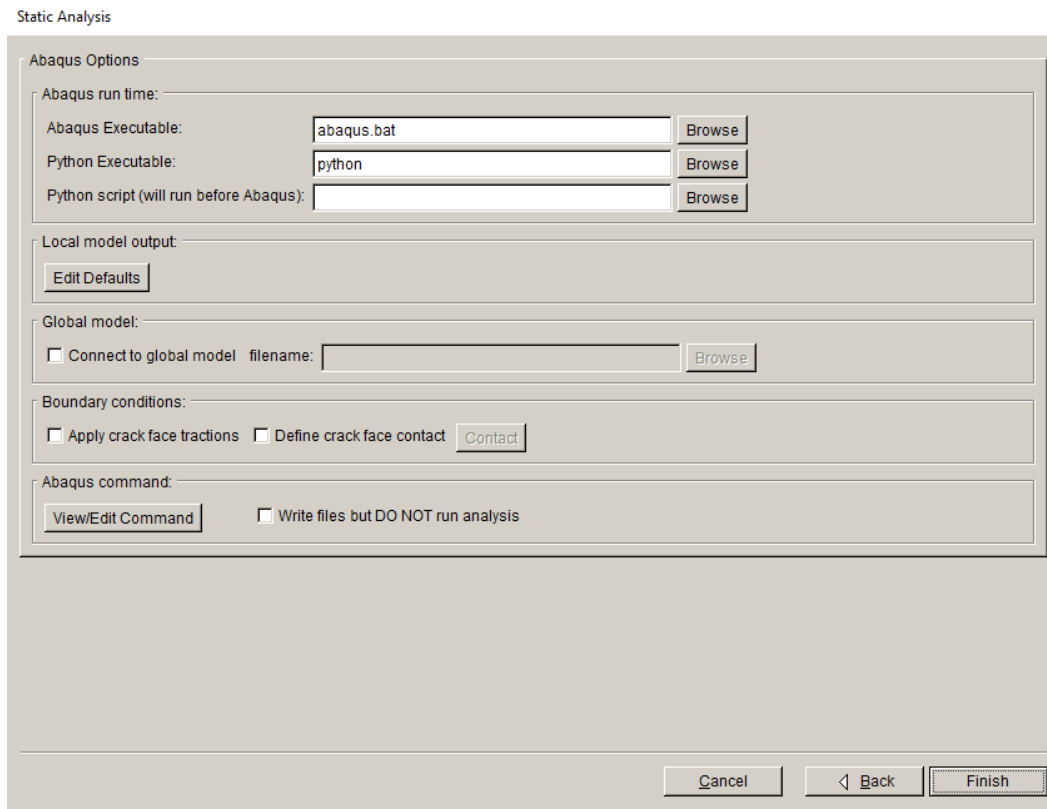


Figure 2.16a Static Analysis wizard second panel – ABAQUS output options.

2.4.2 ANSYS Analysis

If you are using ANSYS, the next panel of the wizard, Fig 2.16b, allows you to specify the ANSYS analysis options. The Apply crack face tractions box is checked automatically. There is no global model. Click **Finish** when ready to proceed with the ANSYS analysis. When ANSYS is done, proceed to Step 5. You can monitor the FRANC3D CMD window as well as the ANSYS *.log* and *.err* files.

Static analysis

ANSYS Options

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☐ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact ☐ Initial strain

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 2.16b Static Analysis wizard second panel – ANSYS output options.

2.4.3 NASTRAN Analysis

If you are using NASTRAN, the next panel of the wizard, Fig 2.16c, allows you to specify the NASTRAN analysis and output options. The Apply crack face tractions box is checked automatically. There is no global model. Click **Finish** when ready to proceed with the NASTRAN analysis. When NASTRAN is done, proceed to Step 5. You can monitor the FRANC3D CMD window as well as the NASTRAN *.f06* and *.msg* files.

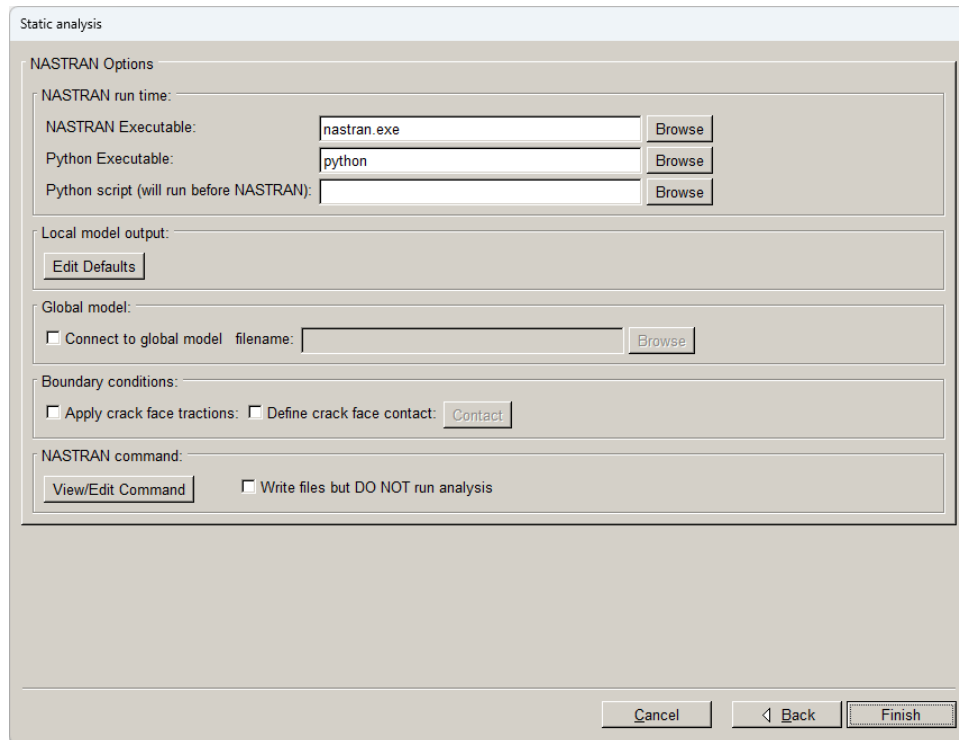


Figure 2.16c Static Analysis wizard second panel – NASTRAN output options.

2.5 Step 5: Compute SIFs

We compute stress intensity factors (SIFs) for this crack. If you ran the analysis directly from FRANC3D, then the results (*.dtp* or *.pch*) file is read automatically. If you run the analysis on a different PC, you should read the results file using **File** → **Read Results**.

From the FRANC3D menu, select **Cracks** → **Compute SIFs**. When the Compute SIFs dialog is displayed, Fig 2.17, you can select the **Advanced** button and make sure the reference temperature is correct (right side image in Fig 2.17). Select **Finish** and the SIFs Plot dialog is displayed, Fig 2.18. The initial plot shows the sum of K_I for all load cases. You can view all stress intensity factor (SIF) modes for the four load cases using the Analysis Load Step dropdown-menu. Fig 2.19 shows the K_I curves for the four load cases.

Note that load 2 has a negative K_I . This is the compression load step, and we did not include CFC in the analysis, so the crack faces penetrate each other giving a negative crack opening. In the next section, we will turn on CFC and redo the analysis.

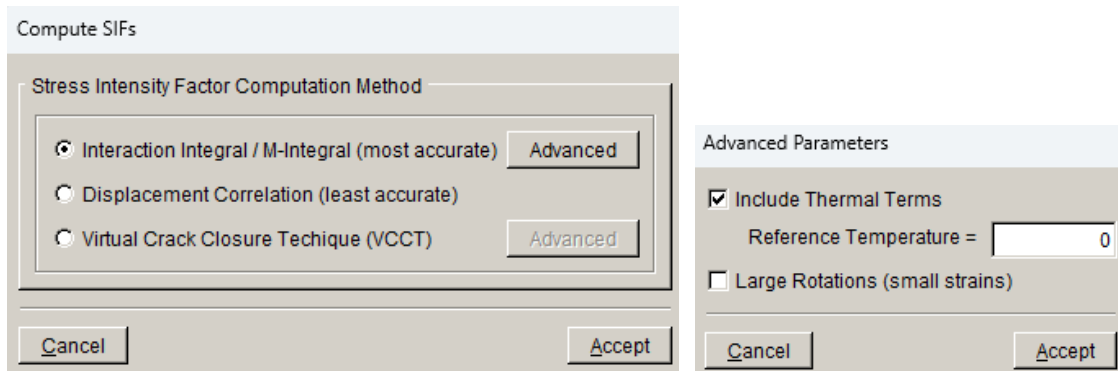


Figure 2.17 Compute SIFs and Advanced Parameters dialogs.

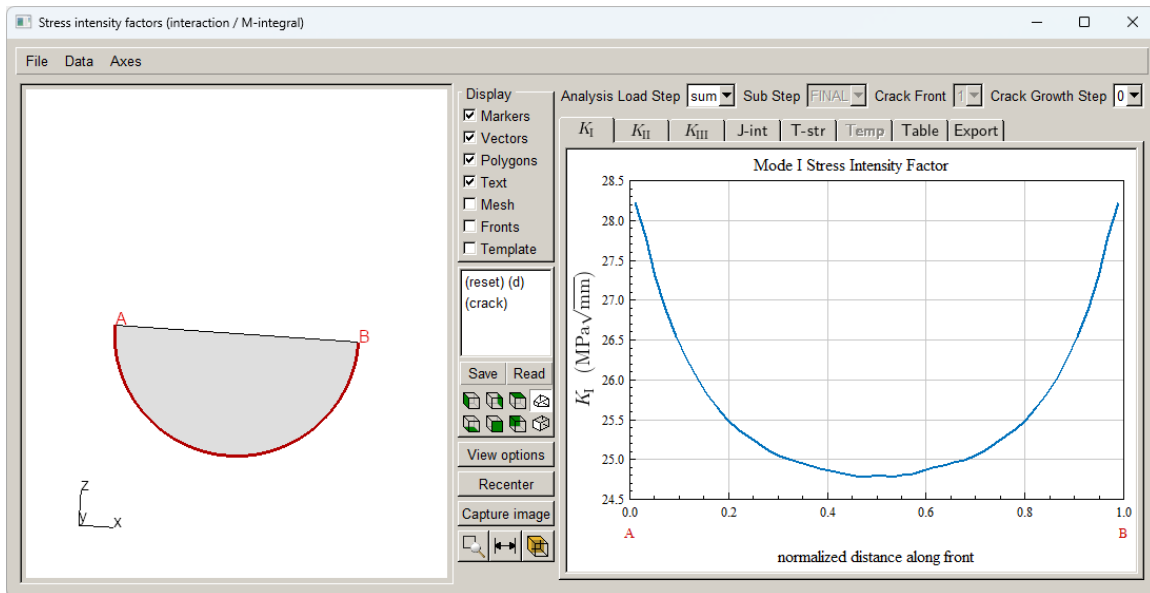


Figure 2.18 Stress Intensity Factor dialog – sum of Mode I SIFs.

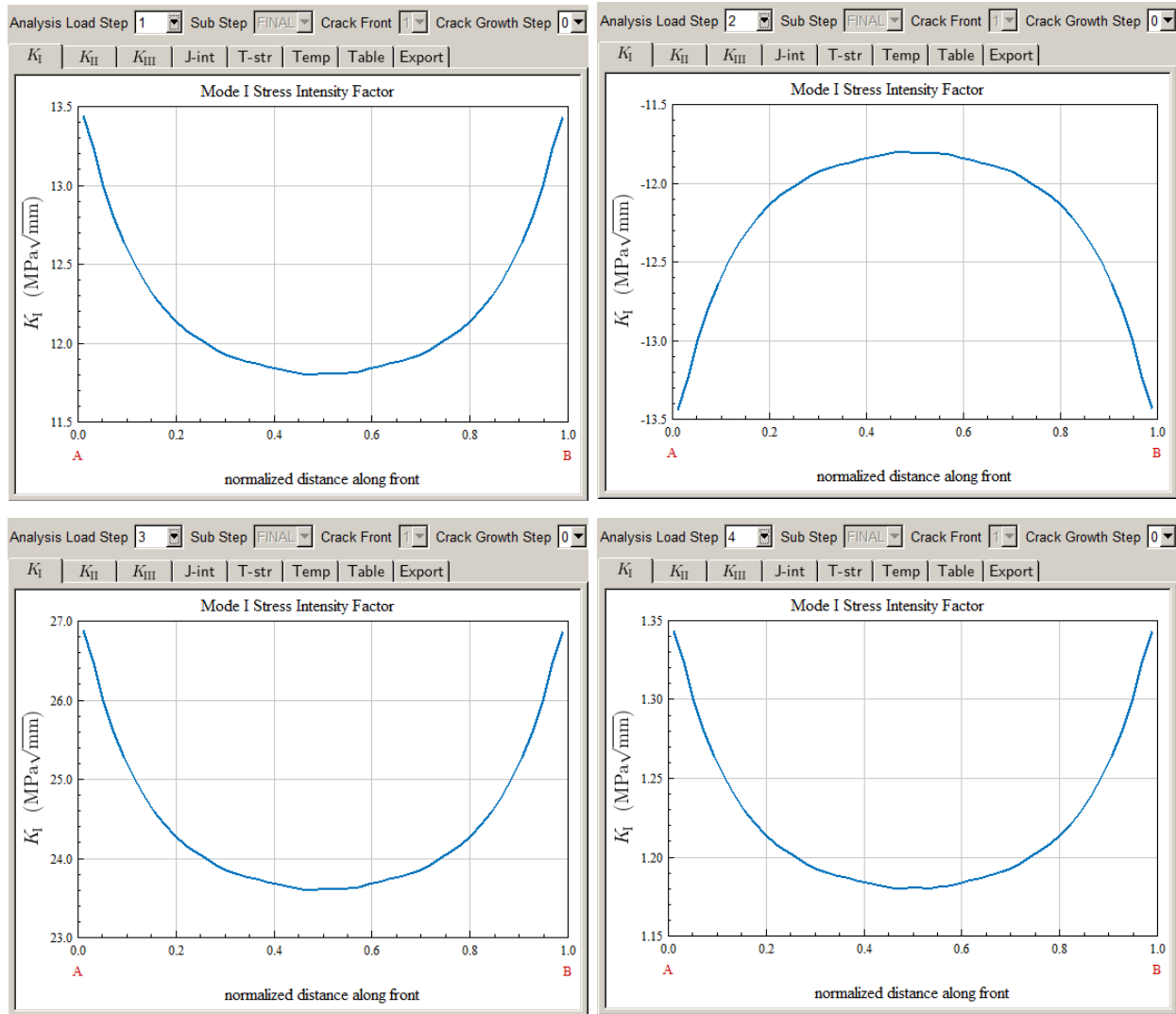


Figure 2.19 Stress Intensity Factor dialog – Mode I SIFs for each of the 4 load cases.

2.6 Step 6: Re-run static crack analysis with Crack Face Contact

We rerun the analysis with crack face contact (CFC). From the FRANC3D menu, select **Analysis** → **Static Crack Analysis**. The first panel of the wizard (see Fig 2.15) lets us specify the file name for the FRANC3D database; call it *Cube_LC_CFC.fdb*. Select **Next** after entering the file name. Proceed to the appropriate sub-section for your analysis code.

2.6.1 ABAQUS Analysis

If you are using ABAQUS, the next panel, Fig 2.20a, allows you to specify the ABAQUS analysis and output options. The Apply crack face tractions box is checked automatically. We can check the Define crack face contact box, which will enable the **Contact** button. Click the **Contact** button and the dialog shown in Fig 2.21a is displayed. You must ensure that these parameters do not conflict with any ABAQUS data already in the model. We are creating a new surface interaction, so we must enter the surface interaction name (cfc_si in Fig 2.21a). The coefficient of friction is 0.5; the other parameters are left at their default values. Click **Accept**, and then click **Finish** in the previous panel (see Fig 2.20a) when ready to proceed with the ABAQUS analysis.

The screenshot shows the 'Static Analysis' wizard, third panel, titled 'Static Analysis'. The panel is divided into several sections:

- Abaqus Options:**
 - Abaqus run time:**
 - Abaqus Executable:
 - Python Executable:
 - Python script (will run before Abaqus):
 - Local model output:**
 -
 - Global model:**
 - ☐ Connect to global model filename:
 - Boundary conditions:**
 - ☒ Apply crack face tractions
 - ☒ Define crack face contact
 - Abaqus command:**
 - ☐ Write files but DO NOT run analysis

At the bottom right, there are three buttons: , , and .

Figure 2.20a Static Analysis wizard third panel – ABAQUS output options.

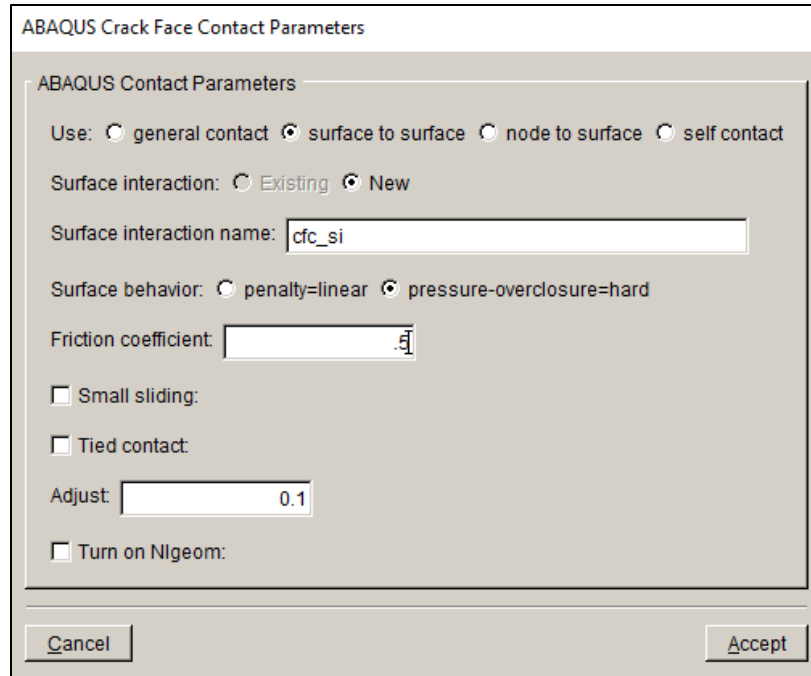


Figure 2.21a ABAQUS contact options.

2.6.2 ANSYS Analysis

If you are using ANSYS, the next panel, Fig 2.20b, allows you to specify the ANSYS analysis and output options. The Apply crack face tractions box is checked automatically. We can check the Define crack face contact box, which enables the **Contact** button. Click the **Contact** button and the dialog shown in Fig 2.21b is displayed. In practice, you must make sure that these parameters do not conflict with ANSYS data already in the model. We can leave all the defaults except we set the coefficient of friction to be 0.5. Click **Accept** and then click **Finish** in the previous panel (see Fig 2.20b) when ready to proceed with the ANSYS analysis.

Static analysis

ANSYS Options

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☐ Connect to global model filename:

Boundary conditions:

☒ Apply crack face tractions ☒ Define crack face contact ☐ Initial strain

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 2.20b Static Analysis wizard third panel – ANSYS output options.

ANSYS Crack Face Contact Parameters

ANSYS Contact Parameters

Material id: Friction coef:

Real Constant id:

Normal penalty stiffness: (negative for absolute value)

Penetration tolerance: (negative for absolute value)

ET 170 id: ET 174 id:

Contact Algorithm: ☒ augmented lagrange ☐ penalty ☐ mpc ☐ lagrange + penalty ☐ lagrange

Gap/Penetration Adjustment: ☐ none ☐ close gap ☐ reduce pene ☐ close/reduce ☒ default/cont

Asymmetric Contact: ☒ off ☐ on

Initial Penetration/Gap: ☒ include ☐ exclude

Contact Stiffness Update: ☒ each iteration ☐ each load step ☐ each iteration of substep

Contact behavior: ☒ standard ☐ rough ☐ no_separation ☐ bonded ☐ bonded_always

Number of substeps: max min:

☐ Include Crack Front Nodes:

Figure 2.21b ANSYS contact options.

2.6.3 NASTRAN Analysis

If you are using NASTRAN, the next panel, Fig 2.20c, allows us to specify the NASTRAN analysis and output options. The Apply crack face tractions box is checked automatically. We need to check the Define crack face contact box, which will enable the **Contact** button. Click the **Contact** button and the dialog shown in Fig 2.21c is displayed. In general, you will need to make sure that these parameters do not conflict with any NASTRAN data already in the model. We leave all the default values except for the coefficient of friction, which is set to 0.5. Click **Accept** and then click **Finish** in the previous panel (see Fig 2.20c) when ready to proceed with the NASTRAN analysis.

The image shows a software dialog box titled "Static analysis". It contains several sections for configuring NASTRAN options:

- NASTRAN Options:**
 - NASTRAN run time:**
 - NASTRAN Executable:
 - Python Executable:
 - Python script (will run before NASTRAN):
- Local model output:**
 -
- Global model:**
 - ☐ Connect to global model filename:
- Boundary conditions:**
 - ☐ Apply crack face tractions: ☒ Define crack face contact:
- NASTRAN command:**
 - ☐ Write files but DO NOT run analysis

At the bottom right, there are three buttons: , , and .

Figure 2.20c Static Analysis wizard third panel – NASTRAN output options.

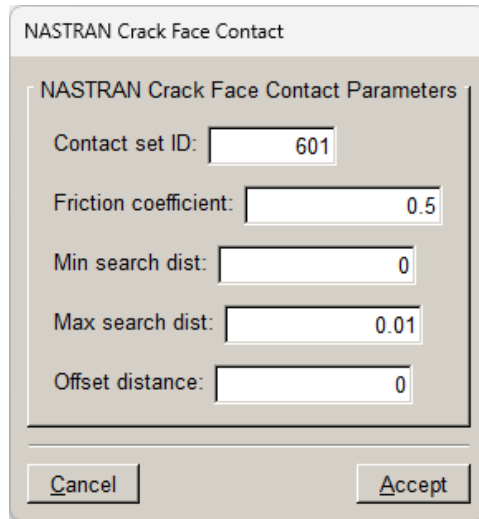


Figure 2.21c NASTRAN contact options.

2.7 Step 7: Compute SIFs with Crack Face Contact

Once the analysis is finished, we compute stress intensity factors. If you ran directly from FRANC3D, then the model already exists and the results (*.dtp* or *.pch*) file will be read automatically. From the FRANC3D menu, select **Cracks** → **Compute SIFs**. When the Compute SIFs dialog is displayed, Fig 2.22, select **Finish** to display the SIFs Plot dialog, Fig 2.23. The initial plot shows the sum of K_I for all load cases.

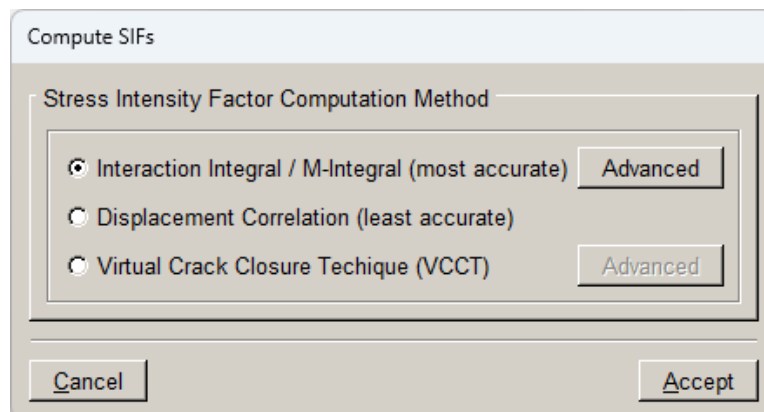


Figure 2.22 Compute SIFs and Advanced Parameters dialogs.

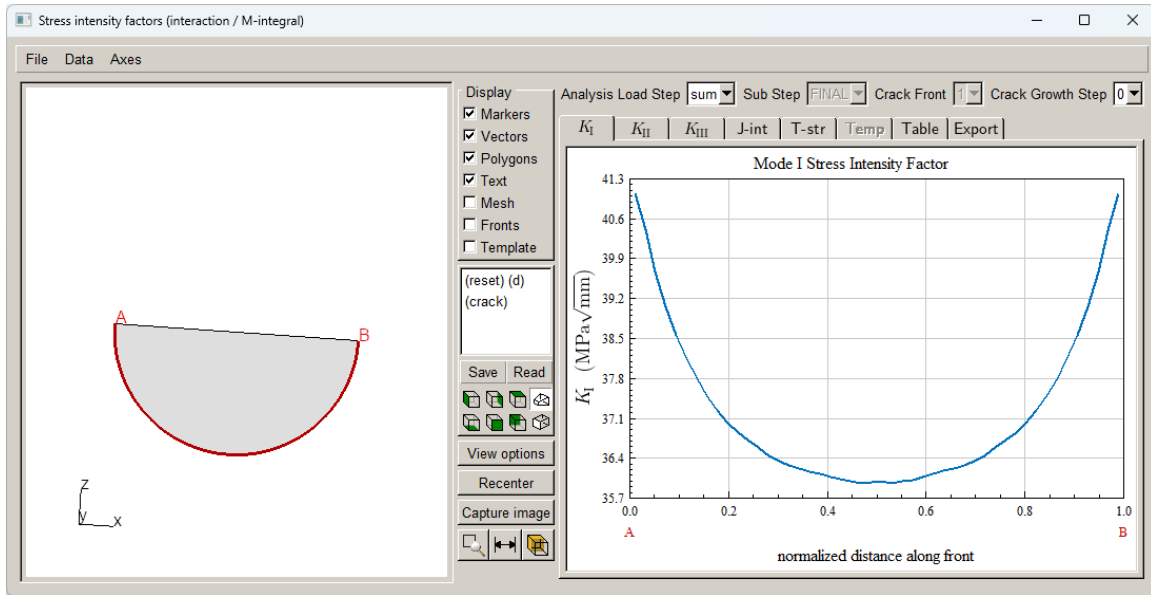


Figure 2.23 Stress Intensity Factor dialog – sum of Mode I SIFs.

Fig 2.24 shows the K_I values for the second load case. Note that the value is close to 0.0. To verify that the SIF results from the M-integral are reasonable, we compare them to the displacement correlation SIFs. In the Compute SIFs dialog, choose Displacement Correlation, Fig 2.25. The Mode I SIFs for load 2 are shown in Fig 2.26. The results are comparable to those for M-integral, not exactly 0.0, but close. You can try different mesh refinement and/or different contact options to improve the results, to get Mode I SIFs closer to zero.

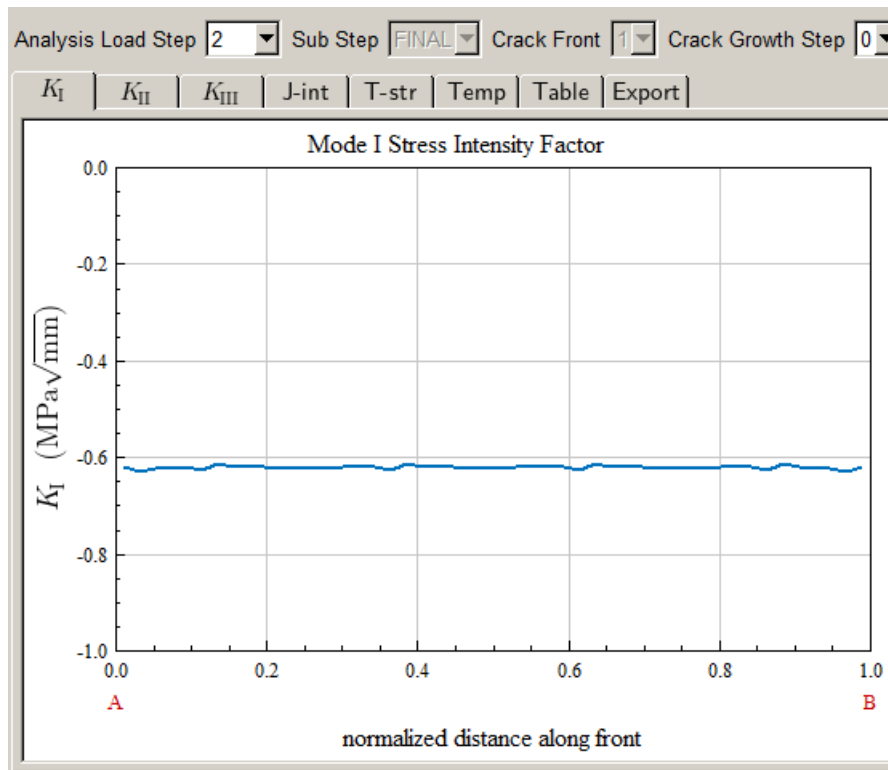


Figure 2.24 Stress Intensity Factor dialog – M-integral Mode I SIFs for the second load case.

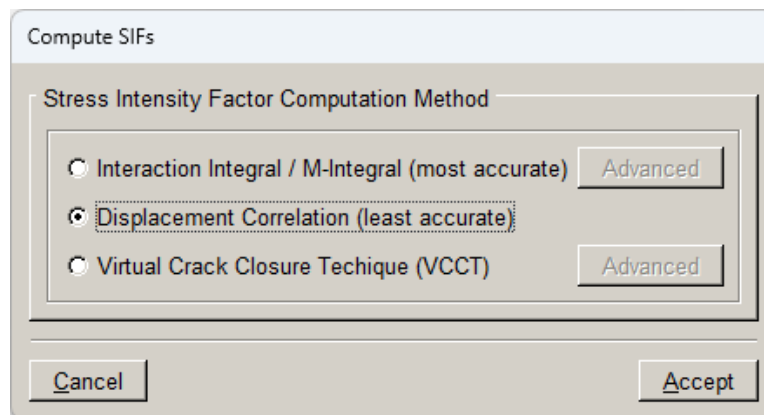


Figure 2.25 Compute SIFs dialog using Displacement Correlation.

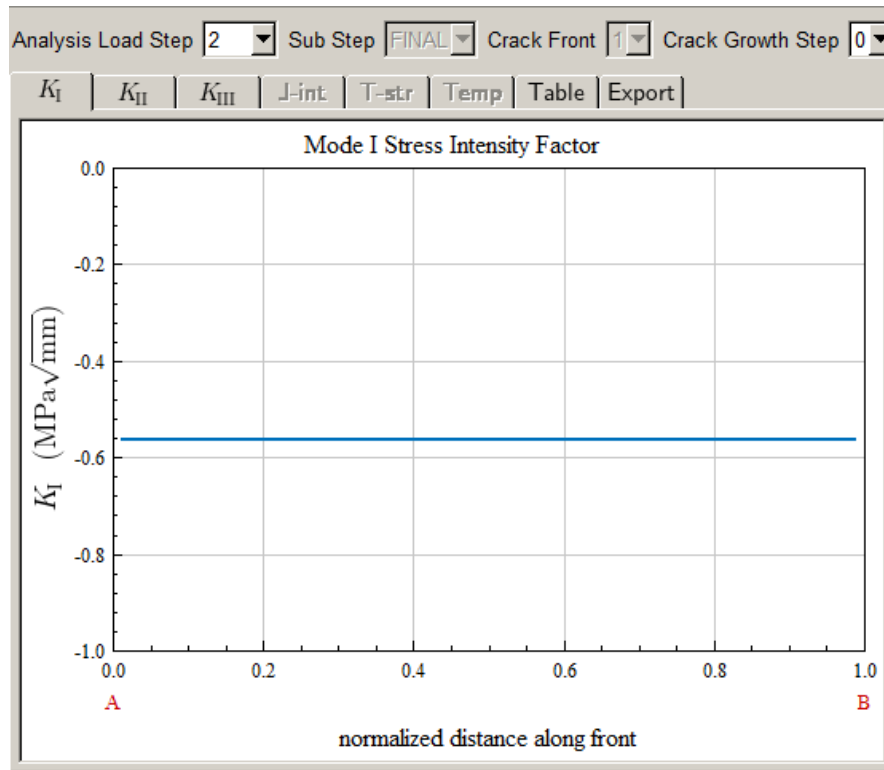


Figure 2.26 Stress Intensity Factor dialog – Displacement Correlation Mode I SIFs for second load case

2.8 Step 8: Apply Surface Treatment Residual Stress

To simulate residual stress due to a surface treatment, such as shot-peening, FRANC3D allows you to define a 1-D stress profile as a function of distance from a surface. We demonstrate this using the cube model with only one far-field tension load step (same model as tutorial #1).

Start with the FRANC3D GUI (see Fig 2.1) and select **File** → **Import**. In the dialog (see Fig 2.2), choose Complete Model. Switch the Mesh File Type radio button (see Fig 2.3) to the appropriate FE type. Select the appropriate “cube” file, and then select **Next**. Retain all the boundary condition surfaces (see Figs 2.4 – 2.6). The model will appear as in Fig 2.7.

Insert the same crack from file as before (see Figs 2.8 – 2.12). At this stage, we could do a static analysis to verify that the model is correct and compute the SIFs for the far-field loading only;

the Mode I SIF should be the same as shown in the top left panel of Fig 2.19. We leave this as an exercise for the reader. The next step is to define the surface-treatment residual stress.

From the FRANC3D main menu, select **Loads** → **Crack Face Pressure/Traction**, Fig 2.27. Click on **Add** in the dialog box shown in the left panel of Fig 2.28. Choose Surface Treatment Residual Stress, as shown in the right panel of Fig 2.28, and select **Next**. The dialog shown in Fig 2.29 is displayed. We use **Read From File** to import the residual stress distribution from a *.txt* file.

Using the Open File dialog shown in Fig 2.30, select the file *surf_treat_res.txt*, and then select **Accept**. The distribution is read and displayed in the dialog, Fig 2.31. Select **Next** on this dialog panel to choose the surface of the model that is treated. The dialog shown in Fig 2.32 appears with all surfaces shown as blue; use the Shift key and the left mouse button to select the front surface (+z face) of the model. Once selected, the surface is colored red/pink. Select **Next** and the original Add Traction dialog is displayed with one entry, Fig 2.33; select **Accept**.

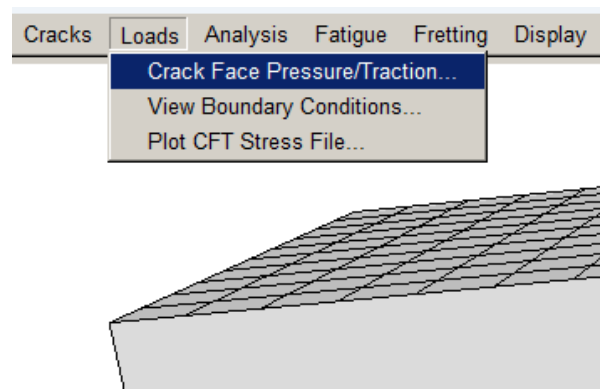


Figure 2.27 Loads – Crack Face Pressure/Traction menu item selected.

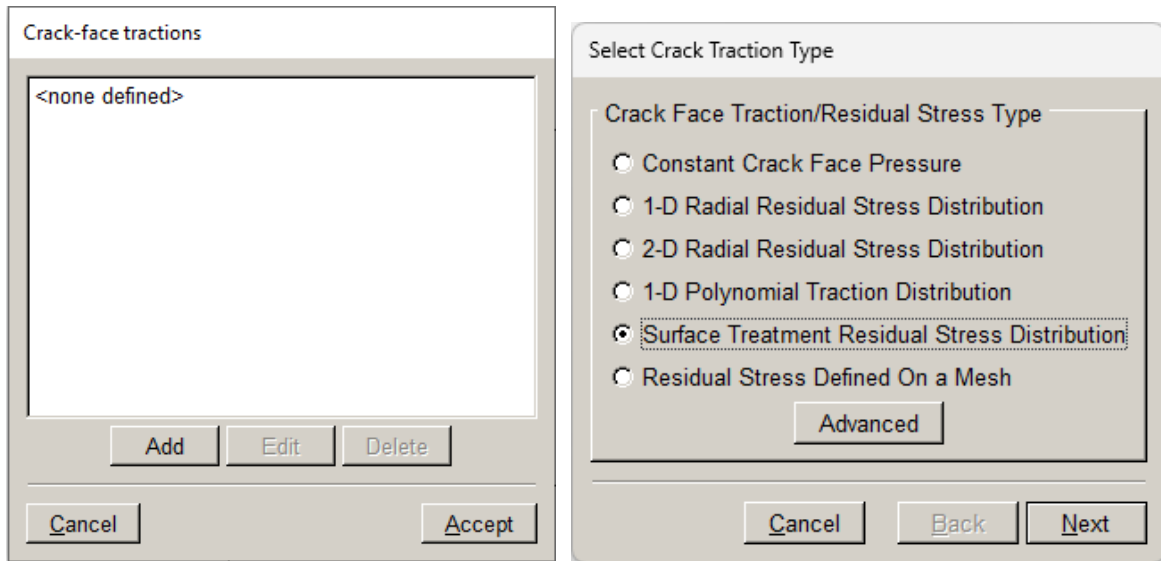


Figure 2.28 Crack Face Traction wizard panels; surface treatment selected.

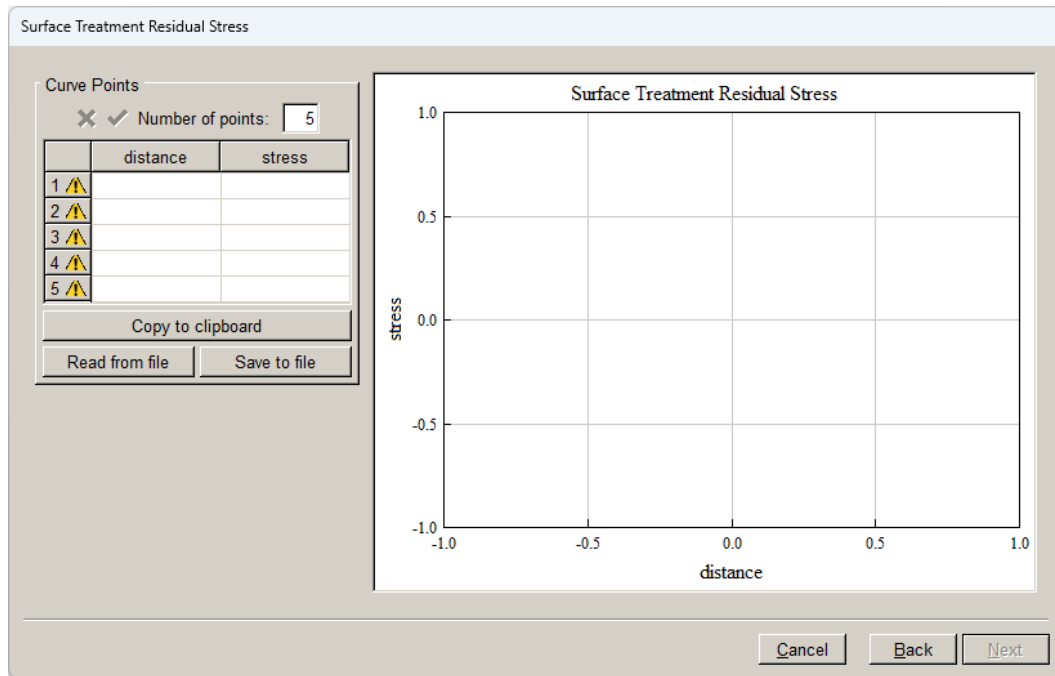


Figure 2.29 Surface treatment residual stress panel.

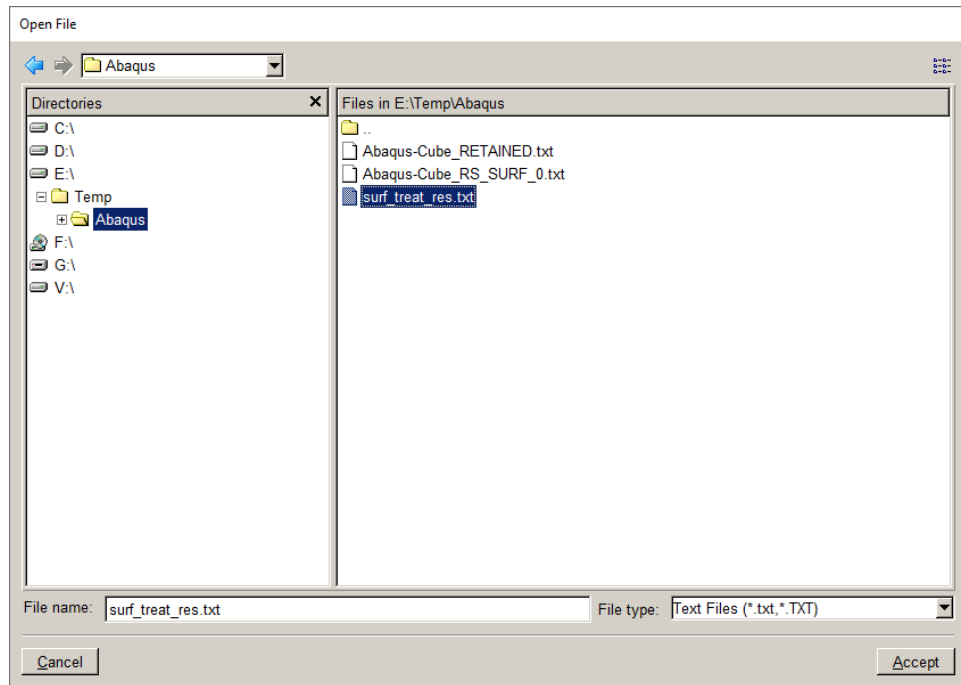


Figure 2.30 Select file for surface treatment residual stress.

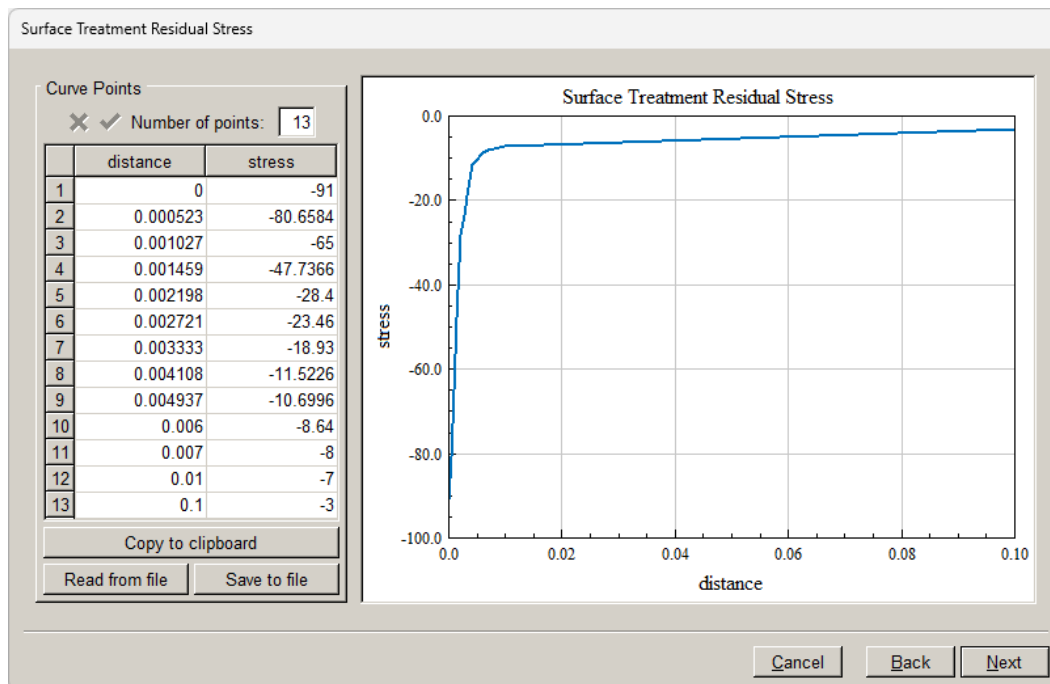


Figure 2.31 Surface treatment residual stress panel - residual stress distribution displayed.

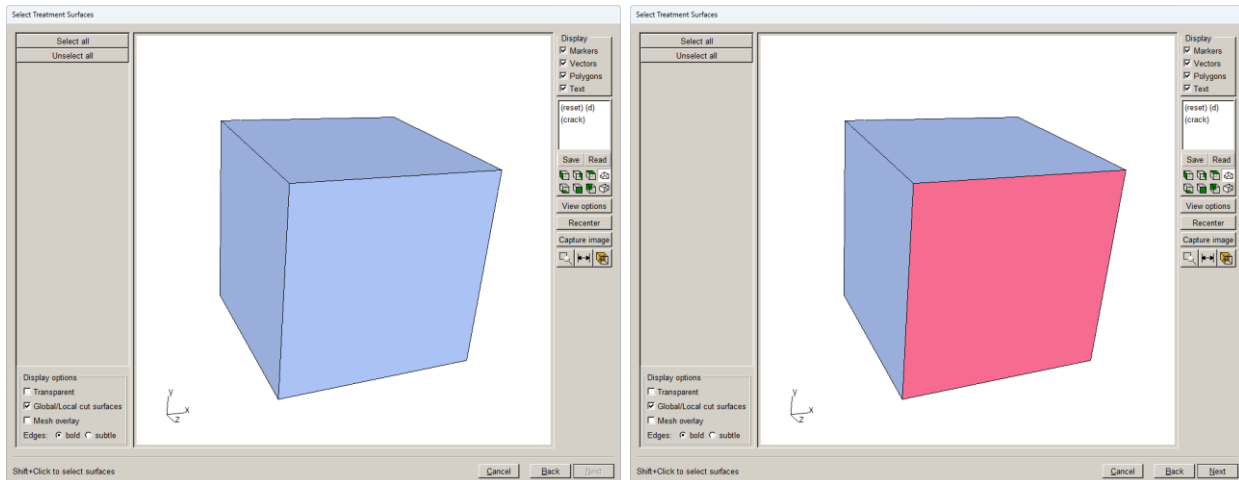


Figure 2.32 Surface treatment residual stress panel to select the treated surface.

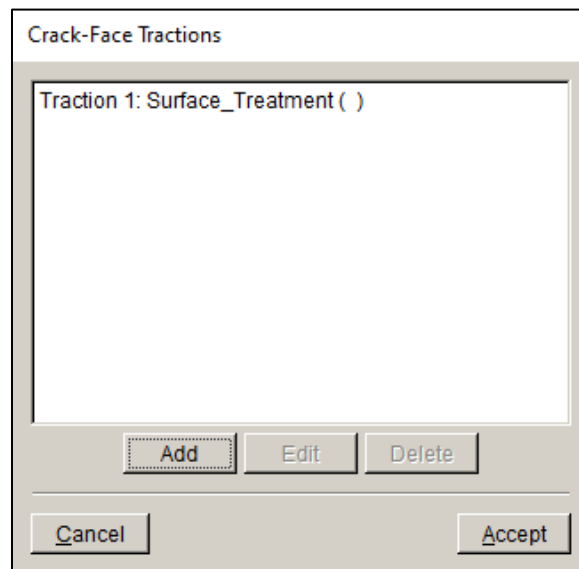


Figure 2.33 Crack Face Traction wizard panel - surface treatment added.

From the FRANC3D menu, select **Analysis** → **Static Crack Analysis**. Specify the file name for the FRANC3D database; we call it *Cube_surf_treat.fdb*; select **Next** after entering the file name. The next panel (see Fig 2.16) allows you to specify the analysis and output options. Select **Finish** when ready to proceed with the analysis.

When the analysis is finished, from the FRANC3D menu, select **Cracks** → **Compute SIFs**. When the Compute SIFs dialog is displayed select **Finish** and the SIFs Plot dialog is displayed, Fig 2.34. The initial plot shows the sum of K_I for both load cases. The lower two panels of Fig 2.34 show the K_I values for the first and second load cases separately. The values for the first load are the same as previously computed. The second load produces a negative SIF indicating that this residual stress tends to keep the crack from opening.

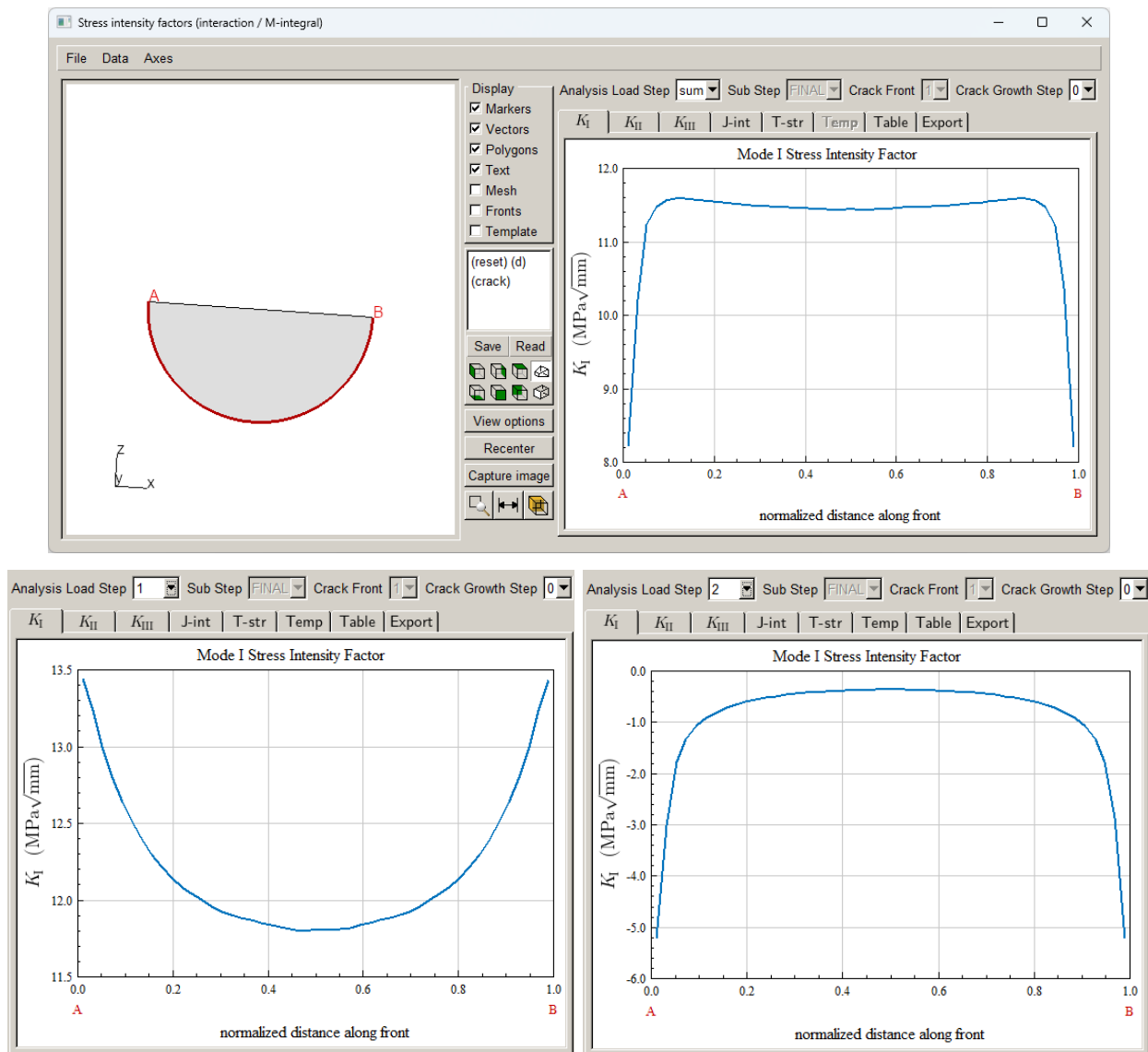


Figure 2.34 Mode I SIF plots showing the sum (top) and the individual load step values; surface treatment load step produces a negative SIF (bottom right).

2.9 Step 9: Apply Tractions from Mesh-Based Stress

The mesh-based stress option provides the most flexibility for defining CFT. We demonstrate this option using the simple cube model with only one far-field tension load step (same model as tutorial #1).

2.9.1 *Uncracked Model Stress*

The uncracked “stress” model will typically be the same as the model that we use for crack growth simulations. If it is different, it must capture the stresses where the crack is located. In addition to the FE specific results files described in the subsections below, a *.dtp* file can also be used to store the nodal stresses.

2.9.1.1 *ANSYS Mesh-Stress*

To create ANSYS mesh-based stresses, we start with the cube model and define a varying far-field stress on the top surface of the cube, Fig 2.35. This creates a stress field that varies in the x-direction, Fig 2.36.

FRANC3D can read the ANSYS stress listing, Fig 2.37. In ANSYS, turn off PowerGraphics, change the format for the listing if needed (using the ‘/format’ command), create the nodal stress component listing (using the ‘prnsol,s’ command), and save the listing to a file; we call it *Ansys_Cube_surf_gradient.str*. The stress listing must be in the global Cartesian coordinate system. We also save the corresponding *Ansys_Cube_surf_gradient.cdb* file.

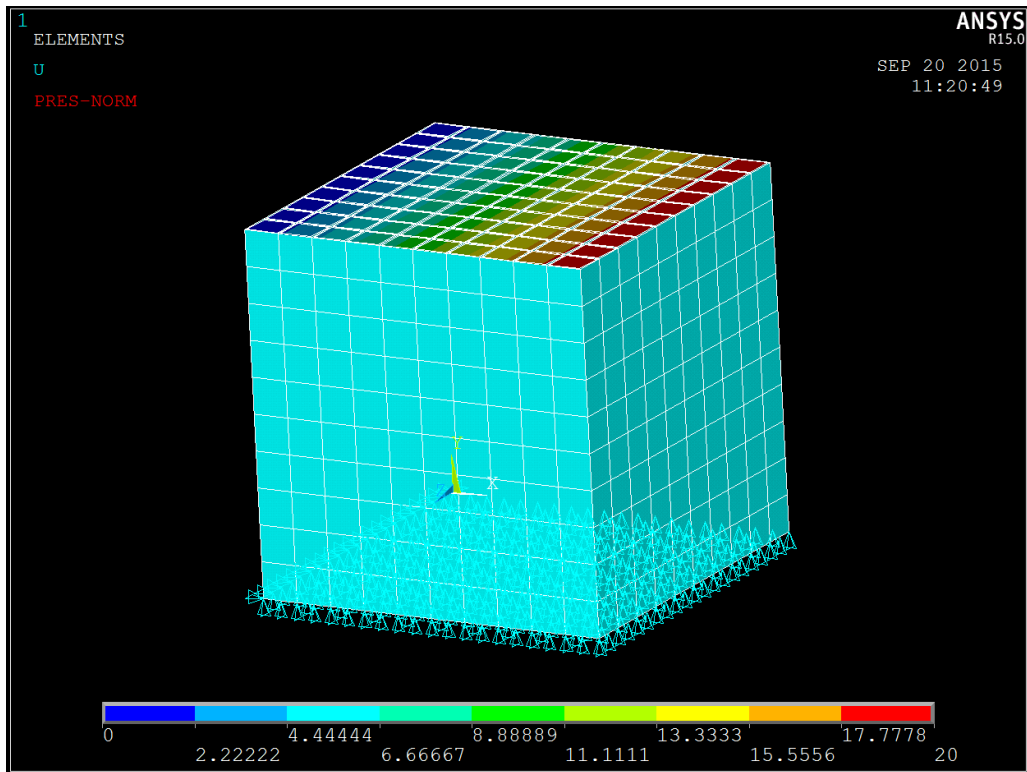


Figure 2.35 Arbitrary far-field surface pressure.

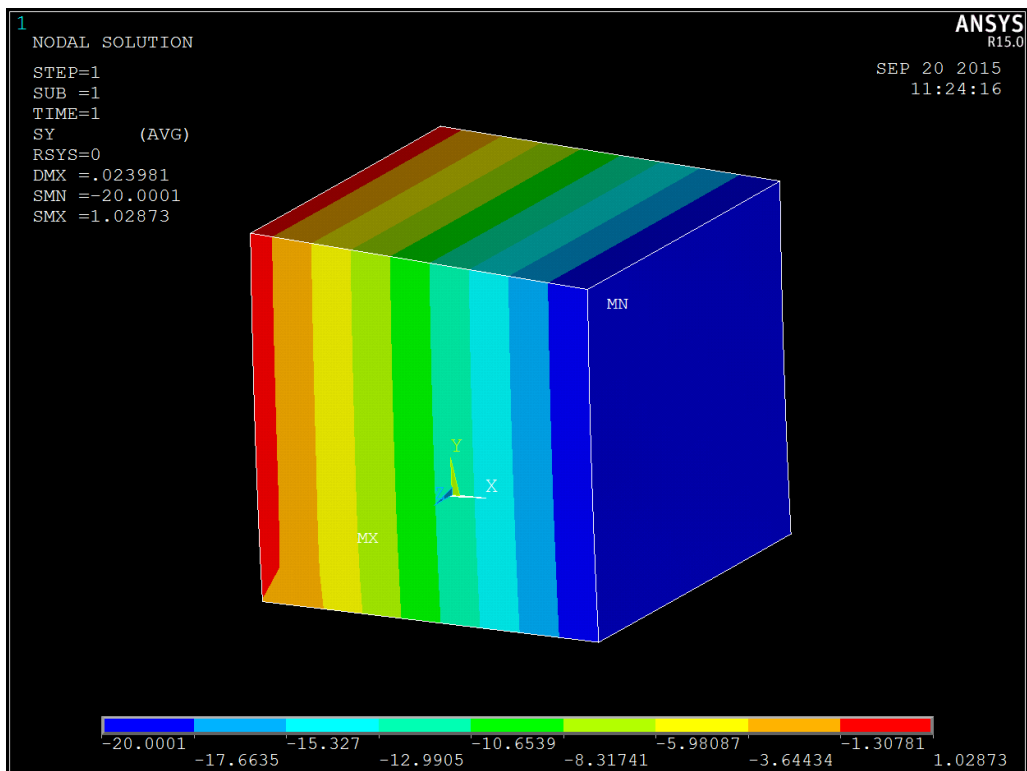


Figure 2.36 Stress in the y-direction for the arbitrary far-field surface pressure.

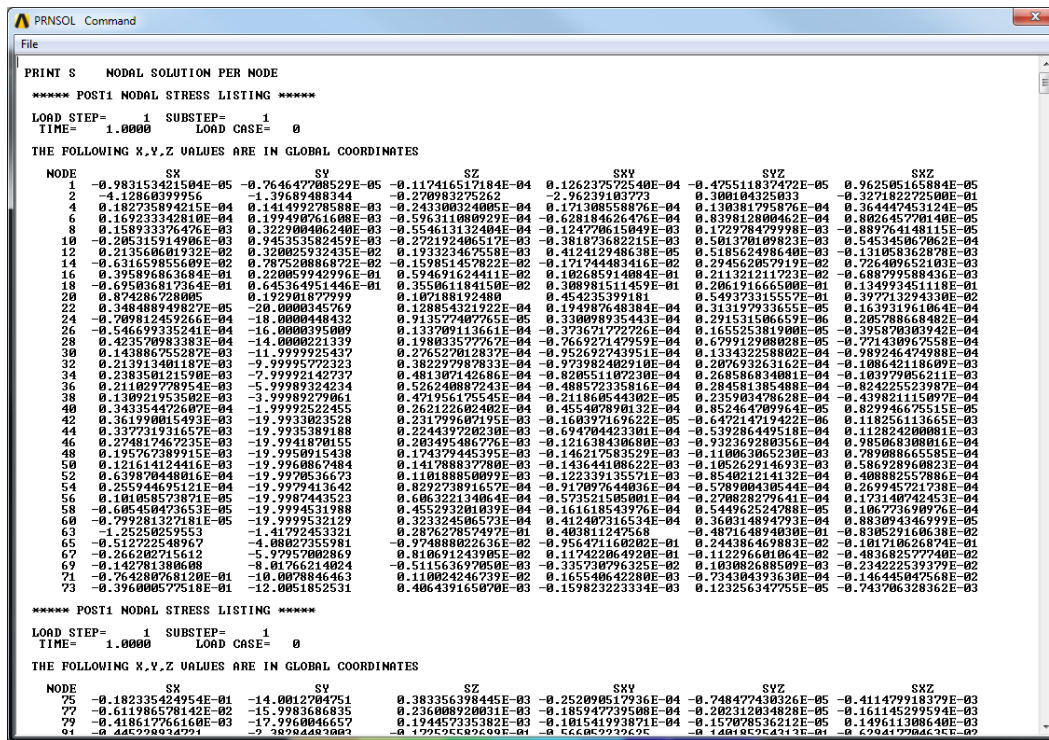


Figure 2.37 Nodal stress component listing.

2.9.1.2 ABAQUS Mesh-Stress

This section describes how to use ABAQUS to create mesh-based stresses. The model is created with a varying far-field stress on the top surface of the cube. The resulting y-direction stress contours are shown in Fig 2.38; the stresses are comparable to the ANSYS stresses (see Fig 2.36).

To generate a .fil file from ABAQUS, add the following lines to the .inp file just before the “*End Step”:

```
** output displacements and stress to fil results file
*File Format, ASCII
*Node File, Frequency=1
U
*El File, position=averaged at nodes, Frequency=1
S
```

Then submit the ABAQUS job for analysis. Note that if you use ABAQUS CAE, you can add these lines using the ABAQUS CAE menu, under **Model** and **Edit Keywords**.

An additional change should be made in ABAQUS CAE to flatten the *.inp* file; remove *Part, *Instance and *Assembly data:

```
>>> mdb.models["Abaqus_Cube"].setValues(noPartsInputFile=ON)
```

This eliminates any confusion with node and element numbering. This should be done prior to running the analysis.

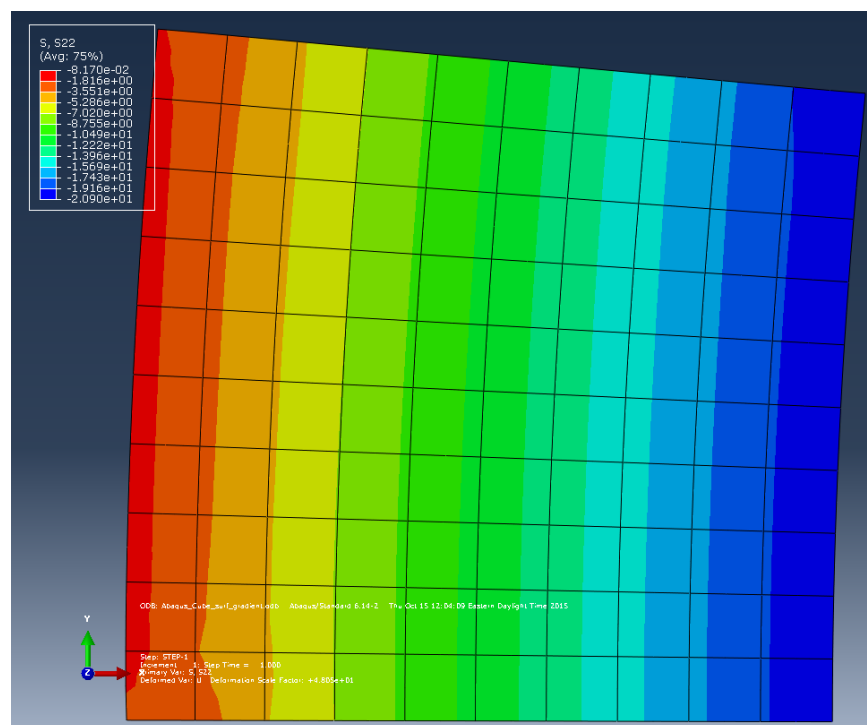


Figure 2.38 ABAQUS stress in y-direction.

Instead of using the *.fil* file, you can write the stress data to a *.rpt* file from ABAQUS CAE. Using the **Report** menu in CAE, choose **Field Output**. The dialog in Fig 2.39 is displayed; select Stress components and click **OK**. This assumes that you requested stress as a field output, which you do before submitting the analysis. We request unique nodal stress values in the report. By default, ABAQUS CAE saves the data to *abacus.rpt*, you can change this using the Setup tab or just remember to rename the *.rpt*; we use *Abaqus_Cube_surf_gradient.rpt* (this name is used for the *.inp* and *.fil* files also).

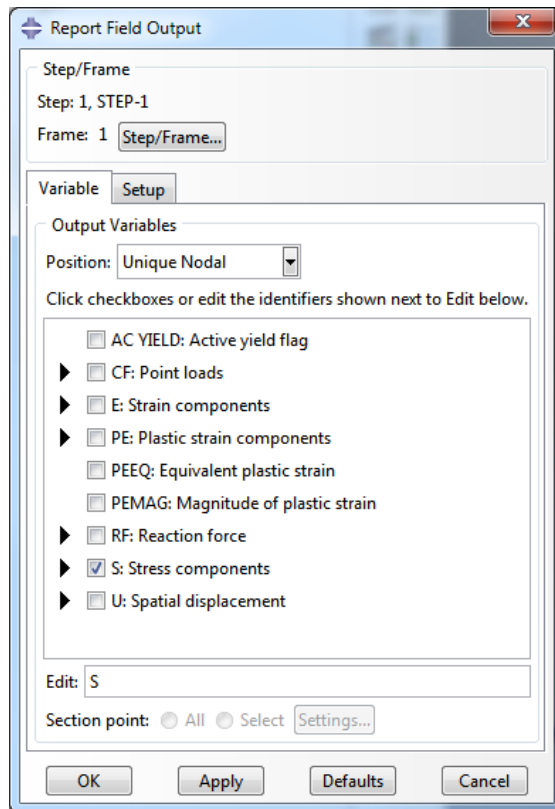


Figure 2.39 ABAQUS report field output dialog.

2.9.1.3 NASTRAN Mesh-Stress

If you use NASTRAN to create stresses, FRANC3D reads *.bdf* (or *.nas*) files along with *.pch* files with the nodal stress data.

2.9.2 Apply Mesh-Based Stress as Crack Face Traction

Start with the FRANC3D GUI (see Fig 2.1) and select **File** → **Import**. In the dialog (see Fig 2.2), choose Complete Model. Switch the Mesh File Type radio button (see Fig 2.3) to the appropriate FE input. Select the appropriate “*cube*” file, and then select **Next**. As before, retain all the boundary condition surfaces (see Figs 2.4 – 2.6). The model will appear as in Fig 2.7.

Insert the same crack from file as before (see Figs 2.8 – 2.12). At this stage, we could do a static analysis to verify that the model is correct and compute the SIFs for the far-field loading only; the Mode I SIF should be the same as was shown in the top left panel of Fig 2.19. We leave this as an exercise for the reader. The next step is to define the mesh-based crack surface traction.

From the FRANC3D main menu, select **Loads** → **Crack Face Pressure/Traction**, Fig 2.40. Click on **Add** in the dialog box shown in the left panel of Fig 2.41. Choose Residual Stress Defined on a Mesh, as shown in the right panel of Fig 2.41, and select **Next**. The dialog shown on the left side of Fig 2.42 is displayed. We select the files for the mesh and the stress listing. There is only one load step in the stress listing. Select **Finish**; the original Add Traction dialog is displayed with one entry, Fig 2.42 – right side; select **Accept**.

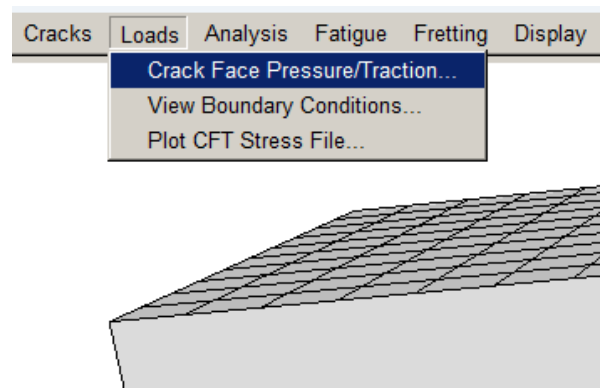


Figure 2.40 Loads – Crack Face Pressure/Traction menu item selected.

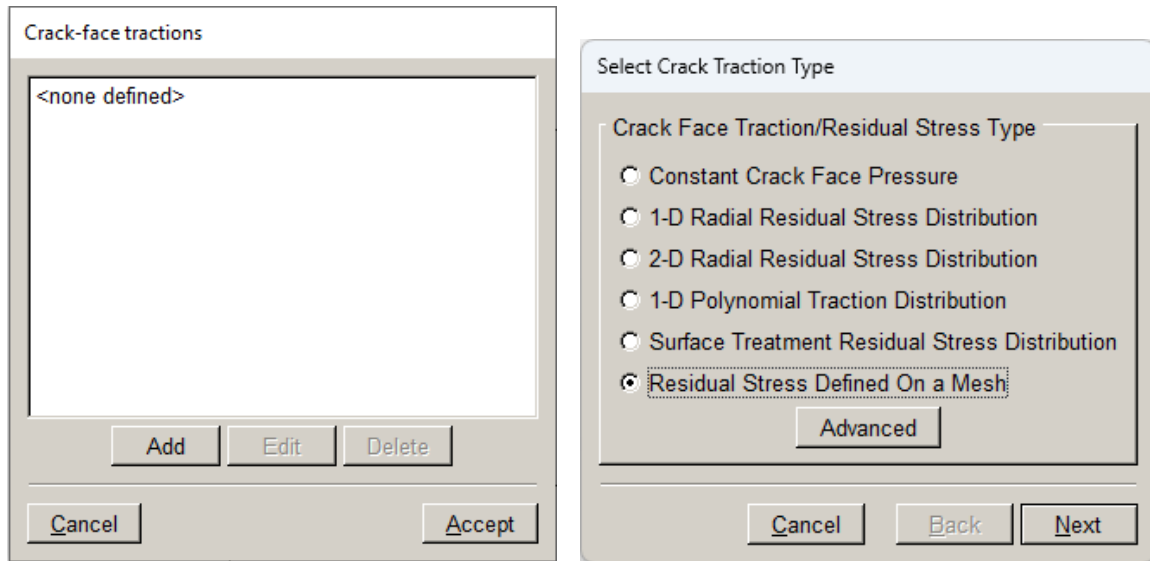


Figure 2.41 Crack Face Traction wizard panels; Residual Stress Defined on a Mesh selected.

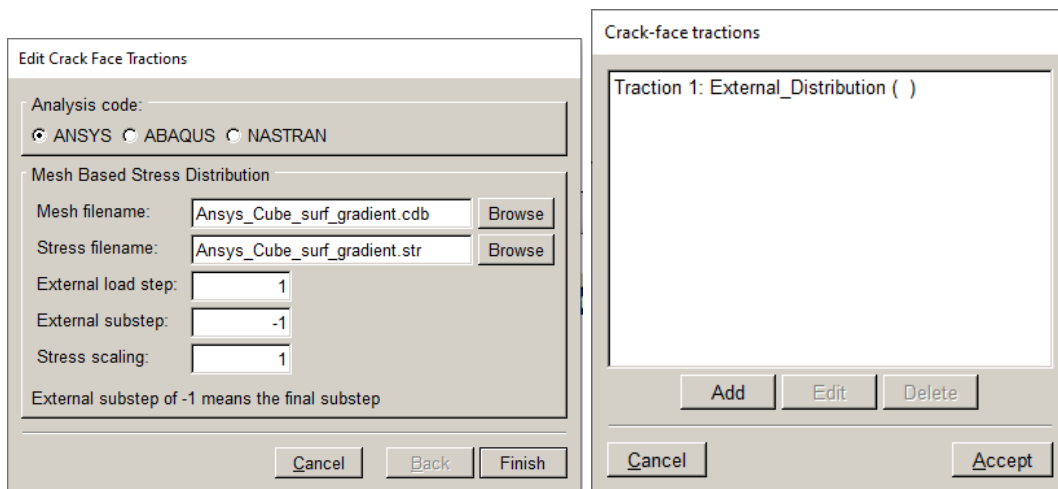


Figure 2.42 Crack Face Traction wizard panels for residual stress defined on a mesh.

Note that we chose to apply ANSYS mesh-based stress; the ANSYS stress can be applied to the ABAQUS, ANSYS or NASTRAN crack model.

From the FRANC3D menu, select **Analysis** → **Static Crack Analysis**. Select **Next** after entering the file name. The next panel allows you to set the analysis options (see Fig 2.16). Click **Finish** when ready to proceed with the analysis.

Once the analysis is finished, from the FRANC3D menu, select **Cracks** → **Compute SIFs**. When the Compute SIFs dialog is displayed, select **Finish** and the SIFs Plot dialog is displayed, Fig 2.43. The initial plot shows the sum of K_I for both load cases. The lower two panels of Fig 2.43 show the K_I values for the first and second load cases separately. The SIF for the first load is the same as previously computed. The second load produces a negative SIF with a gradient due to the mesh-based CFTs.

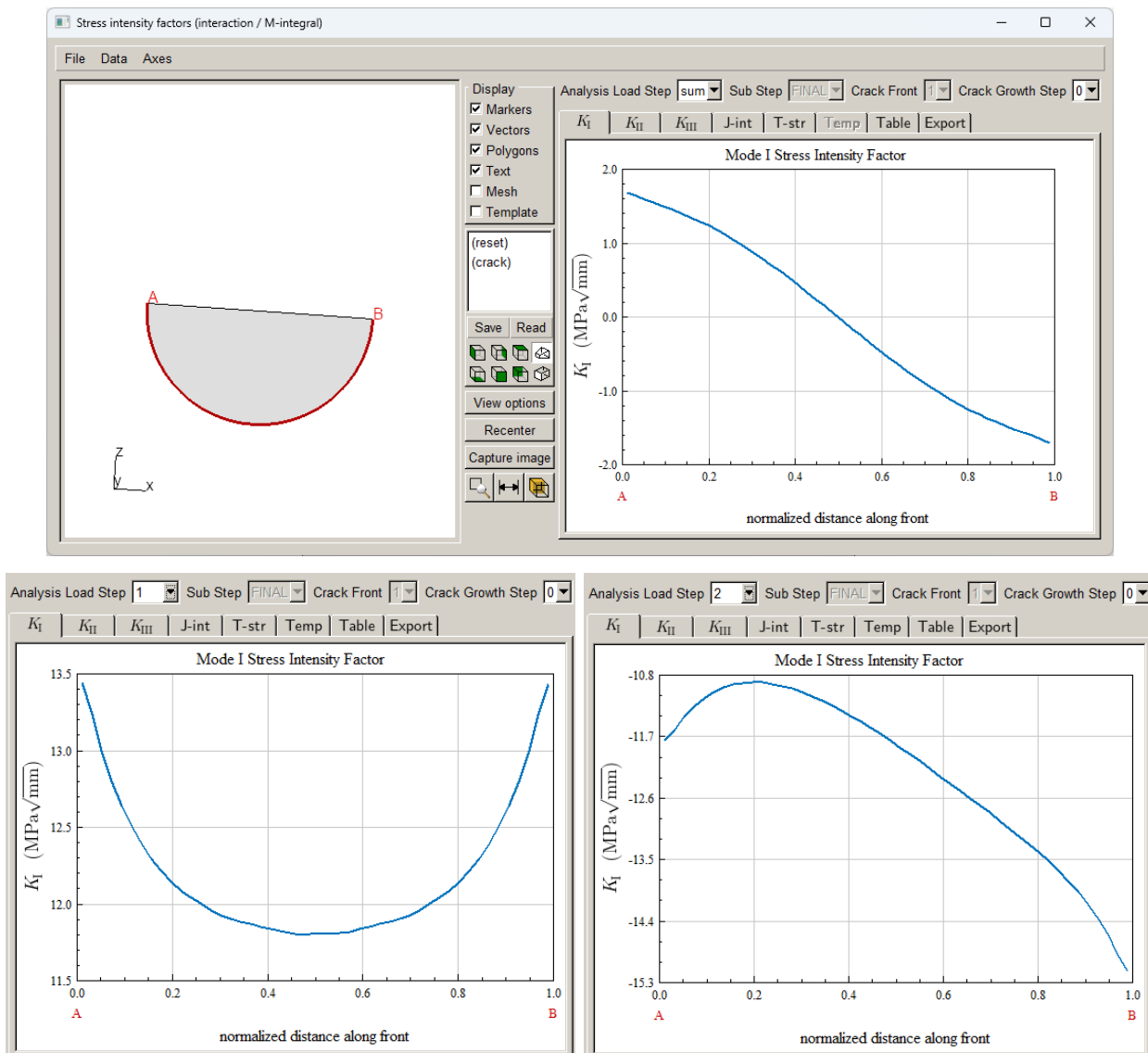


Figure 2.43 Mode I SIF plots showing the sum (top) and the individual load step values; the mesh-based stress produces a negative SIF (bottom right).

2.10 Step 10: Apply Tractions and Temperature from Mesh

The mesh-based stress CFT option allows one to include temperature. We demonstrate using the simple cube model with only one far-field tension load step (same model as tutorial #1). The only change to the base model is the addition of temperature-dependent material properties; these will be described later. The stress and temperature for the CFT load steps will be defined next using the ABAQUS cube model.

2.10.1 Uncracked ABAQUS Cube Stress and Temperature

We start with the ABAQUS model from Section 2.9.1.2. The model did not include temperature or temperature dependent material properties. So, we modify the model to include a reference temperature of 0 degrees, using:

```
*Initial Conditions, type=TEMPERATURE  
all_nodes, 0.0
```

Load step #1 has a surface pressure with a gradient in the x-direction, and we add a simple temperature boundary condition to this load step, using:

```
*TEMPERATURE, OP=NEW  
all_nodes, 1.000000000e+03
```

Next, we add a second load step, which has uniform unit tension on the top surface and a temperature of:

```
*TEMPERATURE, OP=NEW  
all_nodes, 1.000000000e+02
```

The material properties are modified to be:

```
*Material, name=MATERIAL-1  
*ELASTIC, TYPE=ISOTROPIC  
1.00000000e+04, 3.00000000e-01, 0.00000000e+00  
9.00000000e+03, 2.90000000e-01, 1.00000000e+03  
*EXPANSION  
3.00000000e-07, 0.00000000e+00  
3.00000000e-07, 1.00000000e+03
```

The ABAQUS analysis of this model produces stress, temperature and displacement results shown in Figs 2.44-2.49. These results can be exported from ABAQUS CAE to two *.rpt* files or saved to a single *.dtp* file. The model itself is written as an *.inp* file. The results files are described next.

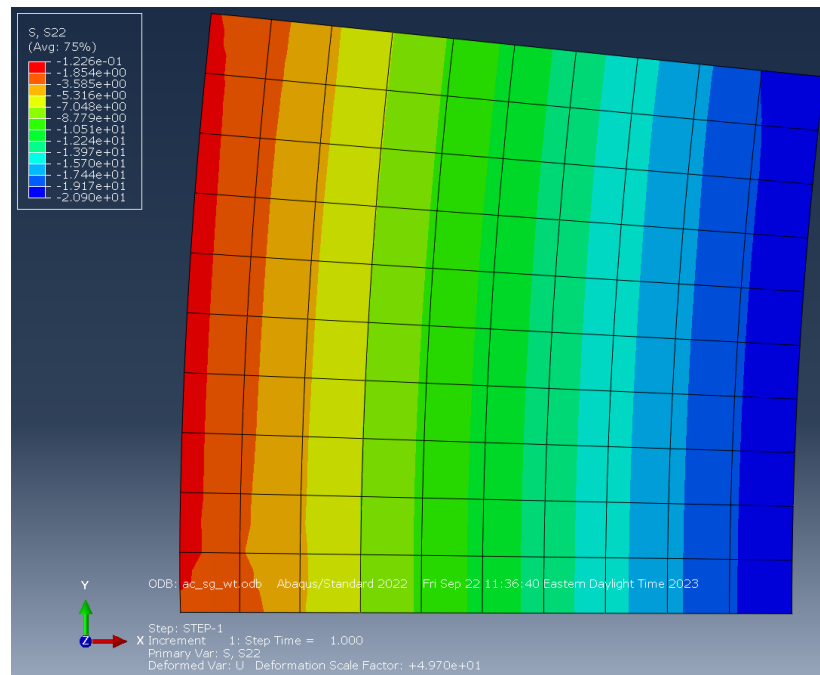


Figure 2.44 S22 (y) stress component for load step #1.

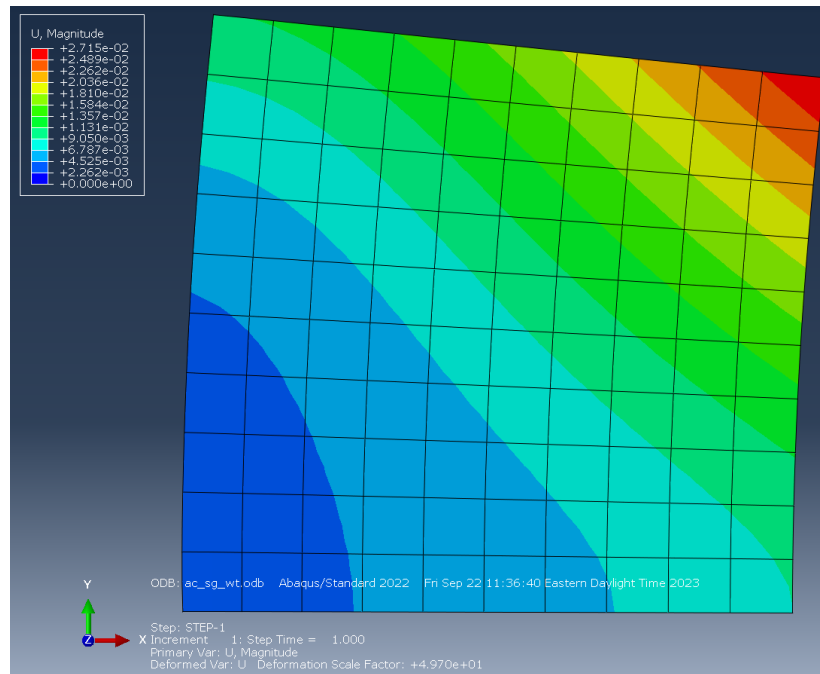


Figure 2.45 Displacement magnitude for load step #1.

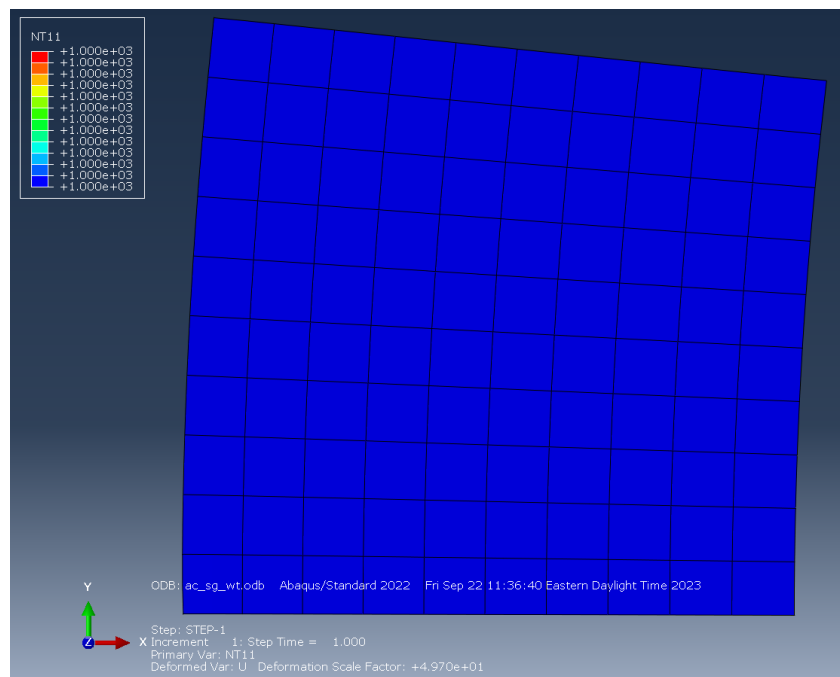


Figure 2.46 Temperature for load step #1.

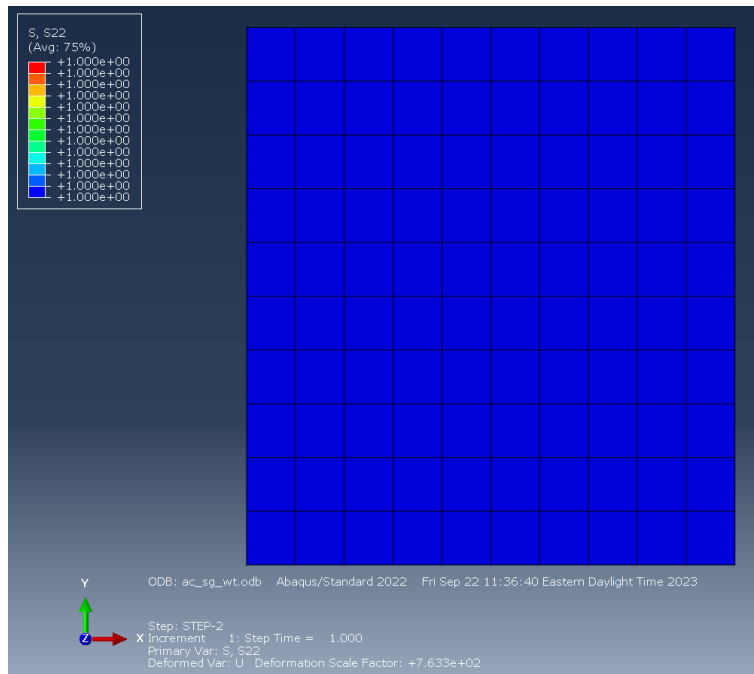


Figure 2.47 S22 (y) stress component for load step #2.

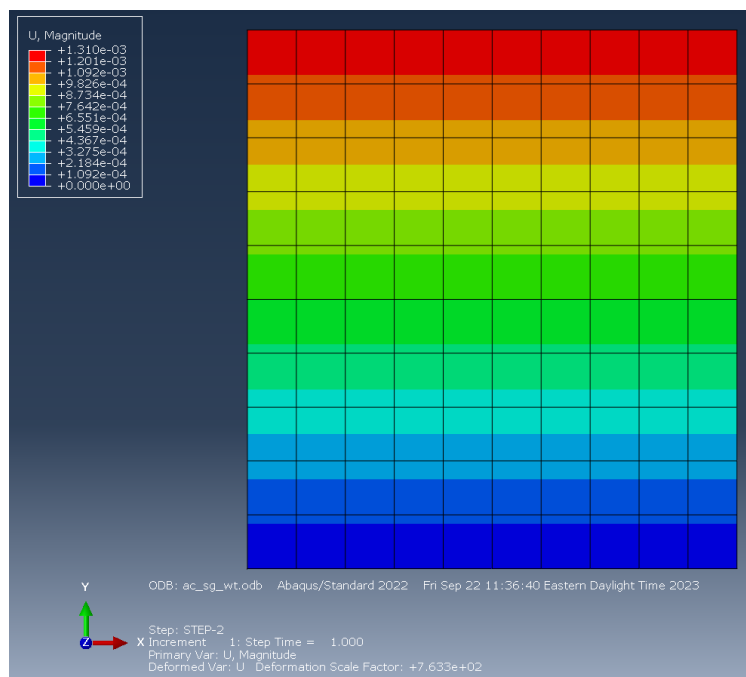


Figure 2.48 Displacement magnitude for load step #2.

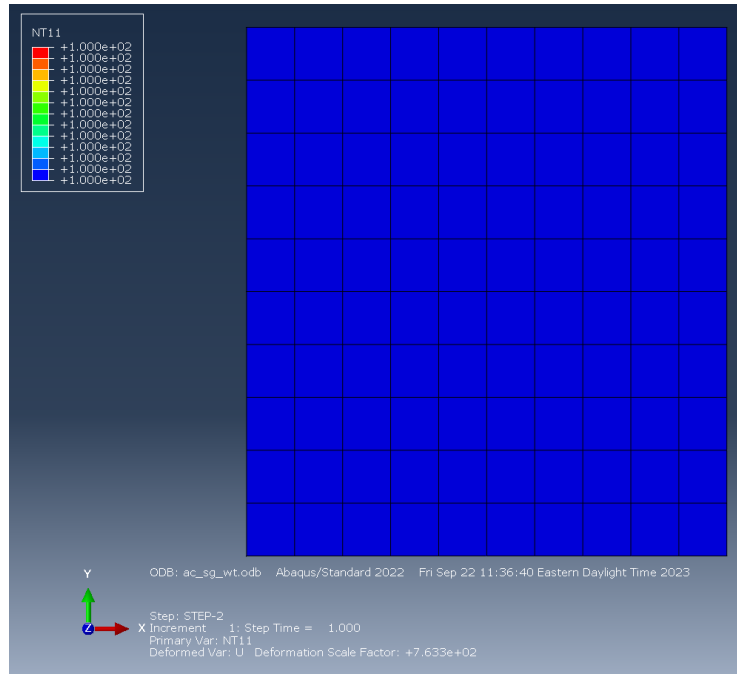


Figure 2.49 Temperature for load step #2.

2.10.1.1 ABAQUS .rpt Files

ABAQUS CAE can be used to write the results to a *.rpt* file for frame #1 of each load step, Fig 2.50. Note that we write all displacement and stress components to the file. FRANC3CD will read the *.rpt* file; it expects all components of stress and displacement. You can use the *ac_sg_wt_ls1.rpt* in the Tutorial #2 archive as an example.

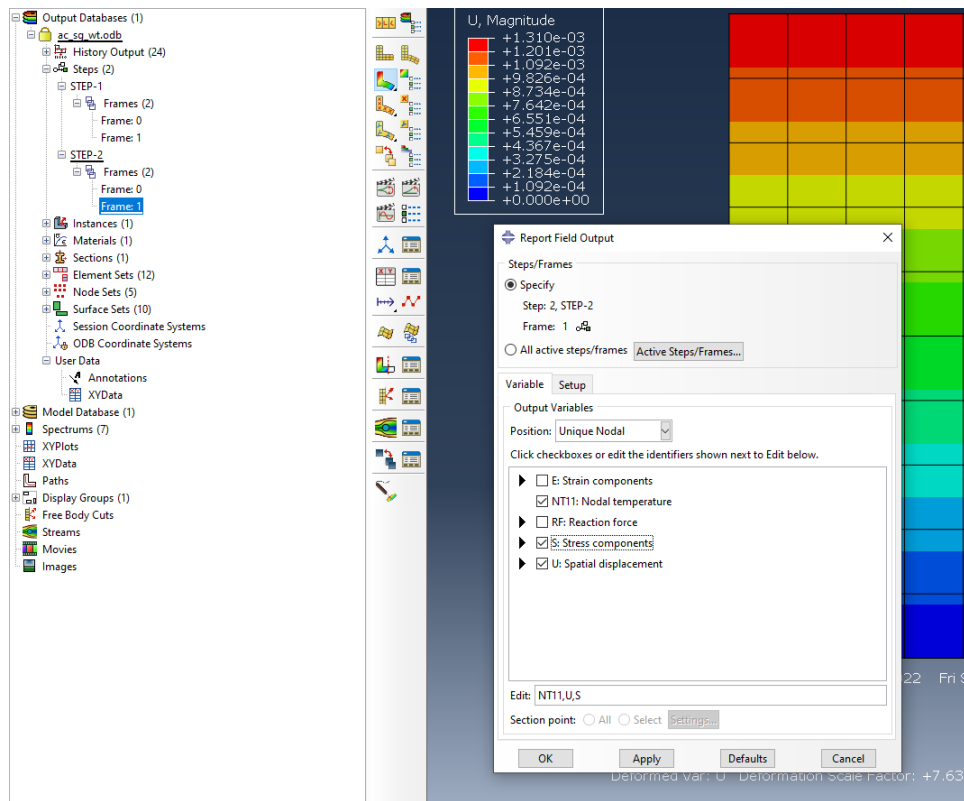


Figure 2.50 ABAQUS .rpt file written with temperature, displacement and stress.

2.10.1.2 .dtp File

The results can also be extracted from the ABAQUS .odb file using a Python script to create a single .dtp file. A Python script (*extract_stress_to_dtp.py*) is included with the Tutorial #2 archive. The corresponding .dtp file (*ac_sg_wt.dtp*) is also included in the archive. The .dtp file contains results for temperature, displacement, and stress for both load steps.

2.10.2 Apply Mesh-Based Temperature and Stress as Crack Face Traction

For this example, we are using the tutorial #1 cube model, but it is modified to include a reference temperature of 0.0 degrees and temperature-dependent material properties (as given above).

Start with the FRANC3D GUI (see Fig 2.1) and select **File** → **Import**. In the dialog (see Fig 2.2), choose Complete Model. Switch the Mesh File Type radio button in the Select Import Mesh File window (see Fig 2.3) to the appropriate FE input type. Select the appropriate “*cube*” file, and then select **Next**. As before, retain all the boundary condition surfaces (see Figs 2.4 – 2.6). The model will appear as was shown in Fig 2.7. Remember that the base model has uniform tension with a magnitude of 10 on the top surface.

Insert the same crack from file as before (see Figs 2.8 – 2.12). At this stage, we could do a static analysis to verify that the model is correct and compute the SIFs for the far-field loading only; the Mode I SIF should be the same as was shown in the top left panel of Fig 2.19. We leave this as an exercise for the reader. The next step is to define the mesh-based-CFT and temperature.

From the FRANC3D main menu, select **Loads** → **Crack Face Pressure/Traction**, Fig 2.51. Click on **Add** in the dialog box shown in the left panel of Fig 2.52. Choose Residual Stress Defined on a Mesh, as shown in the right panel of Fig 2.52, and select **Next**. The dialog shown on the left side of Fig 2.54 is displayed. Select the files for the mesh and for the stress. Fig 2.53 shows the *.dtp* file is selected, and load step #1 in the *.dtp* file is specified. If you use the *.rpt* file, there will only be results for one load step in each file. Select **Next** to display the dialog on the right side of Fig 2.53. In this dialog, choose Same file as external stresses, turn off Allow thermal expansion, and then select **Finish**.

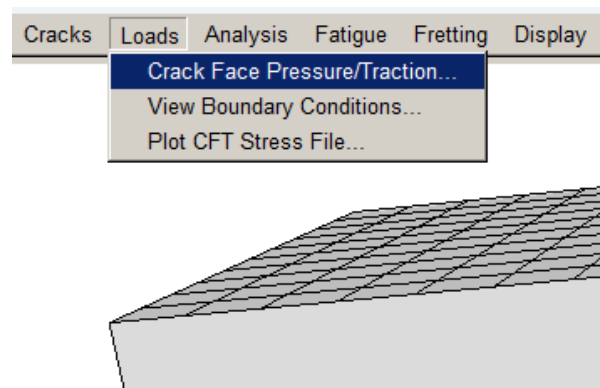


Figure 2.51 Loads – Crack Face Pressure/Traction menu item selected.

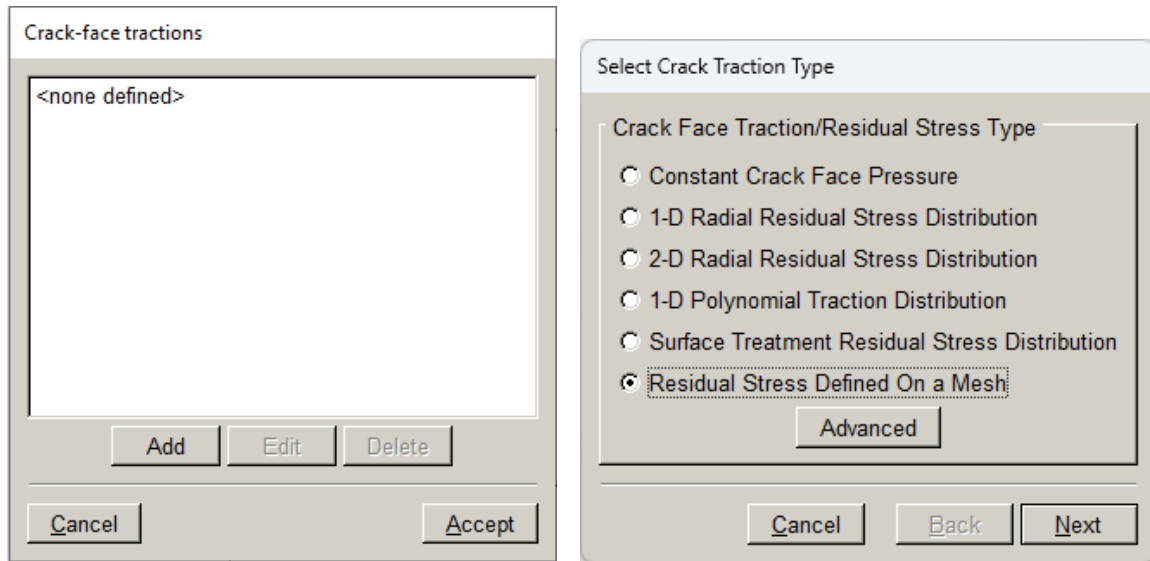


Figure 2.52 Crack Face Traction wizard panels; residual stress defined on a mesh selected.

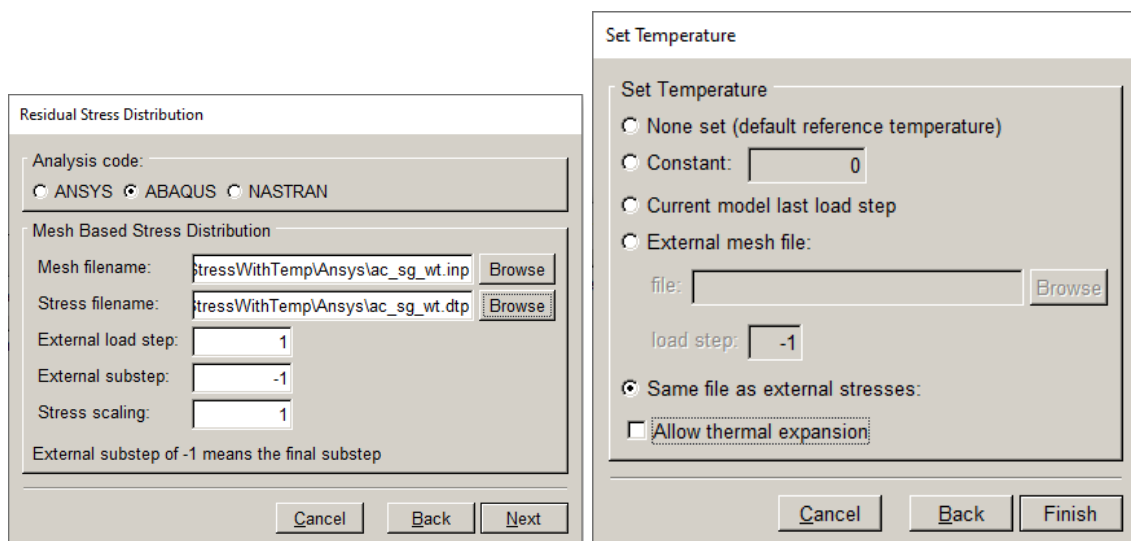


Figure 2.53 Crack Face Traction wizard panels for residual stress defined on a mesh.

The original CFT dialog is displayed with one entry, Fig 2.54. Click on the **Add** button again to add the second CFT load step. Select the Residual Stress Defined on a Mesh again and select the same ABAQUS *.inp* file and the same *.dtp* file. This time, however, set the external load step as 2 (right panel of Fig 2.54). The temperature settings for the second CFT load step are the same as the first. The final CFT dialog will show two entries when you are done, Fig 2.55.

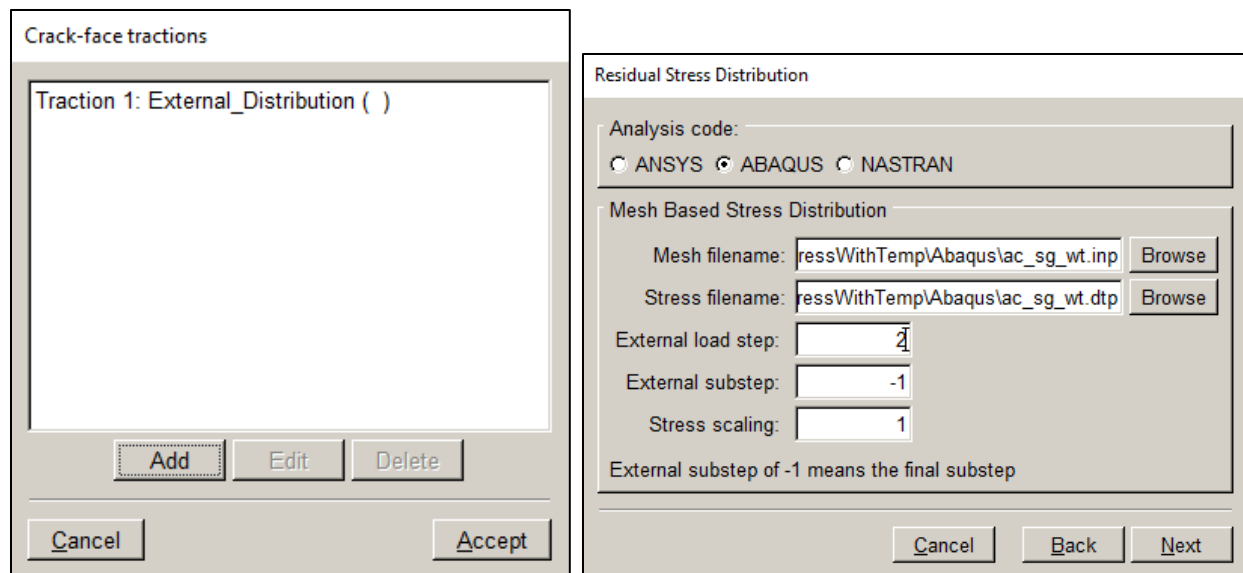


Figure 2.54 Crack Face Traction wizard panels for adding the second CFT.

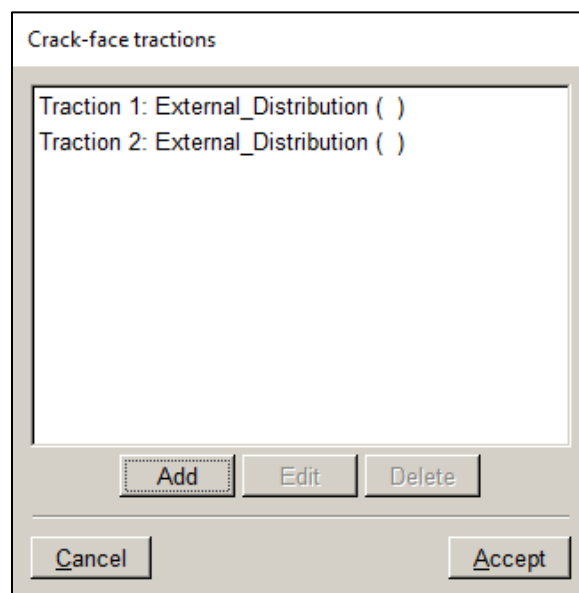


Figure 2.55 Crack Face Traction dialog with two entries.

Note that we chose to apply ABAQUS mesh-based stress and temperature; the ABAQUS stress can be applied to the ABAQUS, ANSYS or NASTRAN crack model.

From the FRANC3D menu, select **Analysis** → **Static Crack Analysis**. Select **Next** after entering the file name. The next panel allows you to specify the analysis options (see Fig 2.16). Click **Finish** when ready to proceed with the analysis.

Once the analysis is finished, from the FRANC3D menu, select **Cracks** → **Compute SIFs**. The initial plot shows the sum of K_I for the three load cases, Fig 2.56.

You can also verify that the ABAQUS results have the correct temperature for each load step using ABAQUS CAE. CFTs with temperature settings are part of Tutorial #4 also and are discussed in Section 12 of the User's Guide.

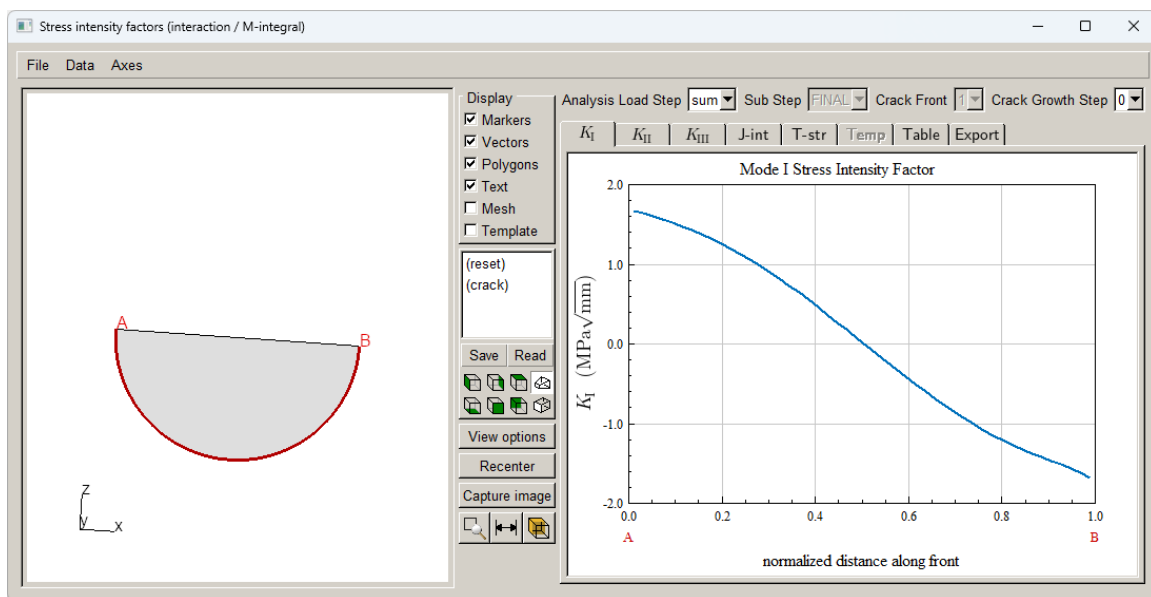


Figure 2.56 Mode I SIF plots showing the sum of the SIFs for the three load steps.

3.0 Tutorial 3: Two Cubes Glued Together

In Tutorial 3, we use two cubes that are tied together to illustrate how to simulate crack growth in models where there is contact or constraint between parts of the model.

3.1 Step 1: Create the FE Model

We use the same initial cube geometry and boundary conditions from tutorial #1 and add a second adjacent cube, Fig 3.1, with slightly different material properties. The second cube has an elastic modulus of 20,000, which is double that of the first cube; the Poisson ratio is the same. The second cube is constrained in the y-direction and has a surface traction (negative pressure) of 10 on the upper surface, which is the same as the first cube.

3.1.1 ABAQUS Model

For ABAQUS, we define a *Tie constraint between the two cubes. The right-side image of Fig 3.1a shows the constrained surface. The constraint settings are shown in Fig 3.2a. The ABAQUS file is exported as *doublecube_abaqus.inp*.

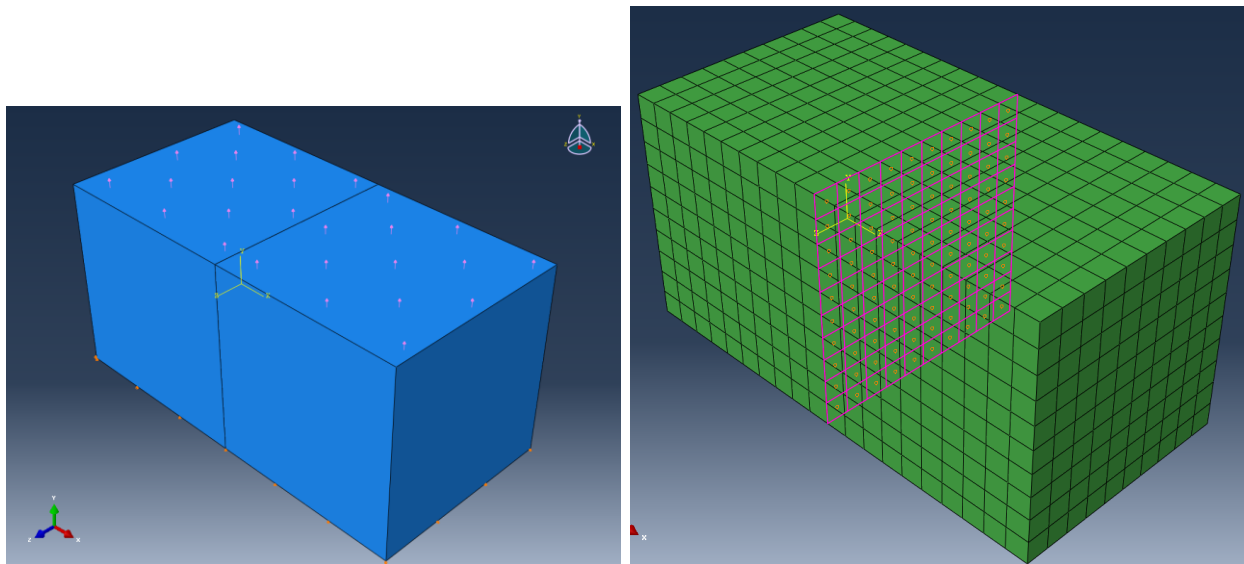


Figure 3.1a ABAQUS double cube model with constraint surface between the cubes.

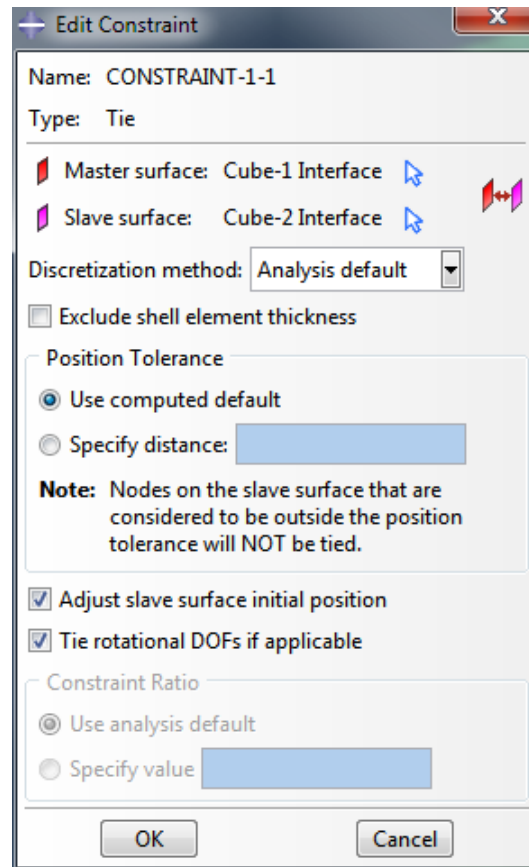


Figure 3.2a ABAQUS tie constraint dialog.

You could also use contact instead of constraint. The surfaces will be the same. The contact can be defined as tied-contact so the surfaces remain glued together.

3.1.2 ANSYS Model

For ANSYS, we define contact between the two cubes. The lower image of Fig 3.1b shows the contact elements. The analysis uses always-bonded contact, Fig 3.2b. The ANSYS file is exported as *doublecube_ansys.cdb*.

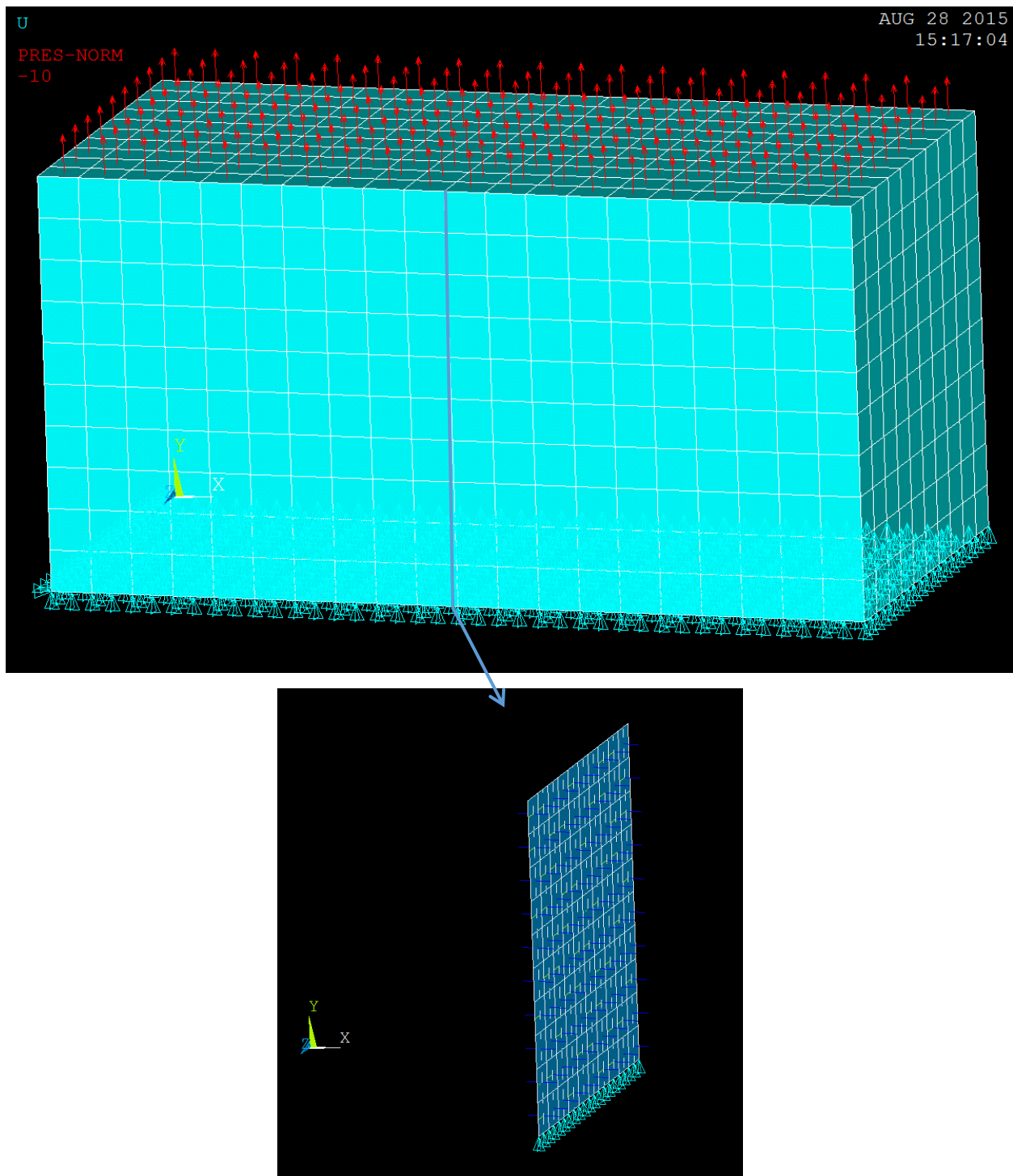


Figure 3.1b ANSYS double cube model with contact surface between the cubes.

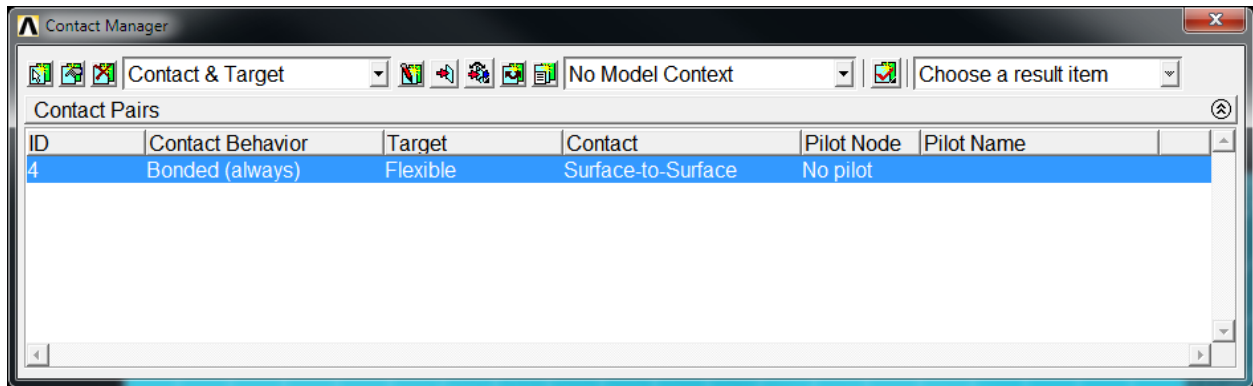


Figure 3.2b ANSYS contact wizard, with always-bonded contact selected.

3.1.3 NASTRAN Model

For NASTRAN, we define contact between the two cubes, Fig 3.1c. The analysis uses bonded contact, Fig 3.2c. The NASTRAN file is saved as *doublecube_nastran.nas*. Note that the NASTRAN model was meshed with tetrahedral elements rather than brick elements.

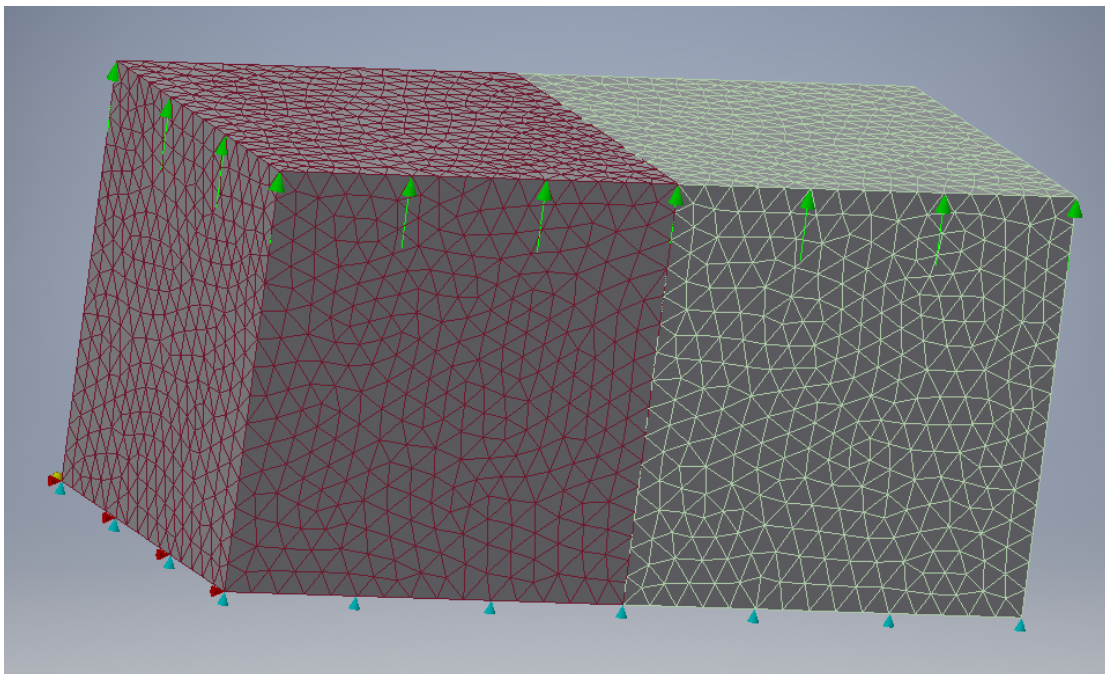


Figure 3.1c NASTRAN double cube model with contact surface between the cubes.

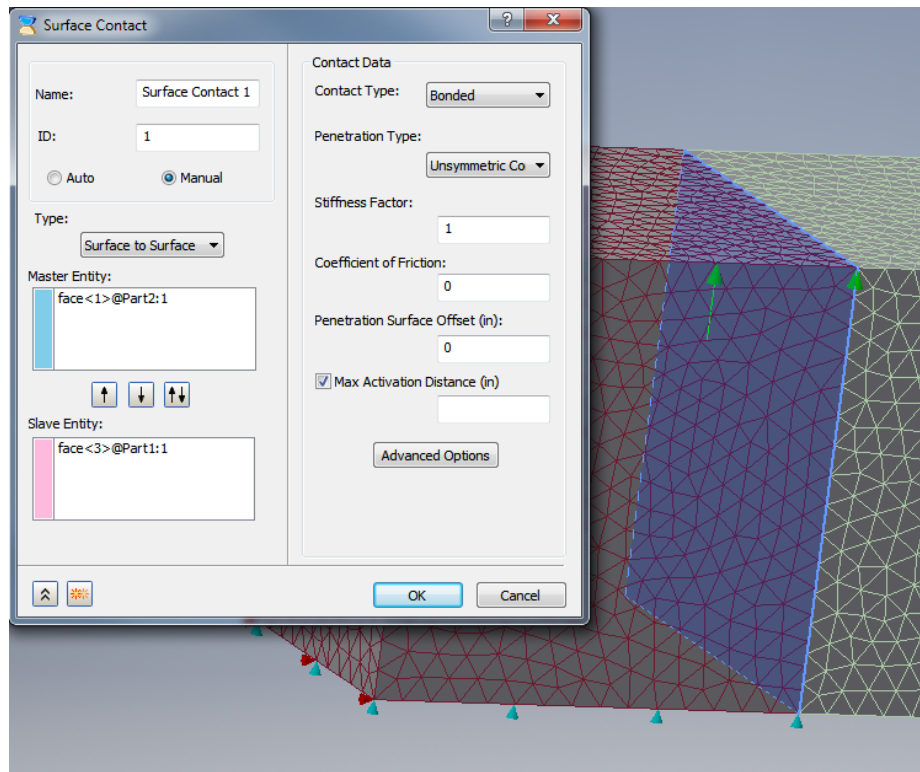


Figure 3.2c NASTRAN contact surface with bonded contact selected.

3.2 Step 2: Import FE Model into FRANC3D

Import the FE model into FRANC3D as a full model; local+global cases are examined later in Section 3.8

Step 2.1: Importing FE Model

Start with the FRANC3D GUI and select **File** → **Import**. In the Select Type of Import panel, choose Complete Model and select **Next**. Switch the Mesh File Type radio button to the appropriate FE type and select the appropriate “*doublecube*” file, Fig 3.3. Select **Next**.

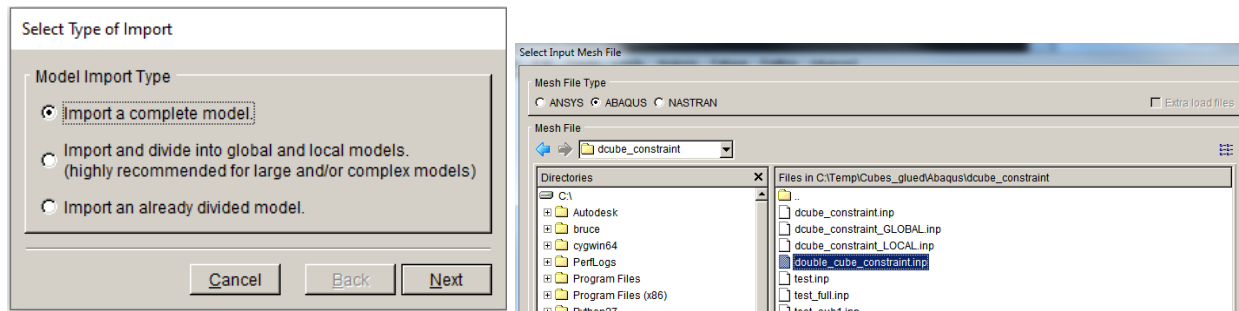


Figure 3.3 FRANC3D FE model import – ABAQUS file selected.

The Select Retained BC Surfaces dialog is displayed, Fig 3.4. The surfaces with boundary conditions and the contact surfaces are highlighted in blue. Click Select all to retain all the surfaces with boundary conditions and then check the constraint/contact surfaces between the cubes. Select **Next** and set the units; use SI units of mm and MPa.

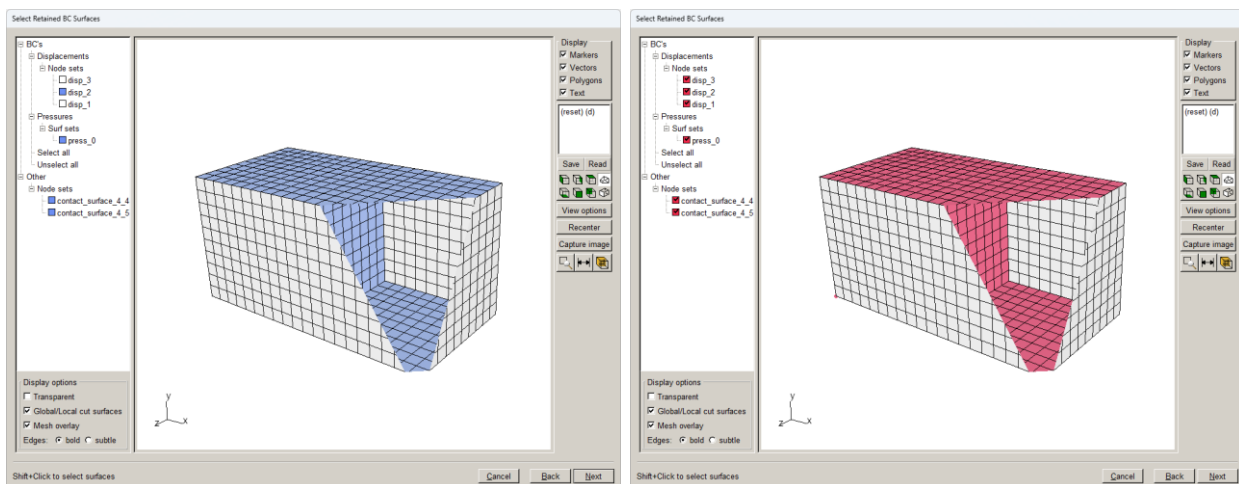


Figure 3.4 FRANC3D select retained surfaces.

3.3 Step 3: Insert a Crack

Insert a surface penny crack with radius of 1.0 in the second cube, Fig 3.5. The template mesh radius is left at the default value of 0.1. The resulting remeshed model is shown in Fig 3.6; the contact surface mesh facets between the two cubes are retained.

Note that the contact surface mesh does not have to be retained, and if the crack is located such that the contact surface must be remeshed, we must not retain it. Here we retain it for simplicity and demonstration. We show what happens if we do not retain it in Section 3.7.

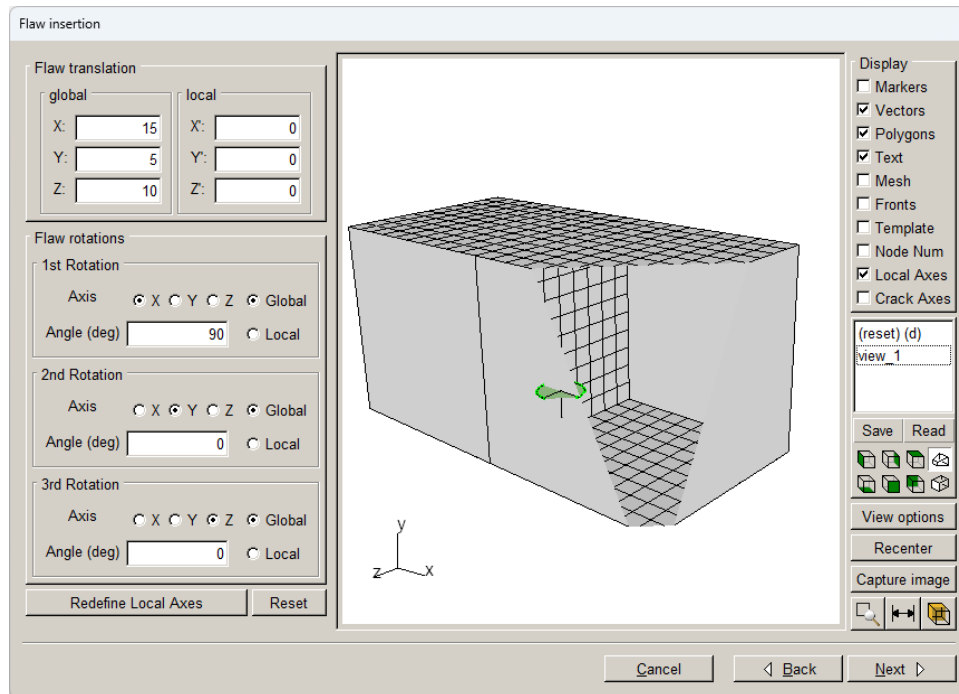


Figure 3.5 FRANC3D new crack insertion panel showing location and orientation.

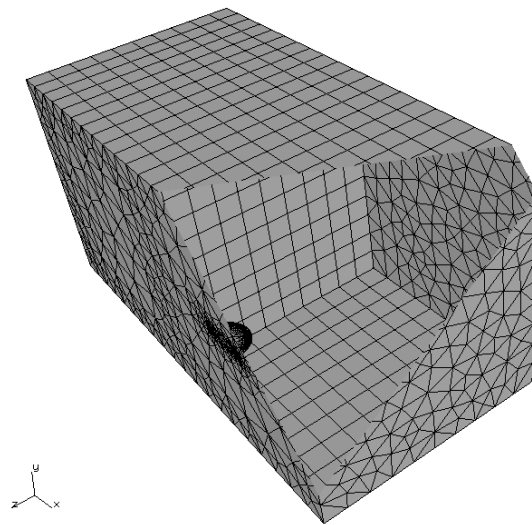


Figure 3.6 Remeshed cracked model, with contact surface mesh facets retained.

3.4 Step 4: Static Crack Analysis

Run a static analysis. From the FRANC3D **Analysis** menu, select **Static Crack Analysis**. Set the file name, we use *constraint_cubes_crack.fdb*. All analysis code default settings are used.

3.4.1 ABAQUS

The *.inp* file written by FRANC3D includes ABAQUS commands to regenerate the tie constraint between the two cubes.

3.4.2 ANSYS

The *.cdb* file written by FRANC3D includes ANSYS commands to regenerate the always-bonded contact between the two cubes.

3.4.3 NASTRAN

The *.bdf* file written by FRANC3D includes NASTRAN data to regenerate the glued contact between the two cubes.

3.5 Step 5: Compute SIFs

The resulting Mode I SIF plot is shown in Fig 3.7; the curve is asymmetric, and the values are slightly higher than for the base model.

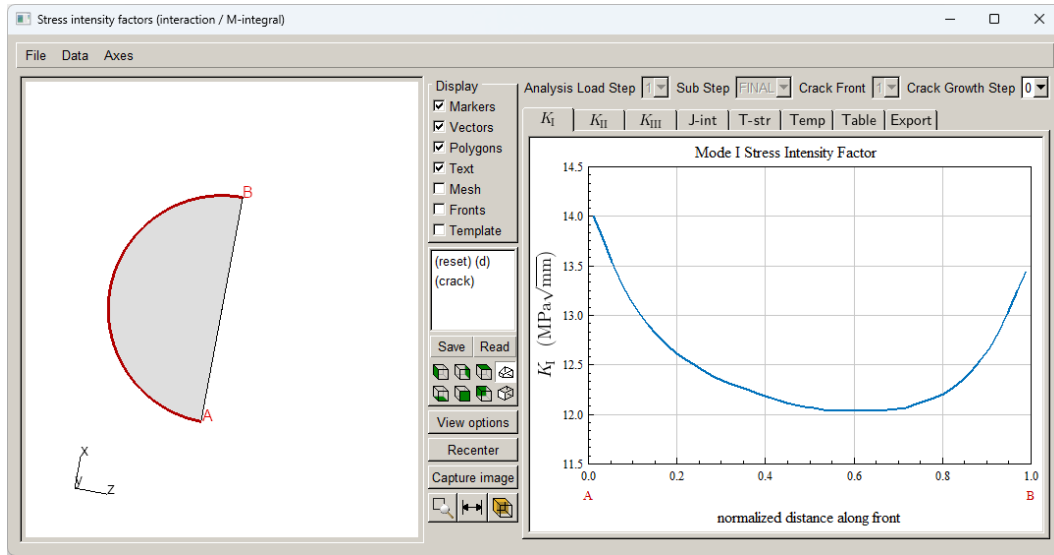


Figure 3.7 Mode I SIFs for double_cube with contact/constraint between cubes.

3.6 Plot Deformed Shape

We can plot the deformed shape in FRANC3D. From the **Display** menu select **View Response**, Fig 3.8. Fig 3.9 shows that the displacement is not uniform; the first cube deforms about twice as much as the second, which is expected as it has a lower elastic modulus. The tie constraint (or bonded contact) between the two cubes is maintained.

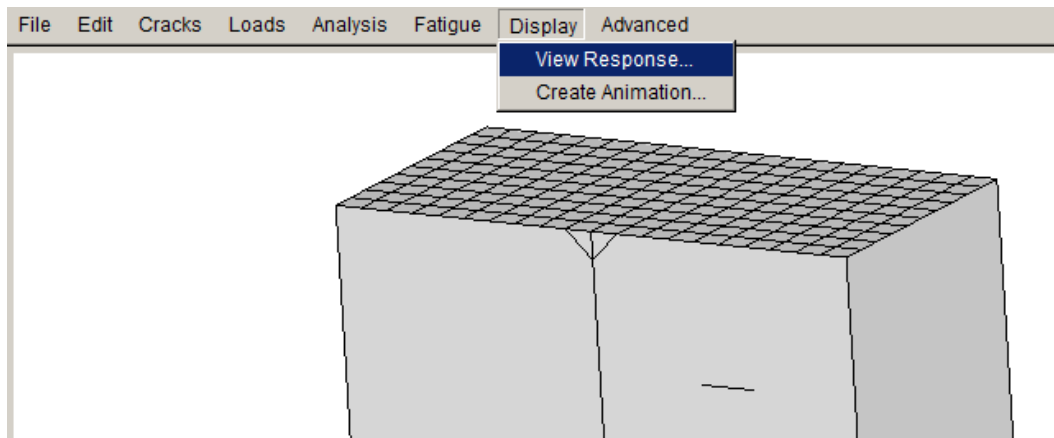


Figure 3.8 FRANC3D View Response menu option.

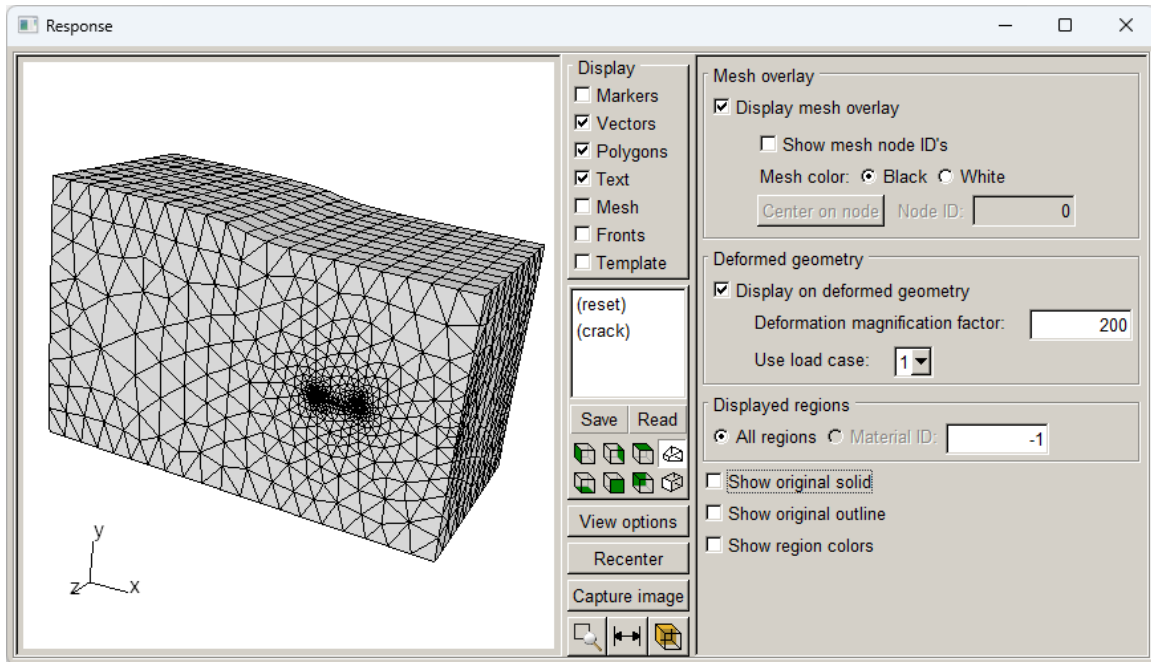


Figure 3.9 Deformed shape – 200x magnification.

3.7 Contact surface mesh not Retained

If we choose not to retain the mesh on the constraint/contact surface when importing the FE model into FRANC3D, do not select the contact surfaces, Fig 3.10; note that both sides of the interface are unselected. Note that you can use the **Clipping** tool (see Section 2 of the Reference document) to see inside the model or use saved camera positions.

Insert the same crack, as in Step 3 (Section 3.3), into the second cube and run the static crack analysis. The resulting surface mesh, Fig 3.11, has a remeshed tie/contact surface. The resulting Mode I SIFs, Fig 3.12, are practically the same as computed previously (see Fig 3.7).

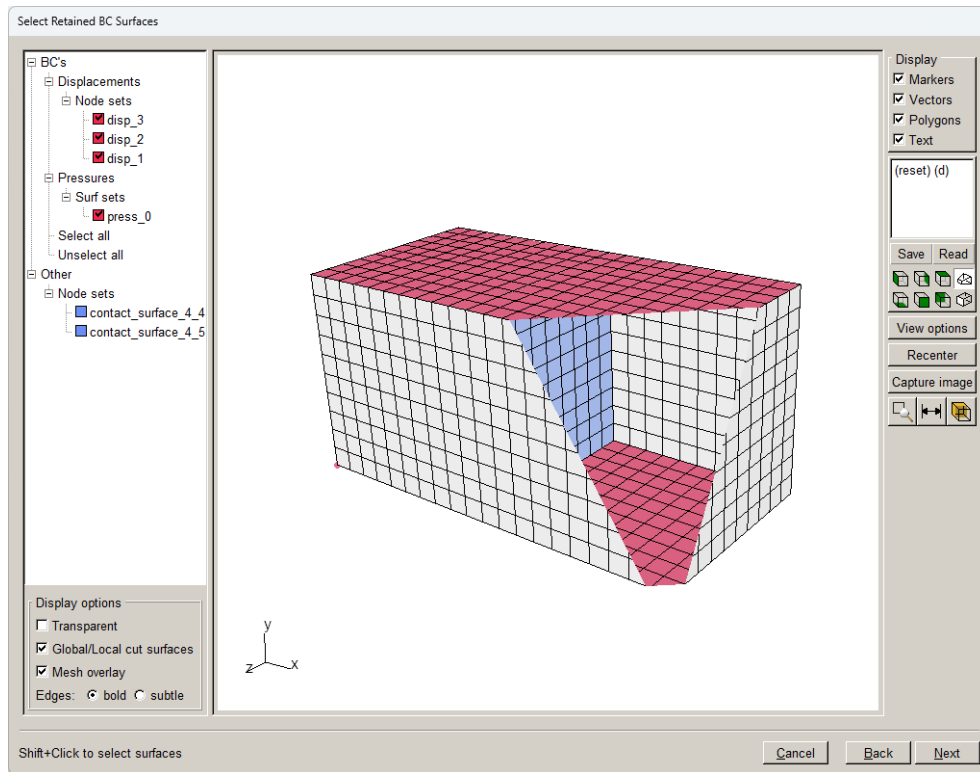


Figure 3.10 Import FE double_cube model with constraint/contact surface unselected.

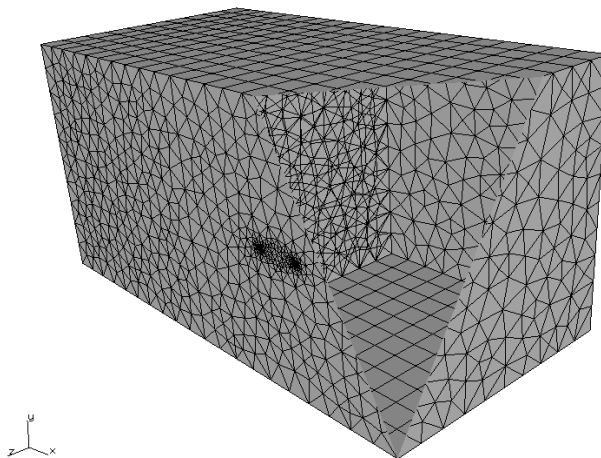


Figure 3.11 Double_cube model with unretained constraint/contact surface remeshed.

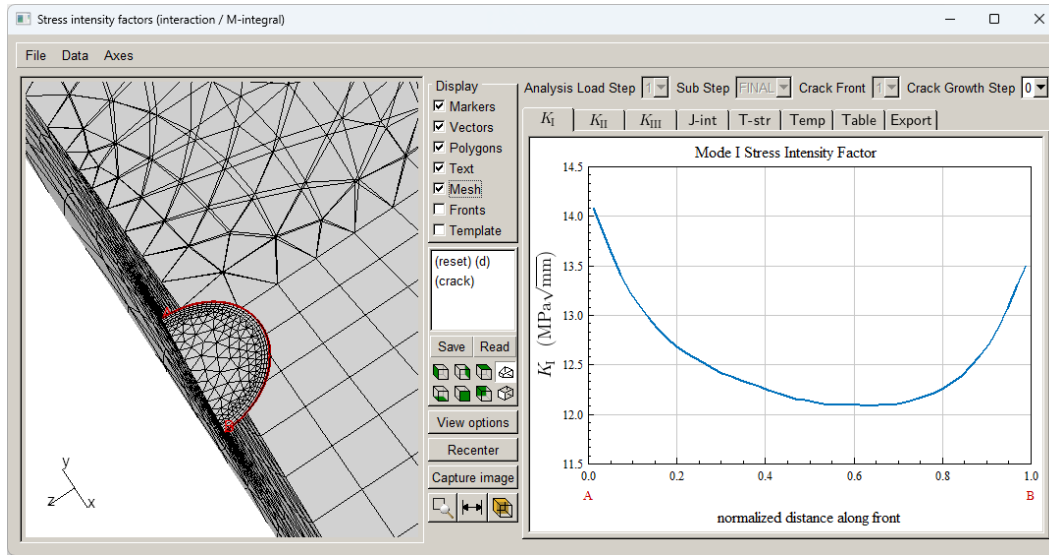


Figure 3.12 Mode I SIFs for double_cube with remeshed contact/constraint between cubes.

3.8 Local / Global Cases

For most models, we want to do a local submodel extraction. When there is contact, there are additional considerations. The simplest choice is to extract a local model that avoids the contact so that all contact surfaces and associated information goes with the global model. We leave this case for the user to try on their own.

Sometimes it is not possible to avoid the contact, and we have three choices: 1) part of one of the contact surfaces is in the local model and the remainder is in the global, or 2) one side of the contact surface is in the local and the other side of the contact is in the global, or 3) one side of the contact surface is in the local plus there is a cut-surface.

We use the ANSYS double-cube model from above to illustrate these three cases; the FRANC3D dialogs and settings are similar for ABAQUS and NASTRAN.

3.8.1 Local model has part of the contact surface

In the first case, the local model includes a portion of one side of the contact. We use the FRANC3D import and divide option when reading the double-cube model. In the submodeler tool, select material #2, Fig 3.13, and **Crop**. Then using the Rubberband Box, select a portion of the material #2 region, Fig 3.14, and **Crop**.

This element selection will include part of one side of the contact surface; this will be clear when selecting the sets/surfaces to retain. Select **Next** after the local portion is cropped.

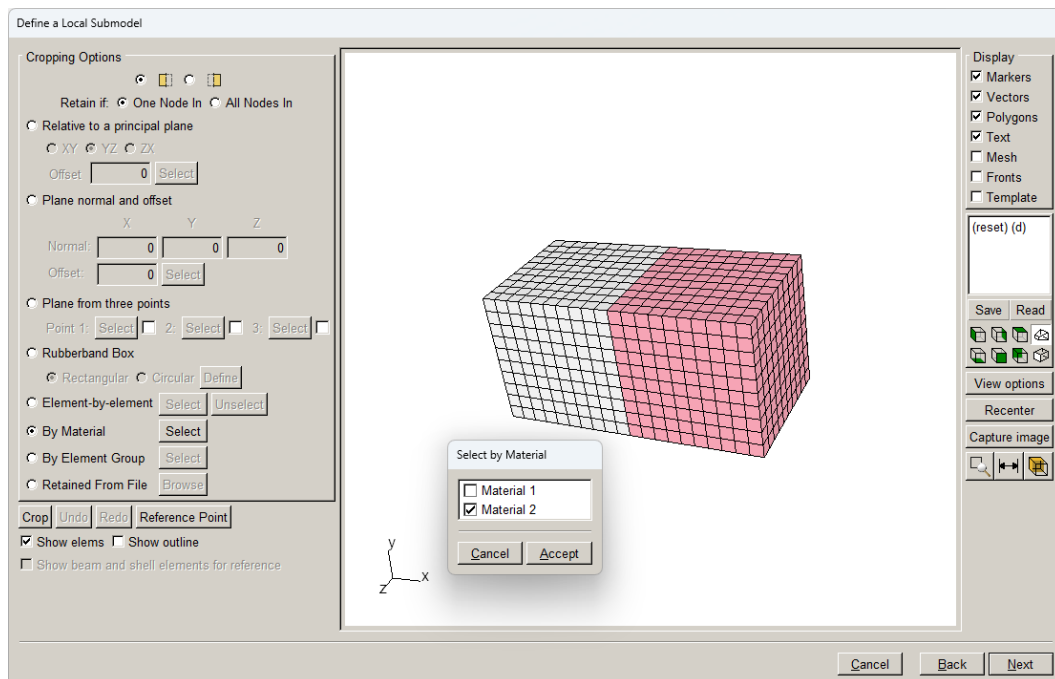


Figure 3.13 Material #2 selected for cropping.

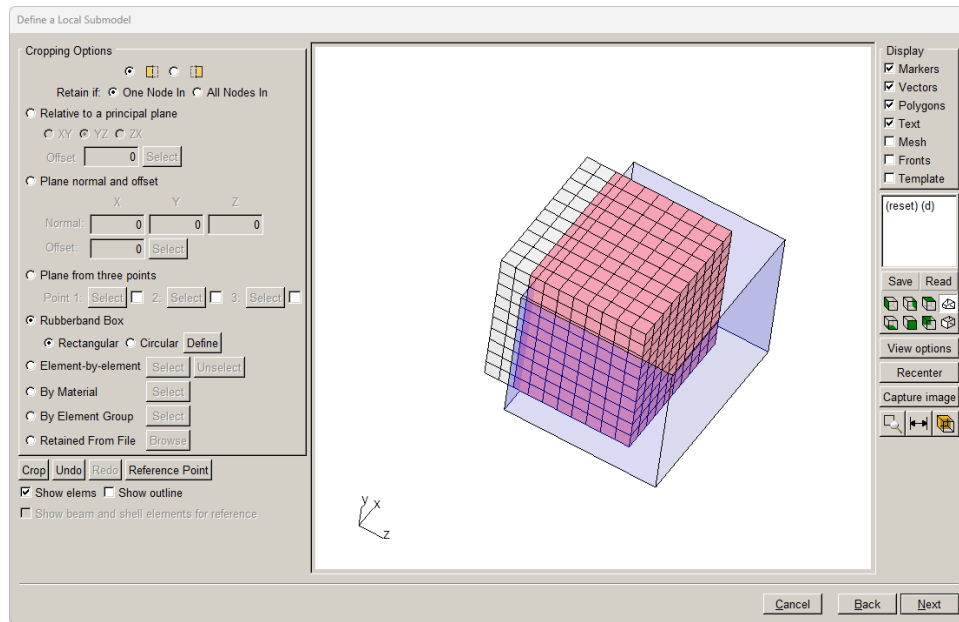


Figure 3.14 Part of material #2 selected for cropping.

Save the local and global files using the default file names and continue to the Select Retained BC Surfaces dialog, Fig 3.15. Depending on the crack location, you might not want to retain the contact surface nodes. For this example, we retain the contact surface, Fig 3.16, the cut-surface, and the surfaces with boundary conditions; the crack is not being inserted (or propagated) into the contact surface. Select **Next** and set the units; the model will be displayed in the main window, Fig 3.17. The facets on the BC surfaces (top and bottom), as well as the facets on the cut-surface and the facets on the contact surface are all preserved.

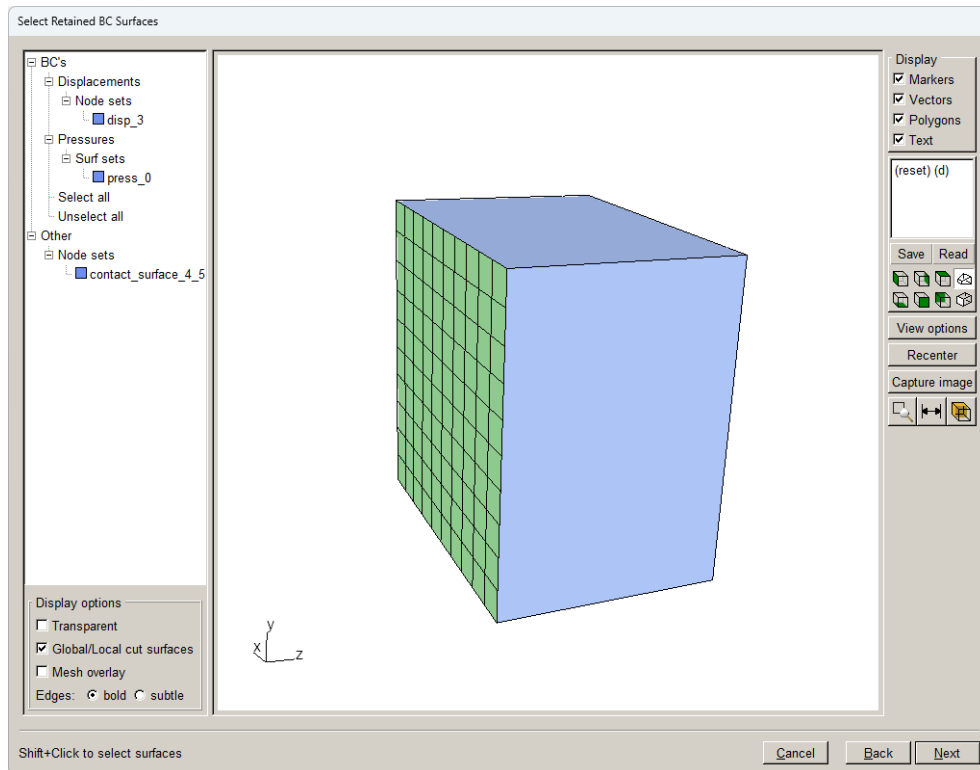


Figure 3.15 Select Retained BC Surfaces dialog.

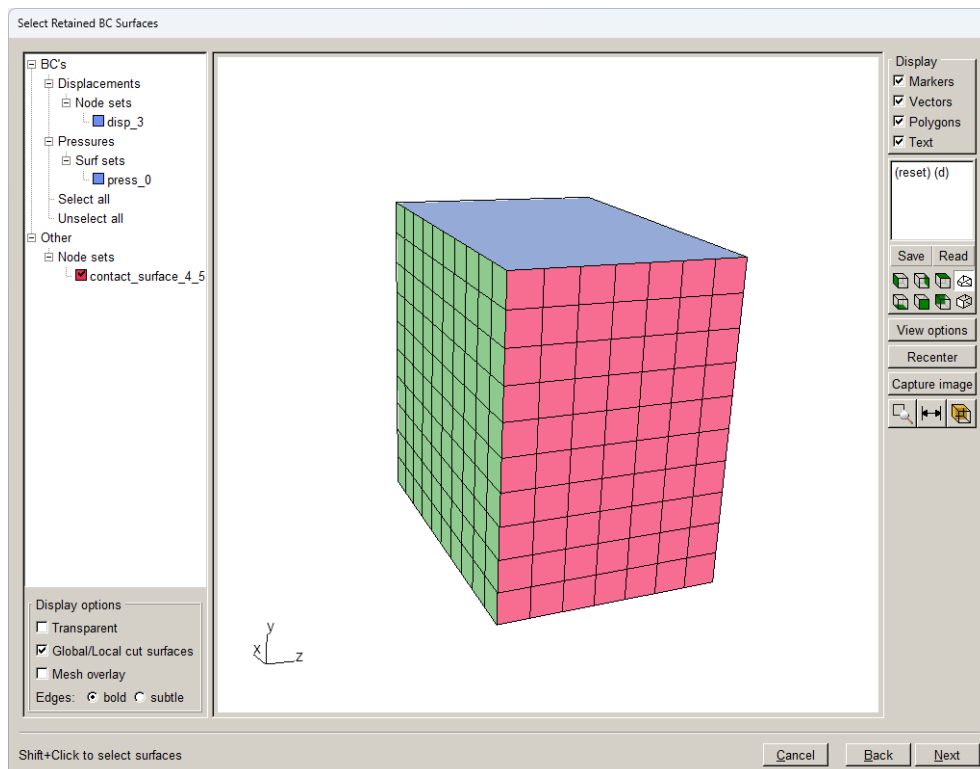


Figure 3.16 Select Retained BC Surfaces dialog with cut surface and selected contact surface.

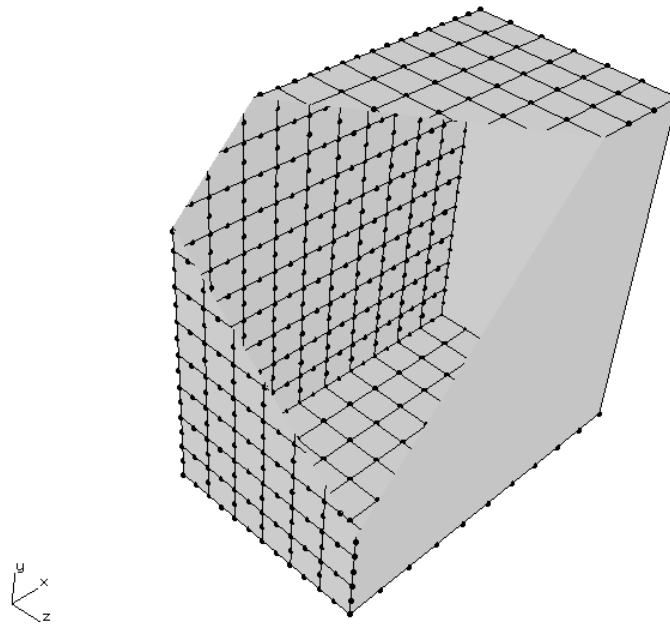


Figure 3.17 Local model with retained nodes and facets.

We insert the same crack as in Section 3.3, Fig 3.18. The resulting remeshed crack model is shown in Fig 3.19; we have pushed the front cutting plane into the model to show the crack surface.

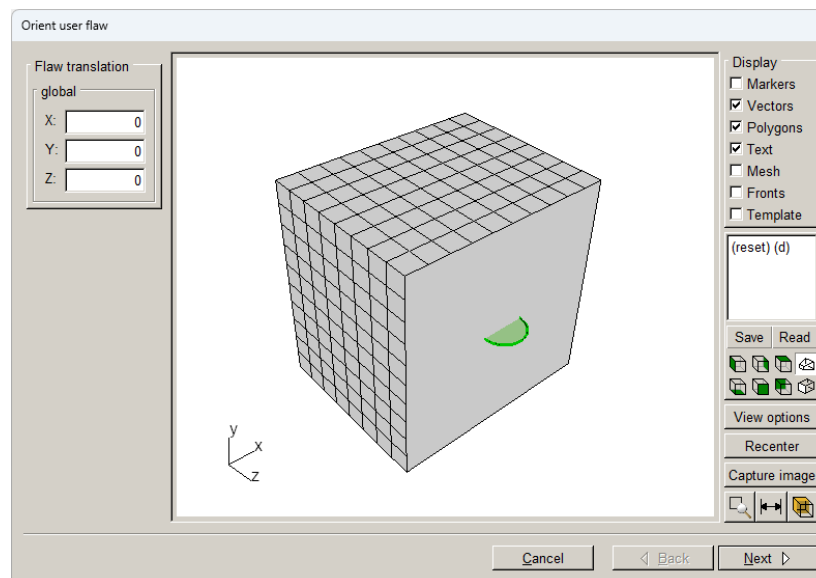


Figure 3.18 Surface crack inserted into local model.

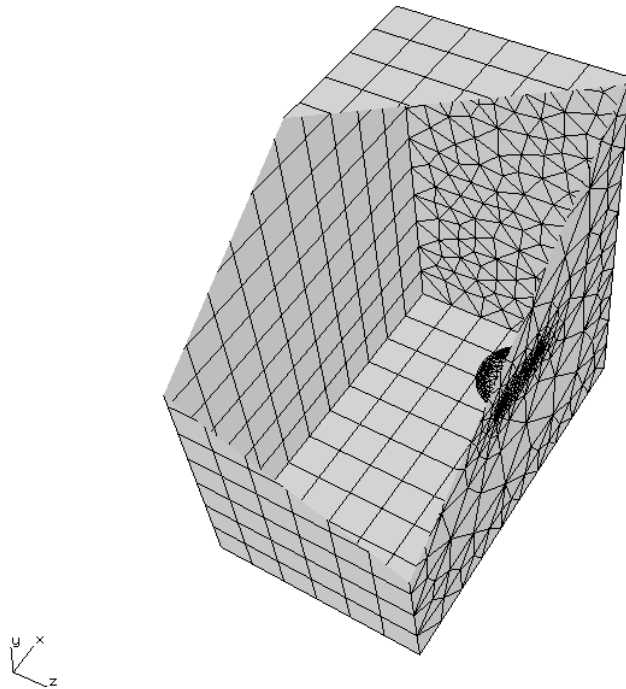


Figure 3.19 Surface mesh for cracked local model.

Once the crack is inserted, we do a static crack analysis. Fig 3.20 shows the local/global connection dialog when using the ANSYS FE model and solver. The AUTO_CUT_SURF and GLOBAL_CONNECT_SURF will be connected by merging the nodes.

The contact surface is automatically regenerated by FRANC3D using the local portion of the contact surface in the cracked region plus the rest of this surface and the mating surface, both of which are part of the global model.

The SIFs are consistent with those shown in Section 3.5. Fig 3.21 shows the deformed shape of the full model in ANSYS, and Fig 3.22 shows the Mode I SIFs.

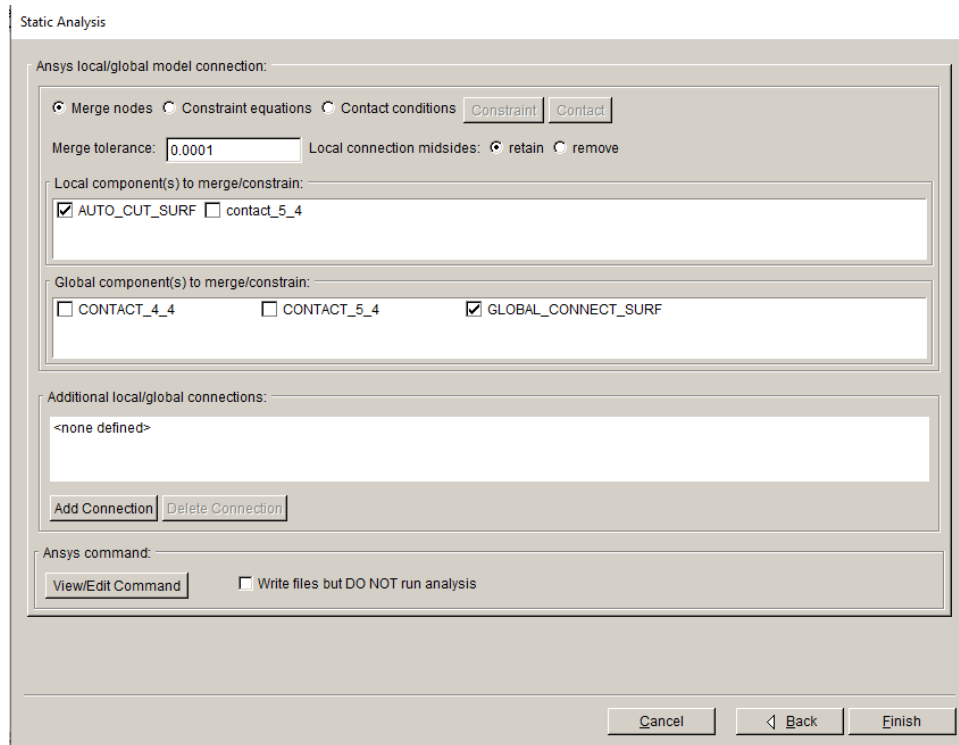


Figure 3.20 Local/global connection panel.

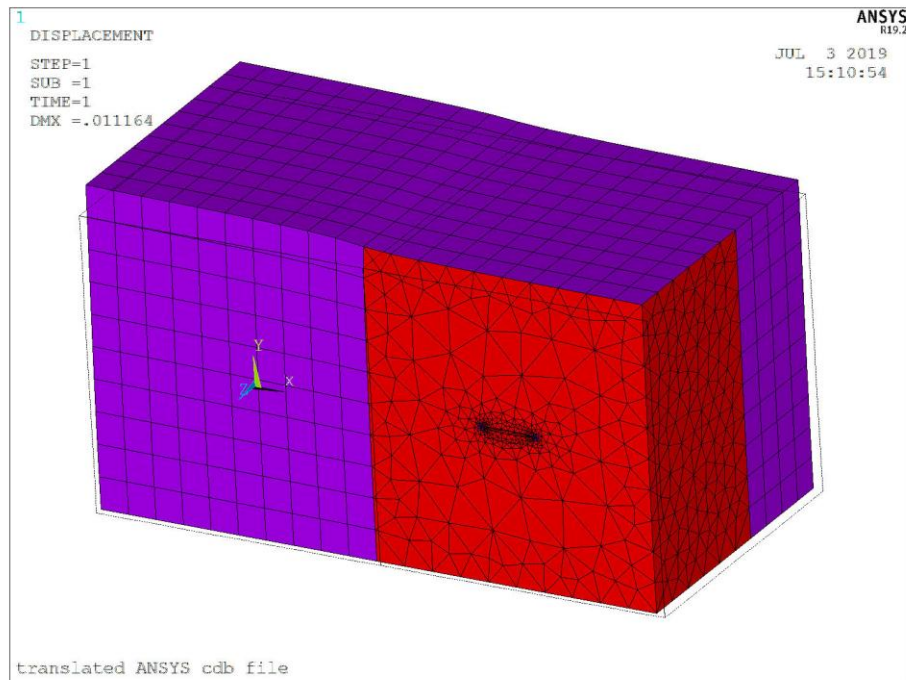


Figure 3.21 Deformed shape for full model.

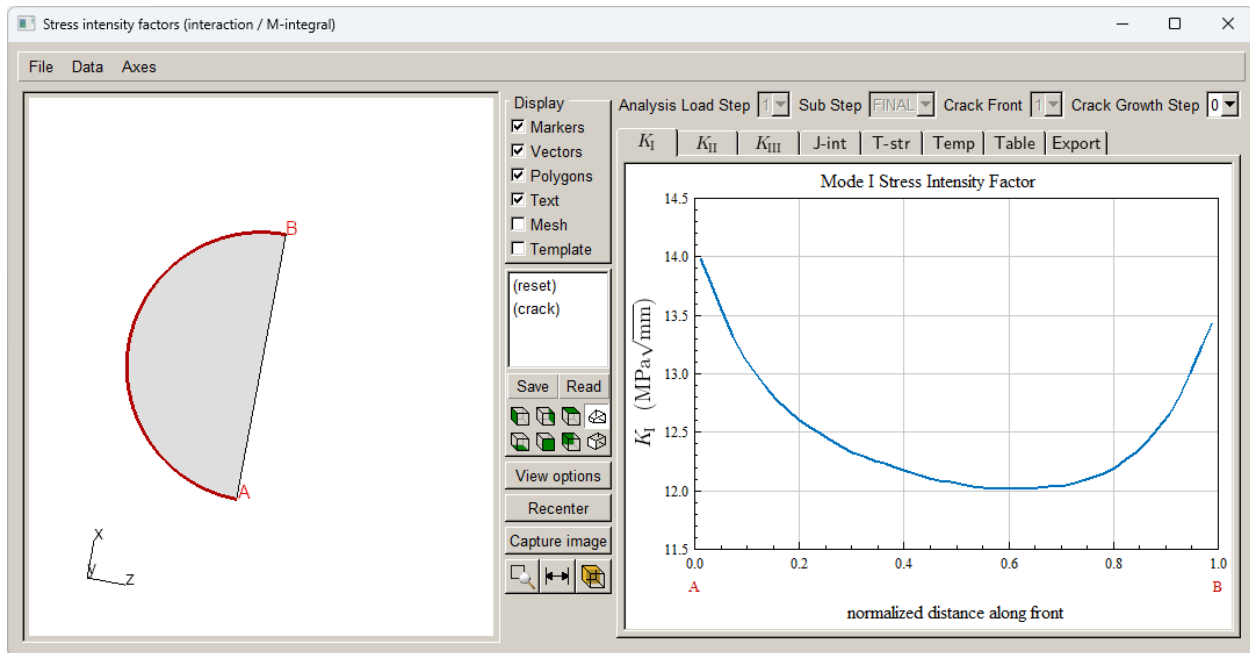


Figure 3.22 Mode I SIFs for case #1.

3.8.2 Local model has one side of the contact surface

For the second case, we do import and divide, but we stop cropping after selecting material #2 (see Fig 3.13). The entire second cube is the local model, and the entire contact surface from this second cube is contained within the local portion.

With this selection cropped, proceed with saving the local and global model files, and then proceed to the Select Retained BC Surfaces dialog, Fig 3.23.

There is no AUTO_CUT_SURF node set in this case. One side of the contact surface is included in the contact_5_4 node set. We retain this node set for simplicity. The top and bottom BC surfaces are retained also. The resulting local model is shown in Fig 3.24.

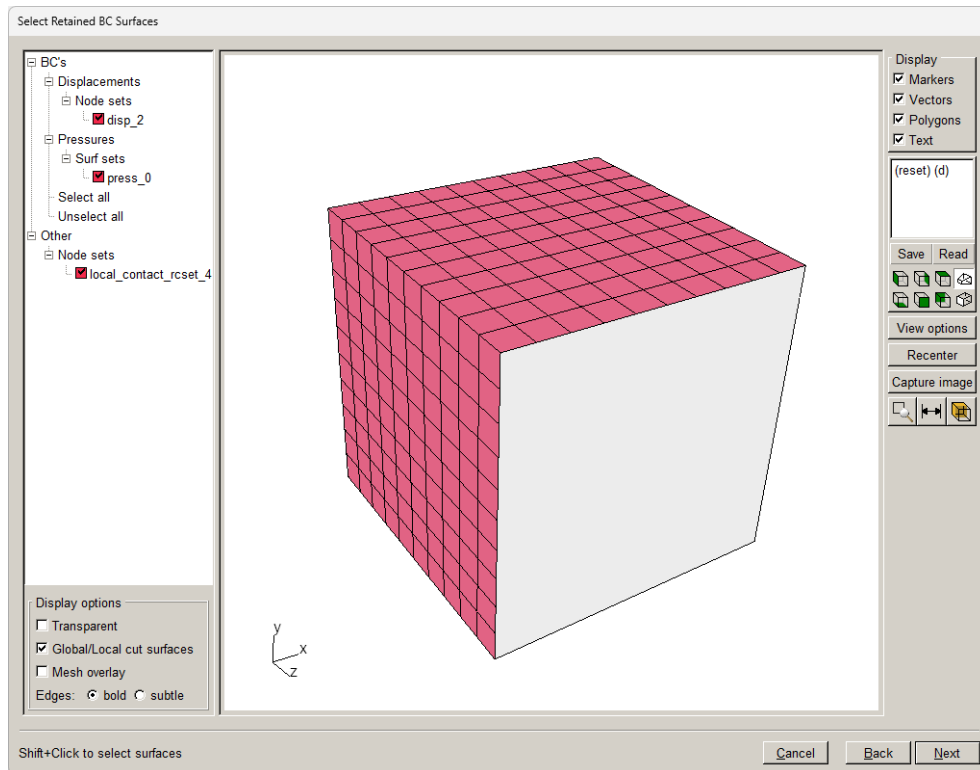


Figure 3.23 Select retained BC surfaces for material #2 selection.

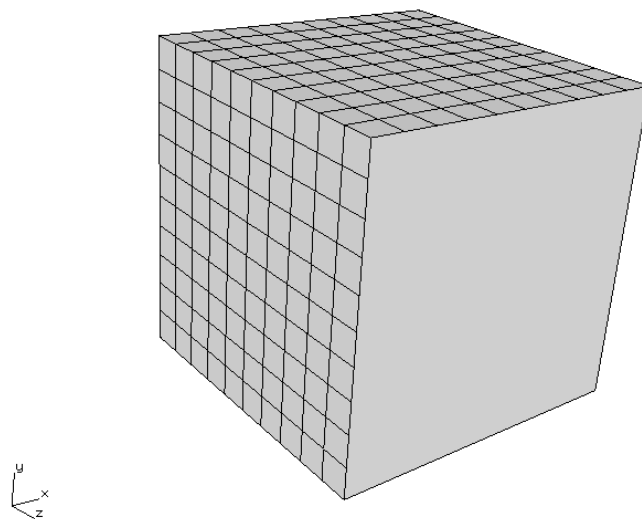


Figure 3.24 Local model for material #2 selection.

We insert the same crack as before, Fig 3.25; the resulting cracked meshed model is shown in Fig 3.26.

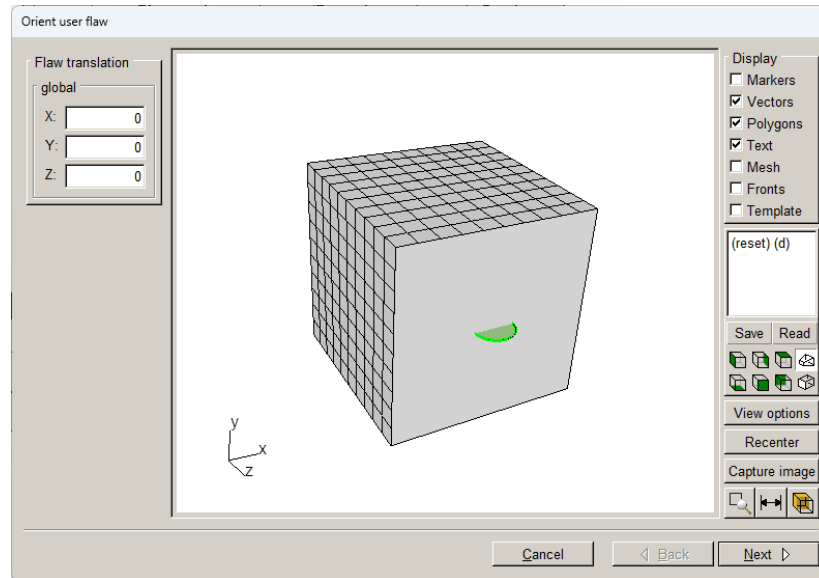


Figure 3.25 Local model for material #2 selection.

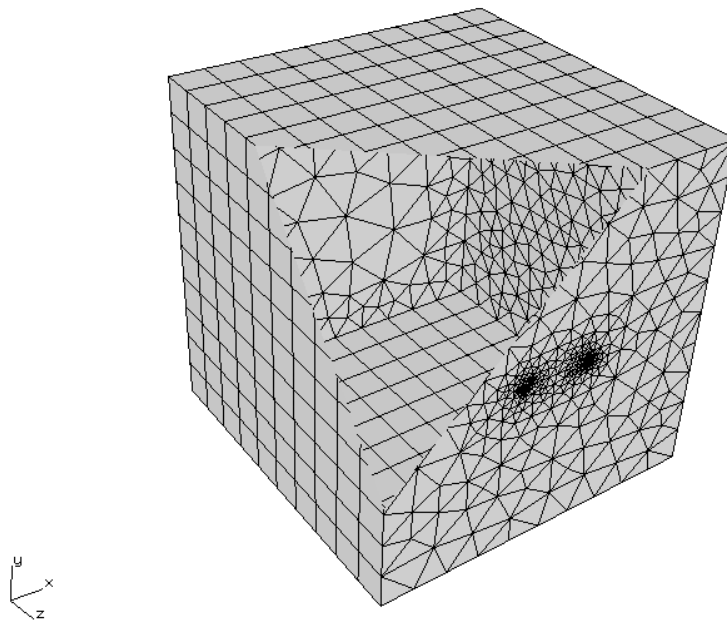


Figure 3.26 Surface mesh for cracked local model.

Once the crack is inserted and remeshed, we can proceed with the static crack analysis. The local/global connection panel for the ANSYS FE model and solver is shown in Fig 3.27. In this case, there is no cut surface. Instead, we connect the local and global using Contact conditions. We could also use Constraints, but we will use bonded contact to be consistent with the uncracked ANSYS model.

Select the **Contact** button to display the contact settings, Fig 3.28. The contact behavior is set to bonded always. Changes to other contact settings might improve the results; you should check the results in ANSYS to verify that the cracked model deformation is consistent with the original uncracked model.

Mode I SIFs are shown in Fig 3.29 and are consistent with previous results.

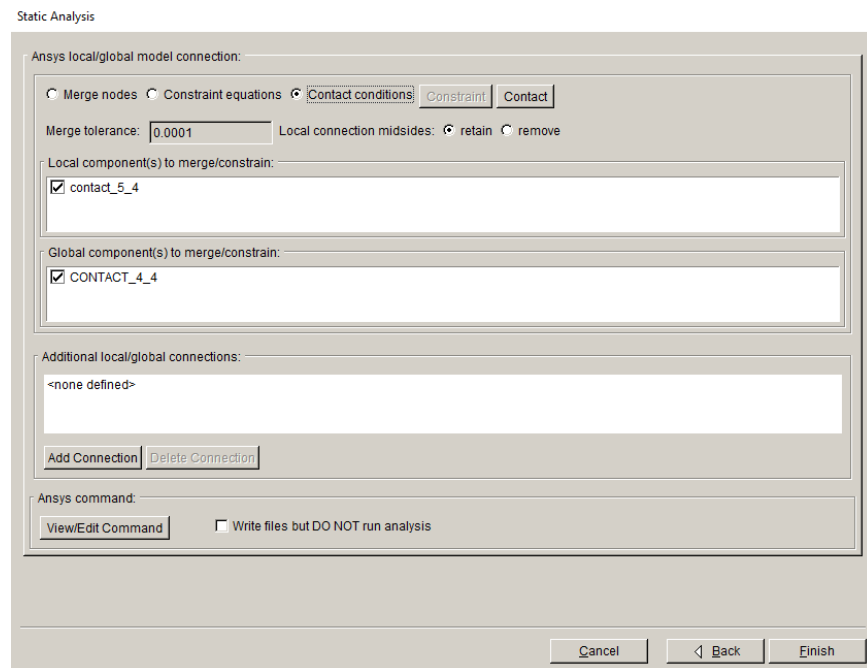


Figure 3.27 Local/global ANSYS connection using contact.

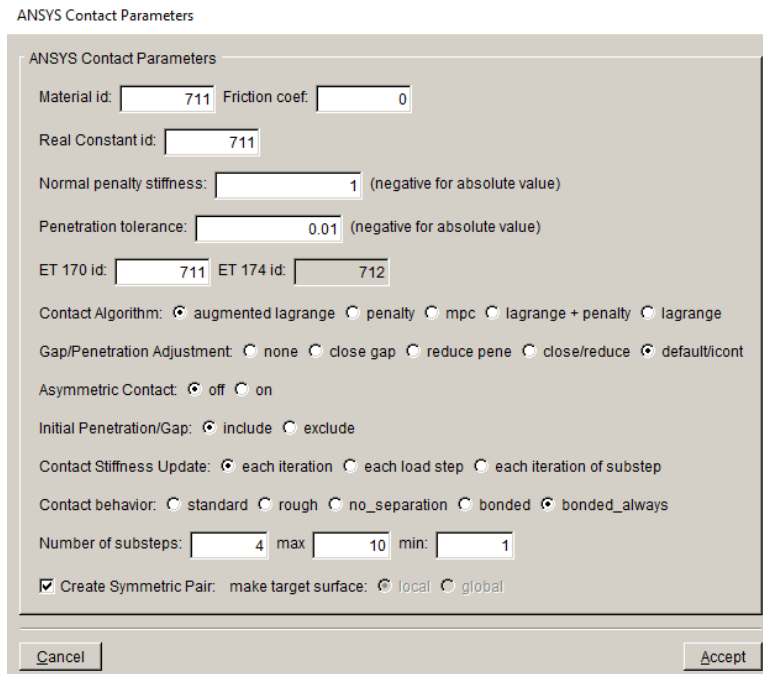


Figure 3.28 Local/global ANSYS contact settings.

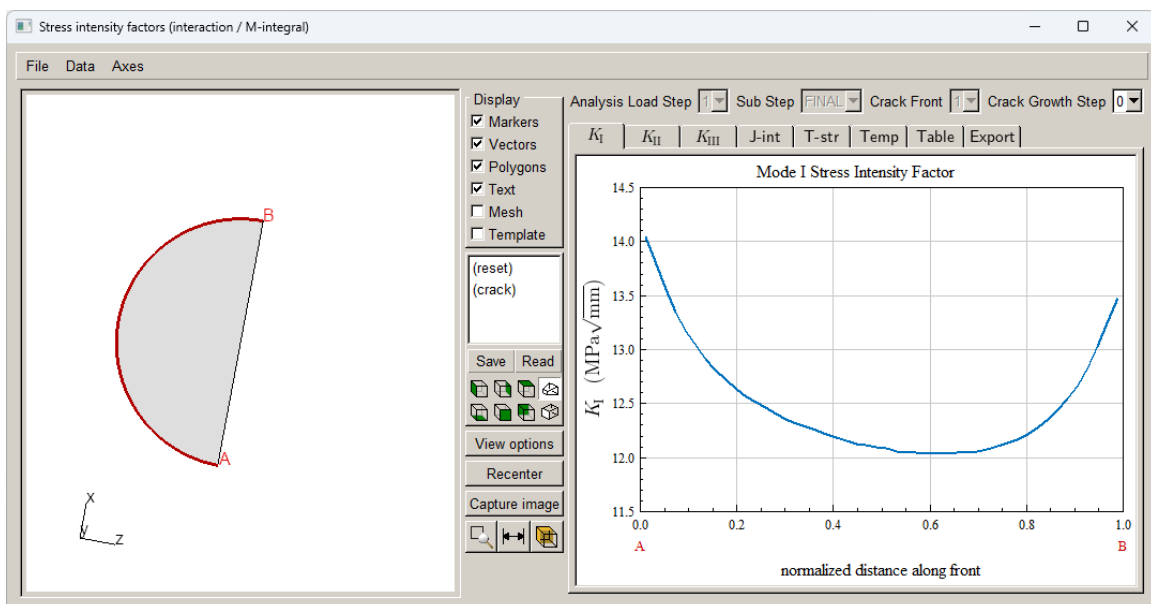


Figure 3.29 Mode I SIFs for case #2.

3.8.3 Local model has one side of the contact surface plus a cut-surface

For the third case, we again do import and divide, but the second selection consists of the region shown in Fig 3.30. This selection contains the entire contact surface of the second cube and has a cut surface. This local model will require “extra” connection settings in FRANC3D for the static crack analysis.

With the crop selection in Fig 3.30, proceed with saving the local and global model files, and then proceed to the Select Retained BC Surfaces dialog, Fig 3.31. We will retain the top and bottom BC surfaces along with the cut-surface and the contact surface.

The resulting local model is shown in Fig 3.32. We insert the same crack as before, Fig 3.33. The resulting remeshed cracked local model is shown in Fig 3.34. Once the crack is inserted, we can proceed with the static analysis. The local/global connections are a little more complicated in this case.

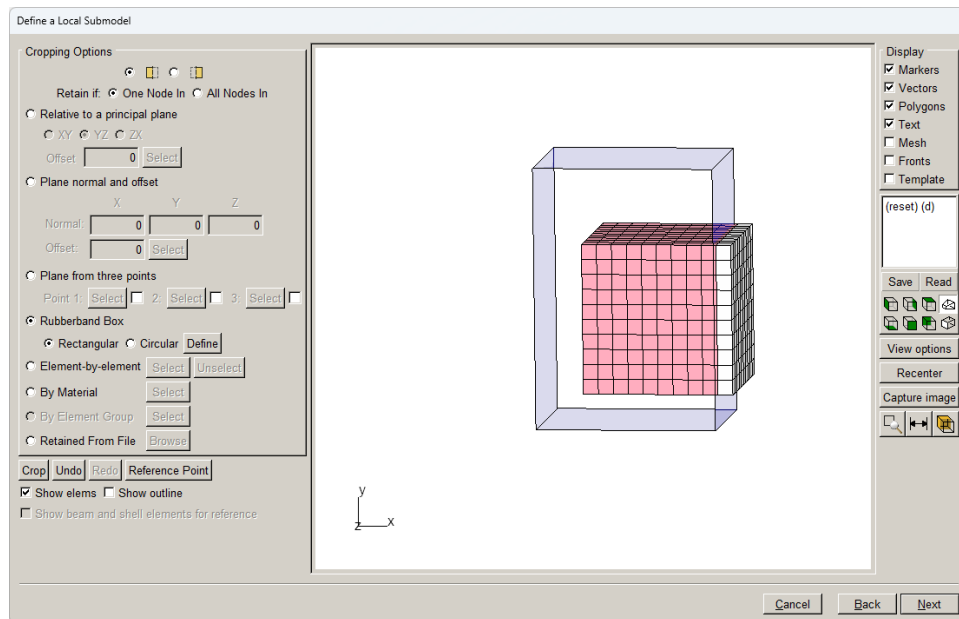


Figure 3.30 Select part of material #2.

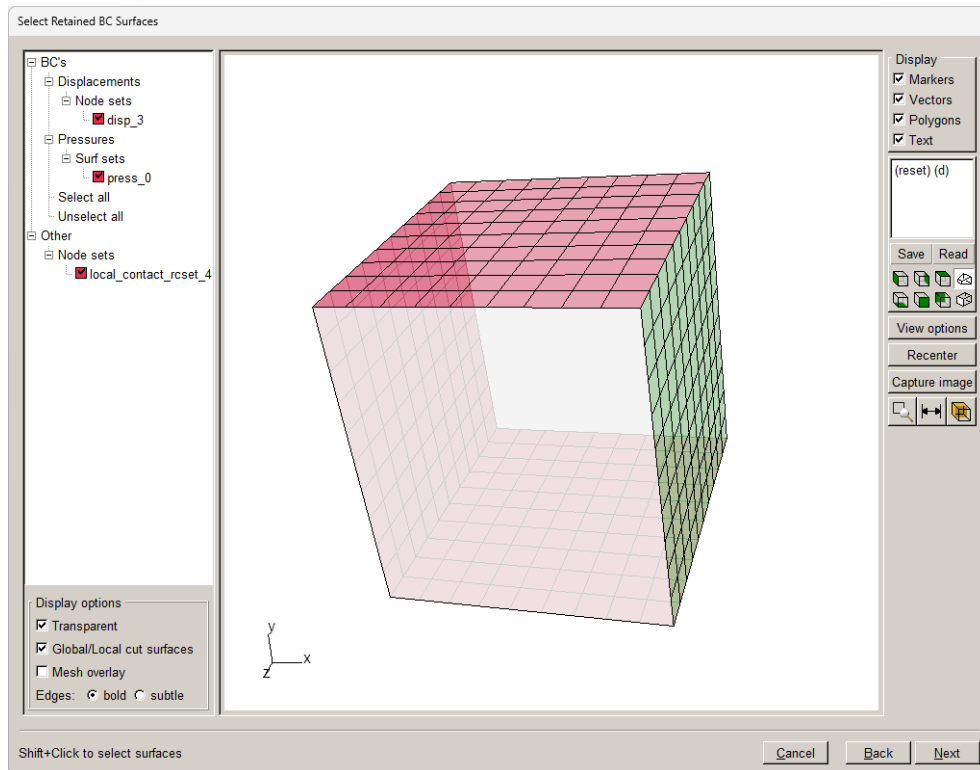


Figure 3.31 Retain BC surfaces along with cut-surface and contact surface.

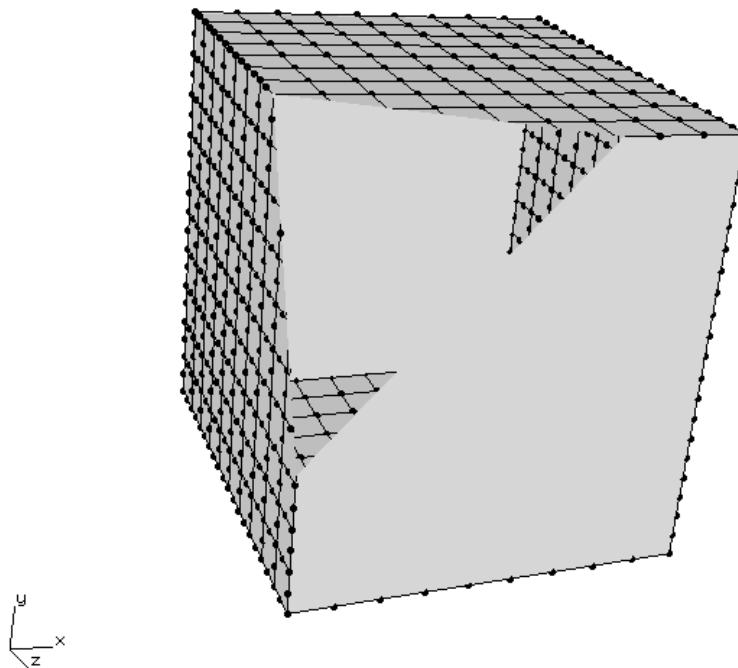


Figure 3.32 Local model with BC surfaces, cut surface and contact surface retained.

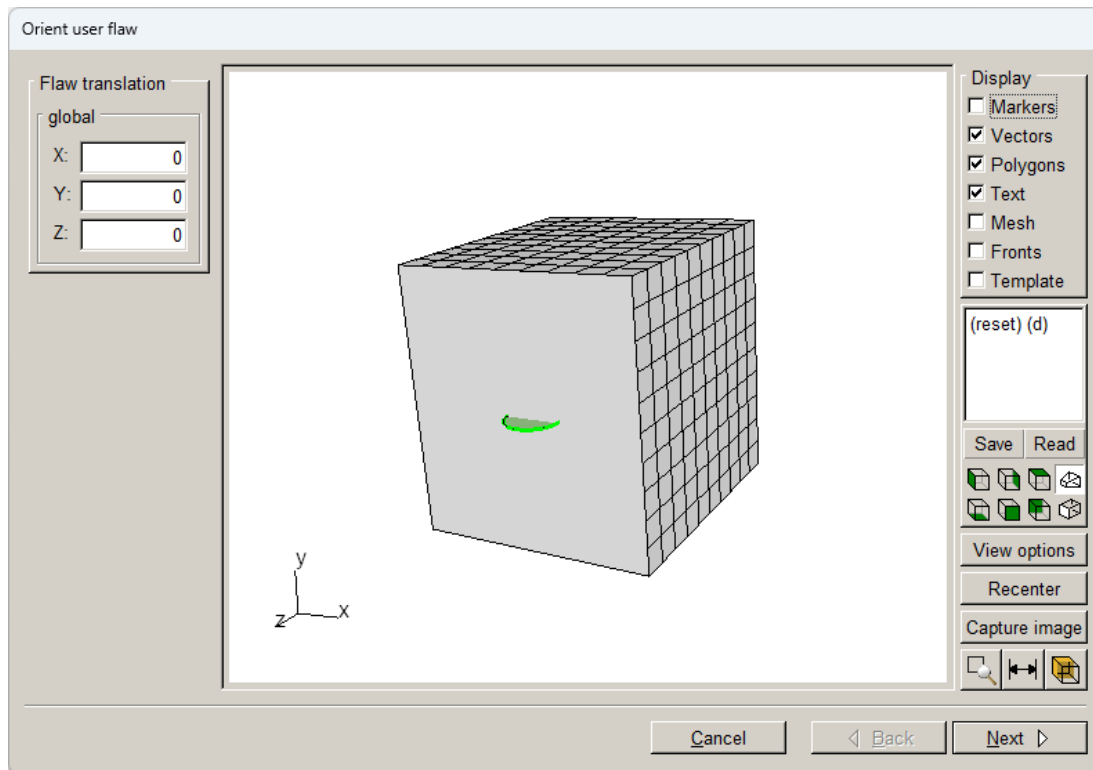


Figure 3.33 Crack inserted into local model.

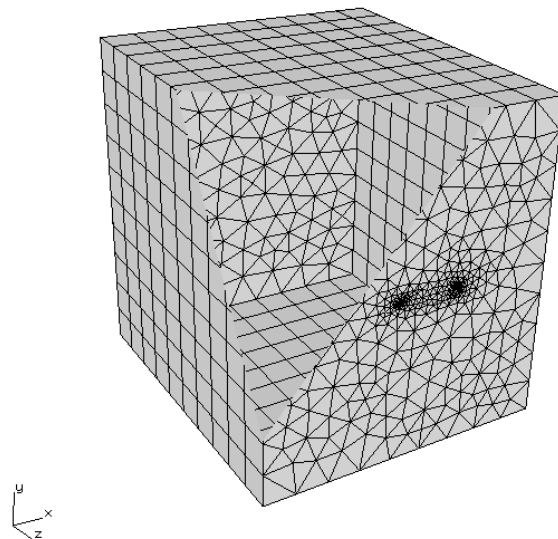


Figure 3.34 Resulting surface mesh for local crack model.

Fig 3.35 shows the initial local/global connection panel for the ANSYS FE model and solver, where the AUTO_CUT_SURF and GLOBAL_CONNECT_SURF sets are selected for node merging.

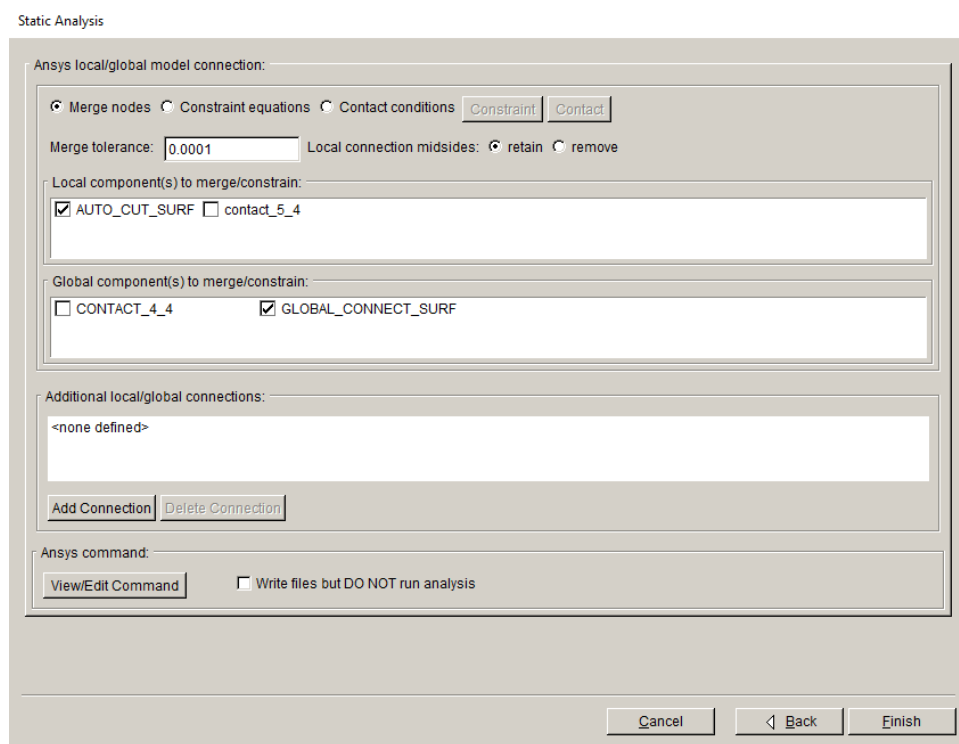


Figure 3.35 Initial local/global connection panel.

Because we completely separated the contact surfaces in the local and global portions, we must also add an “extra” connection. Select the **Add Connection** button to display the dialog in Fig 3.36.

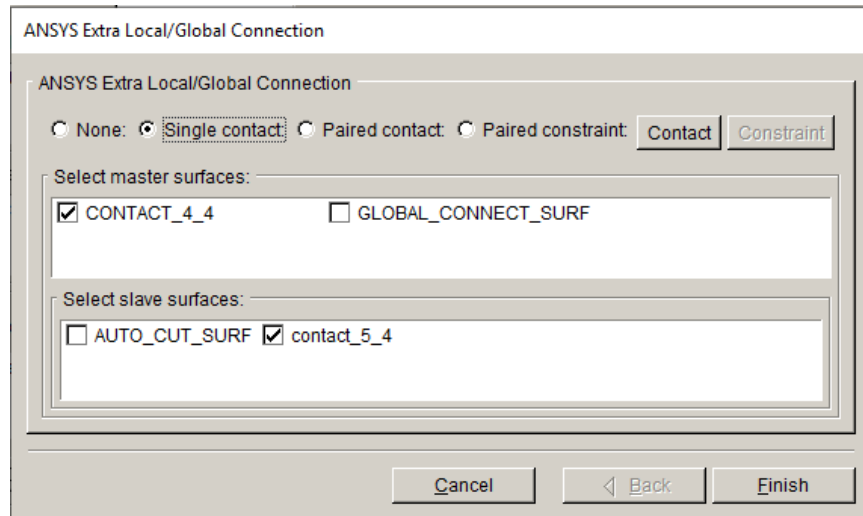


Figure 3.36 Extra connection dialog.

Switch the connection type to Single contact and then select the **Contact** button to display the dialog in Fig 3.37. Switch the Contact behavior to bonded_always.

Select the **Accept** button in Fig 3.37 to return to the top level local/global connection panel. It will now include the extra connection that was just defined, Fig 3.38. Proceed with the analysis and verify that the ANSYS results are consistent with results for the original uncracked model.

The Mode I SIFs for this case are shown in Fig 3.39 and are consistent with previous results.

ANSYS Contact Parameters

ANSYS Contact Parameters

Material id: Friction coef:

Real Constant id:

Normal penalty stiffness: (negative for absolute value)

Penetration tolerance: (negative for absolute value)

ET 170 id: ET 174 id:

Contact Algorithm: ☒ augmented lagrange ☐ penalty ☐ mpc ☐ lagrange + penalty ☐ lagrange

Gap/Penetration Adjustment: ☐ none ☐ close gap ☐ reduce pene ☐ close/reduce ☒ default/cont

Asymmetric Contact: ☒ off ☐ on

Initial Penetration/Gap: ☒ include ☐ exclude

Contact Stiffness Update: ☒ each iteration ☐ each load step ☐ each iteration of substep

Contact behavior: ☐ standard ☐ rough ☐ no_separation ☐ bonded ☒ bonded_always

Number of substeps: max min:

☒ Create Symmetric Pair: make target surface: ☒ local ☐ global

Figure 3.37 Extra connection contact settings.

Static Analysis

Ansyes local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☒ retain ☐ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF ☐ contact_5_4

Global component(s) to merge/constrain:

☐ CONTACT_4_4 ☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

contact_1_CONTACT_4_4_contact_5_4

Ansyes command:

☐ Write files but DO NOT run analysis

Figure 3.38 Final local/global connection panel.

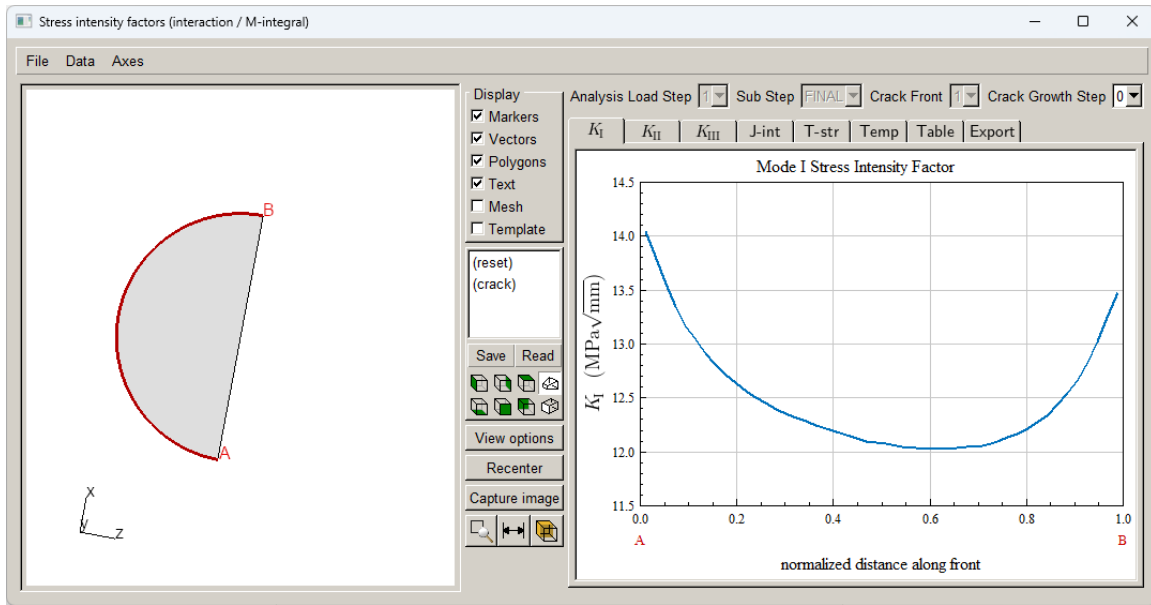


Figure 3.39 Mode I SIFs for case #3.

Tutorial #7 includes an extra contact connection, and Section 11 of the User's Guide includes additional information on local + global connections.

4.0 Tutorial 4: Disk with Rotation and Temperature

In Tutorial 4, we simulate crack growth in a simple disk model that includes temperature variation and rotation. In Step 1, we briefly describe the model and the thermal analysis that produces the temperature variation. Once the temperatures are defined, we run a structural analysis that includes rotation. From this structural model, we proceed with the crack growth simulation. This tutorial shows how temperature can affect SIFs.

4.1 Step 1: Create the Disk Model

The disk is created as a quarter-symmetry model. Heat transfer analysis is conducted to generate nodal temperatures, and these temperatures are then applied, along with mechanical loads (rotation) and constraints, to create a model that can be used for crack growth. The following three subsections describe the steps for ABAQUS, ANSYS and NASTRAN.

4.1.1 ABAQUS Disk

The first step is to build the FE model and to define the material properties. The model is shown in Fig 4.1a. The inner disk radius is 1, the outer radius is 10 and the width is 1.

We define both thermal and structural properties, which consist of density, specific heat, enthalpy, thermal expansion, conductivity, elastic modulus, and Poisson ratio. The material can be temperature dependent; for this analysis we use constant values for all data except for the elastic modulus, Fig 4.2a. Density is set to 1.0 and enthalpy is set to 0.

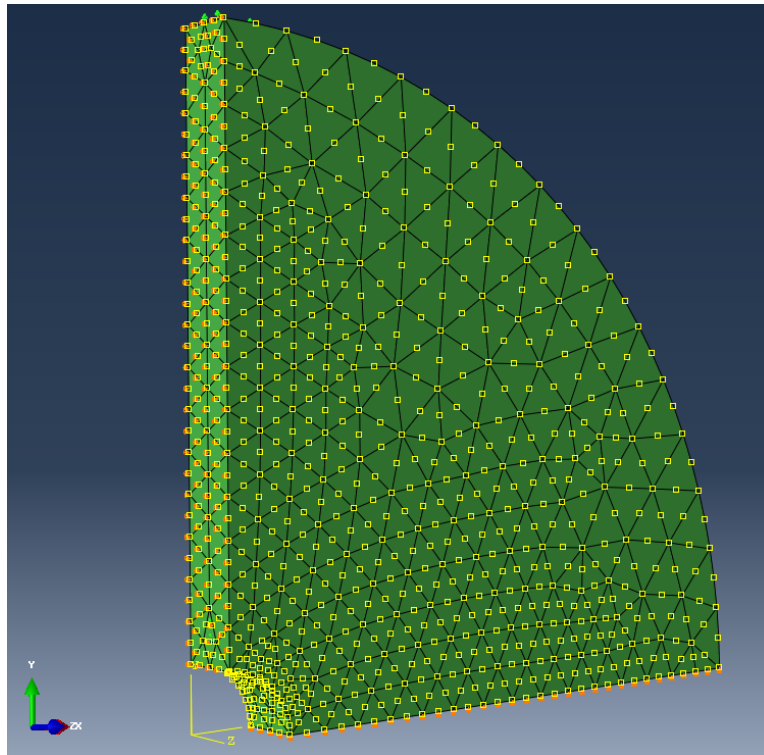


Figure 4.1a ABAQUS disk geometry.

Edit Material

Name: MAT_1
Description:

Material Behaviors

Density
Elastic
Expansion

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic Suboptions

☒ Use temperature-dependent data
Number of field variables: 0
Moduli time scale (for viscoelasticity): Long-term

☐ No compression
☐ No tension

Data

	Young's Modulus	Poisson's Ratio	Temp
1	10000	0.25	0
2	5000	0.25	1000

OK Cancel

Edit Material

Name: MAT_1
Description:

Material Behaviors

Density
Elastic
Expansion

General Mechanical Thermal Electrical/Magnetic Other

Density

Distribution: Uniform
☒ Use temperature-dependent data
Number of field variables: 0

Data

	Mass Density	Temp
1	1	0
2	1	1000

OK Cancel

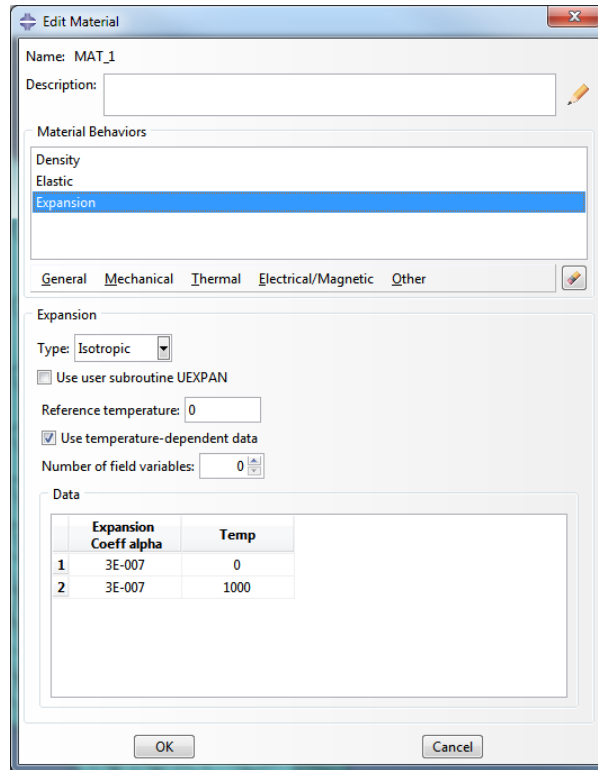


Figure 4.2a ABAQUS disk material properties.

4.1.1.1 ABAQUS Disk Thermal Analysis

The first analysis is for heat transfer, so we set the element type (DC3D10) and create a volume mesh. The model is meshed with all tetrahedral elements, Fig 4.3a. The boundary conditions consist of temperatures on two surfaces; temperature at the inner radius of the disk is set to 1000 and temperature at the outer radius is set to 100, Fig 4.3a. The resulting nodal temperatures are shown in Fig 4.4a. These nodal temperatures are applied as initial conditions for the structural analysis.

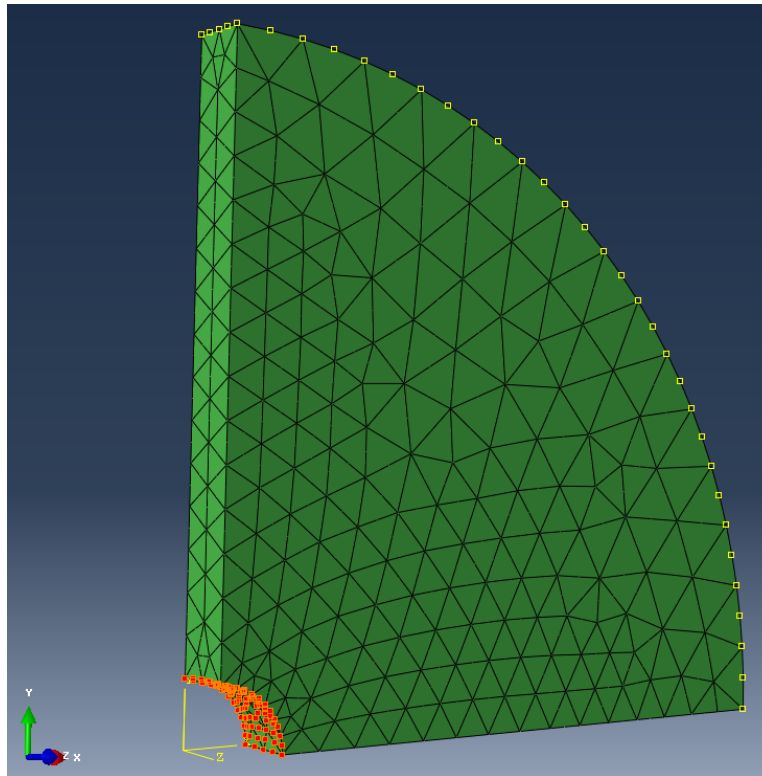


Figure 4.3a Volume mesh and thermal boundary conditions.

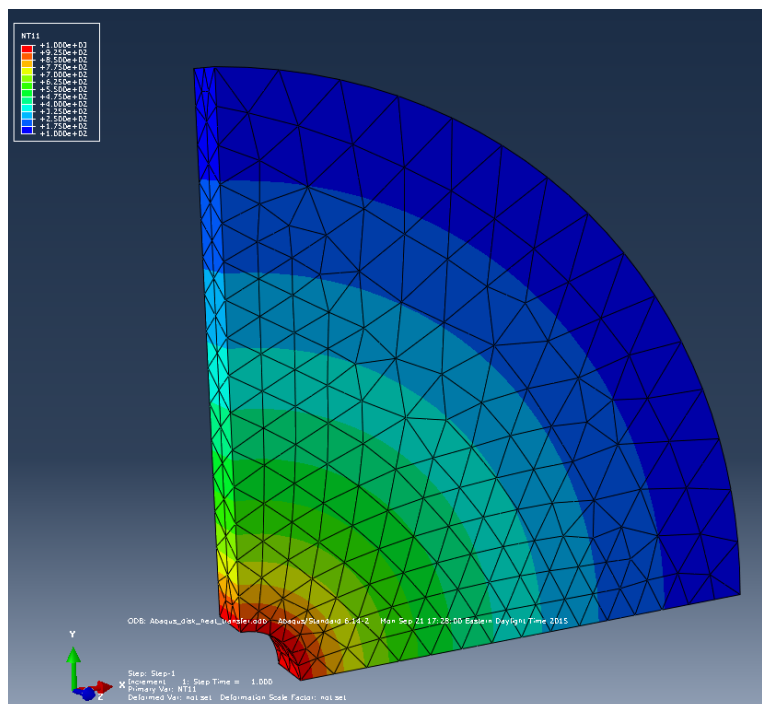


Figure 4.4a Resulting nodal temperature contours.

4.1.1.2 ABAQUS Disk Structural Analysis

We switch the element type from DC3D10 to C3D10 for the structural analysis. The boundary conditions include rollers on the (symmetry) surfaces at $x=0$ and $y=0$. We also fix a point in the z -direction to prevent rigid body motion and apply the nodal temperatures from the thermal analysis as initial conditions. Finally, we define the rotational velocity (centrif) as 0.2809 about the z -axis. The model with boundary conditions is shown in the left panel of Fig 4.5a. The resulting maximum principal stress contours from the structural analysis are shown in the right panel of Fig 4.5a.

We save the ABAQUS *.cae* file in case we need to re-analyze the model, and we create an *.inp* file for use with FRANC3D. The global and local model portions could be created in ABAQUS. However, for this tutorial, we rely on the FRANC3D submodeling tool, so we just archive the full model. We also save the ABAQUS stress listing as *Abaqus_Disk.rpt* for use in Section 4.10

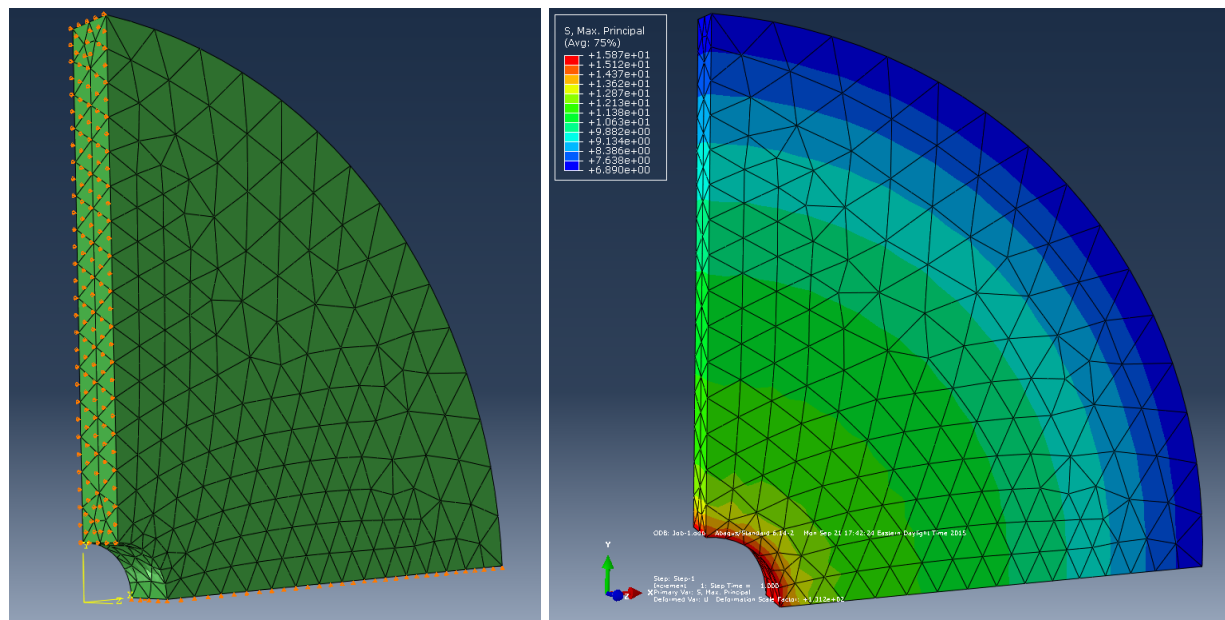


Figure 4.5a Boundary conditions for structural analysis – left panel, and resulting maximum principal stress contours – right panel.

4.1.2 ANSYS Disk

The first step is to build the FE model and to define the material properties. The outline of the disk model is shown in the left panel of Fig 4.1b. The inner radius is set to 1.0 and the outer radius is set to 10. The disk is set to be one unit wide; the final geometry is shown in the right panel of Fig 4.1b.

We define both thermal and structural properties, which consist of density, specific heat, enthalpy, thermal expansion, conductivity, elastic modulus, and Poisson ratio. The material can be temperature dependent; for this analysis we use constant values for all data except for the elastic modulus, Fig 4.2b. Density is set to 1.0 and enthalpy is set to 0.

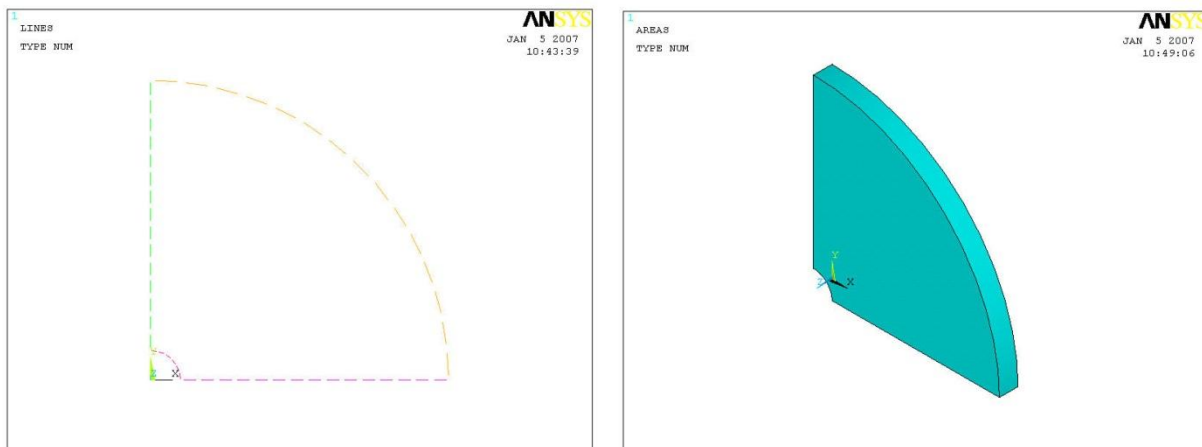
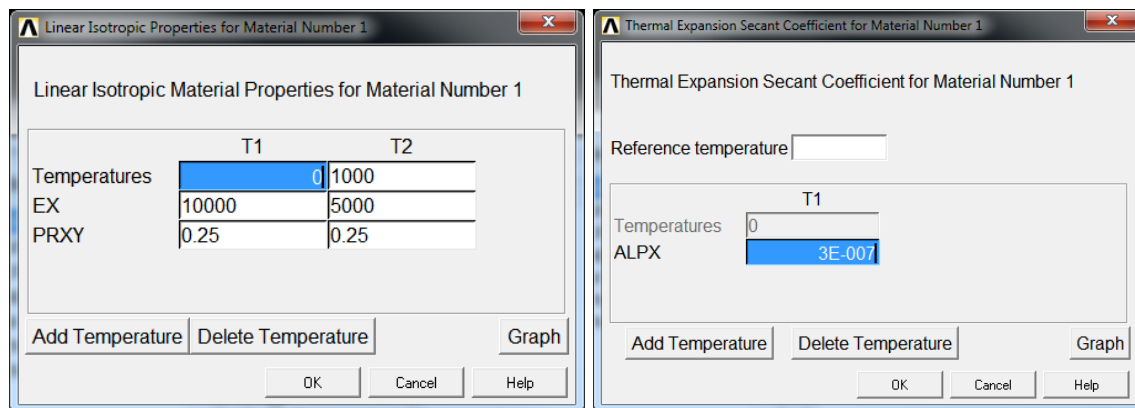


Figure 4.1b ANSYS disk geometry.



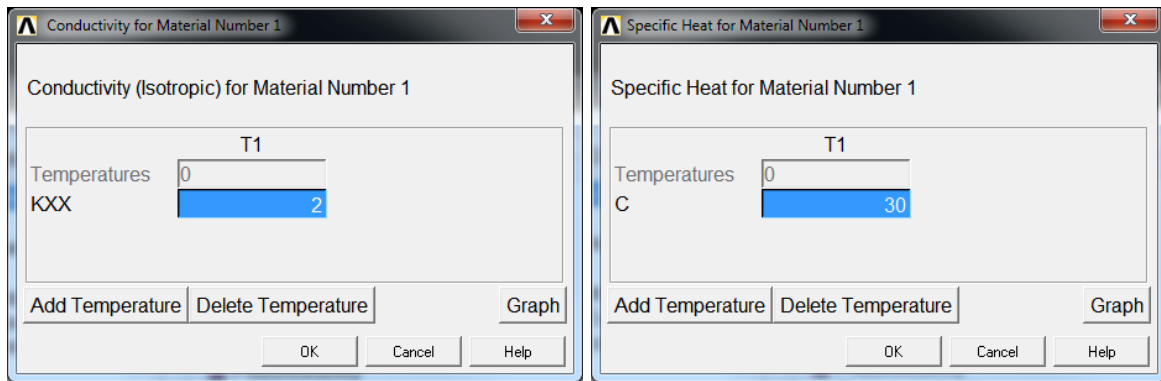


Figure 4.2b ANSYS disk material properties.

4.1.2.1 ANSYS Disk Thermal Analysis

The first analysis is for heat transfer, so we set the element type (Solid87) and create a volume mesh. The model is meshed with all tetrahedral elements, left panel of Fig 4.3b. The boundary conditions consist of temperatures on two surfaces; temperature at the inner radius of the disk is set to 1000 and temperature at the outer radius is set to 100, right panel of Fig 4.3b. The resulting nodal temperatures are shown in Fig 4.4b. These nodal temperatures are applied as initial conditions for the structural analysis.

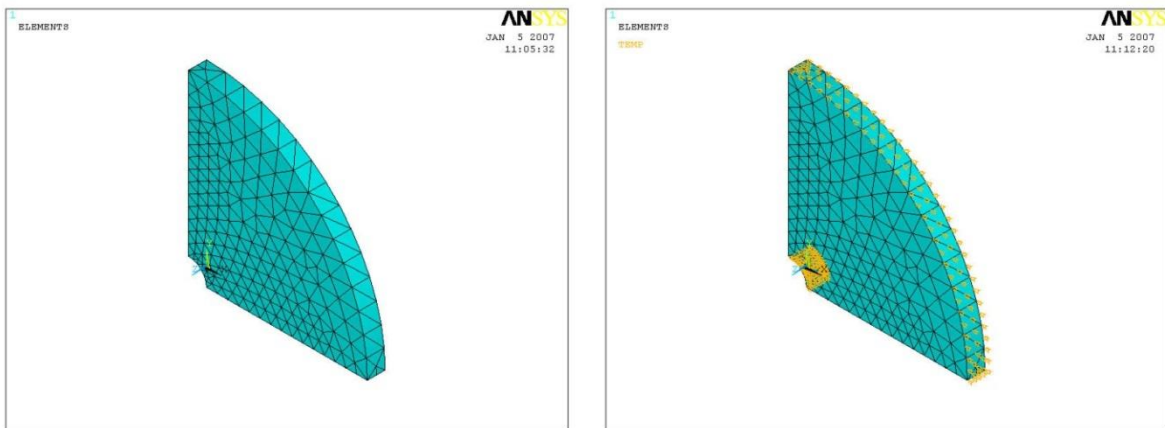


Figure 4.3b Volume mesh and thermal boundary conditions.

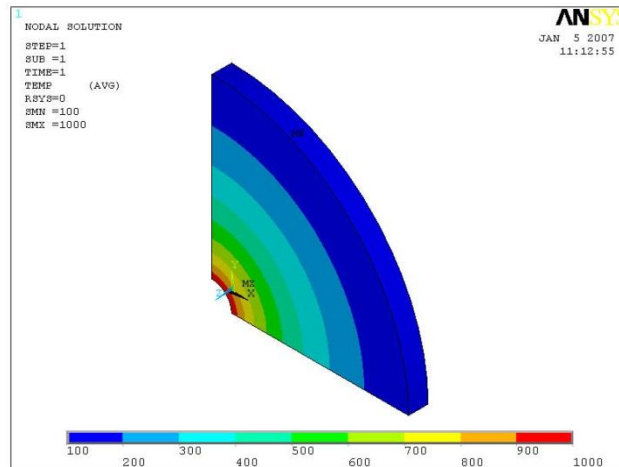


Figure 4.4b Resulting nodal temperature contours.

4.1.2.2 ANSYS Disk Structural Analysis

Switch the element type in the model from Solid87 to Solid92 for the structural analysis. The boundary conditions include rollers on the (symmetry) surfaces at $x=0$ and $y=0$. We also fix a point in the z -direction to prevent rigid body motion, and we apply the temperatures from the heat transfer analysis as initial conditions. Finally, we define the rotational velocity (ω) as 0.53 about the z -axis. The resulting model with boundary conditions should appear as in the left panel of Fig 4.5b. The resulting maximum principal stress contours from the structural analysis are shown in the right panel of Fig 4.5b.

We save the ANSYS *.db* file in case we need to re-analyze the model. We also archive the model to a *.cdb* file for use with FRANC3D. The global and local model portions could be created in ANSYS. However, for this tutorial we rely on the FRANC3D submodeling tool, so we just archive the full model; use “cdwite,DB,file_name,cdb”. We also save the ANSYS stress listing as *Ansys_Disk.str* for use in Section 4.10.

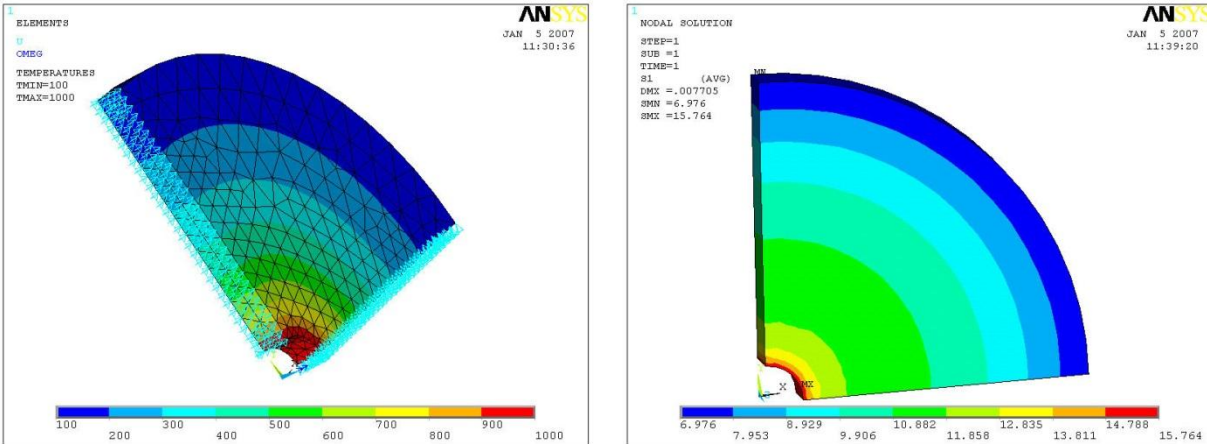


Figure 4.5b Boundary conditions for structural analysis – left panel, and resulting maximum principal stress contours – right panel.

4.1.3 NASTRAN Disk

The first step is to build the FE model and to define the material properties. The outline of the disk model is shown in the left panel of Fig 4.1c. The inner radius is set to 1.0 and the outer radius is set to 10. The disk is set to be one unit wide; the final geometry is shown in the right panel of Fig 4.1c.

We define both thermal and structural properties, which consist of density, specific heat, enthalpy, thermal expansion, conductivity, elastic modulus, and Poisson ratio. The material can be temperature dependent; for this analysis we use mostly constant values, Fig 4.2c. The elastic modulus is 10,000 at temperature=0 and 5,000 at temperature=1000. Poisson ratio is constant at 0.25. The density is set to 1.0. The coefficient of thermal expansion is set to $3e-7$.

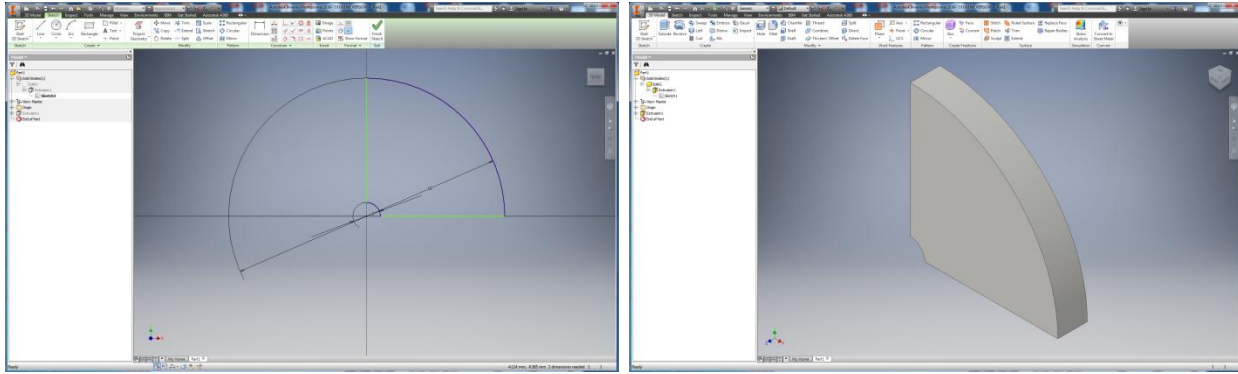


Figure 4.1c NASTRAN disk geometry.

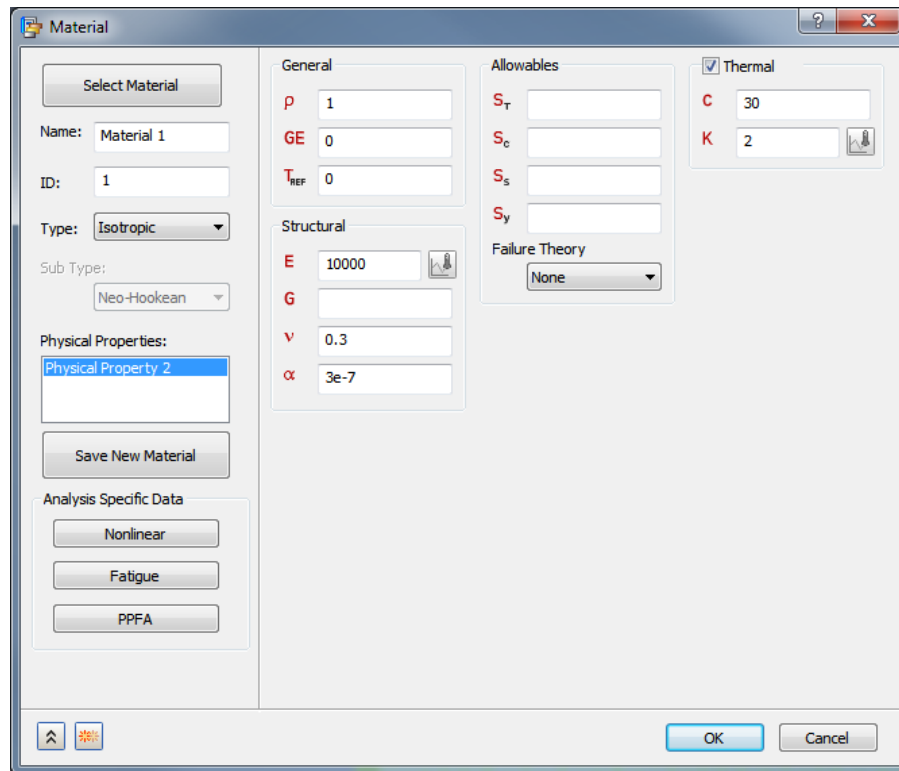


Figure 4.2c NASTRAN disk material properties.

4.1.3.1 NASTRAN Disk Thermal Analysis

The first analysis is for heat transfer. The model is meshed with all tetrahedral elements, Fig 4.3c. The boundary conditions consist of temperatures on two surfaces; temperature at the inner radius of the disk is set to 1000 and temperature at the outer radius is set to 100. The nodal

temperatures for the steady-state heat transfer are shown in Fig 4.4c. These temperatures are applied as initial conditions for the structural analysis.

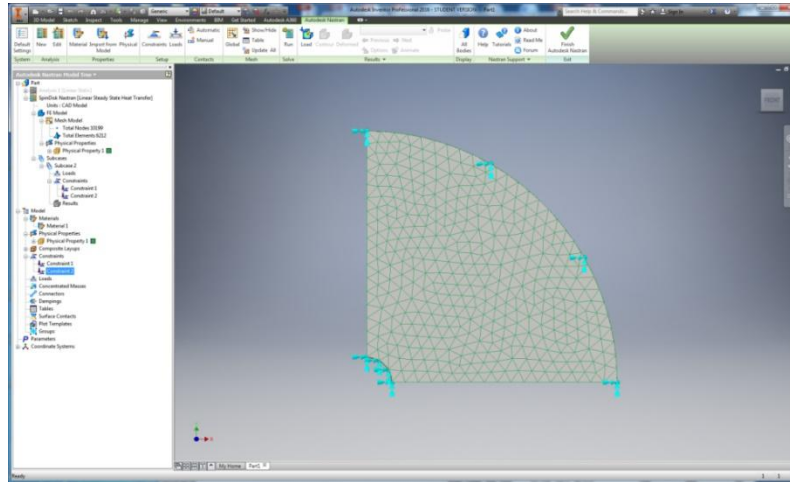


Figure 4.3c Volume mesh and thermal boundary conditions.

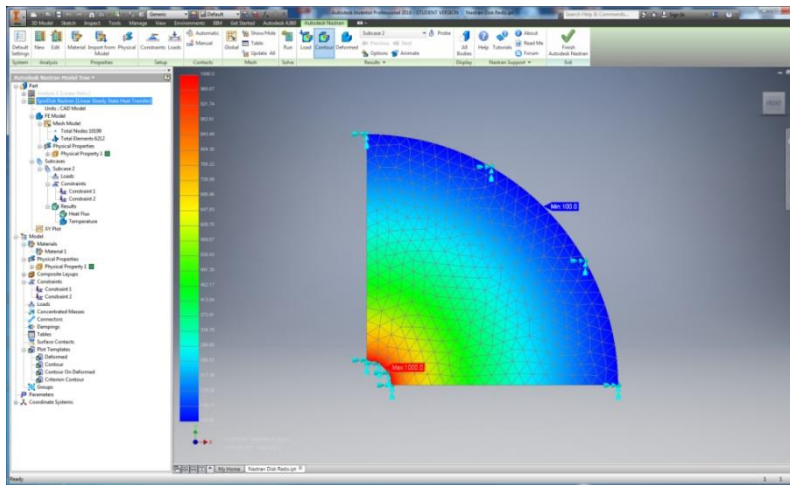


Figure 4.4c Resulting nodal temperature contours.

4.1.3.2 NASTRAN Disk Structural Analysis

The boundary conditions include rollers on the (symmetry) surfaces at $x=0$ and $y=0$. We also fix a point in the z -direction to prevent rigid body motion, and we apply the temperatures from the heat transfer analysis as initial conditions. Finally, we define the rotational velocity (rforce) as

0.08435 about the z-axis. The resulting model with boundary conditions should appear as in the left panel of Fig 4.5c. The resulting maximum principal stress contours from the structural analysis are shown in the right panel of Fig 4.5c.

We save the Inventor files in case we need to re-analyze the model. We also archive the model to a *.nas* file for use with FRANC3D; this is done automatically by Inventor when running the NASTRAN analysis. The global and local model portions could be created in NASTRAN. However, for this tutorial we rely on the FRANC3D submodeling tool, so we just archive the full model. We also save the NASTRAN stress listing as *Nastran_Disk.pch* for use in Section 4.10; take note of the subcase ID for the stress results if it is not set to 1.

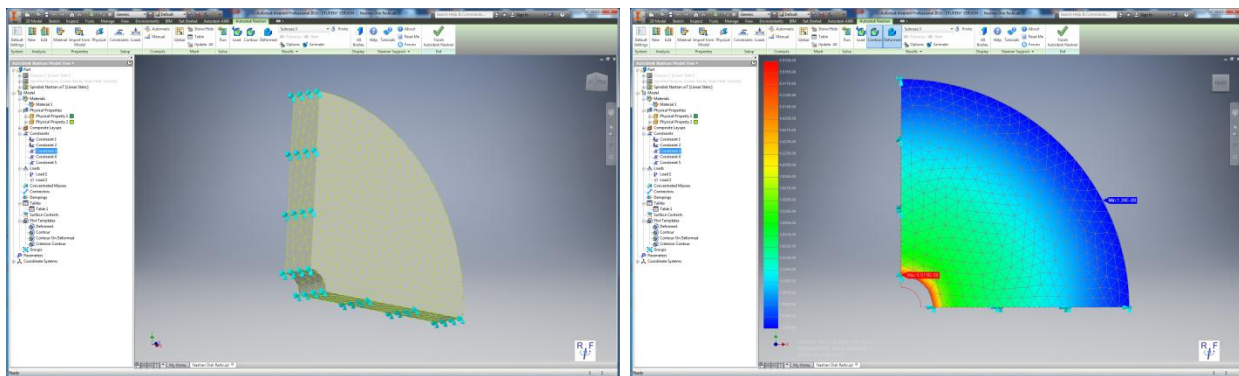


Figure 4.5c Boundary conditions for structural analysis – left panel, and resulting maximum principal stress contours – right panel.

4.2 Step 2: Import Disk into FRANC3D

Once the structural model is ready, we can proceed with the crack growth simulation. Start with the FRANC3D GUI and select **File** → **Import**. In the Select Type of Import panel, choose the Import and divide into global and local models radio button and select **Next**. Switch the Mesh File Type radio button to the appropriate FE type and select the file name for the model, Fig 4.6, and click **Next**.

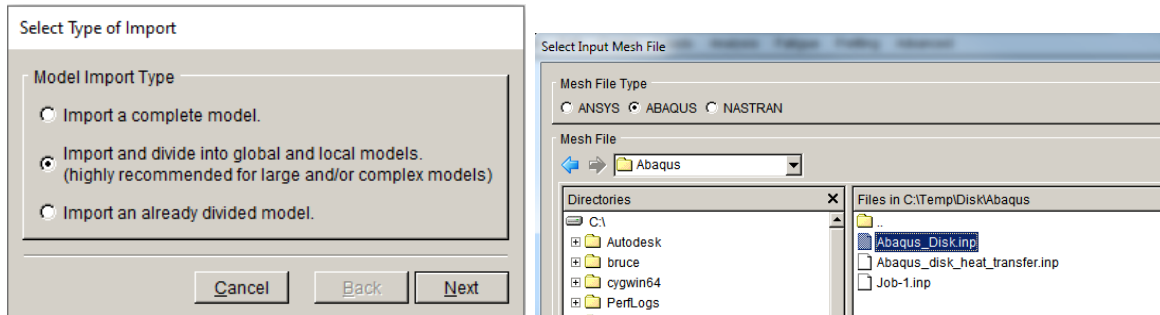


Figure 4.6 FRANC3D FE model import – ABAQUS selection.

Create a local model using the Rubberband Box tool and the **Crop** button in the Define Local Submodel window, Fig 4.7. Select **Next** and save the local and global model files using the default file names. In the Select Retained BC Surfaces dialog, Fig 4.8, use Select all to retain the two surfaces with displacement constraints, and then select **Next**. Set the FEM units so that they are consistent with the FE model units; in this case, the temperature units are also required. SI units of mm, MPa and C are used.

The model will be displayed in the FRANC3D main window, and a crack can be inserted into the local model.

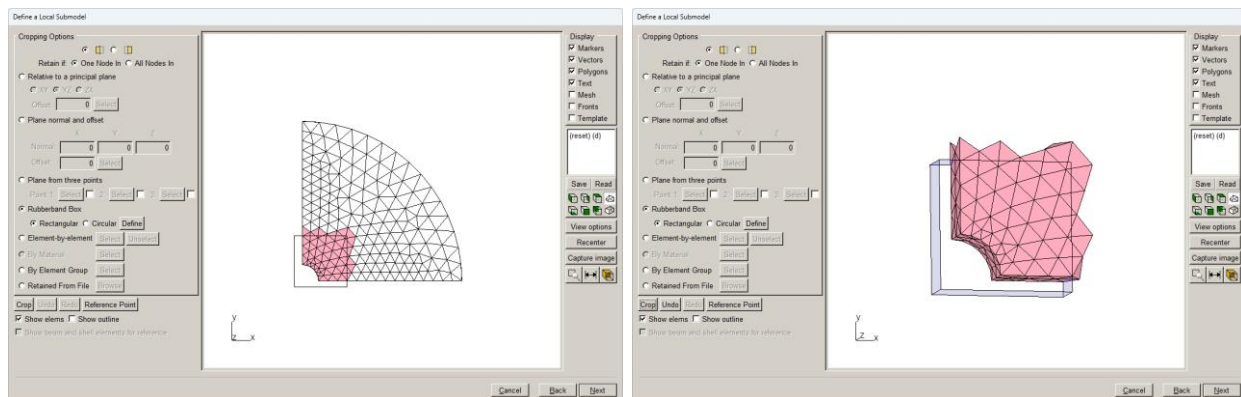


Figure 4.7 FRANC3D FE model import.

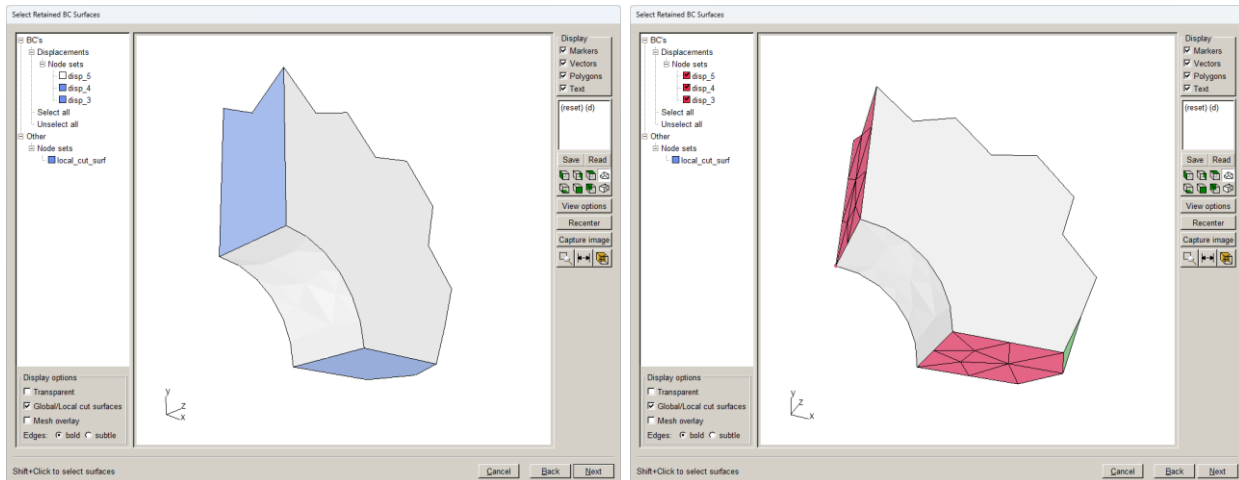


Figure 4.8 FRANC3D FE model import – BC surface selection.

4.3 Step 3: Insert Initial Crack

Insert a circular crack with a radius of 0.1, Fig 4.9. The crack is located at the front of the disk, rotated 45 degrees from the xz-plane, Fig 4.10. The crack front template has a radius of 0.01, Fig 4.11. The resulting remeshed crack model is shown in Fig 4.12.

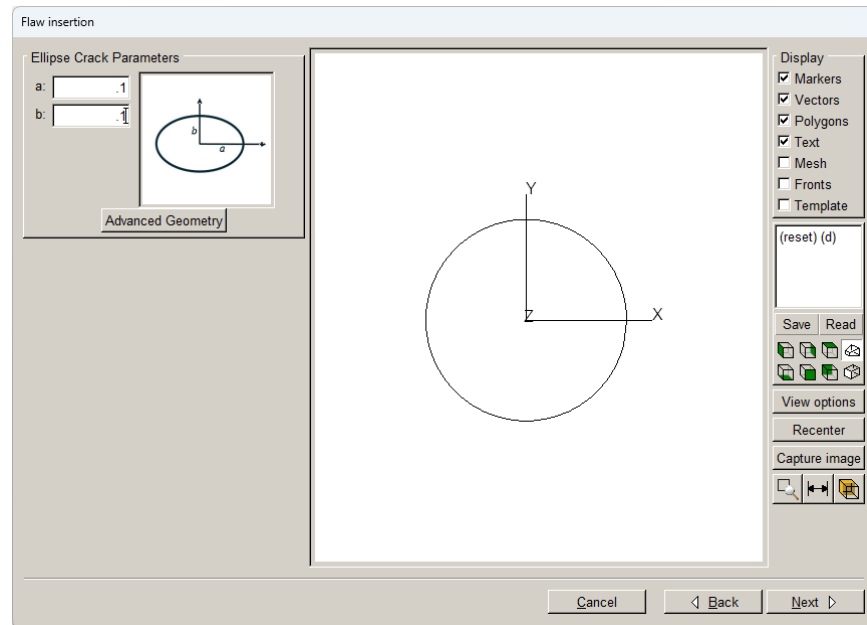


Figure 4.9 Elliptical crack dimension dialog.

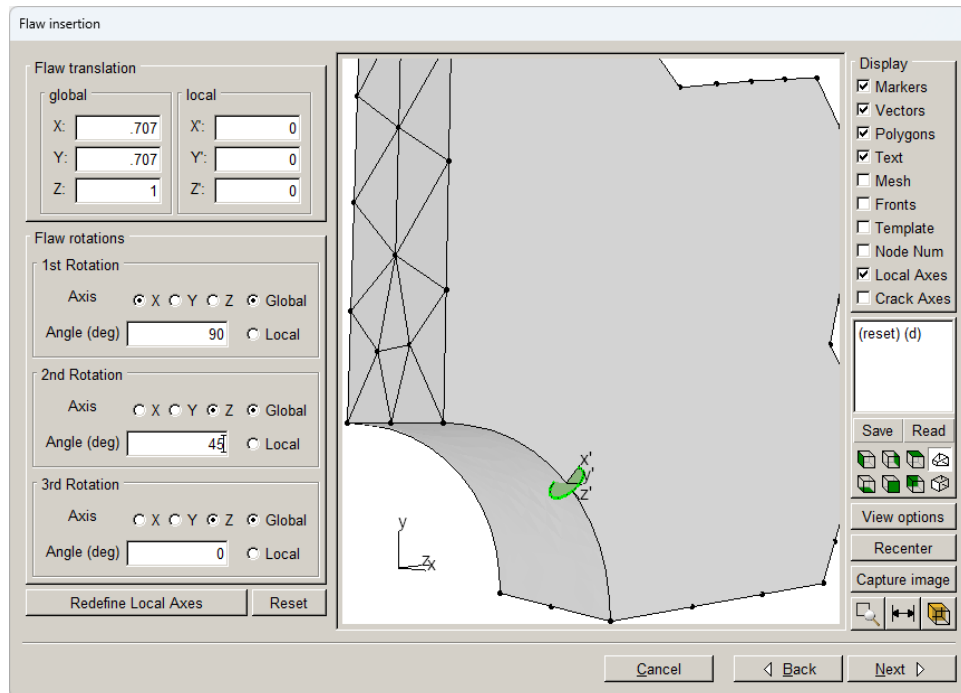


Figure 4.10 Crack location and orientation dialog.

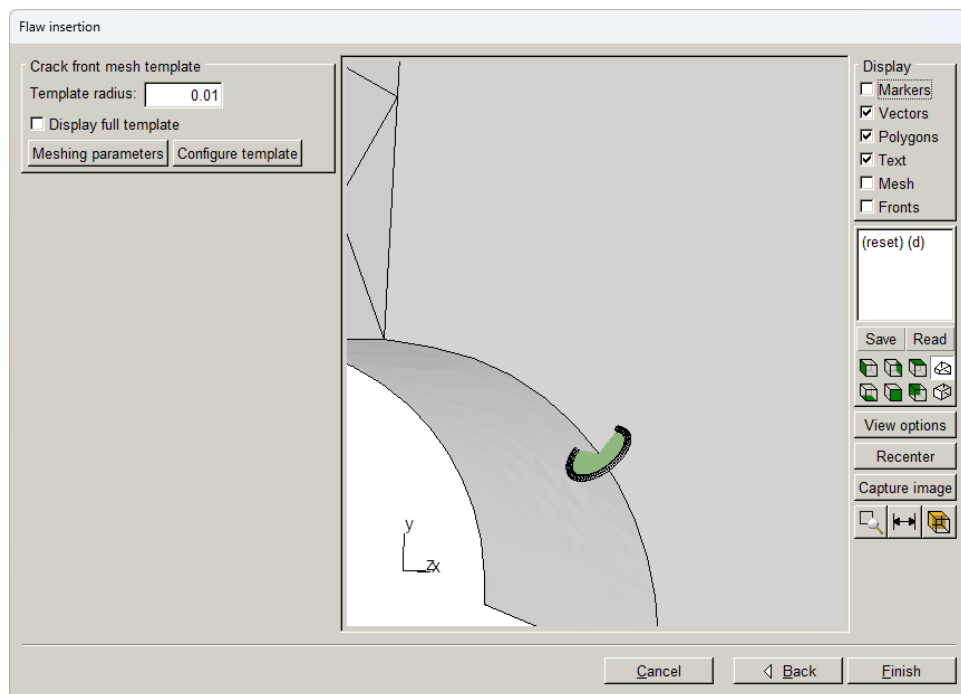


Figure 4.11 Crack front template dialog.

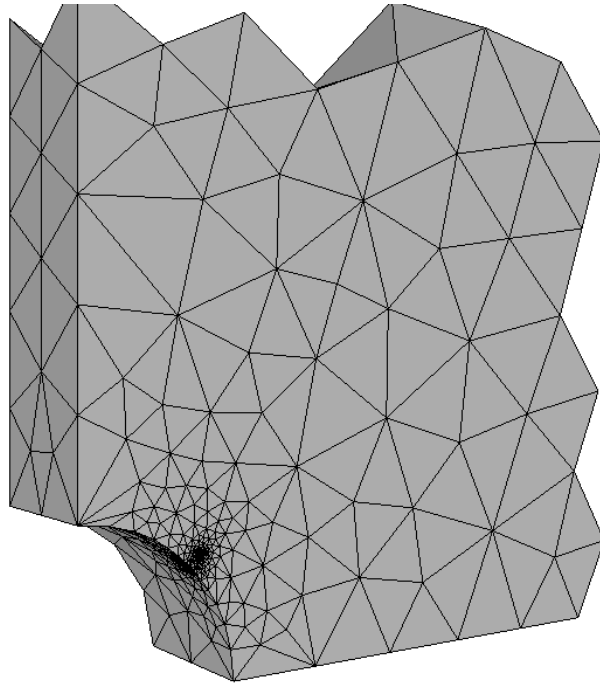


Figure 4.12 Final meshed crack model.

4.4 Step 4: Static Crack Analysis

We perform a static crack analysis. Choose **Analysis** → **Static Crack Analysis** from the FRANC3D menu. Provide a file name and proceed to the appropriate sub-section for your analysis code.

4.4.1 ABAQUS Solver

The ABAQUS options are shown in Fig 4.13a. The Connect to global model option is checked automatically. Select **Next**.

The next dialog box has options for connecting the local and global portions, Fig 4.14a. We use node merging with the automatically selected local and global cut-surfaces. Click **Finish** to begin the ABAQUS analysis.

Static Analysis

Abaqus Options

Abaqus run time:

Abaqus Executable:

Python Executable:

Python script (will run before Abaqus):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact

Abaqus command:

☐ Write files but DO NOT run analysis

Figure 4.13a Static analysis options for ABAQUS.

Static Analysis

Abaqus local/global model connection:

☒ Merge nodes ☐ Tie constraint ☐ Contact conditions ☒ Global = Master

Local node set/surface(s) to merge/constrain:

☒ AUTO_CUT_SURF ☐ disp_2 ☐ disp_3 ☐ disp_4

Global node set/surface(s) to merge/constrain:

☐ Show all labels

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

<none defined>

Abaqus command:

☐ Write files but DO NOT run analysis

Figure 4.14a Static analysis ABAQUS local + global model connection dialog.

4.4.2 ANSYS Solver

The ANSYS options are shown in Fig 4.13b. The Connect to global model option is checked automatically. Select **Next**.

The next dialog box has options for connecting the local and global portions, Fig 4.14b. We use node merging with the automatically selected local and global cut-surfaces. Click **Finish** to begin the ANSYS analysis.

The screenshot shows a dialog box titled "Static analysis" with a tab labeled "ANSYS Options". The dialog is organized into several sections:

- ANSYS run time:** Contains fields for "ANSYS Executable:" (with a file path and a "Browse" button), "ANSYS license:" (a dropdown menu showing "ansys"), "Python Executable:" (with "python" and a "Browse" button), and "Python script (will run before ANSYS):" (with an empty field and a "Browse" button").
- Local model output:** Contains an "Edit Defaults" button.
- Global model:** Contains a checked checkbox for "Connect to global model" and a "filename:" field with the value "Ansys_Disk_GLOBAL.cdb" and a "Browse" button.
- Boundary conditions:** Contains checkboxes for "Apply crack face tractions" and "Define crack face contact" (with a "Contact" button), and a checkbox for "Initial strain" (with a "Define" button).
- ANSYS command:** Contains a "View/Edit Command" button and a checkbox for "Write files but DO NOT run analysis".

At the bottom of the dialog are three buttons: "Cancel", "< Back", and "Next >".

Figure 4.13b Static analysis options for ANSYS.

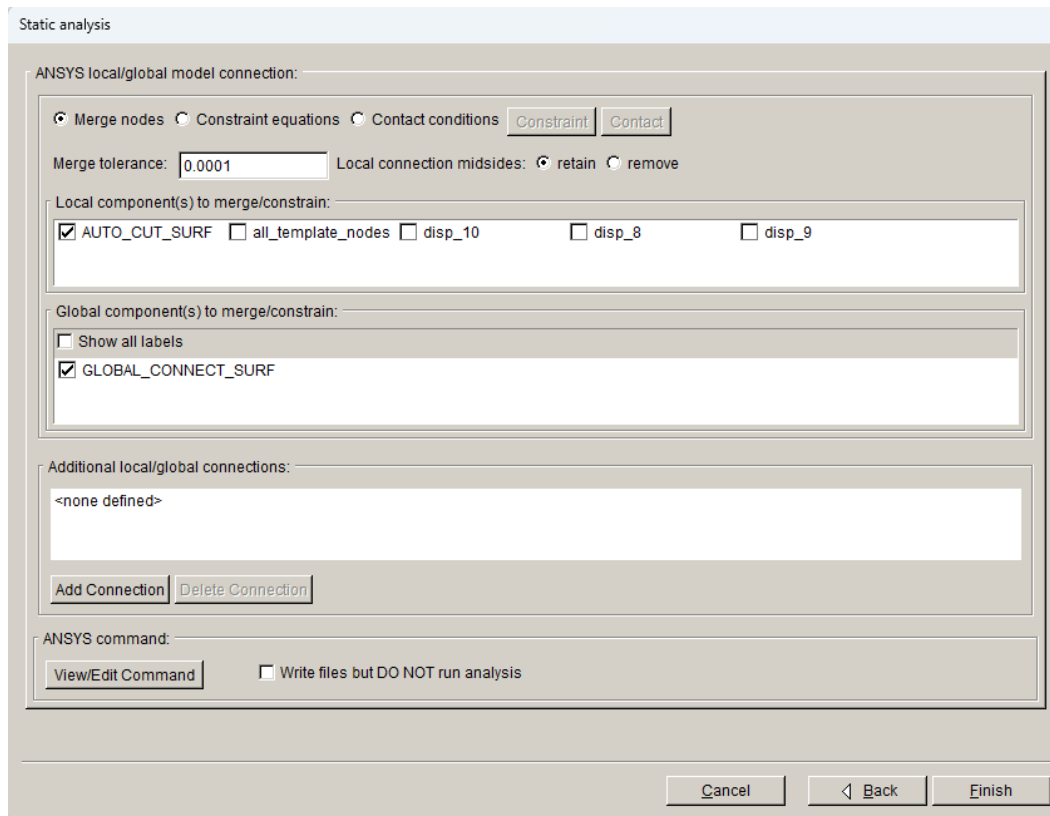


Figure 4.14b Static analysis ANSYS local + global model connection dialog.

4.4.3 NASTRAN Solver

The NASTRAN options are shown in Fig 4.13c. Note that the NASTRAN *.pch* results file does not contain temperatures; instead, the temperatures are extracted from the model boundary condition data. The Connect to global model option is checked automatically. Select **Next**.

The next dialog box has options for connecting the local and global portions, Fig 4.14c. We use node merging with the automatically selected local and global cut-surfaces. Click **Finish** to begin the NASTRAN analysis.

NASTRAN Options

NASTRAN run time:

NASTRAN Executable:

Python Executable:

Python script (will run before NASTRAN):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions: ☐ Define crack face contact:

NASTRAN command:

☐ Write files but DO NOT run analysis

Figure 4.13c Static analysis options for NASTRAN.

NASTRAN local/global model connection:

☒ Merge local and global nodes ☐ Constrain surfaces

Local node sets to merge/constrain:

☒ AUTO_CUT_SURF ☐ disp_2 ☐ disp_3 ☐ disp_4 ☐ disp_5
☐ disp_6 ☐ disp_7 ☐ disp_8

Global node sets to merge/constrain:

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

NASTRAN command:

☐ Write files but DO NOT run analysis

Figure 4.14c Static analysis NASTRAN local/global model connection dialog.

4.5 Step 5: Compute SIFs

Once the analysis has finished, we compute SIFs; choose **Cracks** → **Compute SIFs** from the FRANC3D menu. The dialog, Fig 4.15, allows you to specify the method for computing SIFs. If you use the M-integral, you can make sure that the Include Thermal Terms option is selected and that the Reference Temperature is 0 degrees by clicking the **Advanced** button. The reference temperature is set automatically when importing the initial uncracked model. For an ANSYS model, the “TREF” value is used, and for an ABAQUS model, the “*Initial Conditions, type=TEMPERATURE” is used.

The SIFs based on the M-integral are shown in Fig 4.16 and the SIFs based on Displacement Correlation (DC) are shown in Fig 4.17. It should be noted that the SIFs do not differ much, indicating that the SIFs are computed correctly. If your results differ significantly, you should ensure that the nodal temperatures and reference temperature are correct.

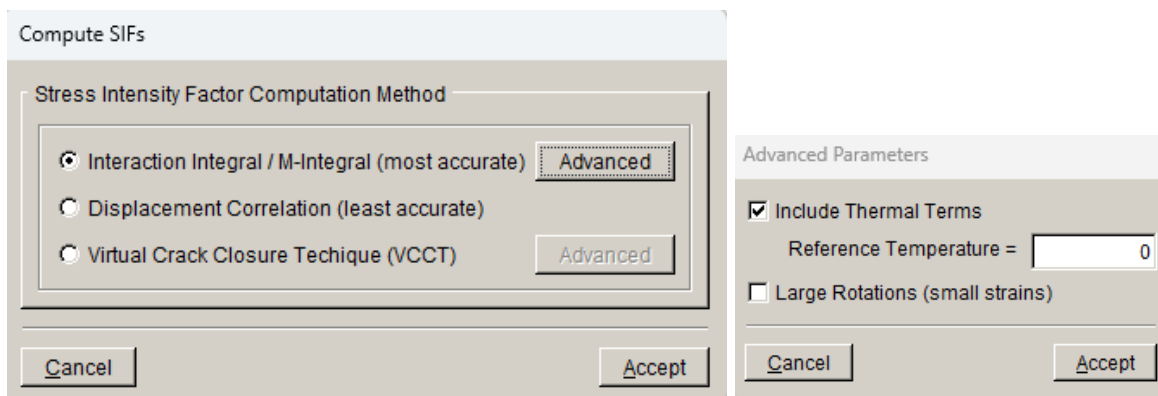


Figure 4.15 Compute SIFs dialog.

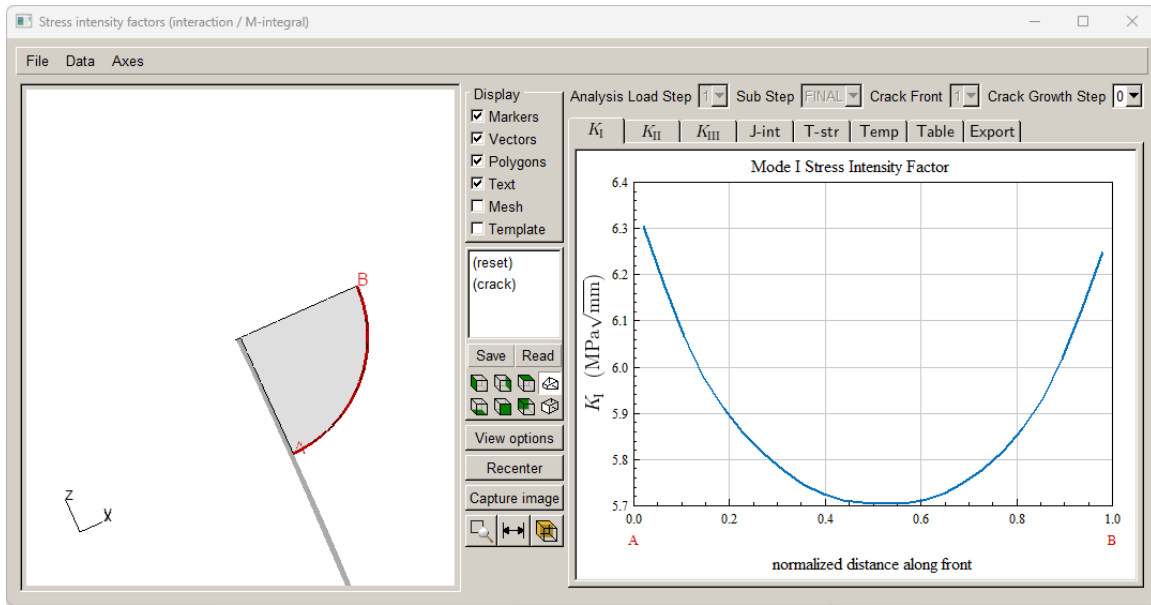


Figure 4.16 M-Integral SIFs.

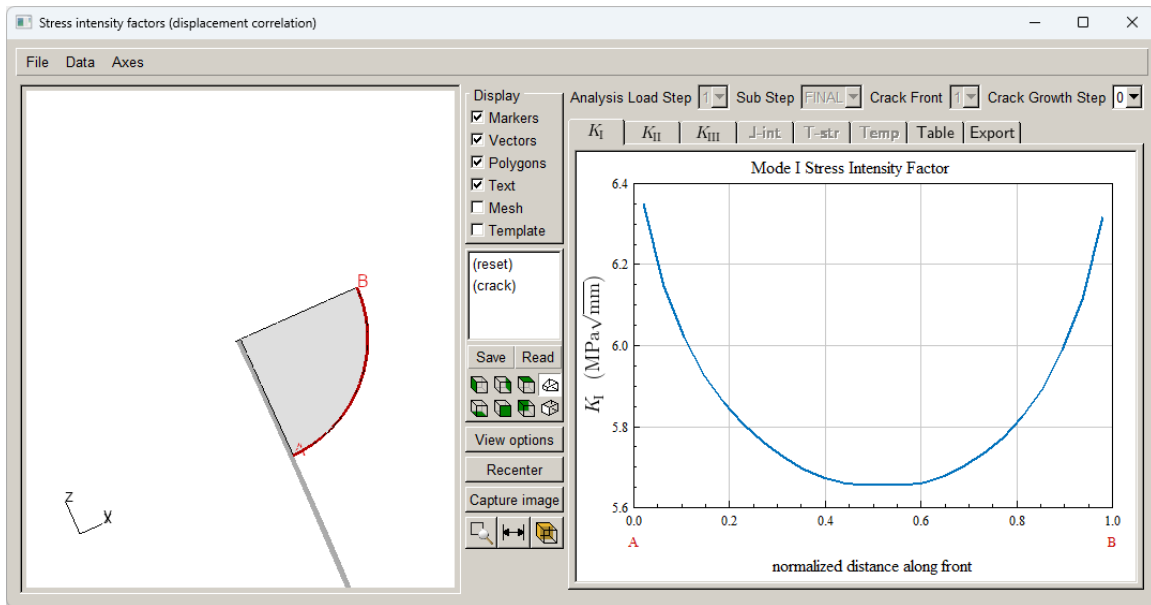


Figure 4.17 Displacement correlation SIFs.

4.6 Step 6: Crack Growth

The next step is to propagate the crack. First, recompute the SIFs with the M-integral with the correct settings (if needed). Then, choose **Cracks** → **Grow Crack** from the FRANC3D menu.

We switch to quasi-static growth, Fig 4.18, and select **Next**. We leave all the defaults in the next two panels, Fig 4.19 - 20, select the one FEM load step, and then click **Finish**.

Figure 4.18 Crack growth first wizard panel.

Figure 4.19 Crack growth second wizard panel.

Quasi-static growth parameters

Power law growth parameter

$$\Delta a_i = \Delta a_{\text{median}} (K_i / K_{\text{median}})^n$$

n:

Mixed-mode equivalent K

☒ $K^{\text{equiv}} = K_I$ ☐ $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{mII} K_{II})^2 + (\gamma_{mIII} K_{III})^2}$

γ_{II} : γ_{III} :

sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

FEM Load Steps

Step	Sub	Load mult
1		1 Edit

Figure 4.20 Crack growth third wizard panel.

The extension is set to 0.015, Fig 4.21. The template radius is set to 0.01, Fig 4.22. Select **Finish** when ready to start the process of crack growth, insertion and remeshing.

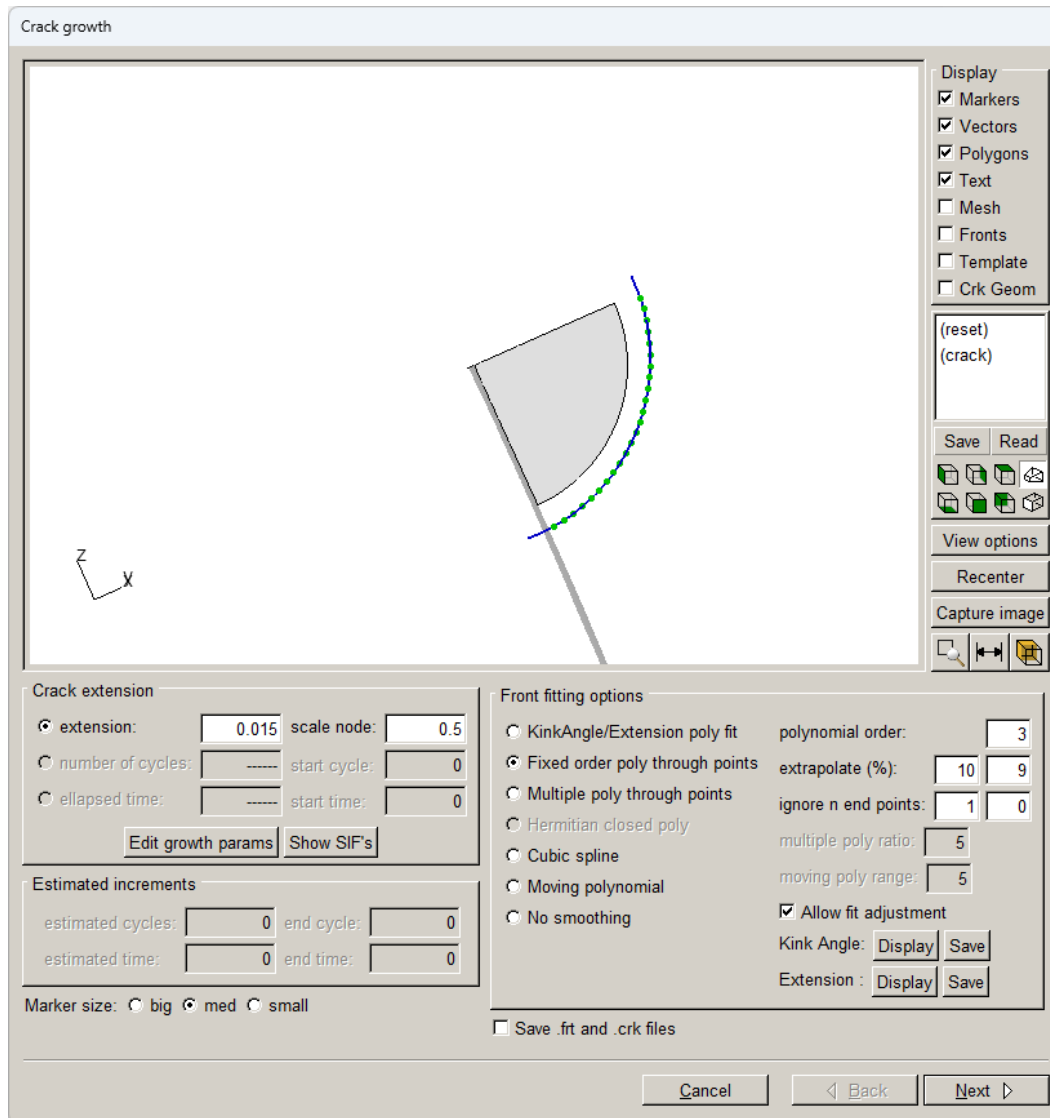


Figure 4.21 Crack growth third wizard panel.

The ends of the crack front and template should extend beyond the model surface at every step of growth; this sometimes requires small adjustments to the fitting parameters depending on the crack front and the model surface geometry; FRANC3D should do this automatically.

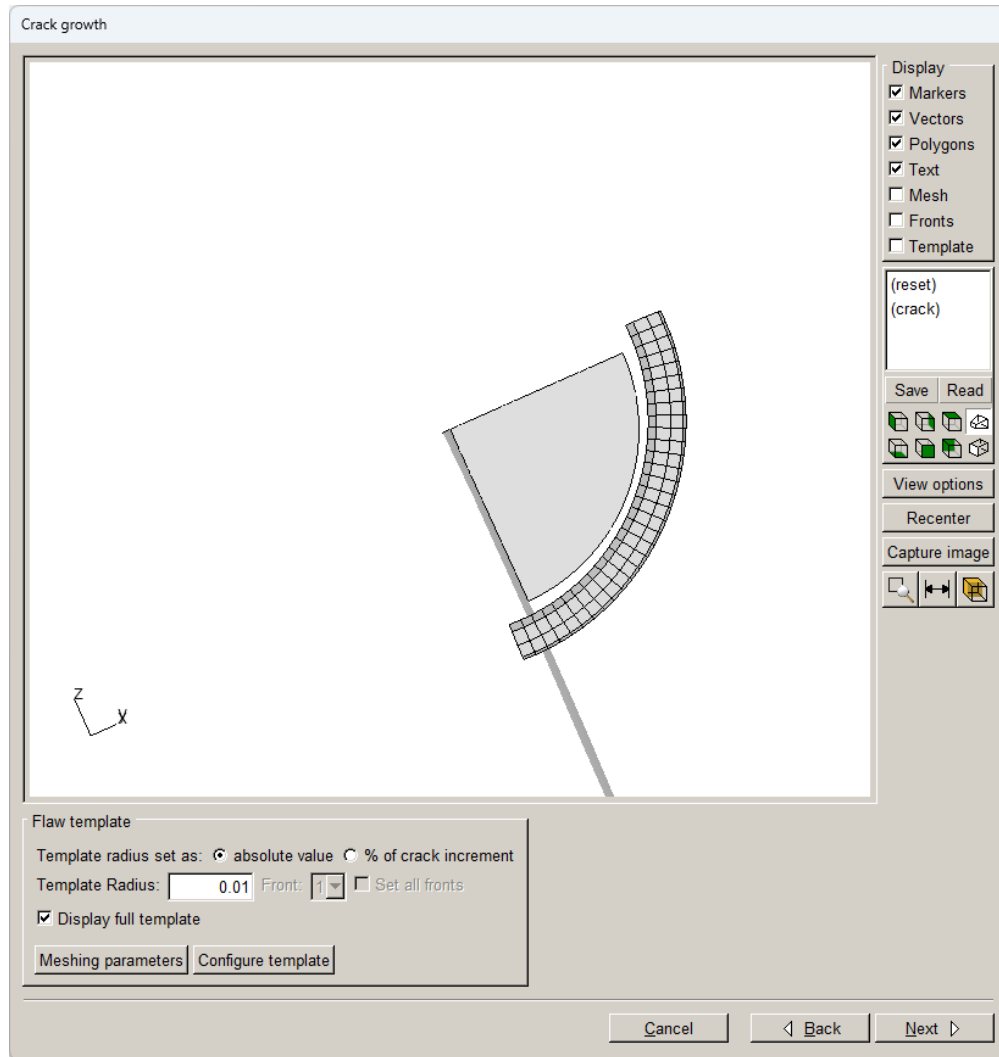


Figure 4.22 Crack growth wizard – final panel.

4.7 Step 7: Automatic Crack Growth

We can continue doing static analyses and manually growing the crack for each step, or we can do automatic crack growth. We proceed with automatic crack growth at this point. Select **Crack Growth Analysis** from the **Analysis** menu. A series of dialog panels with parameters/settings you have just seen in Step 6, when manually growing the crack, will be displayed. We specify ten crack growth steps with a constant growth increment of 0.015 and a template radius of 0.01, Fig 4.23.

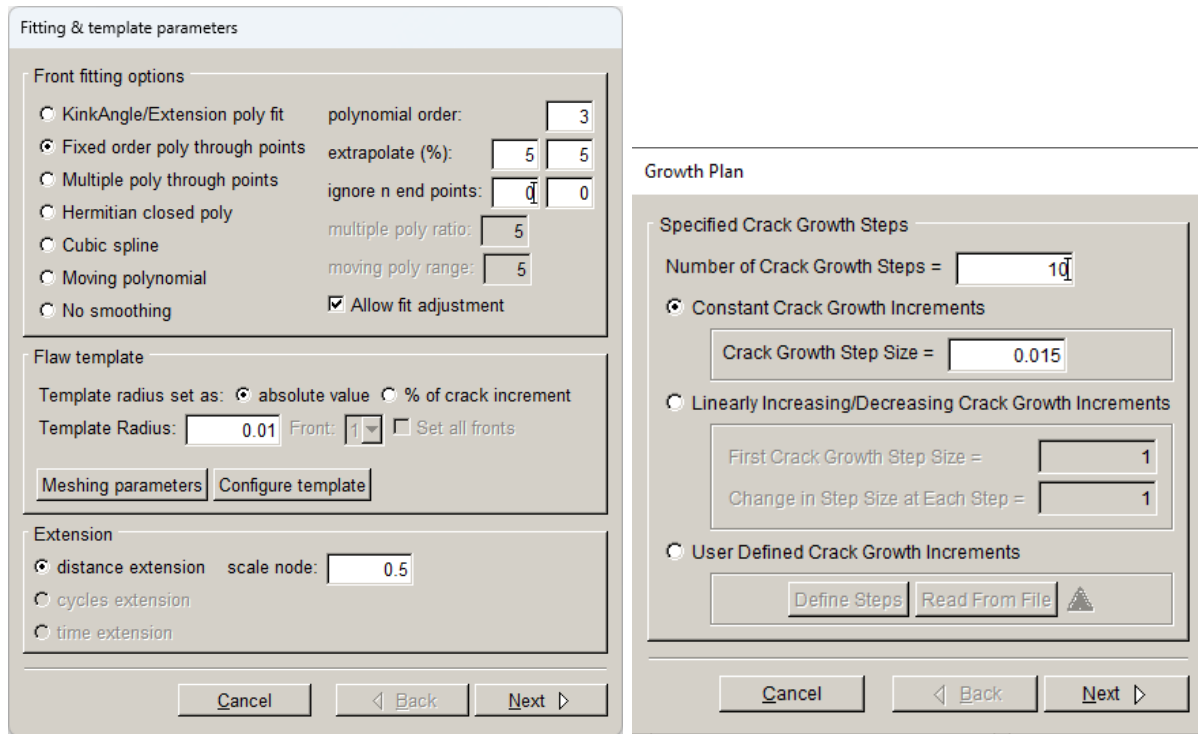


Figure 4.23 Crack growth plan for automatic crack growth.

Select **Next** to set the analysis options. Set the Current crack growth step to 1, as we have extended the crack but have not yet analyzed it, Fig 4.24; this dialog will depend on your analysis code. All the default settings should be okay; select **Finish** to start the automatic growth.

4.8 Step 8: Analysis Results

After 10 steps of crack growth using a constant increment of 0.015, the crack appears as in Fig 4.25. You can monitor the FRANC3D CMD window for any error messages as well as monitoring the analysis code log and error files. The SIF history is shown in Fig 4.26.

Crack growth analysis

ANSYS Options

File Names

Base filename: (avoid spaces and special characters)

Current crack growth step:

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact ☐ Initial strain

ANSYS command:

Figure 4.24 Crack growth step number and base file name dialog for automatic crack growth – using ANSYS.

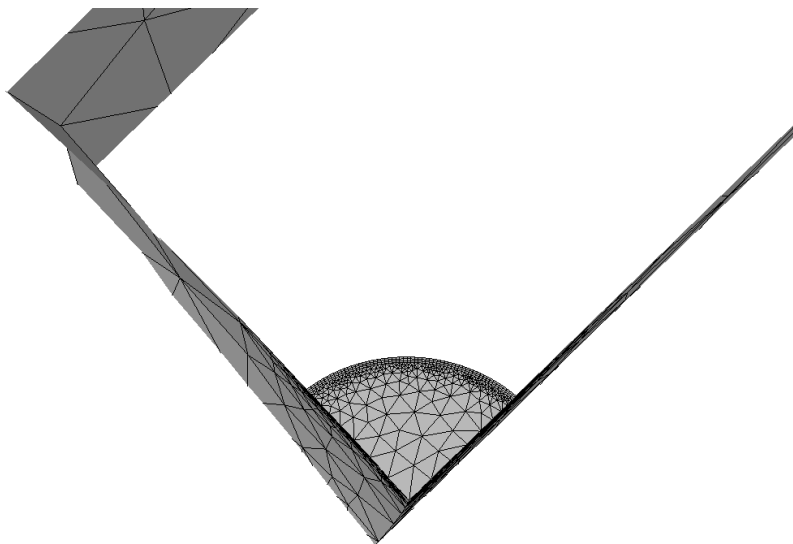


Figure 4.25 Crack configuration after 10 steps of automatic crack growth.

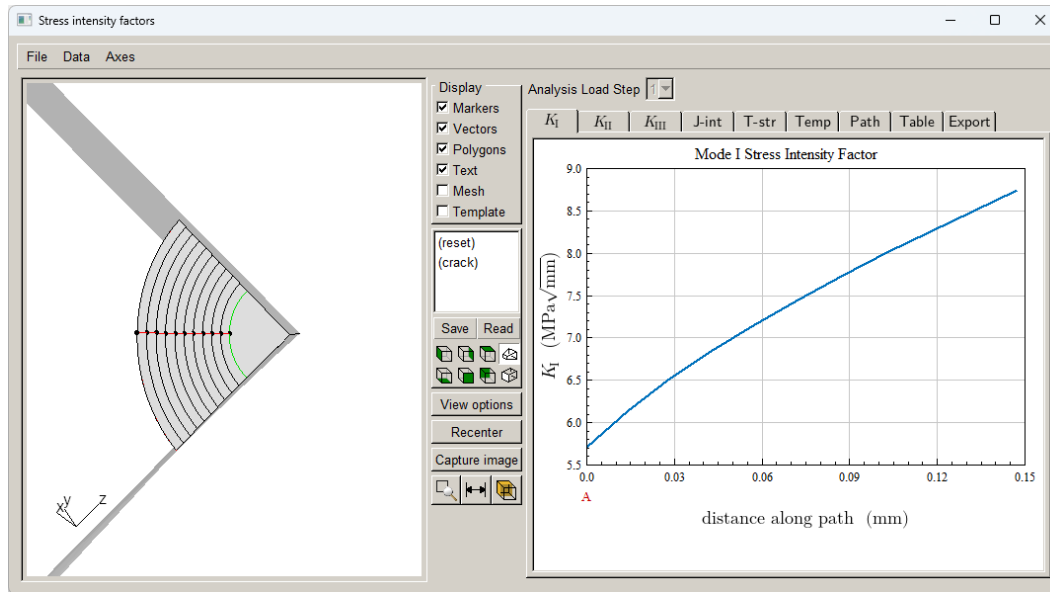


Figure 4.26 Mode I SIF history after 10 steps of automatic crack growth.

The FE analysis results can be examined to ensure that temperatures and other results are consistent with the original uncracked model results.

4.8.1 ABAQUS crack model results

We first examine the temperature contours to verify that the temperatures are mapped correctly, Fig 4.27a. The deformed shape of the crack at step #11 is shown in Fig 4.28a. You can compare the overall deformed shape with that of the uncracked model.

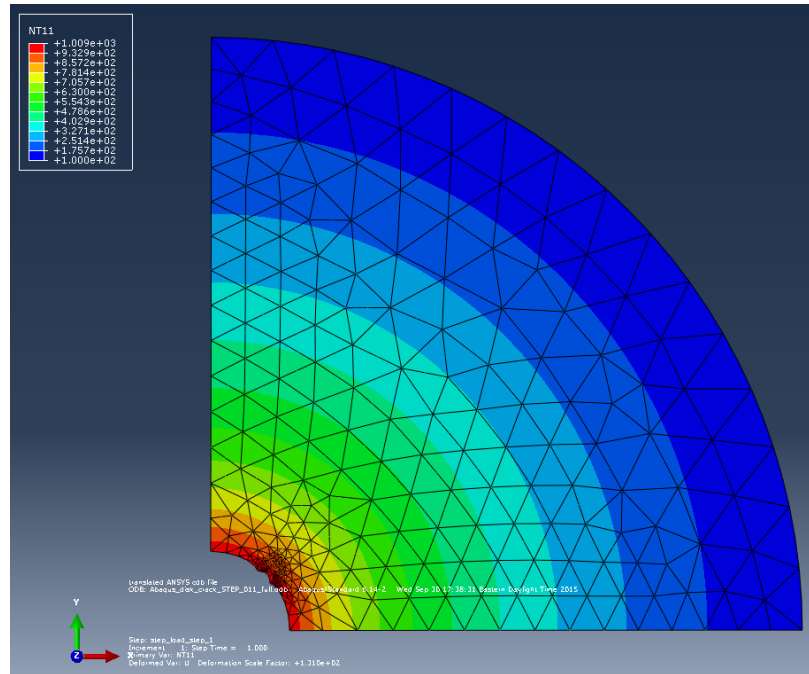


Figure 4.27a Contours of structural temperatures at crack step #11.

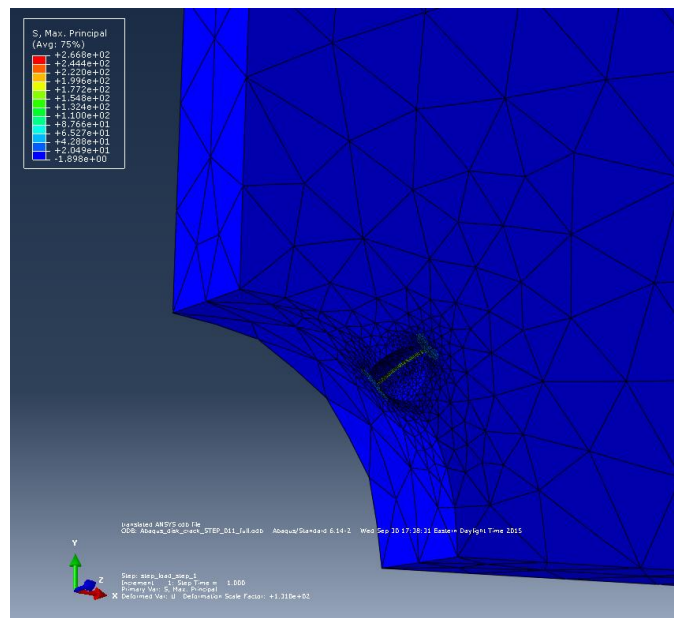


Figure 4.28a Deformed shape of crack step #11.

4.8.2 ANSYS crack model

We examine the temperature contours in ANSYS to verify that these are mapped correctly, Fig 4.27b. The deformed shape of the crack at step #11 is shown in Fig 4.28b. You can compare the overall deformed shape with that of the uncracked model.

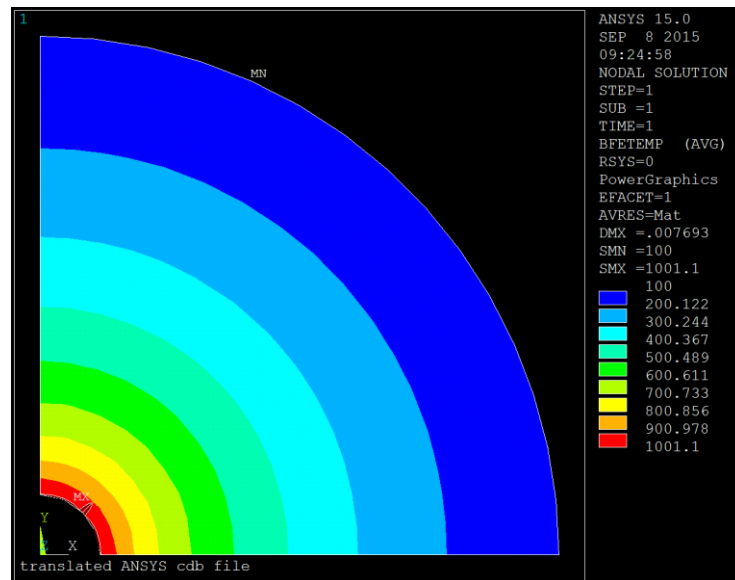


Figure 4.27b Contours of structural temperatures at crack step #11.

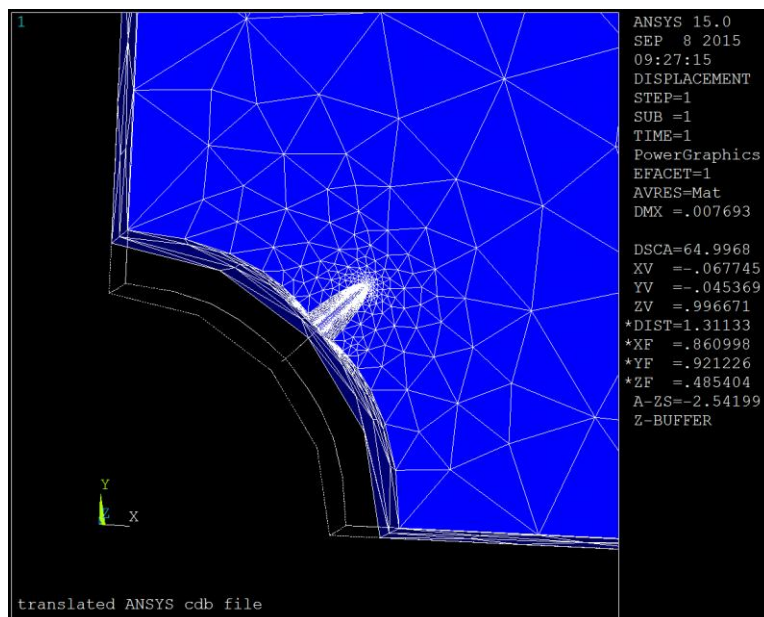


Figure 4.28b Deformed shape of crack step #11.

4.8.3 NASTRAN crack model

We examine the temperature contours in NASTRAN to verify that these are mapped correctly, Fig 4.27c. The deformed shape of the crack at step #11 is shown in Fig 4.28c.

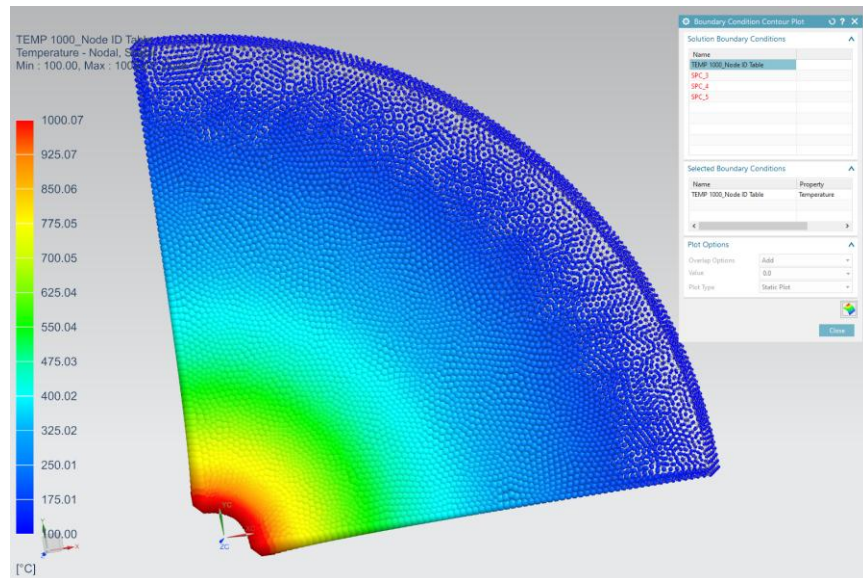


Figure 4.27c Contours of structural temperatures at crack step #11.

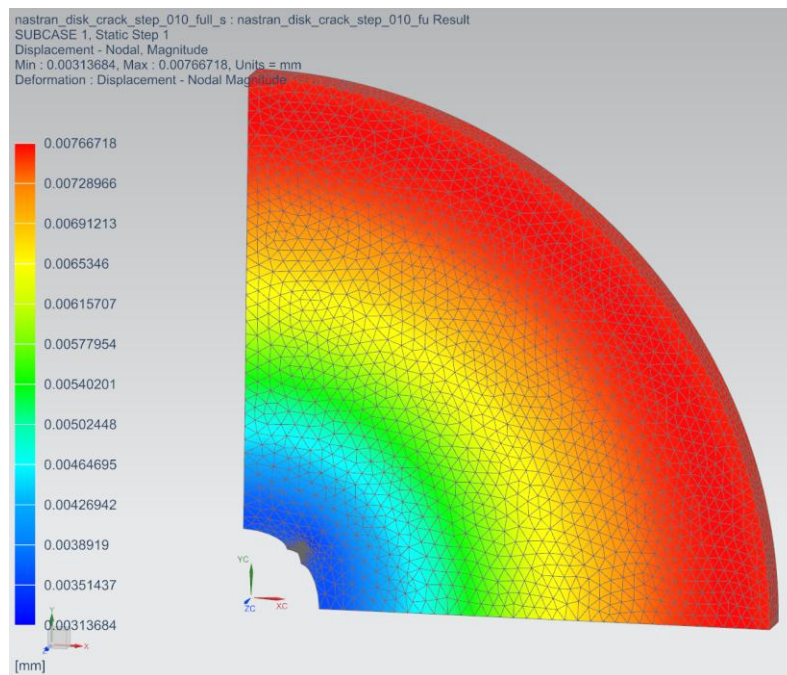


Figure 4.28c Deformed shape of crack step #1.

4.9 Step 9: Fatigue Cycles

We can continue to grow this crack and/or we can compute fatigue cycles at this stage.

We compute fatigue life or cycles based on the current crack growth. From the FRANC3D menu, select **Fatigue** → **Fatigue Life Predictions**. If the SIFs have not been computed yet, the Compute SIFs dialog will be displayed; use the M-integral and select **Finish**. The Fatigue Life dialog, Fig 4.29, is displayed.

In this dialog, the crack fronts are displayed on the left and the window on the right side should be blank (assuming that lifing parameters have not been defined previously). You must **Set** (or **Read**) **Parameters**. Selecting **Set Parameters** displays the dialog shown in Fig 4.30.

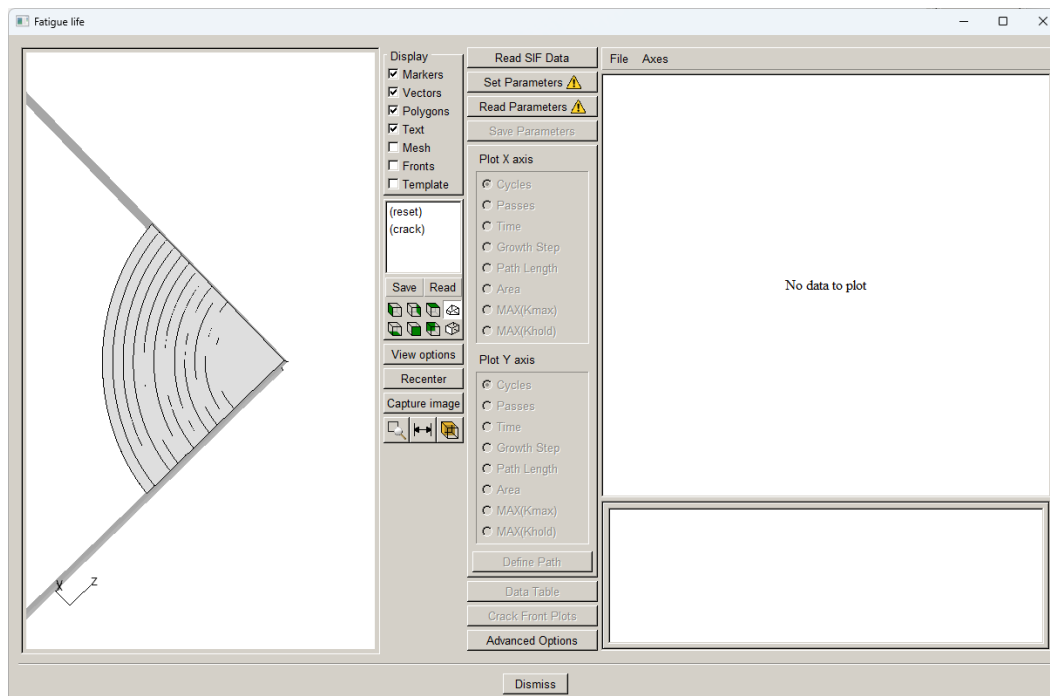


Figure 4.29 Fatigue Life dialog

Growth Parameters

Units used in FE model
 stress: MPa length: mm temp: C time: sec Change

Crack growth load schedule

New schedule Read from file Wizard View/Edit Save to file

Crack growth rate model

New model Read from file View/Edit da/dN View/Edit da/dt Save to file

Mixed-mode equivalent K

☒ $K^{\text{equiv}} = K_I$
☐ $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{II} K_{II})^2 + (\gamma_{III} K_{III})^2}$
☐ $K^{\text{equiv}} = K_{\text{RSS}}$
 γ_{II} γ_{III}
 sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective delta K

☒ $\Delta K_{\text{eff}} = K_{\text{max}} - K_{\text{min}}$
☐ $\Delta K_{\text{eff}} = K_{\text{max}} - \max(K_{\text{min}}, 0)$
☐ Use load interaction model Set/Edit parameters

Integration options

☒ Accelerated counting
 ☐ Constant K for time integration

Dynamic pairing metric

☒ $\Delta K_{\text{eff}}^{\text{equiv}}$
☐ da/dN
☐ Simplified HCF/LCF pairing

Cancel Accept

Figure 4.30 Fatigue Life parameters

The FEM model units are mm and MPa and Celsius; select the **Change** button to display the dialog in Fig 4.31. Set the units and select **Accept**.

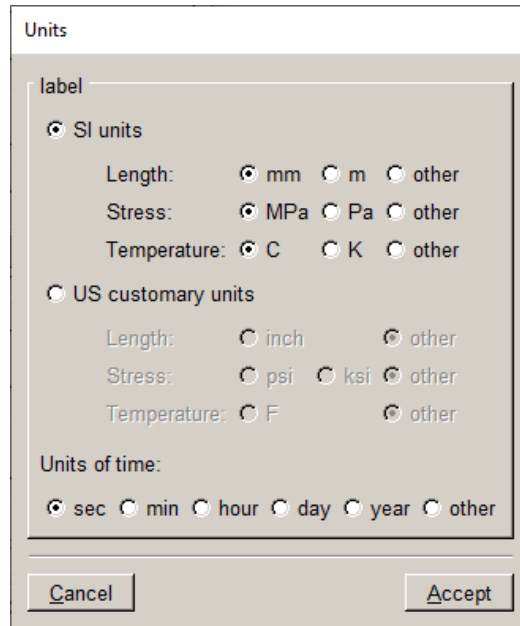


Figure 4.31 FEM units dialog

The crack growth load schedule is defined next. Select the **New Schedule** button (see Fig 4.30) to display the dialog in Fig 4.32. Select the Schedule and then select the **Add** button; see right side image of Fig 4.32. This will display the dialog shown in Fig 4.33; we use a simple cyclic load schedule where the applied load produces K_{max} , while K_{min} is zero. Select **Accept** to return to the Load Schedule, Fig 4.34.

The Repeat count is set to FOREVER, Fig 4.34. For the temperature tab, Temperature dependent crack growth is selected and Use temperature from K_{max} is chosen. Select **Accept** to finish the load schedule.

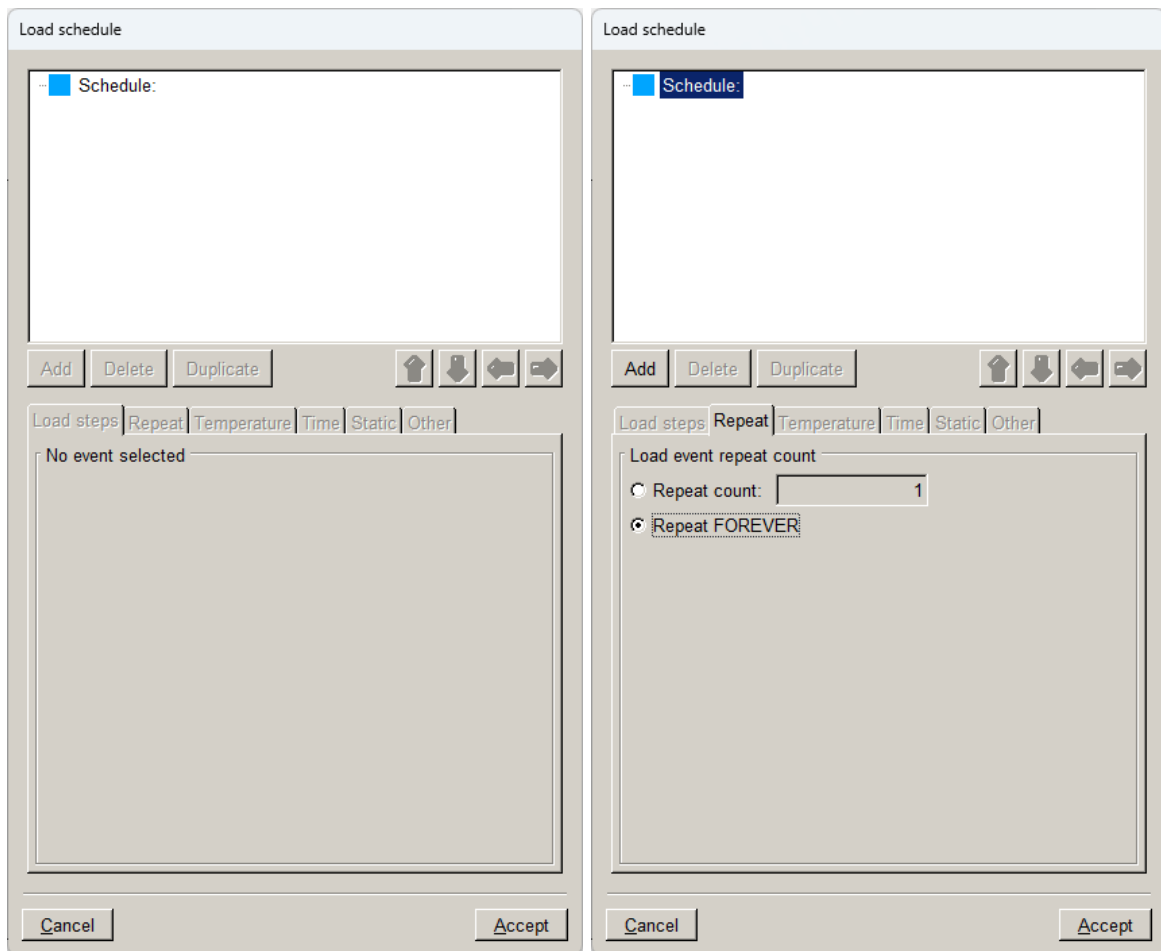


Figure 4.32 New Load Schedule dialog

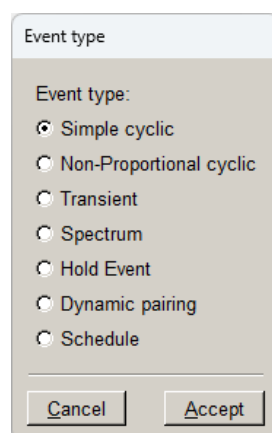


Figure 4.33 Event type dialog

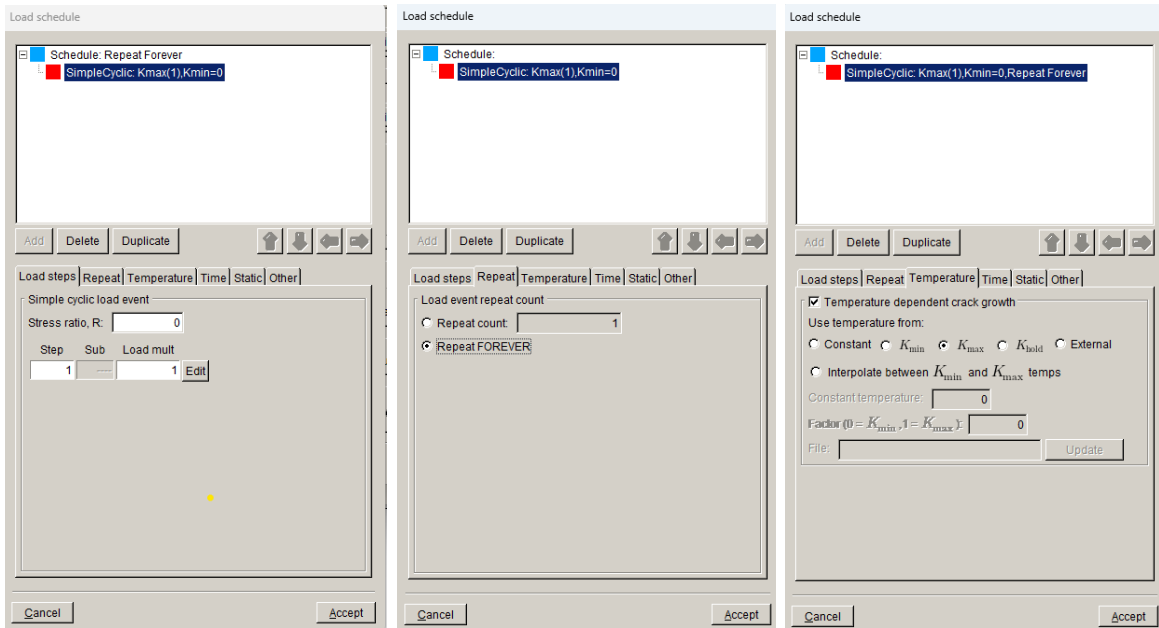


Figure 4.34 Simple cyclic load schedule repeated forever with temperature from Kmax

The crack growth rate model is defined next. Select the **New Model** button (see Fig 4.30) to display the dialog in Fig 4.35. Choose the cyclic loading growth rate model and select **Next** to display the dialog in Fig 4.36.

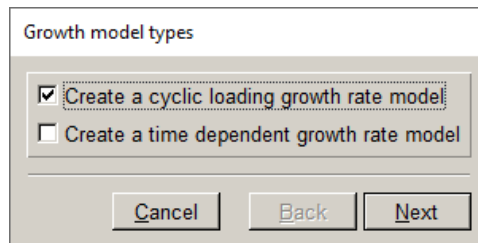


Figure 4.35 Growth model type dialog

We use a simple Paris growth model, which is the default; select **Next** to display the dialog in Fig 4.37. The growth rate model will be temperature dependent. Select **Next** to display the Paris growth model dialog, Fig 4.38.

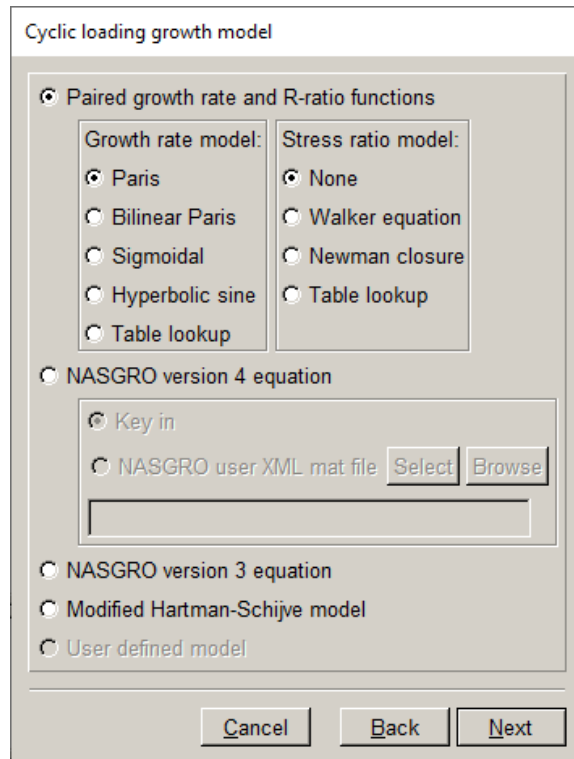


Figure 4.36 Cyclic loading growth model dialog

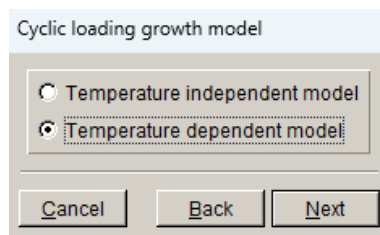


Figure 4.37 Temperature dependent growth rate model

Set the units for the Paris growth model. Select the **Change** button to display the Units dialog, and set the units the same as the FEM units that we set earlier, which were mm, MPa and C. Set the Number of temperatures to 2. In the fields for temperature, C, n, DKth and Kc, enter the appropriate values, Fig 4.38. Select **Next** to return the main dialog (see Fig 4.30). The other options in Fig 4.30 are left at their default values. Select **Accept** to return to the Fatigue Life dialog, Fig 4.39.

Cyclic loading growth model

Paris growth model:
 $da / dN = C \Delta K^n$

Model label (optional):

Description (optional):

Units:

Number of temperatures: ☒ Interpolate ☐ Closest ☐ Next highest ☐ Extrapolate ☒ Capped

	Temperature	C	n	DKth	Kc
1	100	1e-10	3	1	100
2	1000	1.5e-10	3.5	1	115

Figure 4.38 Paris temperature dependent growth rate model dialog.

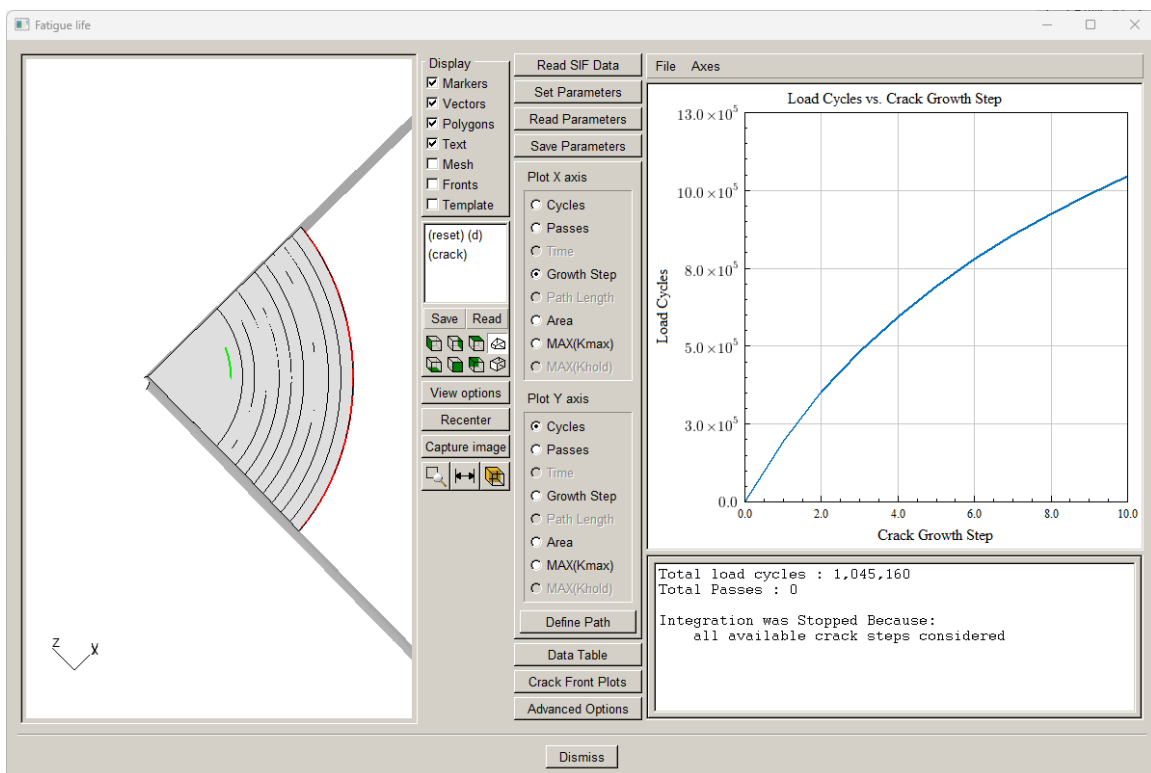


Figure 4.39 Fatigue Life dialog showing Cycles vs Crack Growth Step

4.10 Apply Crack Face Traction (CFT)

Tutorial #2 included steps to apply crack face pressure or traction (CFT). We examine additional temperature settings for CFTs in this subsection.

4.10.1 Unloaded FE Model

First, we want to verify that the uncracked stress (saved during Step 1), when applied as CFT, provides the same SIFs as the actual applied boundary conditions. Edit the uncracked FE file and remove all the rotation and temperature boundary conditions (if there were pressures or forces, we would remove those also). Only the displacement boundary conditions are retained; if the displacement values were non-zero, FRANC3D would set those to zero for the CFT load step. Save the FE file with a different name when finished editing.

Import the ‘unloaded’ FE model into FRANC3D and insert the crack as we did in Steps 2 and 3. Once the crack is inserted and meshed, we can apply the CFT.

Select the **Loads** → **Crack Face Pressure/Traction** menu option. In the CFT dialog, click the **Add** button, Fig 4.40; switch to the Residual Stress Defined on a Mesh, Fig 4.40. Choose either the *Abaqus_Disk.inp* and *Abaqus_Disk.rpt* files, Fig 4.41, or corresponding ANSYS or NASTRAN files, and then click **Next**. Click **Accept** to close the CFT dialog.

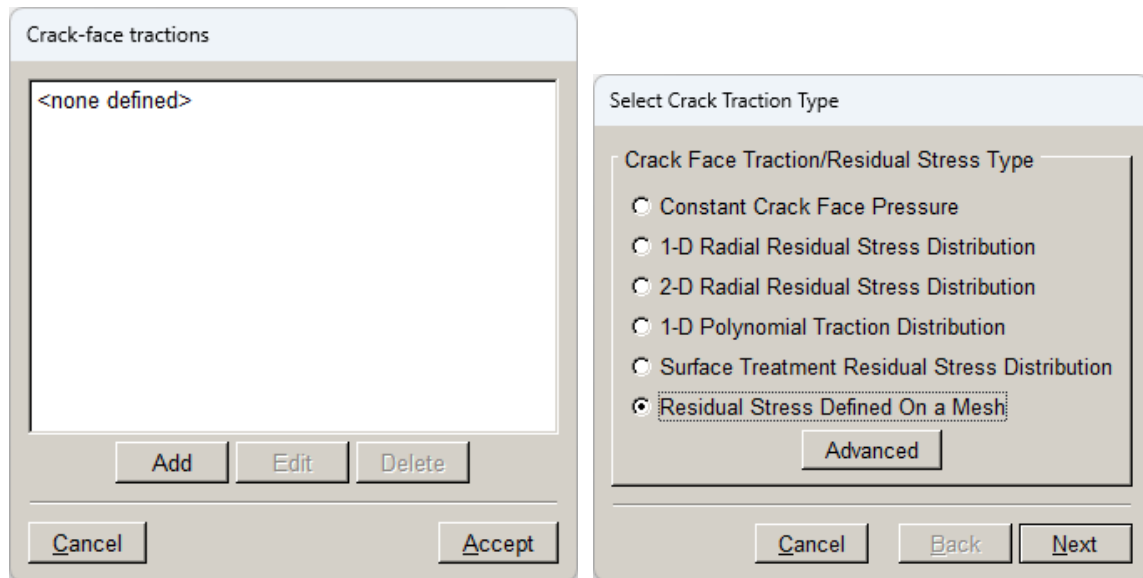


Figure 4.40 Add CFT dialog panels.

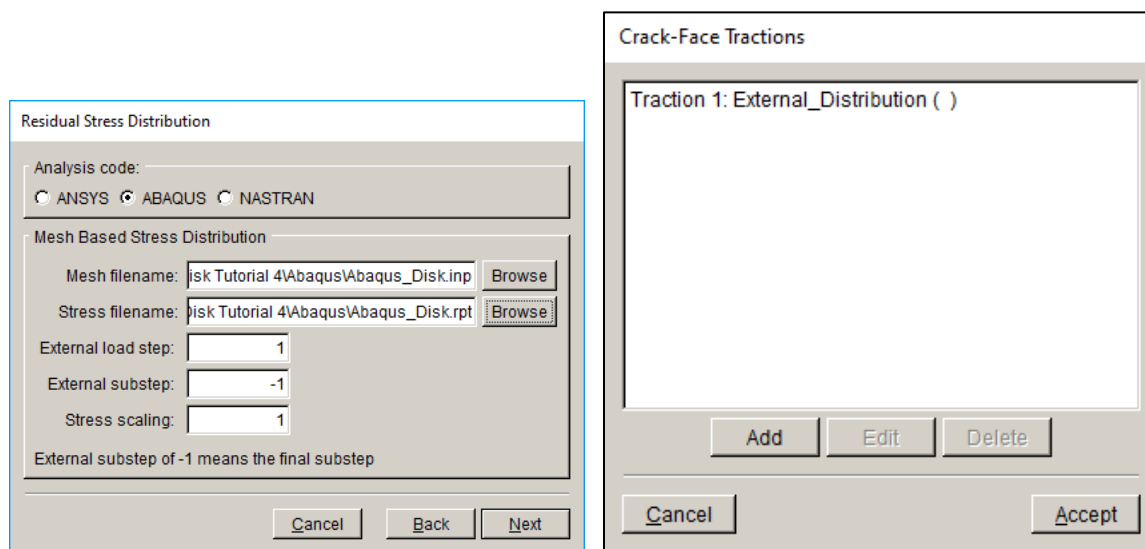


Figure 4.41 Residual Stress CFT dialog panels.

Run the static crack analysis and then compute SIFs using the M-integral, Fig 4.42. The SIFs are displayed in Fig 4.43. In this case, the CFT load is applied in a separate load step from the base model load step. There are no thermal terms as we deleted the temperature from the uncracked FE model.

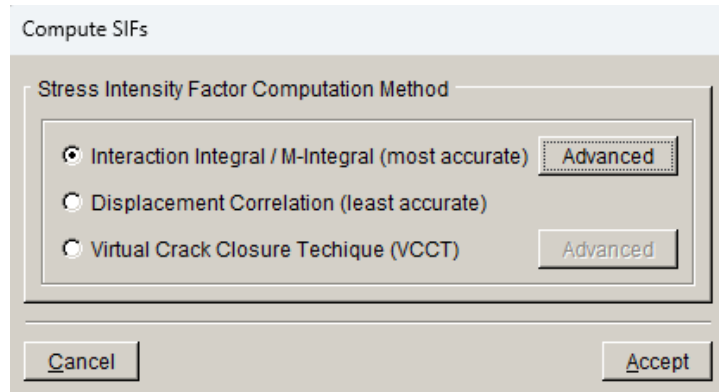


Figure 4.42 Compute SIFs dialog.

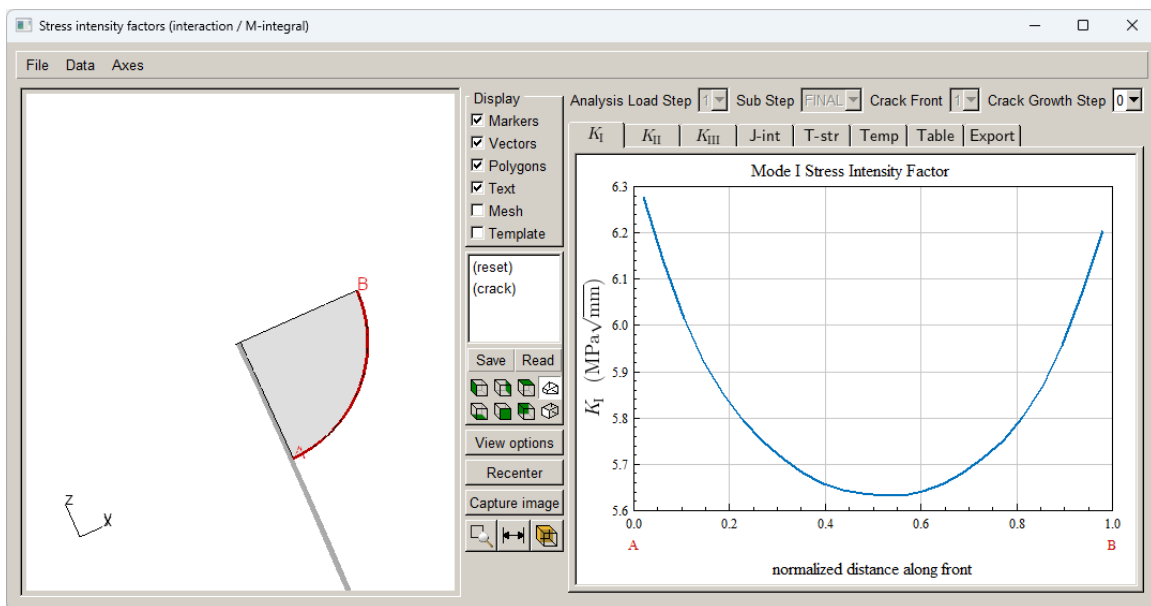


Figure 4.43 M-Integral based SIFs for CFT.

To verify the M-integral terms are computed correctly, we can plot the displacement correlation SIFs. These values are usually within a few percent of the M-integral values and are based only on displacements. Fig 4.44 shows the DC SIFs, and the curve is almost identical to that in Fig 4.43.

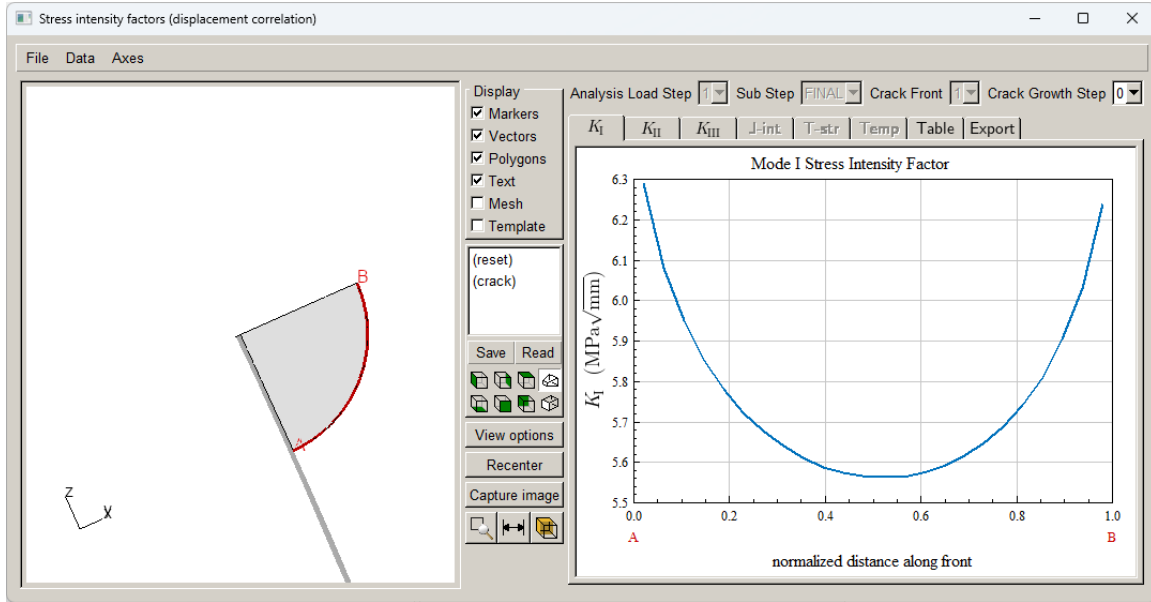


Figure 4.44 DC based SIFs for CFT.

4.10.2 Original FE Model

In this section, the CFT is added to the original model. Start with the original uncracked (with loads) FE file and repeat Steps 2 and 3 to import the model and insert the crack.

Once the crack is inserted, add the CFT as shown in Figs 4.40-41. An additional dialog is presented after specifying the mesh and stress files, Fig 4.45. We set the temperature as constant and 0, which is the reference temperature. If the Allow thermal expansion option is on, turn it off as the CFT will already include the effects of the thermal stress/strain.

Run a static crack analysis and compute SIFs. The base load step SIFs are shown in Fig 4.46, and the CFT load step SIFs are shown in Fig 4.47. The SIFs are almost identical, which is what we expect.

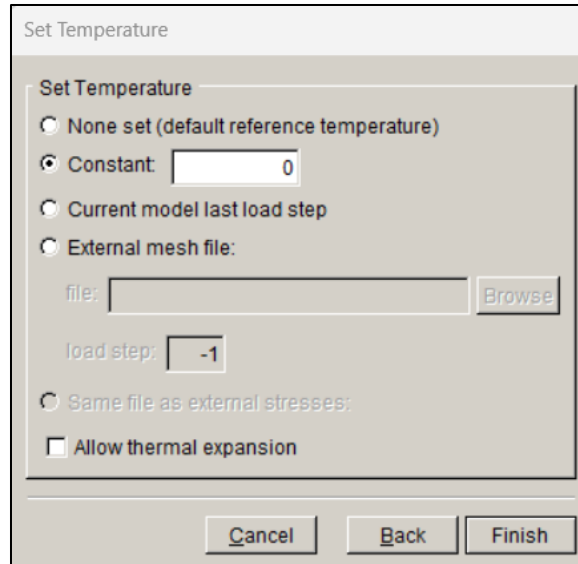


Figure 4.45 CFT temperature setting dialog.

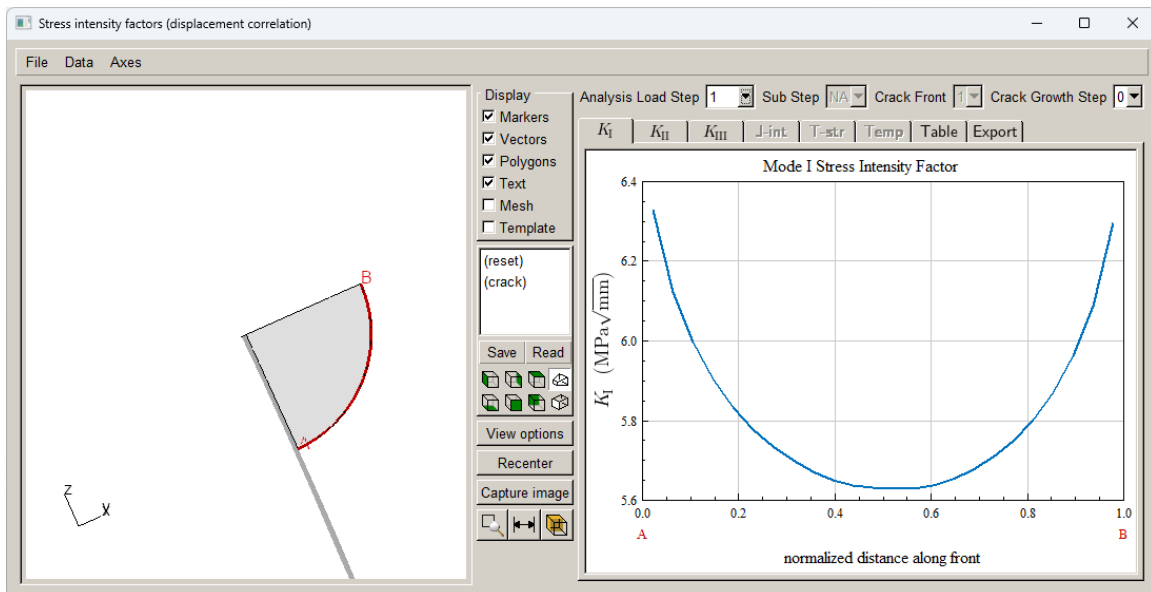


Figure 4.46 M-integral based SIFs for base load step.

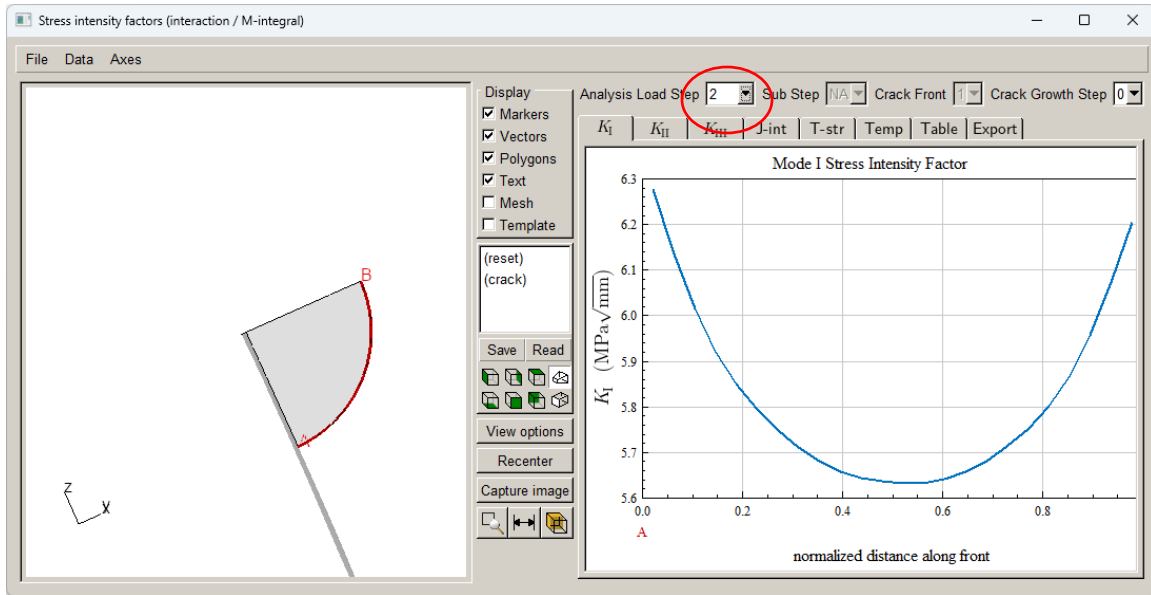


Figure 4.47 M-integral based SIFs for CFT load step.

Note that the CFT SIFs are computed based on temperatures that are set to 0.0, which is the reference temperature for the analysis. The CFT load step has temperature set to 0.0, and the temperature dependent material properties at this temperature will be used.

If temperatures are different from the reference temperature, then the user must determine whether the Allow thermal expansion should be on or off.

Section 12 of the User's Guide provides additional discussion of CFTs and the effects of temperature.

5.0 Tutorial 5: Multiple Cracks in a Plate

In Tutorial 5, we simulate multiple cracks using a simple plate model. Sections 5.1-5.7 describe the process of inserting and growing multiple cracks in a single region.

Section 5.8.1 describes potential issues you might encounter when growing a crack around a geometric corner of the plate. Section 5.8.2 describes how to grow a crack through the back surface of the plate to create a “through” crack.

Section 5.9 describes how to create multiple submodel portions for multiple cracks.

5.1 Step 1: Create the Plate Model

The plate model is created using each of the analysis codes; proceed to the appropriate subsection.

5.1.1 ABAQUS Plate

The plate dimensions are defined as: 20 x 50 x 5. The bottom surface is fixed in the y-direction, and the top surface has a traction of 10. Additional constraints on the bottom left edge prevent motion in the x-direction, and the node at the origin is fixed in the z-direction also, Fig 5.1a.

The material properties are shown in Fig 5.2a. The model is meshed with 20-noded brick elements. Once the model has been defined and analyzed in ABAQUS, the model can be saved to a *.inp* file; we call it *Abaqus_plate.inp*.

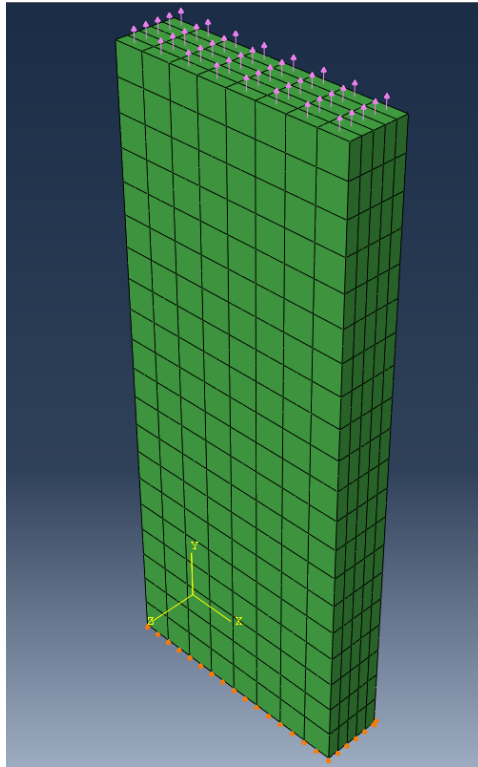


Figure 5.1a ABAQUS plate model.

Edit Material

Name: MATERIAL-PLATE

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic

Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

No compression

No tension

Data

	Young's Modulus	Poisson's Ratio
1	10000	0.3

OK Cancel

Figure 5.2a ABAQUS plate model material properties.

5.1.2 ANSYS Plate

The plate dimensions are defined as: 20 x 50 x 5. The bottom surface is fixed in the y-direction, and the top surface has a traction of 10. Additional constraints on the bottom left edge prevent motion in the x-direction, and the node at the origin is also fixed in the z-direction, Fig 5.1b.

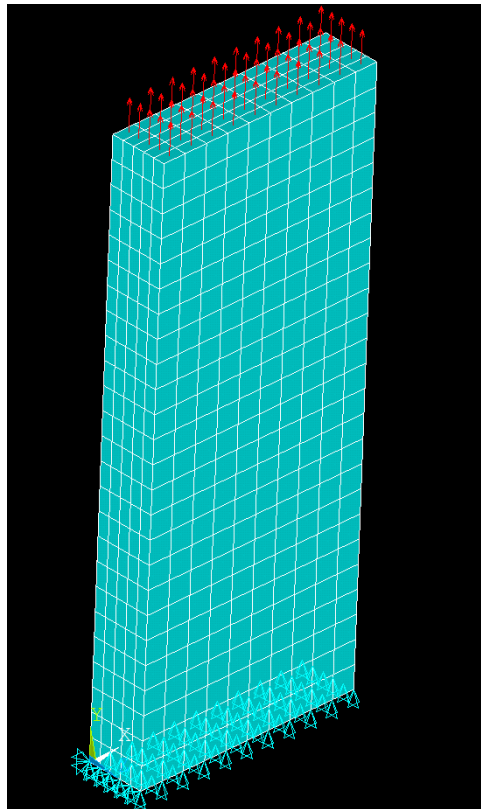


Figure 5.1b ANSYS plate model.

The material properties are shown in Fig 5.2b. The model is meshed with 20-noded brick elements. Once the model has been defined and analyzed in ANSYS, the model can be archived to a *.cdb* file; we call it *Ansys_plate.cdb*.

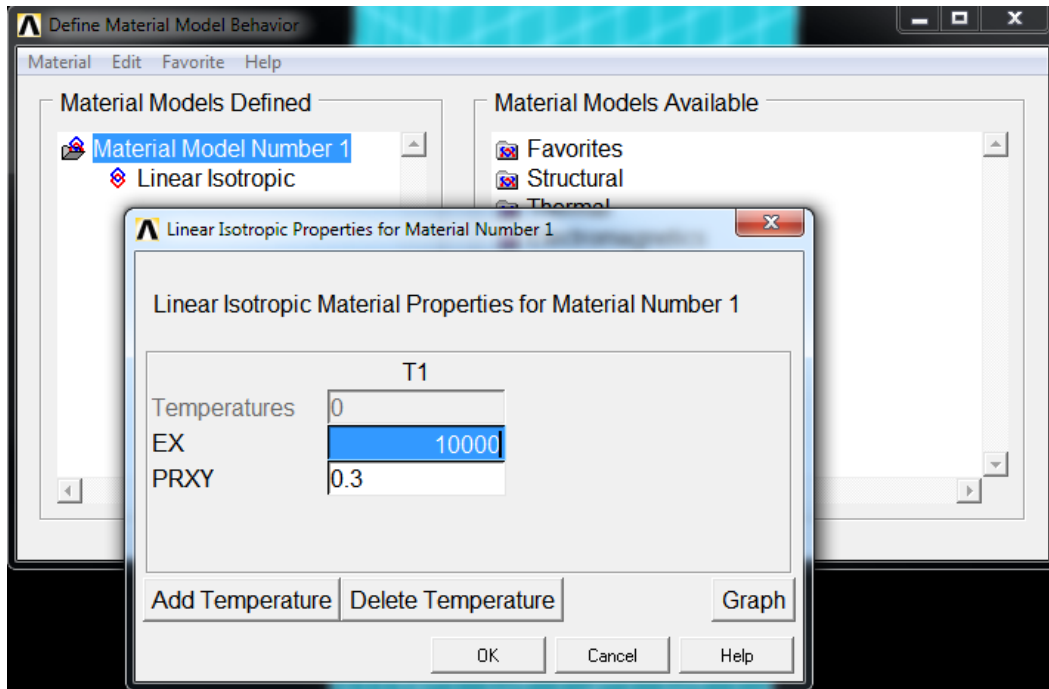


Figure 5.2b ANSYS plate model material properties.

5.1.3 NASTRAN Plate

The plate dimensions are defined as: 20 x 50 x 5. The bottom surface is fixed in the y-direction, and the top surface has a traction of 10. Additional constraints on the bottom left edge prevent motion in the x-direction, and the node at the origin is also fixed in the z-direction, Fig 5.1c.

The material properties are shown in Fig 5.2c. The model is meshed with 10-noded tetrahedral elements. Once the model has been defined and analyzed in NASTRAN, the model is saved to a *.nas* or *.bdf* file; we call it *Nastran_plate.nas*.

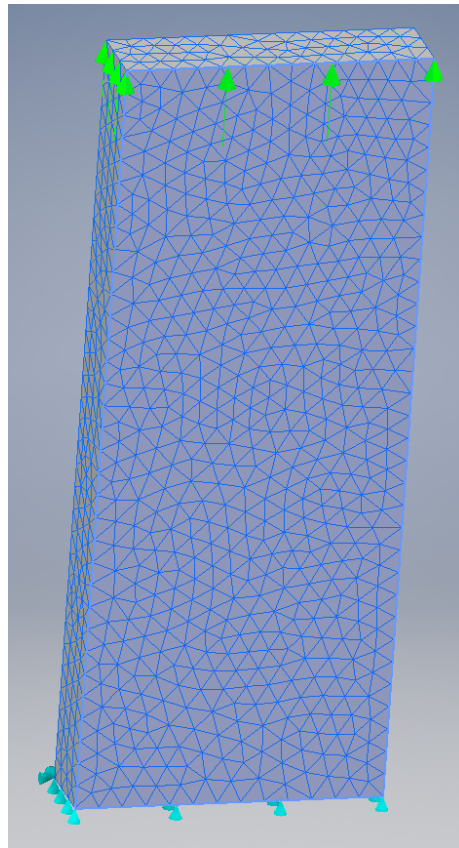


Figure 5.1c NASTRAN plate model.

Material

Select Material

Name: Material 1

ID: 1

Type: Isotropic

Sub Type: Neo-Hookean

Physical Properties:

Physical Property 1

Save New Material

Analysis Specific Data

Nonlinear

Fatigue

PPFA

General

ρ

GE 0

T_{ref}

Structural

E 10000

G

ν 0.3

α

Allowables

S_T

S_c

S_s

S_y

Failure Theory

None

Thermal

C 5e+8

K 16.3

OK Cancel

Figure 5.2c NASTRAN plate model material properties.

5.2 Step 2: Import Plate Model into FRANC3D

Once the plate model is created, proceed with the crack growth simulation. From the FRANC3D menu, select **File** → **Import**. We choose to divide the plate into local and global portions, Fig 5.3 – left side. Select **Next** and then choose the appropriate plate model, Fig 5.3 – right side.

We will extract the mid-portion of the plate using the rectangular Rubberband Box in the submodeling tool, Fig 5.4. Select **Crop** and **Next** in this dialog to extract the portion of the model selected. Save the local and global model files using the default names. There are no boundary conditions on this middle portion, however, there might be node sets, Fig 5.5; just select **Next** in the Select Retained BC Surfaces dialog. Set the FE units; SI units of mm and MPa are okay. Select **Finish**.

The model is then displayed in the FRANC3D main window, and the cut-surface mesh facets are retained, Fig 5.6.

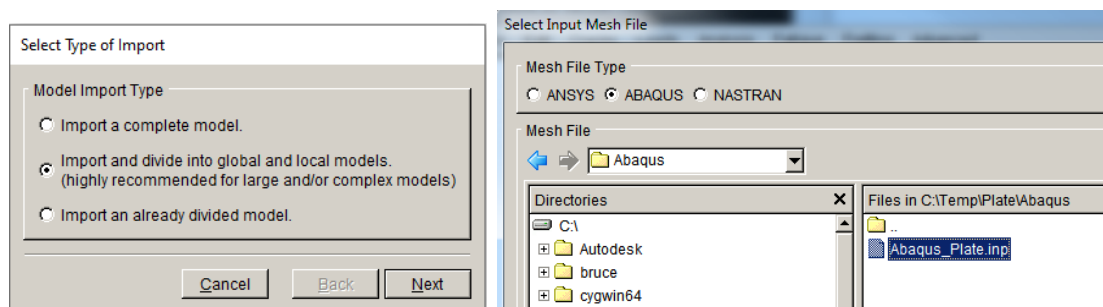


Figure 5.3 FRANC3D model import of plate model – ABAQUS file selected.

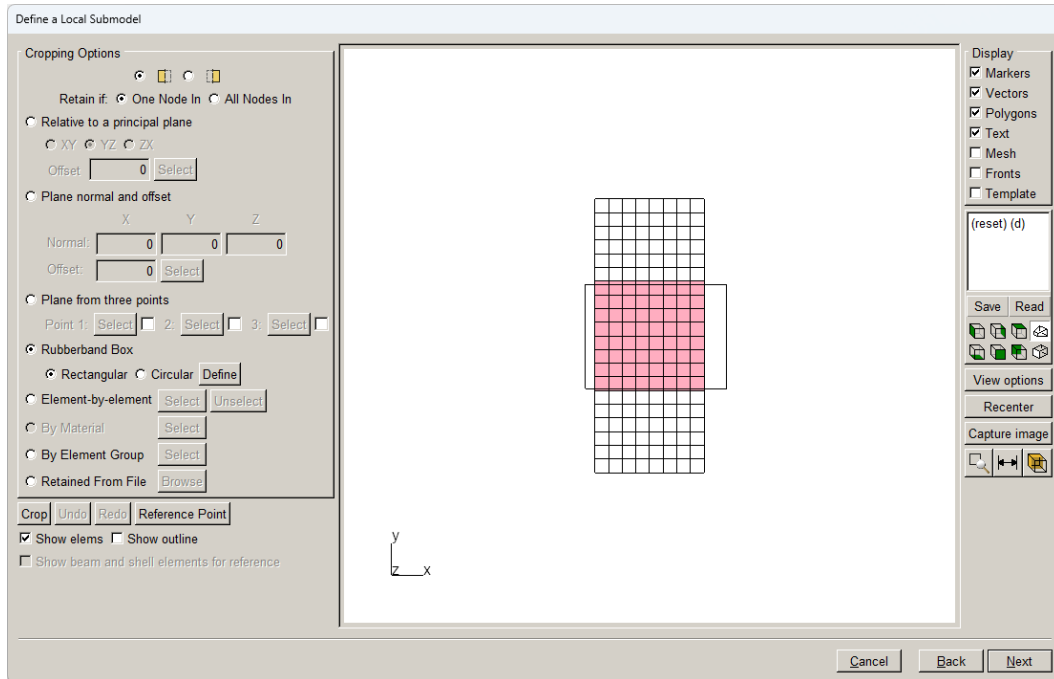


Figure 5.4 FRANC3D submodel extraction for plate model.

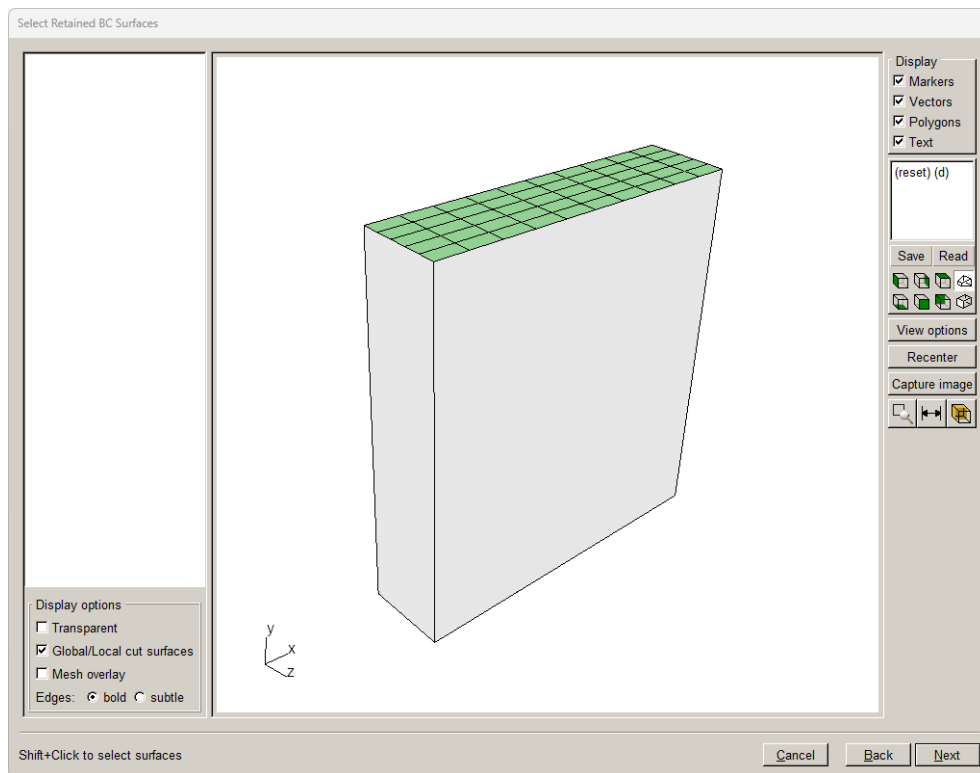


Figure 5.5 FRANC3D Select Retained Surfaces.

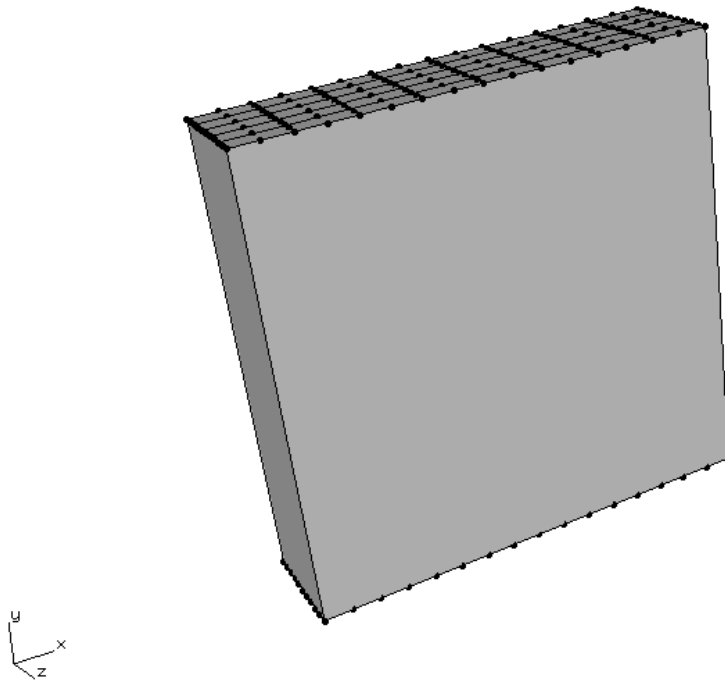


Figure 5.6 Local plate model with cut-surface mesh facets retained.

5.3 Step 3: Insert Multiple Cracks

From the FRANC3D menu, select **Cracks** → **Multiple Flaw Insert**, Fig 5.7. The dialog shown in Fig 5.8 is displayed. Use the **Add** button to define each crack that will be added.

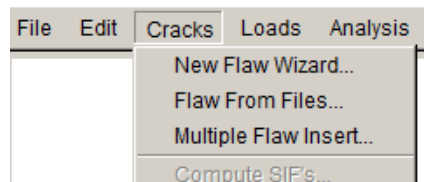


Figure 5.7 Multiple Flaw Insert menu selected.

Note that one could also define single cracks using the **New Flaw Wizard** menu, save each crack to a file without adding it to the model, and then use the **Flaw From Files** menu option to add the multiple *.crk* files.

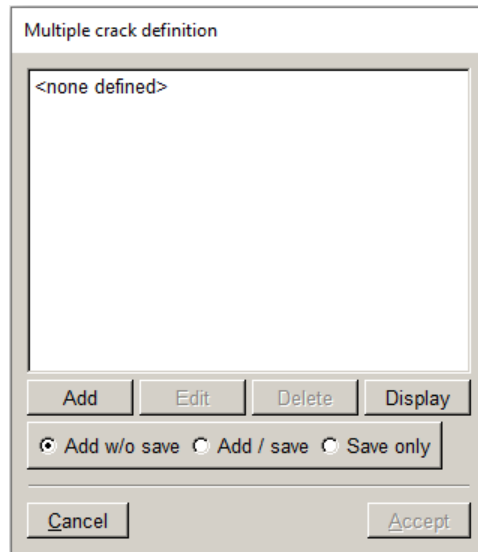


Figure 5.8 Multiple crack definition dialog.

After selecting **Add**, the usual flaw insertion wizard panels are displayed. Figs 5.9 – 5.12 provide the parameters for the first crack to be added, which is a center-through crack. This crack has two fronts. The crack geometry must be defined such that intersections between the crack and model can be computed; we set $g=5.2$ as the plate is 5 units thick.

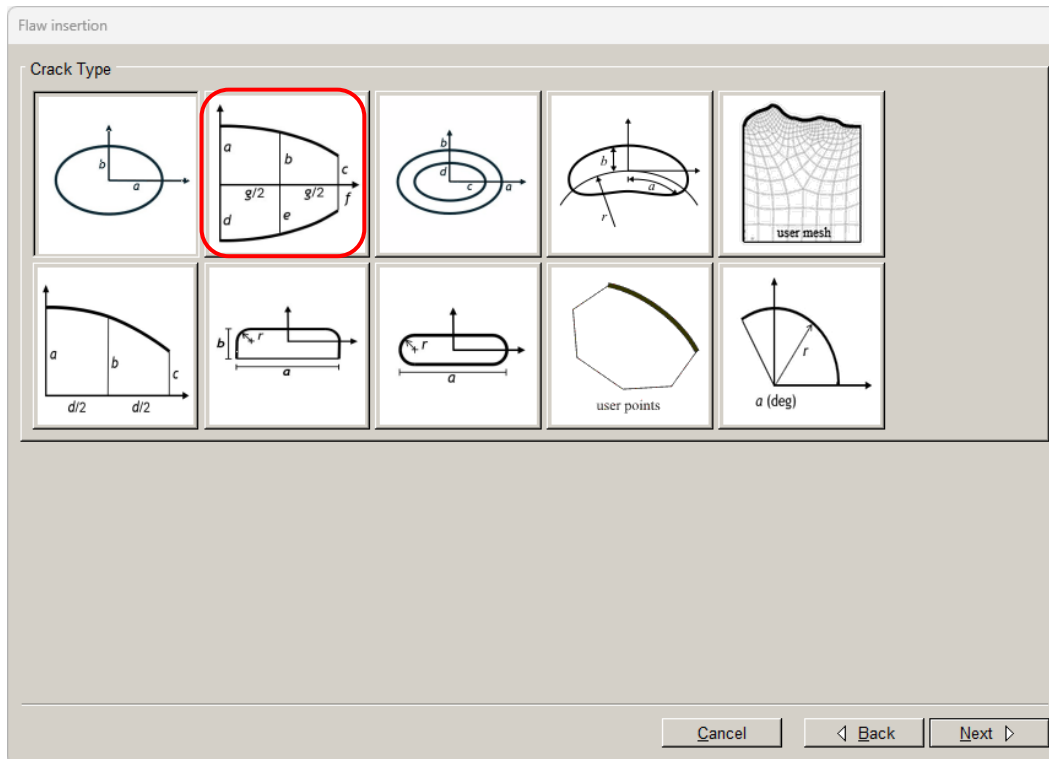


Figure 5.9 Flaw definition – middle through crack selected.

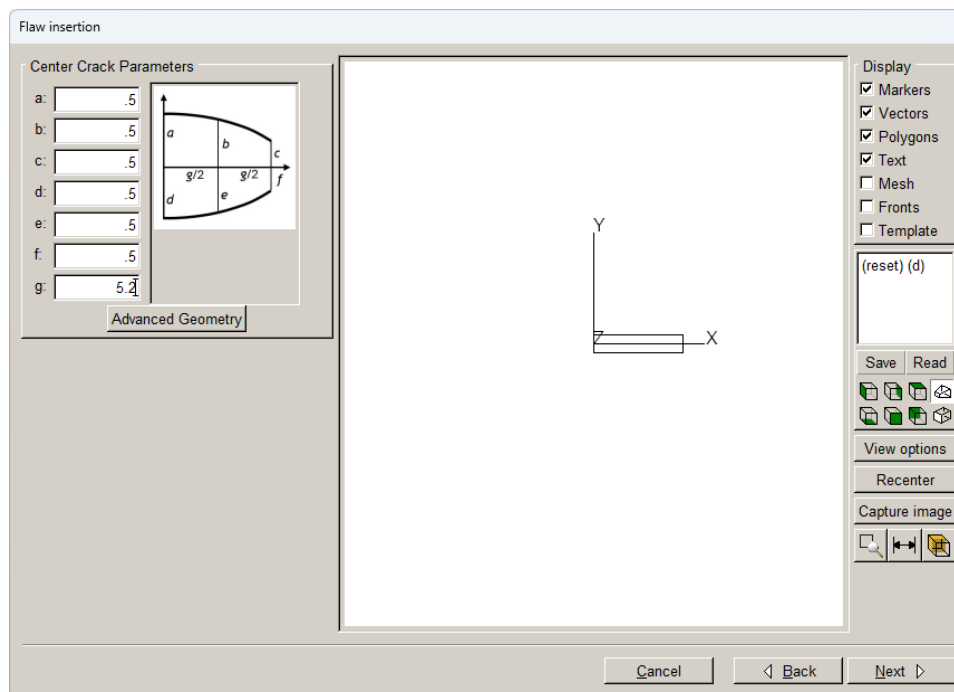


Figure 5.10 Middle through crack dimensions.

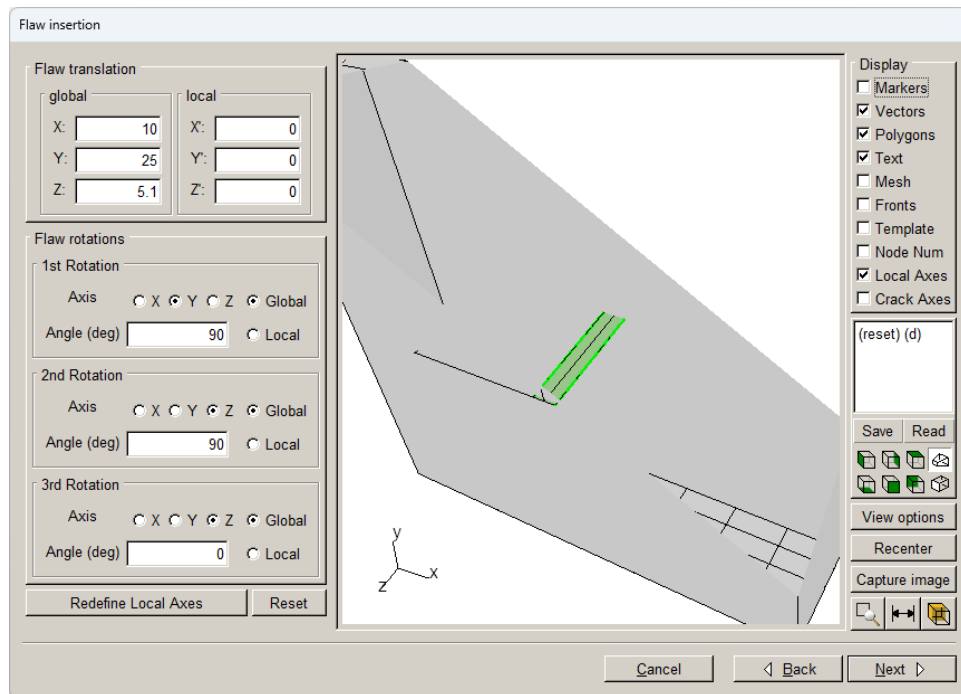


Figure 5.11 Middle through crack orientation.

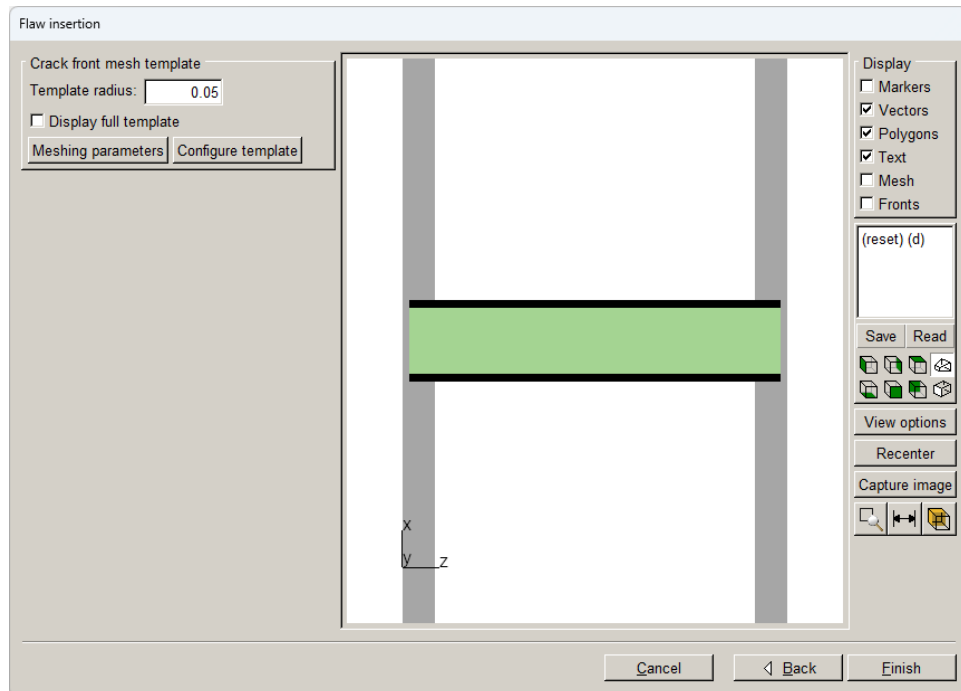


Figure 5.12 Middle through crack template.

Click the **Meshing Parameters** button in Fig 5.12 to display the dialog in Fig 5.13. Turn off the Do coarsen crack mouth mesh so that we generate a more refined mesh to help improve the results. A coarse tetrahedral mesh near the crack front can lead to a rough SIF curve.

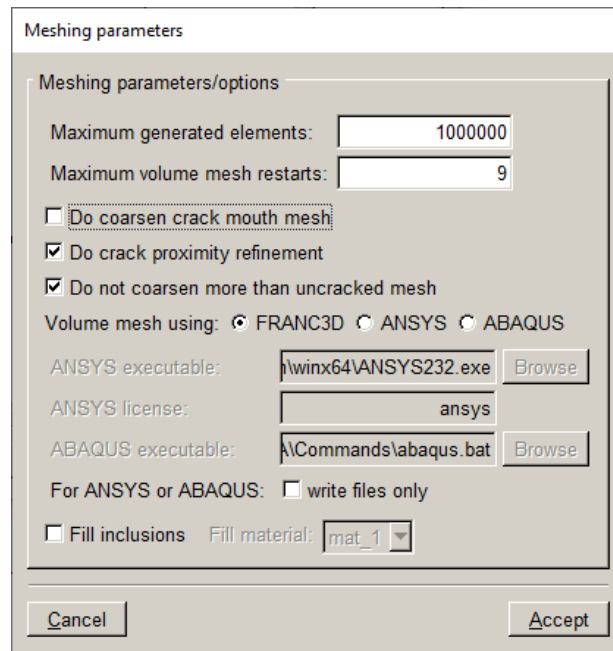


Figure 5.13 Meshing parameters dialog with Do coarsen crack mouth mesh turned off.

It is not required that you change the meshing parameters, the defaults will work. Once the template settings are done, select **Finish** (see Fig 5.12), and the Multiple Crack Definition dialog shows the first crack name, Fig 5.14. Select **Add** again to add the second crack, which will be an edge through crack, with one crack front, Fig 5.15.

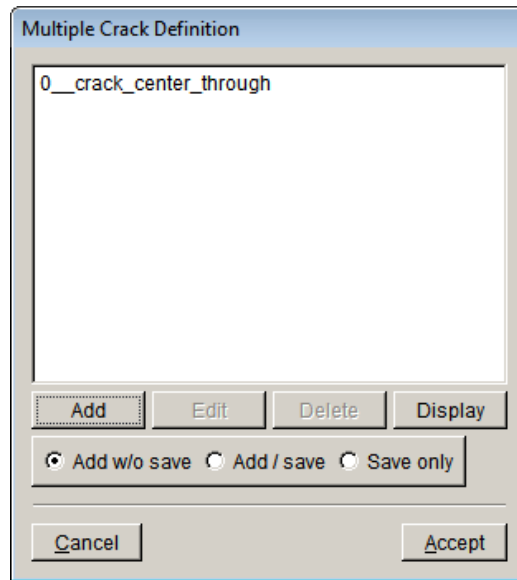


Figure 5.14 Multiple crack definition dialog with one crack defined.

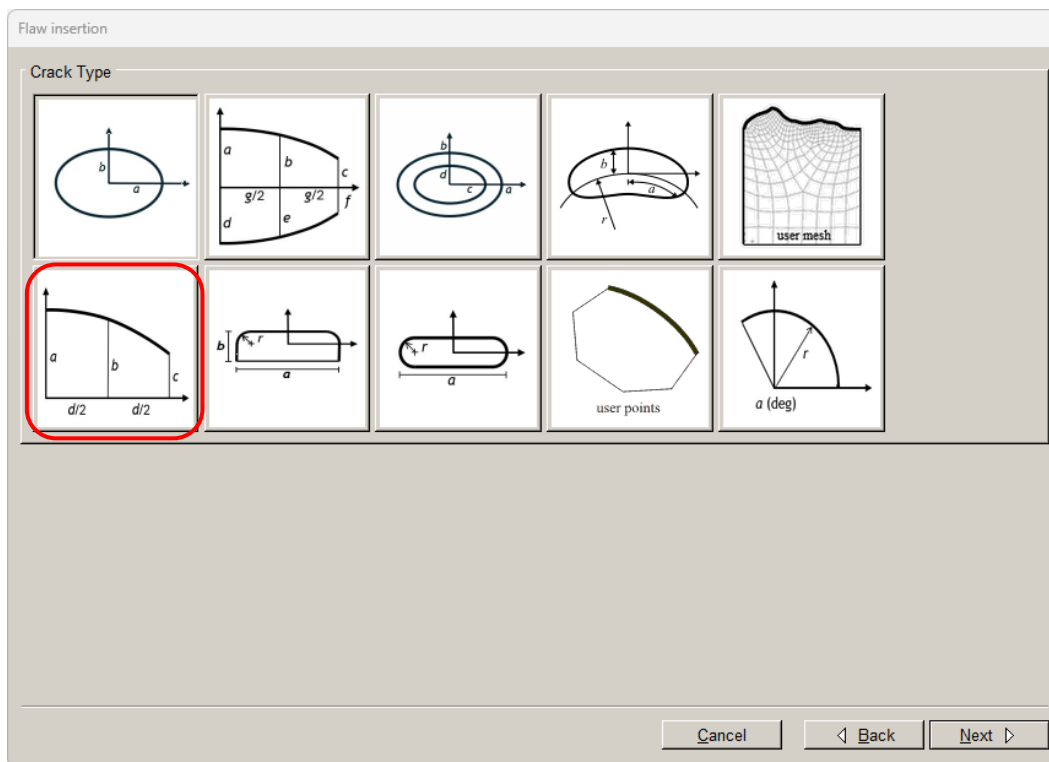


Figure 5.15 Flaw definition – edge through crack selected.

The crack width is defined such that it is bigger than the plate (5.2 compared to 5), and the crack depth is more (0.55) than 0.5, Fig 5.16. You must orient the crack so that the front falls inside the model, Fig 5.17; if you are unsure of the orientation, select **Next** and look at the template, Fig

5.18. If you set the orientation incorrectly, select **Back** to fix it. When you are done, select **Finish**. The Multiple Crack Definition dialog now has two crack names, Fig 5.19.

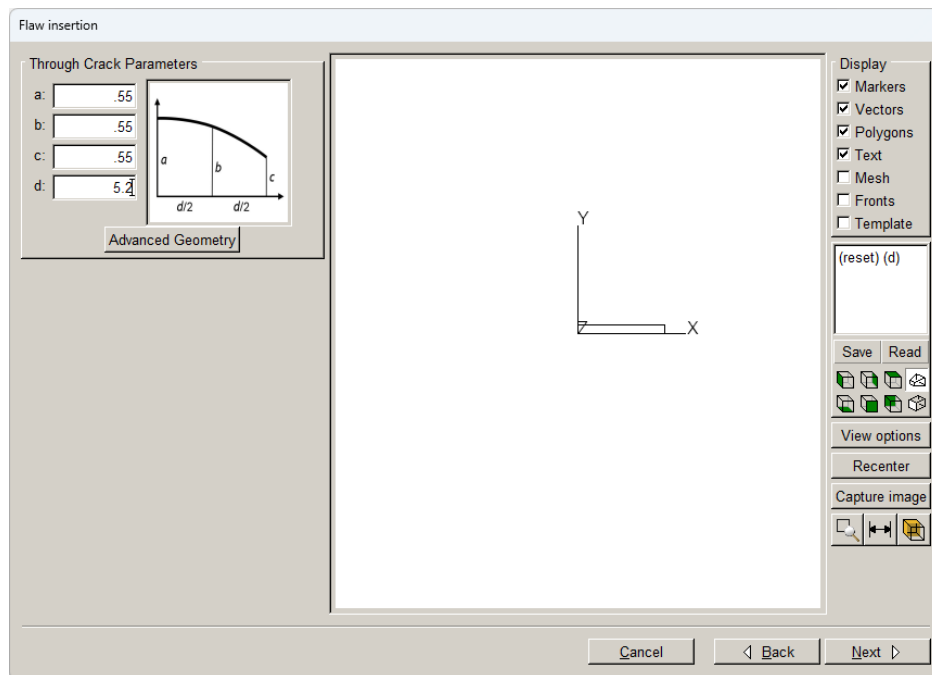


Figure 5.16 Edge through crack dimensions.

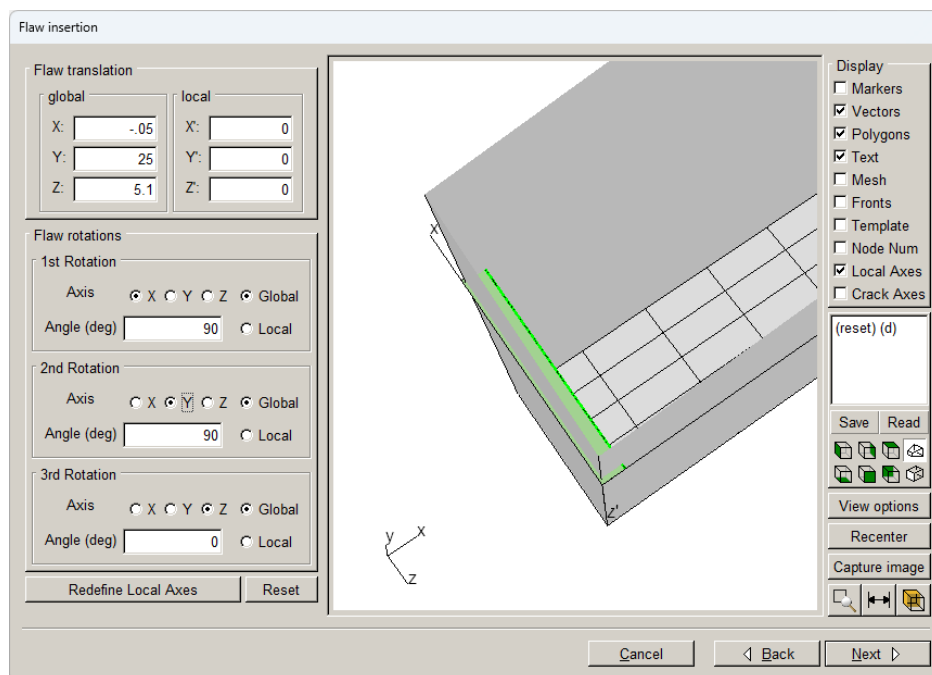


Figure 5.17 Edge through crack orientation.

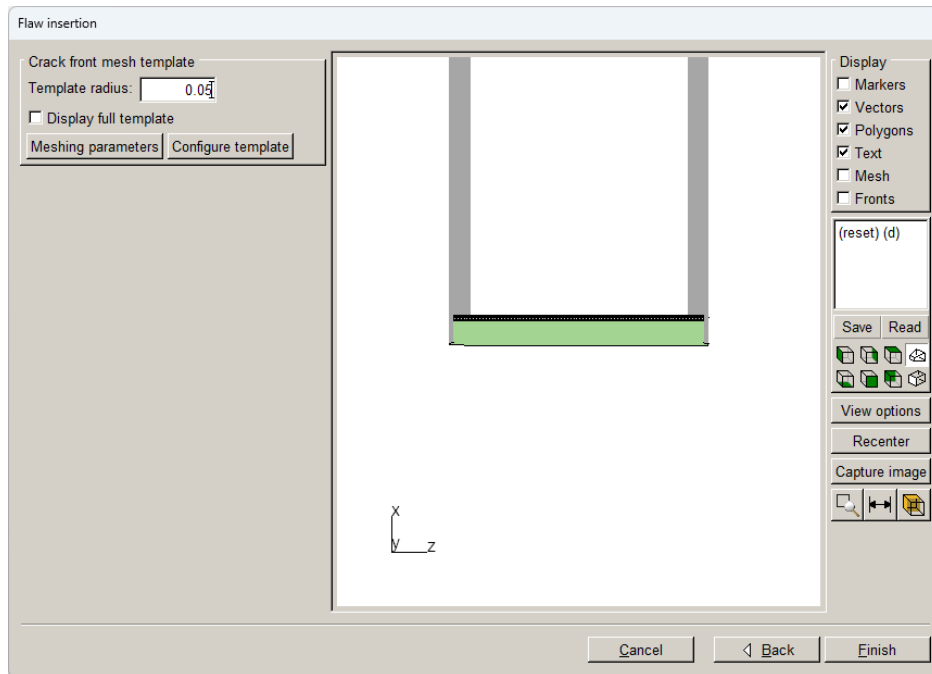


Figure 5.18 Edge through crack template.

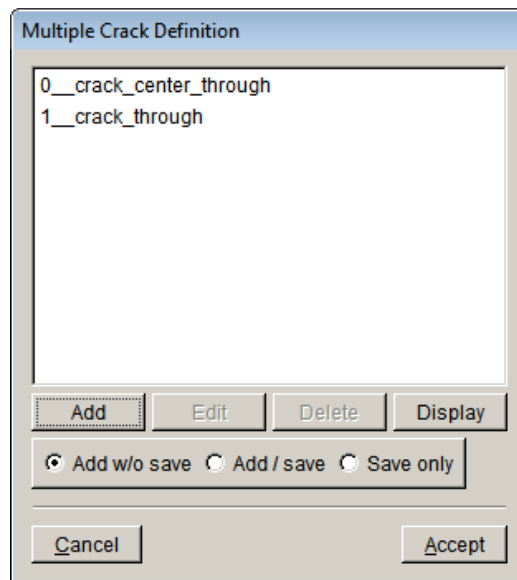


Figure 5.19 Multiple crack definition dialog with two cracks defined.

Select **Add** again to add the third crack, which will be a long shallow crack, Fig 5.20. Note that we use this crack instead of a high-aspect ratio ellipse as the geometry at the ends is better for the

template mesh. The crack geometry and location are defined such that intersection with the plate surface can be computed, Fig 5.21 - 5.23. Select **Finish**, Fig 5.23, to return to the Multiple Crack Definition dialog, which now shows three crack names, Fig 5.24.

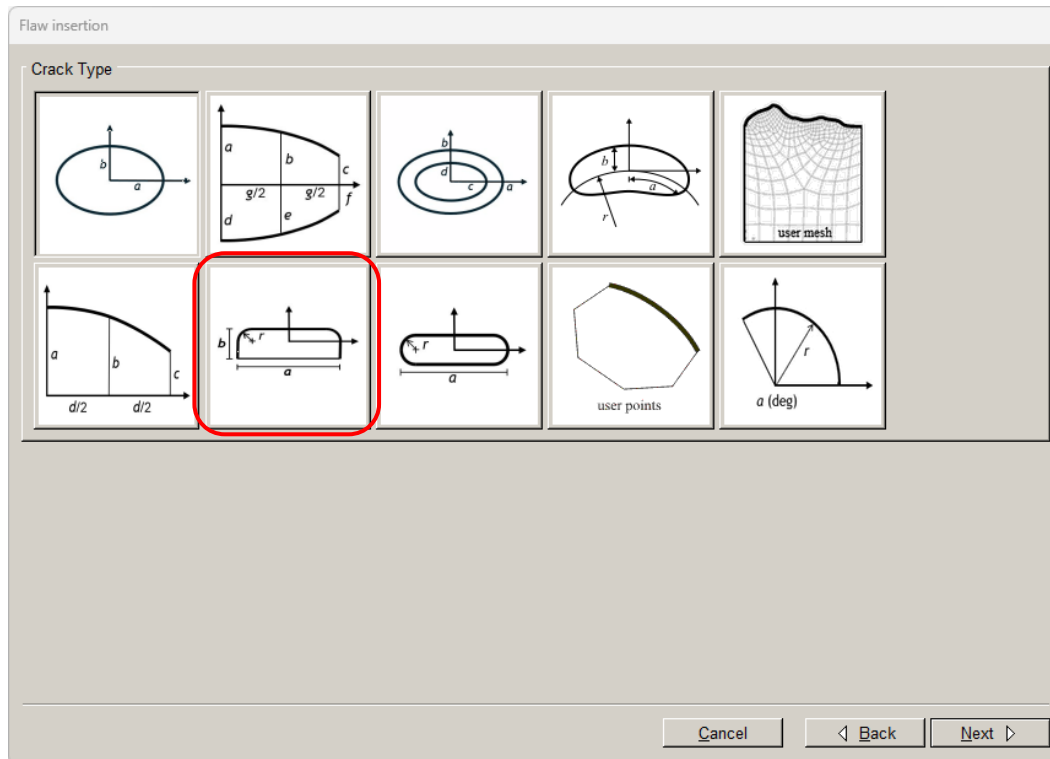


Figure 5.20 Flaw definition – long shallow crack selected.

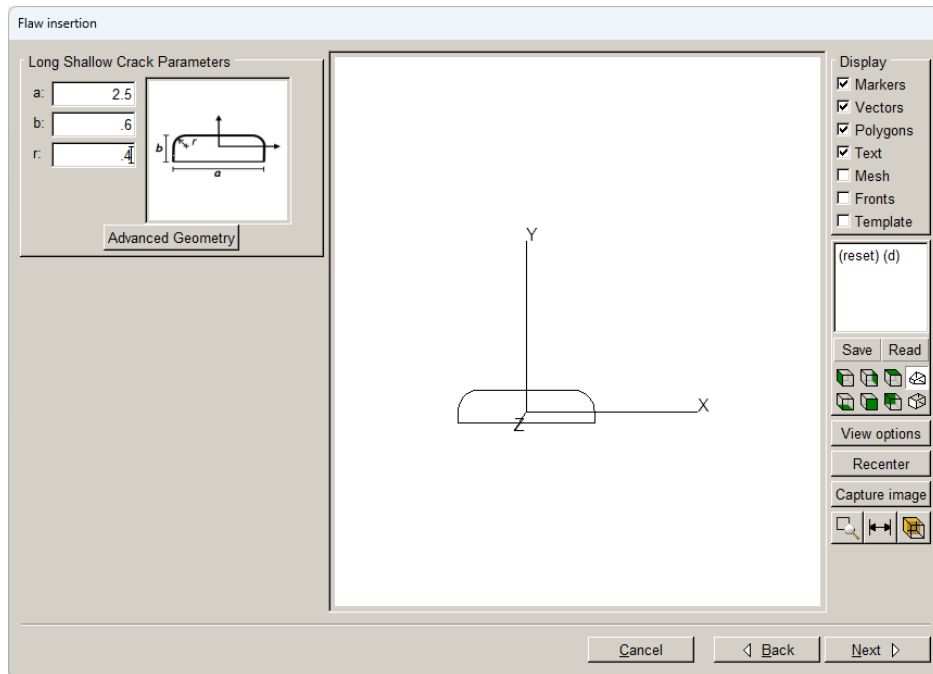


Figure 5.21 Long shallow crack dimensions.

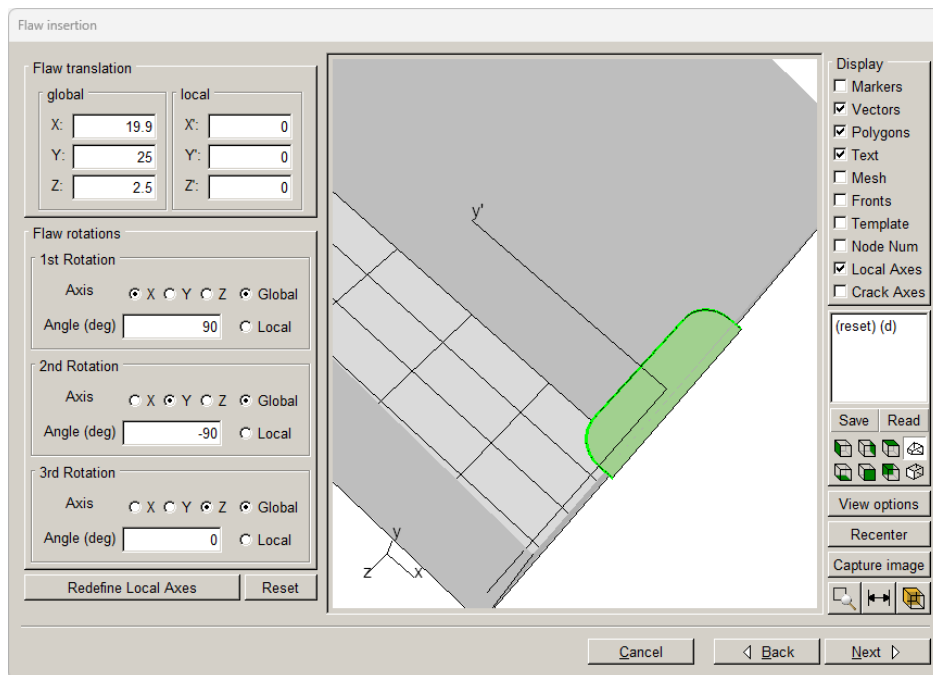


Figure 5.22 Long shallow crack orientation.

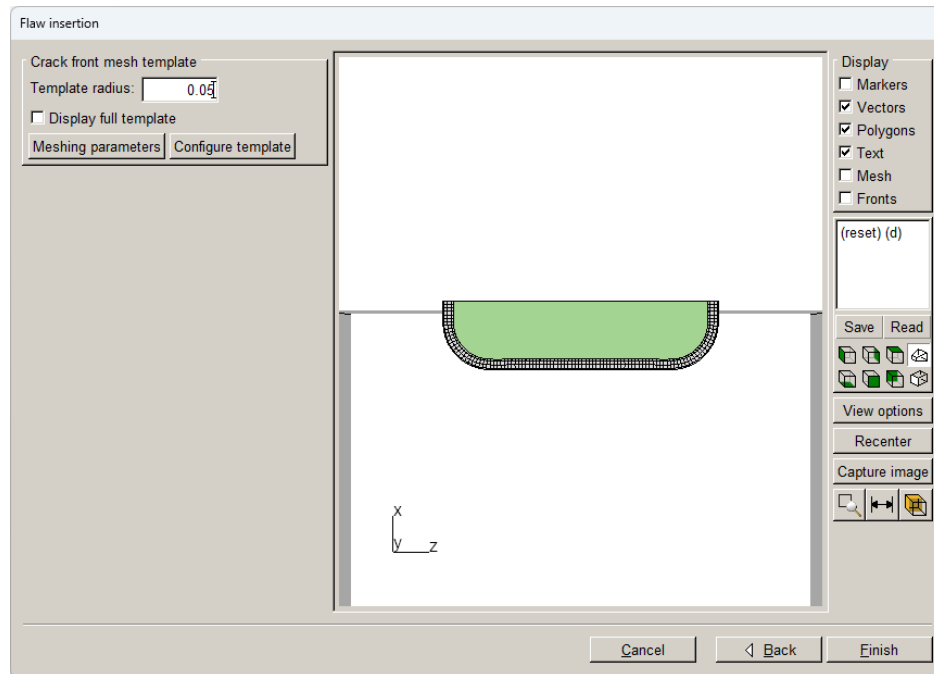


Figure 5.23 Long shallow crack template.

Select the **Display** button, Fig 5.24, to show all three cracks in the model, Fig 5.25. The cracks should not be overlapping or intersecting. Select **Finish** to close the display.

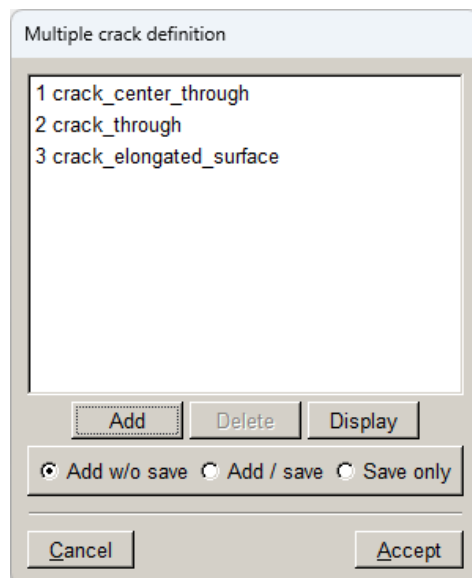


Figure 5.24 Multiple crack definition dialog with three cracks.

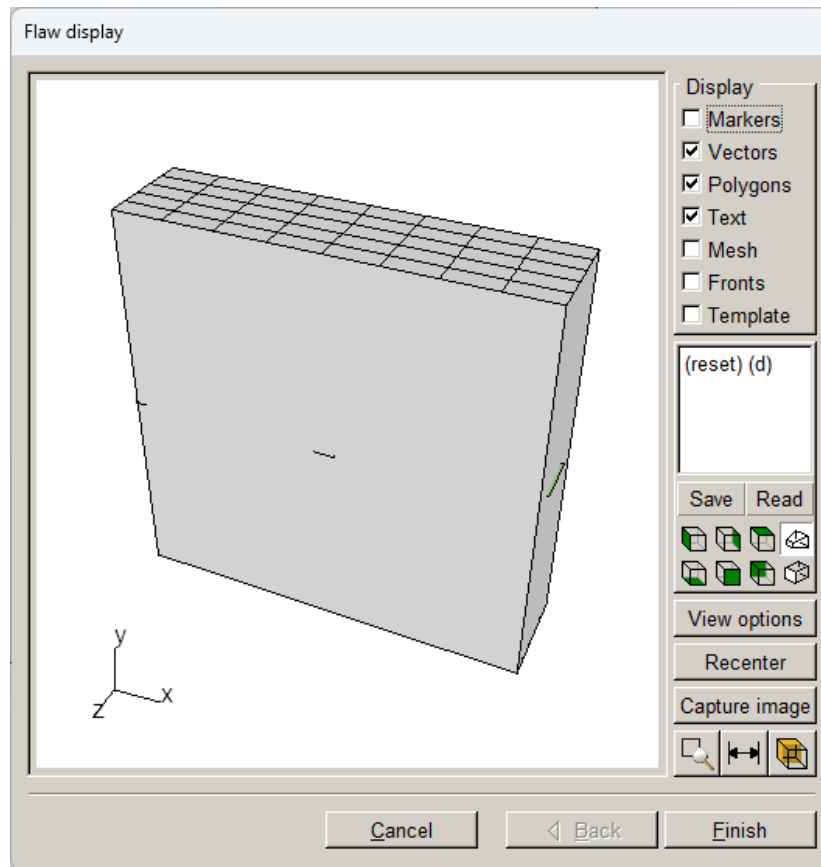


Figure 5.25 Multiple cracks displayed together.

Select **Accept** (see Fig 5.24), to start the crack insertion and remeshing process. Note that you can use the Add / save radio button to save the cracks to a *.crk* file if you think you might need to restart from this initial configuration; you might want to vary the mesh refinement (template radius) for instance. The remeshed model is shown in Fig 5.26, and we can proceed with the static crack analysis.

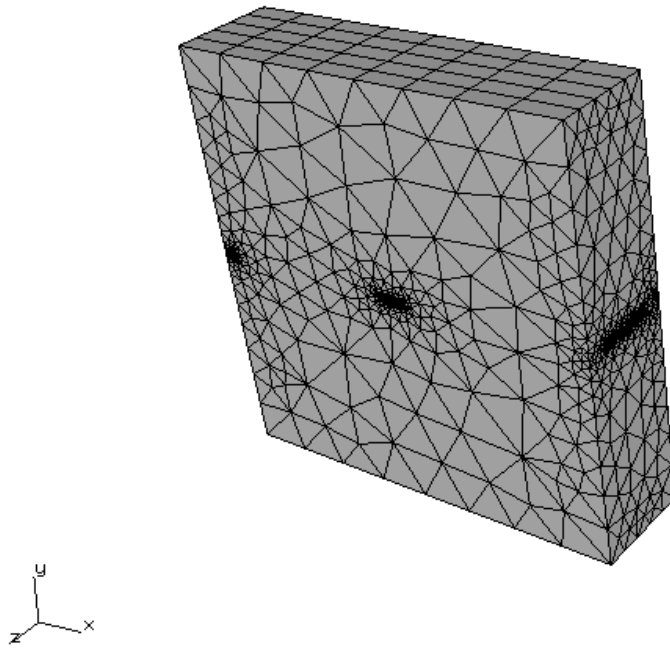


Figure 5.26 Remeshed model with three cracks inserted.

5.4 Step 4: Static Crack Analysis

Follow the same procedure as in the previous tutorials to run the analysis. Select **Static Crack Analysis** from the FRANC3D menu, set the file name, set the analysis options, Figs 5.27a-c, check the local + global connection, Figs 5.28a-c, and select **Finish** to start the analysis in the background.

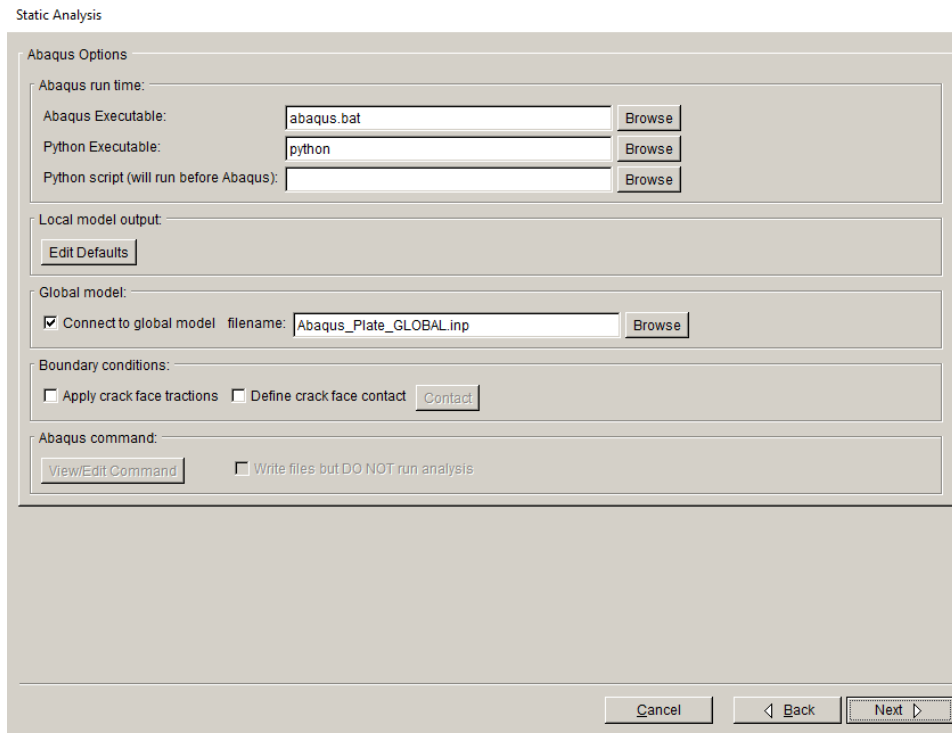


Figure 5.27a Static analysis wizard panel to set ABAQUS solver options.

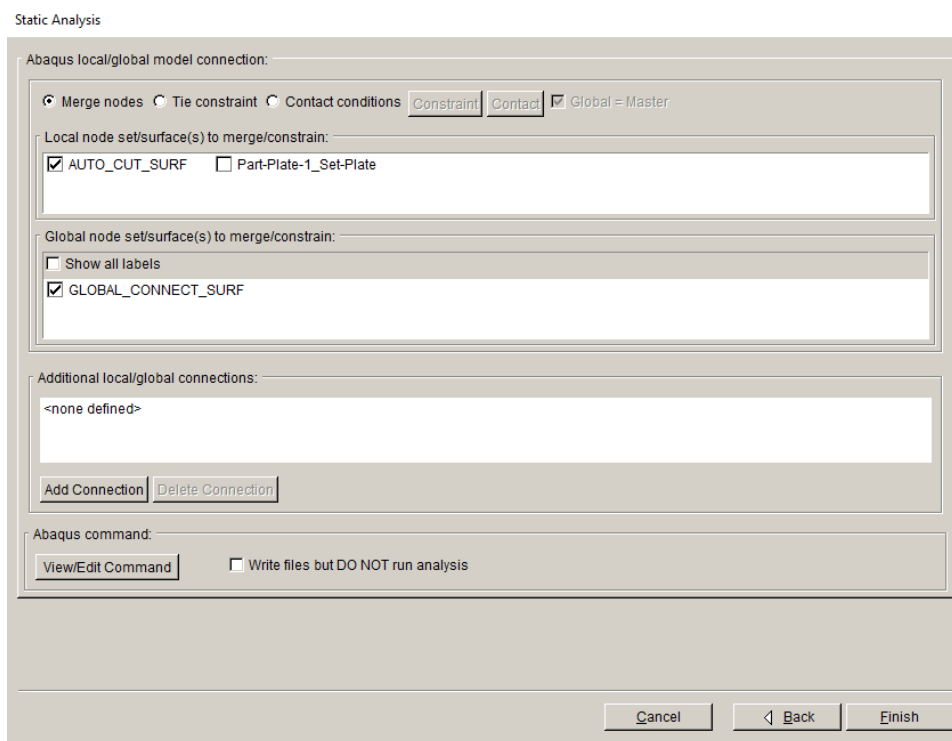


Figure 5.28a ABAQUS analysis wizard panel to set local + global connection.

Static analysis

ANSYS Options

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact ☐ Initial strain

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 5.27b Static analysis wizard panel to set ANSYS solver options.

Static analysis

ANSYS local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☒ retain ☐ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF ☐ all_template_nodes

Global component(s) to merge/constrain:

☐ Show all labels

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 5.28b ANSYS analysis wizard panel to set local + global connection.

Static analysis

NASTRAN Options

NASTRAN run time:

NASTRAN Executable:

Python Executable:

Python script (will run before NASTRAN):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions: ☐ Define crack face contact:

NASTRAN command:

☐ Write files but DO NOT run analysis

Figure 5.27c Static analysis wizard panel to set NASTRAN solver options.

Static analysis

NASTRAN local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Local node sets to merge/constrain:

☒ AUTO_CUT_SURF

Global node sets to merge/constrain:

☒ GLOBAL_CONNECT_SURF ☐ disp_1 ☐ disp_2 ☐ disp_3

Additional local/global connections:

<none defined>

NASTRAN command:

☐ Write files but DO NOT run analysis

Figure 5.28c NASTRAN analysis wizard panel to set local / global connection.

5.5 Step 5: Compute SIFs for Multiple Crack Fronts

Once the analysis has finished, select **Compute SIFs** from the FRANC3D menu. The SIF dialog allows you to select specific crack fronts, Fig 5.29. The front is identified in the image on the left by the red **A** and **B**. As you grow the crack(s), you will have SIF curves from all four crack fronts for all crack growth steps. If there were multiple load steps, as in Tutorial #2, you would have separate SIF curves for each of those also.

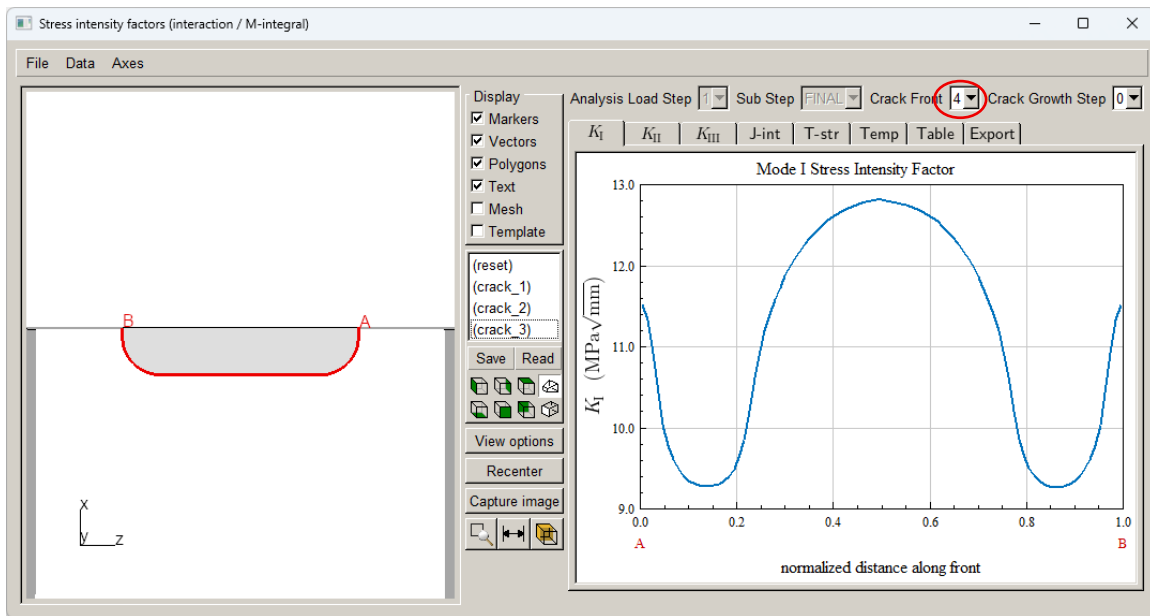


Figure 5.29 Mode I SIF for crack front #4.

5.6 Step 6: Grow Multiple Cracks

Select **Grow Cracks** from the FRANC3D menu. We switch to quasi-static growth and leave all the other parameters at their default values. The resulting crack growth, using a median extension of 0.15, is shown in Fig 5.30. The growth for all crack fronts is scaled based on the specified increment and the relative SIFs (and growth rates) along all crack fronts. Setting the median extension to 0.15 ensures that all crack fronts will extend. Note that you can turn off growth for a specific crack front by unchecking the grow box (bottom left of Fig 5.30).

Each front has its own fitting parameters. In this case, this is important as the crack front on the left side can easily be fit by a 3rd order polynomial (2nd order would work too), Fig 5.31, but the crack front on the right must use a different type of fit, Fig 5.32. For this crack front, a cubic spline or moving polynomial fit works, but we use the multiple polynomial (you can set the order to 4 for this front if the fit is better).

Note that FRANC3D might not track crack front IDs perfectly, so as the cracks grow, the IDs might change, although this is unlikely.

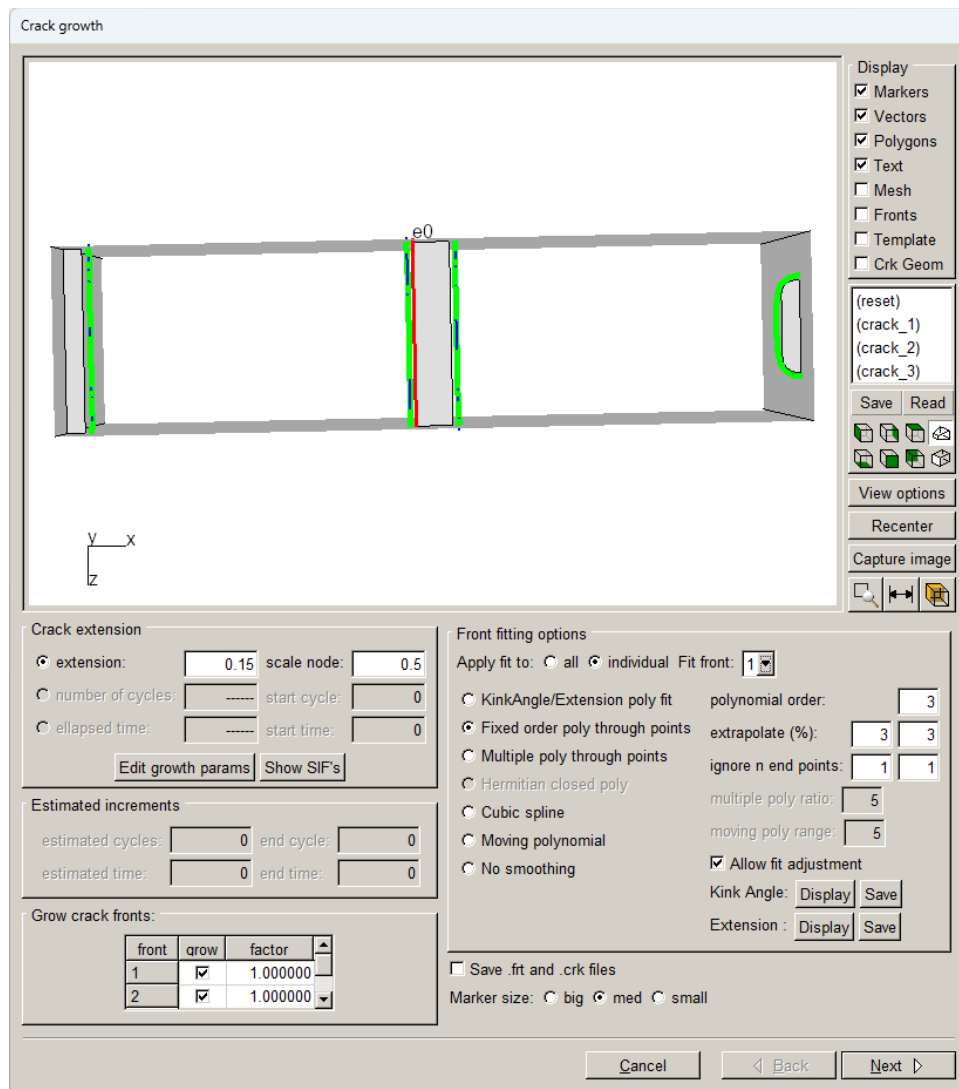


Figure 5.30 Crack growth wizard panel.

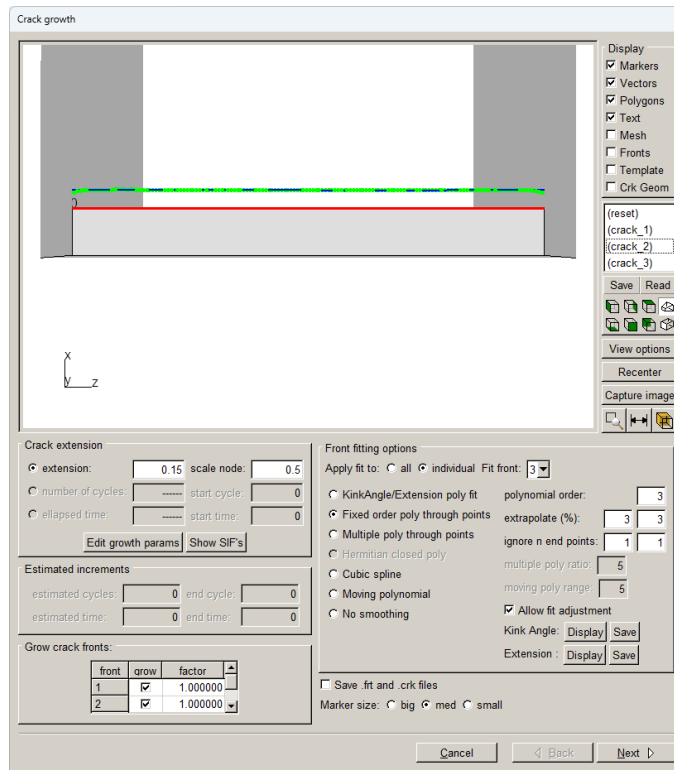


Figure 5.31 Crack growth wizard panel – showing left-side crack front.

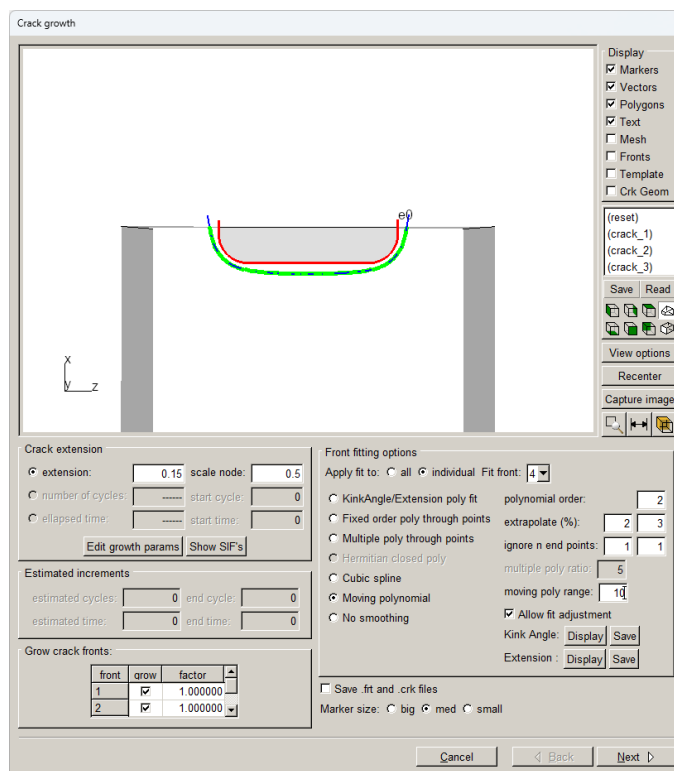


Figure 5.32 Crack growth wizard panel – showing crack front #4.

Once you have checked the fit for all fronts, you can proceed to the next panel to set the template mesh parameters, Fig 5.33, and then proceed with the actual process of growing and remeshing. The template radius of 0.05 is the same for all fronts, but it can be set differently for each front.

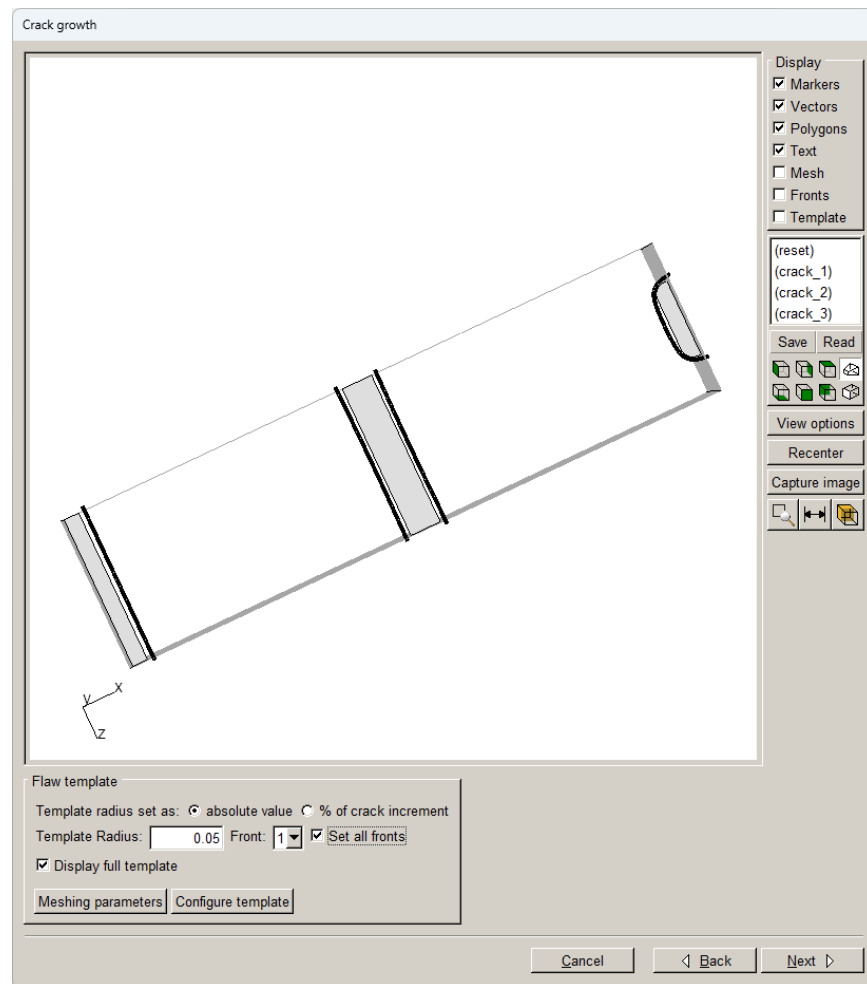


Figure 5.33 Crack growth wizard panel – showing crack front templates.

5.7 SIFs History for Multiple Cracks

After the crack growth and remeshing has been completed in Step 6, we set up five steps of automatic crack growth analysis using the same growth settings. Once the five steps are

completed, you can plot the SIFs for all cracks, Figs 5.34-35. Using the dropdown tabs at the top of the dialog, you can switch between the steps of crack growth and between crack fronts.

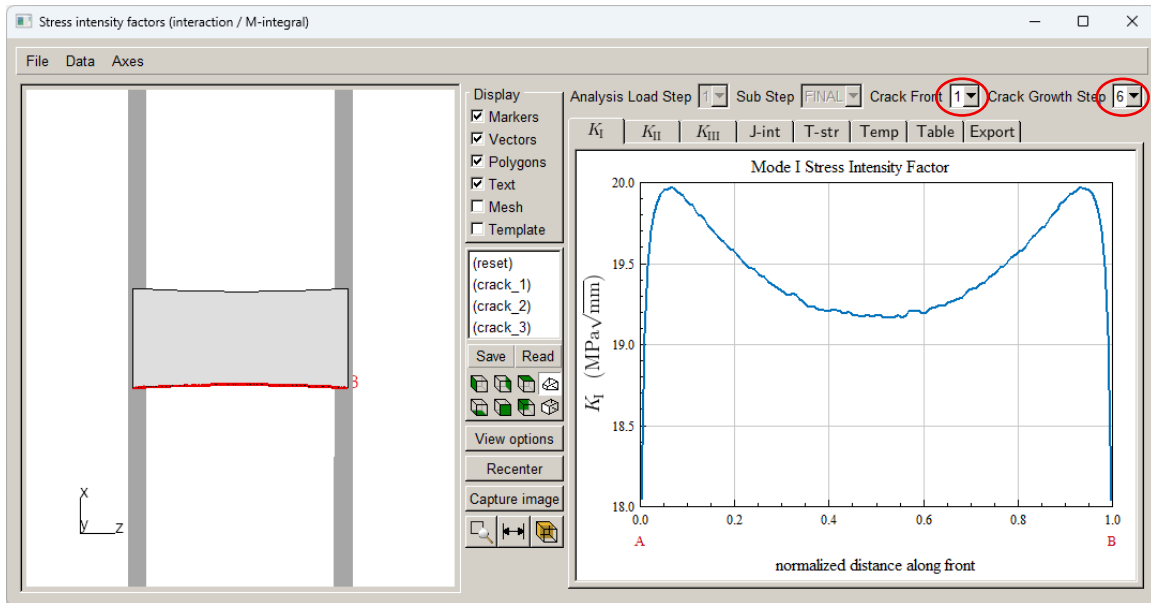


Figure 5.34 SIFs dialog – SIFs for growth step 6 of front 1.

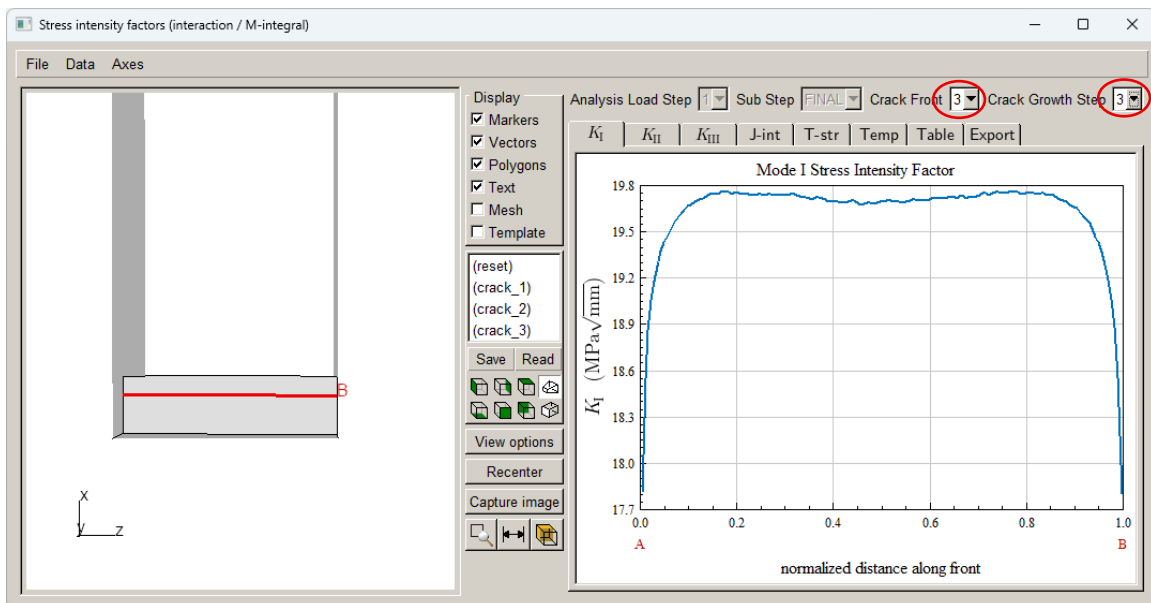


Figure 5.35 SIFs dialog – SIFs for growth step 3 of front 3.

If you want to plot all SIFs for all growth steps for a specific crack front together in a single plot, use the **Cracks** → **SIFs For All Fronts** menu option, Fig 5.36. You can switch between the crack fronts using the dropdown tab at the top. You can also plot the SIF history along a path through any of the crack fronts using the **Cracks** → **SIFs Along a Path** menu option, Fig 5.37.

Fatigue cycles can be computed also; we leave this as an exercise for the reader.

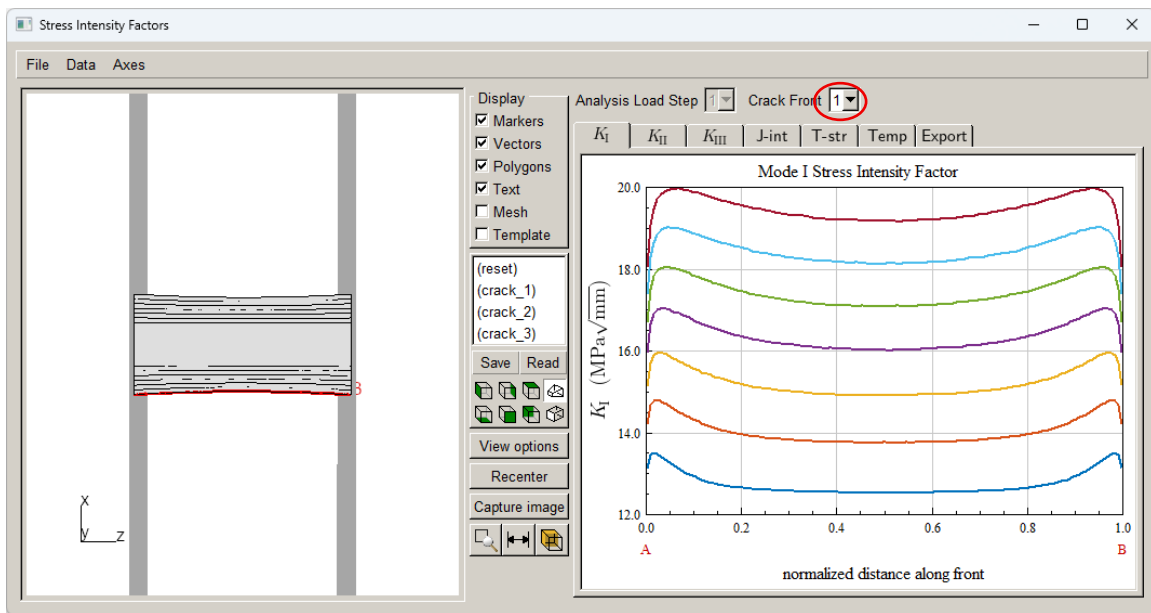


Figure 5.36 SIFs For All Fronts dialog – for crack front 2.

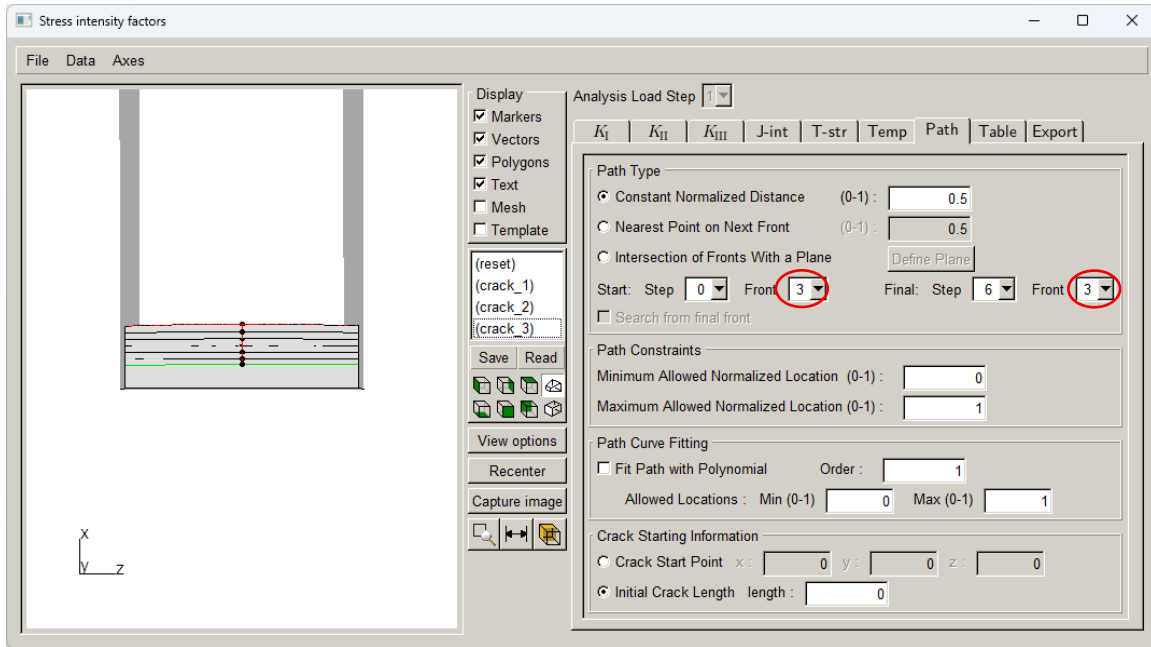


Figure 5.37 SIFs Along a Path dialog – for crack front 3.

5.8 Crack Growth Transitions

Crack growth around geometric corners or through surfaces can sometimes fail depending on the choice of template radius and volume meshing as the crack front reaches the ‘opposing’ model surface. The next two sub-sections show how one can deal with these cases.

5.8.1 Grow a Crack around a Corner

In this section, we describe how to grow a crack around a 90-degree corner, creating a through-edge-crack from a corner-crack shape. We could use the model from the previous steps, as crack 4 would need to grow around the corners of the plate, but we will simplify things by using a single corner-crack.

Import the local portion of the plate model that was created during Step 2. We can import as “already divided” or we can subdivide again and choose the retained element listing (*_Plate_RETAINED_ELEMS.txt*). The model should appear as was shown in Fig 5.6.

We insert a single crack, using the elliptical shape from the flaw library. The crack radius is 1.0. It is located as was shown in Fig 5.38. The mesh template settings are left at their default values.

An initial static crack analysis is completed to obtain the initial SIFs, Fig 5.39. The crack growth is done using the quasi-static growth model with $n=2$. The initial growth and fitting parameters are shown in Fig 5.40. The mesh template parameters are left at their default values.

To keep things simple, 24 steps of automated crack growth are completed, using the same median extension (extension=0.15 and scale node=0.5) .

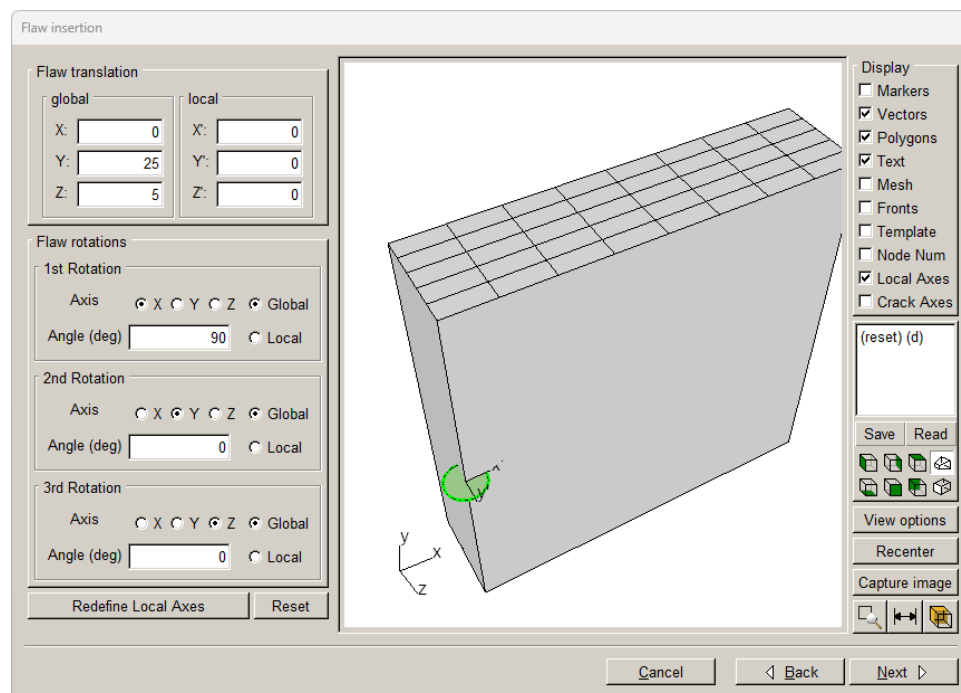


Figure 5.38 Single corner crack location and orientation.

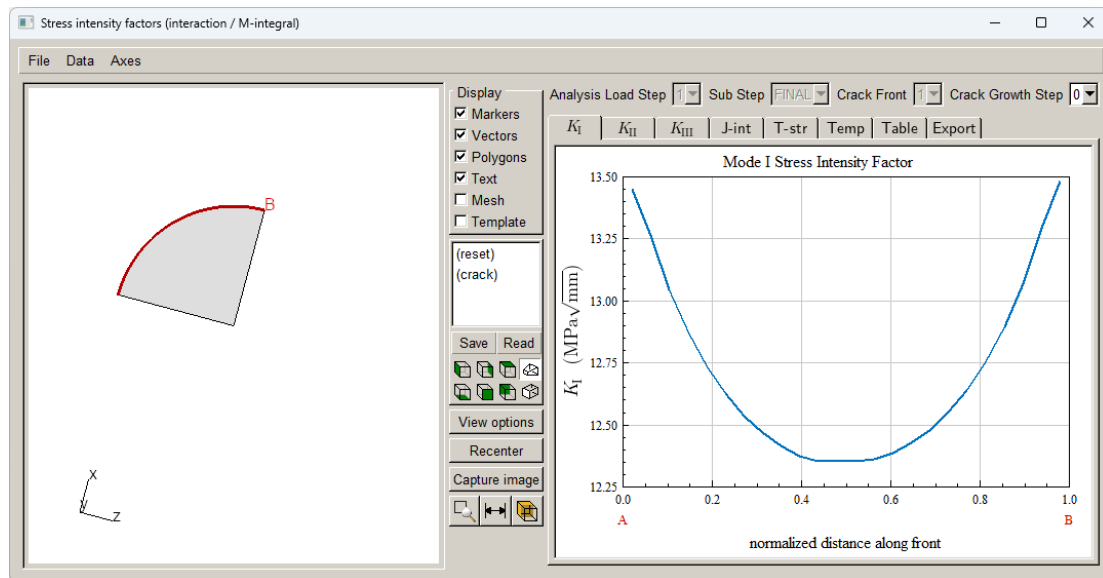


Figure 5.39 Single corner crack SIFs.

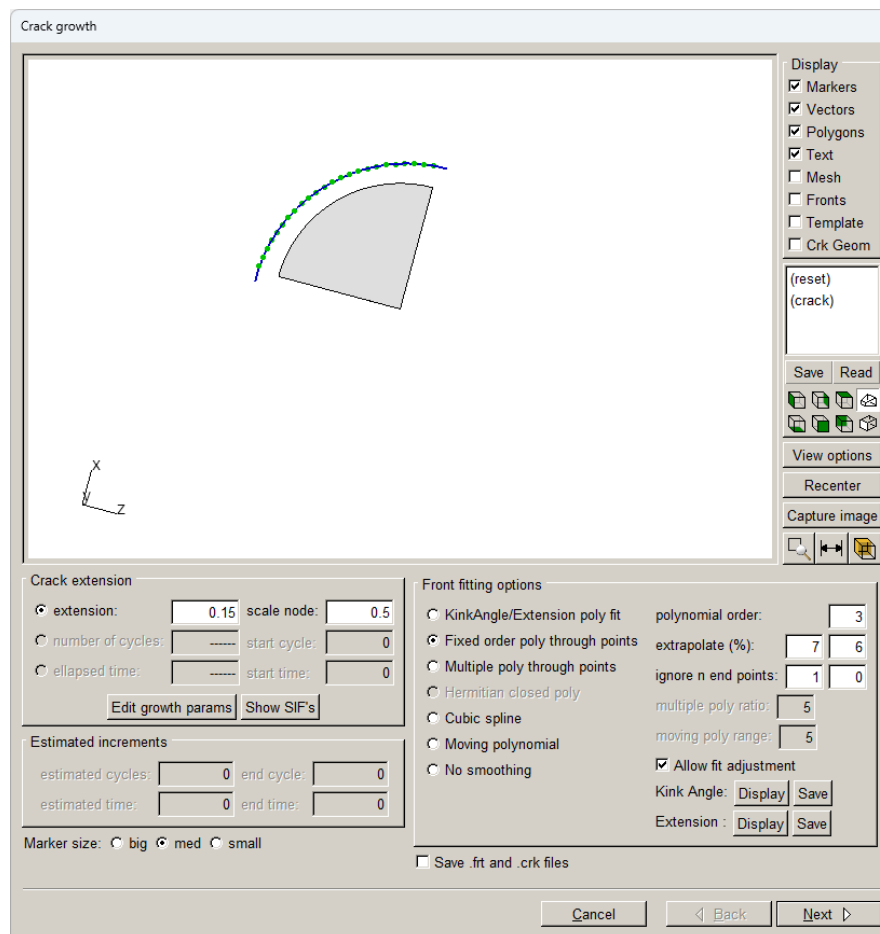


Figure 5.40 Single corner crack initial growth.

At step #24, the crack front is close to the back surface. The crack will transition from a corner crack to an edge crack, and this can happen automatically, Fig 5.41, but if you want to capture SIF values as the crack transitions, then some “manual” crack growth steps and static analyses can be performed.

Fig 5.42 shows a series of manual crack growth steps where the growth increment is adjusted so that the crack front transitions in a user-controlled manner around the corner. Note that when the crack front is at the corner, and when it intersects the model surface at shallow angles, we can turn on Simple Intersections for the template mesh. This forces the template to be pulled back from the model surface; FRANC3D should turn this on automatically as needed. If the back surface of the plate is not being refined enough when the crack front is close, you can edit the meshing parameters and turn on Do crack proximity refinement (see top right panel of Fig 5.42).

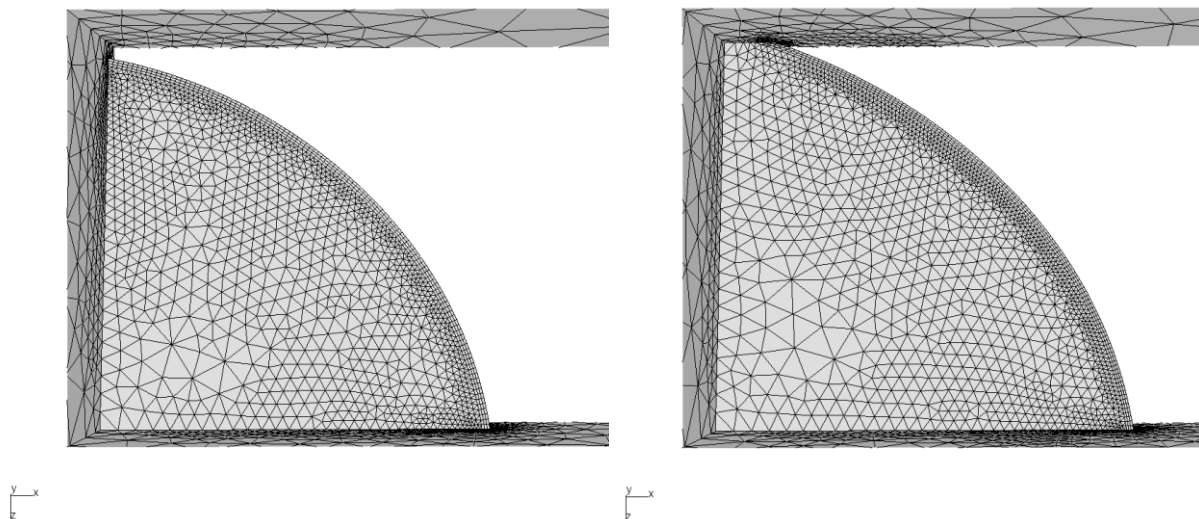


Figure 5.41 Single corner crack after 24 steps (left) and after 26 steps (right) of growth.

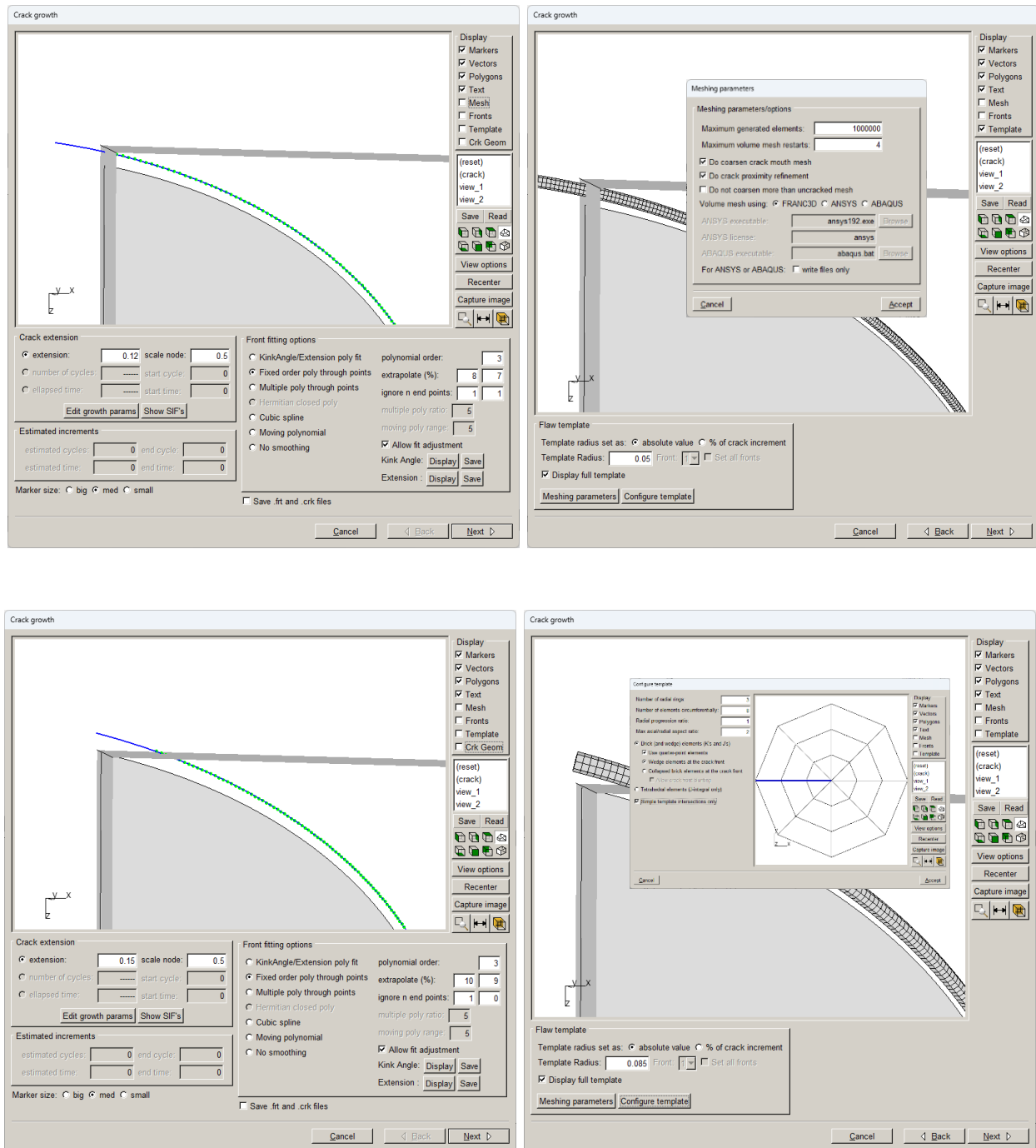


Figure 5.42 Manual steps of growth for single corner crack as it transitions around a corner.

5.8.2 Grow Crack through a Back Surface

In this section, we describe how to grow a crack through a back-surface, creating a center-through-crack with two fronts from a surface-penny crack in a plate. The steps to grow a crack front through a back-surface are like those for transitioning a crack around a corner. We use manual crack growth and static analysis steps when the crack front gets close to the surface and as it breaks through. We describe the steps here using the same plate model but with a single surface-penny crack inserted on the front surface of the plate.

Import the local portion of the plate model that was created during Step 2. We can import as “already divided” or we can subdivide again and choose the retained element listing (*_Plate_RETAINED_ELEMS.txt*). The model will appear as was shown in Fig 5.6.

We insert a single crack, using the elliptical shape from the flaw library. The crack radius is 1.0. It is located as shown in Fig 5.43. The mesh template settings are left at their default values.

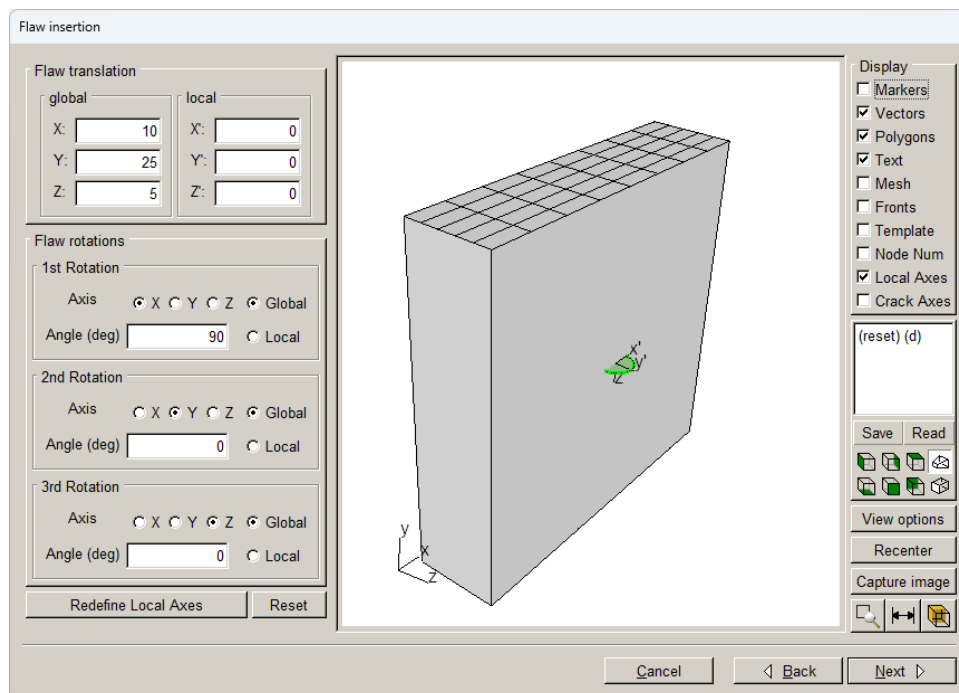


Figure 5.43 Single surface crack location and orientation.

An initial static crack analysis is completed to obtain the initial SIFs, Fig 5.44. The crack growth is done using the quasi-static growth model with $n=2$. The initial growth increment and fitting parameters are shown in Fig 5.45. To keep things simple, automated crack growth is completed using a constant median extension for 10 steps, and then a slightly larger extension for another 10 steps, *etc.*, until the crack front is close to the plate back-surface.

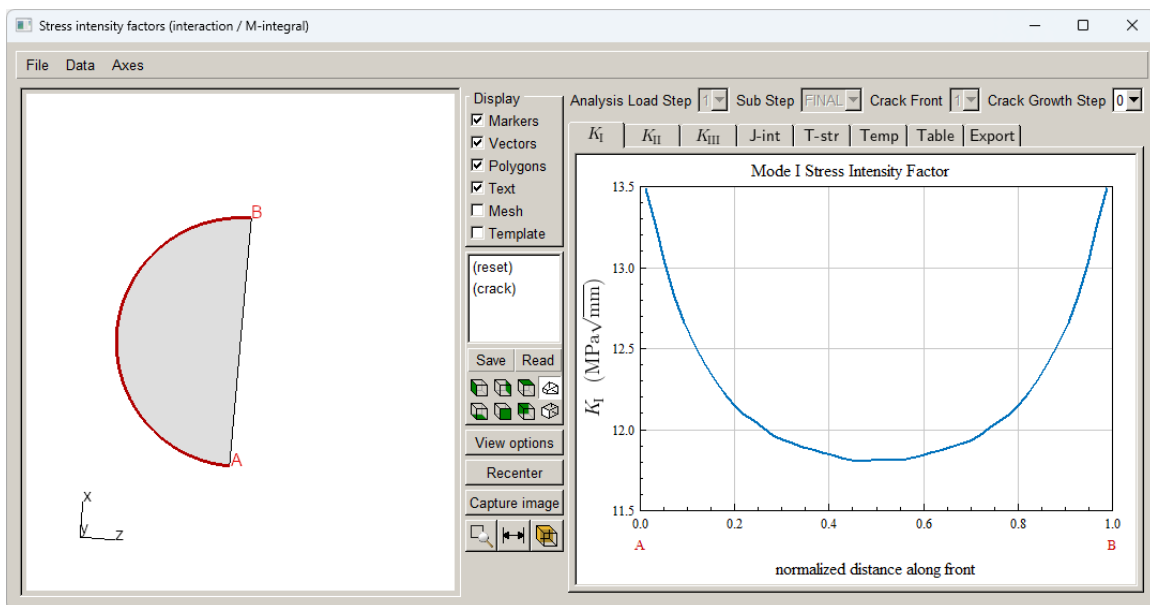


Figure 5.44 Single surface crack SIFs.

When the front reaches the back surface, the crack transitions from a surface crack to a center-through crack; this might happen automatically, but it might require manual steps of growth. If the user wants to capture more SIF values as the crack transitions, then some manual crack growth steps and static analyses should be performed.

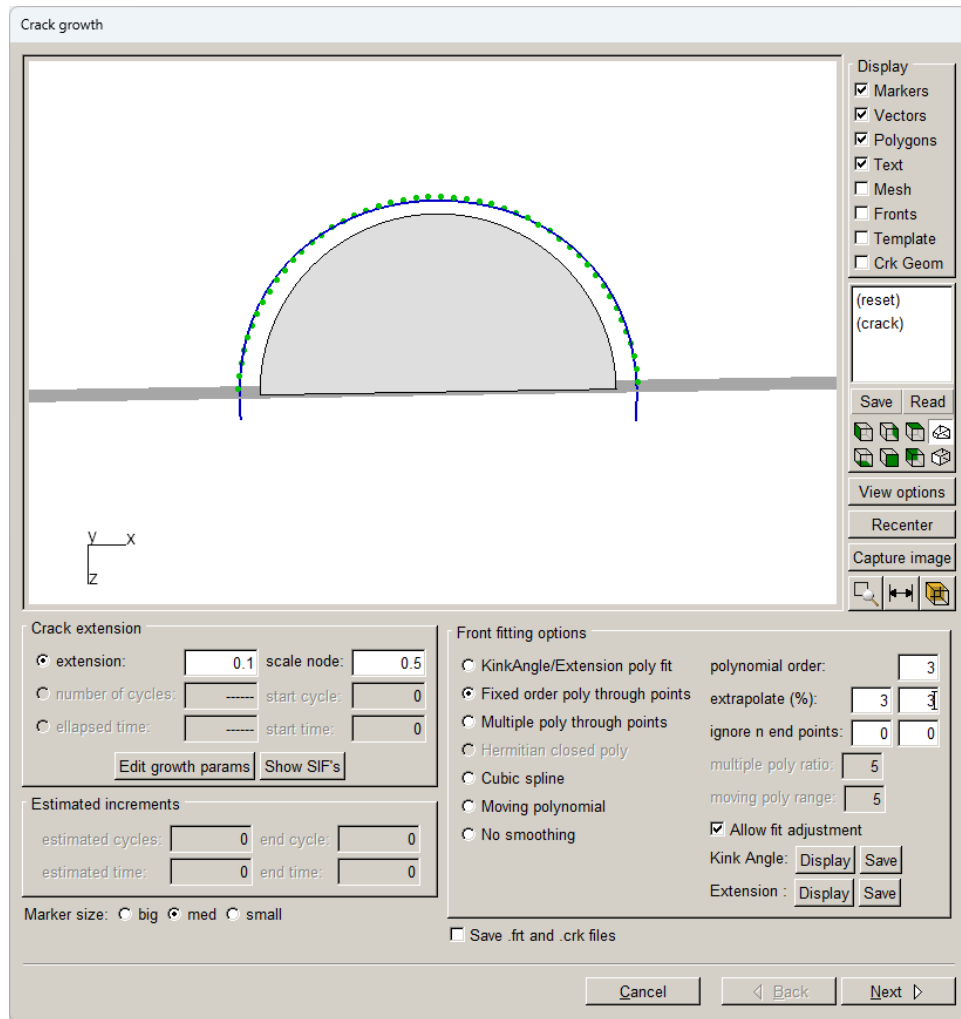


Figure 5.45 Surface crack growth parameters.

Fig 5.46 shows a series of manual crack growth steps where the specified growth increment is adjusted so that the crack front transitions in a user-controlled manner toward and through the back-surface. If the back surface of the plate is not refined enough when the crack front is close, you can edit the meshing parameters and turn on Do crack proximity refinement.

The crack increment and template radius can be reduced to get the front close to the back surface before breaking through, but there needs to be enough room to volume-mesh (top left image). Eventually, the crack must be propagated through the back surface (middle left image). Note that when the crack front intersects the model surface at shallow angles, we can turn on Simple

Intersections for the template mesh (middle right image). This forces the template to be pulled back from the model surface; FRANC3D should turn this on automatically as needed. Simple Intersections can be turned off once the front advances a step or two.

Sections 8 and 9 of the User's Guide provide additional examples and information to assist with crack growth.

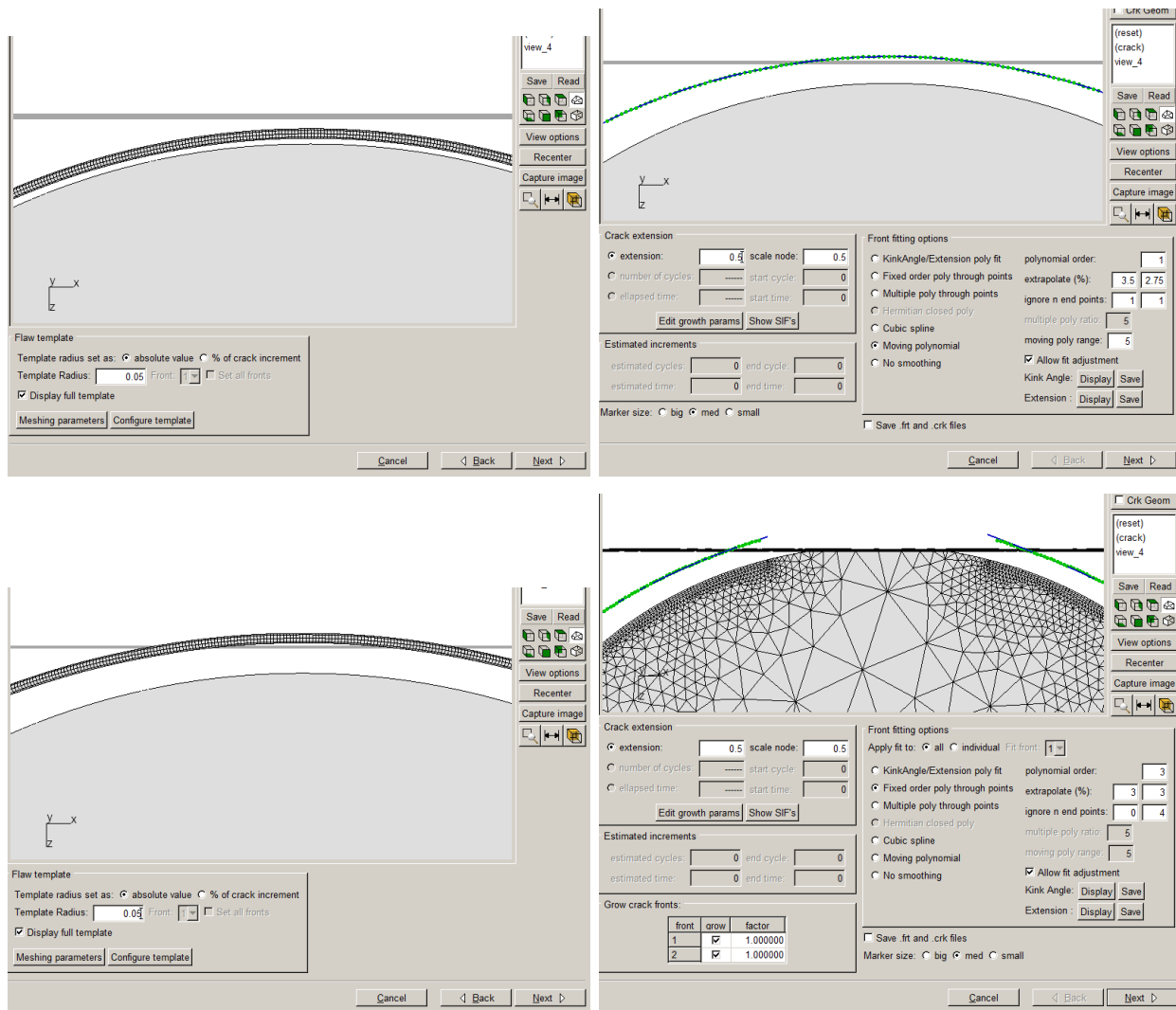


Figure 5.46 Surface crack in a plate breaks through the back surface.

5.9 Grow Multiple Cracks in Multiple Submodel Portions

In this section, we describe how to extract multiple separate portions of a full model when creating the local model, which might be useful when inserting multiple cracks if the cracks are far apart.

5.9.1 Step 1: Import Plate Model into FRANC3D

From the FRANC3D menu, select **File** and **Import**. We Import and divide the plate model, Fig 5.47. First extract the mid-portion of the plate using the **Rubberband** tool in the **Submodeler**, Fig 5.48. Select **Crop** and then continue with the subsequent Rubberband and **Crop** selections as shown in Figs 5.49-51. Note that we switch the cropping option to highlight the inverse selection in Figs 5.49-51.

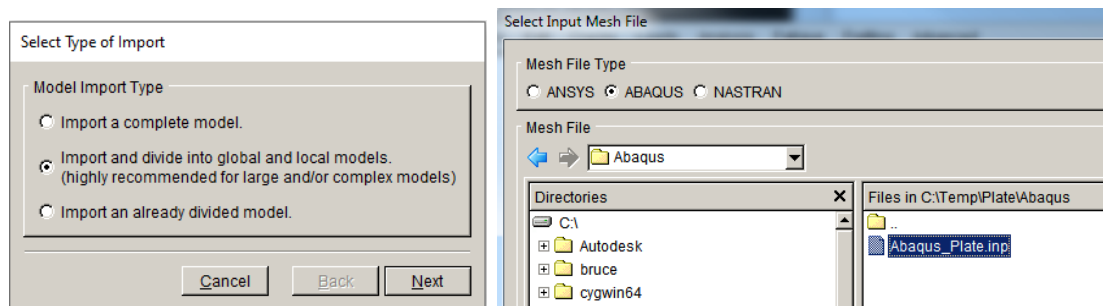


Figure 5.47 FRANC3D model import of plate model.

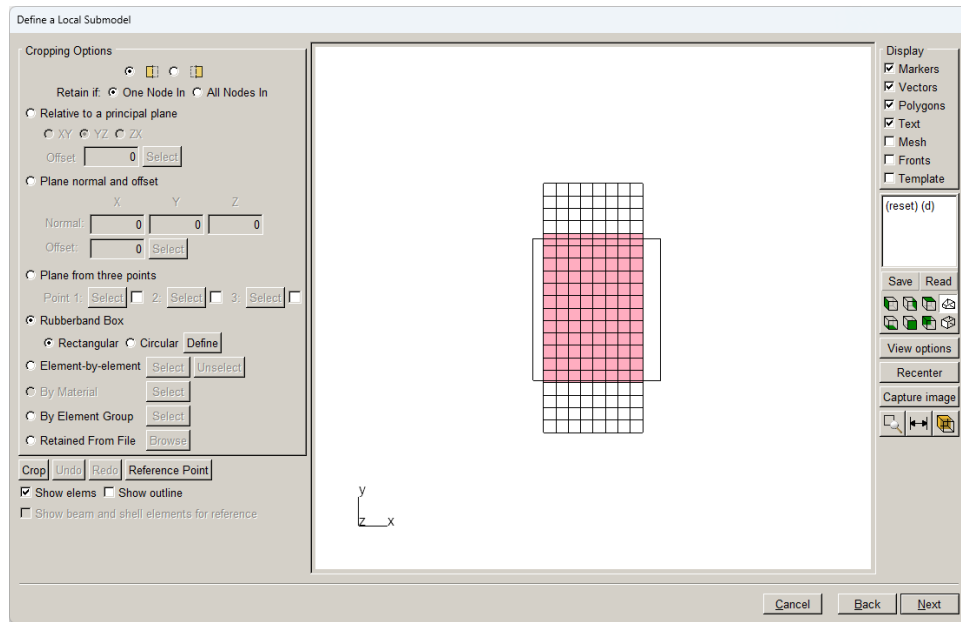


Figure 5.48 FRANC3D first submodel extraction for plate model.

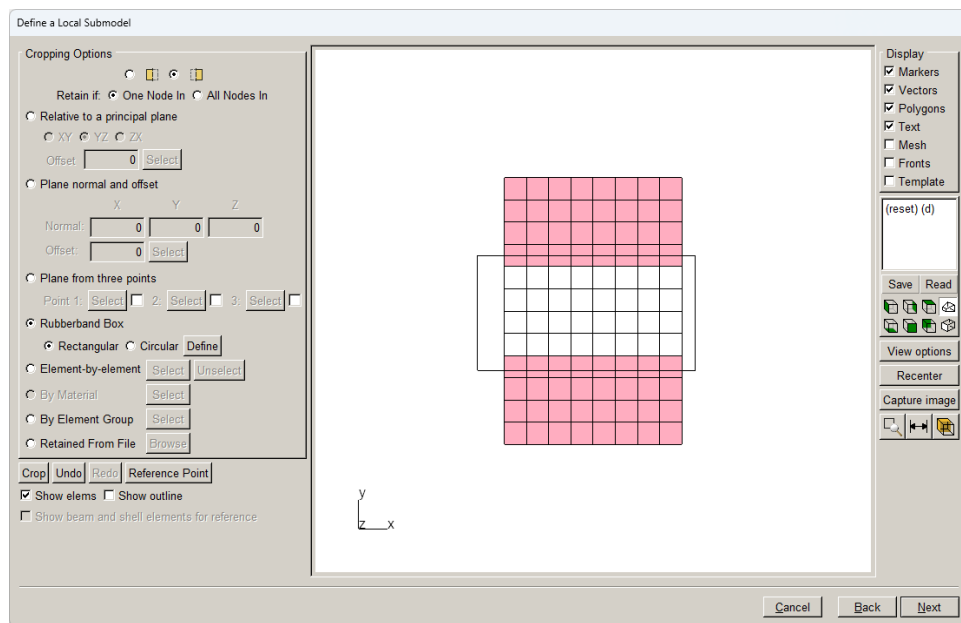


Figure 5.49 FRANC3D second submodel extraction for plate model.

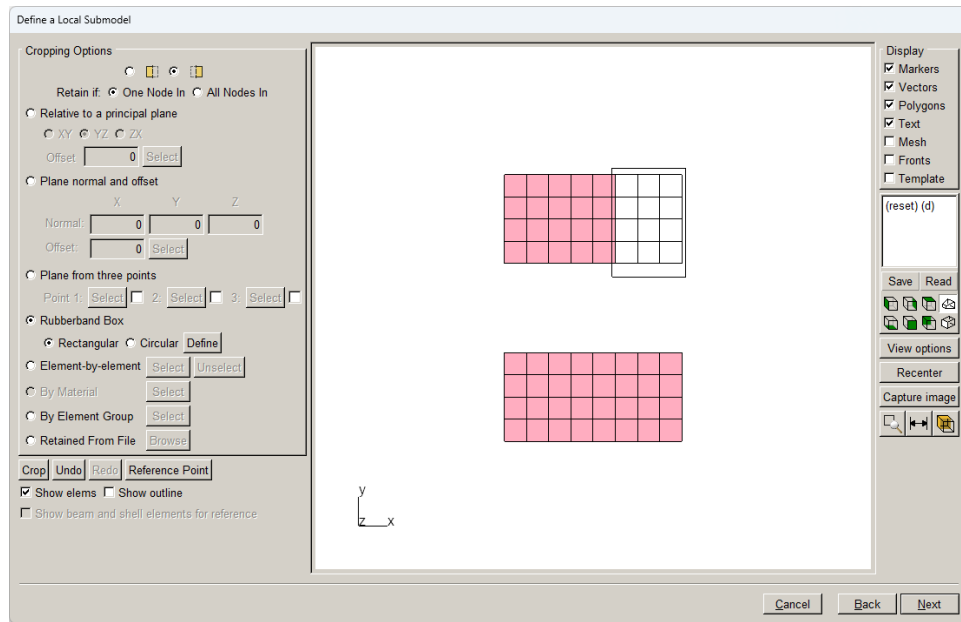


Figure 5.50 FRANC3D third submodel extraction for plate model.

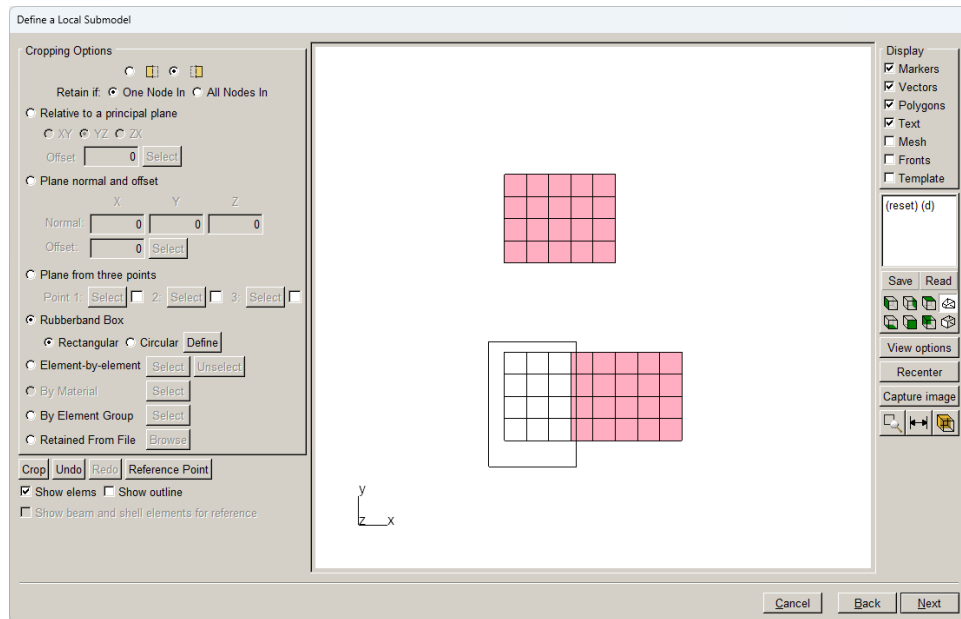


Figure 5.51 FRANC3D fourth submodel extraction for plate model.

Select **Next** after selecting **Crop**. Save the local and global files using the default names. The Select Retained BC Surfaces dialog is displayed, Fig 5.52. There are no boundary conditions on these two portions, so select **Next**, set the units, and then **Finish**.

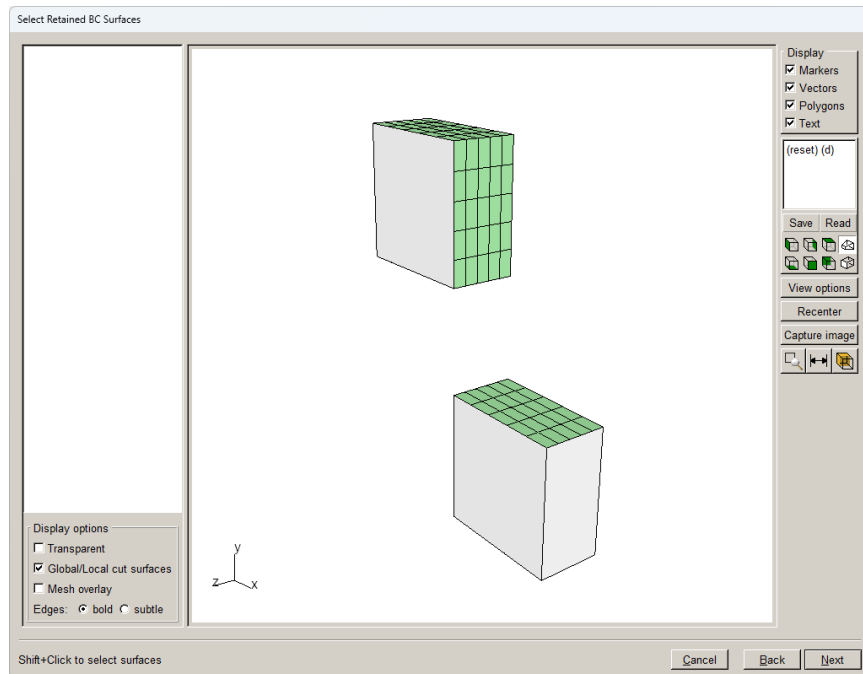


Figure 5.52 FRANC3D Select Retained BC Surfaces dialog.

The model will be displayed in the FRANC3D main model window, Fig 5.53. The cut-surface mesh facets for both portions are retained. The next step is to insert edge-cracks into the two portions.

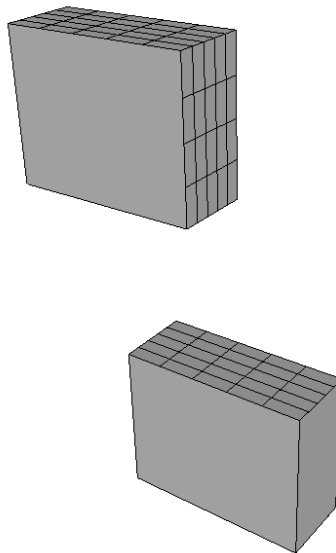


Figure 5.53 FRANC3D model with two portions of the full model.

5.9.2 Step 2: Insert Multiple Cracks

From the FRANC3D menu, we select **Cracks** and **Multiple Flaw Insert**. The dialog shown in Fig 5.54 is displayed. Use the **Add** button to define the two edge cracks. Fig 5.55 shows the dimensions for both edge cracks that will be inserted. Figs 5.56 and 5.57 show the location, orientation, and mesh template for crack #1. Figs 5.58 and 5.59 show the location, orientation, and mesh template for crack #2.

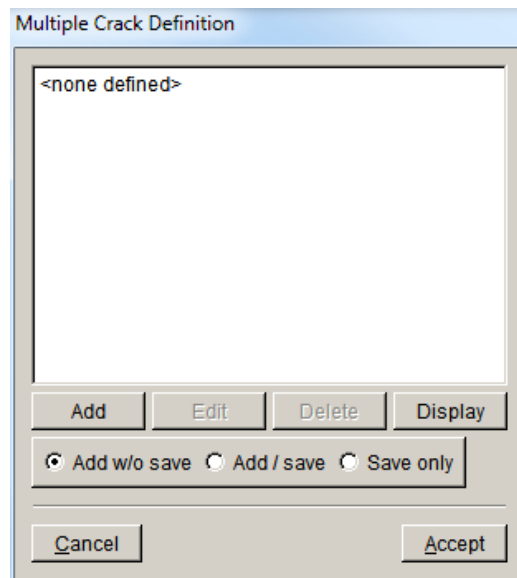


Figure 5.54 Multiple crack definition dialog.

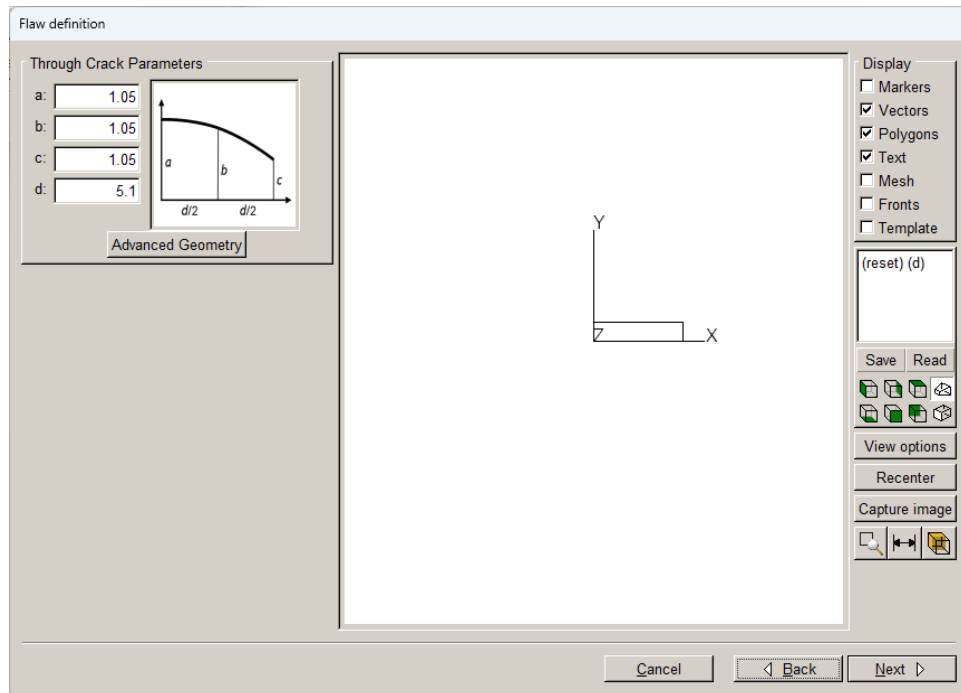


Figure 5.55 FRANC3D edge through crack definition.

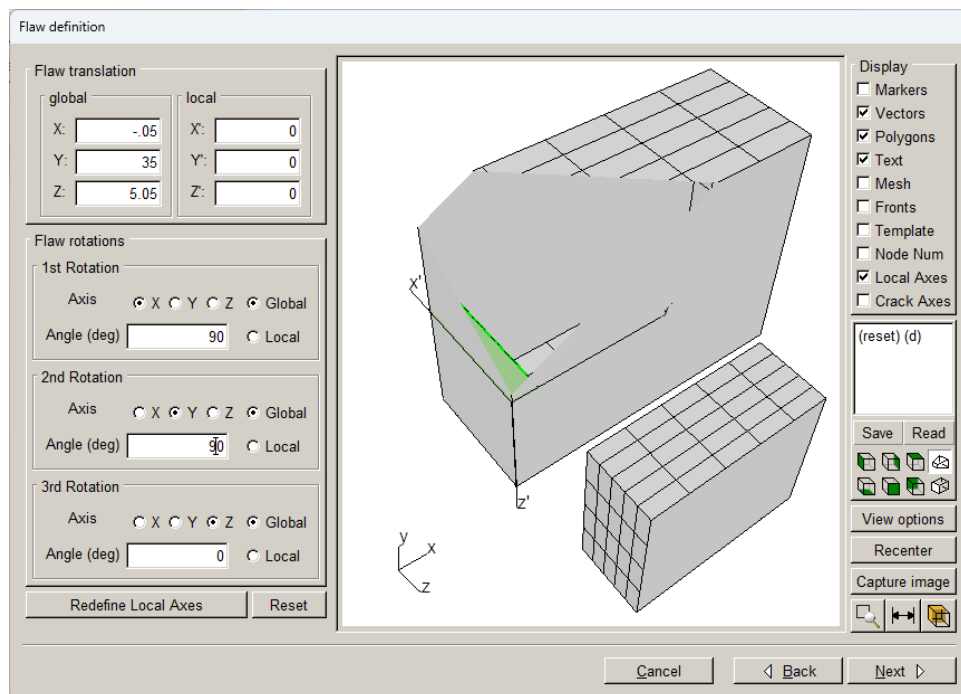


Figure 5.56 FRANC3D edge through crack location and orientation – crack #1.

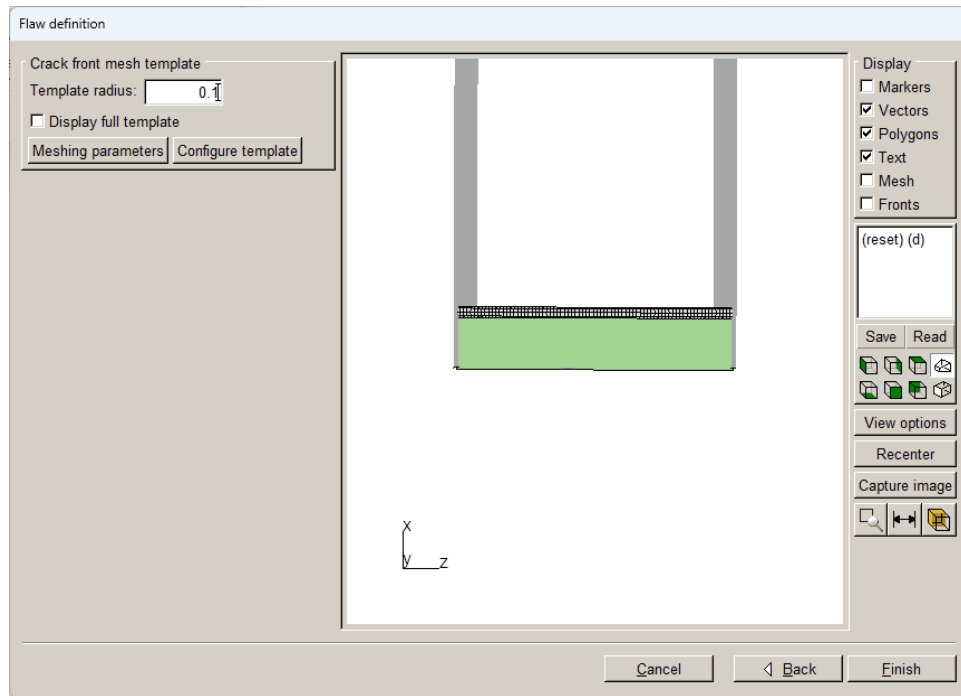


Figure 5.57 FRANC3D edge through crack mesh template – crack #1.

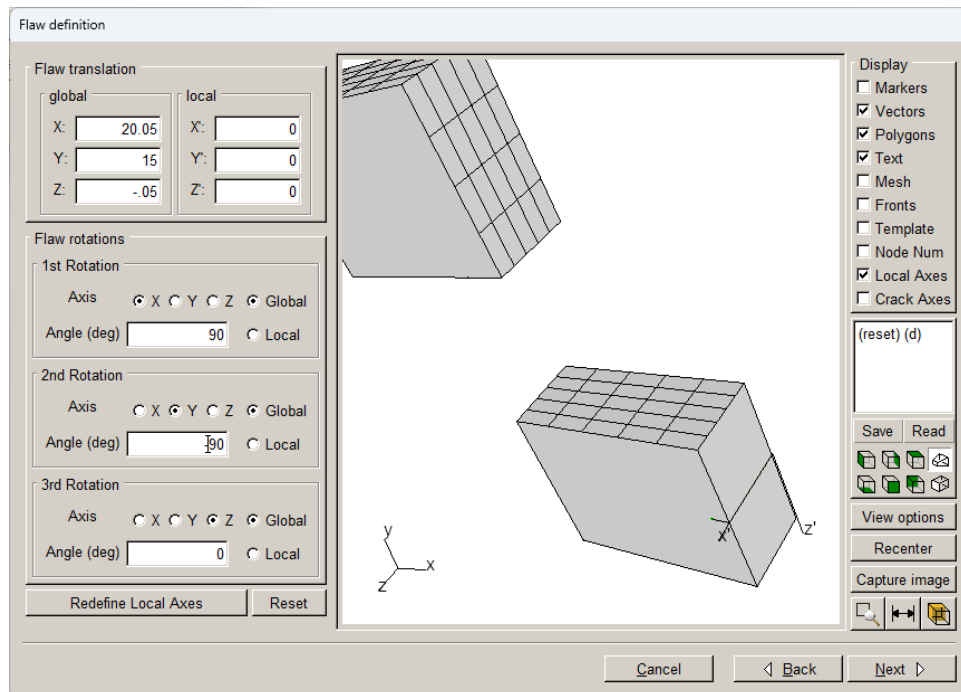


Figure 5.58 FRANC3D edge through crack location and orientation – crack #2.

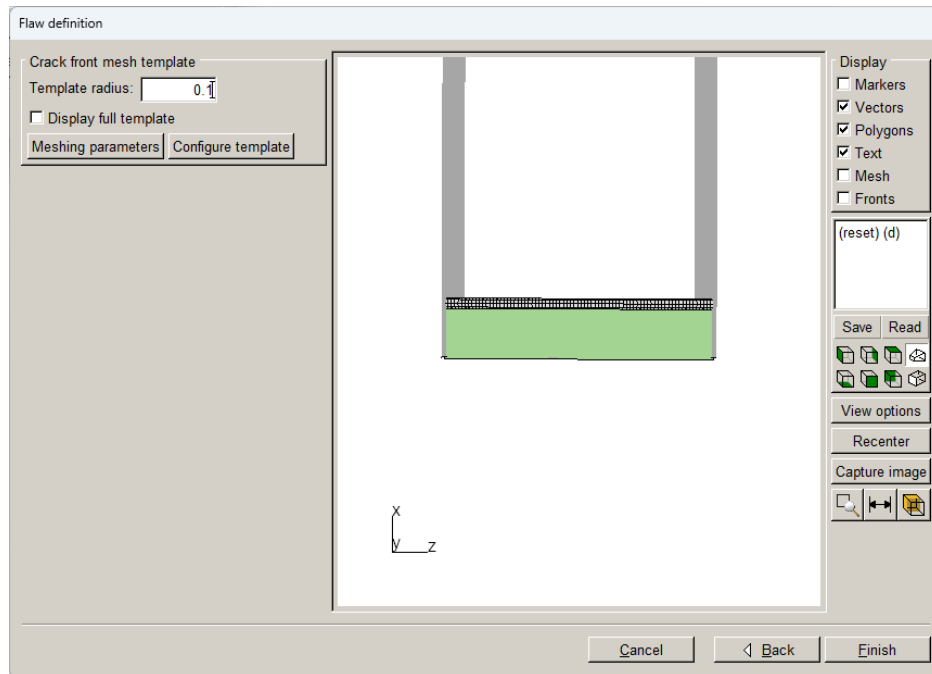


Figure 5.59 FRANC3D edge through crack mesh template – crack #2.

After defining both edge cracks, the **Multiple Crack** dialog shows the two crack names, Fig 5.60. Click on **Display** in this dialog to show the cracks in the model, Fig 5.61. Click **Accept** (in the dialog in Fig 5.58) to finish the insertion and remeshing. The resulting model is shown in Fig 5.62.

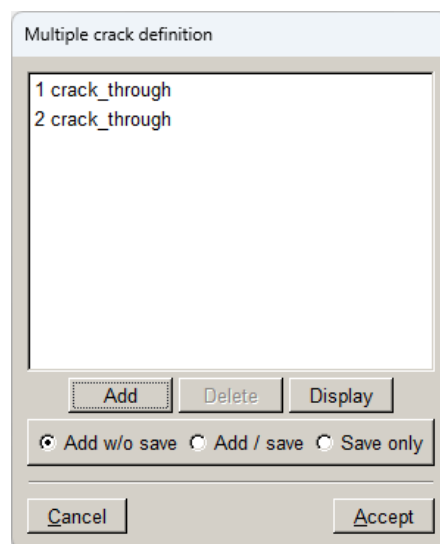


Figure 5.60 FRANC3D multiple crack insertion dialog.

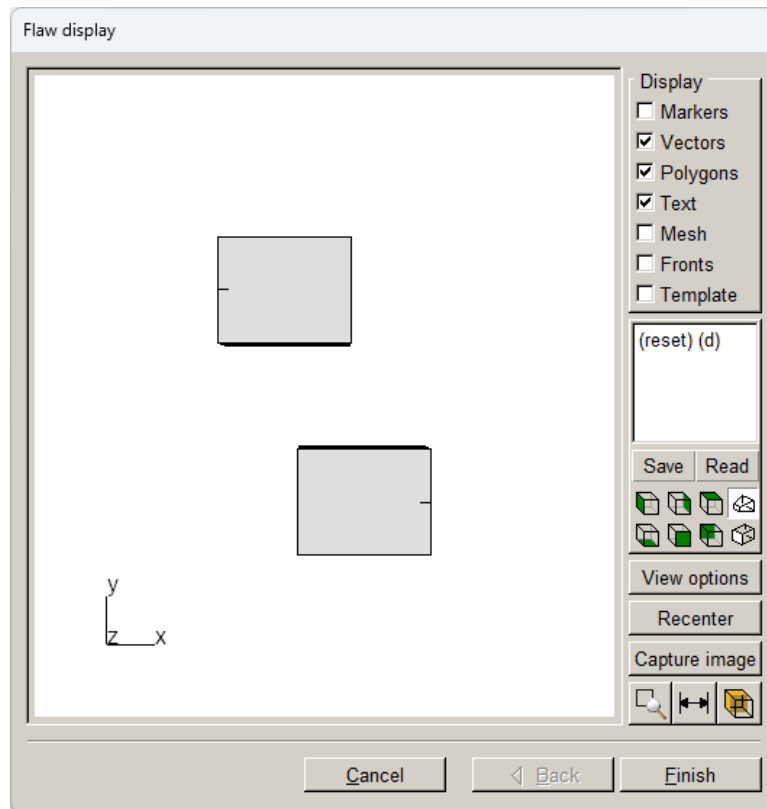


Figure 5.61 FRANC3D multiple crack insertion display.

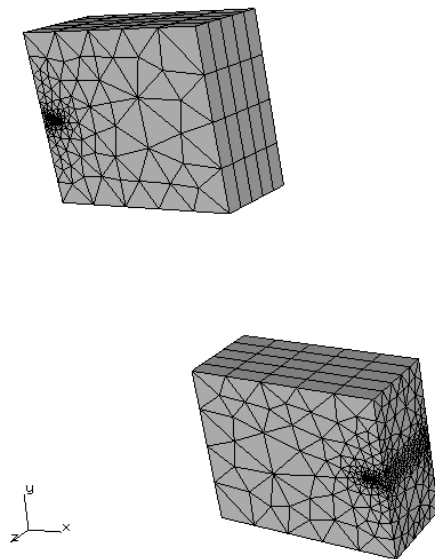


Figure 5.62 FRANC3D model with edge cracks in two separate model portions.

5.9.3 Step 3: Crack Analysis

Follow the same procedure as in the previous tutorials to run the static crack analysis. Select **Static Crack Analysis** from the FRANC3D menu, set the file name, make sure the local + global connection is set, and run the analysis.

Once the analysis has finished, select **Compute SIFs** from the FRANC3D menu. The SIF dialog allows you to select specific crack fronts to display SIF plots. The front is identified on the left by the red **A** and **B**. Fig 5.63 shows the SIF for crack #1, and Fig 5.64 shows the SIF for crack #2. If your SIF curves are not as smooth as the curves in Figs 5.63-64, try changing the meshing parameters; you can try the parameters shown in Fig 5.65 if you are using ABAQUS.

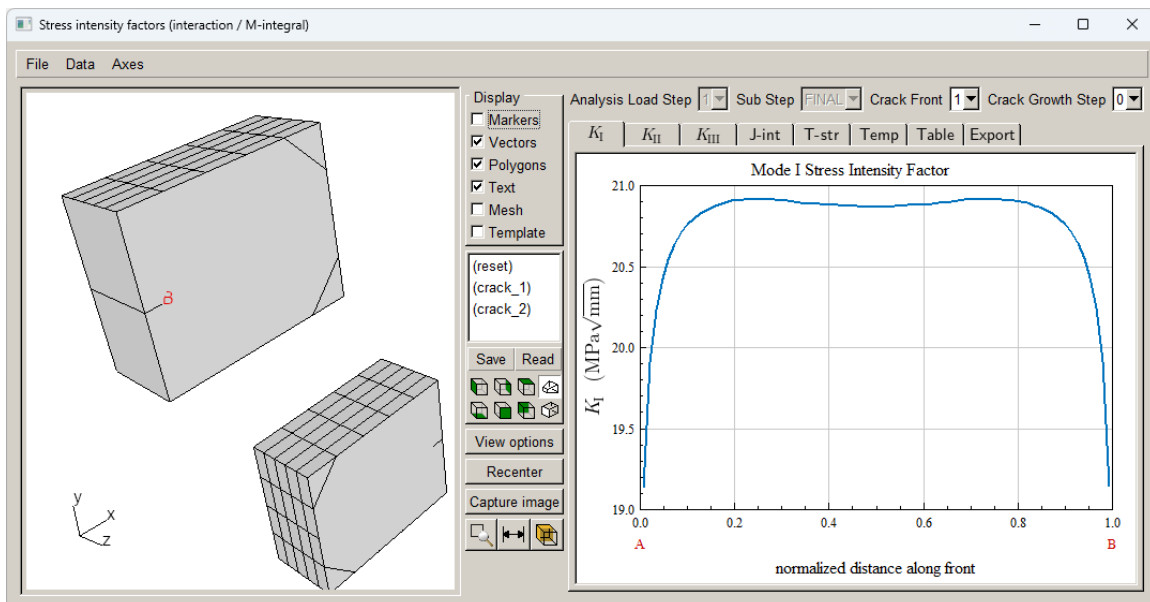


Figure 5.63 Mode I SIF for crack #1.

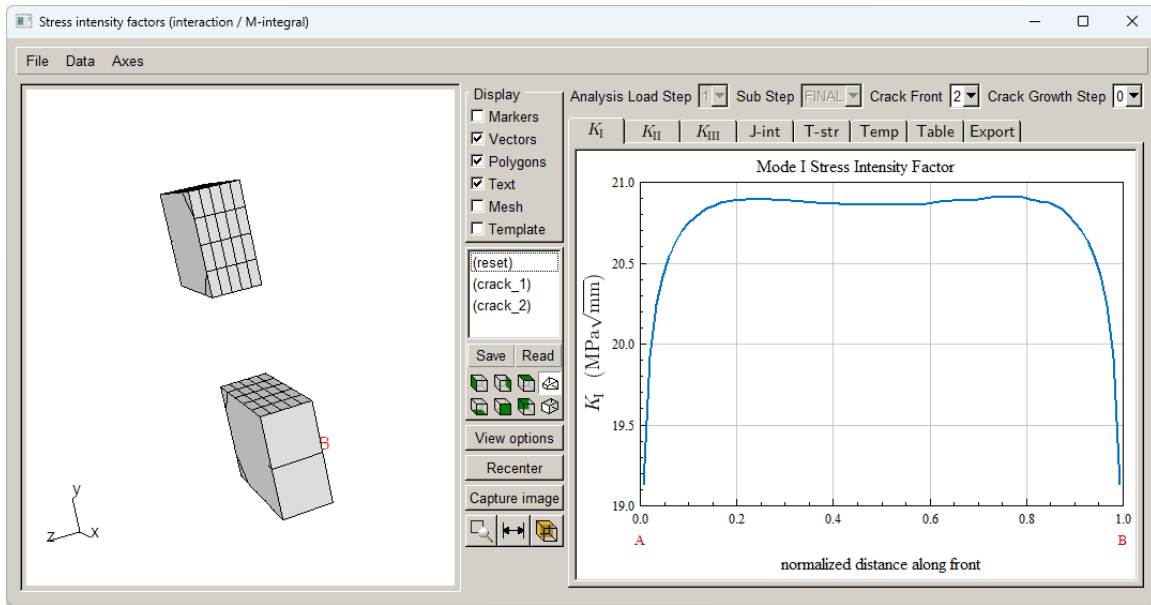


Figure 5.64 Mode I SIF for crack #2.

The cracks can be propagated as described previously; see Section 5.6 for example. Fig 5.66 shows example crack growth. Subsequent crack growth is left as an exercise for the reader.

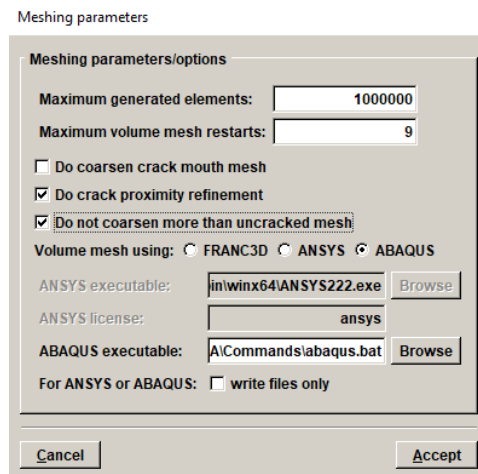


Figure 5.65 Meshing parameters dialog.

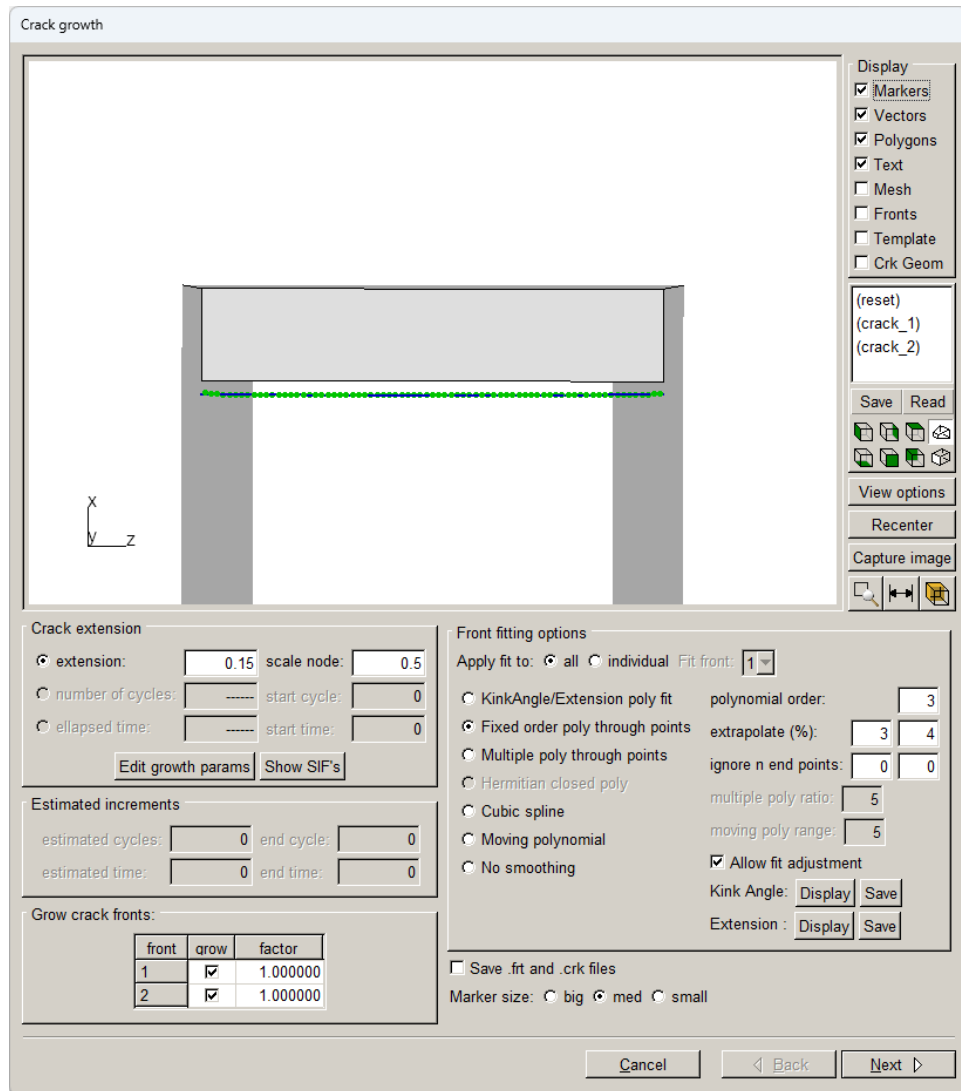


Figure 5.66 Crack growth dialog.

6.0 Tutorial 6: Computing Fatigue Life Cycles

In Tutorial 6, we describe the **Fatigue** module in more detail by presenting an example simulation of a surface crack in a plate under cyclic loading. We can compare the computed number of cycles to experimental values from: *Manu, C. (1980) Three-dimensional finite element analysis of cyclic fatigue crack growth of multiple surface flaws, PhD Thesis, Cornell University.*

The plate and initial crack dimensions are shown in Fig 6.1. The loading is uniaxial tension. The physical testing produced the crack growth seen in Fig 6.2. The crack grows through the plate, eventually becoming a center-through crack with two crack fronts.

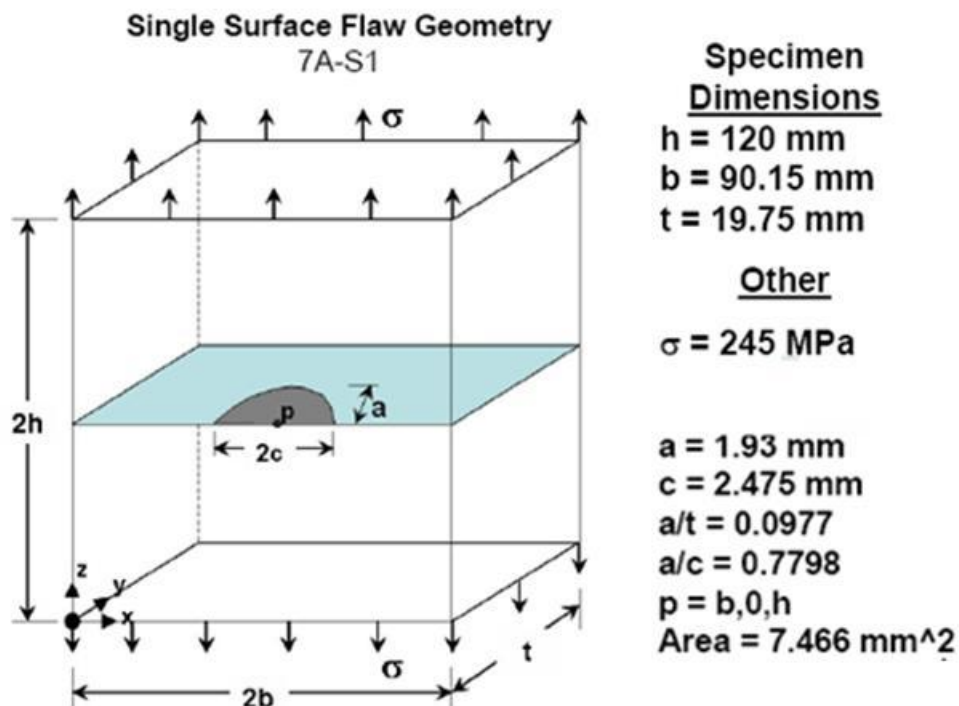


Figure 6.1 Single surface crack in plate under tension

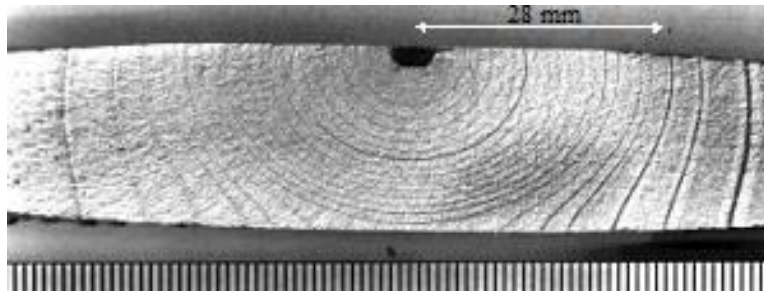


Figure 6.2 Single surface crack in plate under tension (from Manu, 1980).

Fig 6.3 shows the simulated crack fronts. The fatigue cycles are computed using the resulting SIF history and appropriate crack growth rate material data. For units of MPa and mm, the Paris parameters for this material include $C = 1.14e-15$, $m = 3.85$, $\Delta K_{th} = 121 \text{ MPa}\cdot\text{mm}^{1/2}$, and $K_{Ic} = 6949 \text{ MPa}\cdot\text{mm}^{1/2}$.

Fatigue cycles can be compared to previous simulations and experimental observations, Fig 6.4, from: Barlow, K., 2008, *Propagation Simulation of Multiple Cracks in a Tensile Specimen*, International Conference on Fatigue Damage of Structural Materials VII, Hyannis, MA.

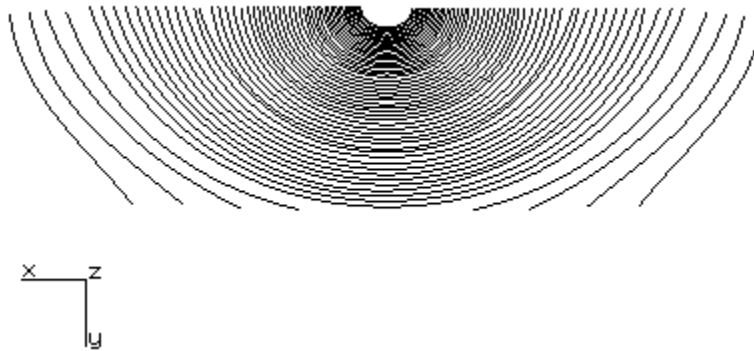


Figure 6.3 FRANC3D simulated crack growth for single surface crack in a plate in tension.



Crack Growth Rate through the Thickness Single Surface Flaw

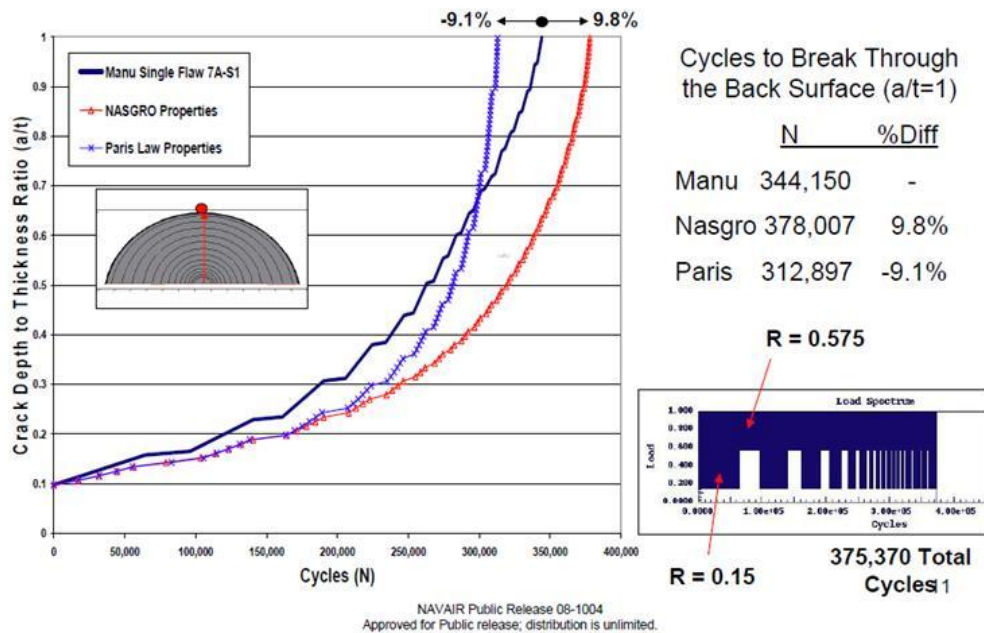


Figure 6.4 Crack growth cycles for single surface crack in plate under tension.

6.1 Step 1: Perform Crack Growth

ABAQUS, ANSYS and NASTRAN models of the plate are available. We do not differentiate between them in this tutorial.

The first step is to import the FE model of the plate into FRANC3D. We import and divide the full model using the Submodeling tool, Fig 6.5. There are no boundary conditions on the resulting local model. The FE units are mm and MPa; temperature is not considered.

Once the local model is created, it is displayed in the FRANC3D main modeling window with the cut-surface facets retained, Fig 6.6.

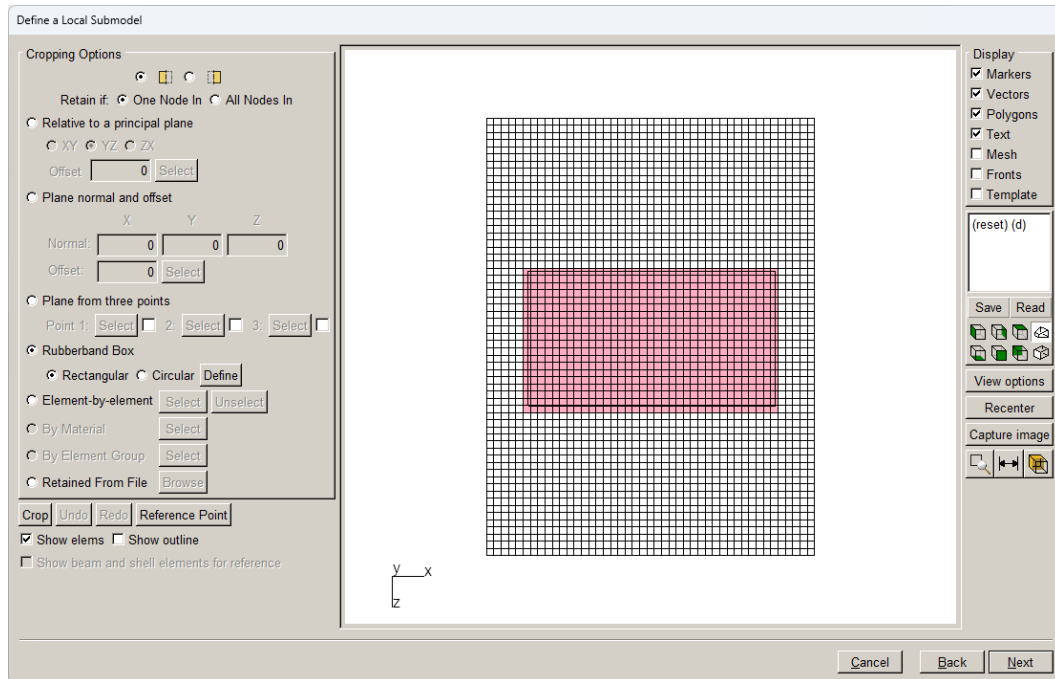


Figure 6.5 Local and global model portions.

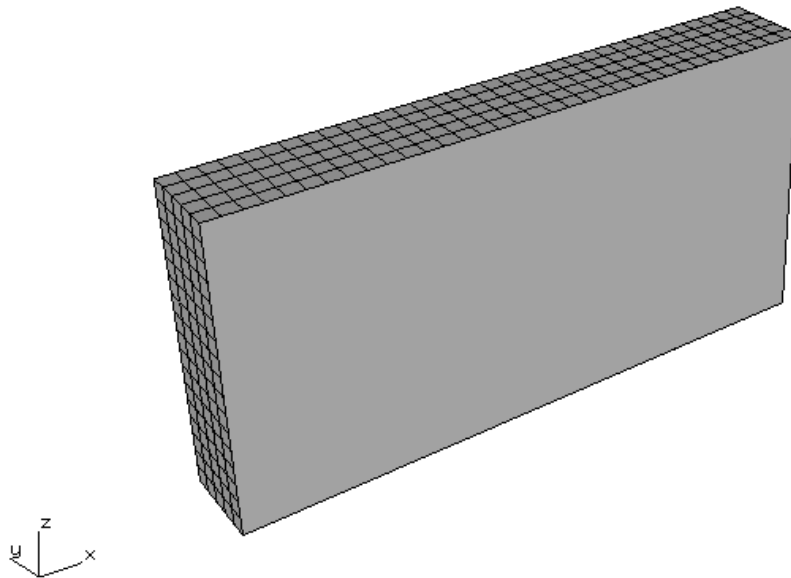


Figure 6.6 Local model portion.

The semi-elliptical surface crack dimensions are defined in Fig 6.1. The location and orientation are shown in Fig 6.7. We use a template radius of 0.15 and turn off the Do coarsen crack mouth mesh flag in the Meshing Parameters dialog to produce a more refined mesh near the crack front.

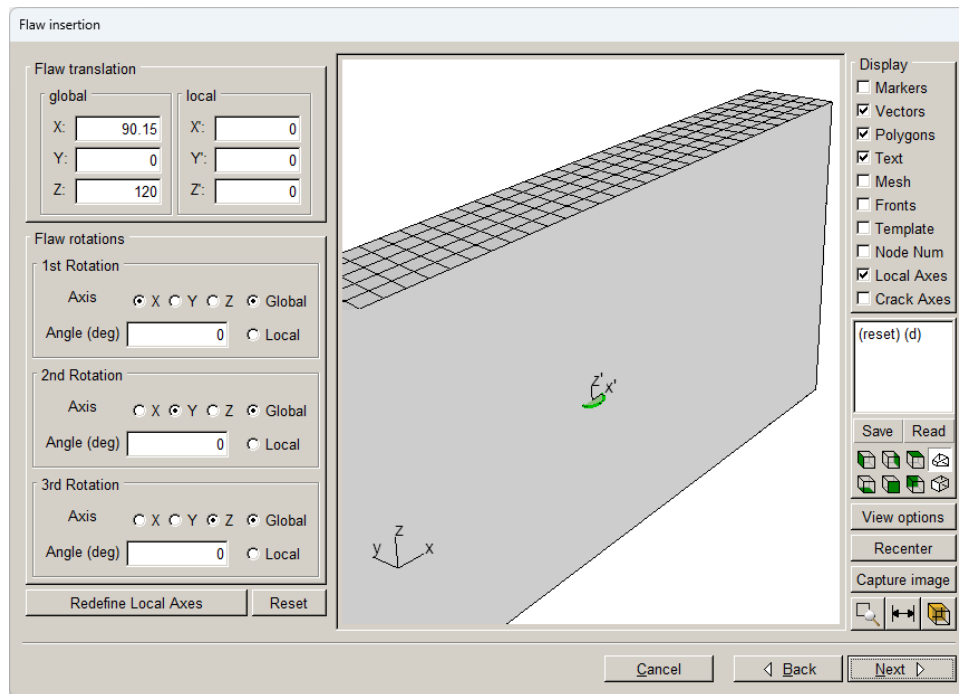


Figure 6.7 Surface crack location and orientation.

A static crack analysis is completed after crack insertion is completed. When the analysis has finished, SIFs are computed. The SIFs can be compared with the Raju-Newman handbook solution, Fig 6.8. The solutions differ by less than 1% for most of the crack front; the ends and midpoint differ by less than 2%.

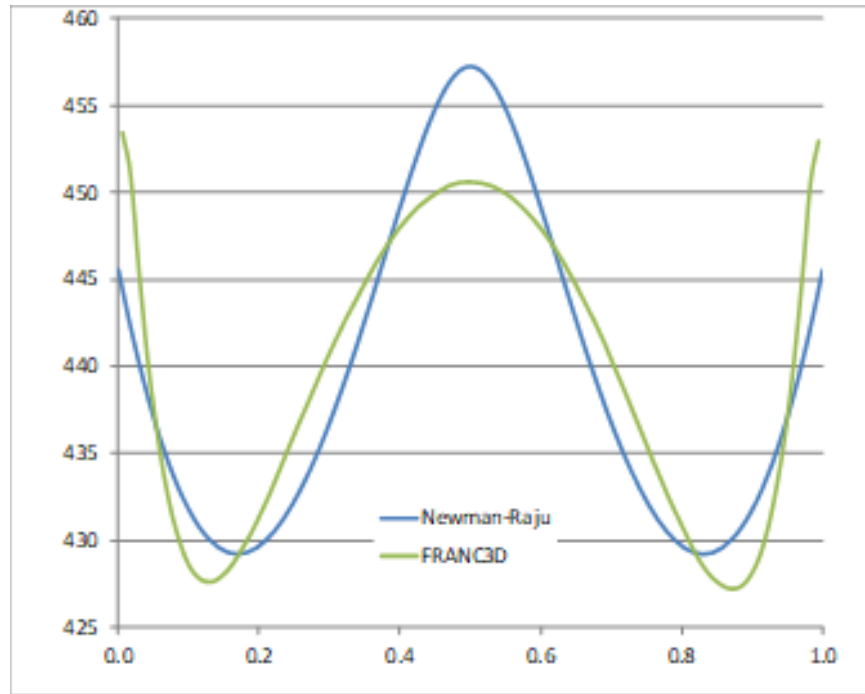


Figure 6.8 Mode I SIF for initial semi-elliptical surface crack.

Crack growth is simulated using a relatively large number of steps with small increments of growth. The growth is based on constant amplitude loading with the Paris crack growth rate material data given above. Variable amplitude loading options are discussed in Step 3.

Using the **Crack Growth Analysis** menu option, the crack is propagated automatically. The series of dialog boxes in Figs 6.9-13 show the growth model and plan settings. Subcritical crack growth is chosen, Fig 6.9. The kink angle default settings are used, Fig 6.10. The fatigue growth model consists of a simple cyclic load event and Paris growth model, Figs 6.11-13.

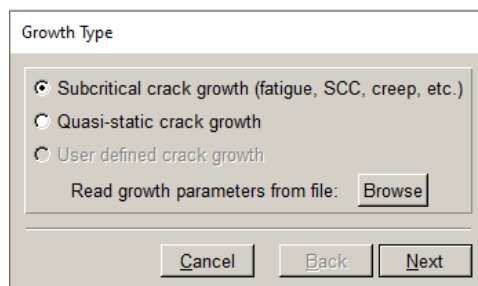


Figure 6.9 Growth type dialog.

Kink angle model

Kink angle model

☒ Max tensile stress (MTS) $\max(K_I^r(\theta))$

☐ Max shear stress (MSS) $\max\left(\sqrt{(\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2}\right)$

☐ Generalized stress $\max(\text{MTS}, \text{MSS})$

☐ Strain energy release rate $\max\left(K_I^r(\theta)^2 + (\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2\right)$

☐ Planar $\theta = 0$

☐ User defined model

☐ Kink angle limit Maximum kink angle (deg):

Non-proportional kink angle parameter ☒ ΔK ☐ $K_{\max}$

Mixed mode eta factors η_{II} : η_{III} :

Crack growth resistance

☐ Anisotropic toughness

Figure 6.10 Kink angle dialog.

Subcritical Growth Parameters

Units used in FE model

stress: MPa length: mm temp: C time: sec

Crack Growth Load Schedule

Crack Growth Rate Model

Mixed-mode equivalent K

☒ $K^{\text{equiv}} = K_I$ ☐ $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{II}K_{II})^2 + (\gamma_{III}K_{III})^2}$

γ_{II} : γ_{III} :

sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective Delta K

☒ $\Delta K_{\text{eff}} = K_{\max} - K_{\min}$ ☐ $\Delta K_{\text{eff}} = K_{\max} - \max(K_{\min}, 0)$

☐ Use load interaction model

Integration Options

☒ Accelerated counting ☐ Constant K for time integration

Figure 6.11 Fatigue growth parameters dialog.

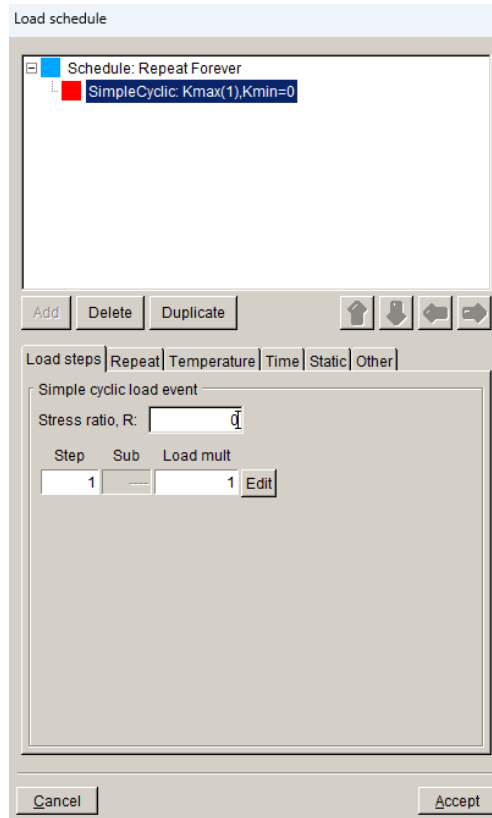


Figure 6.12 Fatigue growth parameters dialog.

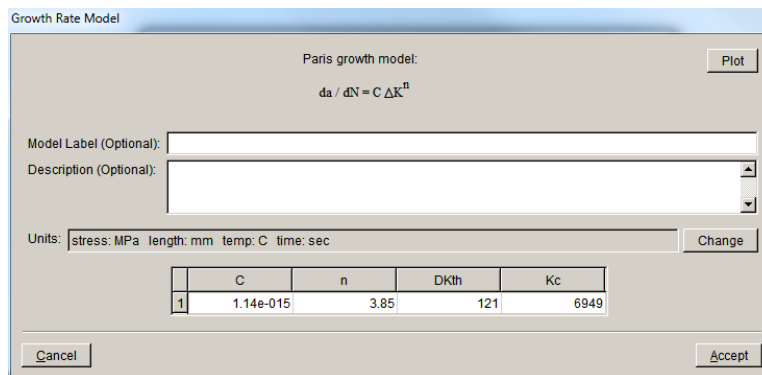


Figure 6.13 Paris growth model dialog.

Fig 6.14 shows reasonable crack front fitting options that can be used in the Fitting & Template Parameters dialog, Fig 6.15 – left side. Fig 6.15 (right side) shows the Growth Plan dialog with the stopping criteria set to maximum number of cycles; you can choose other or additional stopping criteria.

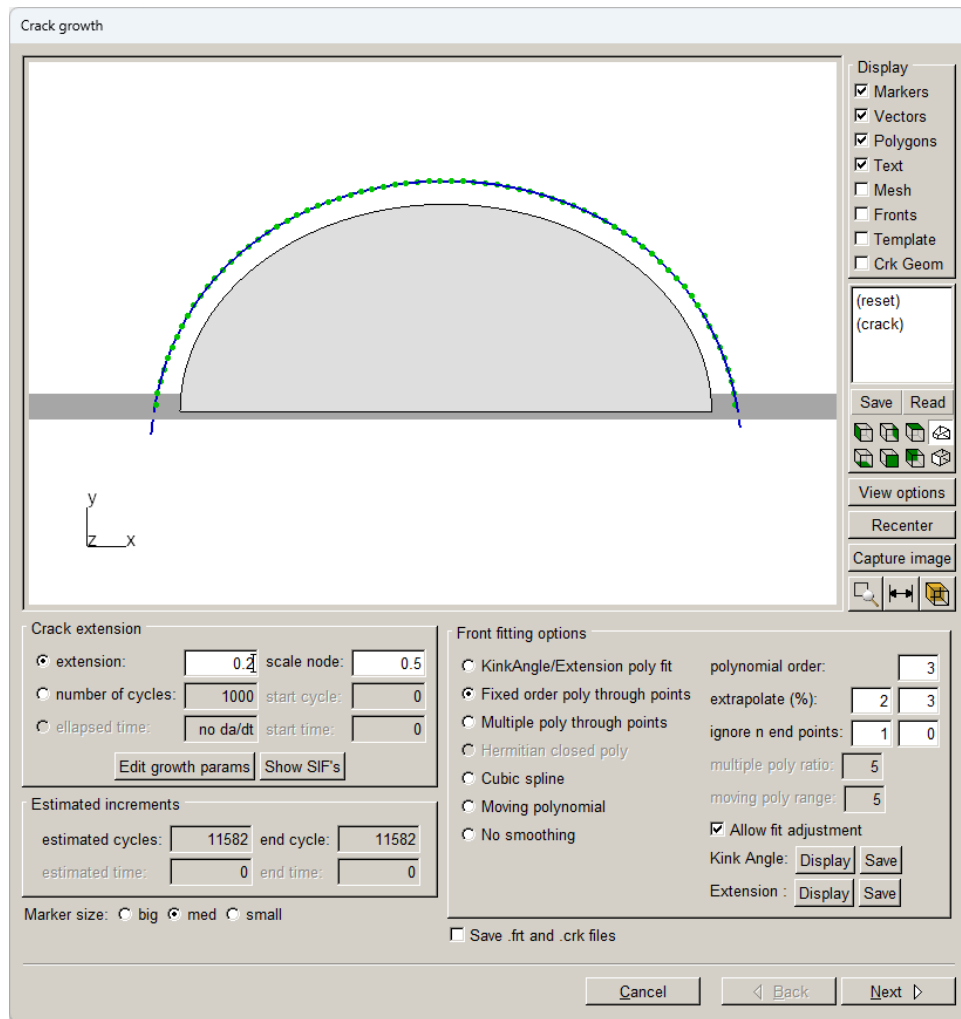


Figure 6.14 Crack front fitting dialog.

Once the growth plan is set, the automatic crack growth simulation can be completed. The stopping criteria of 400,000 cycles, which was based on the numbers from the experiment will never be reached (discussed below). At some point, the crack growth simulation will stop; this might happen as the crack breaks through the back surface.

Fig 6.16 shows the Mode I SIF history for a path through the first 48 crack fronts. The fatigue life computations are described in the next step. Note that the crack front breaks through the back surface of the plate on the next step of crack growth, leading to two crack fronts.

Fitting & Template Parameters

Front Fitting Options

☐ KinkAngle/Extension Poly Fit

polynomial order:

☒ Fixed Order Poly Through Points

extrapolate (%):

☐ Multiple Poly Through Points

ignore n end points:

☐ Hermitian Closed Poly

multiple poly ratio:

☐ Cubic Spline

moving poly range:

☐ Moving Polynomial

☒ Allow fit adjustment

☐ No Smoothing

Flaw Template

☒ use crack-front template

Template radius set as: ☒ absolute value ☐ % of crack increment

Template Radius:
 Front:
☐ Set All Fronts

☐ Simple Intersections Only

Extension

☒ distance extension

scale node:

☐ cycles extension

☐ time extension

Growth Plan

Growth Termination Criteria

☒ Completed load schedule

☒ Critical stress intensity factor

☒ Threshold stress intensity factor

☐ Crack front extension failure

☐ Maximum number of growth steps

Steps:

☒ Maximum number of load cycles

Cycles:

☐ Maximum elapsed time

Time:

☐ Maximum crack depth

Depth:

Specified Crack Growth Steps

☒ Constant Crack Growth Increments

Crack Growth Step Size =

☐ Linearly Increasing/Decreasing Crack Growth Increments

First Crack Growth Step Size =
 Change in Step Size at Each Step =

☐ User Defined Crack Growth Increments

Figure 6.15 Fitting and growth plan dialogs.

Figure 6.16 Mode I SIF history.

190

6.2 Step 2: Compute Fatigue Crack Growth Cycles

The crack growth in Step 1 was performed using a constant amplitude loading with Paris fatigue parameters. When we select **Fatigue** → **Fatigue Life Predictions** from the FRANC3D main menu, the dialog shown in Fig 6.17 is displayed. The cycles versus crack growth step number are plotted by default.

The number of cycles is computed at each step of growth and stored in the FRANC3D database; the number of cycles is the average of the values computed at each mid-side node along the crack front. See Section 9 of the Reference document for additional information.

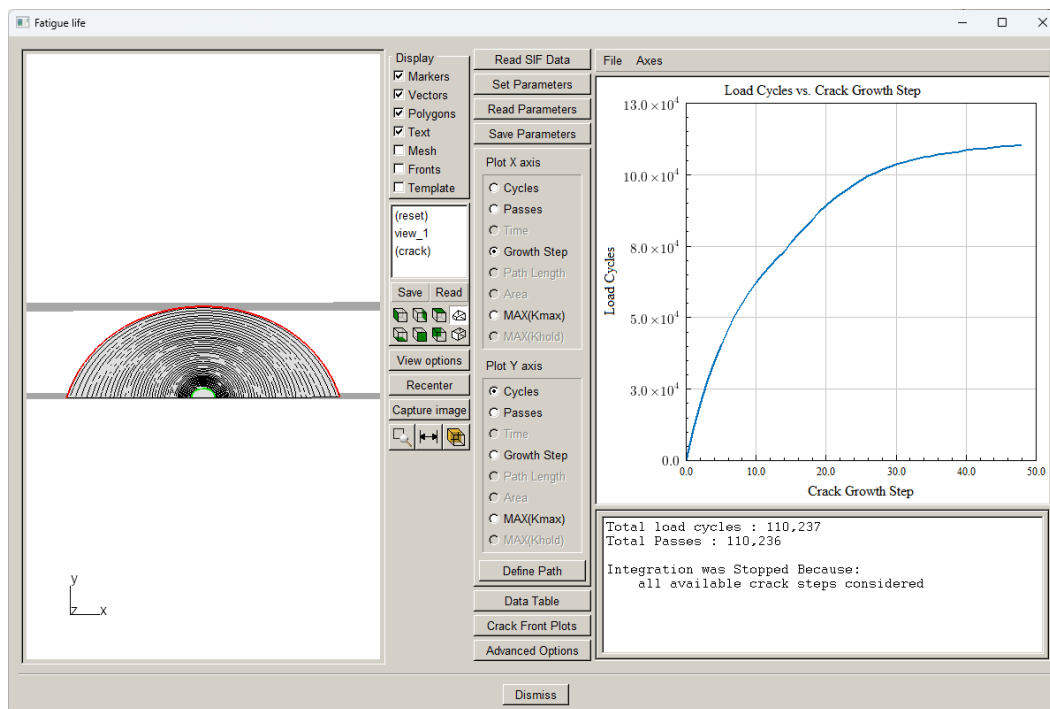


Figure 6.17 Mode I SIF history – cycles vs crack step number.

The Fatigue Life dialog allows us to plot cycles versus crack length, Fig 6.18-19, where the length might be the crack length measured on the model surface; this length was measured

during the experiments. Regardless of the path chosen, the number of cycles is consistent, assuming the path goes through all crack fronts.

The number of cycles is $\sim 110K$ for the current crack, where the front is close to the back surface of the plate but has not yet broken through; there is about 0.5 mm between the deepest point on the crack front and the back surface. This cycle-count is based on constant amplitude loading with $R=0$, which is not representative of the experiment.

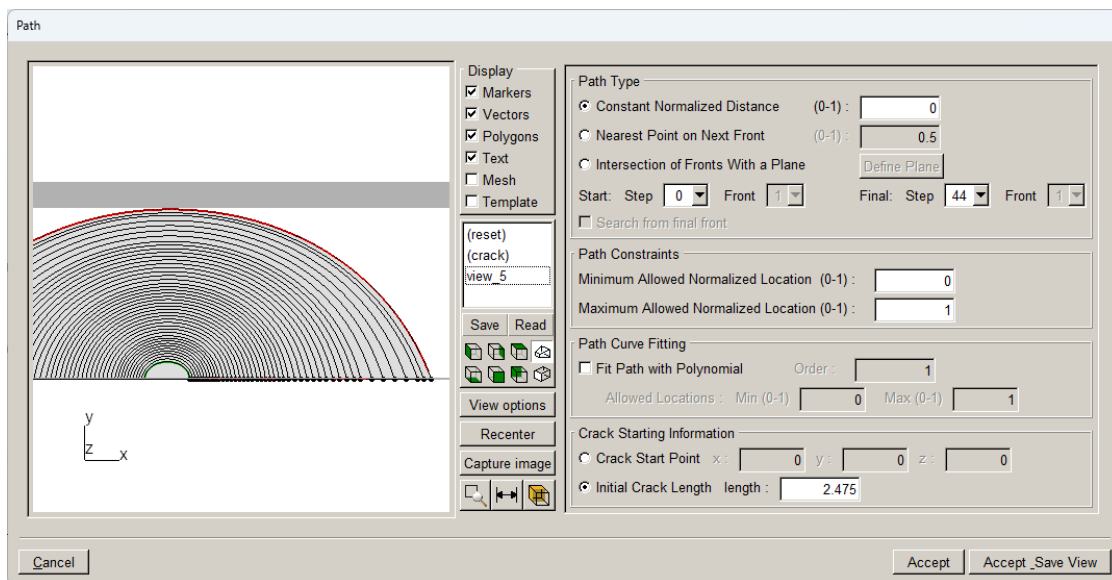


Figure 6.18 Mode I SIF history – define a path near the model surface.

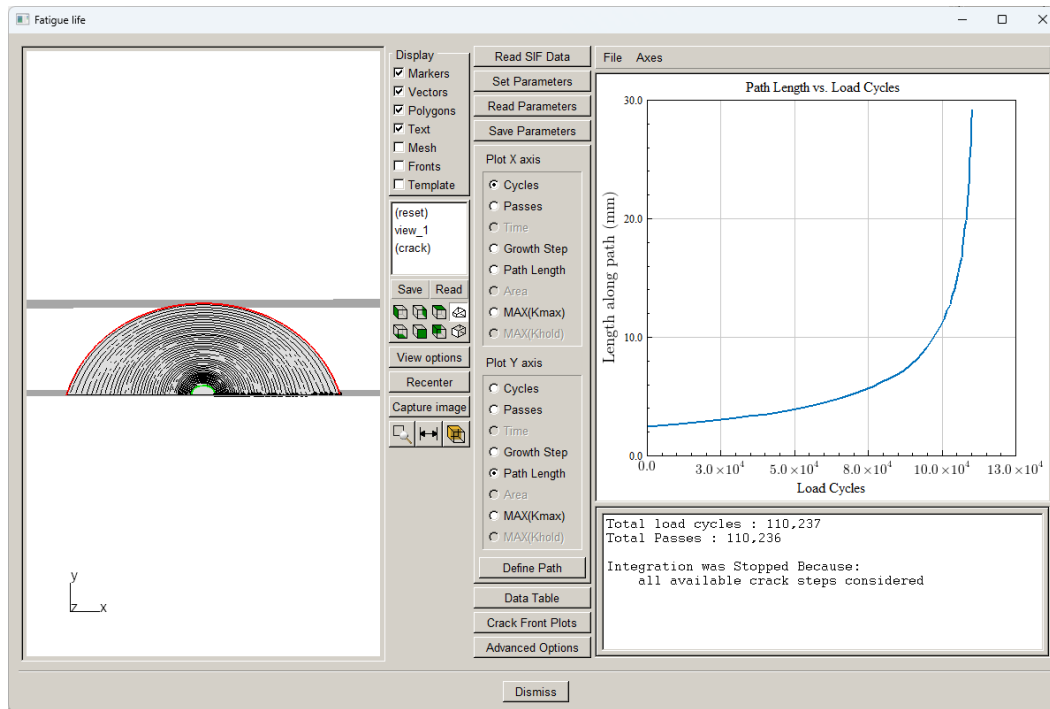


Figure 6.19 Mode I SIF history – cycles vs crack length.

6.3 Crack Growth and Fatigue Options

FRANC3D allows quasi-static and subcritical crack growth. Quasi-static growth has been used in earlier tutorials and is described in Section 6.5.2 of the Reference document. Subcritical growth can include cyclic fatigue, stress corrosion and time dependent crack growth.

For cyclic fatigue, the load schedule consists of load events. Load events include: 1) simple cyclic, 2) non-proportional cyclic, 3) transient, 4) spectrum, 5) hold (time) event, 6) dynamic pairing, and 7) sub-schedules, Fig 6.20.

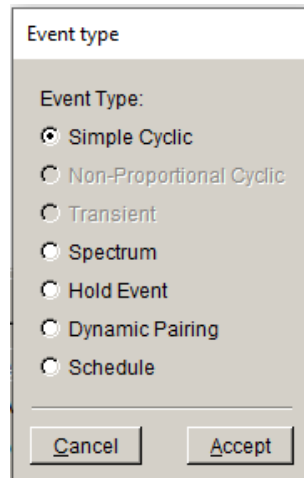


Figure 6.20 Cyclic fatigue load schedule events.

6.3.1 Constant Amplitude Loading

Simple cyclic constant amplitude loading was used in Step 1 and 2 above, where K_{max} is the computed SIF, and $K_{min}=0$ as we set $R=0$ ($R=K_{min}/K_{max}$). Thus, ΔK is equal to K_{max} .

The computed number of cycles is significantly less than the number measured in the experiment, which is expected as the ΔK value is higher, leading to higher crack growth rates.

If we modify the R value in the Fatigue Life dialog, Fig 6.21, (use the **View/Edit** button for the Crack Growth Load Schedule in Fig 6.11) and set $R=0.15$ (see Fig 6.4), the computed cycles increase to 206,182. We can also set R to 0.575, and in that case the number of cycles is 2,973,207.

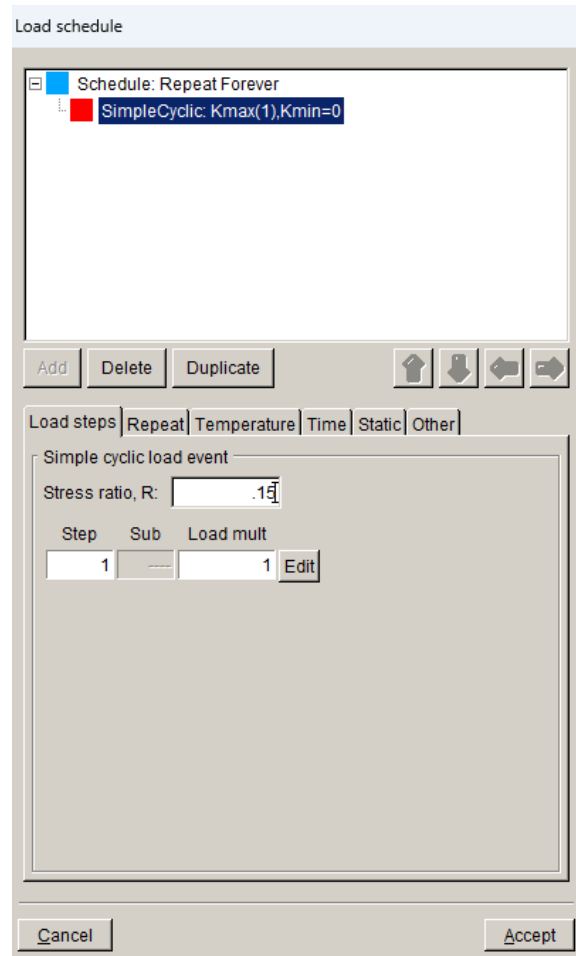


Figure 6.21 Fatigue Life Load Schedule dialog with R set to 0.15.

6.3.2 Variable Amplitude Loading

Constant amplitude loading cannot capture the number of cycles in the experiment, because the experiment did not use a constant amplitude. Fig 6.4 showed an “average” of the actual experimental loading. Before we try the load spectrum with our model, we briefly look at the FRANC3D event types.

A load schedule can include one or many load steps (or load cases). For example, for a jet engine component, a user might define a set of load steps where each step has a different temperature distribution and a different angular velocity; each load step represents a different

time point in a typical flight (e.g., ground idle, takeoff/climb, cruse, maneuver, decent, reverse thrust).

In FRANC3D, a user constructs a ‘Load Sequence’ (or mission profile) that describes a typical mission or flight. For each event in the sequence, an optional load and temperature multiplier can be applied, giving a sequence table such as:

<u>Load_Step</u>	<u>Load_Mult</u>	<u>Comment</u>
1	1.0	Ground idle (cold)
3	1.0	Climb
2	1.0	Initial cruse
4	0.8	Maneuver 1
2	1.1	Second cruse
4	1.2	Maneuver 2
2	1.0	Third cruse
5	1.0	Descent
1	1.0	Ground idle (hot)

FRANC3D computes SIFs for each of the events in the sequence and then steps through those SIFs and finds the ΔK and R for each reversal (e.g., from ‘off’ to ‘ground idle’). For each ΔK and R, FRANC3D finds $da = da/dN(\Delta K, R, T)$, and sums the da's for each event in the sequence to get total da for one pass through the sequence. Note that da is the change in crack length at a point on the front, and each point will have its own da.

Each pass through the sequence represents one flight. FRANC3D passes through the sequence as many times as necessary to predict crack length as a function of number of cycles. One can convert cycles to flights by dividing the cycles by the number of events in the schedule.

One of the possible ‘events’ in the schedule is a spectrum load. The spectrum load option involves a load history that defines the stress level at a (typically) large number of time points. This load history is “counted” using a "rainflow" or other method to produce a list of load

reversals (stress_max and stress_min pairs). This is a counted load spectrum. There are industry standard spectra (e.g., FALSTAFF, TWIST, TURBISTAN and HELIX). We will use the spectrum event in the next subsection.

FRANC3D steps through the spectrum one load reversal at a time. For a crack front, FRANC3D computes the sum of the Ks from one or more analysis load steps (as per the user's input) and multiplies the nominal K by the stress_max and stress_min values to get K_max and K_min for the current load reversal. FRANC3D then computes ΔK and R, and then $da = da/dN(\Delta K, R, T)$.

Another possible ‘event’ is the transient. The transient event involves a series of load steps that mimic a ‘moving’ load or a load that changes in time. An example of this is gear tooth contact; as two gears rotate, the load on a tooth changes in magnitude and location as the tooth comes into and goes out of contact with the opposing tooth. Another example is the changing load on a railway wheel as it comes into and goes out of contact with the track as the wheel revolves. In these cases, the user defines a set of load steps in (time) sequence. FRANC3D computes the SIFs for each load step and computes an increment of crack growth, both extension and direction based on the Kmax and Kmin for the transient event.

6.4 Spectrum Loading for Manu Model

The applied (measured) load spectrum from the Manu experiment was shown in Fig 6.4; this is an “average” of the actual spectrum. It consists of a history of ‘blocks’ of constant amplitude load cycles, where the ‘blocks’ have different R values ($R = \text{minimum load} / \text{maximum load}$) and different numbers of cycles. There are two different R values (0.15 and 0.575), and these values separate the ‘blocks’ of cycles. There are 47 ‘blocks’ and a total of 375,370 cycles defined in the ‘average’ spectrum.

This load spectrum can be read into FRANC3D and used to compute cycles. The load schedule in the Fatigue Life dialog is set to Spectrum instead of Simple Cyclic, Fig 6.22. After selecting the Spectrum event, the **Select a load spectrum** button in the Load Schedule dialog, Fig 6.22 – right panel, can be selected to read the Manu load spectrum data from the file, called *avg.txt*.

The spectrum file is ASCII text with lines of:

min max count

where min and max are load factors and count is the number of cycles in the ‘block’; the total cycle count in *avg.txt* is 374,000. The spectrum is shown in Fig 6.23. Note the max and min can be reversed in the file. After selecting the **Finish** button, Fig 6.23 – right side, there are buttons to **Edit** and **Display** the spectrum. Click on **Display** to show the load versus cycle plot, Fig 6.24.

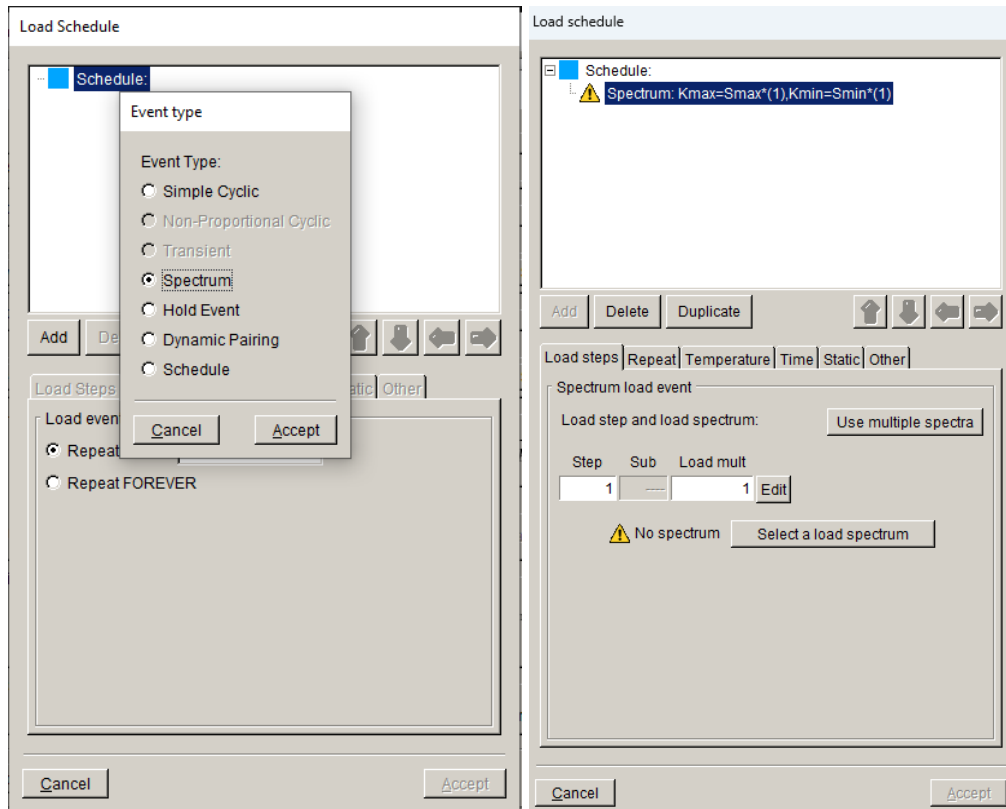


Figure 6.22 Fatigue Life Load Schedule dialog with Spectrum selected.

The spectrum max and min values are used as multipliers to the actual loads. In this model, the maximum applied load is used in the FE model, so the spectrum values are less than or equal to 1.0 to obtain the min and max loads. One could apply a unit load to the FE model and then specify the max and min applied loads in the spectrum.

Spectrum file delimiter

Delimiters

☒ Values separated by spaces or tabs

☐ Values separated by commas

Data preview

☐ Ignore first line of data

```

0.15 1.000000 64500
0.575 1.000000 32350
0.15 1.000000 22610
0.15 1.000000 21100
0.575 1.000000 20590
0.15 1.000000 29280
0.575 1.000000 15320
0.15 1.000000 18680
0.575 1.000000 9570
0.15 1.000000 13000
0.575 1.000000 6600
0.15 1.000000 9350
0.575 1.000000 4850
0.15 1.000000 6600
0.575 1.000000 3600
0.15 1.000000 5800
0.575 1.000000 3420
0.15 1.000000 5580
0.575 1.000000 2950
0.15 1.000000 5460

```

Cancel Back Next

Define spectrum file columns

Select the Max and Min columns

Min	Max	Count
0.15	1.000000	64500
0.575	1.000000	32350
0.15	1.000000	22610
0.15	1.000000	21100
0.575	1.000000	20590
0.15	1.000000	29280
0.575	1.000000	15320
0.15	1.000000	18680
0.575	1.000000	9570
0.15	1.000000	13000
0.575	1.000000	6600
0.15	1.000000	9350
0.575	1.000000	4850
0.15	1.000000	6600
0.575	1.000000	3600
0.15	1.000000	5800
0.575	1.000000	3420
0.15	1.000000	5580
0.575	1.000000	2950
0.15	1.000000	5460

Cancel Back Finish

Figure 6.23 Variable amplitude load spectrum – for Manu *avg.txt*.

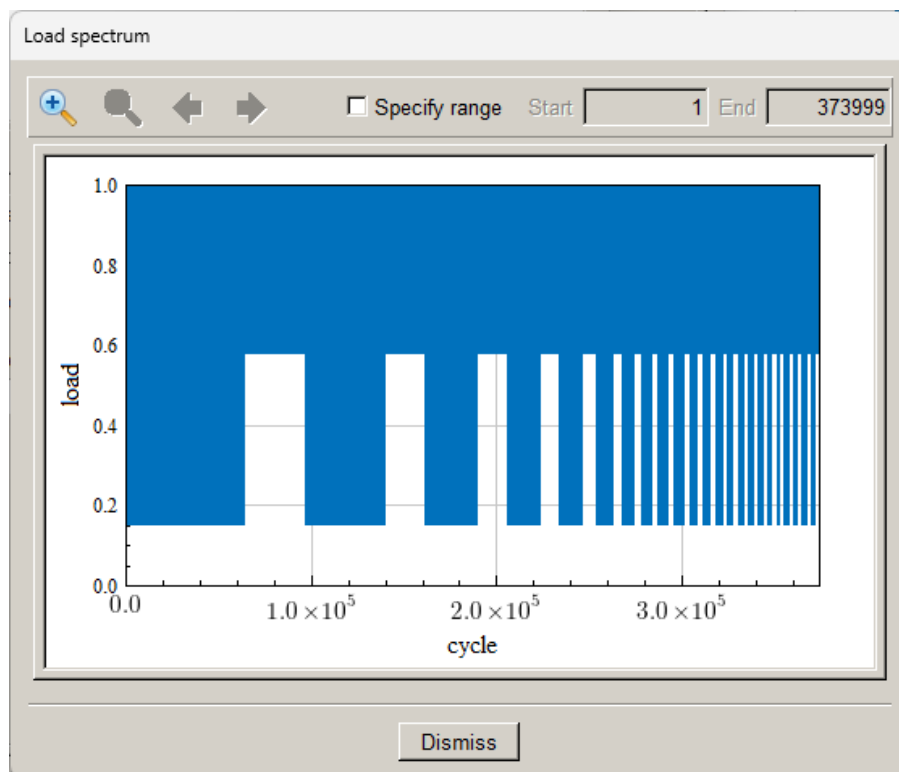


Figure 6.24 Spectrum display.

For this spectrum, the number of cycles computed by FRANC3D is ~298K, Fig 6.25, which is closer to the measured cycles. The spectrum in *avg.txt* is not the exact load spectrum as the data has been simplified a little. The NASGRO model, Fig 6.26-27, which was used by Barlow (2008), provides a better prediction than the Paris model, Fig 6.28.

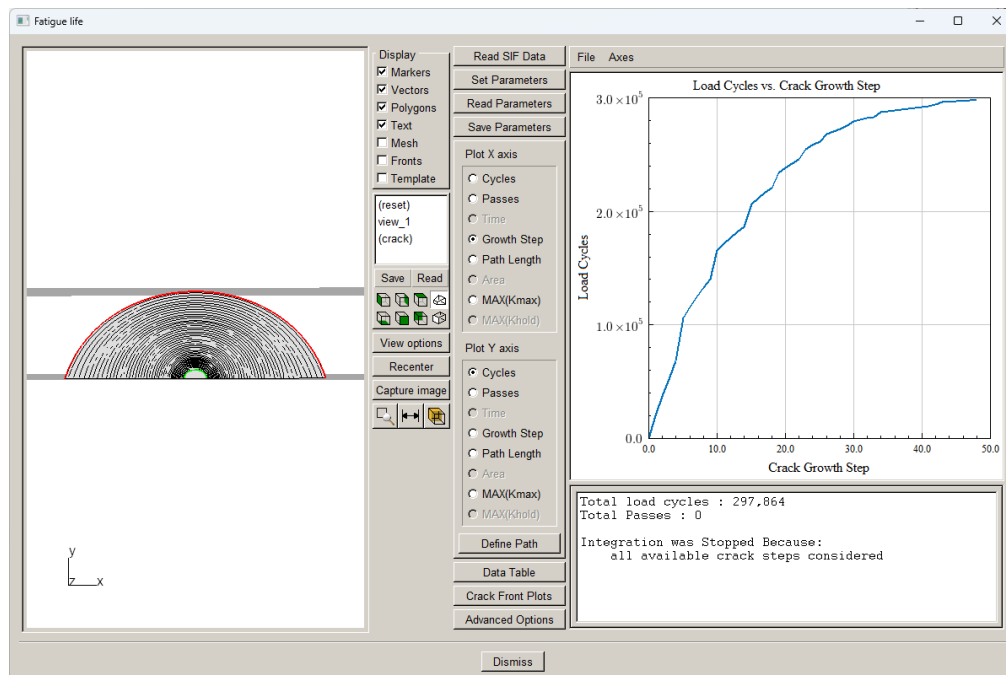


Figure 6.25 Fatigue Life dialog – using the Manu “average” spectrum.

Growth rate model

NASGRO v3 growth model: Plot

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

Model label (optional):

Description (optional):

Units: stress: MPa length: mm temp: C time: sec Change

UTS:	Yield:	K1e:	K1c:	Ak:	Bk:	a0:	C:	K critical: ☒ K1e ☐ K1c ☐ f(K1c,t)
620.55	275.8	9730	6950	1	0.5	0.0381	3.63218e-13	
n:	p:	q:	DK0:	Cth:	alpha:	Smax/Flow:	Thickness:	
3	0.25	0.25	121.625	1	2.5	0.3	0	

Material properties database

Cancel Accept

Figure 6.26 NASGRO V3 annealed 304 stainless steel crack growth rate data.

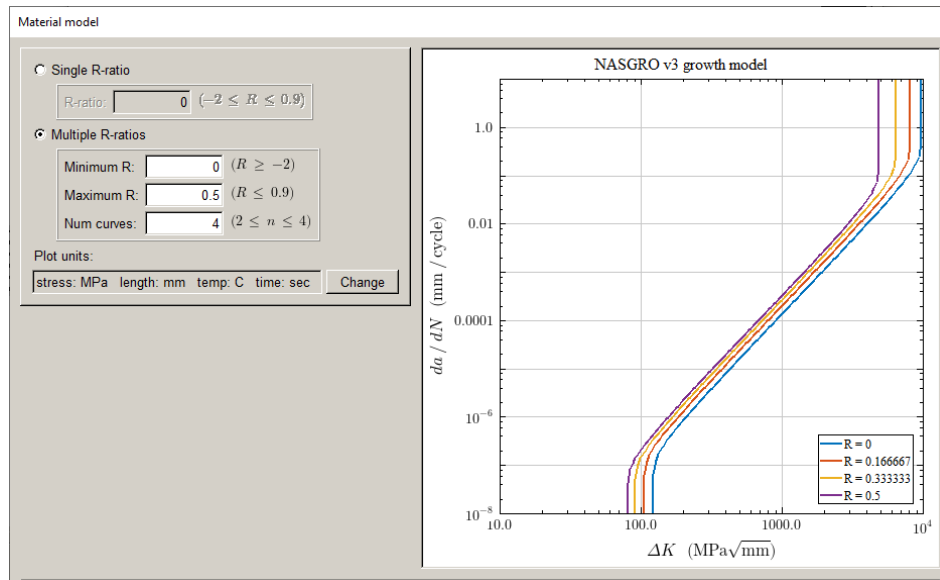


Figure 6.27 NASGRO V3 annealed 304 stainless steel crack growth R-dependence.

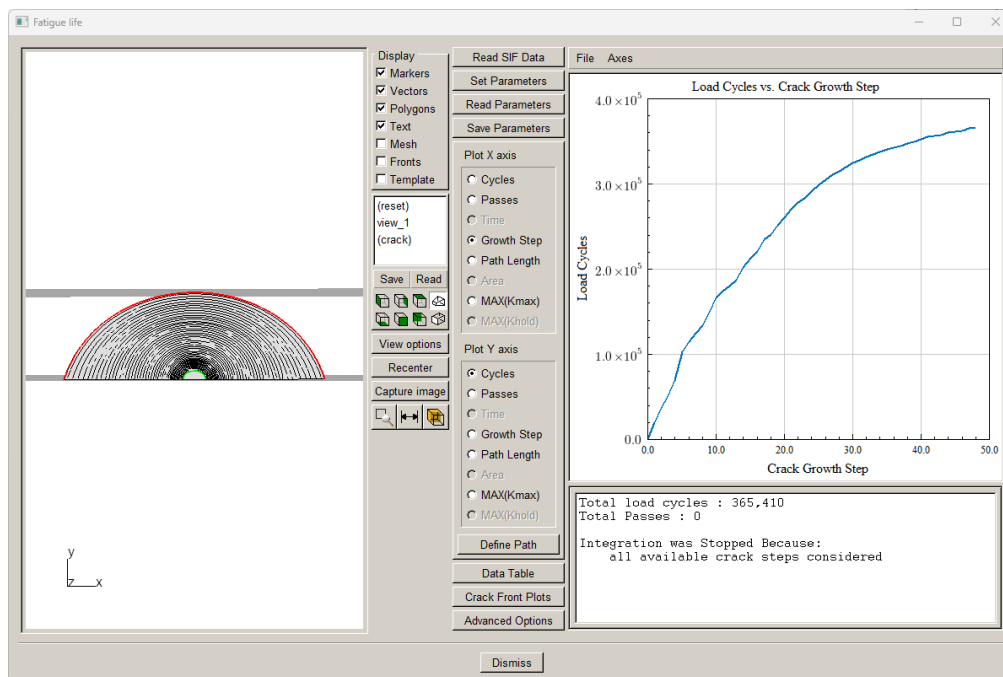


Figure 6.28 Fatigue Life dialog – using the Manu “average” spectrum and NASGRO data.

The fatigue module is described in Section 9 of the Reference document, and the subcritical crack growth models are described in Section 6.5. Section 10 of the User’s Guide describes aspects of the load event scheduling. Tutorials 10 and 14 in this document describe additional load schedule options.

7.0 Tutorial 7: Fretting Fatigue

Tutorial 7 describes the **Fretting Fatigue** module by presenting an example simulation, using a FE model of a fretting test rig, Fig 7.1.

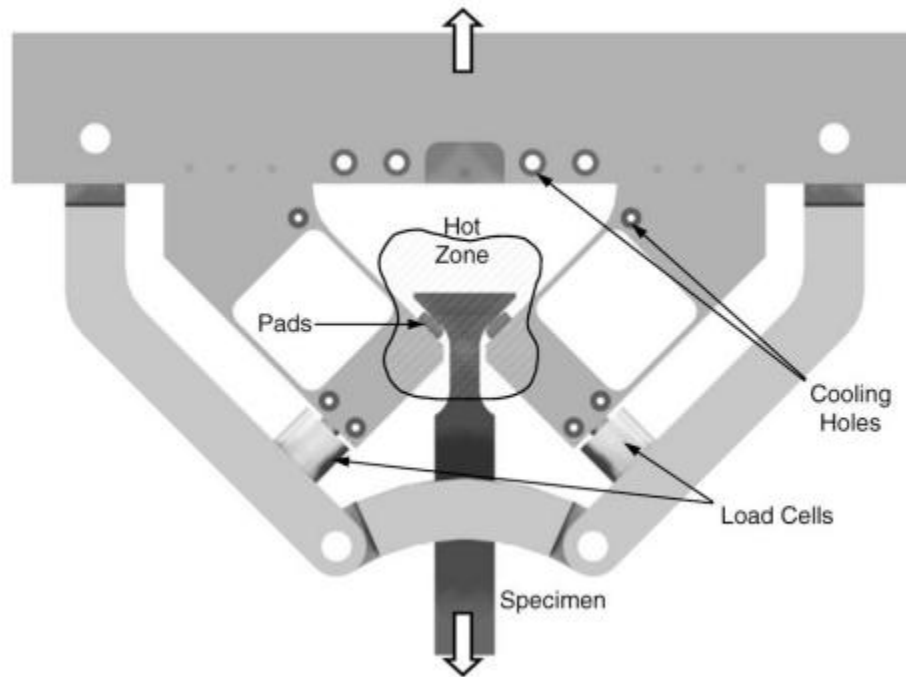


Figure 7.1 Fretting test rig (from: Golden, IJF, 31, 2009).

7.1 FE Model

Only ABAQUS and ANSYS versions of this model are described.

7.1.1 ABAQUS Model

The ABAQUS model is shown in Fig 7.2a. Using symmetry, only one quarter of the actual model is required. There are two load steps applied; the deformed shapes for both steps are shown in Fig 7.3a. The specimen is pulled downward and then partially unloaded; the load ratio

(R) is 0.1. There is contact between the specimen and fretting pad; the coefficient of friction is 0.3.

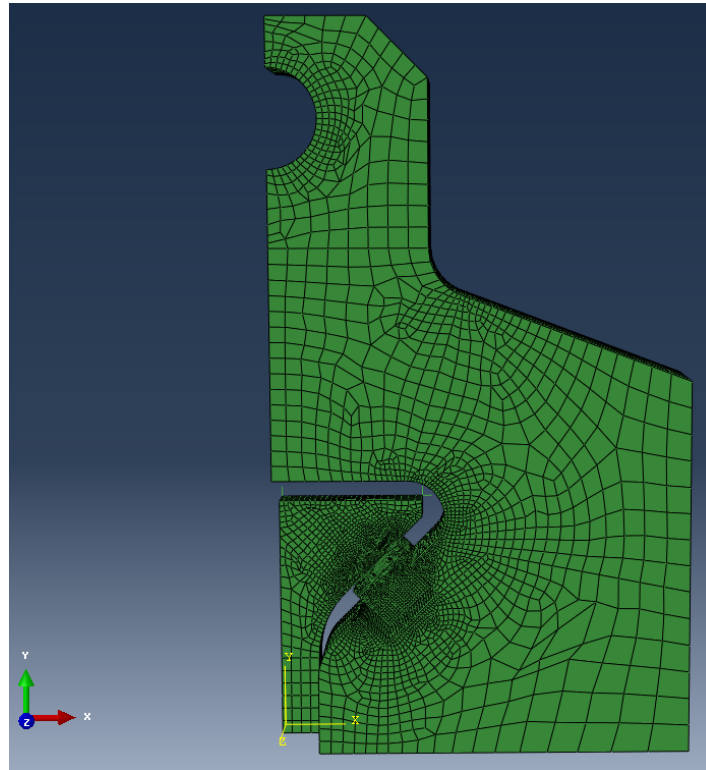


Figure 7.2a ABAQUS model of fretting test rig.

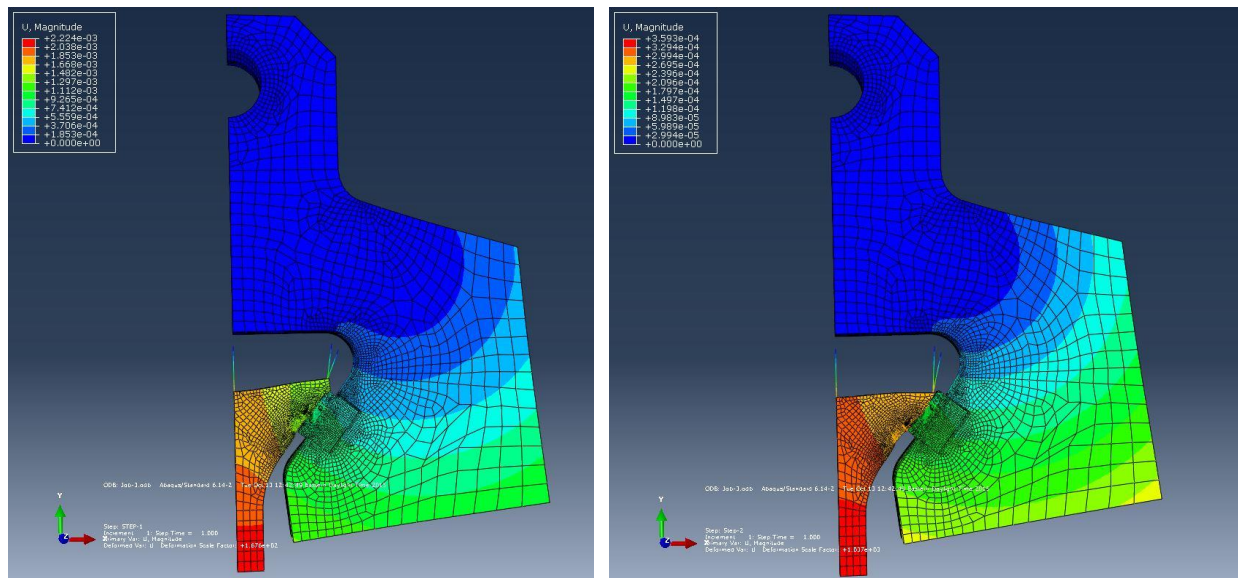


Figure 7.3a Fretting test rig deformed shape for load step #1 (left) and #2 (right).

Results can be written to a *.fil* file by adding the following commands to the ABAQUS *inp* file or using the ABAQUS CAE **Edit Model Keywords** option:

```
*File Format, ASCII
*Node File, Frequency=1
U
*El File, position=averaged at nodes, Frequency=1
S,E
*Contact File, Frequency=1
CDISP, CSTRESS
```

This set of commands is repeated for the second load step, minus the “**File Format, ASCII*” line. These lines can be added just before the **End Step* lines in the *.inp* file. Another option is to use an ABAQUS Python script to extract results from the *.odb* file into a *.dtp* file; a script is available from the FAC website.

An additional change should be made in ABAQUS CAE to flatten the *.inp* file and *.fil* files, removing **Part*, **Instance* and **Assembly* data:

```
>>> mdb.models["fretting_rig"].setValues(noPartsInputFile=ON)
```

This eliminates any confusion with node and element numbering.

7.1.2 ANSYS Model

The ANSYS model is shown in Fig 7.2b. Using symmetry, only one quarter of the actual model is required. There are two load steps applied; the deformed shapes for both cases are shown in Fig 7.3b. The specimen (darker blue colored piece in Fig 7.2b) is pulled downward and then partially unloaded; the load ratio (*R*) is 0.1. There is contact between the specimen and the fretting pad; the coefficient of friction is 0.3.

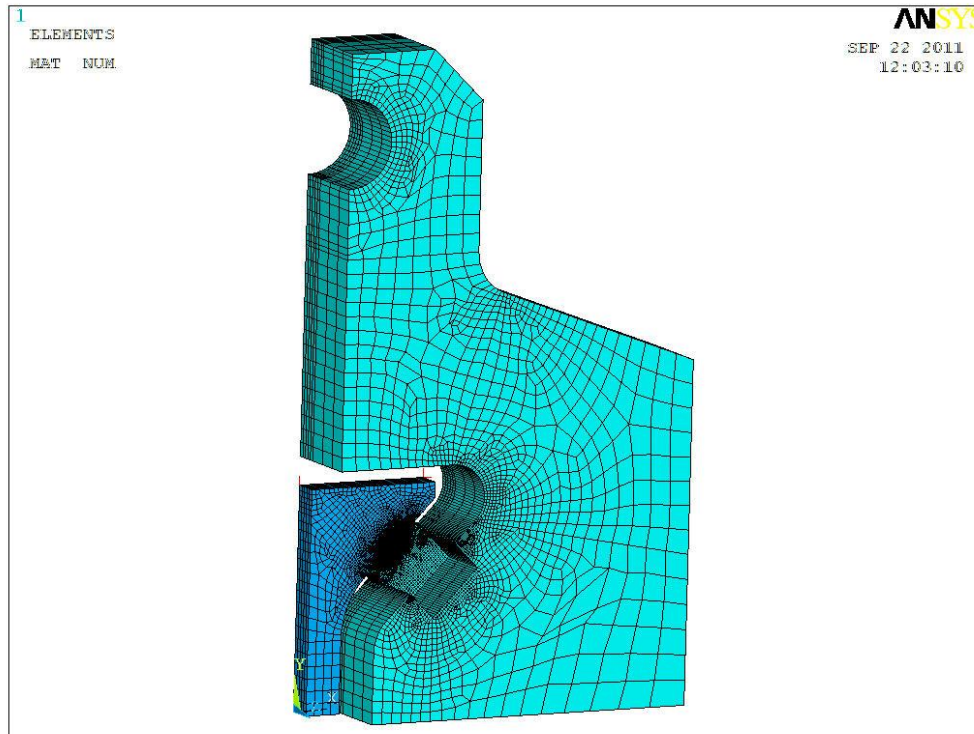


Figure 7.2b ANSYS model of fretting test rig.

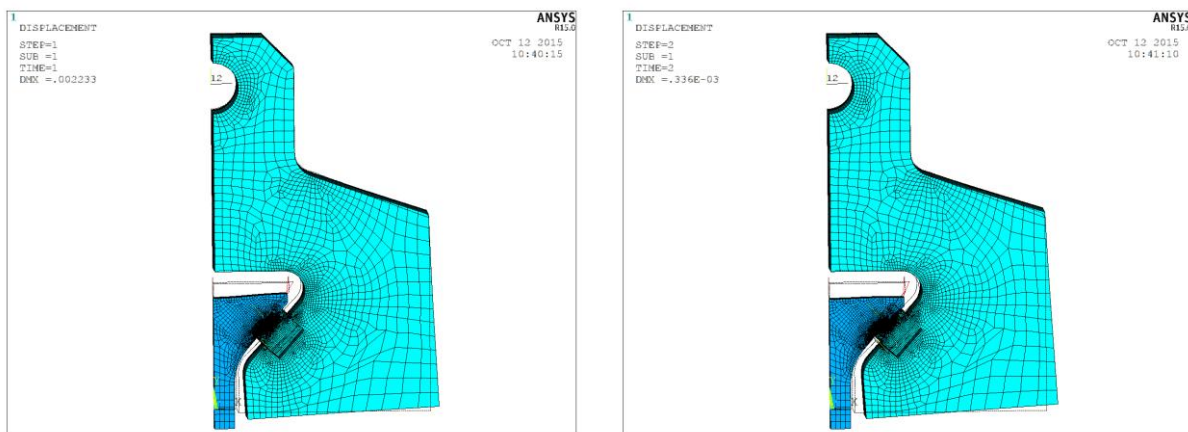


Figure 7.3b Fretting test rig deformed shape for load step #1 (left) and #2 (right).

Using the ANSYS post-processor, nodal results are extracted and saved to ASCII files. The results must be in the global Cartesian coordinate system. The following ANSYS ADPL commands are used:

```
/graphics,off  
rsys,0  
/format,,g,20,12  
prnsol,s  
prnsol,epto  
prnsol,dof  
prnsol,cont
```

Each set of results for each load step are saved to files with the following extensions: *.str*, *.stn*, *.dsp* and *.con*. The base file name should be the same for each extension, we use *fretting_rig_ls1* and *fretting_rig_ls2*. In addition to these results, we must also save the ANSYS model to a *.cdb* file, called *fretting_rig.cdb*. Another option is to use an ANSYS macro to extract results from the *.rst* file into a *.dtp* file; a macro is available from the FAC website.

7.1.3 NASTRAN Model

We do not currently support NASTRAN for fretting analysis.

7.1.4 Results in *.dtp* File

Rather than using the ABAQUS *.fil* file or the multiple ANSYS results files, results can be stored in a single *.dtp* file.

7.2 Step 2: Fretting Model Import

Start FRANC3D and from the main menu, choose **Fretting** → **Read model and results**, Fig 7.4. If the fretting menu option is not displayed, you can turn it on in the Preferences (see Section 5.2.2 of the Reference).

Select the appropriate model file, Fig 7.5. Note that the File type (not shown in Fig 7.5) is switched from ANSYS files to ABAQUS files so we can choose the *.inp* file.

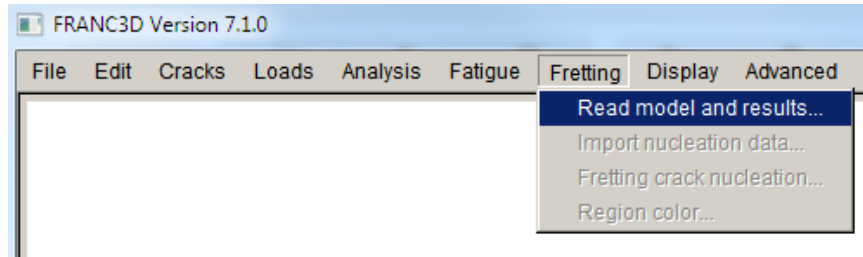


Figure 7.4 FRANC3D menu selection for fretting analysis.

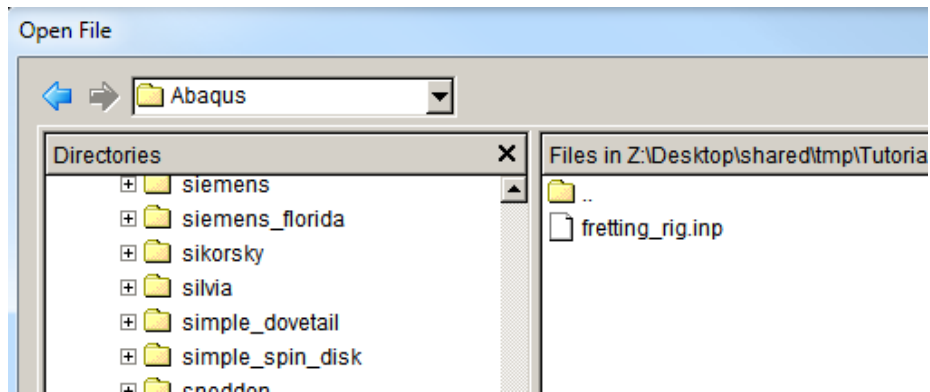


Figure 7.5 FRANC3D mesh model file selector for fretting module – ABAQUS file selected.

7.2.1 Import ABAQUS Results

After selecting the model, you are prompted to select the results files, Fig 7.6a. In this case, we have results for two load steps, corresponding to max and min conditions, stored in a *.dtp* file:

```
NUM STEP 2
LOADSTEP 1
GAP
917  0.00212423852645
1389 0.00211900915019
...
```

Enter the file name (or use the **Browse** button) and then select **Finish** to proceed to the next step, where we specify the fretting contact surface that will be analyzed.

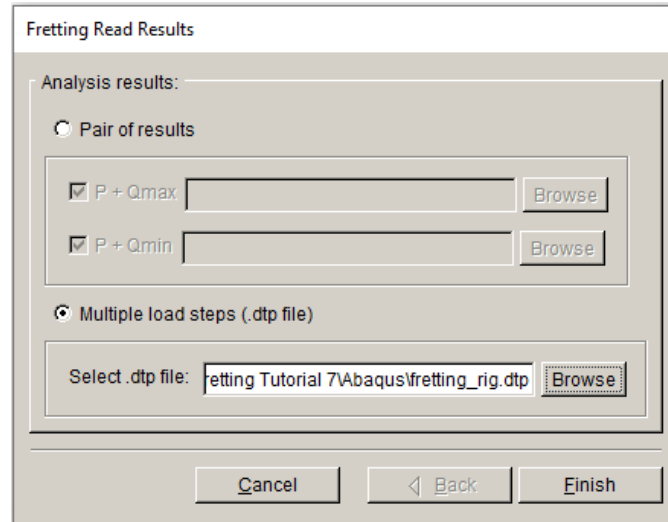


Figure 7.6a FRANC3D - ABAQUS results file selector for fretting module.

7.2.2 Import ANSYS Results

If we are reading the ANSYS *.cdb* file for the FE model (see Fig 7.5), we will read the corresponding results from multiple ANSYS results files for this tutorial, Fig 7.6b.

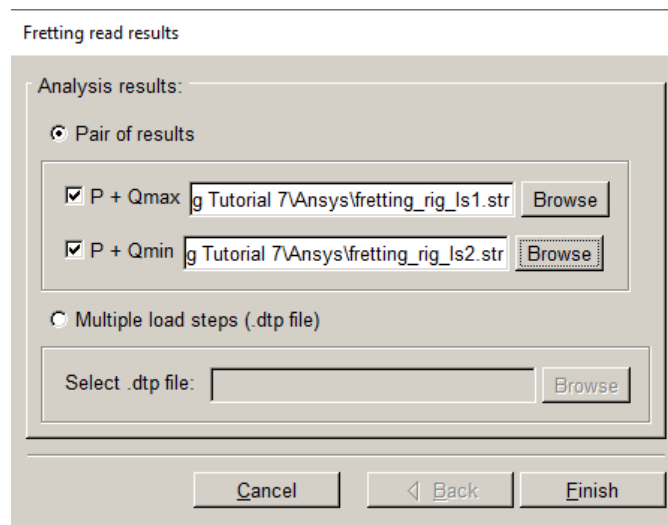


Figure 7.6b FRANC3D - ANSYS results file selector for fretting module.

7.3 Step 3: Specify Fretting Contact Surface

The next dialog, Fig 7.7, allows you to select the contact surface. The contact-pair surfaces are listed on the left and colored blue in the model. Select a contact-pair by checking the box beside the name, Fig 7.8; the corresponding surface is then colored red.

Select **Accept** and the model will be displayed in the FRANC3D main window; the material regions will have distinct colors, Fig 7.9.

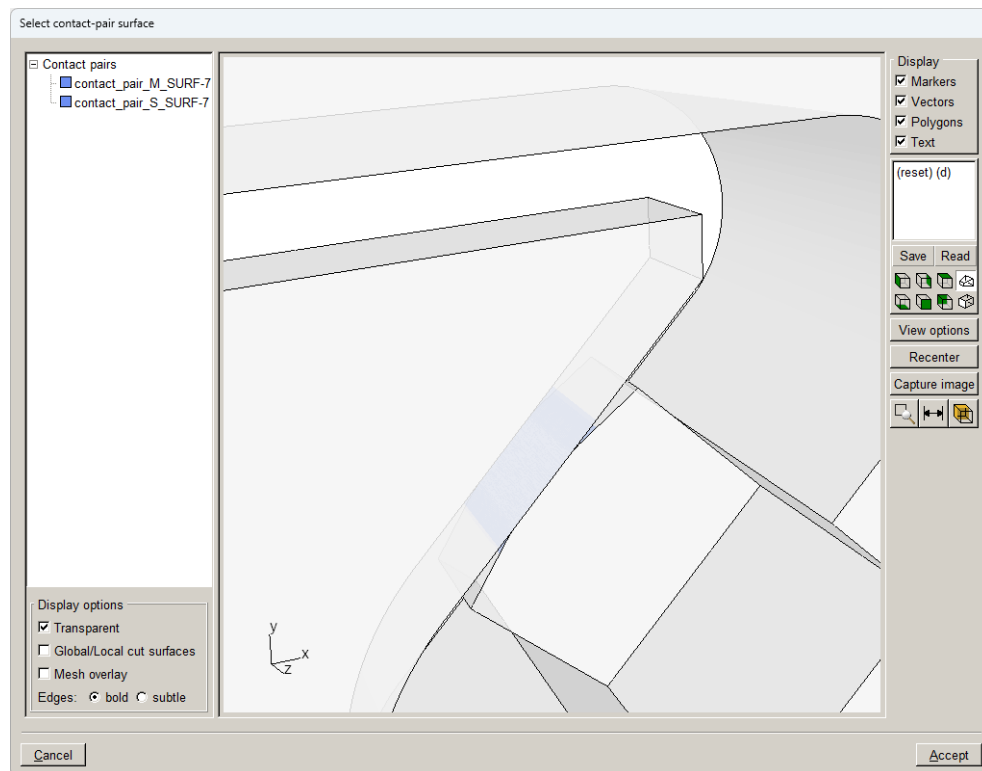


Figure 7.7 Fretting model data filter – listed contact surfaces.

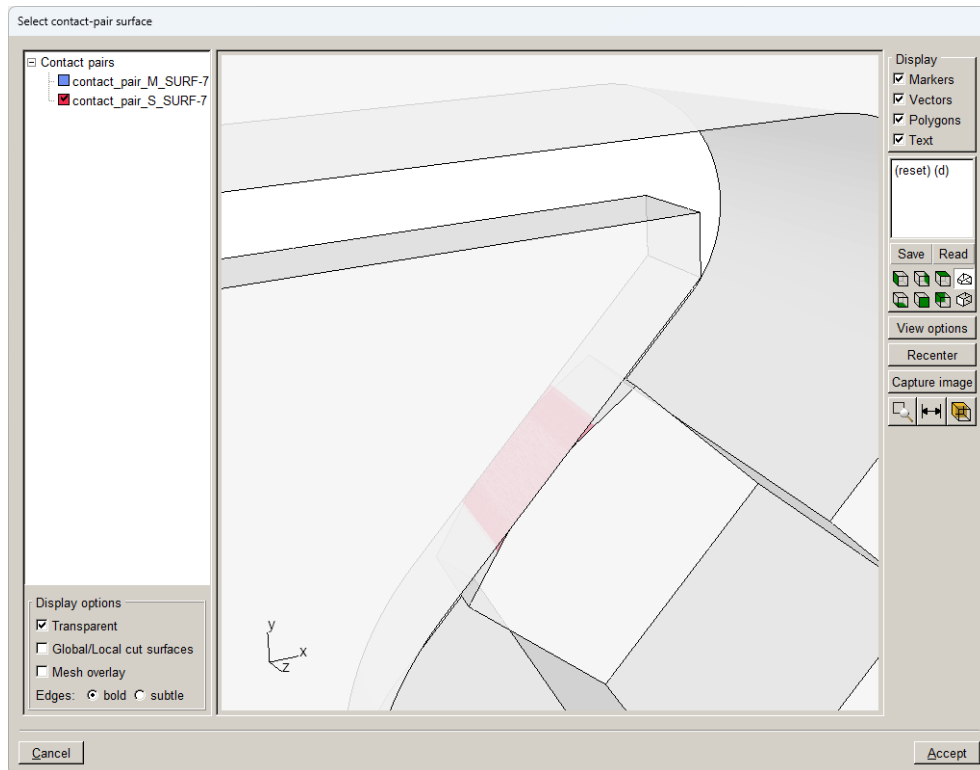


Figure 7.8 Fretting model data filter – selected contact surface.

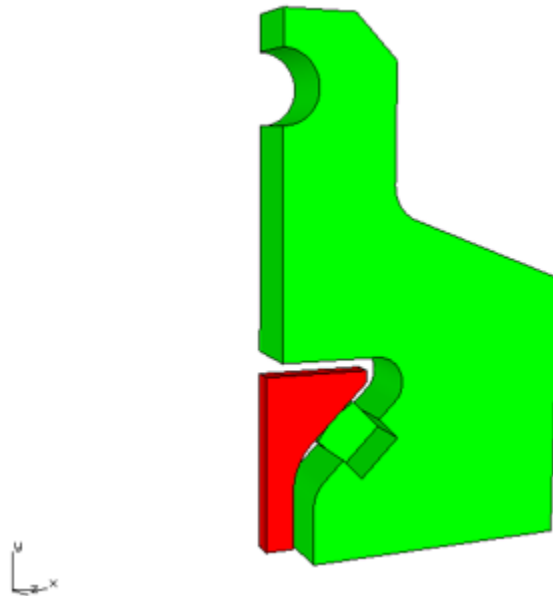


Figure 7.9 Fretting test rig model displayed in FRANC3D.

7.4 Step 4: Fretting Crack Nucleation

The next step is to compute the fretting nucleation parameters. From the FRANC3D **Fretting** menu, select **Fretting crack nucleation**, Fig 7.10. There are five fretting nucleation models available. The models have been described in the literature (see Section 10 of the Reference document); they use the contact analysis results to compute the fretting nucleation parameter. If the data required by a fretting model is not present in the results, the model cannot be selected.

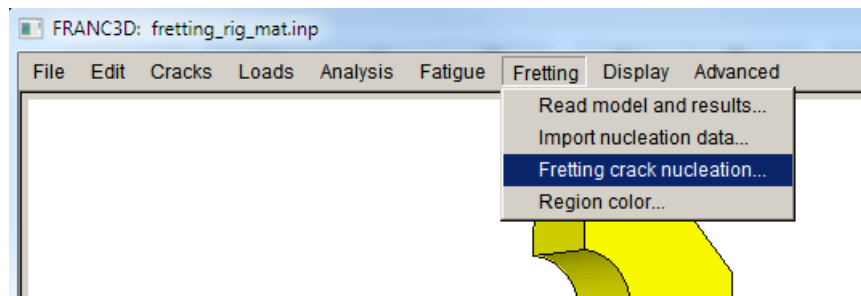


Figure 7.10 Fretting crack nucleation menu option.

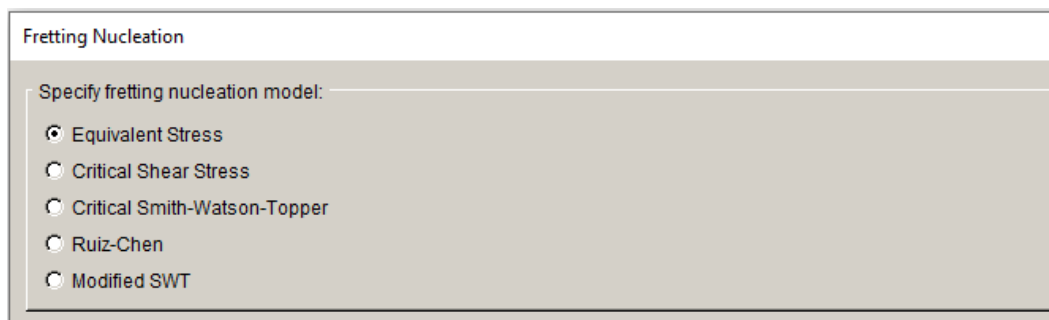


Figure 7.11 Fretting nucleation models.

For this example, we choose the Equivalent Stress model and select **Next**. The next dialog, Fig 7.12, allows you to specify the model parameters to compute the Seq parameter and the number of cycles. FRANC3D computes the Seq values at the nodes in the contact surface region and then computes the number of cycles (N_i), based on the equation shown at the top of the dialog.

Fretting Nucleation

Equivalent stress model:

$$\sigma_{eq} = a N_i^b + c N_i^d$$

Terms: ☒ user-defined ☐ curve-fit

a:

b:

c:

d:

$$\sigma_{eq} = (F_{s2} / F_{s1})^{1/\alpha} \sigma_{eff,max}$$

$$\sigma_{eff} = 0.5(\Delta\sigma_{psu})^w (\sigma_{max})^{1-w}$$

$$F_{s2} = \sigma_{eff} / \sigma_{eff,max} \Delta A$$

α :

F_{s1} :

w :

$\Delta\sigma_{psu}, \sigma_{max}, \Delta A$ from FE results

Compute stress at:

☒ node ☐ element centroid ☐ averaged at node

Surface treatment residual:

☐ Add residual stress

Figure 7.12 Equivalent stress fretting nucleation model parameters.

The Equivalent Stress model requires several material dependent parameters. For this example, the material is Ti-6Al-4V, which corresponds with the default parameters in the dialog; these parameters were obtained from the literature.

Select **Next** to proceed to the final panel, Fig 7.13, which allows you to display the contact surfaces, and color contours of fretting nucleation cycles or nucleation parameter, Fig 7.14. The Seq value is computed from the FE results, and the number of cycles (Ni) is computed based on the given equation.

Fretting equation parameters are mostly material dependent and could be model-contact dependent. Fretting test data can be imported if the equation parameters are not known. Fretting tests provide the number of cycles. FE models of the fretting tests provide the corresponding

Seq values. If sufficient tests are conducted, the data can provide a Seq versus Ni plot, Fig 7.15. This data can be fit to an empirical equation, which can then be used along with computed Seq values from the fretting test-rig FE model to predict the Ni cycles.

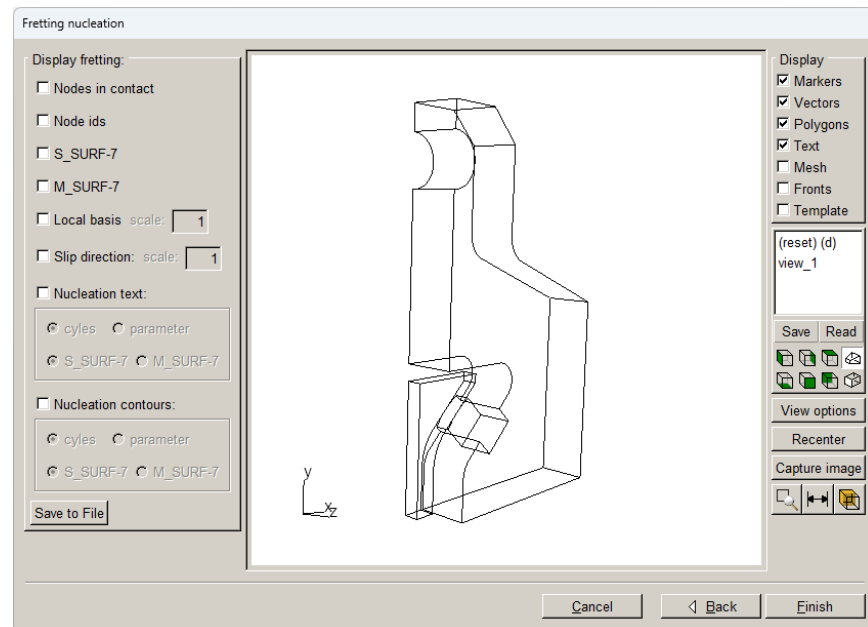


Figure 7.13 Fretting nucleation display window.

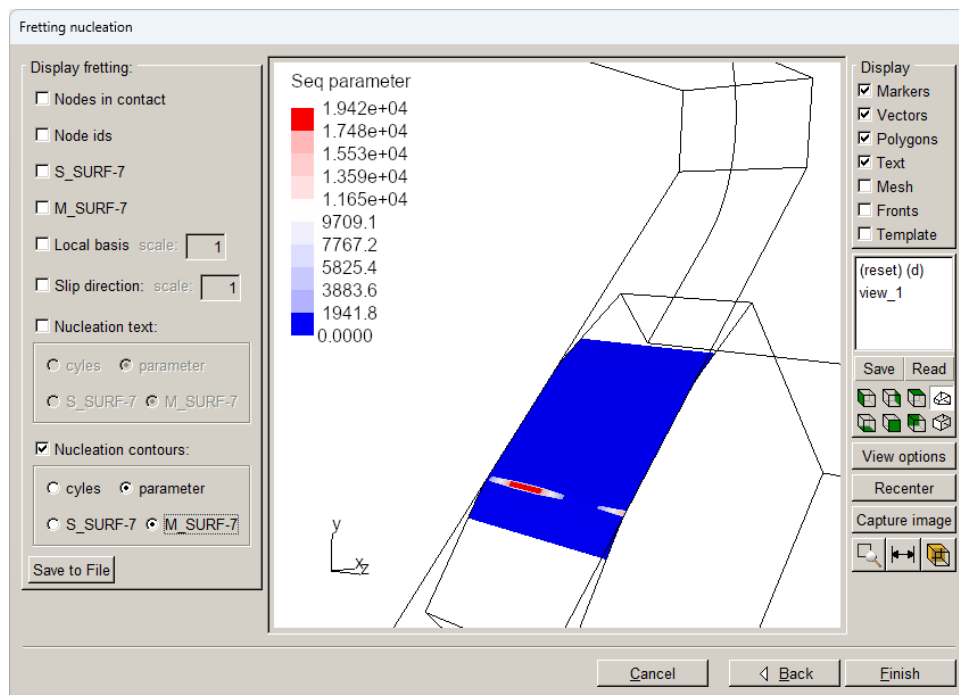


Figure 7.14 Seq contours on fretting specimen surface.

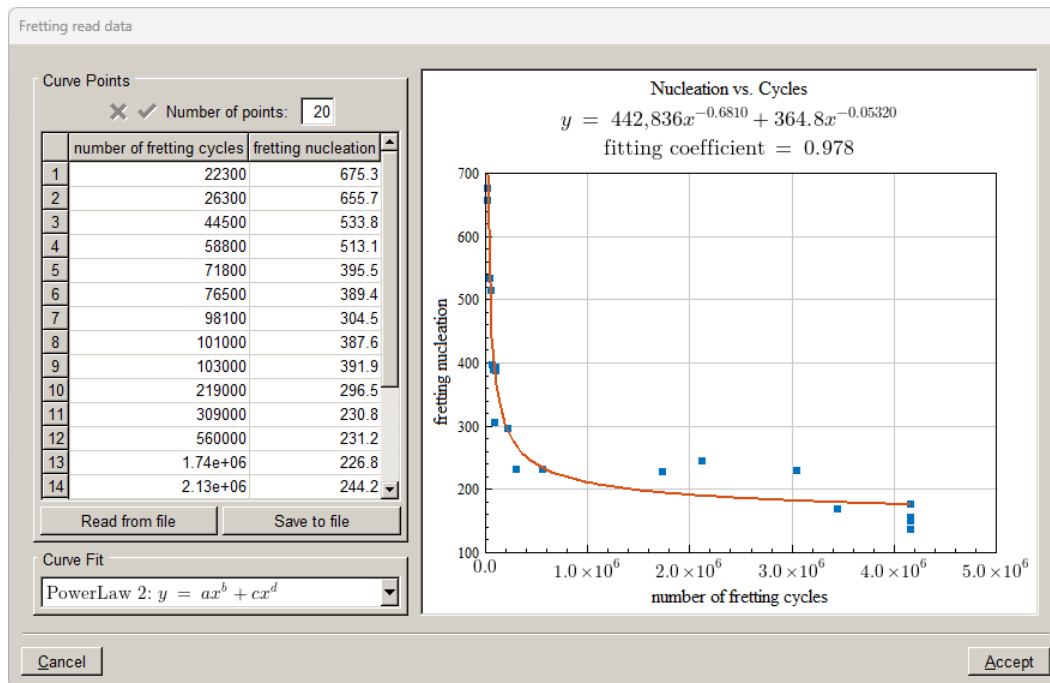


Figure 7.15 Seq vs Ni equation fitting.

7.5 Step 5: Discrete Crack Nucleation

In this section, we assume that fretting fatigue causes crack nucleation. We extract a submodel, insert a crack, and run an analysis.

The first step is to close the fretting model if it is still open; use **File** → **Close**. Next we import the model using **File** → **Import** and choose “import and subdivide”. Select the appropriate fretting model file, Fig 7.16.

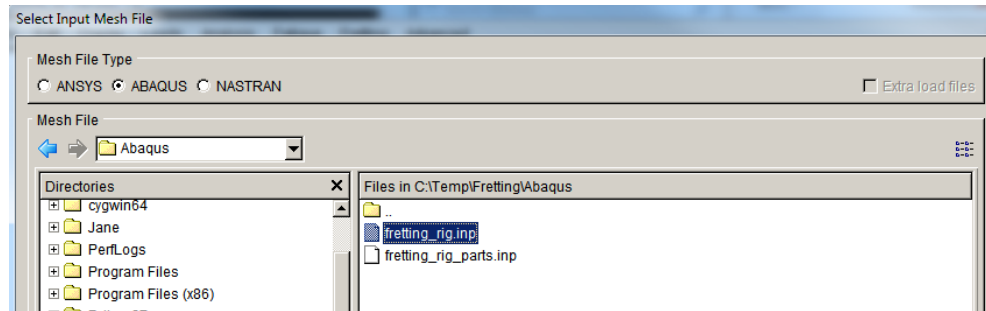


Figure 7.16 Import the fretting rig model – ABAQUS file selected.

The submodeling tool is displayed once the file is imported. The first selection can be done using the By Element Group option; click the **Select** button and pick the *dovetail_specimen* set, Fig 7.17. Use the **Crop** button to remove the remainder of the model, Fig 7.18.

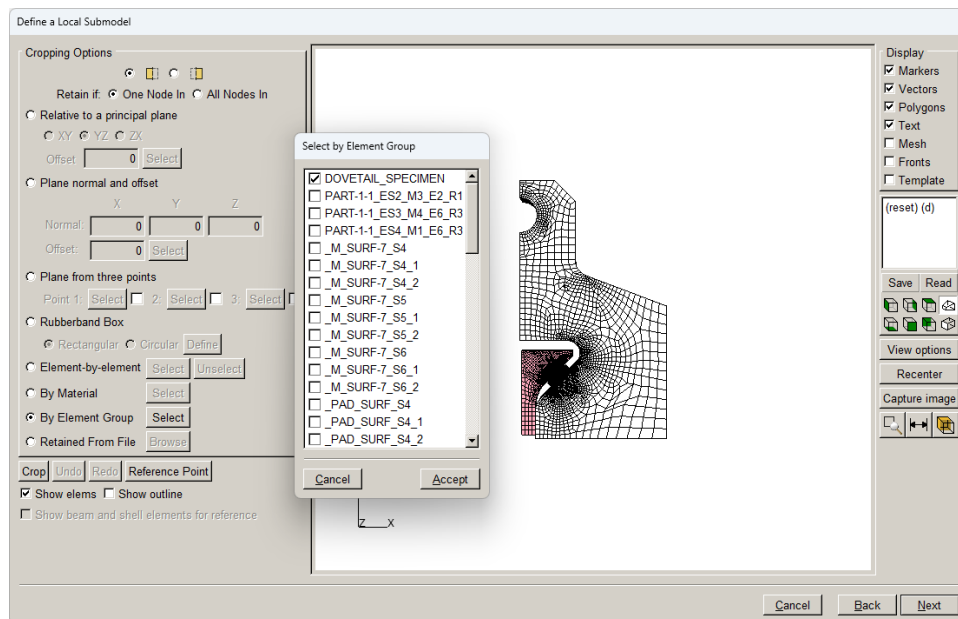


Figure 7.17 Submodel selection based on group selection.

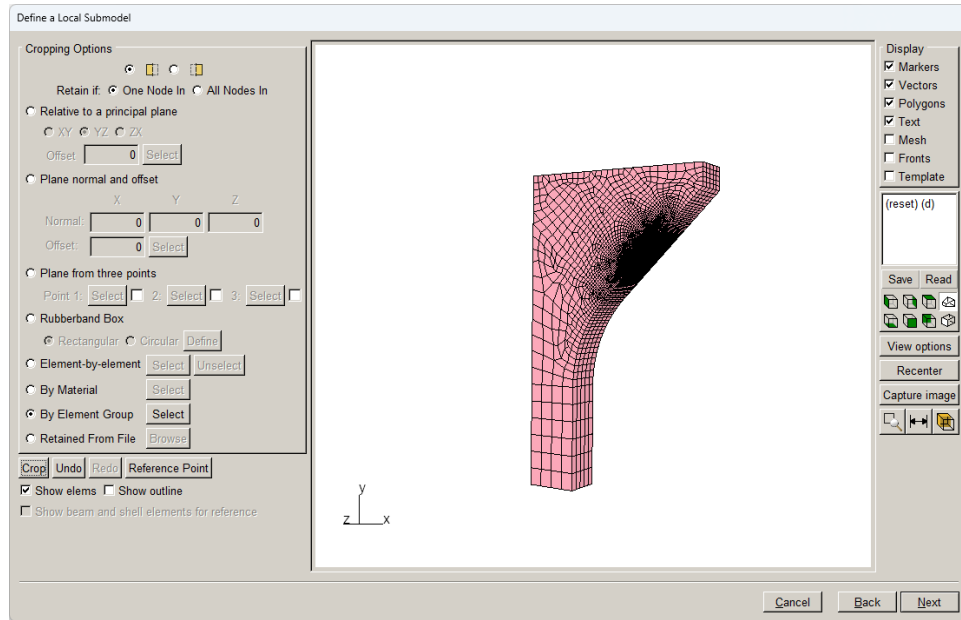


Figure 7.18 Local model after group selection crop.

Next we use the Rubberband Box to select a smaller submodel, Fig 7.19. Note that this submodel includes the entire contact surface region; this is important to remember when we are setting up the static analysis of the full cracked model as we will have to redefine the contact by adding an extra connection.

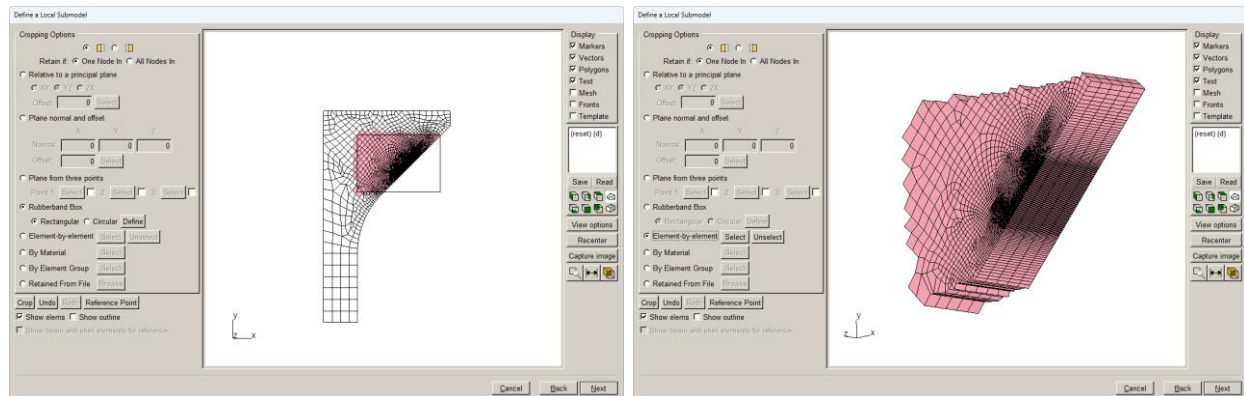


Figure 7.19 Submodel selection based on a Rubberband box.

After extracting the appropriate submodel and saving the local and global files, the dialog in Fig 7.20 is displayed. This allows us to retain the mesh facets where the boundary conditions are

applied. In this case, we do not select these facets as the surface must be remeshed to accommodate the through crack that we will insert. Just select **Finish** in this dialog, and the model will be displayed in the FRANC3D main model window.

A shallow through (or edge) crack is inserted near the lower edge of contact at the location as predicted by the fretting nucleation model. Fig 7.21 shows two views of the crack. Note that for shallow cracks, it is a good idea to change the default meshing parameters when setting the crack front template. Fig 7.22 shows the meshing parameters dialog with Do coarsen crack mouth mesh unchecked. The resulting remeshed cracked submodel is shown in Fig 7.23.

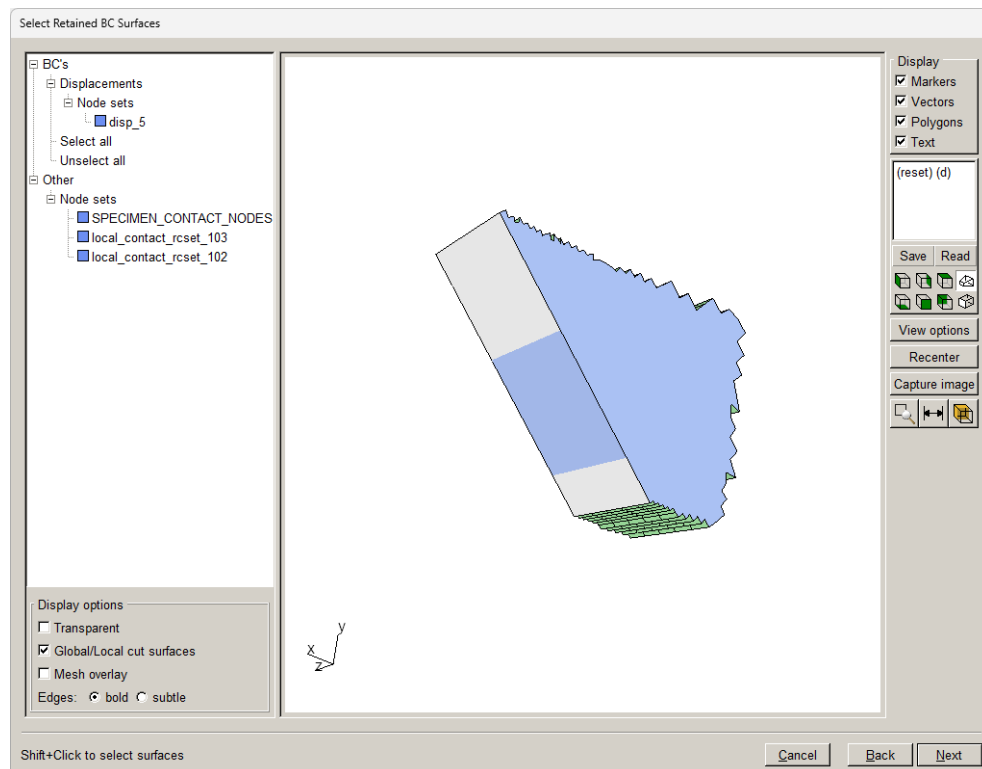


Figure 7.20 Select retained surfaces dialog – do not select this surface.

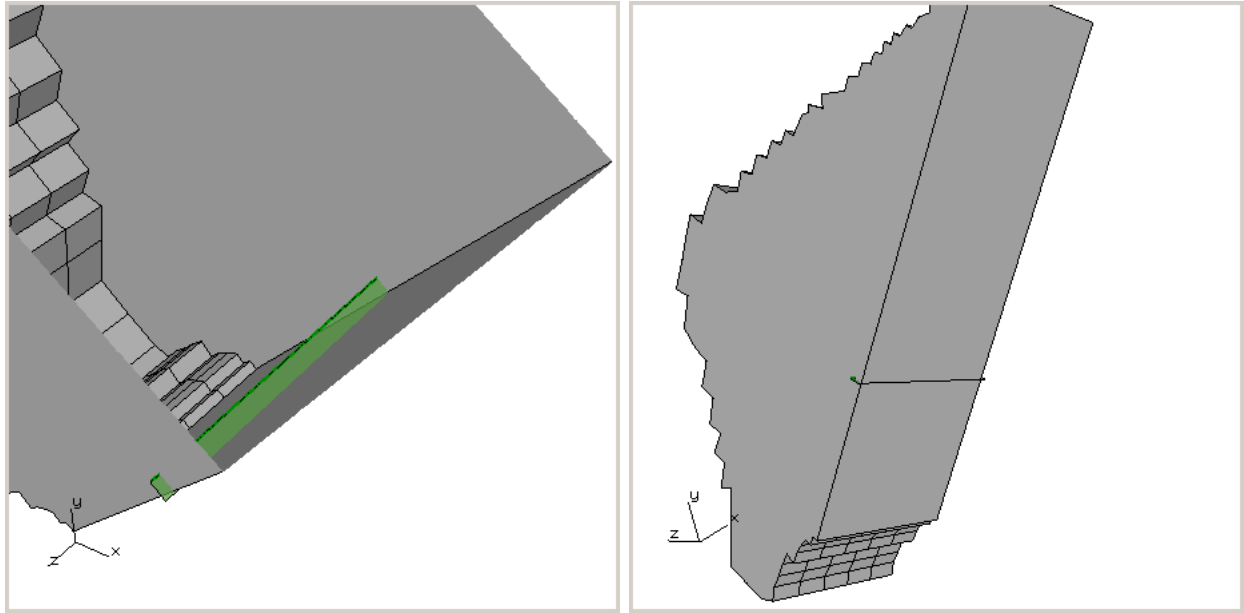


Figure 7.21 Through crack near the lower edge of contact.

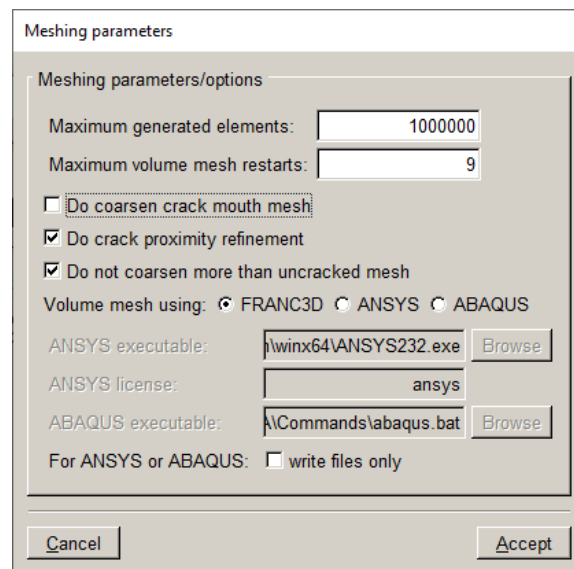


Figure 7.22 Meshing parameters dialog.

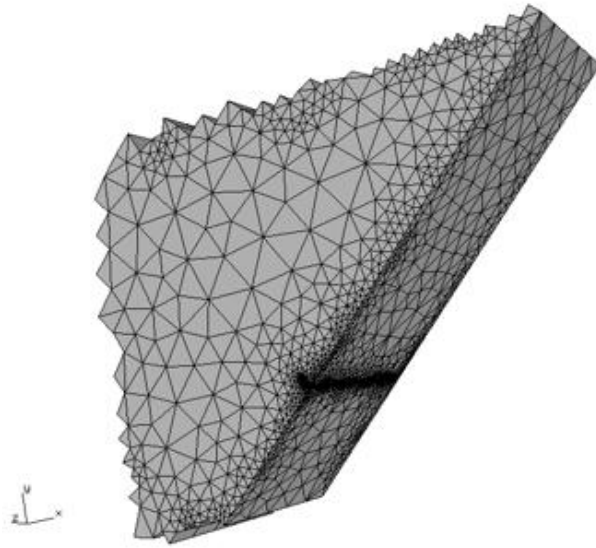


Figure 7.23 Cracked remeshed submodel.

7.6 Step 6: Static Crack Analysis

This local cracked portion of the model is now combined with the global model and solved.

7.6.1 ABAQUS Static Crack Analysis

The Static Analysis dialogs for ABAQUS are shown in Figs 7.24 – 7.27. The global model consists of linear brick elements, and these must be tied to the quadratic elements from FRANC3D using *Tie constraints, Fig 7.25. An extra connection for the contact between the fretting pad and specimen is defined using surface to surface contact, Fig 7.26. The local + global connection dialog will list the extra connection once it is defined, Fig 7.27.

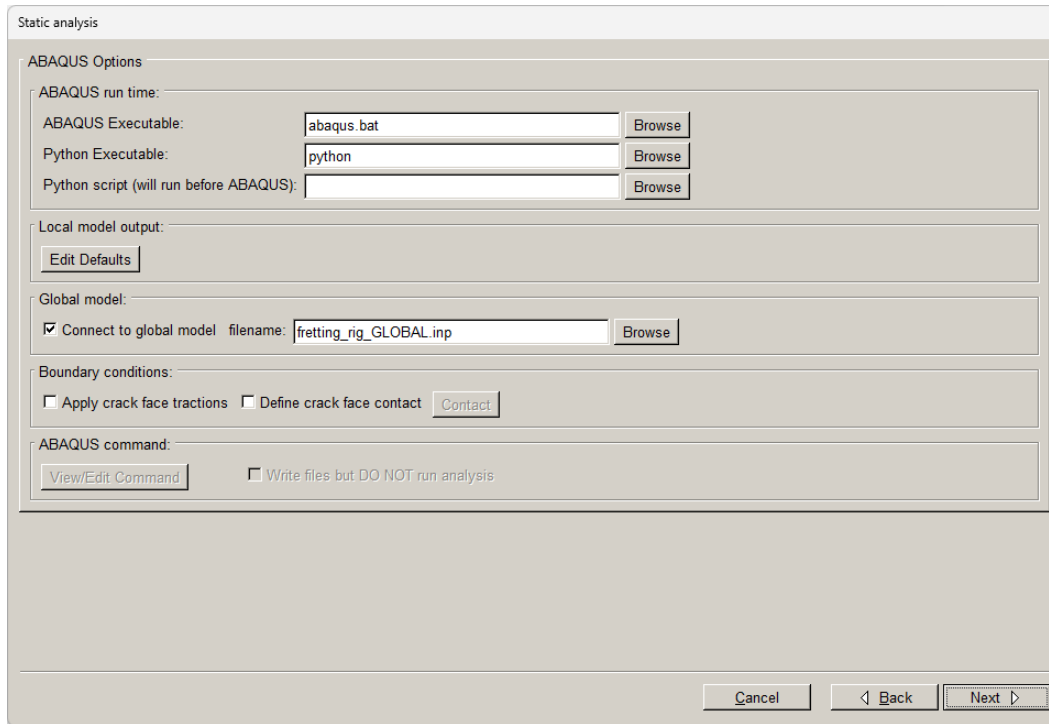


Figure 7.24 Static analysis dialog – ABAQUS options.

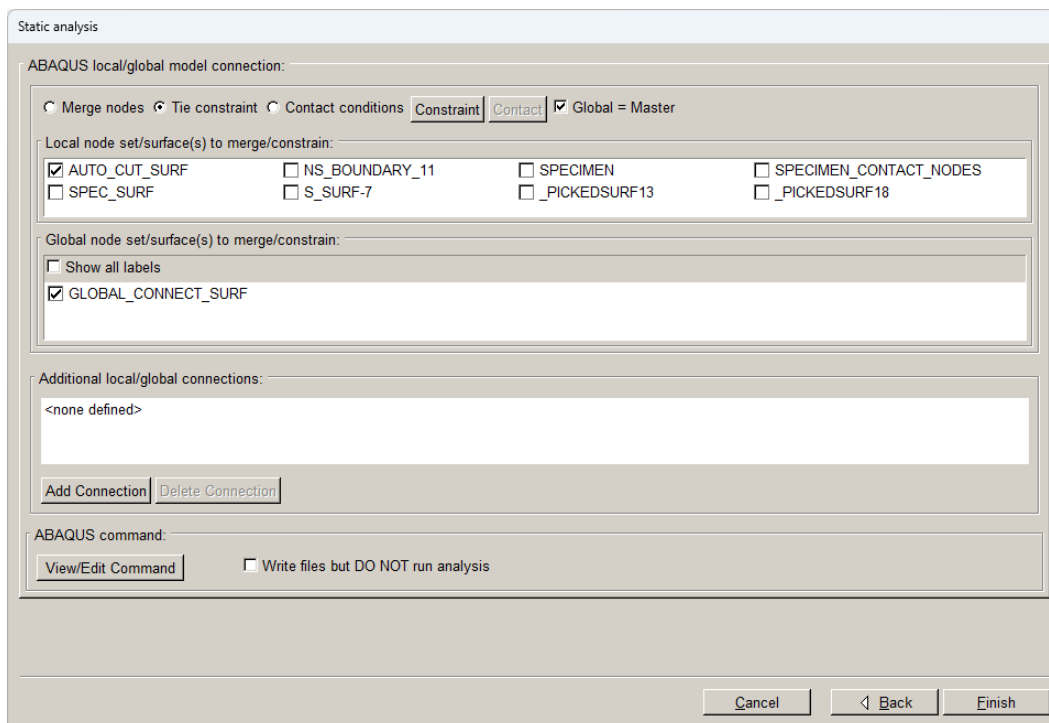


Figure 7.25 Static analysis dialog – ABAQUS local + global connection.

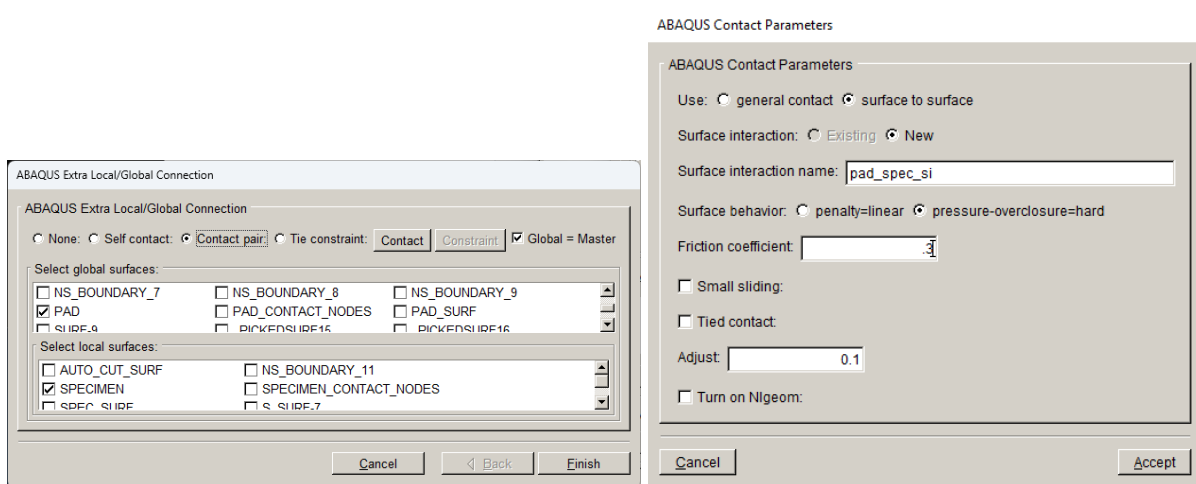


Figure 7.27 Static analysis dialog – ABAQUS extra contact connection.

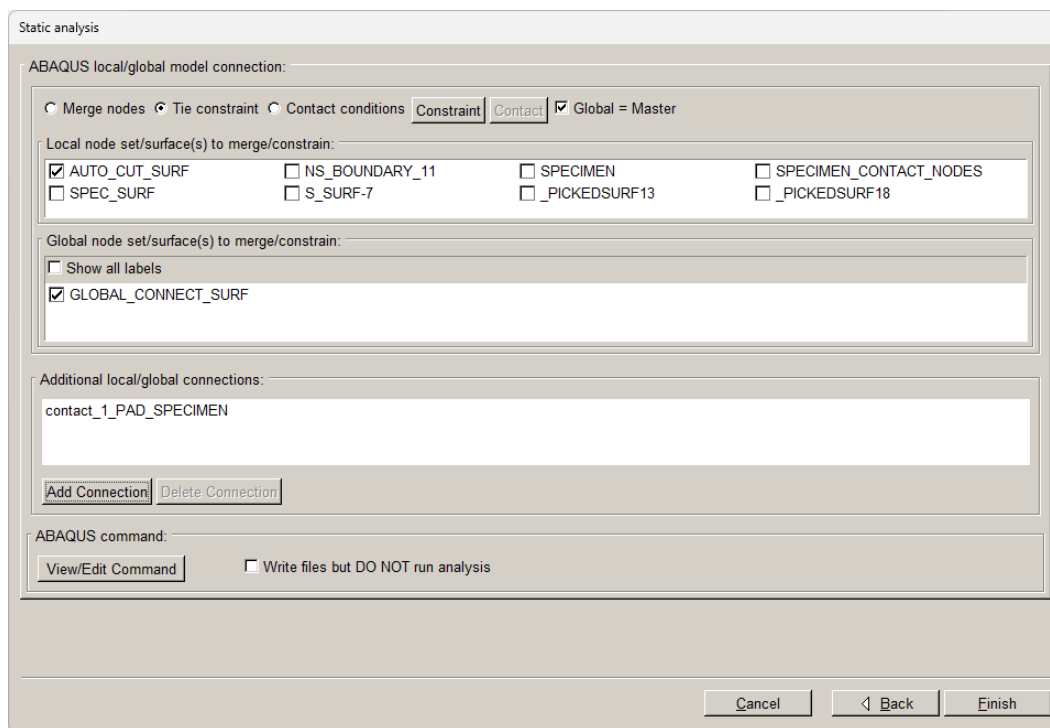


Figure 7.27 Static analysis dialog – ABAQUS local + global plus extra connection.

The resulting contact surfaces are shown in Fig 7.28. The analysis produces results as shown in Fig 7.29, which shows the deformed shape, and Fig 7.30, which shows Mode I SIFs for the first load step.

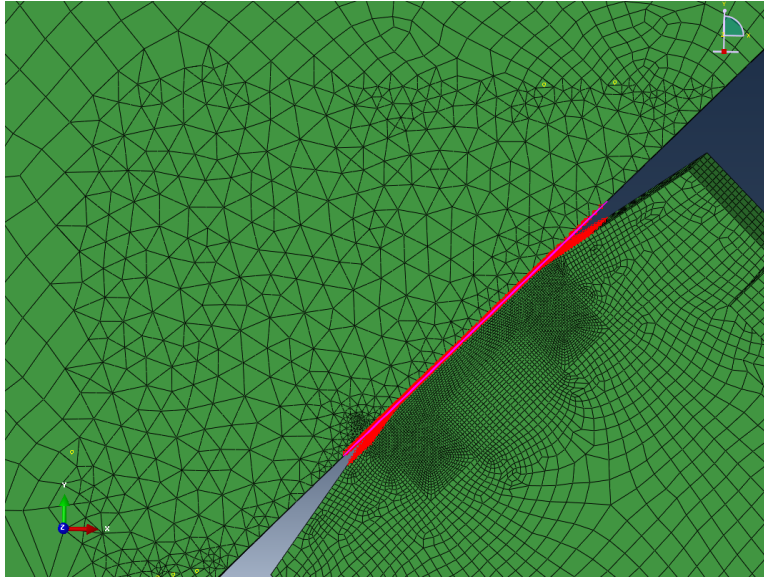


Figure 7.28 Newly defined ABAQUS contact surfaces.

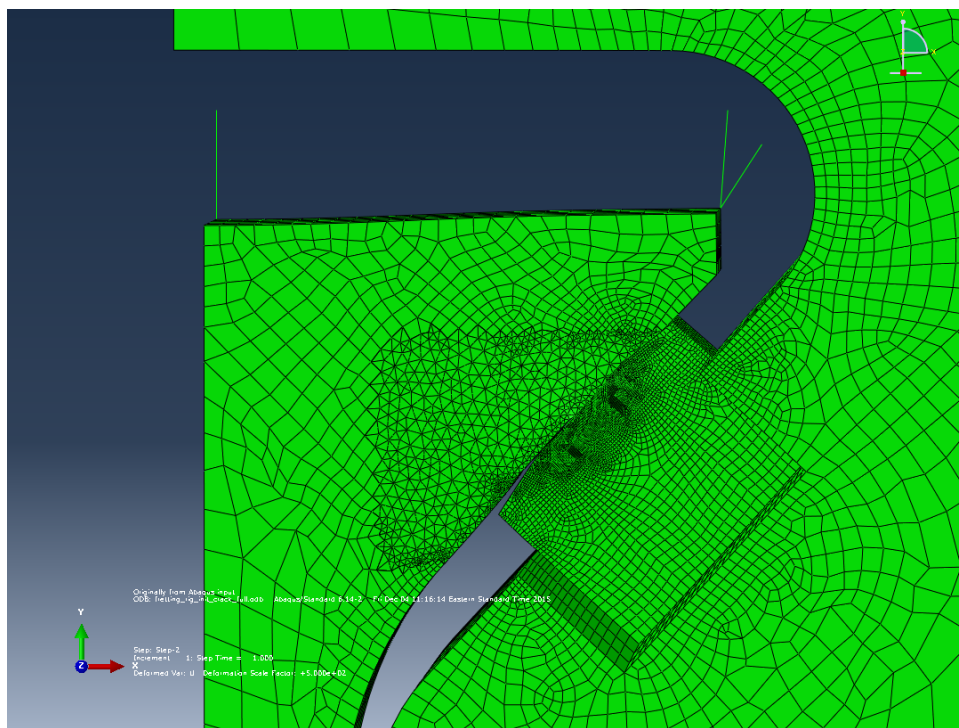


Figure 7.29 ABAQUS deformed shape after analysis – 500x magnification.

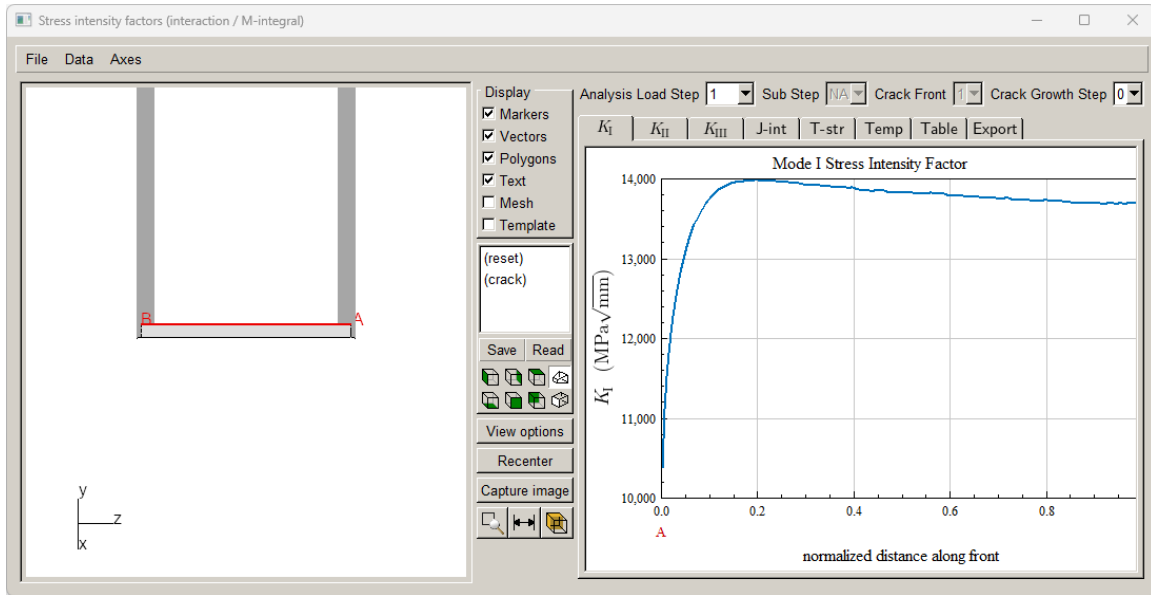


Figure 7.30 Mode SIFs for first load step.

7.6.2 ANSYS Static Crack Analysis

The Static Analysis dialogs for ANSYS are shown in Figs 7.31 – 7.34. The global model has linear brick elements; we can tie these to the quadratic elements from FRANC3D using node-merging with the midside nodes removed. We could also use constraints or bonded-contact.

An extra connection for the contact between the fretting pad and specimen is defined using contact, Fig 7.32-7.33. The local + global connection dialog will list the extra connection once it is defined, Fig 7.34.

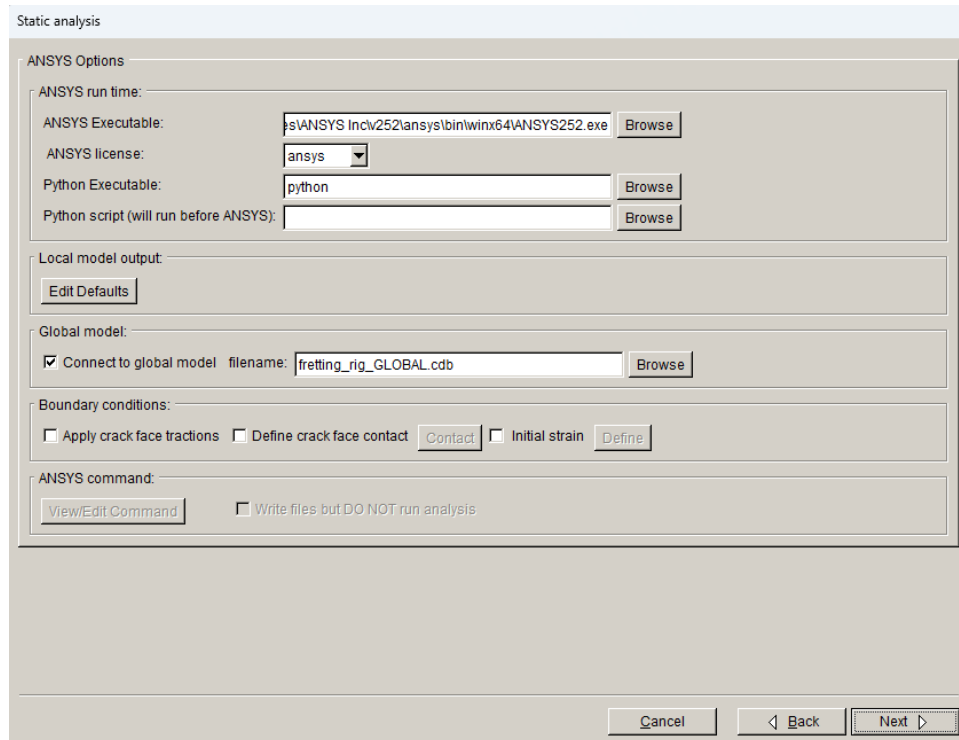


Figure 7.31 Static analysis dialogs – ANSYS options.

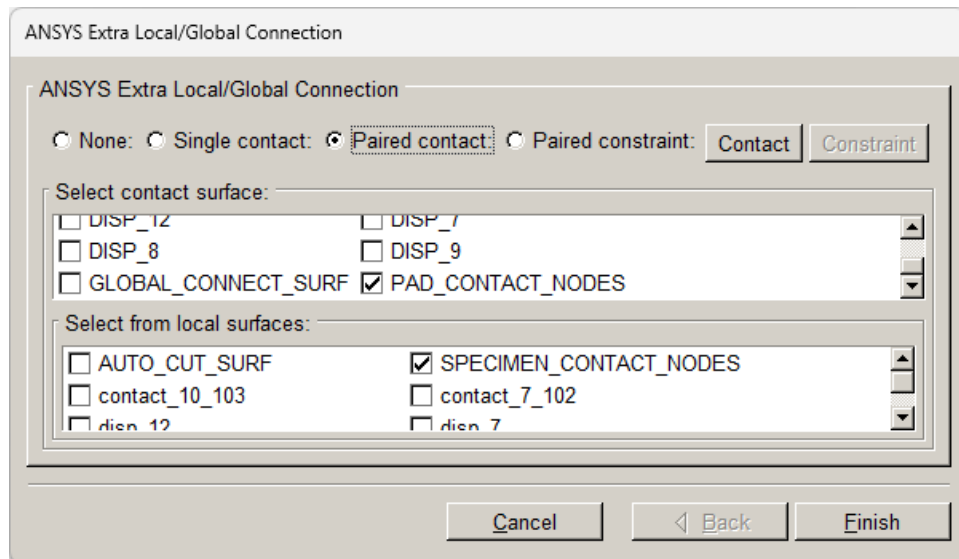


Figure 7.32 Static analysis dialogs – ANSYS extra contact connection.

ANSYS Contact Parameters

ANSYS Contact Parameters

Material id: Friction coef:

Real Constant id:

Normal penalty stiffness: (negative for absolute value)

Penetration tolerance: (negative for absolute value)

ET 170 id: ET 174 id:

Contact Algorithm: ☒ augmented lagrange ☐ penalty ☐ mpc ☐ lagrange + penalty ☐ lagrange

Gap/Penetration Adjustment: ☐ none ☐ close gap ☐ reduce pene ☐ close/reduce ☒ default/cont

Asymmetric Contact: ☒ off ☐ on

Initial Penetration/Gap: ☒ include ☐ exclude

Contact Stiffness Update: ☒ each iteration ☐ each load step ☐ each iteration of substep

Contact behavior: ☒ standard ☐ rough ☐ no_separation ☐ bonded ☐ bonded_always

Number of substeps: max min:

☒ Create Symmetric Pair: make target surface: ☒ master ☐ slave

Figure 7.33 Static analysis dialogs- ANSYS contact settings.

Static analysis

ANSYS local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☐ retain ☒ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF	☐ SPECIMEN_CONTACT_NODES	☐ contact_10_103	☐ contact_7_102
☐ disp_12	☐ disp_7	☐ disp_8	☐ disp_9

Global component(s) to merge/constrain:

☐ Show all labels

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

contact_1_PAD_CONTACT_NODES_SPECIMEN_CONTACT_NODES

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 7.34 Static analysis dialogs – ANSYS local + global connection.

The resulting contact surfaces are shown in Fig 7.35. The analysis produces results as shown in Fig 7.36, which shows the deformed mesh in dark blue with the white dashed lines representing the original shape. Mode I SIFs along the crack front are shown in Fig 7.37, for the first load step.

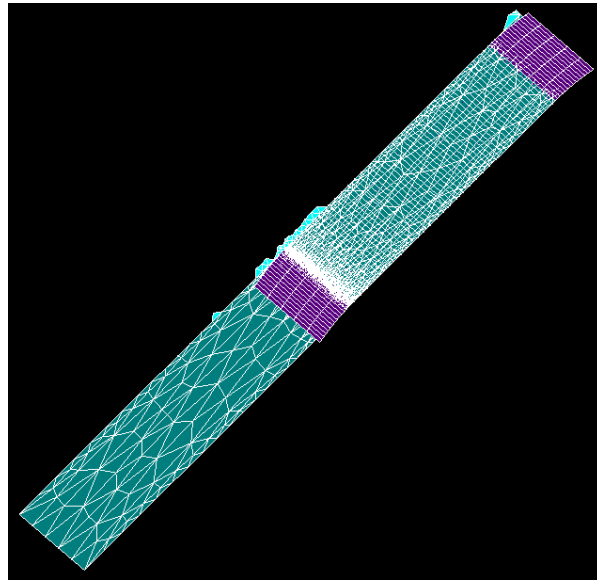


Figure 7.35 New ANSYS contact surfaces.

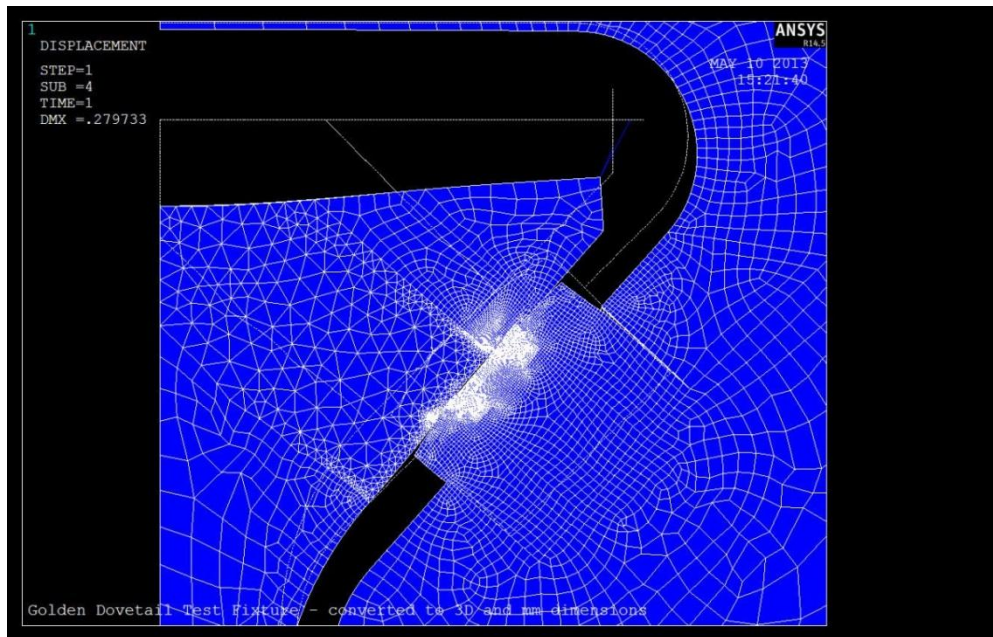


Figure 7.36 ANSYS deformed shape after analysis.

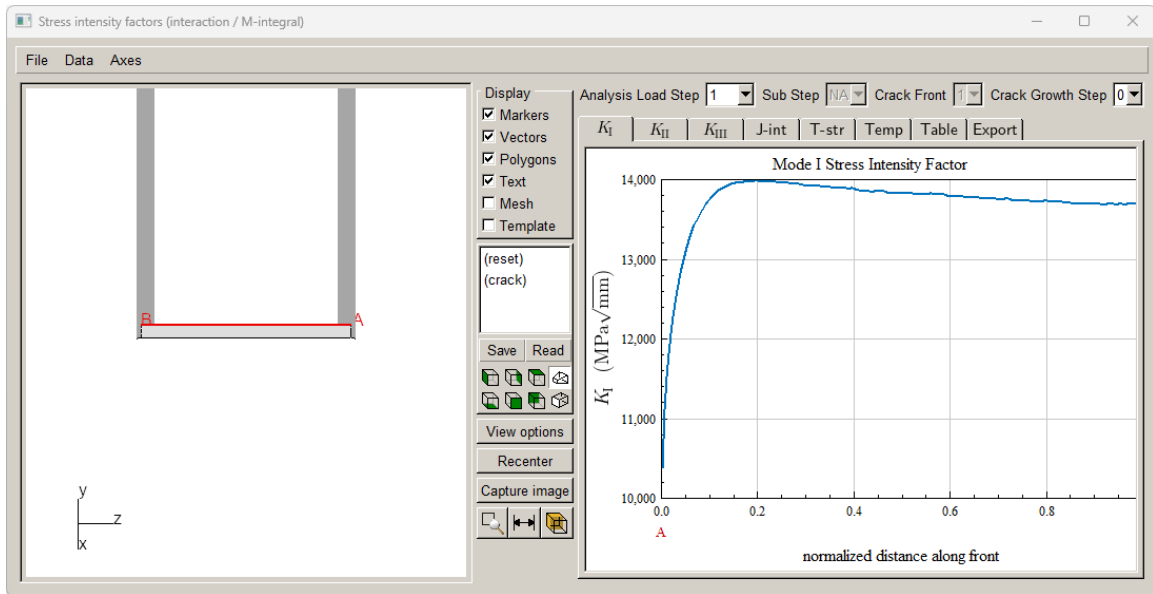


Figure 7.37 ANSYS Mode SIFs for first load step.

8.0 Tutorial 8: Session Log Playback

In Tutorial 8, we describe the session log and its playback. When running the FRANC3D (GUI) program, a session log file is saved for (almost) every menu command that is executed. The session log file is a text file that can be edited, played back in the FRANC3D GUI, played back from the command line, and/or converted to a Python script that can be used with the FRANC3D Python module. The session log file number in the current working directory is incremented each time FRANC3D is started in that directory.

The work directory should be set first; the session log will then be opened in that folder. As an example, the session log file for Tutorial #1 for ABAQUS is given here:

```
# session log for tutorial 1

Submodeler(
    model_type=ABAQUS,
    orig_file_name='Abaqus_Cube.cdb',
    submodel_file_name='Abaqus_Cube_LOCAL.cdb',
    global_file_name='Abaqus_Cube_GLOBAL.cdb',
    elem_file_name='Abaqus_Cube_RETAINED_ELEMS.txt')

OpenMeshModel(
    model_type=ABAQUS,
    file_name='Abaqus_Cube_LOCAL.cdb',
    global_name='Abaqus_Cube_GLOBAL.cdb')

InsertFileFlaw(
    file_name='Cube_Crack.crk')

RunAnalysis(
    model_type=ABAQUS,
    file_name='Abaqus_Cube_crack.fdb',
    flags=[NO_WRITE_TEMP,TRANSFER_BC,NO_CFACE_TRACT,NO_CFACE_CNTCT],
    merge_tol=0.0001,
    connection_type=MERGE,
    command='"abaqus.bat" job=Abaqus_Cube_crack_full -interactive -analysis ',
    global_model='Abaqus-Cube_GLOBAL.inp',
    merge_surf_labels=[AUTO_CUT_SURF],
    global_surf_labels=[GLOBAL_CONNECT_SURF])
```

ComputeSif()

```
SetGrowthParams(  
    growth_type=QUASI_STATIC,  
    load_step_map=["VERSION: 1","MAX_SUB: 1","0 0","LABELS: 1","1 Load Step 1"],  
    kink_angle_strategy=MTS,  
    quasi_static_n=2,  
    quasi_static_loads=["ONE_STEP: 1 NONE 1 1 0"])
```

```
GrowCrack(  
    load_model=QS_EXPONENTIAL,  
    kink_angle_strategy=MTS,  
    const_median_step=0.15,  
    temp_radius_type=ABSOLUTE,  
    temp_radius=0.1)
```

```
AutoGrowth(  
    model_type=ABAQUS,  
    cur_step=1,  
    file_name='Abaqus_Cube_crack',  
    num_steps=5,  
    step_type=SCONST,  
    const_step_size=0.15,  
    check_Kc=true,  
    check_DKth=true,  
    temp_radius_type=ABSOLUTE,  
    temp_radius=0.1,  
    flags=[NO_WRITE_TEMP,TRANSFER_BC,NO_CFACE_TRACT,NO_CFACE_CNTCT],  
    merge_tol=0.0001,  
    connection_type=MERGE,  
    command='"abaqus.bat" cpus=1 job=Abaqus_Cube_crack_STEP_001_full ask_delete=NO -  
interactive -analysis ',  
    global_model='Abaqus-Cube_GLOBAL.inp',  
    merge_surf_labels=[AUTO_CUT_SURF],  
    global_surf_labels=[GLOBAL_CONNECT_SURF])
```

```
WriteSifPath(  
    file_name='cube_crack_sif_hist_p5.sif',  
    load_step=1,  
    flags=[TAB,A,KI,KII,KIII,CRD])
```

The commands and the associated parameters are described in the Commands & Python reference document.

8.1 GUI Playback

The session log can be played back in the FRANC3D GUI. Start FRANC3D and choose **File** → **Playback**. Using the file selector, Fig 8.1, choose the session log file and select **Accept**. FRANC3D will read the log file and process the commands; in this case, an ABAQUS cube model is subdivided, the crack is inserted and analyzed and then propagated for five steps. Finally, the SIF history is extracted and written to a file.

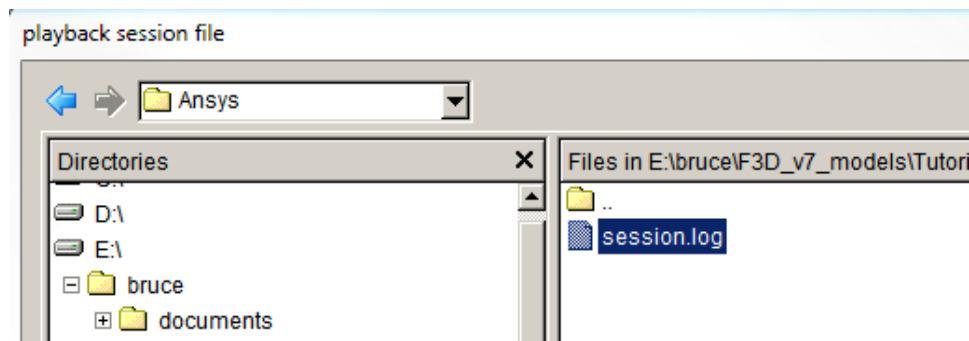


Figure 8.1 Select session log file to play back in FRANC3D.

During playback from the GUI, all the commands are written to a new session log file. This allows FRANC3D to continue writing commands from the GUI once the current session has finished playing.

8.2 Command Line Playback

To run the session log in the background without the GUI, use a CMD or terminal window and execute `franc3d` from the command line as:

```
➤ franc3d -batch session.log
```

This assumes that you are in the folder where the FE model and session log files reside and that FRANC3D is included in your PATH. During playback from the command line, the playback history is saved to a `playback.log` file.

9.0 Tutorial 9: Python Interface

In Tutorial 9, we briefly describe how to use the Python module. The Python module uses Python version 3; it should work with versions between 3.6 and 3.9. Python 2.7 was used with FRANC3D Versions 6 and 7, but Python 2.7 has reached the end-of-life status.

The Command Language & Python Extensions reference document provides additional details on setting appropriate environment variables as well as listing commands and their arguments.

9.1 Python PyF3D Module

The FRANC3D session log (see Section 8) can be converted to a Python script using the Fcl2Py executable that is shipped with FRANC3D. This program takes one argument, which is the session log file name. It reads the commands from the file, converts them to equivalent Python script, and writes the script to stdout, which can be sent to a file. For example:

➤ `C:/examples/Fcl2Py.exe session01.log > ses01.py`

This results in the ses01.py file with the following:

```
#!/usr/bin/python
import sys
sys.path.append("/home/bruce/franc3d") # added manually

import PyF3D

f3d = PyF3D.F3DApp()

# session log for tutorial 1

f3d.Submodeler(
    model_type="ABAQUS",
    orig_file_name='cube.inp',
    submodel_file_name='cube_LOCAL.inp',
    global_file_name='cube_GLOBAL.inp',
    elem_file_name='cube_RETAINED_ELEMS.txt')
```

```
f3d.OpenMeshModel(
    model_type="ABAQUS",
    file_name='cube_LOCAL.inp',
    global_name='cube_GLOBAL.inp')

f3d.InsertFileFlaw(
    file_name='cube_02.crk')

f3d.RunAnalysis(
    model_type="ABAQUS",
    file_name='cube_surf_v7.fdb',
    flags=["NO_WRITE_TEMP", "TRANSFER_BC", "NO_CFACE_TRACT", "NO_CFACE_CNTCT"],
    merge_tol=0.0001,
    connection_type="MERGE",
    command='abaqus job=cube_surf_v7_full -interactive -analysis ',
    global_model='cube_GLOBAL.inp',
    merge_surf_labels=["AUTO_CUT_SURF"],
    global_surf_labels=["GLOBAL_CONNECT_SURF"])

f3d.ComputeSif()
```

You can run this script from the command line:

➤ `python ses01.py`

This assumes that Python is in your PATH and that you are running from the folder where the `ses01.py` and corresponding FE model files reside.

The Python interface allows much more than simply re-running a session log. You can use all the Python capabilities to automate tedious tasks, such as inserting and growing a collection of initial cracks. There are Python example scripts on the FAC website.

The next section describes an additional Python extension that is embedded inside FRANC3D, which allows users to define their own crack growth model. This extension is accessible from the PyF3D module as well as from the GUI.

The PyF3D module is not accessible from the embedded extension.

9.2 Python Extension for User-defined Crack Growth

FRANC3D allows you to define your own crack growth rules using Python extensions. The full set of Python extensions is documented in Section 4 of the FRANC3D Commands & Python manual.

Using the cube model from the base tutorial, we briefly describe how to define Python extensions for crack growth. For this tutorial, we start from an initial crack model that has already been analyzed.

9.2.1 Step 1: Read Cube Crack Model

Start the FRANC3D GUI and select **File** → **Open**. Select the appropriate “cube_crack” model file (for example, for ABAQUS: *Abaqus_Cube_crack.fdb*). Select **Accept**. The model will be read into FRANC3D (along with the *.dtp* file that was created when running the static analysis).

9.2.2 Step 2: Read Python Extension

Use the FRANC3D **Advanced** menu to select the **Read User Extensions** option, Fig 9.1. The file selector dialog is displayed with a list of *.py* files. Select the *user_python.py* file, Fig 9.2. This file is described below.

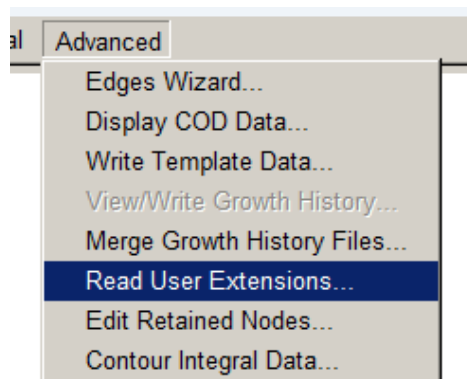


Figure 9.1 Read User Extensions menu option.

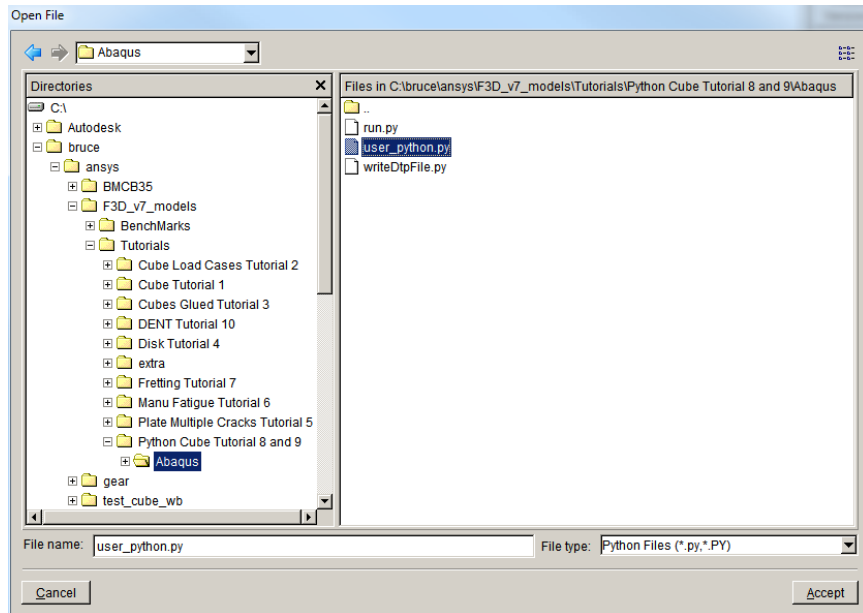


Figure 9.2 Select Python user file.

The *user_python.py* file should be in the working directory. FRANC3D reads the file and displays the list of functions that are included, Fig 9.3. The functions can be turned on or off as needed. Select **Accept** when ready to continue.

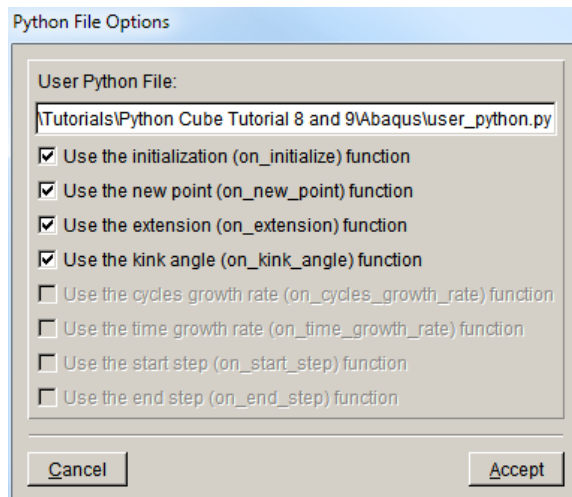


Figure 9.3 User Python functions available in the .py file.

9.2.3 Step 3: Grow Crack

From the FRANC3D **Cracks** menu, select **Grow Crack**, Fig 9.4. Use the default method to compute SIFs, Fig 9.5, and select **Next**. The User defined crack growth option is selected in the Growth Type dialog, Fig 9.6; this option is only available if you have loaded the Python extensions.

The subsequent dialog, Fig 9.7, shows the computed new crack front points. The options in this dialog are mostly the same as for the “built-in” crack growth dialog; the difference is the extension scale factor in the Crack Extension subpanel, which allows the relative extension to be adjusted without having to edit the Python functions to recompute growth. Subsequent crack growth, remeshing and analysis is the same as for the built-in crack growth routines.

Note that if you need to restart/continue crack growth after reading a *.fdb* file, you will need to reload the Python user extensions.

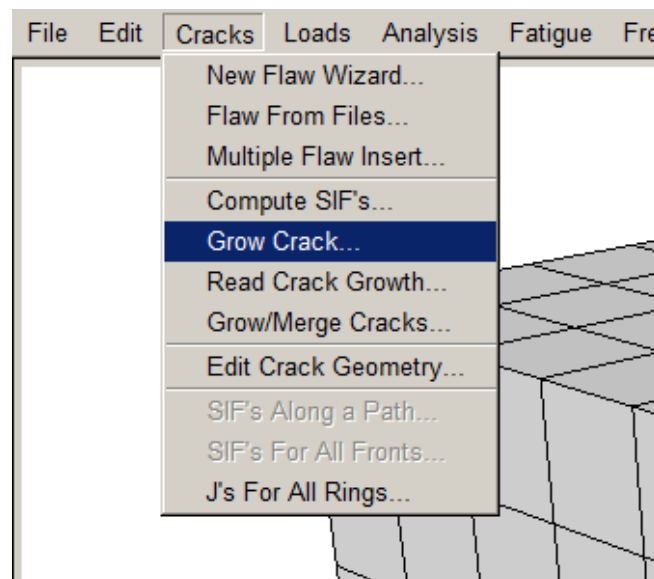


Figure 9.4 Cracks – Grow Crack menu option.

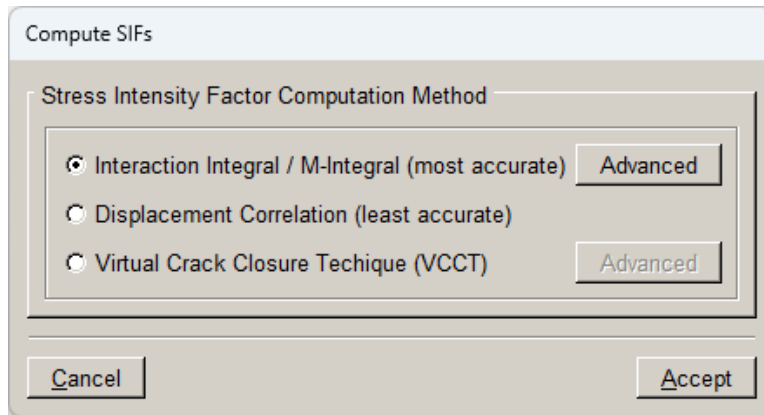


Figure 9.5 Compute SIFs dialog.

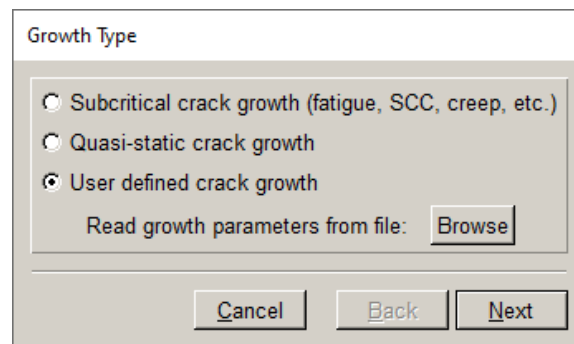


Figure 9.6 Growth Type dialog with User-defined crack growth selected.

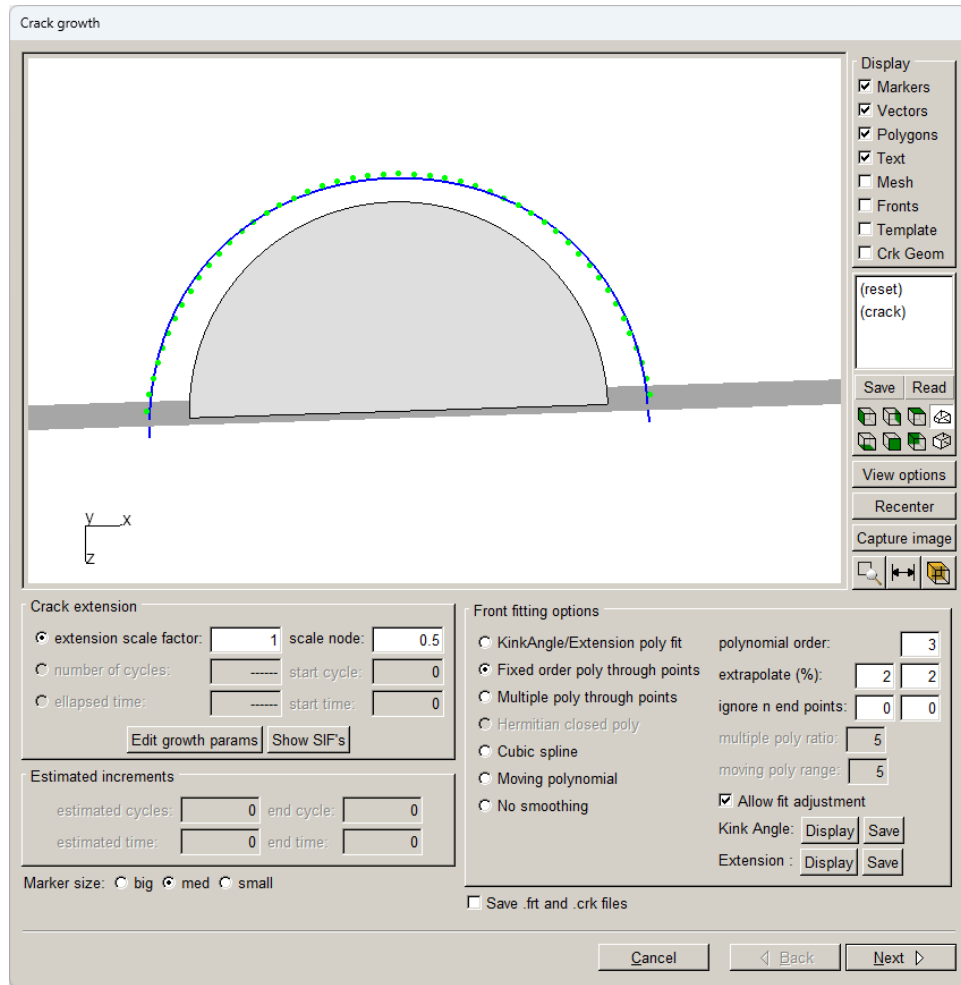


Figure 9.7 Crack Growth dialog for user-defined growth.

9.3 Python Extension File

The Python functions, in the *user_python3.py* file, are listed here. See the Command Language & Python Extensions document for a description of these functions and a list of all the functions. Each “on_” routine that is included in the file can be checked (see Fig 9.3) when the file is read into FRANC3D; the “on_” functions then can call other functions.

```
import math
```

```
# =====
# User defined crack extension function:
```

```

#
# This version computes an extension based on a temperature dependent
# Paris model for a specified number of cycles.
# =====
# -----
# initialization function
# -----

def on_initialize():
    global C_b, C_m, n, cycles

    C_b = 100.0
    C_m = 10.0
    n = 3
    cycles = 1000.0

    return

# -----
# crack extension function
# -----

def on_extension():
    global C_b, C_m, n, cycles

    C = C_m * GetTemp() + C_b
    dadN = C * GetK()[0]**n

    return dadN * cycles

# =====
# User defined kink angle function:
#
# This version computes the Max Tensile Stress criterion
# for the sum of the Ks from all load cases
# =====
# -----
# function to compute the hoop stress
# -----

def hoop_stress(theta, Ksum):

    KIlf = Ksum[0] * (math.cos(theta/2) * (1.0-math.sin(theta/2)**2))
    KIIlf = 0.75 * Ksum[1] * (math.sin(theta/2) + math.sin(3*theta/2))
    return KIlf - KIIlf

# -----

```

```

# kink angle function
# -----

def on_kink_angle():

    # compute the sum of the Ks for all load steps (note
    # load steps are numbered from 1 to n

    Ksum = Vec3D(0,0,0)
    for i in xrange(NumLoadSteps()) : Ksum += GetK(i+1)

    # call the F3D builtin Maximize function to find the
    # angle where the hoop stress is maximum

    max_ang = Maximize(-math.pi/2,math.pi/2,hoop_stress,Ksum)

    return max_ang

# =====
# User defined new point function:
#
# This version computes a new crack front point location
# based on a max hoop stress angle criterion and an assumed
# Paris like growth model
# =====
# -----
# new point function
# -----

def on_new_point():

    # compute the sum of the Ks for all load steps (note
    # load steps are numbered from 1 to n

    Ksum = Vec3D(0,0,0)
    for i in xrange(NumLoadSteps()) : Ksum += GetK(i+1)

    # call the F3D builtin Maximize function to find the
    # angle where the hoop stress is maximum

    angle = Maximize(-math.pi/2,math.pi/2,hoop_stress,Ksum)
    # compute an extension based on a Paris like model
    extend = 0.00005 * Ksum[0]**3.2

    return extend, angle

```

The ‘on_’ functions are turned on by the user when reading the *.py* file (see Section 9.2.2). The ‘on_’ functions can call other functions; for example, the `on_kink_angle()` function uses the `hoop_stress()` function. There are also built-in FRANC3D functions such as `Maximize()` that can be used. The user can define their own functions and use native Python functions.

10.0 Tutorial 10: Fatigue with Time Dependent Growth with a User-Mesh Crack

This tutorial illustrates subcritical growth of a crack under combined cyclic load and time dependent crack growth. The ABAQUS FE model is shown in Fig 10.1. ANSYS and NASTRAN FE models are similar.

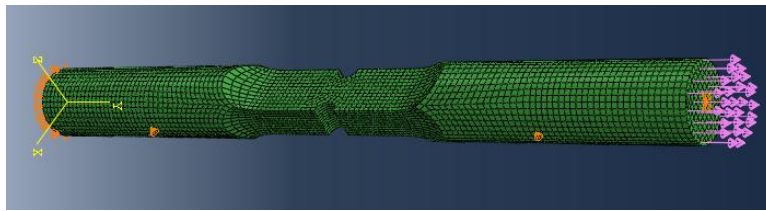


Figure 10.1 DENT specimen.

Two load steps are defined:

1. Tension: 17.4 MPa
2. Compression: -17.4 MPa

Crack growth is assumed to take place under the loading sequence shown in Fig 10.2. $K_{\max}$ and K_{dwell} will be the stress intensity factors associated with load step 1, and $K_{\min}$ will be the stress intensity factor associated with load step 2. Hold time, t_{dwell} , is 20 min/cycle.

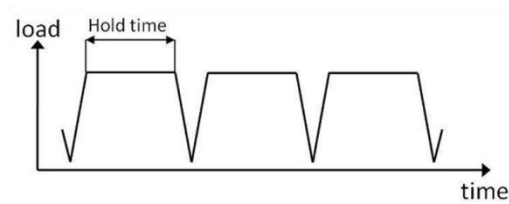


Figure 10.2 A constant amplitude load sequence including hold times.

The local crack extension for each load cycle will be determined from the expression:

$$\frac{da}{dN} = C \cdot \Delta K^m + \int_0^{t_{dwell}} A \cdot K_{dwell}^n dt$$

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

Parameters and constants for the crack growth analysis are:

Paris C 4.e-11
Paris m 3

Hold time A 5e-18
Hold time n 5
t_{dwell} 20

$\Delta K_{\text{threshold}}$ 1
K_c 10,000

with stress intensity factors in MPa*sqrt(mm), time in minutes, and da/dN in mm/cycle.

Temperatures units are set to degrees C, but the temperature is constant and not used.

10.1 SubModel and Crack Insertion

First, import the uncracked FE model file into FRANC3D and define a submodel.

Start FRANC3D and select **Import...** from the **File** menu. We want the default type of import, which allows us to divide the model into local and global models. Select **Next**. Select the appropriate FE file, and then select **Next**.

The **Define a Local Submodel** dialog will appear, Fig 10.3. Use the rectangular Rubberband Box to select the submodel, **C**rop the selection, and select **Next**. We save the local and global files using the default names; select **Next**.

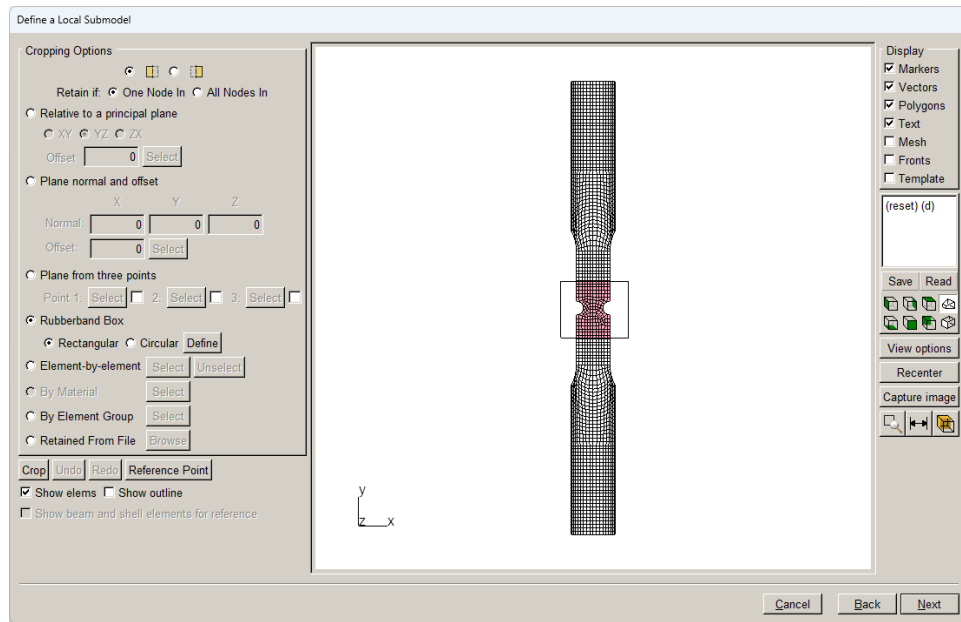


Figure 10.3 The Define a Local Submodel dialog – local submodel selected.

The Select **Retained BC Surfaces** dialog is shown next, Fig 10.4. There are no boundary conditions applied to the submodel, so just select **Next**. Set the SI units as mm, MPa and C and select **Finish**.

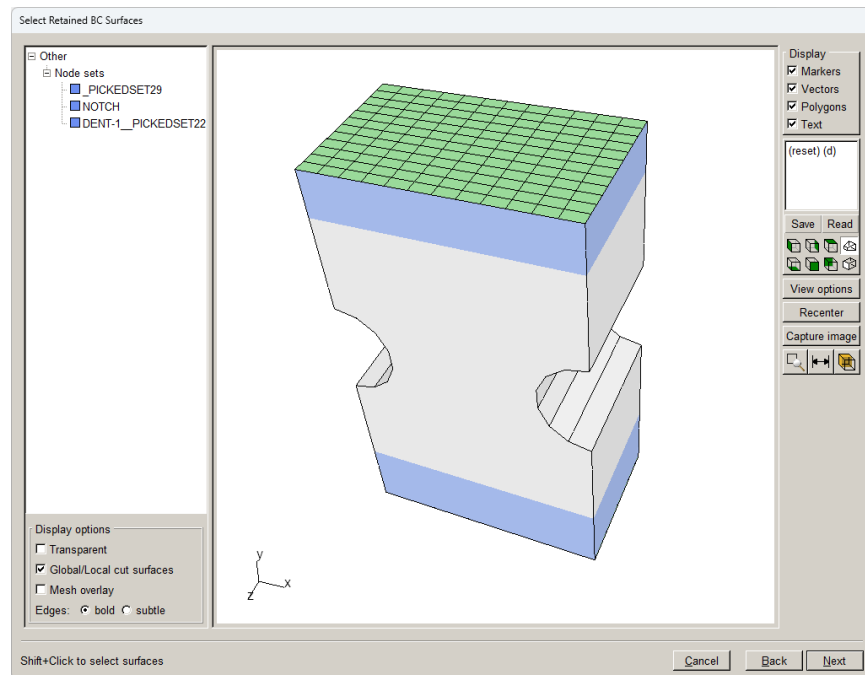


Figure 10.4 The Select Retained BC Surfaces dialog.

Use the **Cracks** and **New Flaw Wizard** menu option. Select Crack (zero volume flaw) in the first panel, and then select the user mesh crack type in the second panel. An ABAQUS surface mesh was created to define the initial crack, Fig 10.5. The crack front node set is defined by the red nodes in Fig 10.5. An ANSYS or NASTRAN surface mesh could have been created and used instead.

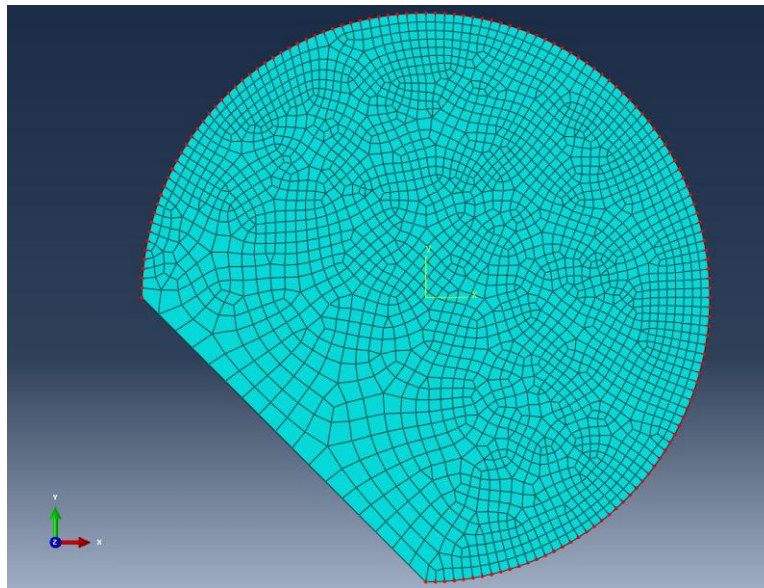


Figure 10.5 ABAQUS surface mesh with crack front node set.

Click the **Read Mesh File** button, Fig 10.6, and select the *dent_cae_crack_wfront.inp* file. Select **Next** to position the crack in the model.

The left panel of Fig 10.7 shows the location and rotation. The template mesh radius is set to 0.025. Select **Finish** to insert the crack and remesh. The remeshed model is shown in Fig 10.8; the Mesh is turned on.

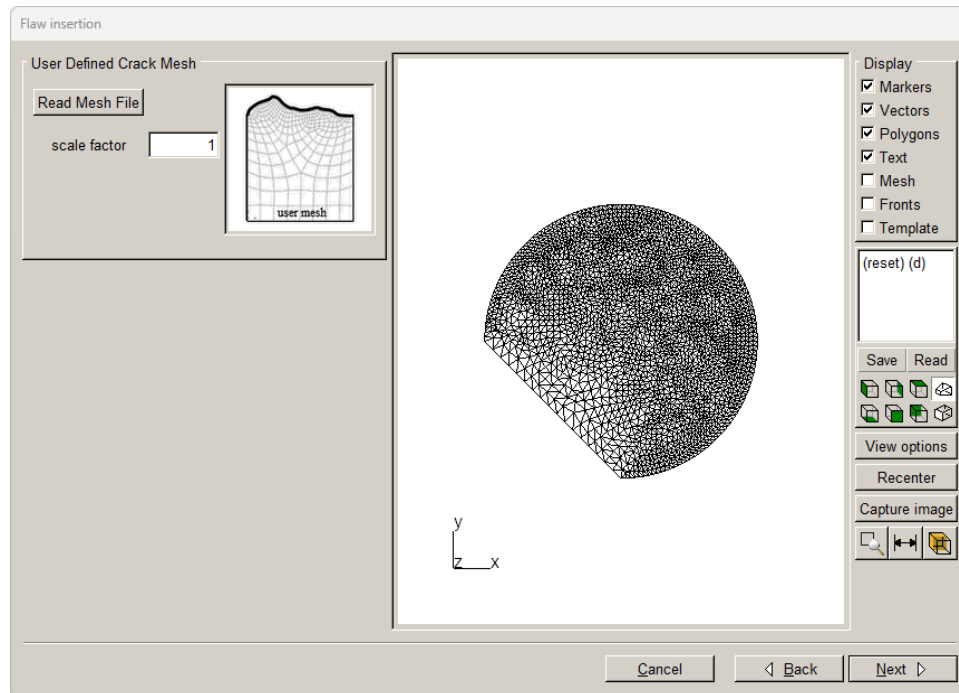


Figure 10.6 User-mesh crack – with front node set indicated.

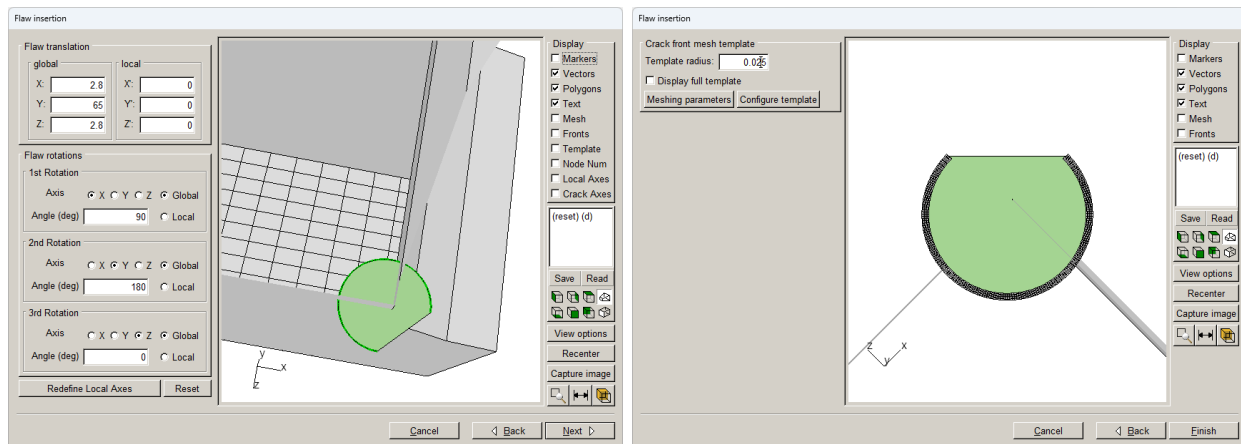


Figure 10.7 Crack location and front template mesh.

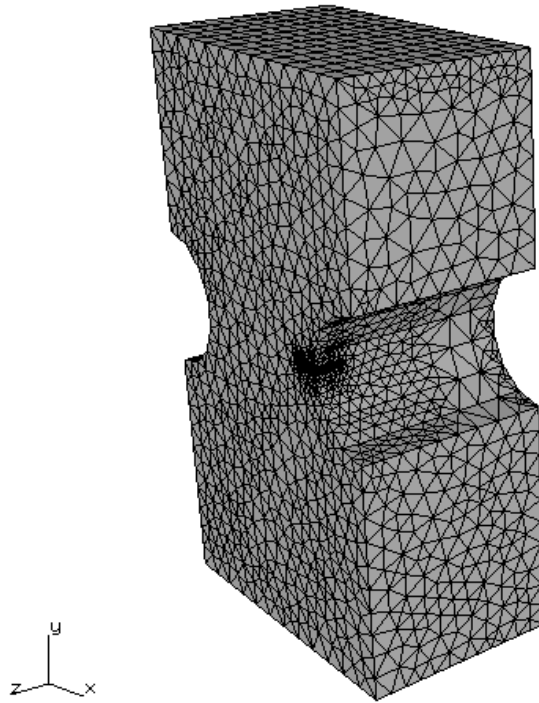


Figure 10.8 The mesh after crack insertion.

10.2 Define Crack Growth Parameters

The next step is to perform a crack growth analysis. Typically, we would run the initial crack using the **Static Crack Analysis** option and then grow the crack manually to ensure that we have the growth model and parameters correct and consistent. For this tutorial, however, we skip immediately to automatic crack growth.

Select **Crack Growth Analysis...** from the **Analysis** menu. The first dialog shown is the Compute SIFs dialog. We use the default values, so select **Next**. The Growth Type dialog is shown next, Fig 10.9. The default option, Subcritical crack growth, is used; select **Next**.

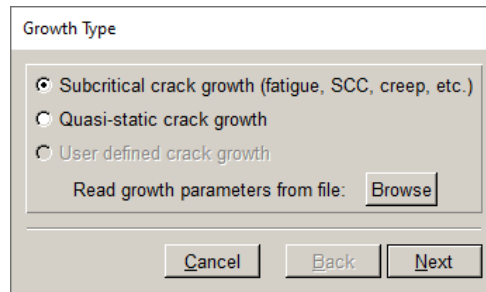


Figure 10.9 The Growth Type dialog.

The next dialog allows you to select the algorithm used to predict the kink angle during crack growth, Fig 10.10. We use the defaults; select **Next**.

Note that for most analyses, the default option, Max Tensile Stress (MTS), works well. This criterion predicts that the crack will grow perpendicular to the maximum local tensile stresses.

When subjected to significant shear loading at the crack front (K_{II} and K_{III}), some materials show a transition from tension dominated crack growth to shear dominated crack growth. The Max Shear Stress (MSS) criterion predicts that the crack will grow in the direction where the shear stress is maximum. The Generalized Stress criterion computes both the tensile and shear stresses and predicts the direction of the greater. The Strain Energy Release Rate option predicts that the crack will grow in a direction that is a combination of tensile and shear stresses. The Planar option forces the crack growth to be planar. See Section 6.5 of the Reference document for more information.

Kink angle model

Kink angle model

☒ Max tensile stress (MTS) $\max(K_I^r(\theta))$

☐ Max shear stress (MSS) $\max\left(\sqrt{(\eta_{II} K_{II}^r(\theta))^2 + (\eta_{III} K_{III}^r(\theta))^2}\right)$

☐ Generalized stress $\max(MTS, MSS)$

☐ Strain energy release rate $\max\left(K_I^r(\theta)^2 + (\eta_{II} K_{II}^r(\theta))^2 + (\eta_{III} K_{III}^r(\theta))^2\right)$

☐ Planar $\theta = 0$

☐ User defined model

☐ Kink angle limit Maximum kink angle (deg):

Non-proportional kink angle parameter ☒ ΔK ☐ K_{max}

Mixed mode eta factors η_{II} : η_{III} :

Crack growth resistance

☐ Anisotropic toughness

Figure 10.10 The Kink Angle Model dialog.

The Subcritical Growth Parameters dialog is displayed next, Fig 10.11. The yellow triangles indicate that parameters must be set before you continue. Select the **Change** button on the right side of the FEM units to display the Units dialog, Fig 10.12. The SI units are MPa, mm, and C; time units are min.

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

Subcritical Growth Parameters

Units used in FE model
 stress: MPa length: mm temp: C time: sec Change

Crack Growth Load Schedule

New Schedule Read From File Wizard View/Edit Save To File

Crack Growth Rate Model

New Model Read From File View/Edit da/dN View/Edit da/dt Save To File

Mixed-mode equivalent K

☒ $K^{\text{equiv}} = K_I$
☐ $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{II} K_{II})^2 + (\gamma_{III} K_{III})^2}$

γ_{II} : γ_{III} :

sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective Delta K

☒ $\Delta K_{\text{eff}} = K_{\text{max}} - K_{\text{min}}$
☐ $\Delta K_{\text{eff}} = K_{\text{max}} - \max(K_{\text{min}}, 0)$

☐ Use load interaction model Set/Edit Parameters

Integration Options

☒ Accelerated counting
 ☐ Constant K for time integration

Cancel Back Next

Figure 10.11 The Subcritical Growth Parameters Dialog.

Specify the units used in FEM model

FEM model units

☒ SI units

 Length: ☒ mm ☐ m ☐ other

 Stress: ☒ MPa ☐ Pa ☐ other

 Temperature: ☒ C ☐ K ☐ other

☐ US customary units

 Length: ☐ inch ☒ other

 Stress: ☐ psi ☐ ksi ☒ other

 Temperature: ☐ F ☒ other

Units of time:

☒ sec ☐ min ☐ hour ☐ day ☐ year ☐ other

Cancel Back Finish

Figure 10.12 The Units dialog.

10.2.1 Load Schedule

Next we define a load schedule. A load schedule does two main things. First, it defines a mapping from the load steps defined in the FE model to stress intensity factors used to predict crack growth rates. Second, it defines the sequence with which these loads will be applied to the model.

Select the **New Schedule** button on the Subcritical Growth Parameters dialog to display the Load Schedule dialog, Fig 10.13.



Figure 10.13 The initial Load Schedule dialog.

A load schedule is comprised of a collection of load events. Load events are organized in a tree-like structure. The root of the tree is a Schedule event. Initially, the upper part of the dialog (Fig 10.13) shows one Schedule event, which is the tree root. A left click on the root event does two things: 1) it makes the **Add** button active, and 2) it brings the Repeat tab to the front in the lower part of the dialog.

Select the **Add** button to display the Event Type dialog, Fig 10.14. There are seven different types of event:

- Simple Cyclic – a simple $\Delta K = K_{\max} - K_{\min}$ cycle, where the $K_{\max}$ and $K_{\min}$ loadings are proportional, so $K_{\min} = R * K_{\max}$, and R is given by the analyst.
- Non-Proportional Cyclic – a $\Delta K = K_{\max} - K_{\min}$ cycle, where the $K_{\max}$ and $K_{\min}$ loadings are not proportional (they come from different load steps), and $R = K_{\min} / K_{\max}$ is computed.
- Transient – For a transient event, a list of analysis load steps, sub-steps (frames), or a combination are specified. FRANC3D will examine all the associated K 's and select $K_{\max}$ and $K_{\min}$ from among these to define $\Delta K = K_{\max} - K_{\min}$.
- Spectrum – For a spectrum event, one or more load spectra are specified. A load spectrum is a list of load multipliers used to define a sequence of $K_{\max}$ and $K_{\min}$ values, which are used to define a sequence of ΔK 's. If multiple spectra are specified, they are superimposed.
- Hold Event – The four load events defined above can include a hold (dwell) time. A hold event allows one to define a hold time without any companion load cycling.
- Dynamic Pairing – A dynamic pairing event will examine all SIFs and dynamically create max-min pairs.
- Schedule – A schedule event can contain child events and allow a tree-like structure to be constructed.

For this analysis, $K_{\max}$ comes from load step 1 and $K_{\min}$ comes from load step 2. Since these are two different load steps, a Non-Proportional Cyclic event is selected. Once the event type is set, the Load Schedule dialog is updated, Fig 10.15; the Load Steps tab is automatically selected. The yellow triangles indicate that the load steps associated with $K_{\max}$ and $K_{\min}$ must be specified. The **Select a load step** button brings up the dialog shown to the right in Fig 10.15. Load step 1 is selected from the “pull-down” list of load steps for the $K_{\max}$ load step. Set load step 2 for $K_{\min}$.

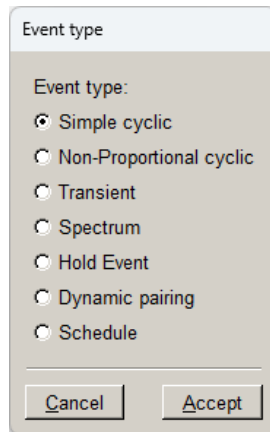


Figure 10.14 The select an event type dialog.

There is only one sub-step (frame) for this step, so the Load Sub-Step option is unavailable. A load and temperature scaling factor can be defined but are not needed here. Also, the **Sum Multiple Steps** button allows $K_{\max}$ and $K_{\min}$ to be defined as the sum of the K 's associated with two or more load steps (each with its own load multiplier). See the Reference document for more details.

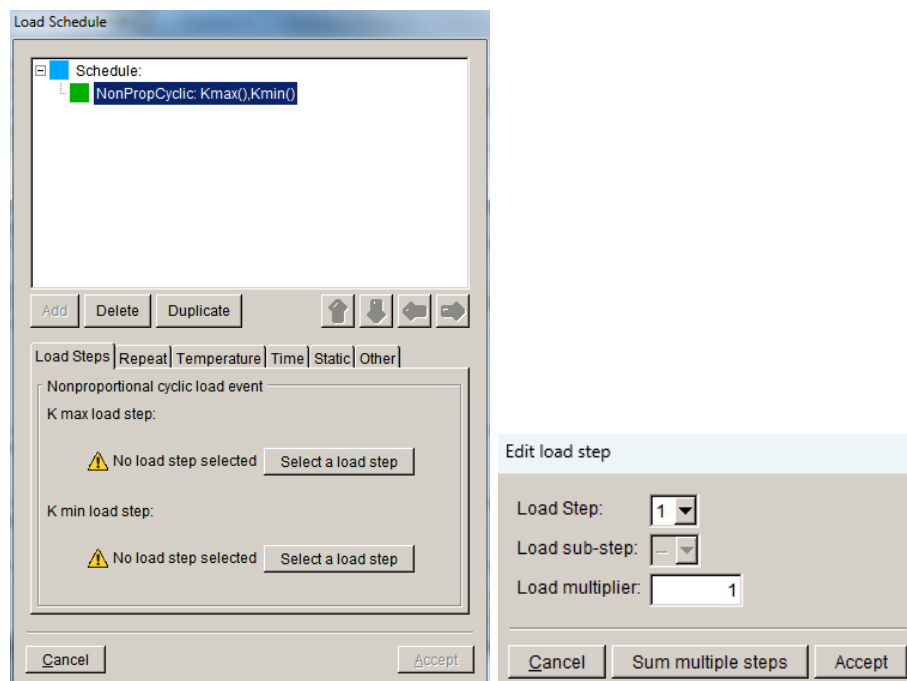


Figure 10.15 Load Schedule dialog with a non-proportional cyclic load event added.

The updated Load Schedule dialog, after selecting the load steps, is shown in Fig 10.16.

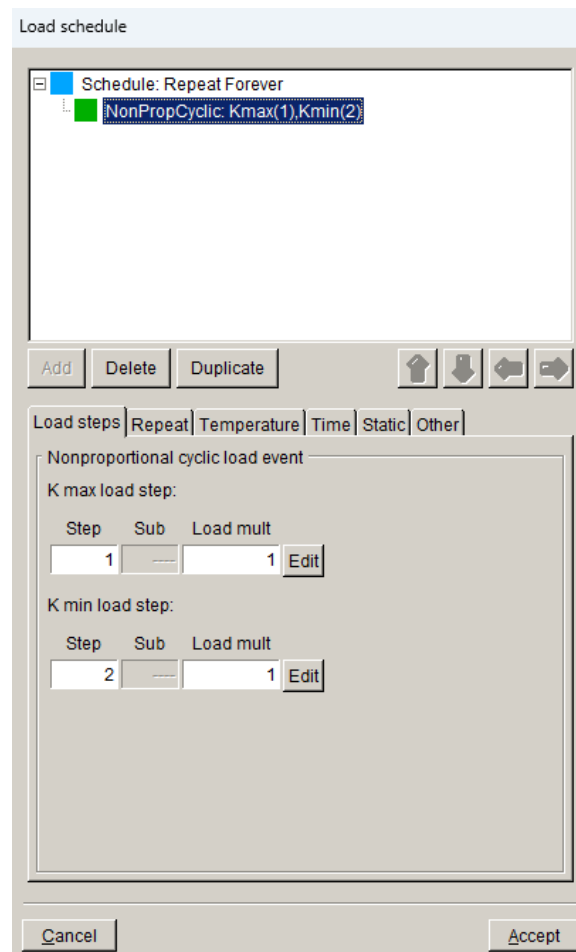


Figure 10.16. The Load Schedule dialog after load steps have been selected.

In this model, temperature is not assigned. The specified crack growth rate equation is not temperature dependent either. However, we briefly describe the settings for a temperature dependent model here for illustration. The Load Schedule dialog with the Temperature tab selected is shown in Fig 10.17, with Temperature dependent crack growth checked.

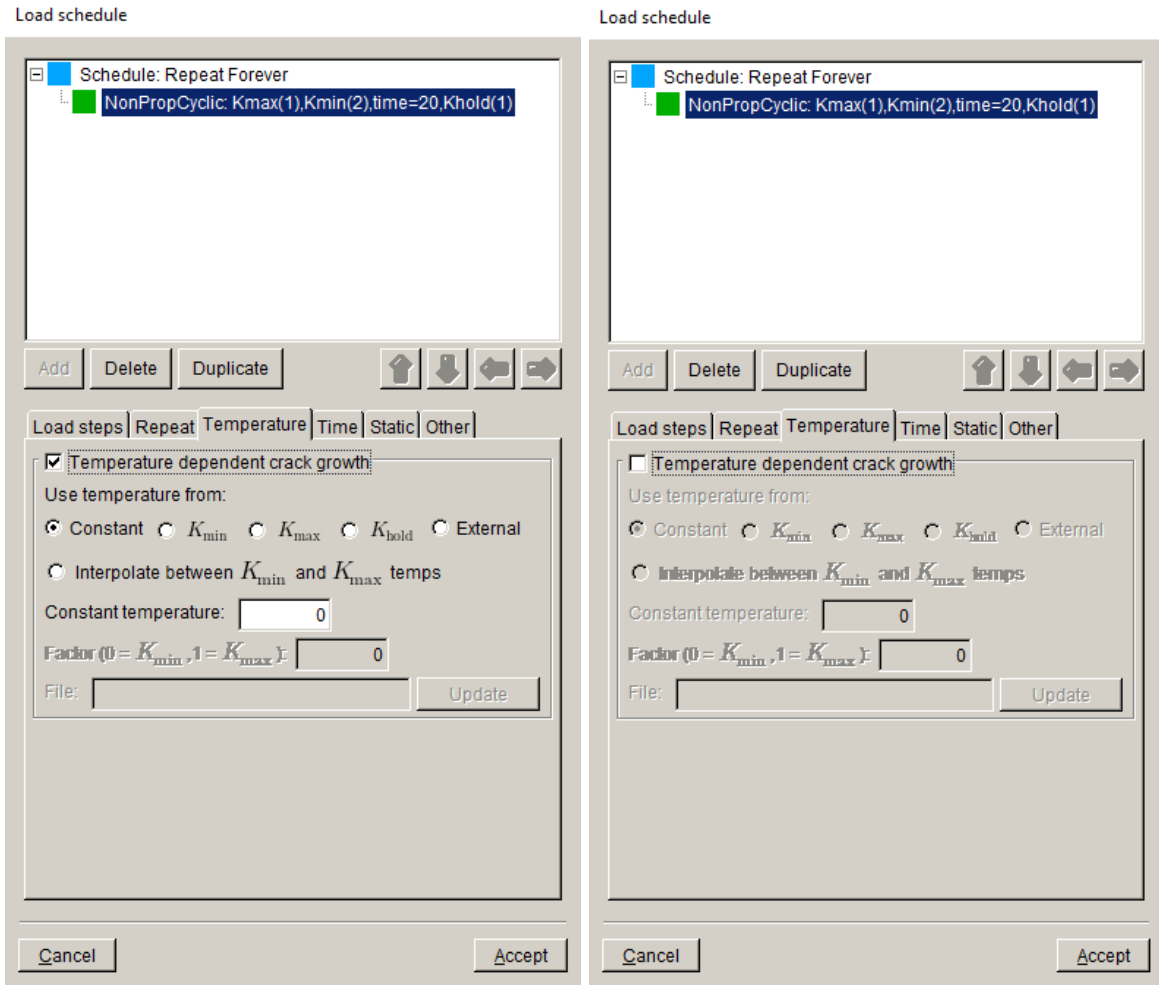


Figure 10.17. The temperature tab of the load schedule dialog.

Because predicting crack growth might involve many different finite element load steps, each of which can have associated temperatures, it is not obvious what temperature to use when evaluating a temperature dependent growth model. FRANC3D provides the following options:

- Constant – a constant temperature, provided by the user, is assumed for all crack front nodes, and is used for evaluating temperature dependent growth models.
- K_{min} – the crack-front temperatures associated with the load step used for determining K_{min} is used when evaluating growth models.

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

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

You should **uncheck the Temperature dependent crack growth option for this tutorial, Fig 10.17 – right panel.**

The analysis includes a hold (dwell) time for each load cycle. Simple Cyclic, Non-Proportional Cyclic, Transient, and Spectrum Load event types all allow a hold time. A hold-time can also be specified as an independent Hold event. In this tutorial, the hold-time is just added to the event.

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

For this example, the hold time is added as part of the *Non-Proportional Cyclic* event. Fig 10.18 shows the Load Schedule dialog after the Time tab has been selected, a hold time of 20 min has been entered, and the load step associated with the hold (load step 1) has been selected.

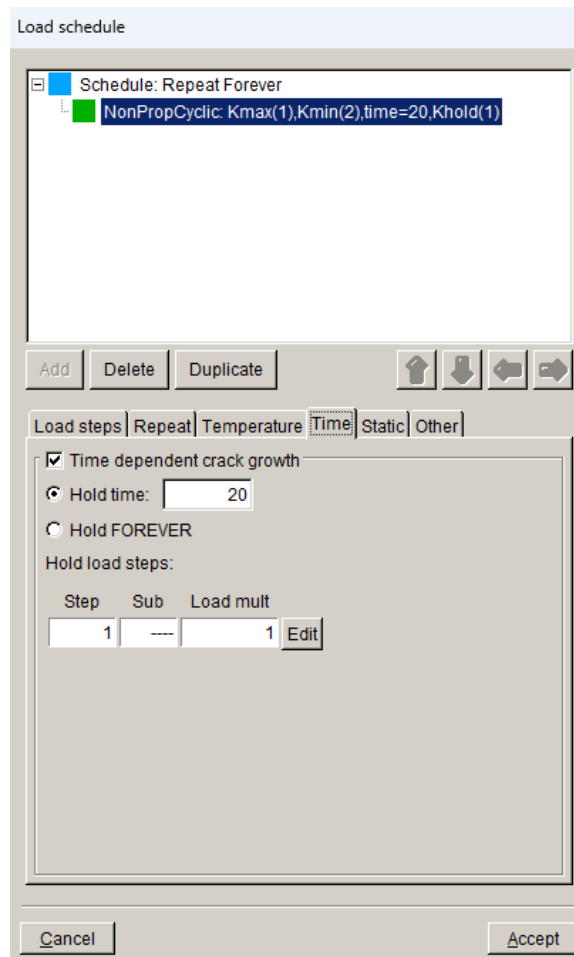


Figure 10.18. The load schedule dialog with the Time tab selected.

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

The Static tab is not used in this analysis. It allows the analyst to specify that the K from a load step (or sum of load steps) is to be added to both $K_{\max}$ and $K_{\min}$. This will not change the ΔK , but it will change the stress ratio, R . For example, this might be useful for models where there is a steady “fixed” load plus an alternating cyclic load, and the crack growth rate model is dependent

on *R*. As an example, a residual stress could be defined as a static load, and defined as a crack face traction load step, with stresses read from external analysis results (*e.g.*, *.inp* and *.odb* files). By default, a load event (load cycle, hold time, or combination) will be applied once. The Repeat tab allows the analyst to specify that the load event should be applied a finite number of times. It also allows one to specify that the event will be repeated **FOREVER**.

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

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

For this example, the load sequence is to be applied FOREVER, this could be specified either in the *Non-Proportional Cyclic* event or in the root *Schedule* event. In such a case, it is best to specify repeat FOREVER for the root *Schedule* event. This will allow FRANC3D to more easily determine if an accelerated cycle counting algorithm can be used when computing a fatigue life.

As shown in Fig 10.19, left click on the root Schedule event in the upper part of the dialog, and select Repeat FOREVER in the lower part. Once this is done, the load schedule is complete, so select **Accept** to return to the original dialog, Fig 10.11.

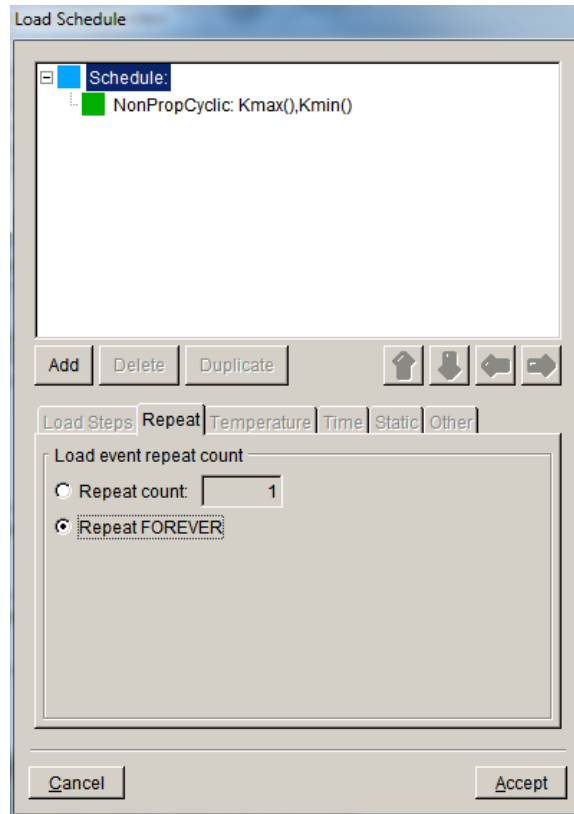


Figure 10.19 Load schedule dialog with Repeat FOREVER selected for the whole schedule.

10.2.2 Crack Growth Rate Model

The Subcritical Growth Parameters dialog (see Fig 10.11) requires a Crack Growth Rate Model. Select the **New Model** button to display the dialog shown in Fig 10.20. For this analysis, we define both a cyclic loading and a time dependent rate model, so check both options.

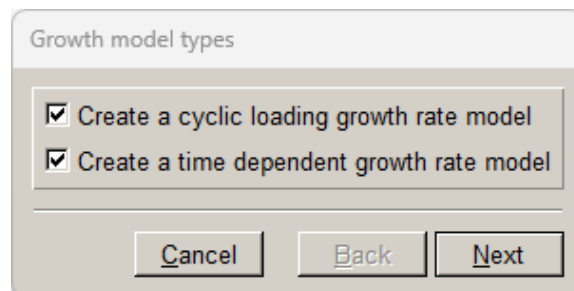


Figure 10.20 The growth model types dialog.

Select **Next** to display the Cyclic Loading Growth Model dialog, Fig 10.21. FRANC3D supports two general types of cyclic growth models: “paired” and “inclusive”. The difference between them is how stress ratio effects are incorporated. For a “paired” model, the shape of the crack growth rate curve (Paris, Sigmoidal, *etc.*) is selected independently from the stress ratio algorithm (Walker, Newman Closure, *etc.*). The “inclusive” models (two different versions of the NASGRO equation) include stress ratio effects.

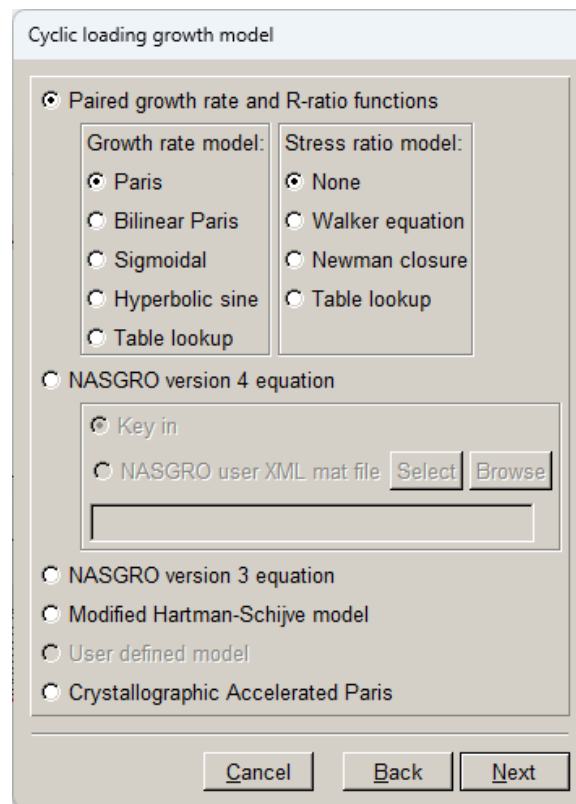


Figure 10.21. The cyclic loading growth model.

For this analysis, the Paris crack growth rate model is selected, with no stress ratio effects. Select **Next**; the next dialog allows you specify either a temperature dependent or independent model, Fig 10.22. For this analysis, the growth rate model is temperature independent. This means that only one set of growth rate parameters (C and m for the Paris model) are required. For temperature dependent models, different sets of growth model parameters are specified for

different temperatures, and FRANC3D interpolates among these values to find the appropriate growth rate for the crack-front temperature. Select **Next**.

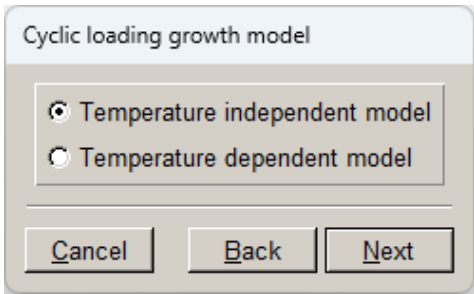


Figure 10.22. Temperature independent model selected.

The dialog shown in Fig 10.23 is displayed. The Model Label, and Description fields are optional. These might be useful if the model parameters are stored in a file for later reuse.

The units must be specified. The units need not be the same as the FE model. If the units are different, FRANC3D will automatically perform unit conversions. However, FRANC3D assumes that the units used within the growth model description (or within the finite element analysis) are self-consistent. For example, for a Paris model the units for C and K_c must match. For this model, we use the same units for the growth rate model as for the FE model. The Paris parameters are entered by selecting the fields and typing in the data.

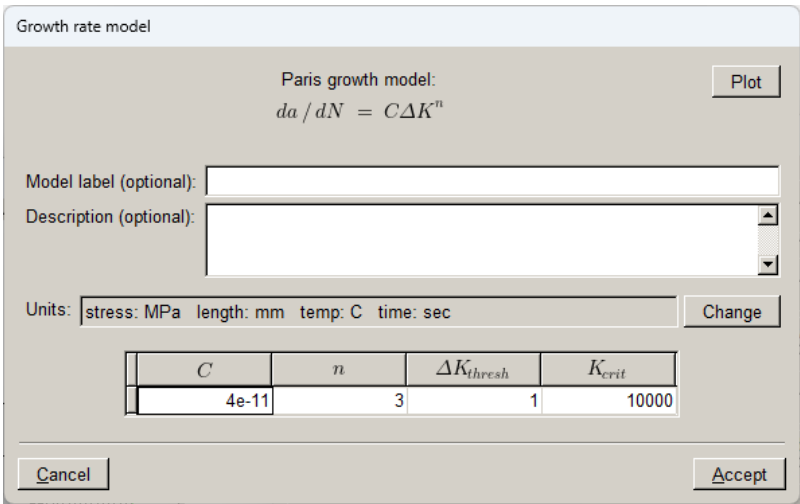


Figure 10.23. The dialog for entering the cycling loading growth model parameters.

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

After selecting **Next**, the dialog to select a time dependent growth model, Fig 10.24 is displayed. For this analysis, a Power Law model is used.

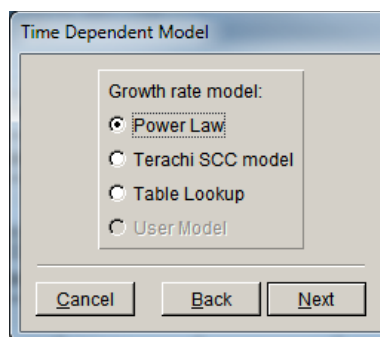


Figure 10.24. Dialog to select a time dependent growth model type.

After selecting **Next**, the dialog to select a temperature dependent or a temperature independent growth model, Fig 10.25, is displayed. In this case a temperature independent model will be used.

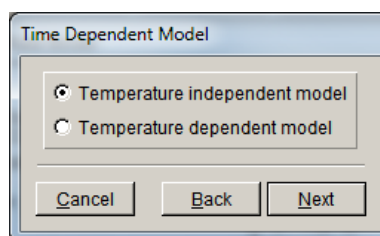


Figure 10.25 Dialog to specify a temperature dependent or independent growth model.

The dialog to specify the power law time dependent growth model parameters is shown in Fig 10.26. Again, as for cyclic loading, Units must be specified; we use the same FE model units.

For time dependent rates, the threshold (K_{th}) is an actual K value rather than a ΔK value. If K_{hold} is below this value for any crack front point, there will be no time dependent crack extension for that point. Select **Finish** to return to the Subcritical Growth Parameters dialog, Fig 10.27.

Growth rate model

Growth rate model:
 $da / dt = AK^m$

Model label (optional):

Description (optional):

Units: stress: MPa length: mm temp: C time: sec Change

A	m	K_{thresh}	K_{crit}
5e-18	5	1	10000

Cancel Accept

Figure 10.26 Dialog box for entering time dependent power law growth model parameters.

10.2.3 Effective dK and Integration

The lower half of the Subcritical Growth Parameters dialog, Fig 10.27, is described next.

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

Subcritical growth parameters

Units used in FE model
 stress: MPa length: mm temp: C time: min Change

Crack growth load schedule
New schedule Read from file Wizard View/Edit Save to file

Crack growth rate model
 da/dN: PARIS-NONE; da/dt: POWER_LAW;
New model Read from file View/Edit da/dN View/Edit da/dt Save to file

Mixed-mode equivalent K
☒ $K^{\text{equiv}} = K_I$ ☐ $K^{\text{equiv}} = \sqrt{K_I^2 + (\gamma_{II} K_{II})^2 + (\gamma_{III} K_{III})^2}$
 γ_{II} : γ_{III} :
 sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective delta K
☐ $\Delta K_{\text{eff}} = K_{\text{max}} - K_{\text{min}}$ ☒ $\Delta K_{\text{eff}} = K_{\text{max}} - \max(K_{\text{min}}, 0)$
☐ Use load interaction model Set/Edit parameters

Integration options
☒ Accelerated counting ☐ Constant K for time integration

Dynamic pairing metric
☒ $\Delta K_{\text{eff}}^{\text{equiv}}$ ☐ da/dN ☐ $\Delta CTOD$

Cancel Back Next

Figure 10.27 Subcritical Growth dialog completed.

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

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

This option is selected for this analysis.

The dialog box, Fig 10.27, also includes two options that affect how the crack growth integrations are performed when computing total life. If the Accelerated counting option is

selected (the default), FRANC3D will try to use an accelerated cycle counting algorithm based on Runge-Kutta integration. If this option is turned off, FRANC3D will perform cycle-by-cycle counting. The accelerated counting might be slightly less accurate than cycle-by-cycle counting, but usually the difference in the computed fatigue lives for the two methods is less than 1-2% of the total cycle count. On the other hand, for analyses where the per cycle crack extension is small, accelerated counting has been shown to give greater than 1000x reductions in the time required to compute total fatigue lives.

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

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

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

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

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

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

Select **Next** (see Fig 10.27) to display the dialog shown in Fig 10.28. You can save the parameters to a file. If a file is saved, it can be read with the **Read growth parameters from file** option in the dialog (see Fig 10.11).

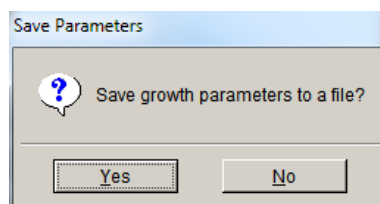


Figure 10.28 Save growth parameters to file option.

10.3 Crack Growth Analysis

The next dialog box, Fig 10.29, has three parts. The upper section allows the analyst to select the method used to fit an analytical curve through a collection of predicted new crack front points. The fitting serves two purposes. First, it tends to smooth out numerical noise in the predicted new crack front point positions. More importantly, the curve is extrapolated to find the exact locations where new crack fronts intersect free surfaces.

The middle part of the dialog allows the analyst to modify parameters regarding crack front templates and other meshing related options. FRANC3D determines a crack-front template radius from the initial crack size, but it might not always be suitable. We will set the template radius to use an absolute value of 0.025.

The lower part of the dialog defines the amount of crack extension for each crack growth step. The default is to specify the amount of crack extension for the point on the crack front that has the median predicted crack growth rate. This is the recommended option. The other two options are to specify either a fixed number of cyclic loads or a fixed amount of elapsed time. While these options are intuitive from an engineering perspective, the relationship between crack extension amount and either a cycle count or elapsed time can be highly nonlinear with respect to the crack size. Unless these values are carefully tuned, one can have a situation, within one analysis, where the predicted crack growth steps are extremely small for small cracks or extremely large for big cracks.

Fitting & Template Parameters

Front Fitting Options

☐ KinkAngle/Extension Poly Fit

polynomial order:

3

☒ Fixed Order Poly Through Points

extrapolate (%):

2

2

☐ Multiple Poly Through Points

ignore n end points:

0

0

☐ Hermitian Closed Poly

multiple poly ratio:

5

☐ Cubic Spline

moving poly range:

5

☐ Moving Polynomial

☒ Allow fit adjustment

☐ No Smoothing

Flaw Template

☒ use crack-front template

Template radius set as:
☒ absolute value
☐ % of crack increment

Template Radius:

0.025

Front:

1

☐ Set All Fronts

☐ Simple Intersections Only

Meshing Parameters

Advanced Options

Extension

☒ distance extension

scale node:

0.5

☐ cycles extension

☐ time extension

Cancel

Back

Next

Figure 10.29 Fitting and template parameters dialog.

The upper part of the next dialog, Fig 10.30, allows the analyst to set the criteria that will be checked for stopping a crack growth simulation. The analysis will stop when the first of these conditions is reached. The first three criteria will always be checked and cannot be turned off. The first says that if the load schedule does not contain any load events with the repeat count set to FOREVER, FRANC3D will take one pass through the load schedule and stop (note that multiple crack growth steps might be required for a full pass through the schedule). The second says that crack growth will stop when $K_{\max}$ for any point on a crack front reaches a critical value. The third says that crack growth will stop if the ΔK (cyclic) and/or $K_{\min}$ (time dependent) for all crack front points fall below their respective threshold values.

Other criteria for stopping are shown in the dialog. With respect to the maximum crack depth, defining a “crack depth” for an arbitrarily shaped crack can be very ambiguous. The current algorithm that FRANC3D uses to determine a crack “depth” is not very robust, and might give surprising results, so it should be used with caution. You can set the Maximum number of growth steps to a large number (> 60) if you have sufficient time and computing resources.

Growth Plan

Growth Termination Criteria

- ☒ Completed load schedule
- ☒ Critical stress intensity factor
- ☒ Threshold stress intensity factor
- ☐ Crack front extension failure
- ☐ Maximum number of growth steps Steps:
- ☐ Maximum number of load cycles Cycles:
- ☐ Maximum elapsed time Time:
- ☐ Maximum crack depth Depth:

Specified Crack Growth Steps

☒ Constant Crack Growth Increments

Crack Growth Step Size =

☐ Linearly Increasing/Decreasing Crack Growth Increments

First Crack Growth Step Size =

Change in Step Size at Each Step =

☐ User Defined Crack Growth Increments



Figure 10.30 Growth termination and step size dialog.

The lower part of this dialog allows you to specify the nominal crack step increment. For this analysis we will use a constant crack increment for each crack growth step. FRANC3D determines this value from the initial crack size, so the suggested size might be poor. We set it to 0.05 for this model.

10.3.1 ABAQUS Analysis

The following dialogs, Figs 10.31a - 32a, allow you to set the base file name for the crack growth models, the analysis code options and the local + global connection.

Crack growth analysis

ABAQUS Options

File Names

Base filename: (avoid spaces and special characters)

Current crack growth step:

ABAQUS run time:

ABAQUS Executable:

Python Executable:

Python script (will run before ABAQUS):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact

ABAQUS command:

Figure 10.31a ABAQUS analysis code options.

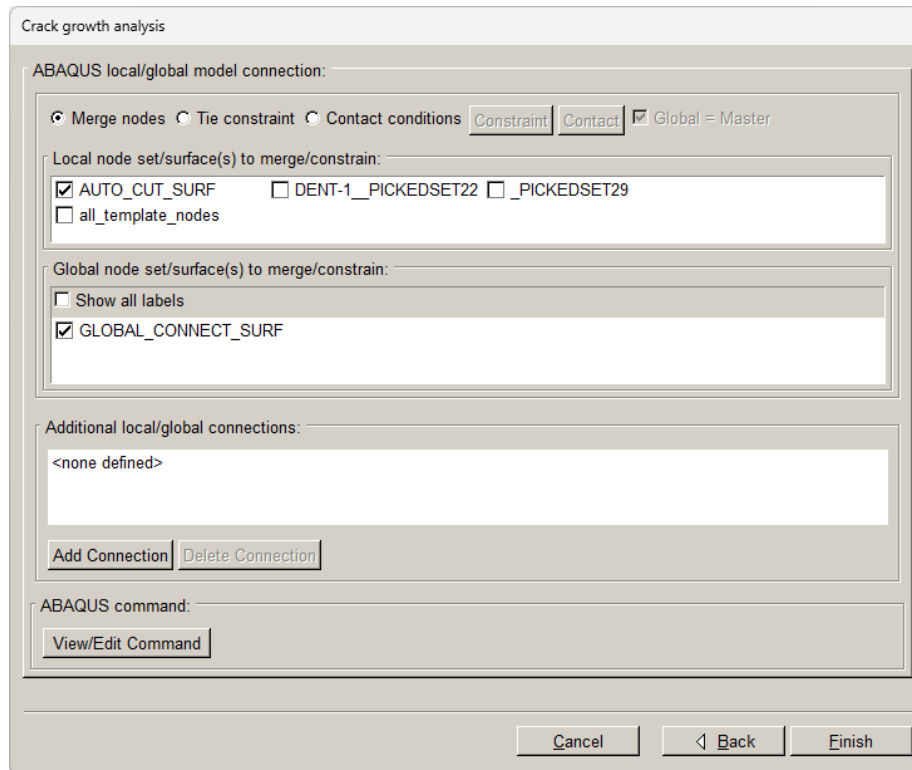


Figure 10.32a ABAQUS options for joining the global and local model.

10.3.2 ANSYS Analysis

The following dialogs, Figs 10.31b – 32b, allow you to set the base file name for the crack growth models, the analysis code options and the local + global connection.

Crack growth analysis

ANSYS Options

File Names

Base filename: (avoid spaces and special characters)

Current crack growth step:

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact ☐ Initial strain

ANSYS command:

Figure 10.31b ANSYS analysis code options.

Crack growth analysis

ANSYS local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☒ retain ☐ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF ☐ DENT-1_PICKEDSET22 ☐ _PICKEDSET29

☐ all_template_nodes

Global component(s) to merge/constrain:

☐ Show all labels

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

ANSYS command:

Figure 10.32b ANSYS options for joining the global and local model.

10.3.3 NASTRAN Analysis

The following dialogs, Figs 10.31c – 32c, allow you to set the base file name for the crack growth models, the analysis code options and the local + global connection.

Crack growth analysis

NASTRAN Options

File Names

Base filename: (avoid spaces and special characters)

Current crack growth step:

NASTRAN run time:

NASTRAN Executable:

Python Executable:

Python script (will run before NASTRAN):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions: ☐ Define crack face contact:

NASTRAN command:

Figure 10.31c NASTRAN analysis code options.

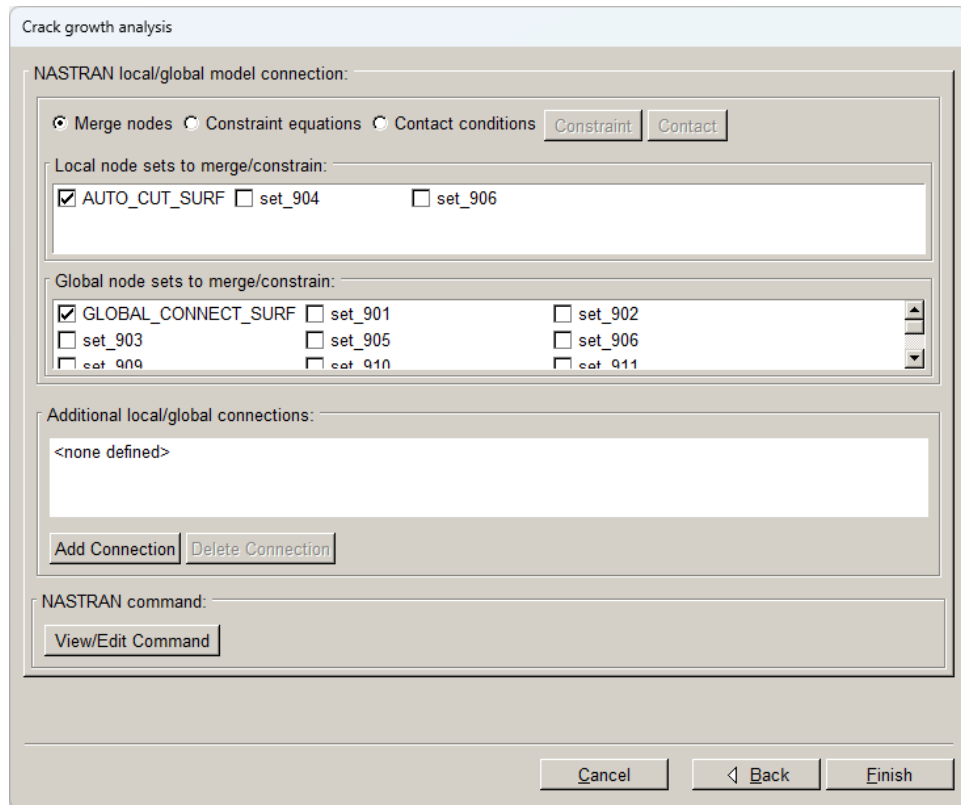


Figure 10.32c NASTRAN options for joining the global and sub-models.

10.4 Life Prediction

Without any user-defined stopping criteria, crack growth should continue for the user-supplied maximum number of steps; growth might fail as the crack transitions around the corner of the model. After sufficient crack growth has been completed, you can examine the SIF history and compute fatigue life. Use the **Cracks** and **SIFs Along a Path** menu to display the SIF history, Fig 10.33.

Use the **Fatigue** → **Fatigue Life Predictions** menu option to display the Fatigue Life dialog, Fig 10.34. The number of load cycles and total time are displayed at the bottom right. Crack growth can continue beyond what is shown here. If you only compute 4000-5000 cycles (for the amount of crack growth shown in Fig 10.34), check your ΔK_{eff} setting.

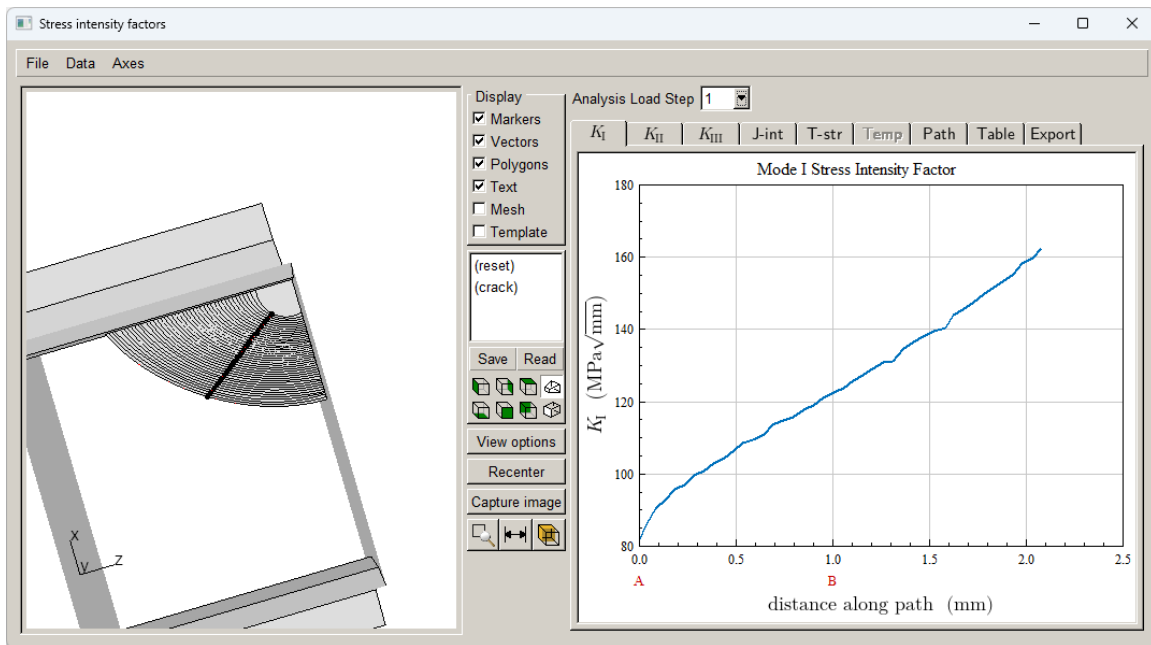


Figure 10.33 SIF history.

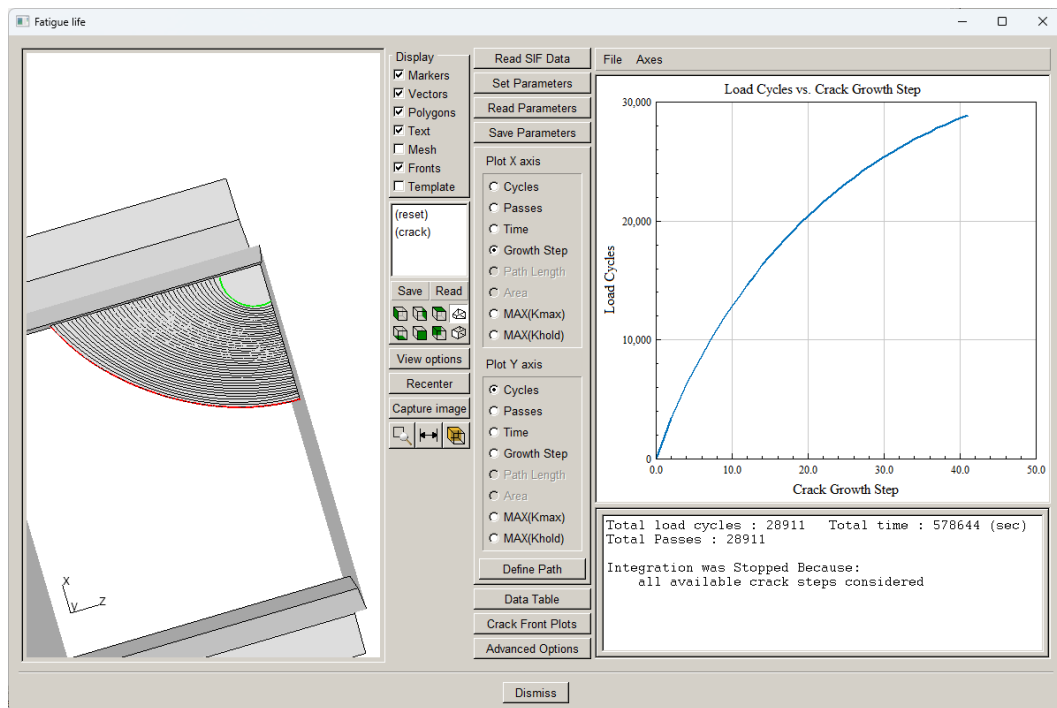


Figure 10.34 Fatigue life.

11.0 Tutorial 11: Stress Corrosion Crack

A simple stress corrosion crack simulation is demonstrated in this tutorial, using a simple 2x2x2 ABAQUS cube model with constraint on the base surface, but no applied loads.

11.1 Import Uncracked Model

Start FRANC3D and select **File** → **Import**. In the dialog, Fig 11.1, select Import a complete model and click **Next**.

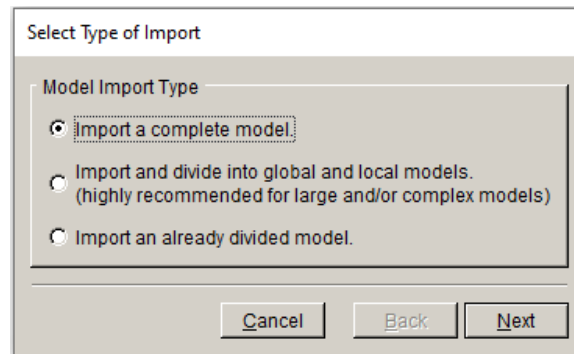


Figure 11.1 Import model type panel.

In the **Select Input Mesh File** dialog, set the Mesh File Type to ABAQUS, and select the file *cube_noload.inp*. The model contains one load step, which consists of displacement boundary conditions that fix all the nodes on the bottom (–y) face of the model. There are no loads.

In the Select Retained BC Surfaces dialog, Fig 11.2, click Select all to retain the mesh facets on the bottom surface. Select **Next** to set the units; we will use US units of inch, psi and F and set the time units as day. Select **Finish** to display the model in the FRANC3D main window.

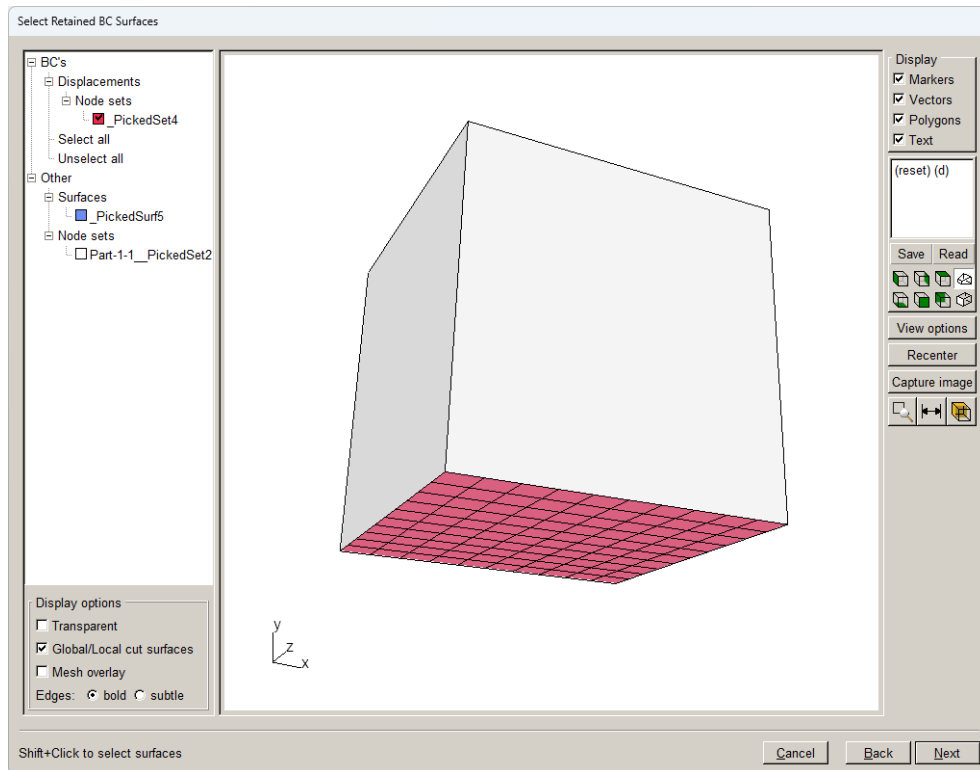


Figure 11.2 Select Retained BC Surfaces dialog showing the $-y$ surface mesh to be retained.

11.2 Insert a Crack

Insert a crack by selecting **Cracks** → **New Flaw Wizard**. On the first panel, select Crack and click **N**ext. On the next panel, the elliptical flaw should be selected already so select **N**ext. Define the elliptical semi-axes lengths of $a = 0.1$ and $b = 0.1$ and select **N**ext. The crack is located at $(0,0,2)$. Specify a flaw rotation of 90 degrees about the X axis, Fig 11.3, and select **N**ext. The default template radius should be prefilled to 0.01, select **F**inish.

The resulting cracked and remeshed model is shown in Fig 11.4.

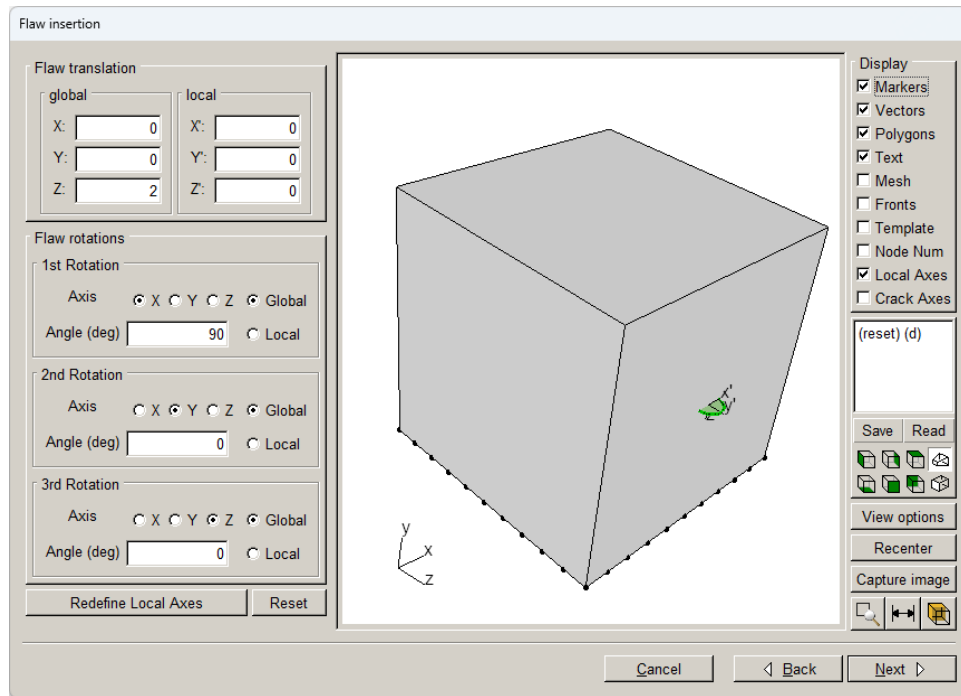


Figure 11.3 Initial crack location and orientation.

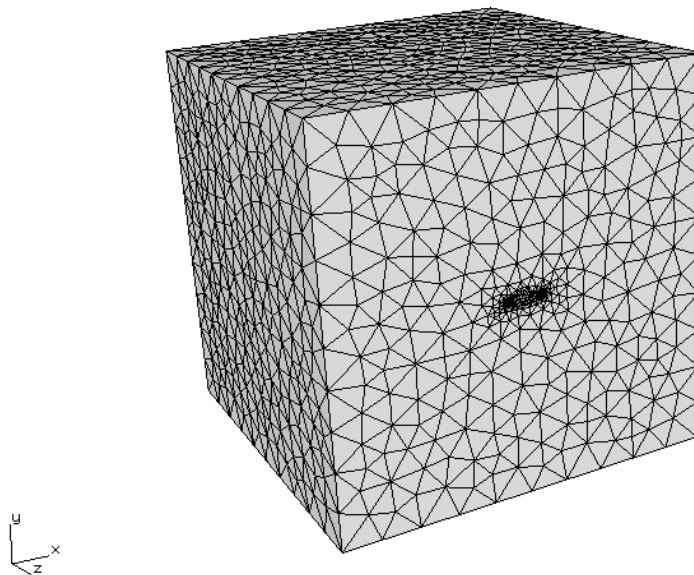


Figure 11.4 The cube model after crack insertion.

11.3 SCC Analysis

Select **Analysis** and **Crack Growth Analysis**. The Compute SIFs dialog is displayed first, Fig 11.5, select **Next**. Fig 11.6 shows the growth type dialog. Select Subcritical crack growth, and then click **Next**. Fig 11.7 shows the King Angle Model dialog. The Maximum Tensile Stress model is preselected so just select **Next**.

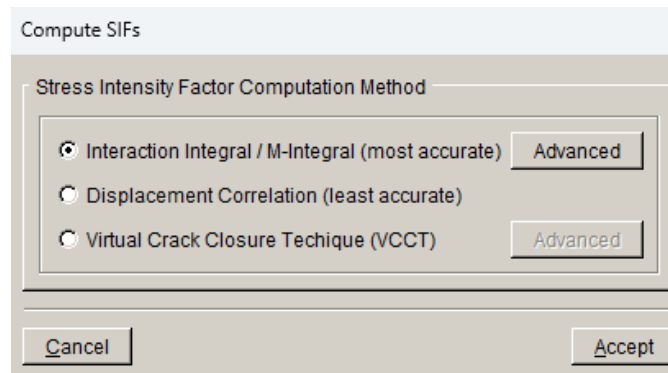


Figure 11.5 Compute SIFs dialog

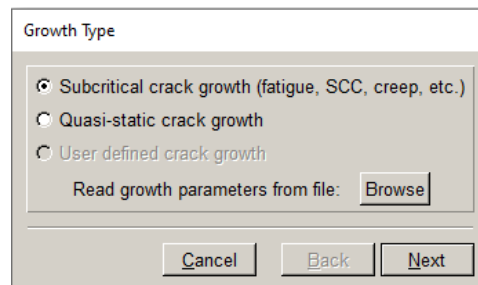


Figure 11.6 Growth Type dialog.

Kink angle model

Kink angle model

☒ Max tensile stress (MTS) $\max(K_I^r(\theta))$

☐ Max shear stress (MSS) $\max\left(\sqrt{(\eta_{II} K_{II}^r(\theta))^2 + (\eta_{III} K_{III}^r(\theta))^2}\right)$

☐ Generalized stress $\max(MTS, MSS)$

☐ Strain energy release rate $\max\left(K_I^r(\theta)^2 + (\eta_{II} K_{II}^r(\theta))^2 + (\eta_{III} K_{III}^r(\theta))^2\right)$

☐ Planar $\theta = 0$

☐ User defined model

☐ Kink angle limit Maximum kink angle (deg):

Non-proportional kink angle parameter ☒ ΔK ☐ K_{max}

Mixed mode eta factors η_{II} : η_{III} :

Crack growth resistance

☐ Anisotropic toughness

Figure 11.7 Kink Angle Model dialog.

The Subcritical Growth Parameters dialog, Fig 11.8, is then displayed. You can check the Units used in FE model by selecting the **Change** button. Fig 11.9 shows the Units dialog; select US customary units, and inch, psi, F, and day.

Set the Crack Growth Load Schedule. The load schedule can be built “by hand”, but we will use the load schedule wizard. Select **Wizard** (see Fig 11.8). Fig 11.10 shows the Schedule Type wizard panel.

Subcritical Growth Parameters

⚠ Units used in FE model
 stress: unset length: unset temp: unset time: unset Change

⚠ Crack Growth Load Schedule
New Schedule Read From File Wizard View/Edit Save To File

⚠ Crack Growth Rate Model
New Model Read From File View/Edit da/dN View/Edit da/dt Save To File

Mixed-mode equivalent K
☒ $K^{equiv} = K_I$ ☐ $K^{equiv} = \sqrt{K_I^2 + (\gamma_{II} K_{II})^2 + (\gamma_{III} K_{III})^2}$
 γ_{II} : γ_{III} :
 sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective Delta K
☒ $\Delta K_{eff} = K_{max} - K_{min}$ ☐ $\Delta K_{eff} = K_{max} - \max(K_{min}, 0)$

Integration Options
☒ Accelerated counting ☐ Constant K for time integration

Cancel Back Next

Figure 11.8 Subcritical Growth Parameters dialog.

Units

label

☐ SI units
 Length: ☒ mm ☐ m ☐ other
 Stress: ☒ MPa ☐ Pa ☐ other
 Temperature: ☒ C ☐ K ☐ other

☒ US customary units
 Length: ☒ inch ☐ other
 Stress: ☒ psi ☐ ksi ☐ other
 Temperature: ☒ F ☐ other

Units of time:
☐ sec ☐ min ☐ hour ☒ day ☐ year ☐ other

Cancel Accept

Figure 11.9 Units dialog.

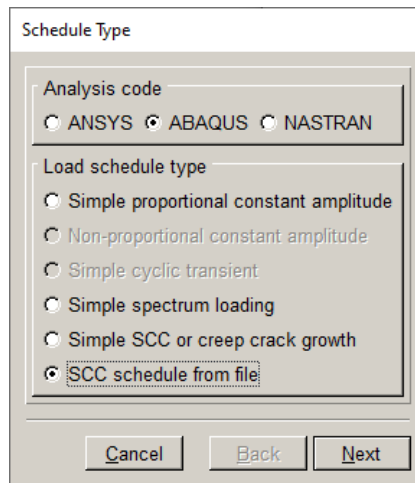


Figure 11.10 Schedule Type panel.

The Analysis code should be set to ABAQUS already because the imported model was an *.inp* file. For the load schedule type, choose SCC schedule from a file. Select **Next**, and then select the **scc_sched.txt** file and select **Next** again to display the data from the file, Fig 11.11.

The content of the **scc_sched.txt** file is:

Displacement File						
Title	(days)Time	inp file name	odb file name	Step#	Inc #	Scale Factor
Cube	2	uncracked.inp	uncracked.odb	1	-1	1
cube2	1	uncracked.inp	uncracked.odb	2	-1	1

Temperature File					(Deg F)
Inp file name	odb file name	Step#	Inc #	Uniform Temp	
-999	-999	-999	-999	300	
Uncracked_temp.inp	uncracked_temp.odb	1	-1	-999	

Note, the file contents are reformatted here to fit the margins in this document, the actual lines in the file are long continuous lines.

The first three lines and the last lines are comment lines that make the file more readable. The actual data is contained in lines 4 and 5. The File Data wizard panel is shown in Fig 11.11.

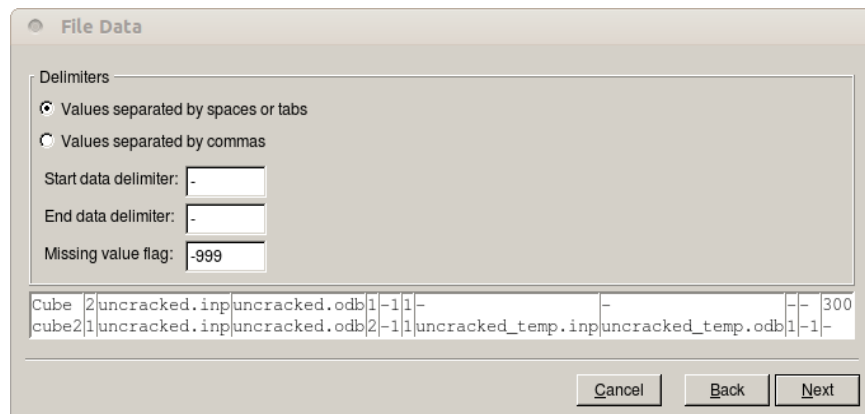


Figure 11.11 The File Data wizard panel.

FRANC3D starts reading from the beginning of the file. Lines will be ignored until a line beginning with the string in the Start data delimiter field is seen. This line will be ignored, but the subsequent line and following lines will be interpreted as data lines until a line beginning with the string in the End data delimiter field is seen. The default value for the delimiter is “-“. The data box near the bottom of the panel displays the lines that will be interpreted as data.

A string can be specified to indicate missing data. The default value is “-999”. In the data display region, missing values will be displayed as “-“. Data columns are defined as follow:

Column 1 (title): A user defined label for each hold time event. This is ignored by FRANC3D.

Column 2 (time): Hold time. This should be specified in the same time units set for the Units used in FE model dialog.

Column 3 (mesh file): The file that defines the mesh model corresponding with the stress file in the next column.

Column 4 (stress file): The file that defines the stresses for the displacement analysis. It is assumed that the (.odb for ABAQUS) file contains averaged nodal stresses.

Column 5 (step #): Stresses for this load step number will be extracted from the *.odb* file and applied as crack face tractions.

Column 6 (frame #): Stresses from this frame within the specified step number will be extracted from the *.odb* file. A value of -1 means that we use the values from the last available frame for this load step.

NOTE: following the ABAQUS convention, load steps are numbered starting with 1. Frames are numbered starting with 0.

Column 7 (scale): A multiplier that will be applied to the stresses extracted from the *.odb* file.

Column 8 (temp file): The base name for the files that define the model and the temperatures for the thermal analysis. It is assumed that the *.inp* file and the corresponding *.odb* file share this base name. It is further assumed that the *.odb* file contains nodal temperatures.

Column 9 (step #): Temperatures for this load step number will be extracted from the *.odb* file and used as crack front temperatures in the crack growth model.

Column 10 (frame #): Temperatures from this frame, within the specified step number, will be extracted from the *.odb* file. A value of -1 means that we use the values from the last available frame for this load step.

Column 11 (uniform temp): Specifies a uniform temperature to use for the crack front rather than extracting values from an *.odb* file.

After selecting **Next**, the dialog shown in Fig 11.12 is displayed.

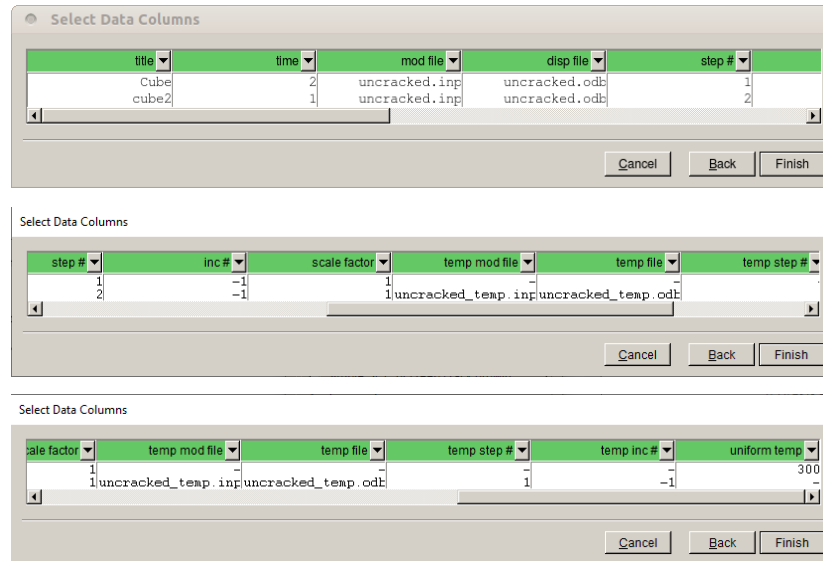


Figure 11.12 Select data columns dialog – single dialog split into three parts.

To make the program more flexible, the data need not be given in the column order described above. If all the data columns are present, they can be given in an order that is more convenient for the person preparing the file. This dialog allows a user to change the definition of the data given in each of the columns. Select **Finish** to continue.

Define a Crack Growth Rate Model by selecting the **New Model** button (see Fig 11.8) to display the dialog box shown in Fig 11.13. This allows one to define a crack growth rate model due to cyclic loading, time dependent crack growth, or both. In this case, select time dependent growth, and then click **Next**.

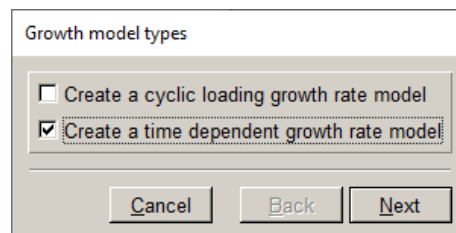


Figure 11.13 Growth Model Types dialog

There are four options for time dependent growth models, as shown in Fig 11.14. Select the Terachi SCC model and **Next**.

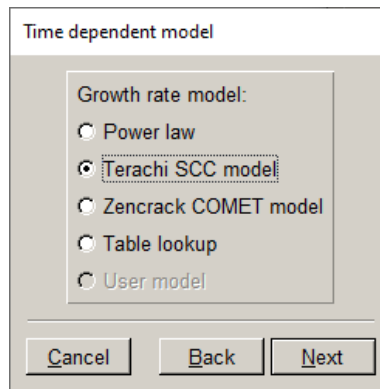


Figure 11.14 Time Dependent Model dialog.

The parameters for the Terachi model are defined in the dialog box shown in Fig 11.15. The model label and description are optional but might be useful if the model is saved and used for other analyses. The units must be set by selecting the **Change** button. For this tutorial, the units are the same as those specified for the FEM analysis (see Fig 11.9), inch, psi, F, and day.

In general, different units can be specified for the FEM analysis and the crack growth rate models and FRANC3D will perform any necessary conversions automatically.

Terachi SCC model:

$$da/dt = A\sigma_{ys}^m K_I^n e^{-Q/RT}$$

Model label (optional):

Description (optional):

Units: stress: psi length: in temp: F time: day Change

A	m	n	Q	K_{thresh}	K_{crit}	σ_{ys}
0.0001	1	2	10000	1	100	32000

Cancel Accept

Figure 11.15 The dialog box for entering parameters for the Terachi SCC model.

For this tutorial, enter the parameters as shown in Fig 11.15. These are not correct values for any known material, but they give crack growth rates that are not excessively large or small for this model.

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

Once parameters have been defined, the resulting crack growth rate model can be displayed by selecting the **Plot** button. The resulting dialog is shown in Fig 11.16. Options are available for displaying the growth rate model for different temperatures and yield stresses and for different units. Use the **Dismiss** button to continue.

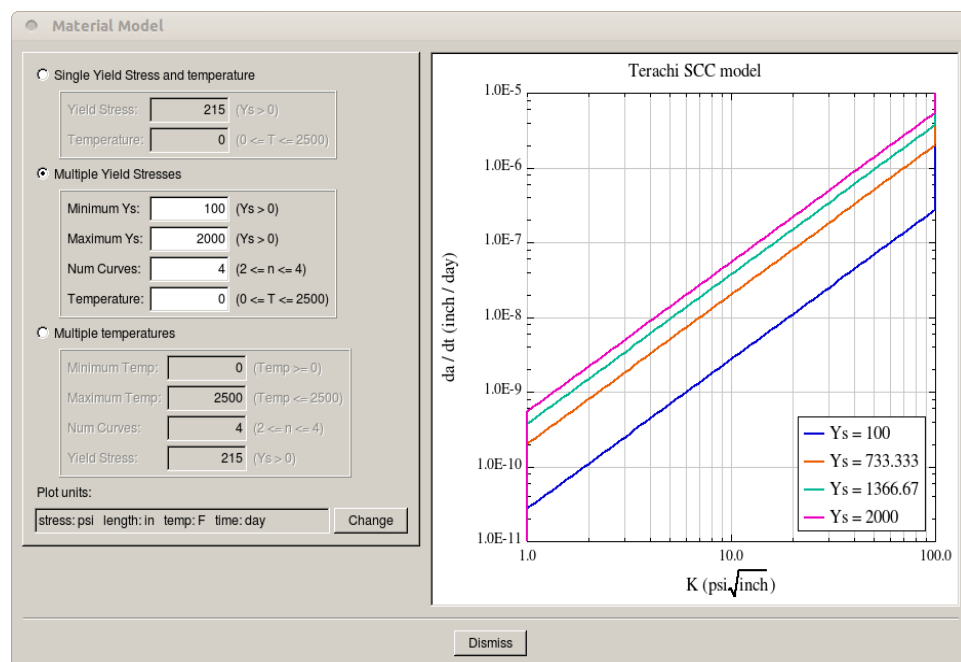


Figure 11.16 The crack growth rate model plot dialog.

After defining the growth model parameters, select **Finish**, and then select **Next** in the top-level dialog (see Fig 11.8). You will be given the option to save the specified crack growth parameters to a file, Fig 11.17. This file can be read for a subsequent analysis using the Read growth parameters from a file option (see Fig 11.6).

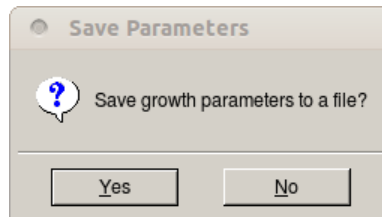


Figure 11.17 Save growth parameters to a file.

Set the crack front Fitting & Template Parameters, Fig 11.18. For this tutorial, most default values will work. Set the template radius to an absolute value of 0.01, and then select **Next**.

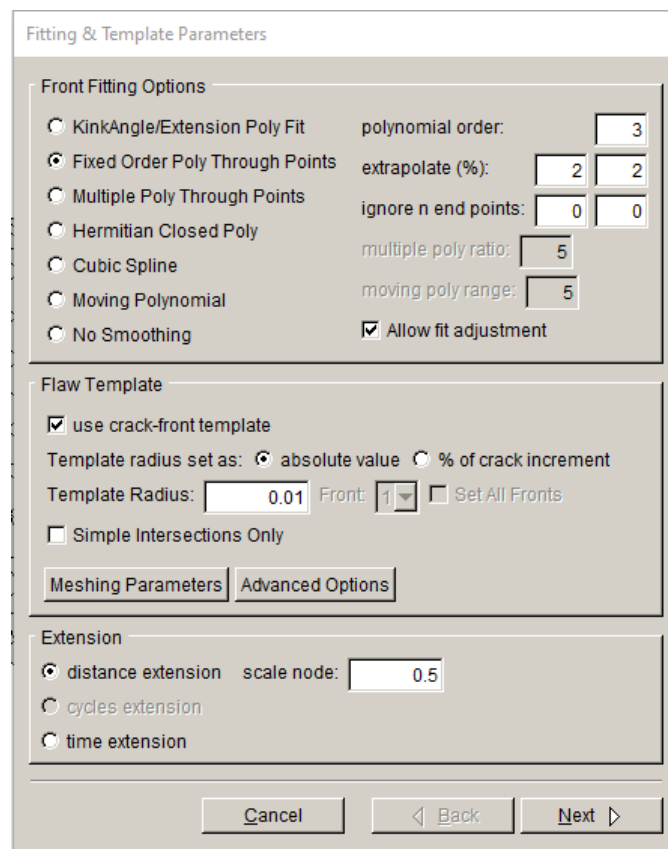


Figure 11.18 Fitting& Template Parameters dialog

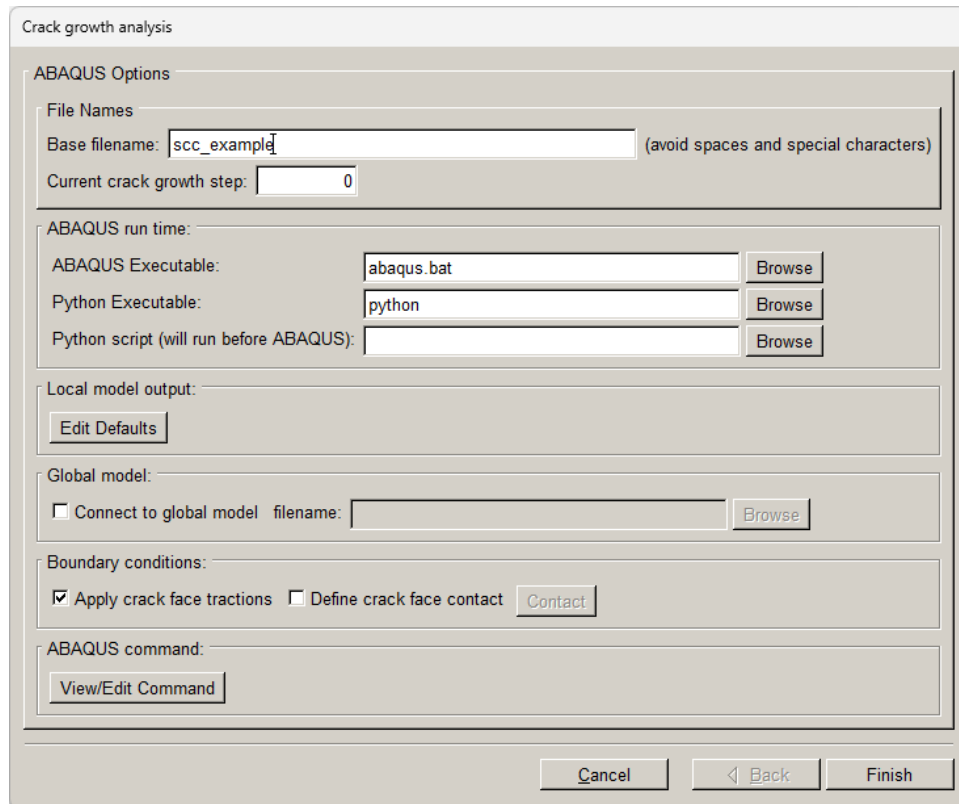
Set the growth plan, Fig 11.19, which includes the crack growth termination criteria and the crack growth step sizes. The crack growth analysis will always stop if we get to the end of the load schedule (after 3 days in this case, two days for the first entry in the scc_sched.txt file, and one day for the second entry), if the stress intensity factor goes above critical, or (not appropriate in this case, but for a cycle load analysis) if the stress intensity factor range goes below threshold.

We set the maximum number of growth steps to 5, just to guard against a runaway analysis for this demonstration. Other termination criteria include maximum number of load cycles (appropriate for cyclic loading), maximum elapsed time, and maximum crack depth; we leave all these unchecked. The default Constant Crack Growth Increment (step size) of 0.02 is appropriate for this analysis. Select **Next**.

The image shows a software dialog box titled "Growth Plan". It is divided into two main sections. The first section, "Growth Termination Criteria", contains several checkboxes: "Completed load schedule" (checked), "Critical stress intensity factor" (checked), "Threshold stress intensity factor" (checked), "Crack front extension failure" (unchecked), "Maximum number of growth steps" (checked, with a value of 5 in the adjacent "Steps:" field), "Maximum number of load cycles" (unchecked, with a value of 0 in the "Cycles:" field), "Maximum elapsed time" (unchecked, with a value of 0 in the "Time:" field), and "Maximum crack depth" (unchecked, with a value of 0 in the "Depth:" field). The second section, "Specified Crack Growth Steps", contains three radio button options. The first option, "Constant Crack Growth Increments", is selected, and its corresponding field "Crack Growth Step Size =" has a value of 0.02. The second option, "Linearly Increasing/Decreasing Crack Growth Increments", is unselected, but its fields "First Crack Growth Step Size =" and "Change in Step Size at Each Step =" both have a value of 1. The third option, "User Defined Crack Growth Increments", is unselected, and below it are two buttons: "Define Steps" and "Read From File" (with a warning triangle icon). At the bottom of the dialog are three buttons: "Cancel", "< Back", and "Next >".

Figure 11.19 Growth Plan dialog.

Set the base file name, Fig 11.20, to *scc_example*. The Current crack growth step is 0 representing the initial crack. The analysis options should not need to be changed, so select **Finish**. FRANC3D will write files and then execute ABAQUS in the background for each step of crack growth.



The image shows the 'Crack growth analysis' dialog box in ABAQUS. It is divided into several sections:

- ABAQUS Options:**
 - File Names:** Base filename: (avoid spaces and special characters). Current crack growth step: .
 - ABAQUS run time:**
 - ABAQUS Executable:
 - Python Executable:
 - Python script (will run before ABAQUS):
 - Local model output:**
 - Global model:** ☐ Connect to global model filename:
 - Boundary conditions:** ☒ Apply crack face tractions ☐ Define crack face contact
 - ABAQUS command:**
- Buttons:**

Figure 11.21 ABAQUS dialog.

When the automatic crack growth simulation has finished, we can examine the results. Select **Fatigue → Fatigue Life Predictions**, and the dialog box shown in Fig 11.22 is displayed. The Plot Time option should be preselected; this shows a plot of Time vs. Crack Growth Step. The plot on the left shows the crack growth. The red crack front is the estimated crack front location when the end of the load schedule is reached or when the analysis has stopped based on the stopping criteria. If you run additional steps of growth, the analysis should stop around growth step #8 when the full (time) schedule is used (or exceeded), Fig 11.23.

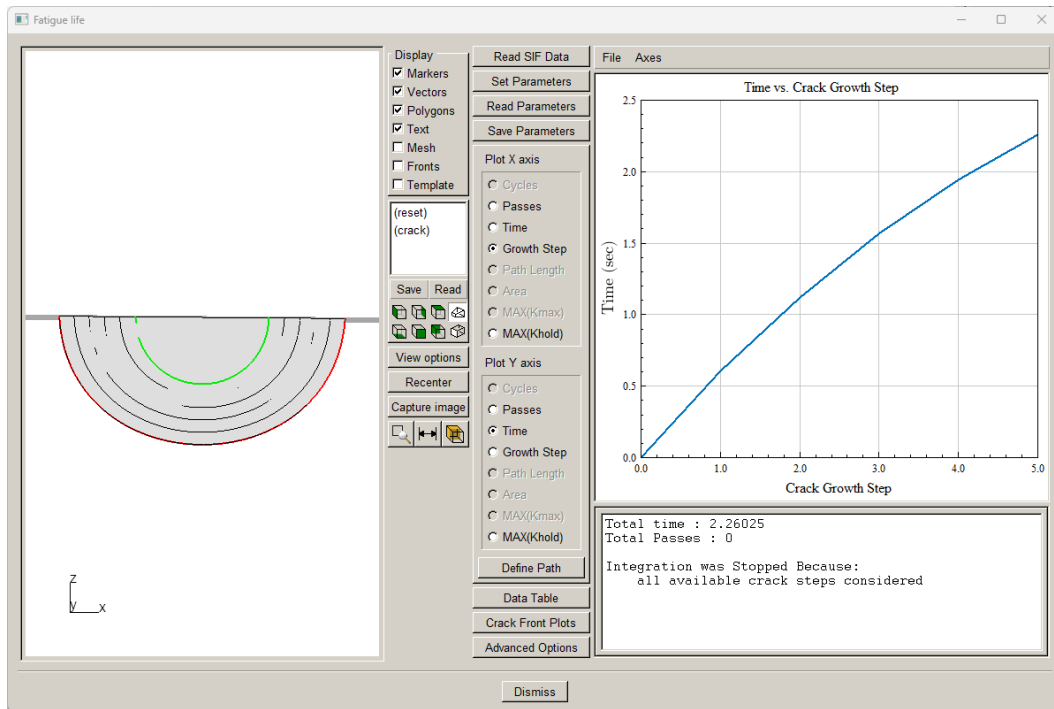


Figure 11.22 Fatigue Life dialog – time versus crack growth step.

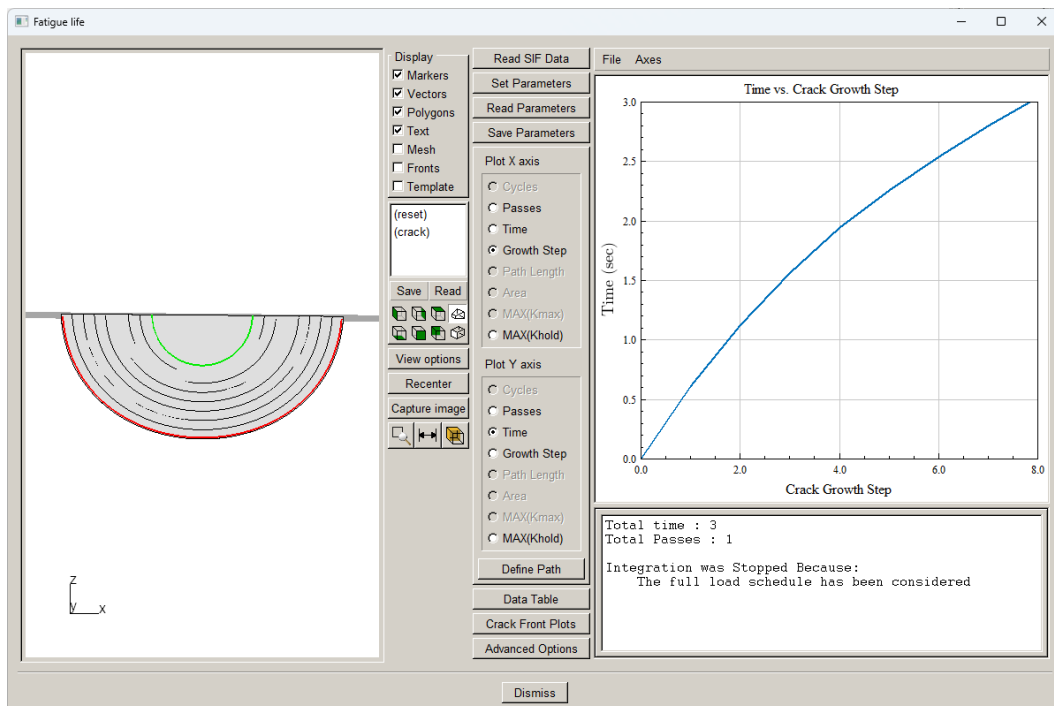


Figure 11.23 Fatigue Life dialog – after additional steps of crack growth.

12.0 Tutorial 12: Symmetry Crack

A simple symmetry-crack simulation is demonstrated in this tutorial, using an ABAQUS cube model with symmetry-constraint on the top surface and tension on the bottom surface, Fig 12.1. A “full” model can be created by doubling this cube (and removing the symmetry constraints); this will allow you to compare with the symmetry model SIFs.

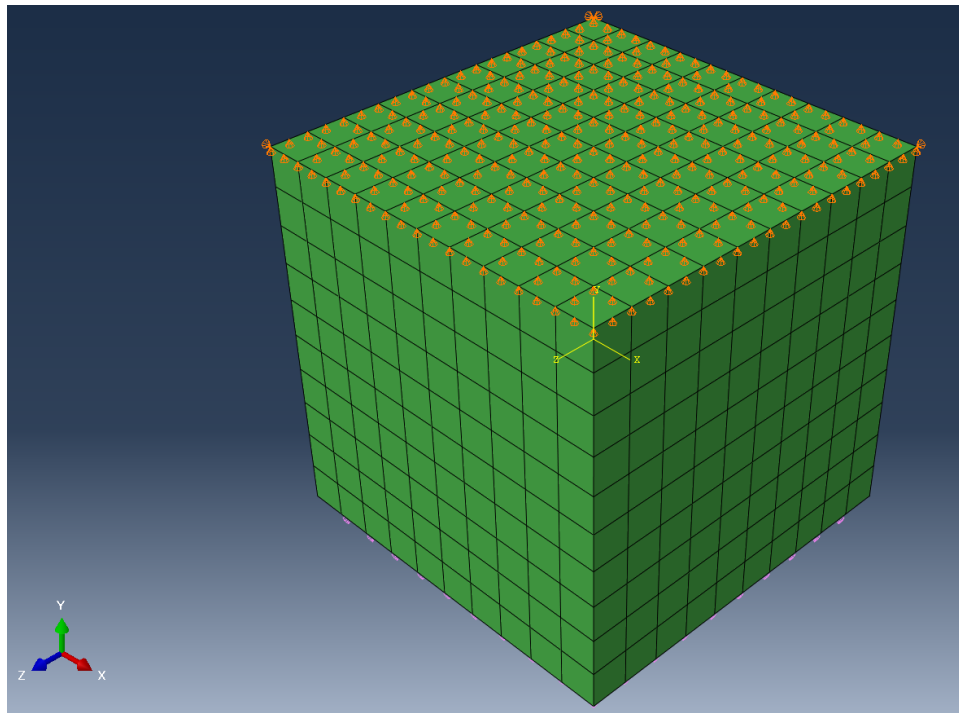


Figure 12.1 ABAQUS cube with symmetry boundary conditions.

12.1 Import Uncracked Model

Start FRANC3D and select **File** → **Import** and select Import a complete model and click **Next**. We select the *half_uncracked.inp* file, rotate the model and pick the bottom surface, Fig 12.2, to retain the mesh. The top surface has y-symmetry-constraint, and we will “insert” the symmetry

crack on this surface. Point constraints, which are far from the crack, on the top surface have been selected as well.

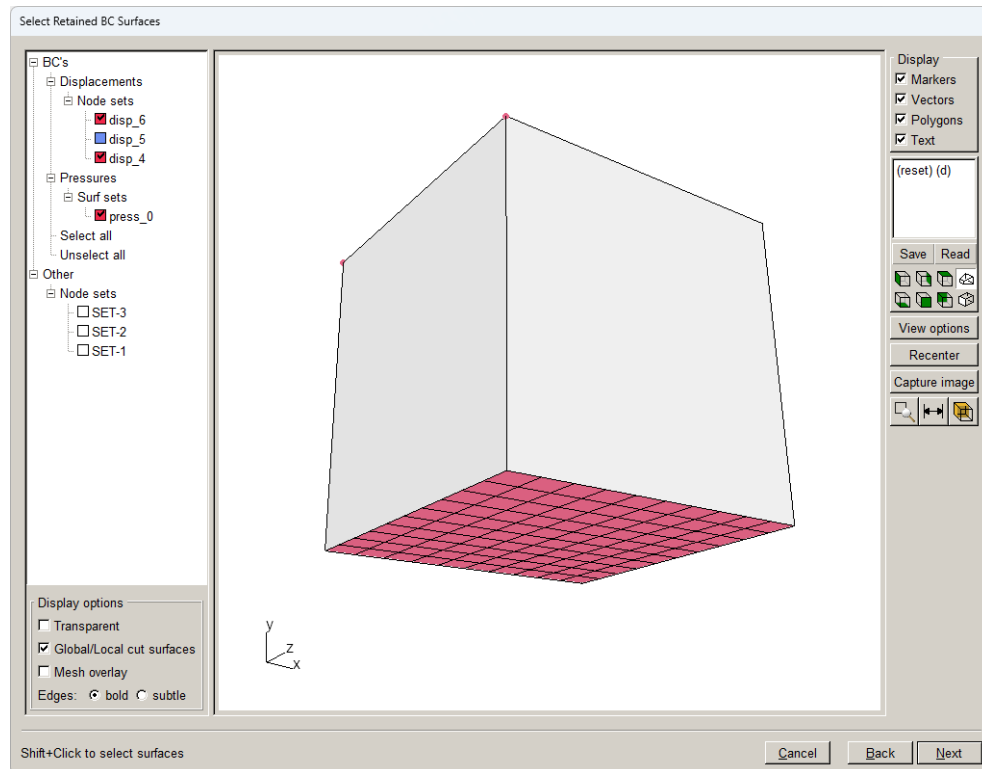


Figure 12.2 Select retained surfaces dialog.

You can set SI units to mm, MPa and C, and then select **Finish** to display the model in the main window.

12.2 Insert Symmetry Crack

Select **Cracks** → **New Flaw Wizard** from the FRANC3D main menu to display the New Flaw Wizard, Fig 12.3. We choose the Symmetry surface crack option and then select **Next**. The next panel lets us choose the surface where the symmetry crack is inserted, Fig 12.4. The top surface is initially colored blue; it is the only surface that can be selected as it is the only surface with symmetric boundary conditions. Select this surface using the Shift key and the left-mouse button (or use the check box); it will be red as shown in Fig 12.4. Then select **Next**.

The following wizard panels are mostly the same as the regular 3D crack panels. We will use the elliptical crack shape with a and b set to 1. The crack orientation panel, Fig 12.5, allows us to position the crack on the selected surface. We set it to create a corner crack at 10, 10, 10. Select **Next** to set the template mesh, and use the default settings, Fig 12.6; then select **Finish**.

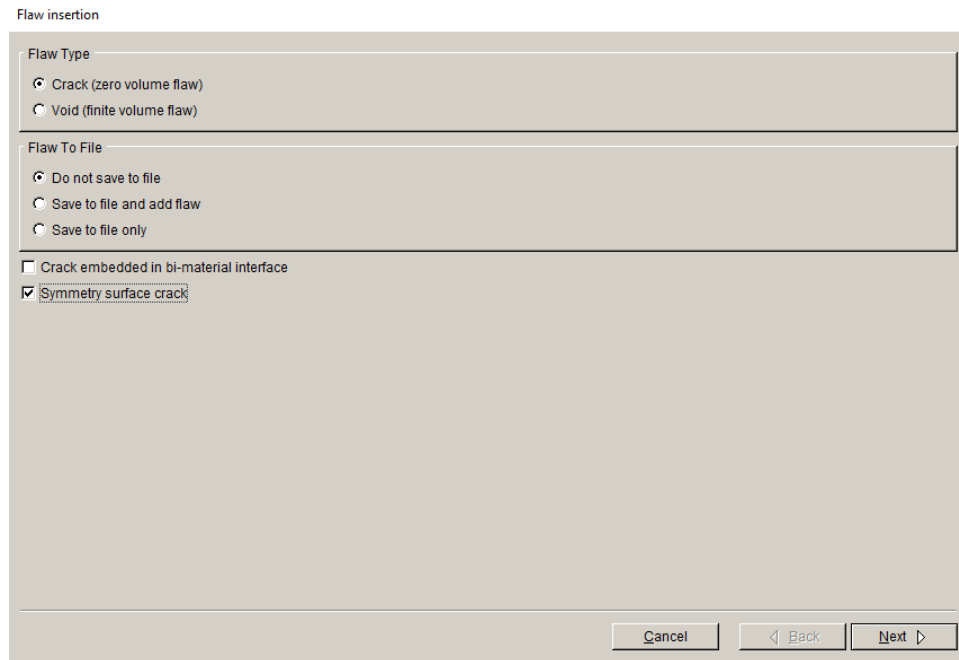


Figure 12.3 Flaw insertion dialog.

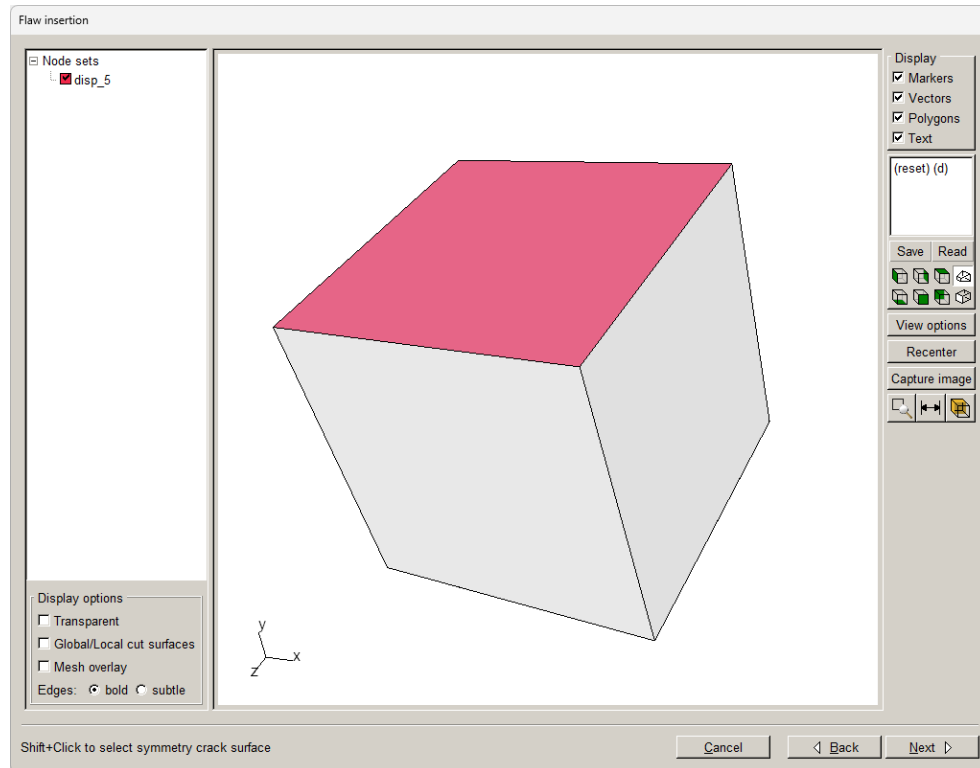


Figure 12.4 Select symmetry surface dialog.

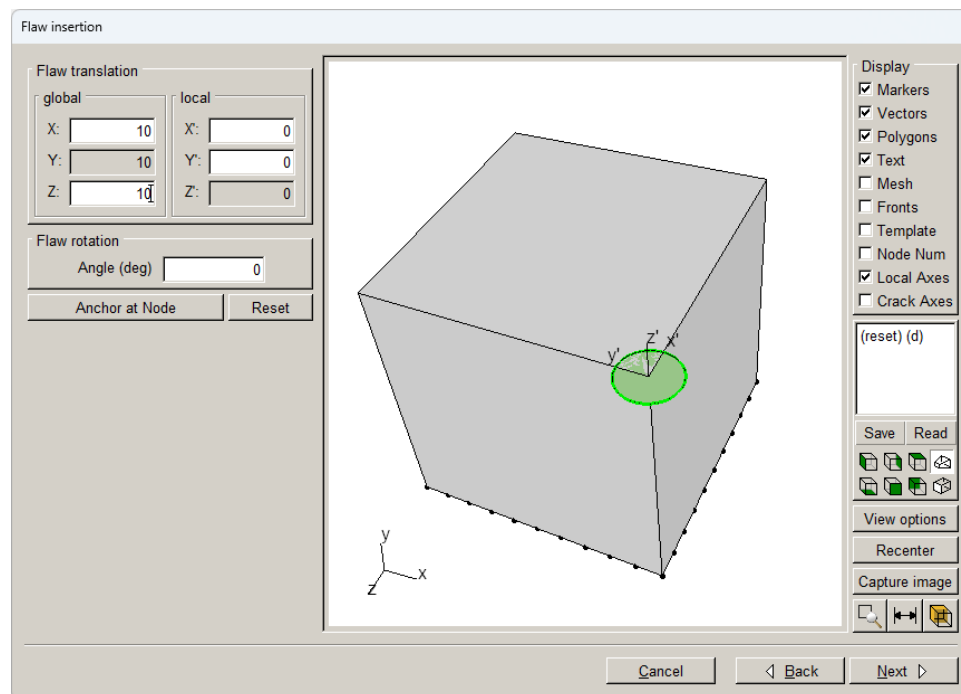


Figure 12.5 Symmetry crack orientation dialog.

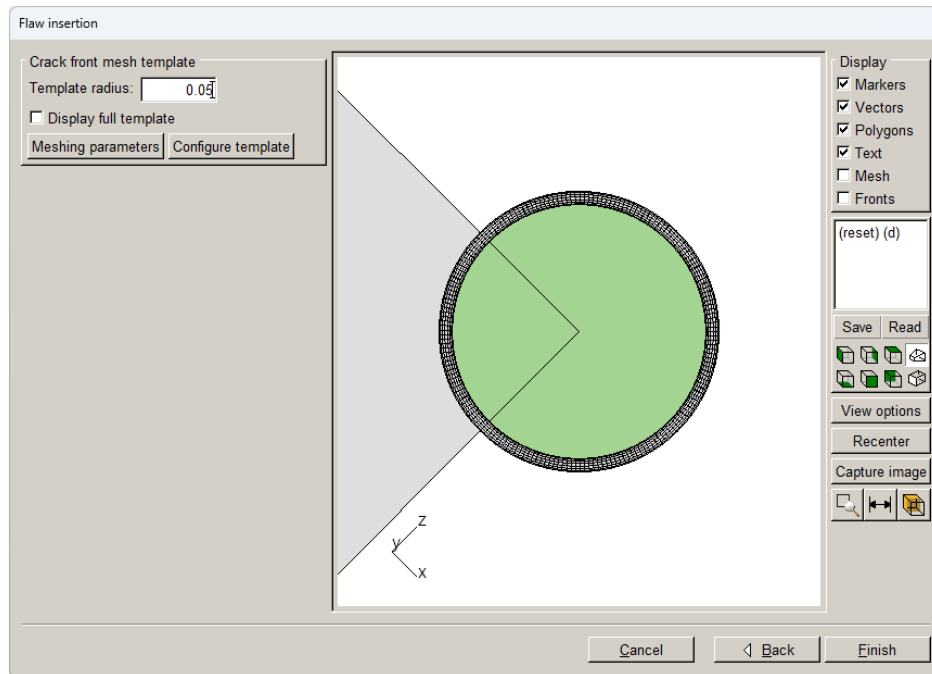


Figure 12.6 Symmetry crack mesh template dialog.

The resulting mesh is shown in Fig 12.7, which also shows a zoomed-in image of the symmetry crack surface mesh.

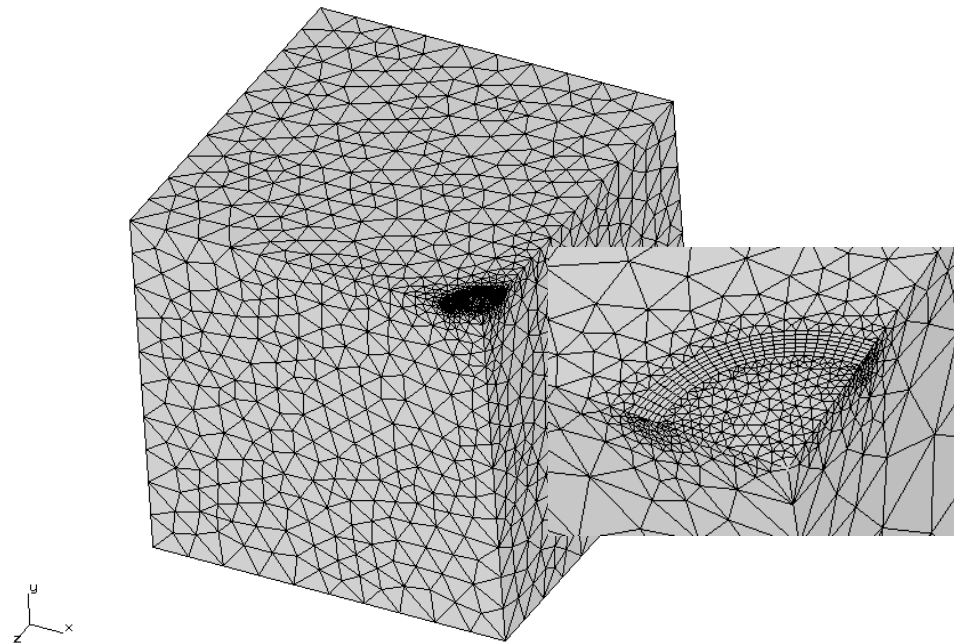


Figure 12.7 Symmetry crack surface mesh.

12.3 Symmetry Crack SIFs

Run a static crack analysis to compute SIFs. When the FE analysis has finished, use the M-integral method to compute the SIFs, Fig 12.8. K_{II} and K_{III} are zero. You can proceed to grow the crack if desired; the only option for kink angle is planar as the crack will always remain on the symmetry surface.

You can compare the SIFs with those from a regular 3D corner-crack in a “full” model, Fig 12.9; there should be negligible difference.

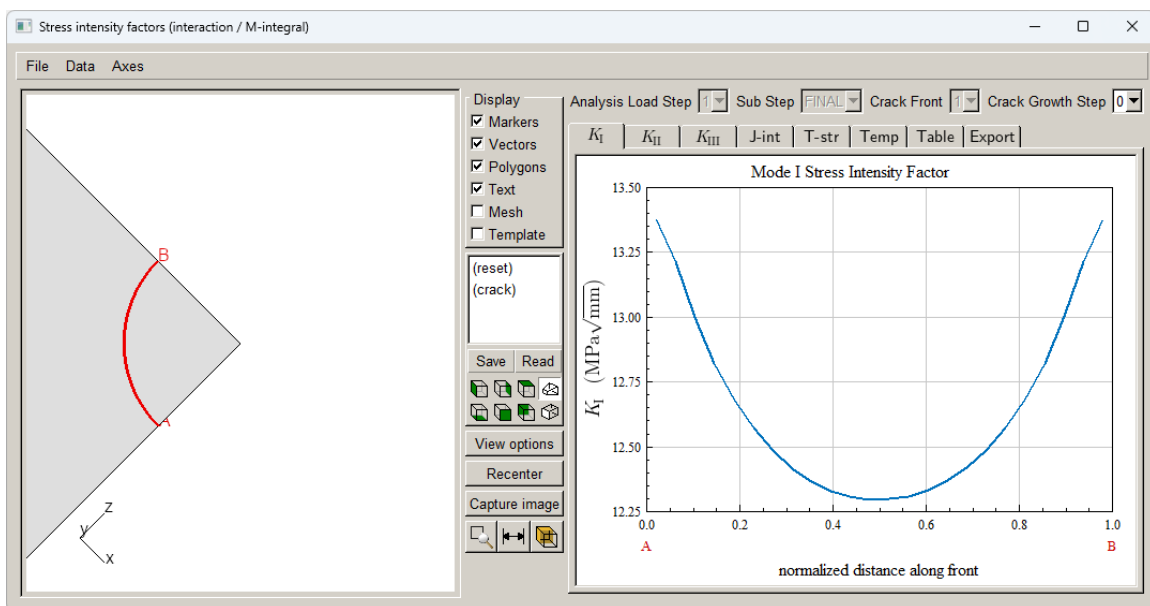


Figure 12.8 Symmetry crack Mode I SIFs.

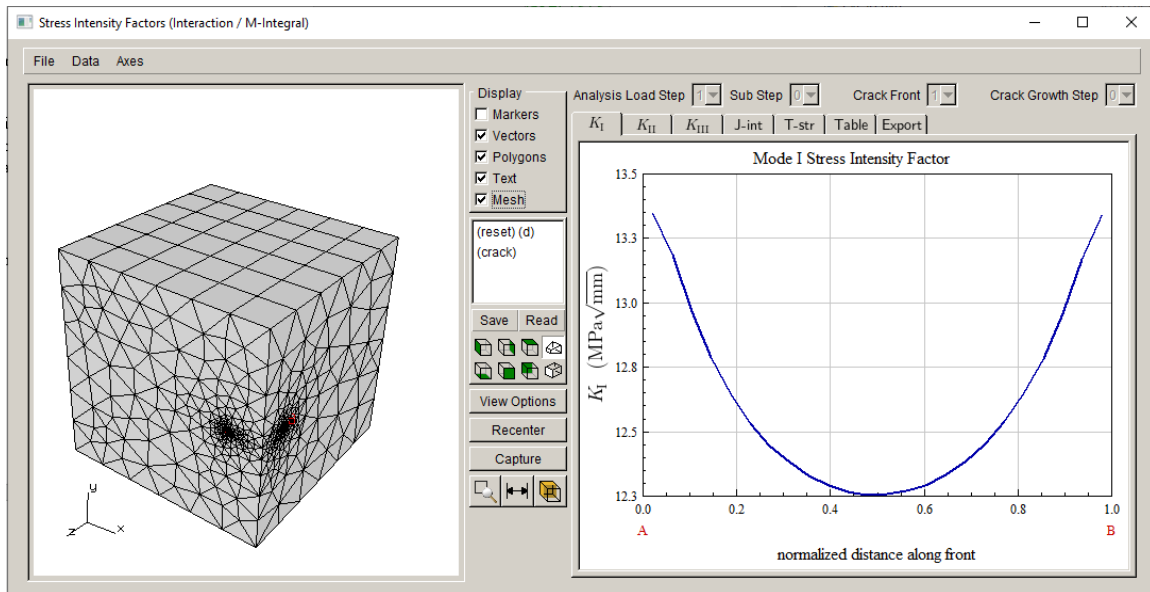


Figure 12.9 Full 3D crack Mode I SIFs.

12.4 Symmetry Crack Growth

Symmetry crack growth is limited to planar growth and is constrained to the original symmetry surface. Otherwise, growth is the same as for a full 3D crack. Twenty steps of growth are completed using an extension increment of 0.15, Fig 12.10, and a quasi-static growth model.

Note that Fronts in the Display box in Fig 12.10, is turned on so that we can see the original crack front as well as the new front points. The fronts for twenty steps of growth are displayed in Fig 12.11. SIF history and fatigue life computations are left as an exercise for the reader.

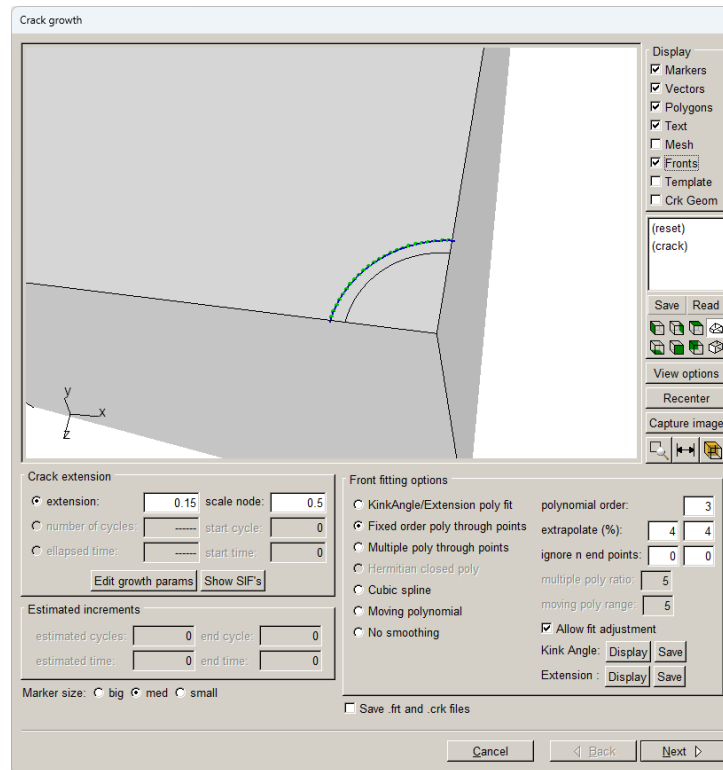


Figure 12.10 Crack growth increment for the symmetry crack.

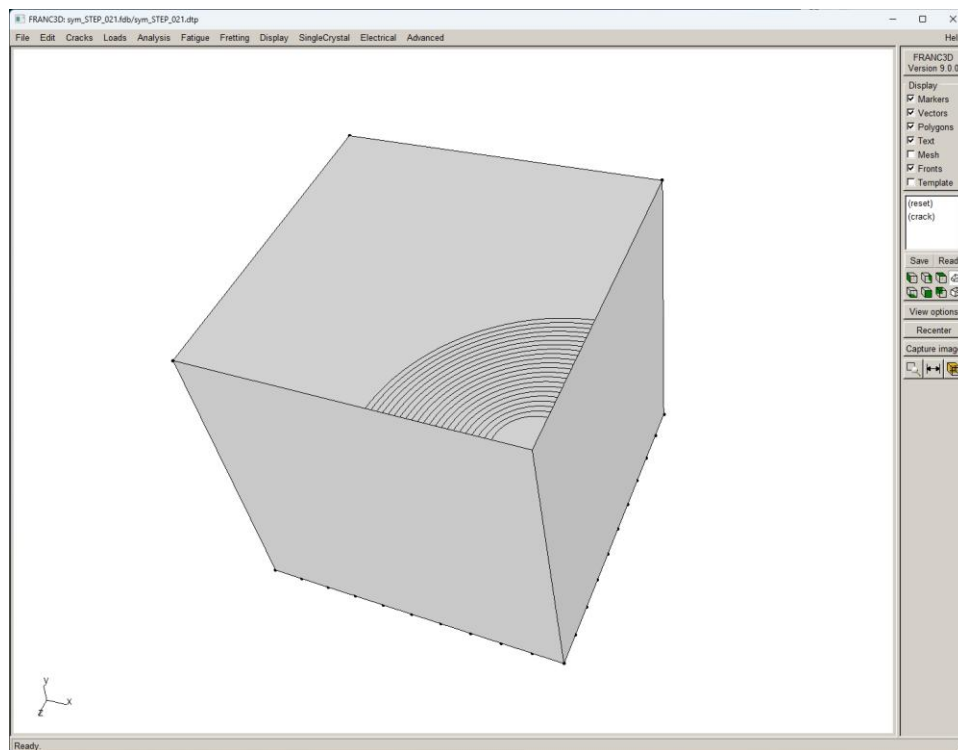


Figure 12.11 Crack growth for the symmetry crack.

13.0 Tutorial 13: Interface Crack

A bi-material crack simulation is demonstrated in this tutorial, using an ABAQUS cube model with the lower half as material #1 and the upper half as material #2, Fig 13.1. The cube is constrained on the bottom and has tension on the top (+y) surface. There are two basic options for crack insertion: 1) a crack embedded in the material interface, and 2) a crack crossing the material interface. The material interface is represented by a single surface, so the materials are “bonded” and share mesh facets on the interface.

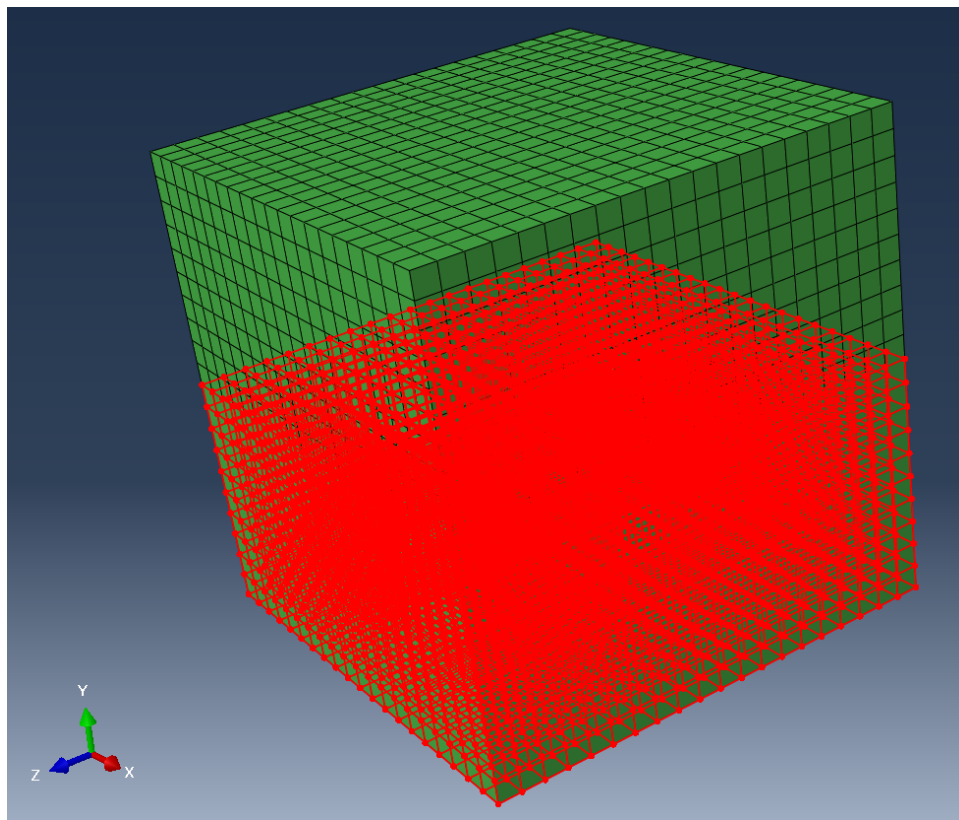


Figure 13.1 ABAQUS cube with material interface.

13.1 Import Uncracked Model

The complete model is imported for simplicity and the top and bottom surfaces are retained, Fig 13.2. Fig 13.2 shows the material interface as well as the retained mesh facets. The mesh facets on the interface are not retained.

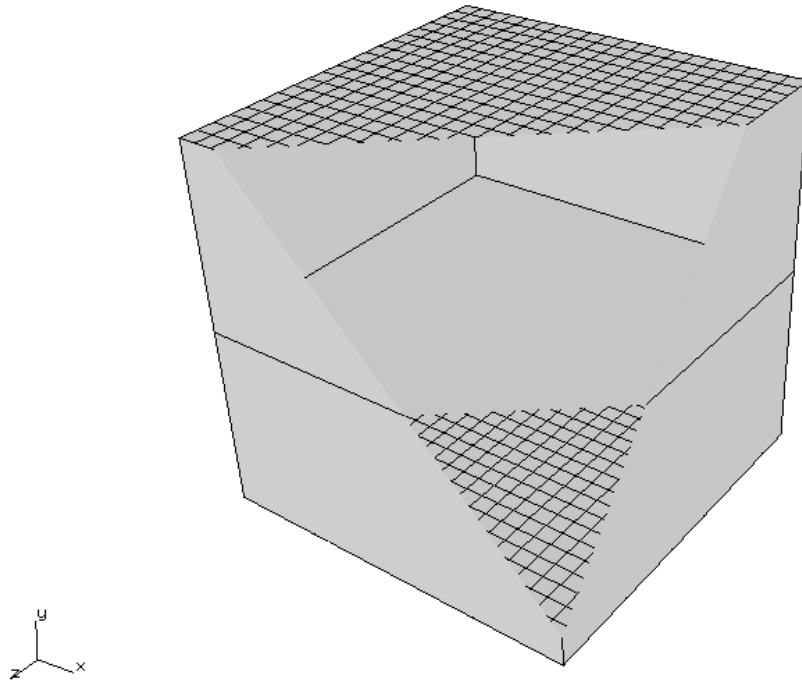


Figure 13.2 Cube with material interface imported into FRANC3D.

13.2 Embedded Interface Crack

When inserting a crack that is embedded in the material interface, the user must check the Crack embedded in bi-material interface box in the Flaw Insertion wizard, Fig 13.3. The rest of the flaw insertion process is the same as usual; in this tutorial, we insert an elliptical crack with dimensions of $a=b=1$, and located at coordinates (5,5,10) and rotated 90 degrees about the global x-axis, Fig 13.4. The template radius is 0.1.

The resulting surface mesh is shown in Fig 13.5; a front view and a cut-away view are shown. The static crack analysis can be performed to obtain the displacements, and the SIFs can be computed, Fig 13.6.

We do not currently support crack growth for this type of crack.

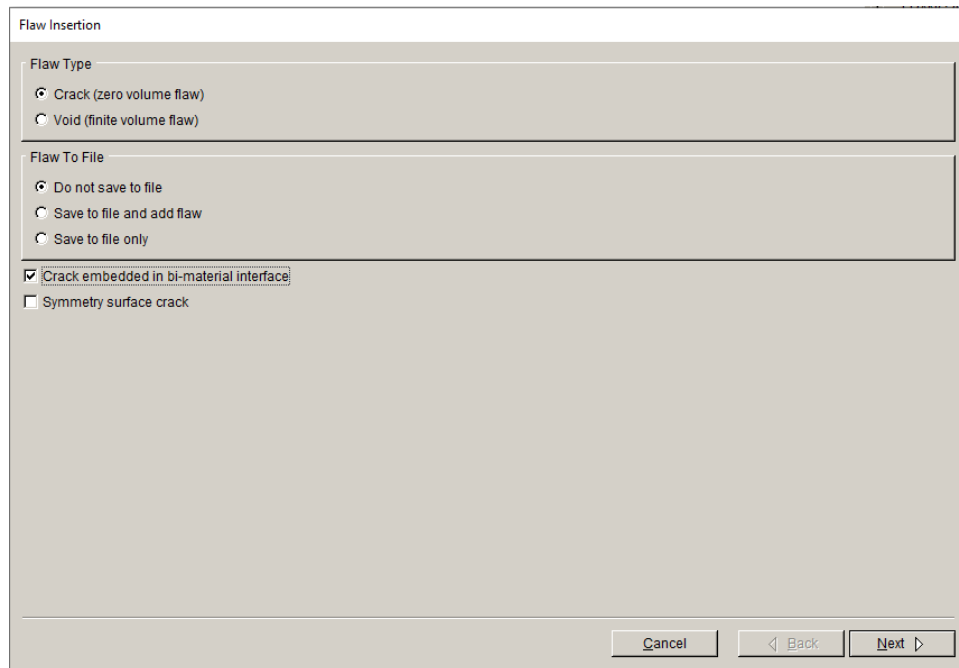


Figure 13.3 Flaw insertion first wizard panel with bi-material interface checked.

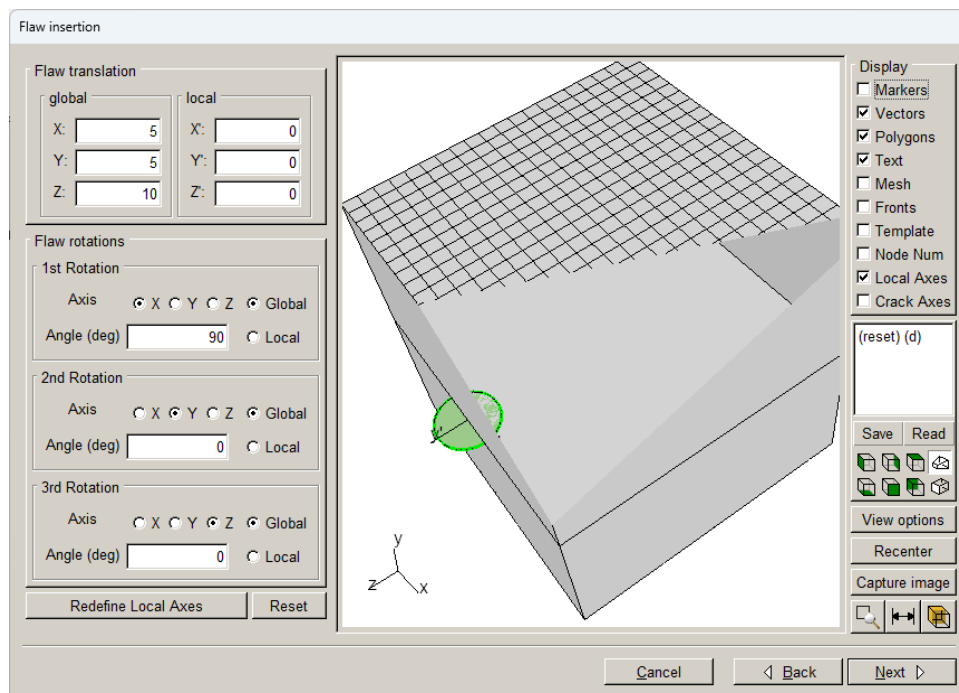


Figure 13.4 Elliptical flaw located in the material interface.

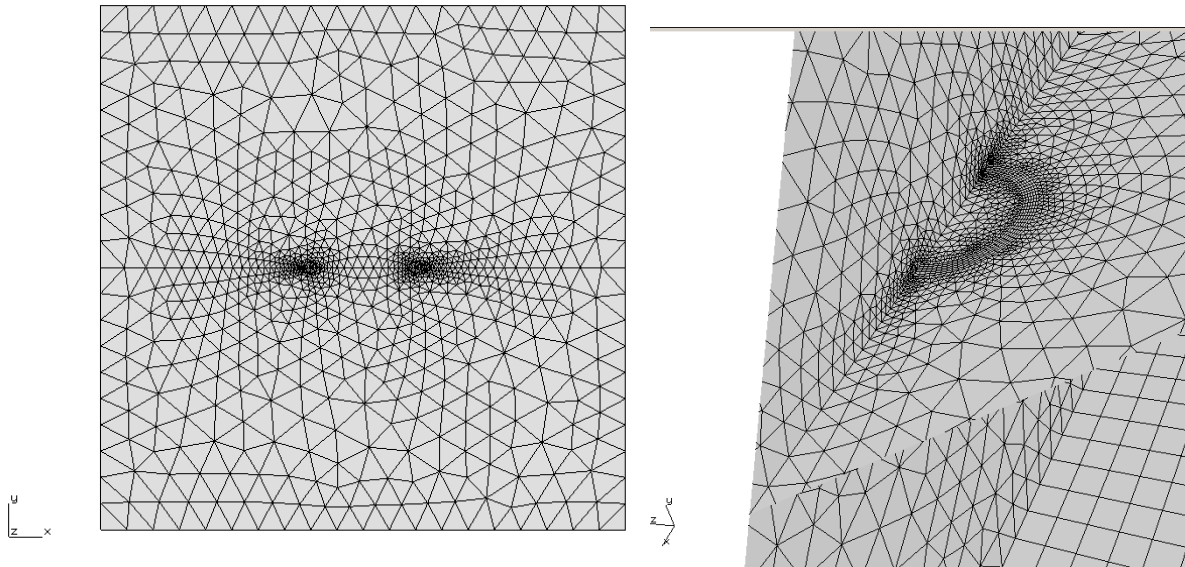


Figure 13.5 Half-penny surface crack embedded in a material interface.

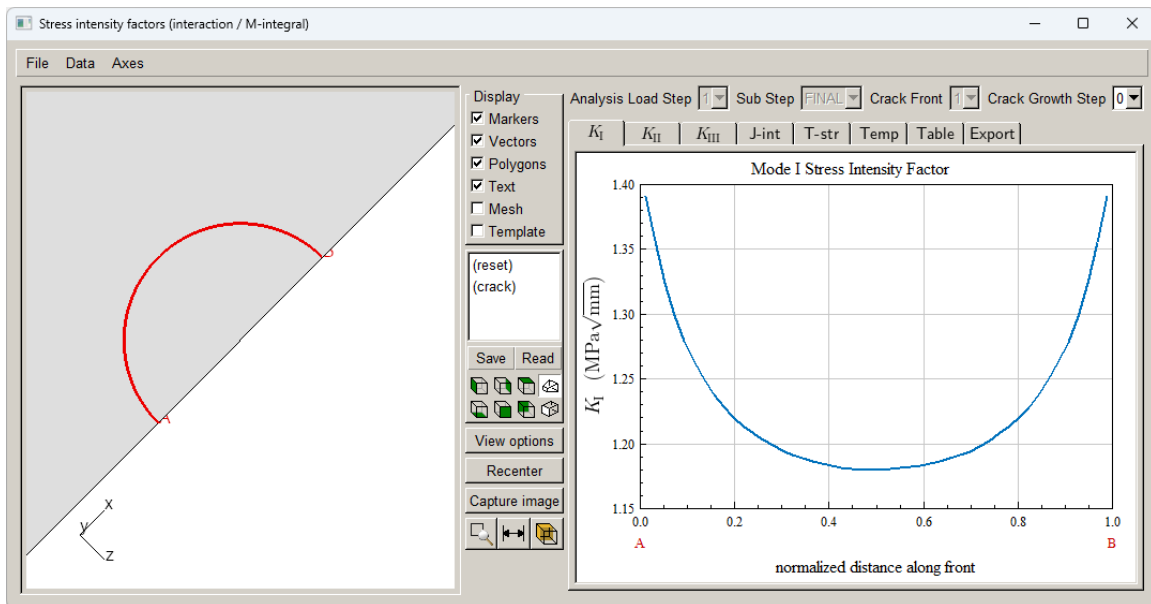


Figure 13.6 Mode I SIFs for a half-penny surface crack embedded in a material interface.

13.3 Crack Crossing an Interface

Inserting a crack that crosses a material interface does not require anything special. The flaw insertion process is the same as usual; in this tutorial, we insert an elliptical crack with

dimensions of $a=b=1$, and located at coordinates (5,5,10) and rotated 90 degrees about the global x-axis and 45 degrees about the global z-axis, Fig 13.7. The template radius is 0.1.

The resulting surface mesh is shown in Fig 13.8; a front view and a cut-away view are shown.

The static crack analysis can be performed to obtain the displacements, and the SIFs can be computed, Fig 13.9; the jump in SIFs at the material interface is clear.

We do not currently support crack growth for this type of crack. Although it might be possible to advance the crack and get a mesh, the interface behavior is not considered.

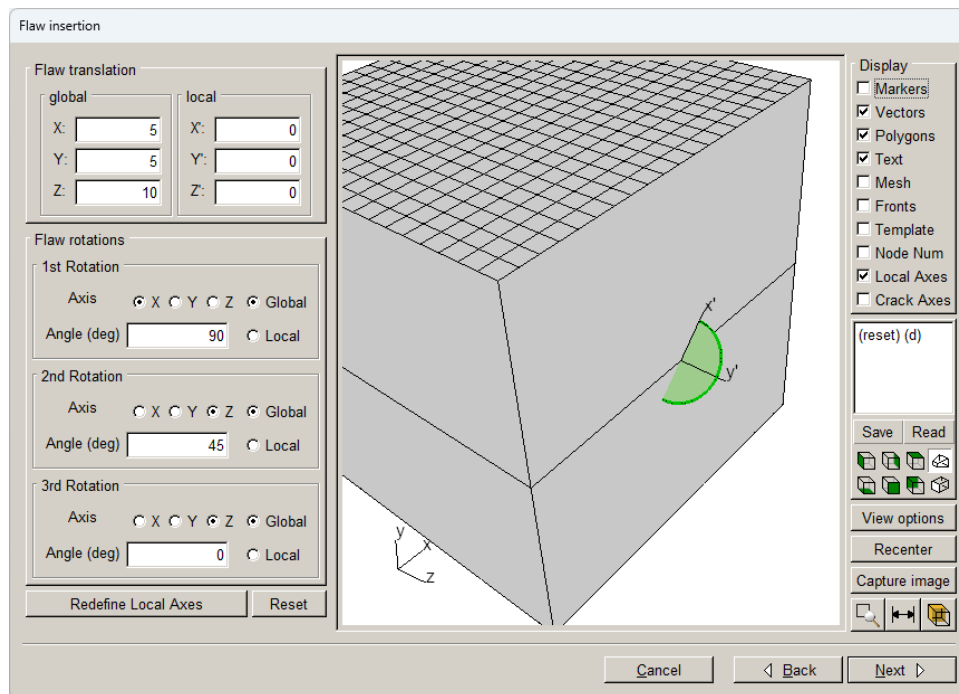


Figure 13.7 Elliptical surface crack crossing a material interface.

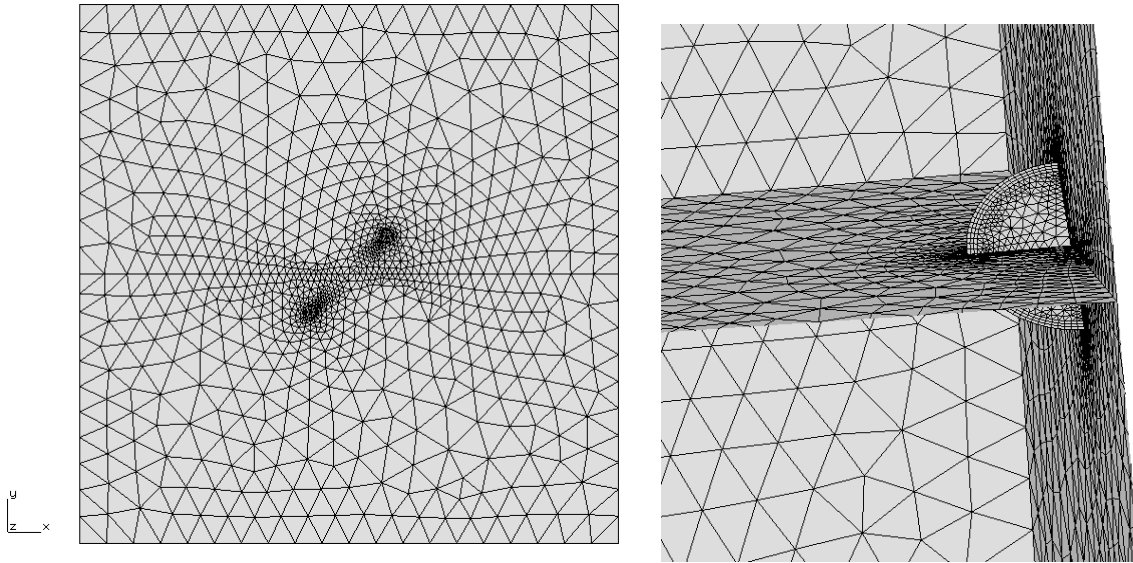


Figure 13.8 Surface mesh for a half-penny surface crack crossing a material interface.

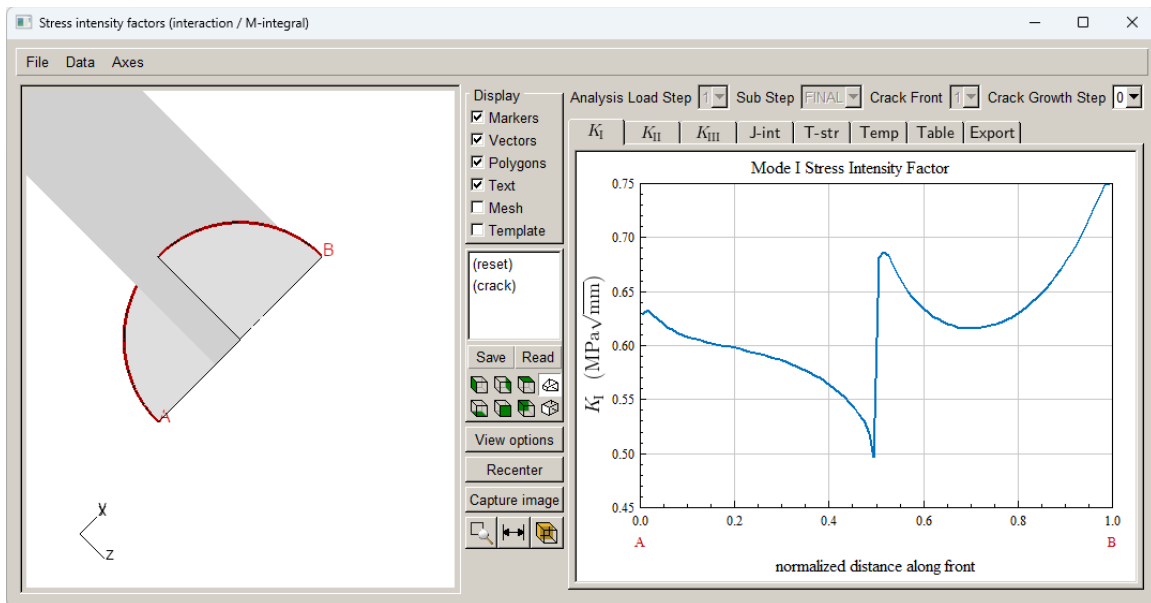


Figure 13.9 Mode I SIFs for a half-penny surface crack crossing a material interface.

14.0 Tutorial 14: LCF + HCF with Dynamic Pairing

Low cycle fatigue plus high cycle fatigue (LCF + HCF) with dynamic pairing is demonstrated in this tutorial, using an ANSYS model, Fig 14.1. The dimensions in mm are provided in Fig 14.1. The material properties are $E=160,000$ MPa, $\nu=0.34$ and $\text{density}=1.2\text{e-}3$. The model consists of a 40-degree sector with symmetric boundary conditions; five static load steps are defined with different values of rotational velocity. There are no temperatures; temperature variation might be included in the future. The sixth load step is a frequency/modal analysis.

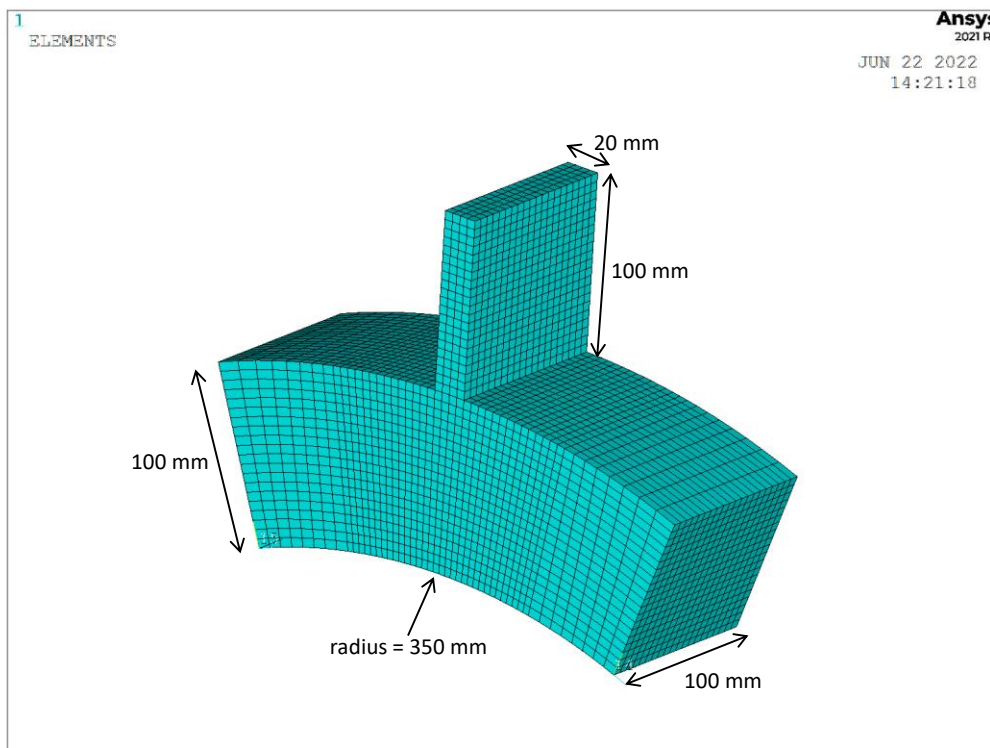


Figure 14.1 ANSYS model.

14.1 Import Uncracked Model

Start FRANC3D and select **File** → **Import** and select Import and divide... and click **Next**. Switch the Mesh File Type radio button in the Select Import Mesh File window to the appropriate FE type and select the file name for the model, Fig 14.2, and click **Next**.

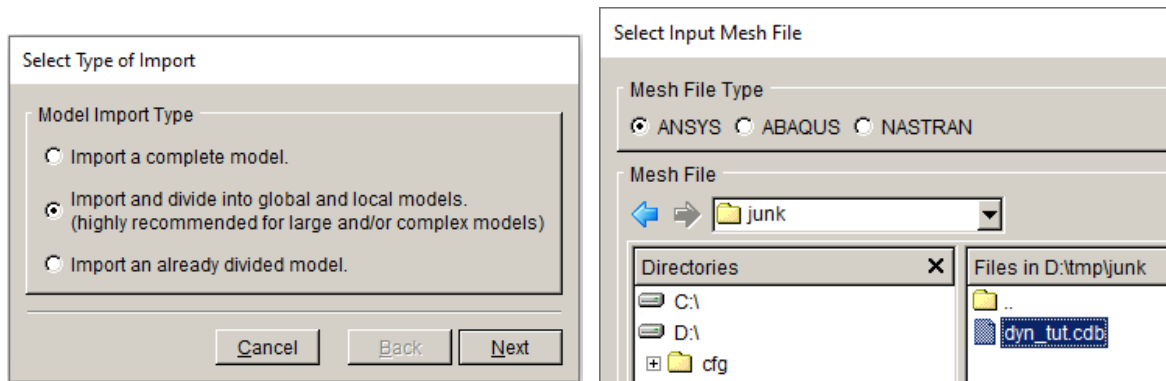


Figure 14.2 FRANC3D FE model import – ANSYS selected.

Create a local model using the Rubberband Box tool and the **Crop** button in the Define Local Submodel window, Fig 14.3. Select **Next** when ready and save the local and global model files using the default file names.

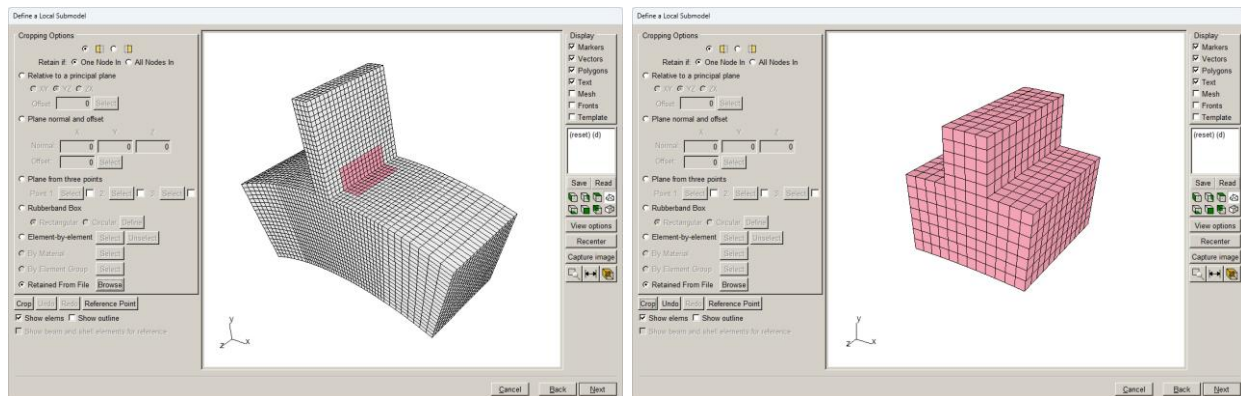


Figure 14.3 FRANC3D FE model import.

In the Select Retained BC Surfaces dialog, Fig 14.4, only the cut-surfaces will be retained, so just select **Next**. Set the FEM units so that they are consistent with the FE model; SI units of mm, MPa and C are selected. The model will be displayed in the FRANC3D main window, and we are ready to insert a crack into the local model.

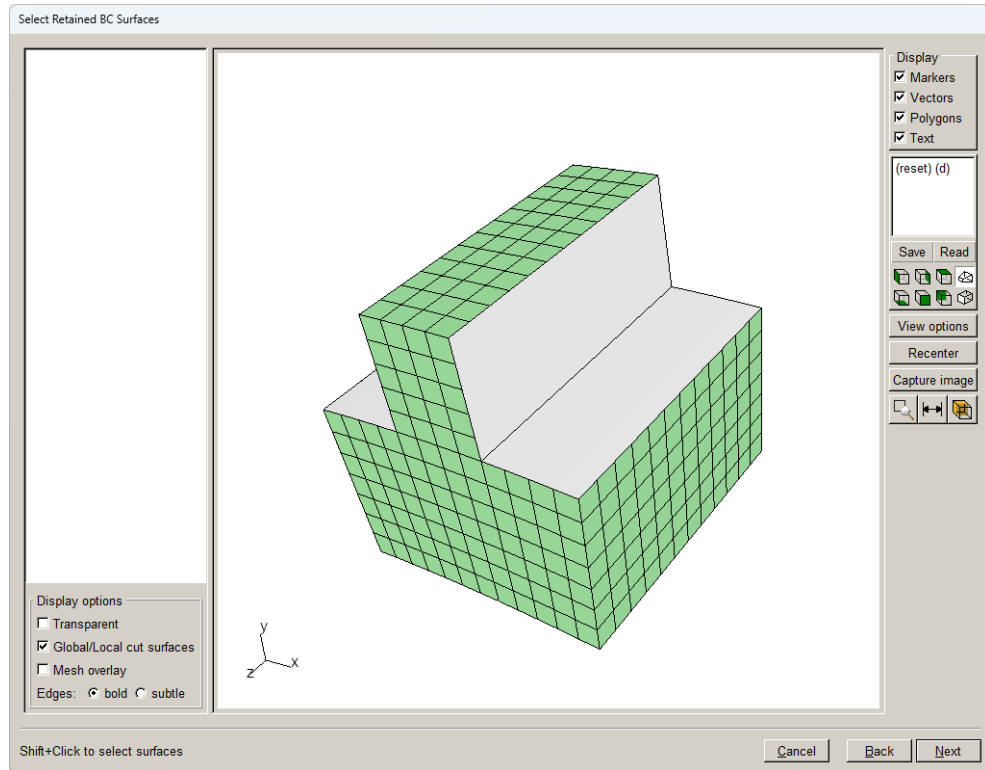


Figure 14.4 FRANC3D FE model import – retained surface selection.

14.2 Step 2: Insert Initial Crack

Insert a circular crack with a radius of 1.5 mm, Fig 14.5. The crack is located near the bottom of the ‘blade’ and rotated 90 degrees about the x-axis, Fig 14.6. The default crack front template radius is 0.15; the resulting remeshed crack model is shown in Fig 14.7.

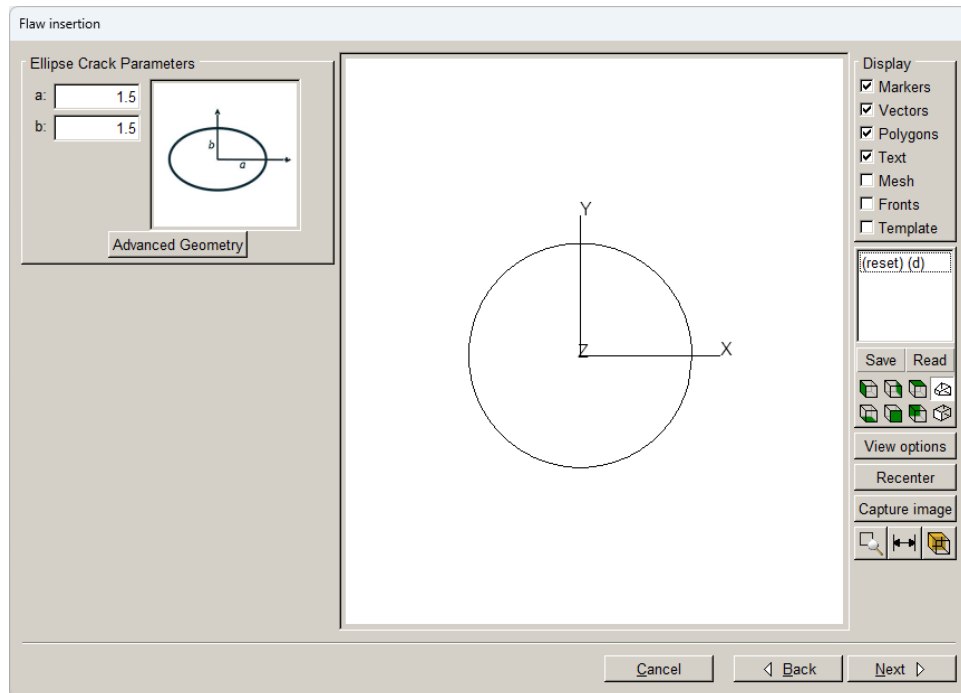


Figure 14.5 Elliptical crack dimension dialog.

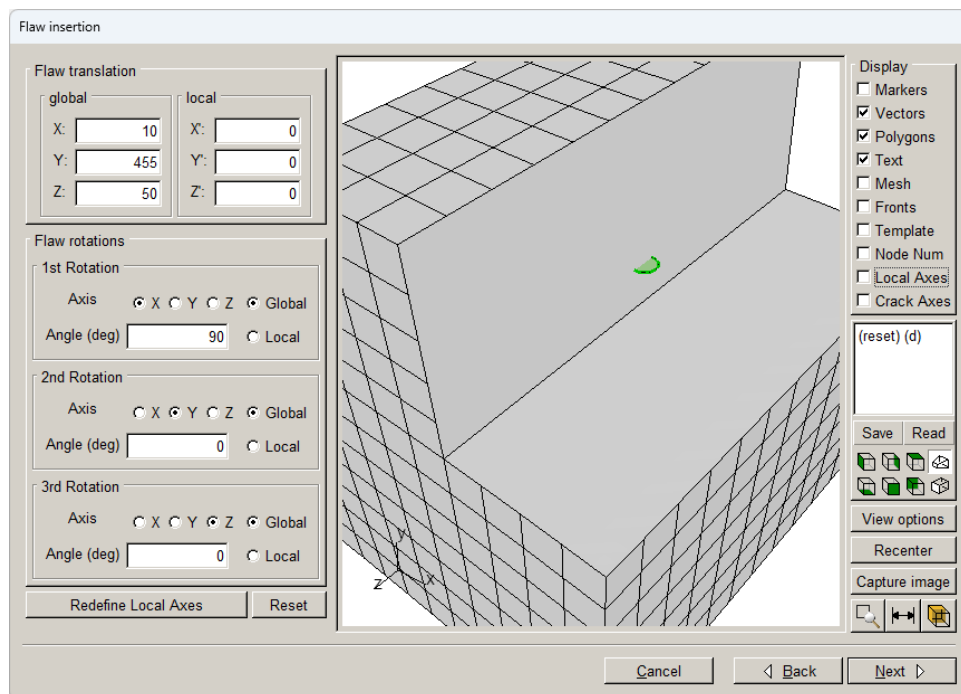


Figure 14.6 Crack location and orientation dialog.

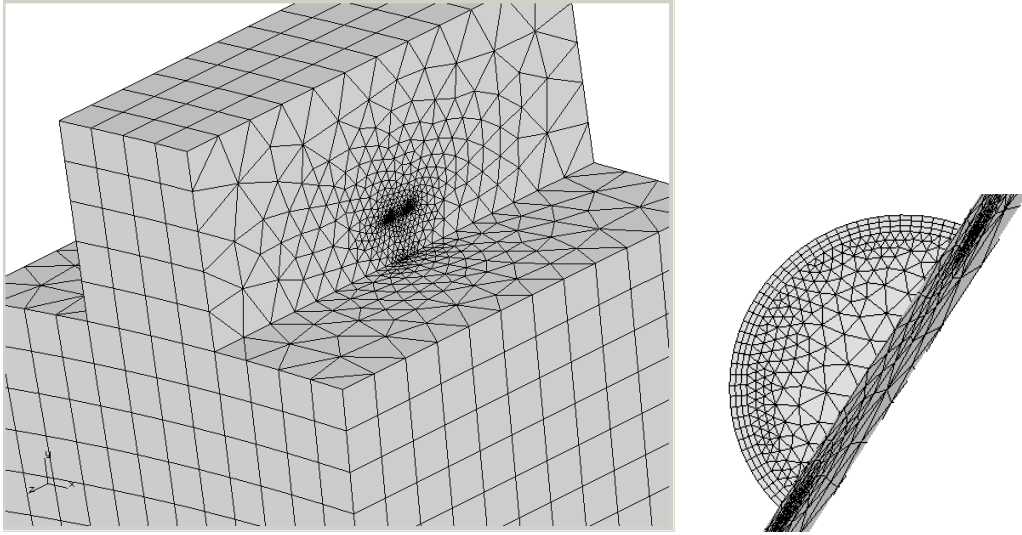


Figure 14.7 Final meshed crack model.

14.3 Step 3: Static Crack Analysis

Next we perform a static crack analysis. Choose **Analysis** → **Static Crack Analysis** from the FRANC3D menu. The ANSYS options are shown in Fig 14.8. The Connect to global model option is checked automatically. Select **Next**.

The next dialog box has options for connecting the local and global portions, Fig 14.9. We use node merging with the automatically selected local and global cut-surfaces. Click **Finish** to begin the ANSYS analysis.

Static analysis

ANSYS Options

ANSYS run time:

ANSYS Executable:

ANSYS license:

Python Executable:

Python script (will run before ANSYS):

Local model output:

Global model:

☒ Connect to global model filename:

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact ☐ Initial strain

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 14.8 Static analysis options for ANSYS.

Static analysis

ANSYS local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☒ retain ☐ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF ☐ all_template_nodes

Global component(s) to merge/constrain:

☐ Show all labels

☒ GLOBAL_CONNECT_SURF

Additional local/global connections:

ANSYS command:

☐ Write files but DO NOT run analysis

Figure 14.9 Static analysis ANSYS local + global model connection dialog.

Note that there are five static load steps plus a frequency/modal load step. The modal load step is set up in the original uncracked model, and the first five modes are requested. Of those first five modes, mode 2 produces bending of the blade, Fig 14.10. This is the only mode that we will consider when computing SIFs and growing the crack.

The displacements from the modal analysis typically must be scaled; that can be done when extracting the displacements from the ANSYS results or it can be done in FRANC3D when defining the load schedule. For this model, we will extract the unscaled displacements for the 2nd mode from ANSYS and then scale the SIFs in FRANC3D. For this 2nd mode, the displacement results for phase angle 0 and 90 degrees are stored in the .dtp file; only one of these are actually required.

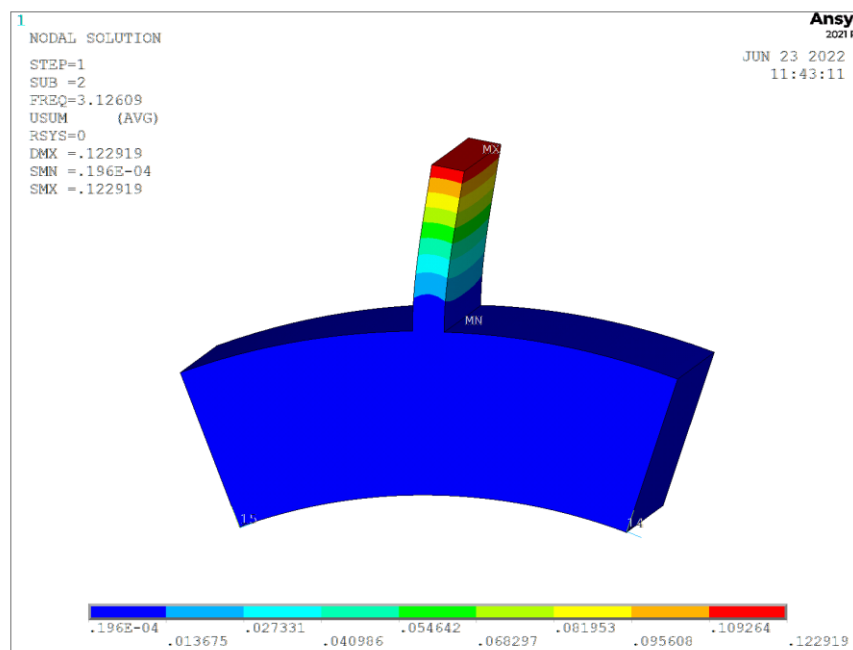


Figure 14.10 Mode 2 “bending” of the uncracked model.

14.4 Step 4: Compute SIFs

Once the ANSYS analysis has finished, we compute the SIFs; choose **Cracks** → **Compute SIFs** from the FRANC3D menu. The dialog, Fig 14.11, allows you to specify the method for computing SIFs. The SIFs based on the M-integral are shown in Fig 14.12a. Note that for load step 6, which is the modal step, there are two sub-steps representing 0- and 90-degree phase angles, Fig 14.12b.

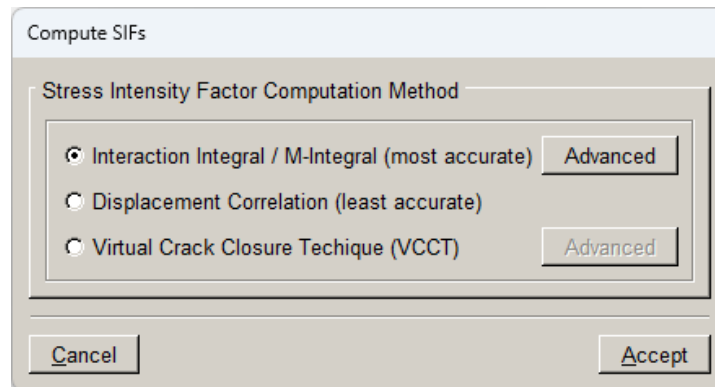


Figure 14.11 Compute SIFs dialog.

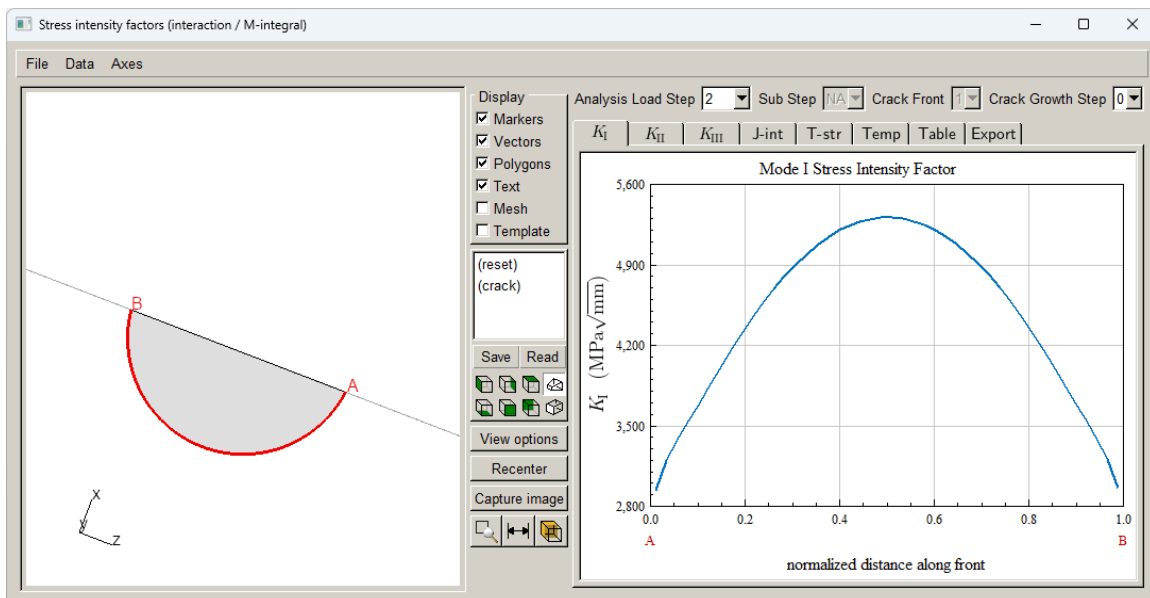


Figure 14.12a M-Integral SIFs for load step 2.

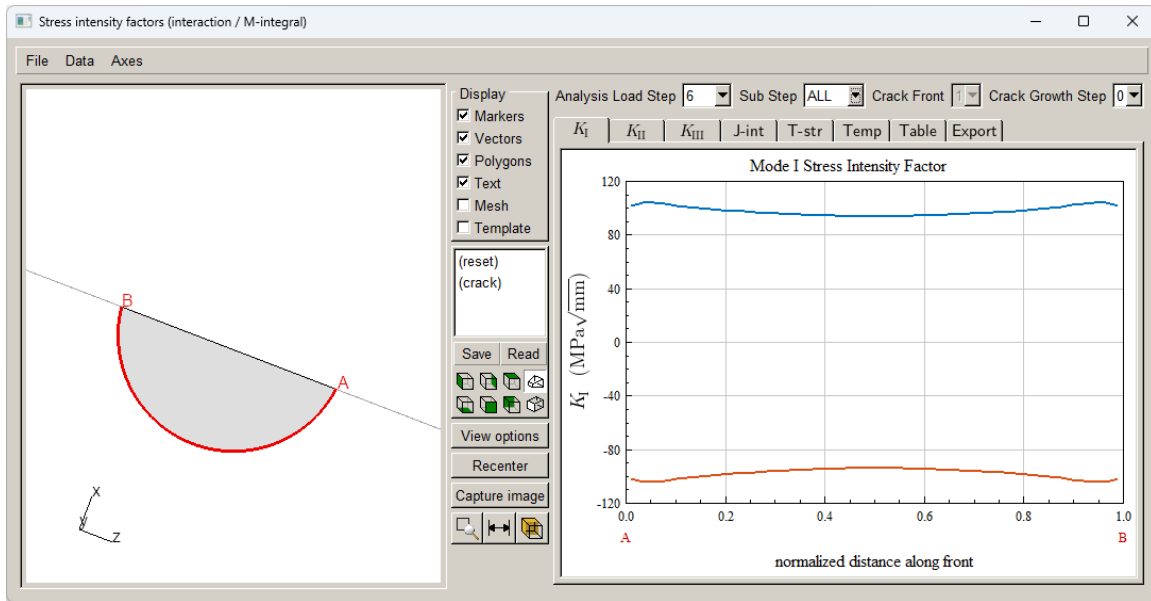


Figure 14.12b M-Integral SIFs for load step 6.

14.5 Step 5: Crack Growth

Next we set up the load schedule and propagate the crack. Choose **Cracks** → **Grow Crack** from the FRANC3D menu. Use Subcritical crack growth, Fig 14.13, and select **Next**. We leave all the defaults in the next panel, Fig 14.14, and select **Next**.

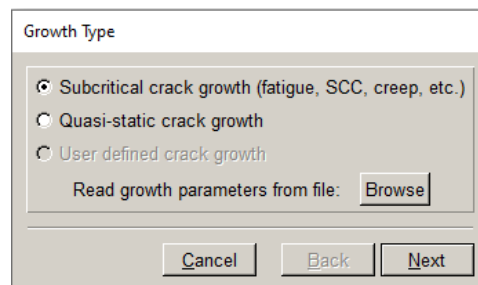


Figure 14.13 Crack growth first wizard panel.

Kink angle model

Kink angle model

☒ Max tensile stress (MTS) $\max(K_I^r(\theta))$

☐ Max shear stress (MSS) $\max\left(\sqrt{(\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2}\right)$

☐ Generalized stress $\max(\text{MTS}, \text{MSS})$

☐ Strain energy release rate $\max\left(K_I^r(\theta)^2 + (\eta_{II}K_{II}^r(\theta))^2 + (\eta_{III}K_{III}^r(\theta))^2\right)$

☐ Planar $\theta = 0$

☐ User defined model

☐ Kink angle limit Maximum kink angle (deg):

Non-proportional kink angle parameter ☒ ΔK ☐ $K_{\max}$

Mixed mode eta factors η_{II} : η_{III} :

Crack growth resistance

☐ Anisotropic toughness

Figure 14.14 Crack growth second wizard panel.

The next panel, Fig 14.15, allows you to set the load schedule and crack growth rate model. First, make sure the FE model units are correct. Then click on **New Schedule** to define the load schedule. A schematic of the load steps and their relative magnitudes is shown in Fig 14.16. The HCF (modal) load step is applied with load step #5 as a static load. Load step #1 produces zero SIFs.

The Load Schedule dialog is filled by selecting the Schedule and clicking the **Add** button, Fig 14.17 – left. Choose the Dynamic Pairing option for the Event type and then click **Accept**. Then select Dynamic Pairing under the Schedule and click the **Add** button, Fig 14.17 – right. We will add Low Cycle Reversals and High Cycle Const(ant) Amp(litude) events.

Subcritical Growth Parameters

Units used in FE model
stress: MPa length: mm temp: C time: sec Change

Crack Growth Load Schedule
New Schedule Read From File Wizard View/Edit Save To File

Crack Growth Rate Model
New Model Read From File View/Edit da/dN View/Edit da/dt Save To File

Mixed-mode equivalent K
☒ $K^{equiv} = K_I$ ☐ $K^{equiv} = \sqrt{K_I^2 + (\gamma_{II} K_{II})^2 + (\gamma_{III} K_{III})^2}$
☐ $K^{equiv} = K_{RSS}$ γ_{II} : γ_{III} :
sign: ☒ from K_I ☐ from K_{II} ☐ from K_{III} ☐ always positive ☐ always negative

Effective Delta K
☒ $\Delta K_{eff} = K_{max} - K_{min}$ ☐ $\Delta K_{eff} = K_{max} - \max(K_{min}, 0)$
☐ Use load interaction model Set/Edit Parameters

Integration Options
☒ Accelerated counting ☐ Constant K for time integration

Dynamic Pairing Metric
☒ ΔK_{eff}^{equiv} ☐ da/dN ☐ $\Delta CTOB$

Cancel Back Finish

Figure 14.15 Crack growth third wizard panel.

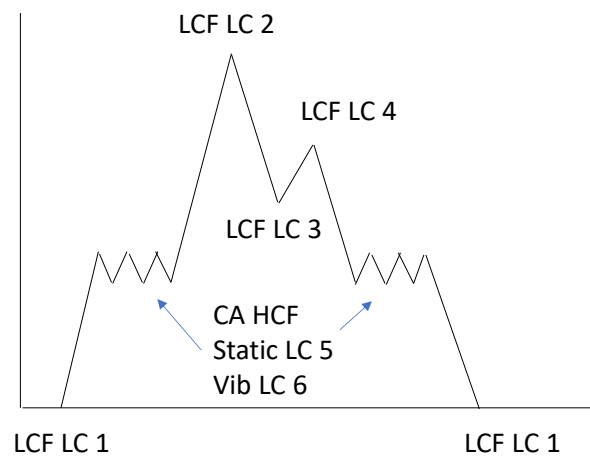


Figure 14.16 Load step schedule schematic.

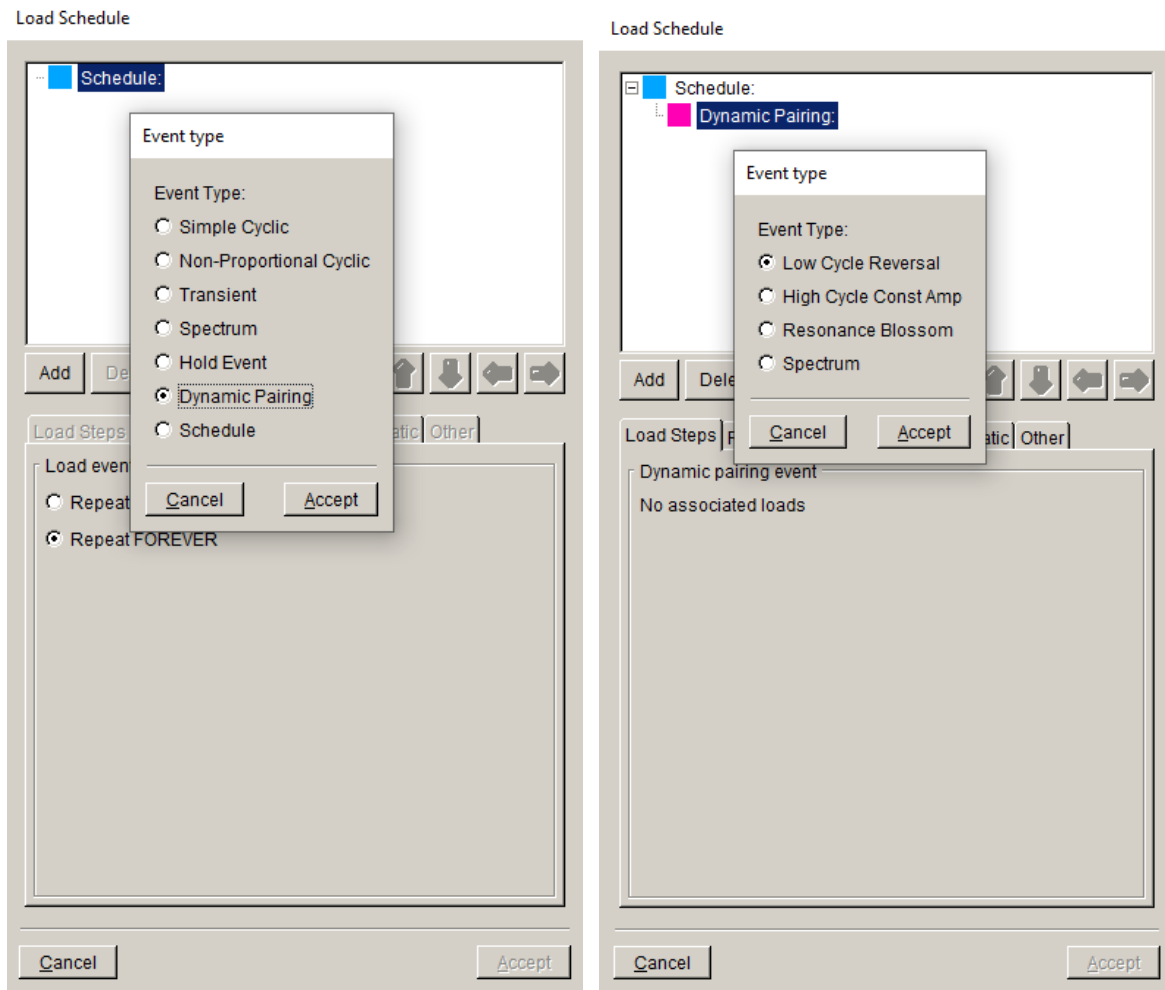


Figure 14.17 Load step schedule dialog.

The finished schedule is shown in Fig 14.18 – left. Note that the top-level Schedule is set to Repeat FOREVER. The HCF event consists of a static load step (#5) and a vibratory load step (#6), Fig 14.18 – right. The vibratory load step is given a load multiplier of 2; the modal displacement scaling (in this case, the SIF scaling) is chosen to give comparable amounts of HCF and LCF crack growth. The vibratory step is set to the MAX substep of loadstep #6; there are displacements for 0 and 90-degree phase angles for mode 2.

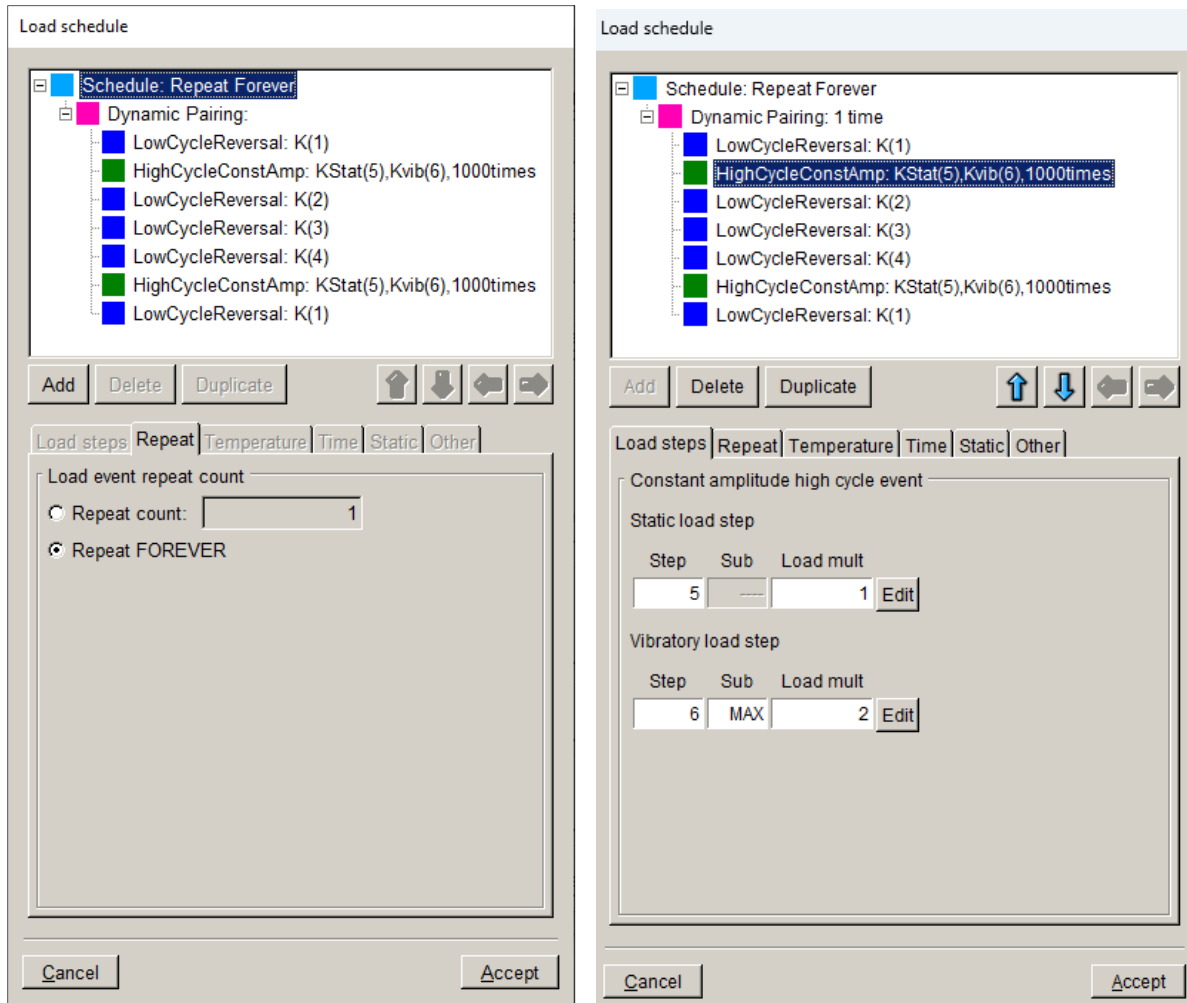


Figure 14.18 Final load step schedule.

A crack growth rate must also be defined. We use a Paris growth model with no R and no temperature dependence. The Paris model parameters are shown in Fig 14.19. This is defined by clicking on **New Model** in Fig 14.15 and choosing a cyclic loading, Paris model that is temperature independent.

Growth rate model

Paris growth model:
 $da/dN = C\Delta K^n$

Model label (optional):

Description (optional):

Units: stress: MPa length: mm temp: C time: sec

C	n	ΔK_{thresh}	K_{crit}
1e-12	2.7	0.1	12000

Figure 14.19 Paris crack growth rate model.

Once the load schedule and growth rate are defined, click **Finish** in Fig 14.15 to see the resulting crack growth. The user-defined extension is set to 0.25, Fig 14.20. The template radius is set to 0.15. Select **Next** when ready to start the process of crack growth, insertion and remeshing.

We continue this process for five steps of crack growth and then compute the fatigue cycles (see section 14.6). It is important to make sure that the modal load step produces consistent results from step to step; check the SIF history for this load step.

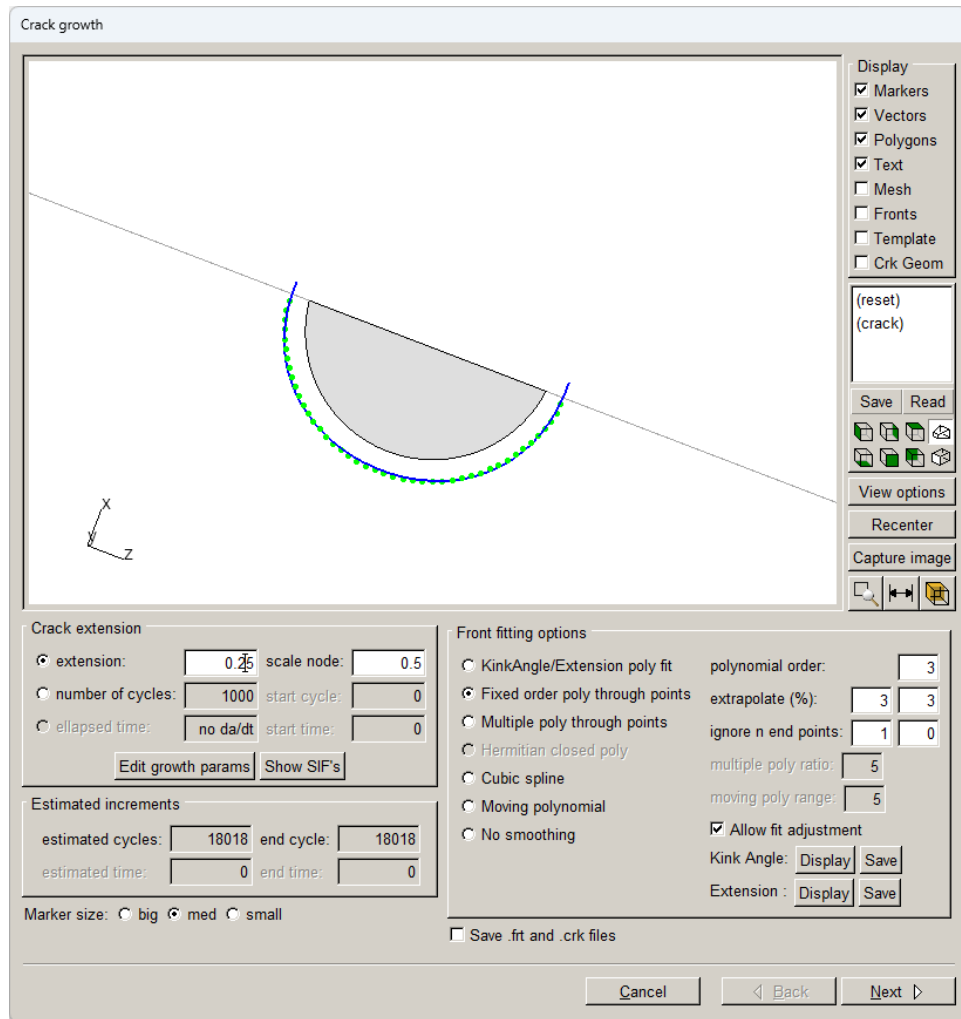


Figure 14.20 Predicted crack growth.

14.6 Step 6: Fatigue Cycles

We compute fatigue cycles based on the current crack growth. From the FRANC3D menu, select **Fatigue** → **Fatigue Life Predictions**. If the SIFs have not been computed yet, the Compute SIFs dialog will be displayed; use the M-integral and select **Finish**. The Fatigue Life dialog, Fig 14.21, is displayed. In this dialog, the crack fronts are displayed on the left and the right side should show computed cycles.

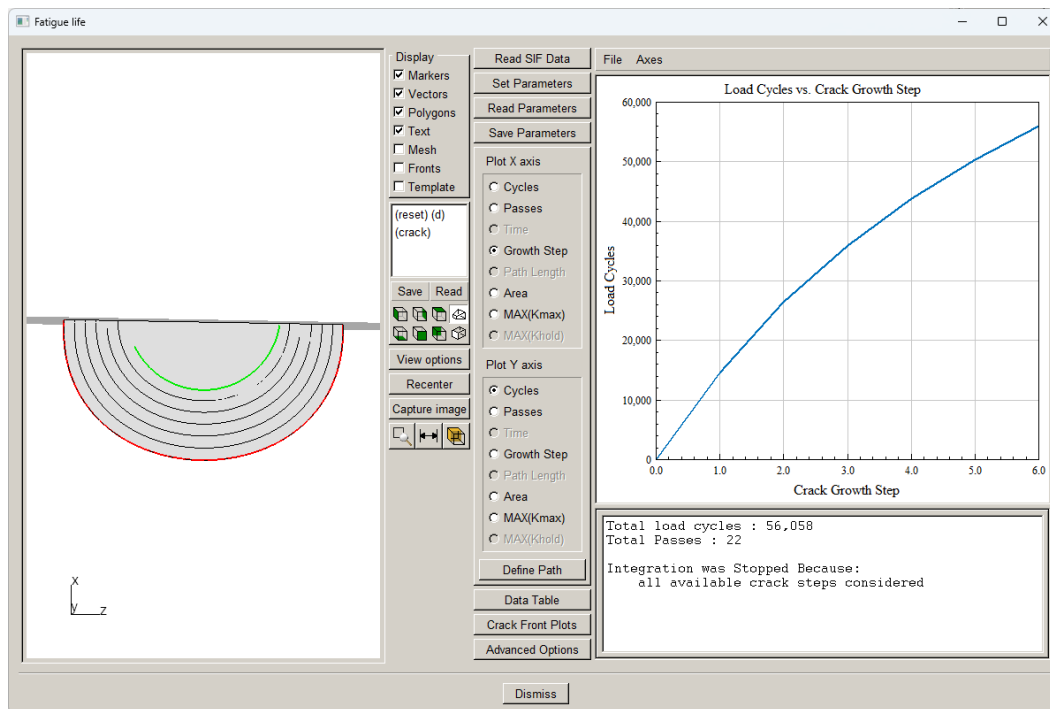


Figure 14.21 Fatigue Life dialog showing computed cycles versus crack step.

14.7 Summary

The K_{eq} load sequence is shown in Fig 14.22 for the mid-point of the initial crack front. The dynamic pairing produces the cycle pairs shown in Fig 14.23.

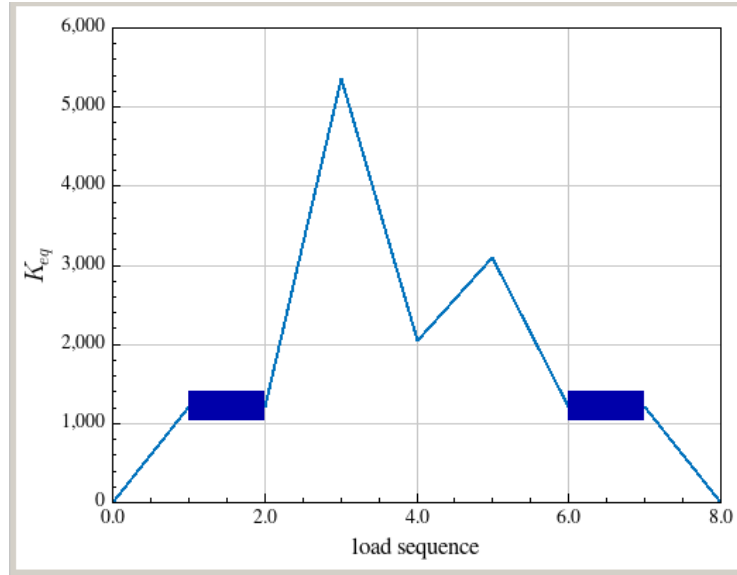


Figure 14.22 K_{eq} versus load sequence.

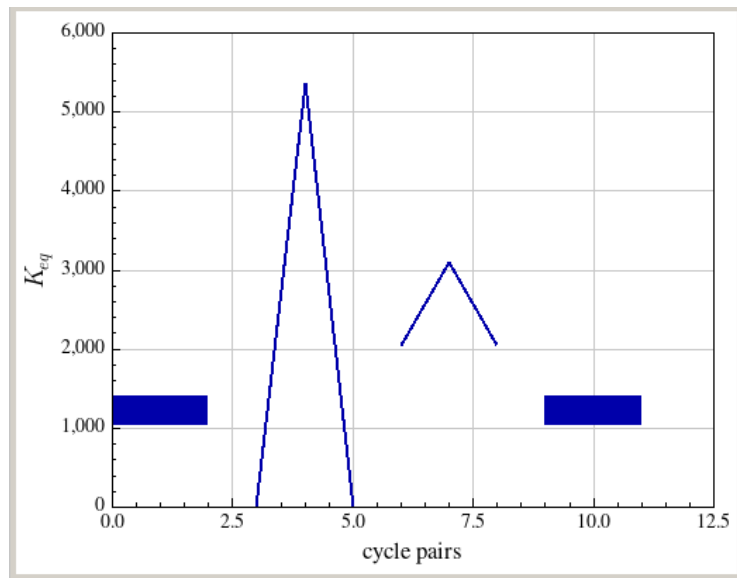


Figure 14.23 K_{eq} versus 'dynamic' cycle pairs.

The resulting cycle pairs #1 and #4 are HCF events. The static SIF ($1213 \text{ MPa} \sqrt{\text{mm}}$) comes from load step #5 and the vibratory SIF ($\pm 94 \text{ MPa} \sqrt{\text{mm}}$) comes from load step #6; the pair is repeated 1000 times.

$$\begin{aligned}
K_{\max} &= 1213 + 2*(94) = 1401 \\
K_{\min} &= 1213 - 2*(94) = 1025 \\
\Delta K &= 376 \\
R &= 0.73 \\
da/dN &= 8.974e-6 \\
da &= 2 * 1000 * 8.974e-6 = 1.795e-2
\end{aligned}$$

da is the growth increment and dN is the cycle increment. The resulting cycle pairs #2 and #3 are LCF events. The computed maximum SIFs are 5352 and 3106 MPa $\sqrt{\text{mm}}$, respectively.

$$\begin{aligned}
K_{\max} &= 5352 \\
K_{\min} &= 0 \\
\Delta K &= 5352 \\
R &= 0 \\
da &= da/dN = 1.167e-2
\end{aligned}$$

$$\begin{aligned}
K_{\max} &= 3106 \\
K_{\min} &= 2050 \\
\Delta K &= 1056 \\
R &= .66 \\
da &= da/dN = 1.459e-4
\end{aligned}$$

The total expected growth for each pass through the load schedule is

$$da = 1.795e-2 + 1.167e-2 + 1.459e-4 = 2.977e-2 \text{ mm}$$

where

1.182e-2 mm would be from LCF growth and

1.795e-2 mm would be from HCF growth

The user-supplied increment of growth (da) is 0.25 mm; this is applied at the point on the crack front where the median K_{eq} occurs. The rest of the crack front points will have extension that is scaled to make the cycle counts equal at all points on the front.

There are 2002 cycles in one pass through the schedule. Approximately 8 passes through the schedule are required to achieve the user-specified increment of growth. So approximately 16,016 total cycles are required; this is the approximate value that is plotted in Fig 14.21 at crack step #1.

15.0 Tutorial 15: Building SIF Histories

There are times when SIF history data from FRANC3D might need to be extracted or combined. This tutorial describes some instances of this along with a Python script that can be used to process and combine the *.fcg* files.

15.1 Resuming Growth with a Larger Submodel

We start with the simple cube model from Tutorial #1, where a LOCAL and a GLOBAL portion were defined and a crack propagated nearly to the boundary of the LOCAL portion, Fig 15.1. If we need crack growth to extend past the cut-boundary of the local submodel, a larger submodel can be selected and crack growth can be continued without having to restart the simulation from the beginning. In this section, we describe the process of extracting the current crack geometry and inserting this crack into a larger portion of the model to resume crack growth.

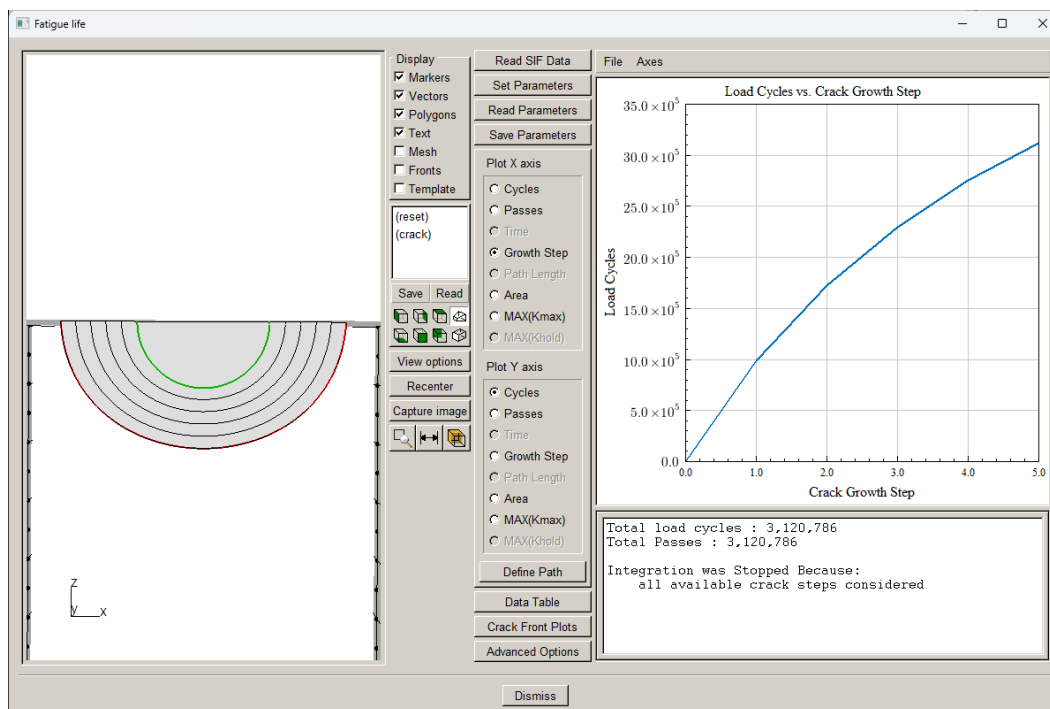


Figure 15.1 Crack propagated nearly to the boundary of the LOCAL model.

15.1.1 Extract and Save Crack Geometry

The crack geometry information for each step of crack growth is saved in the FRANC3D restart (*.fdb*) file. The data appears in this block:

```
FLAWSURF
(
  VERSION: 5
  NUM_SURFS: 763
  SURF: 0
    263 264 356
      6.83559456365193    5.00380971299962    9.72712346638146
    ...
    ...
      4.73891556707121    5.00420103803897    8.3468309877949
  )
```

This data can be copied from the *.fdb* file and saved to a *.crk* file, using any text editor. The *.crk* file can also be saved during a manual growth step using the Save .frt and .crk files option.

At step 5, the crack surface mesh will look like the left-side image in Fig 15.2. The crack geometry looks like the right-side image in Fig 15.2.

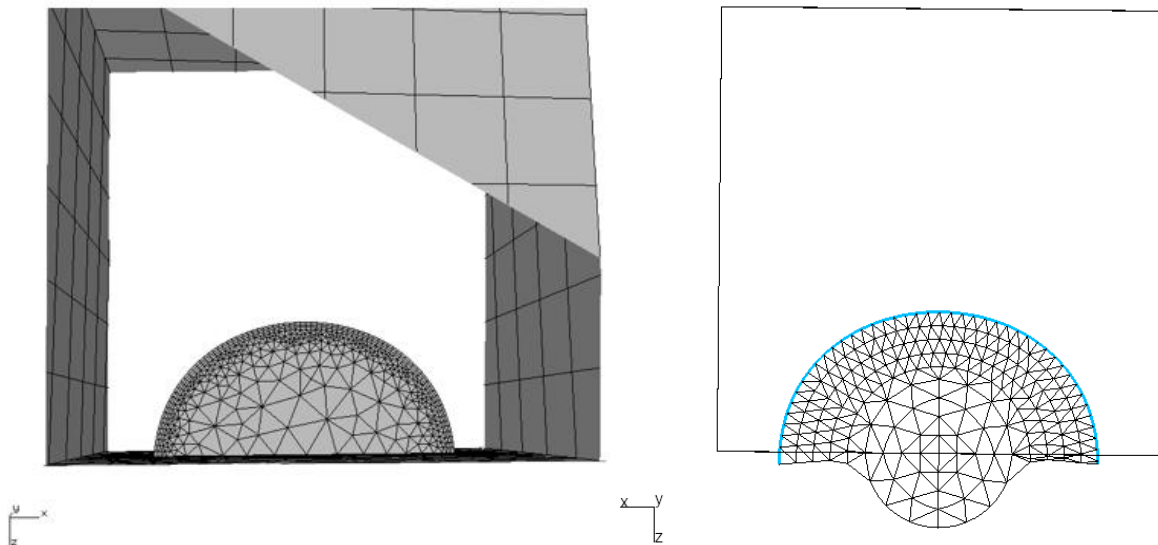


Figure 15.2 Crack step 5 in local submodel; surface mesh and geometry.

15.1.1.1 ANSYS

Open the *Ansys_Cube_subdivide_STEP_005.fdb* file (from Tutorial #1) in a text editor, copy the FLAWSURF data, and save it to a *.crk* file, called *Ansys_Cube_step_5.crk*.

15.1.1.2 ABAQUS

Open the *Abaqus_Cube_subdivide_STEP_005.fdb* file (from Tutorial #1) in a text editor, copy the FLAWSURF data, and save it to a *.crk* file, called *Abaqus_Cube_step_5.crk*.

15.1.1.3 NASTRAN

Open the *Nastran_Cube_subdivide_STEP_005.fdb* file (from Tutorial #1) in a text editor, copy the FLAWSURF data, and save it to a *.crk* file, called *Nastran_Cube_step_5.crk*.

15.1.1.4 SIERRA

Open the *Sierra_Cube_subdivide_STEP_005.fdb* file (from Tutorial #1) in a text editor, copy the FLAWSURF data, and save it to a *.crk* file, called *Sierra_Cube_step_5.crk*.

15.1.2 Restart from Saved Crack Geometry

Start FRANC3D, and from the menu select **File** → **Import**. We could extract a larger submodel region from the cube, but for this tutorial, we import the full cube model and run subsequent crack growth simulations using the full model.

Note: if you create a larger local submodel instead of using the full model, it is best to use a new folder so that you do not overwrite the first set of files.

Select Import a complete model from the dialog, Fig 15.3, set the Mesh File Type and select the appropriate FE file, Fig 15.4. Select **Accept** to continue.

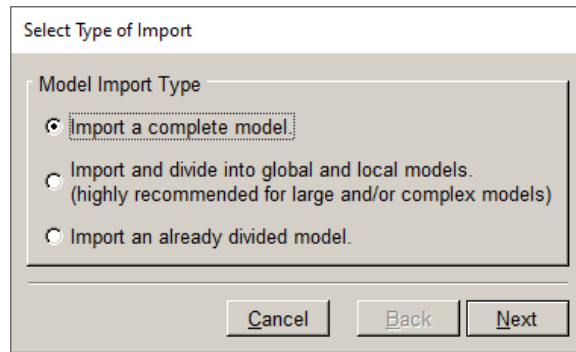


Figure 15.3 FRANC3D model import dialog.

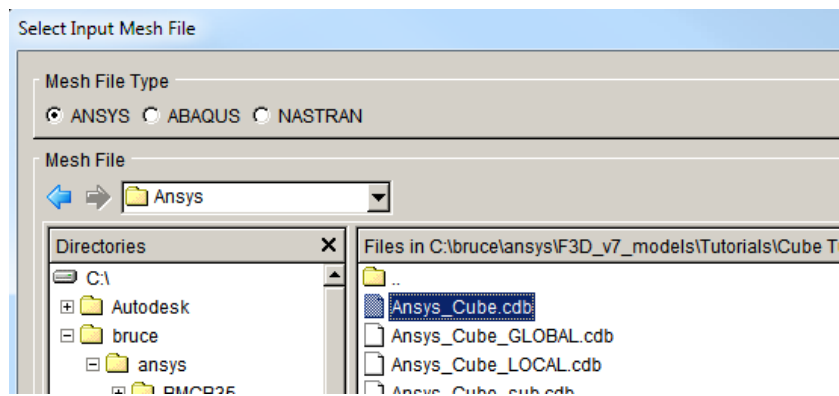


Figure 15.4 FRANC3D mesh import file selector (using ANSYS).

Use Select all in the next dialog, Fig 15.5, to retain all the mesh facets where boundary conditions are applied. Select **Next** when ready; the model will be imported and displayed in the FRANC3D main window. The units are mm and MPa.

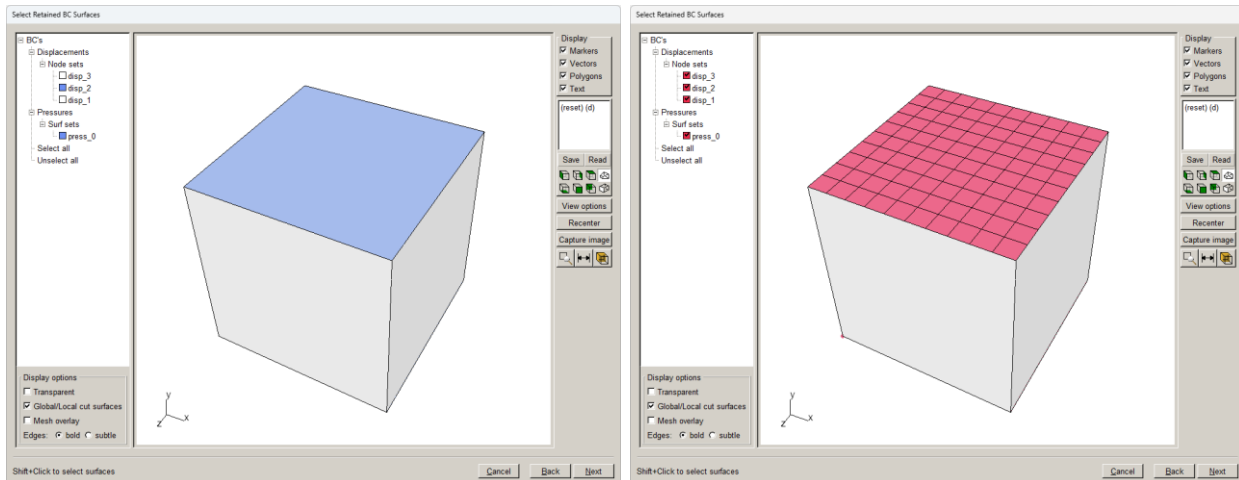


Figure 15.5 Use Select all to retain all highlighted mesh facets.

From the FRANC3D menu, select **Cracks** and **Flaw From Files**. Select the saved *.crk* file, which was extracted and saved from the *.fdb* file. The *.crk* file is read and then displayed in the Orient User Flaw dialog, Fig 15.6. Note that the crack geometry includes the original circular crack and all the subsequent steps of growth. Also note that part of the crack geometry falls outside the model so that intersections can be computed correctly. In general, a crack that is read from a file can be translated in Cartesian space, but for continuing crack growth, we want the exact same location.

Select **Next** to set the mesh template parameters, Fig 15.7; we just use the defaults. Select **Finish** when ready; the crack will be inserted and the model remeshed.

From the FRANC3D menu, select **Static Crack Analysis** and set the analysis parameters. Use a file name that will not overwrite the original crack growth files; for ANSYS, we call it *Ansyz_cube_resume_STEP_005*.

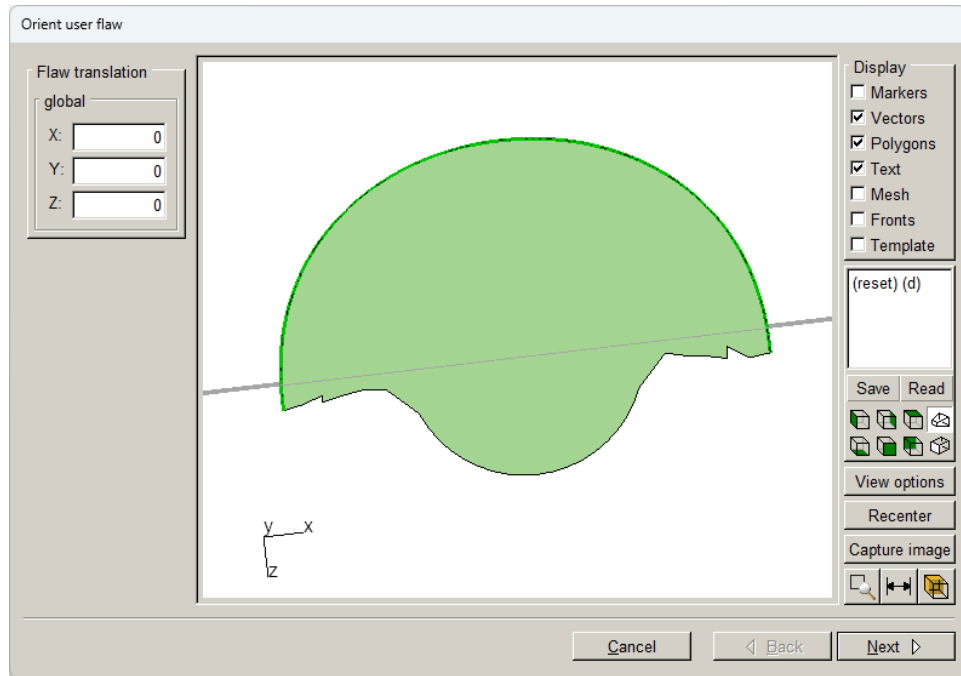


Figure 15.6 Step 5 crack geometry displayed in cube model.

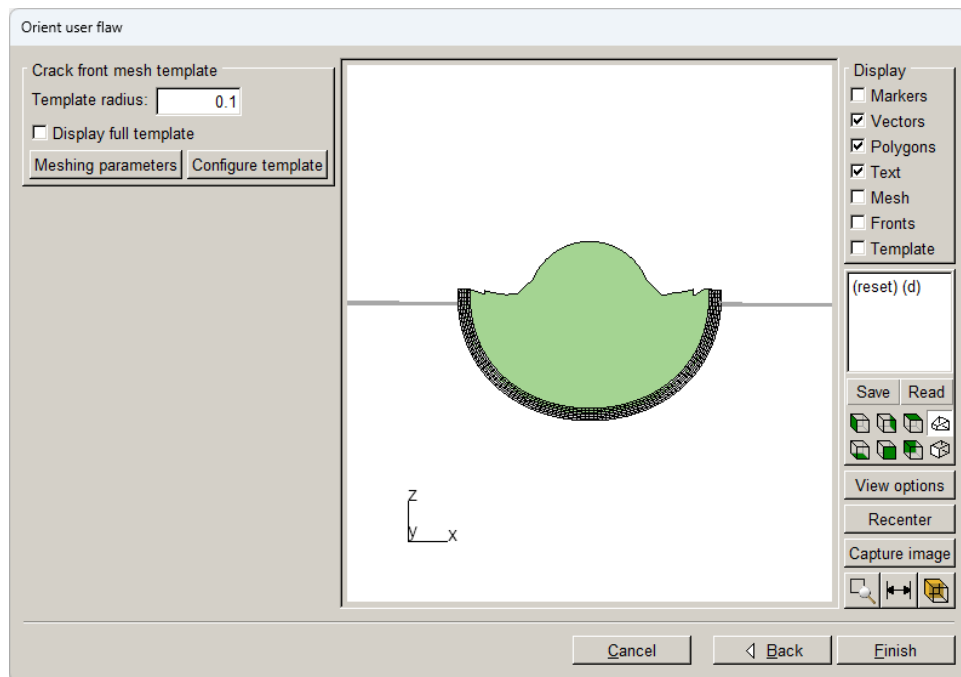


Figure 15.7 Step 5 crack mesh template.

The analysis should produce SIFs that are the same as the SIFs computed for the step_005 using the local submodel region. Fig 15.8 shows Mode I SIFs for the two cases. There are minor differences (at the ends of the curves) because the mesh around the crack front is different (especially near the ends). At this stage, the crack can be propagated further; do another six steps of crack growth using quasi-static growth with a growth step size of 0.2.

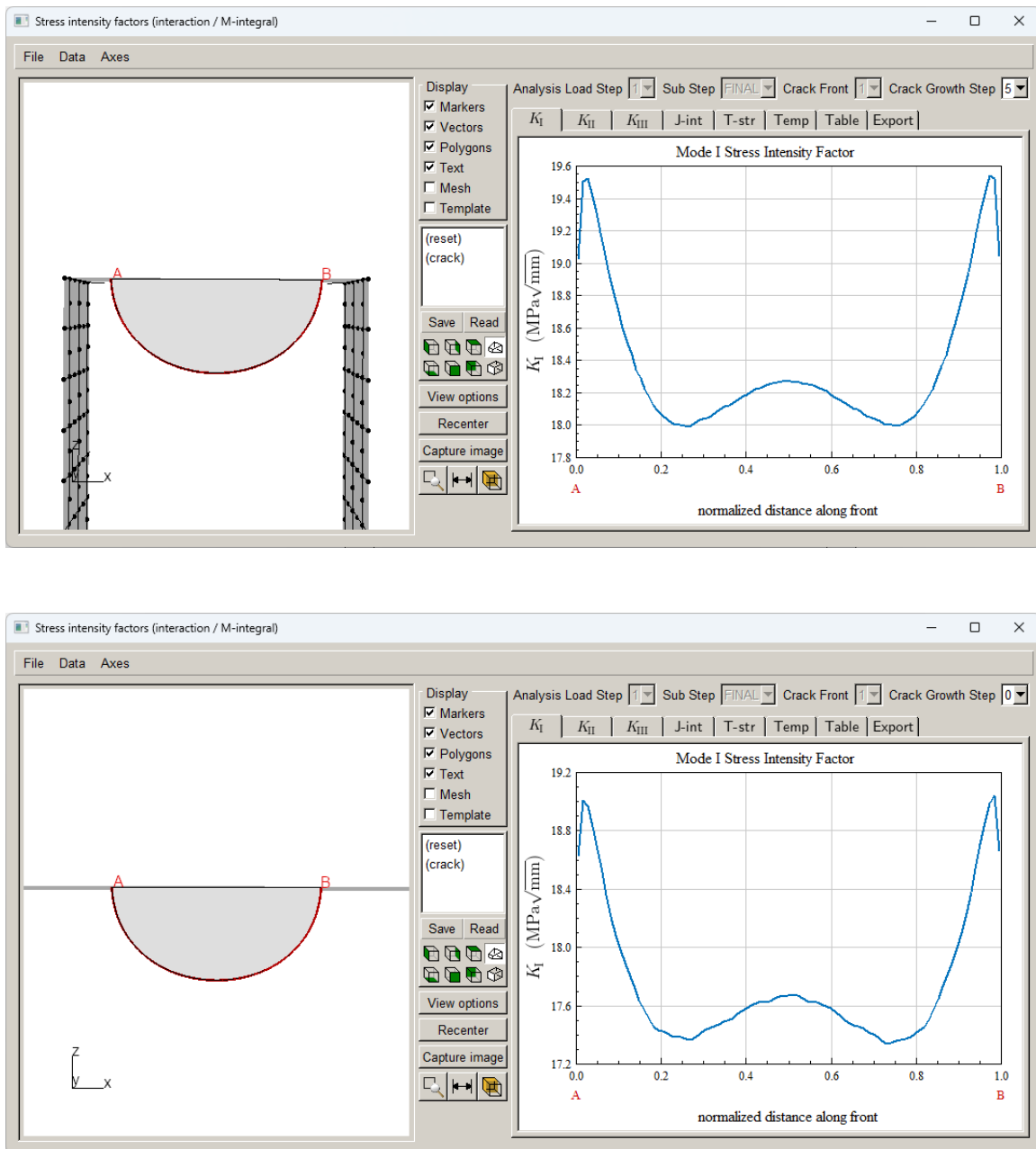


Figure 15.8 Step 5 Mode I SIFs for the original submodel analysis (top) compared with the results using the full model (bottom).

After propagating the crack an additional six steps, we can combine the SIF history for the two analyses. We describe this in the next section using **Create Growth History** under the **Advanced** menu.

15.1.3 Combine SIF Histories

Start FRANC3D, select the **File** → **Open** menu option, and select the *.fdb* file for the first set of analyses (for ANSYS, this is *Ansys_Cube_subdivide_STEP_005.fdb*). From the **Advanced** menu, select **View/Write Growth History**, Fig 15.9. The dialog shown in Fig 15.10 is displayed. Note that the initial crack is labeled as ‘Step 0’, and there are five steps of growth.

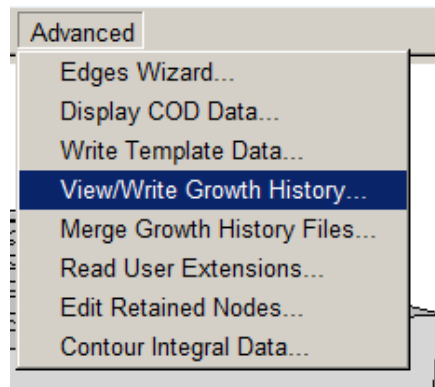


Figure 15.9 FRANC3D Advanced menu → **View/Write Growth History**.

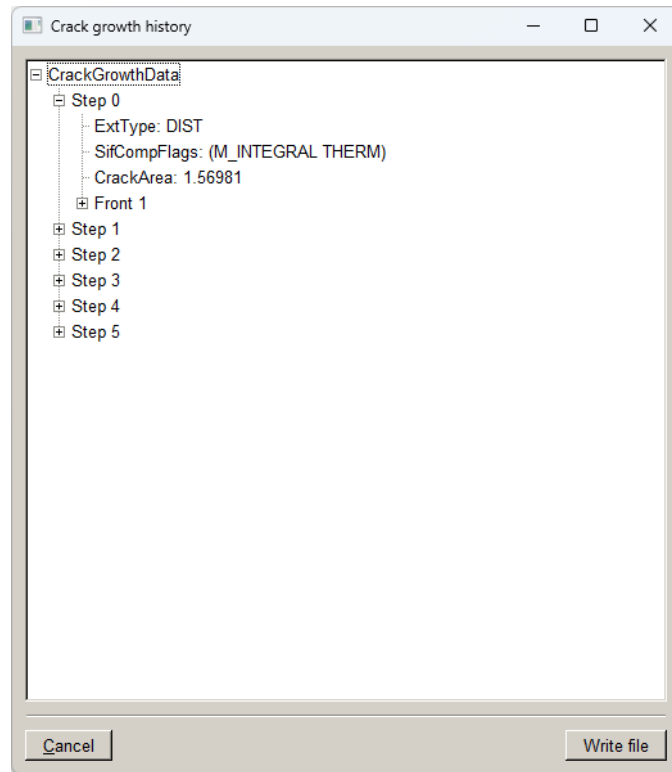


Figure 15.10 Growth History dialog – growth steps 0-5.

Use the **Write file** button in the dialog to save the SIF history to a *.fcg* file (for ANSYS, we use *Ansys_Cube_sub_steps.fcg*). Close the dialog using the **Cancel** button and then close the model in FRANC3D using **File** → **Close**.

Now open the final growth step for the ‘resumed’ set of analyses (for ANSYS, *Ansys_Cube_resume_STEP_006.fdb*) using **File** → **Open**. As with the submodel, save the history using the Growth History dialog. The history is shown in Fig 15.11. Note that ‘Step 0’ in the ‘resumed’ model gives the same SIFs as ‘Step 5’ in the submodel; however, ‘Step 0’ in the ‘resumed’ model includes crack extension data.

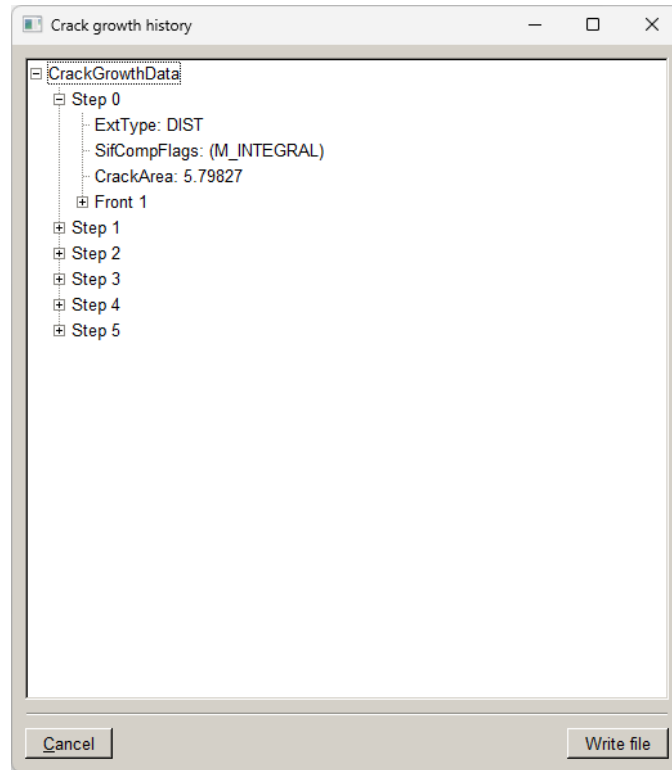


Figure 15.11 Create Growth History dialog – “resume” model fronts for growth steps 0-6.

Save the ‘resumed’ model SIF history to a *.fcg* file (for ANSYS, *Ansys_Cube_resume_steps.fcg*). Close the Growth History dialog and close the model using **File** → **Close**. This leaves the FRANC3D main model window empty. Select the **Advanced** → **Merge Growth History Files** menu option to display the Merge History dialog; it will not have any data, Fig 15.12.

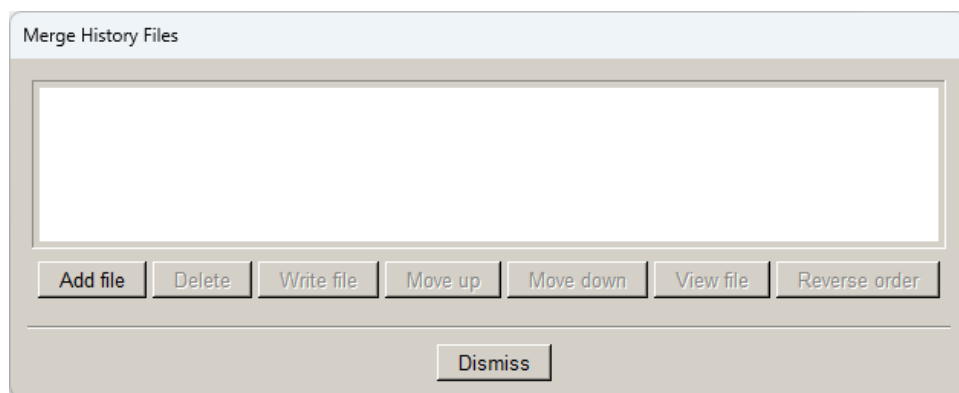


Figure 15.12 Merge Growth History Files dialog.

Use the **Add file** button in the Merge History dialog and select the first *.fcg* file (for ANSYS, *Ansys_Cube_sub_steps.fcg*). Using the same button select the second *.fcg* file (for ANSYS, *Ansys_Cube_resume_steps.fcg*). Use the **Write file** button in the dialog to save the combined SIF history to a *.fcg* file (for ANSYS, *Ansys_Cube_combined_steps.fcg*). The Merge History dialog can then be dismissed.

The combined SIF history data can be imported into the Fatigue Life module using the **Read SIF Data** button, Fig 15.13. You can **Set** or **Read Parameters** to plot the cycles, Fig 15.14. Note that between crack growth step 5 and 6 in this plot, the cycle count does not change; this is because the last front in the initial crack growth simulation is the same as the first front in the resumed simulation, but the last step in the initial simulation does not include extension. The duplicate front (the one without extension) can be removed from the combined history if needed. There are Python scripts available that can be used to edit a *.fcg* file.

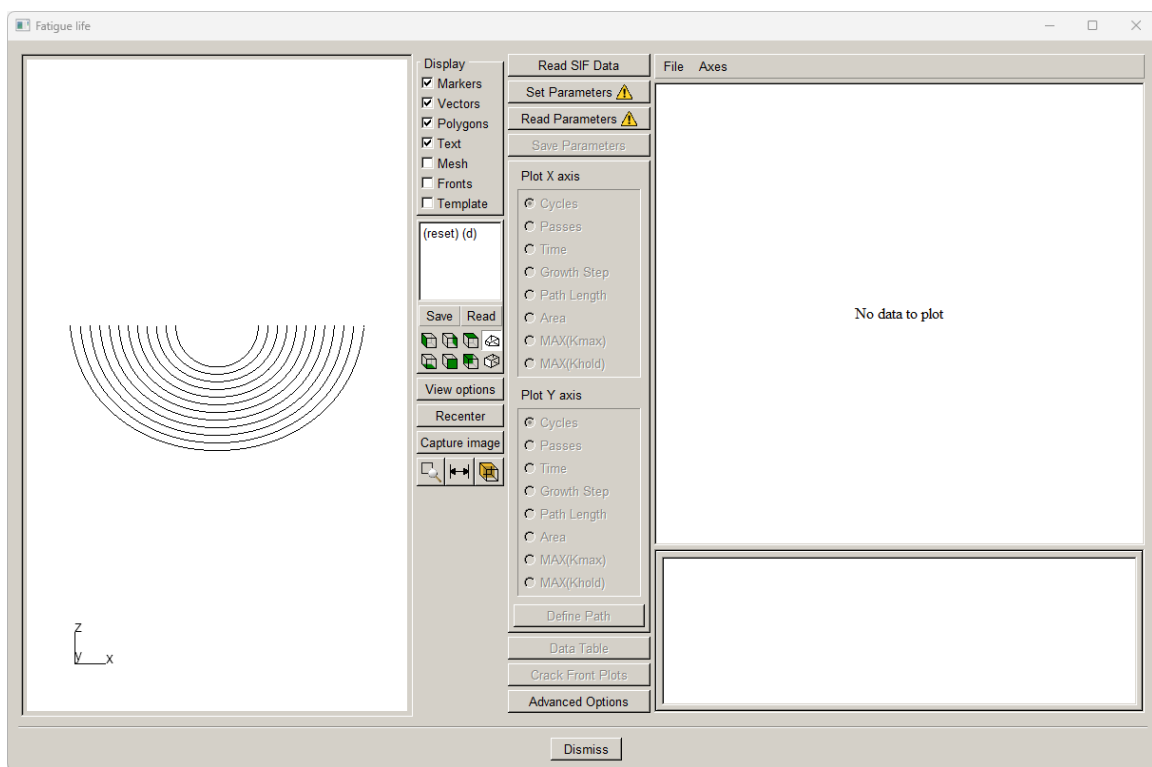


Figure 15.13 Fatigue Life dialog.

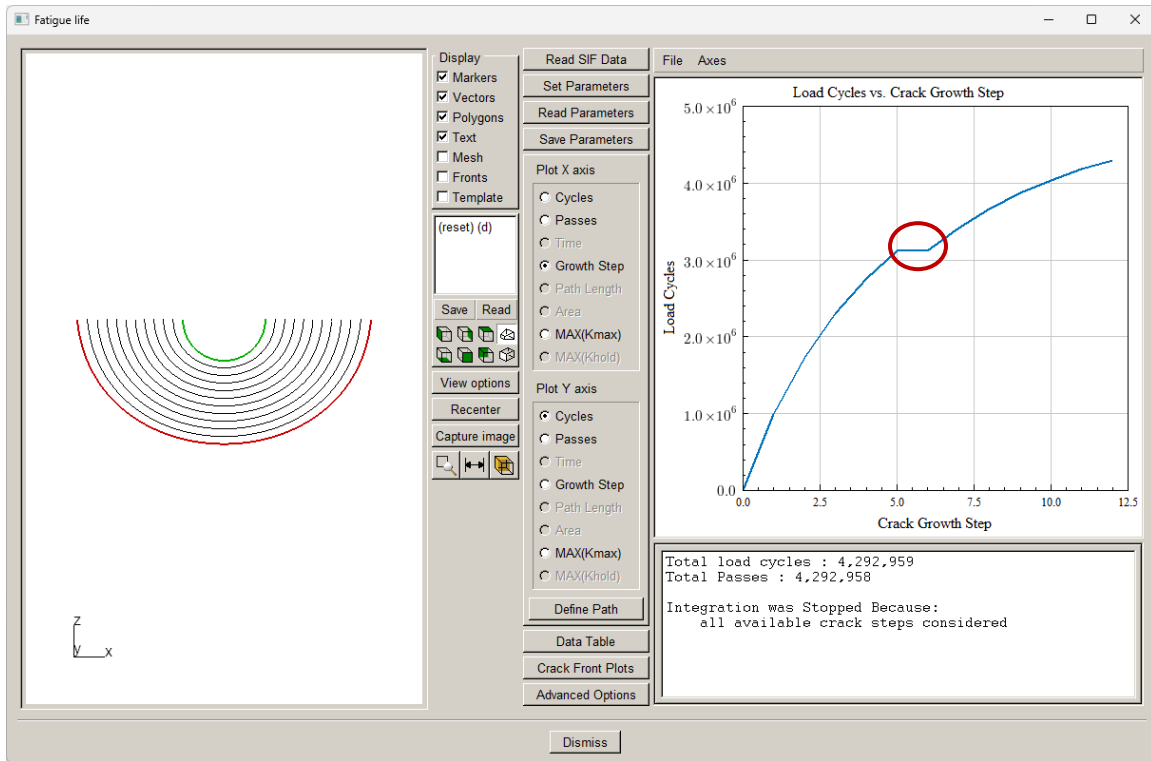


Figure 15.14 Fatigue cycles using Paris model from Tutorial #1.

15.2 Building a SIF History from Multiple Crack Insertions

One can build a history of SIFs from a series of crack insertions rather than growing a crack from the initial crack. In this section, we will insert a series of half-ellipse surface cracks in a cube (the same cube as the previous section). This might represent a series of observed crack fronts from a fatigue test for example.

The simplest way to insert a series of cracks is to use the Python module as described in Section 9. However, for this example we will insert the first crack using the GUI and then use the session log that is created to insert subsequent cracks. We start by importing the complete cube model, Fig 15.15.

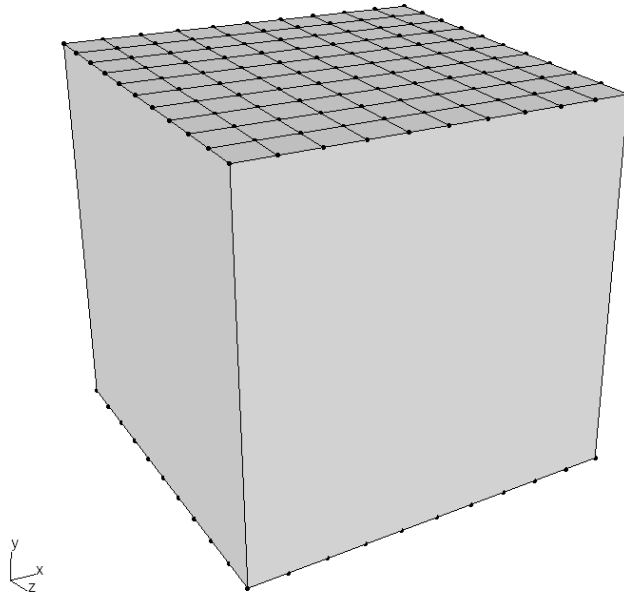


Figure 15.15 Complete cube model.

The first crack is an elliptical surface crack with dimensions of 1.0 and 0.5 (for a and b in the FRANC3D dialog) and centered in the ‘front’ face of the cube, Fig 15.16.

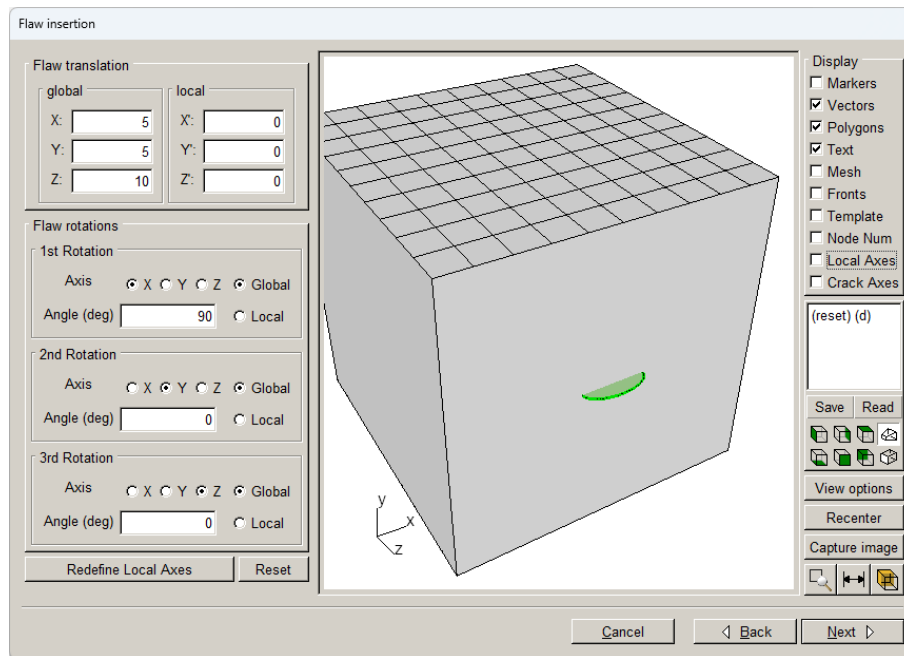


Figure 15.16 Elliptical surface crack inserted.

The template radius is 0.05 and all template parameters are the defaults. After the static crack analysis is done, the SIFs can be computed, Fig 15.17. A *.fcg* file can be saved using the **Advanced** → **View/Write Growth History** menu option and dialog. We can also save the fatigue crack growth data (*.fcg*) file by playback of the WriteCrackData command. In practice, a user would run all the session log commands in batch.

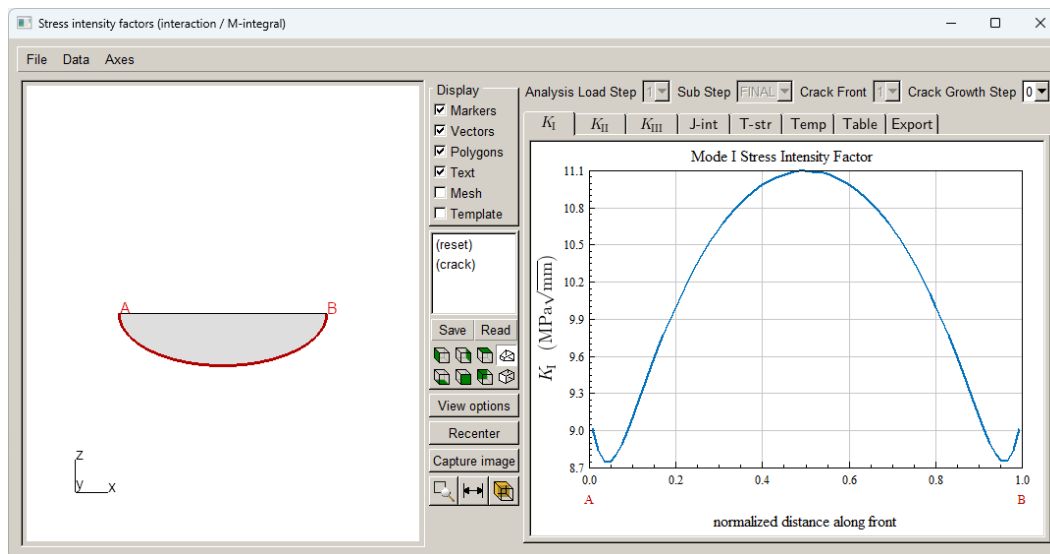


Figure 15.17 SIFs for elliptical crack with (a and b) dimensions 1.0 and 0.5.

The session log file for the simulation is listed here:

```
OpenMeshModel(
  model_type=ANSYS,
  file_name='Ansys_Cube.cdb',
  retained_nodes_file='Ansys_Cube_RETAINED.txt',
  retain_auto_cut_surf=false)
```

```
SetUnits(
  model_length=MM,
  model_stress=MPa,
  temperature=C)
```

```
InsertParamFlaw(
  flaw_type=CRACK,
  crack_type=ELLIPSE,
  flaw_params=[1,0.5],
```

```

temp_radius=0.05,
rotation_axes=[1],
rotation_mag=[90],
translation=[5,5,10])

RunAnalysis(
    model_type=ANSYS,
    file_name='crack_1x05.fdb',
    flags=[NO_CFACE_TRACT,NO_CFACE_CNTCT],
    command="ANSYS242.exe" -b -p ansys -smp -i "crack_1x05.cdb" -o
    "crack_1x05.out",
    del_file_list=[bcs,emat,esav,full,err,ldhi,log,mntr,osav,pcs,rdb,stat,txt,mlv,mode,r001])

ComputeSif(
    do_therm_terms=true,
    ref_temp=0)

WriteSif(
    file_name="crack_1x05.sif",
    crack_step=0,
    load_step=1,
    flags=[TAB,KI,KII,KIII],
    do_therm_terms=true,
    ref_temp=0)

WriteCrackData(
    file_name='c01.fcg')

CloseModel()

```

We can make a copy of this and edit the following:

```

flaw_params in InsertParamFlaw()
file_name in RunAnalysis()
command in RunAnalysis (to match the file_name)
file_name in WriteSif()
file_name in WriteCrackData()

```

After saving the edited commands to a new *.log* file, the new file can be played back from the command line using the *-batch* keyword:

```
> franc3d.exe -batch session2x1.log
```

Once all crack insertions have been completed, the SIF history can be created. We use a Python script that reads and combines *.fcg* files. This script is available from the FAC website; it is #4 in the list and is called *GrowData.py*.

The command to merge the *.fcg* files is:

```
> python.exe GrowthData.py -merge c01.fcg c02.fcg c03.fcg c04.fcg c05.fcg > merged.fcg
```

We can use the **Advanced** → **Merge Growth History Files** menu option to read and merge the *.fcg* files, Fig 15.18. After writing the merged file, you can read and display it using the same menu and dialog, Fig 15.19.

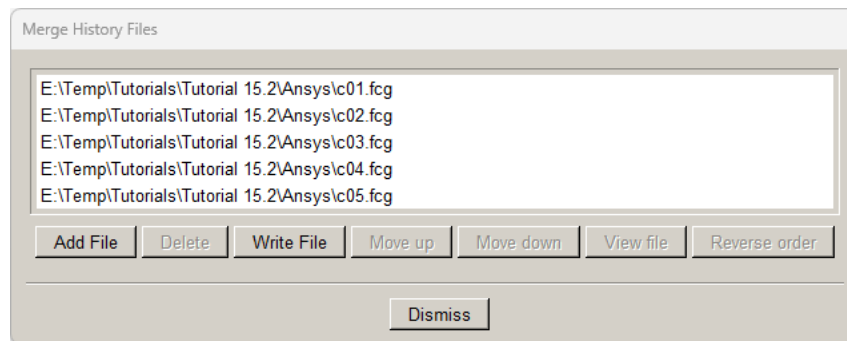


Figure 15.18 Merge Growth History dialog.

The merged data can be imported into the Fatigue Life module, Fig 15.20, using the **Read SIF Data** button; **Set** or **Read Parameters** to plot the cycles, Fig 15.21.

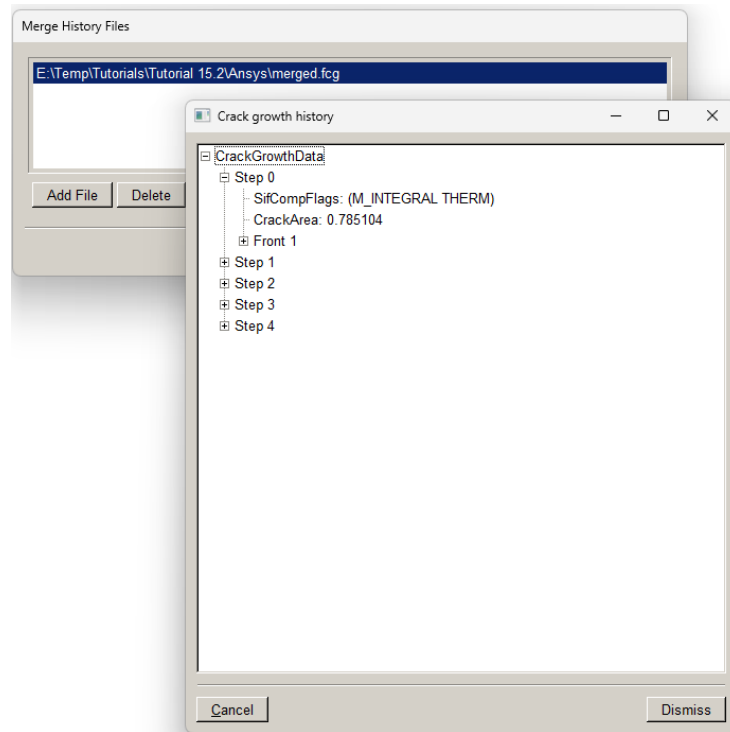


Figure 15.19 Merge Growth History dialog with combined crack fronts.

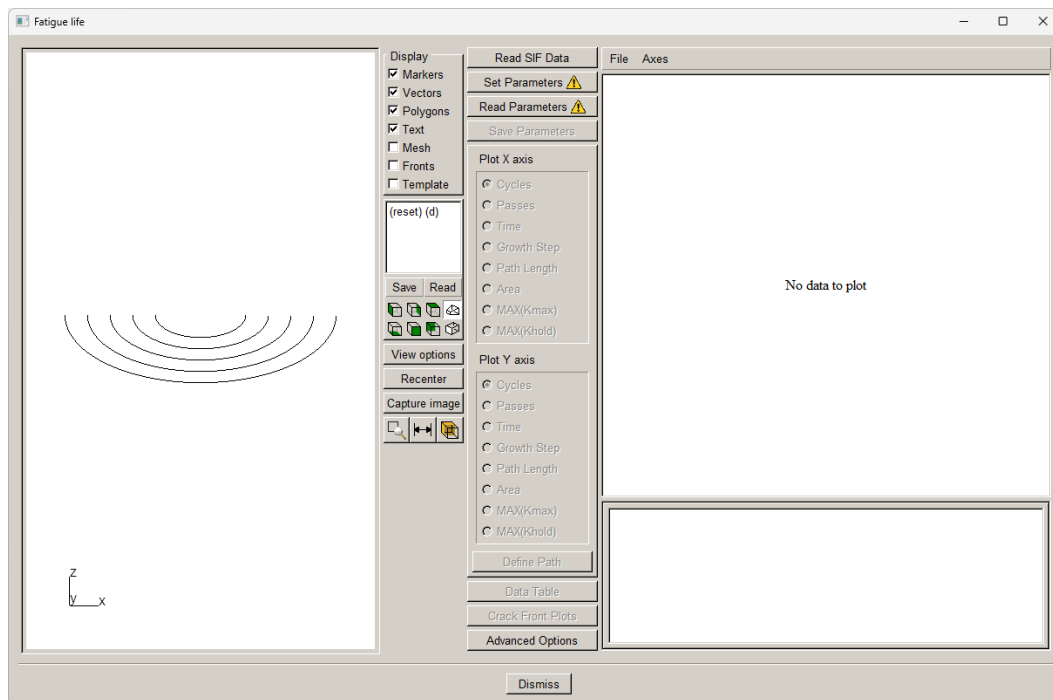


Figure 15.20 Fatigue life dialog with combined crack fronts.

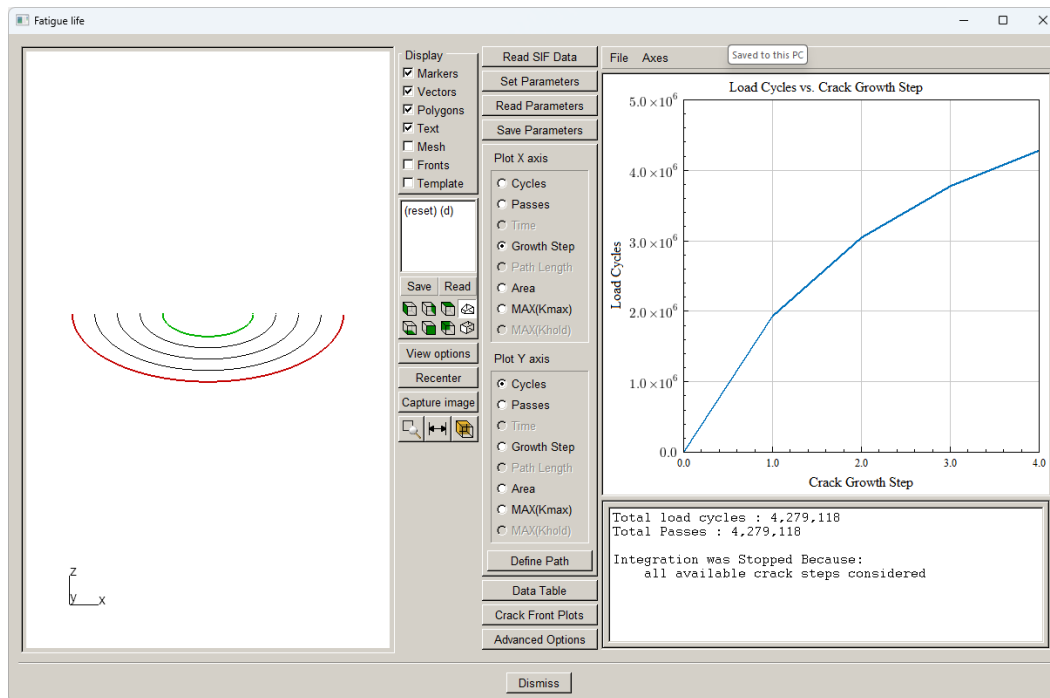


Figure 15.21 Fatigue cycles using Paris model from Tutorial #1.