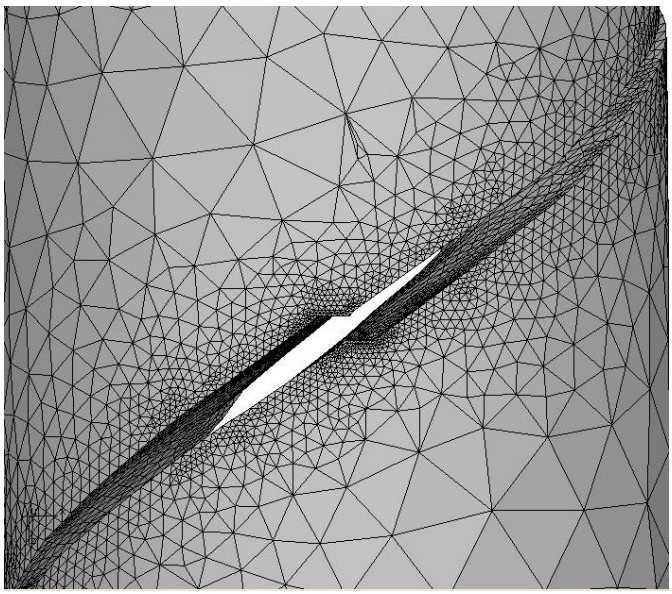
FRANC3D

ABAQUS™ Tutorial

ABAQUS is a registered trademark of Dassault Systèmes www.simulia.com



Version 9.2

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

Revised: Oct 2026

Table of Contents:

1.0 Introduction 4

2.0 Tutorial 1: Crack Insertion and Growth in a Cube 6

2.1 Step 1: Create the ABAQUS FE Model 6

2.2 Step 2: Reading ABAQUS FE Model into FRANC3D 8

Step 2.1: Importing Complete ABAQUS FE Model 8

Step 2.2: Select the Retained Items in the FE Model 10

2.3 Step 3: Importing and Subdividing the Model 13

2.4 Step 4: Insert a Crack 19

Step 4.1: Define a new Crack from FRANC3D Menu 19

Step 4.2: Insert Cracks from Files 25

2.5 Step 5: Static Crack Analysis 28

Step 5.1: Select Static Crack Analysis 28

Step 5.2: Select ABAQUS Analysis Options 30

2.6 Step 6: Compute Stress Intensity Factors 32

2.7 Step 7: Manual Crack Growth 34

Step 7.1: Select Grow Crack 34

Step 7.2: Specify Fitting and Extrapolation 37

Step 7.3: Specify Crack Front Template 39

2.8 Step 8: Automatic Crack Growth 41

Step 8.1: Open FRANC3D Restart File 41

Step 8.2: Select Crack Growth Analysis 41

Step 8.3: Specify Growth Rules 43

Step 8.4: Specify Fitting and Template Parameters 44

Step 8.5: Specify Growth Plan 45

Step 8.6: Analysis Code Settings 46

2.9 Step 9: SIF History and Fatigue Life 48

Step 9.1: Select SIFs Along a Path 49

Step 9.2: Select SIFs For All Fronts 51

Step 9.3: Select Fatigue Life Predictions 51
Appendix A: ABAQUS local model between *Tie constraints 62
Appendix B: ABAQUS model with *Tied solid and shell elements. 73
Appendix C: ABAQUS Keywords 79

1.0 Introduction

This tutorial introduces the fracture simulation capabilities of FRANC3D Version 9 and ABAQUS 2026; other versions of ABAQUS should work; the version should be set in the preferences. FRANC3D is introduced by analyzing a single surface crack in an ABAQUS finite element (FE) model of a cube.

Subsequent tutorials (see the FRANC3D Tutorials #2-15 document) build on this first example and describe additional capabilities and features of the software. It is intended that the user perform operations as they are presented, but you can experiment and consult other reference documentation whenever necessary.

The FRANC3D menu and dialog button selections are indicated by bold text, such as **File** menu. Window regions along with dialog options, fields and labels are underlined. Model names and file names are indicated by *italic text.*

The FRANC3D main window is shown in Fig 1.1. When you start the tutorial, the first thing to do is set the working directory using the **File →Work Directory** menu option, Fig 1.1.

Before or after a model is imported, you can set the FE model units, Fig 1.2. The selected units are applied to plots and figures that FRANC3D displays and are used during crack growth in combination with fatigue crack growth rate data, which also has units.

You can also set preferences; it is helpful to set the ABAQUS executable path. You can view the preferences using the **Edit →Preferences** menu option. The Preferences dialog is described in Section 5.4 of the FRANC3D Reference document.

There is a preference setting for the ABAQUS version. The latest (2025) version uses Python 3, whereas versions older than 2025 used Python 2. FRANC3D generates a *writeDtpFile.py* script so the version setting is important.

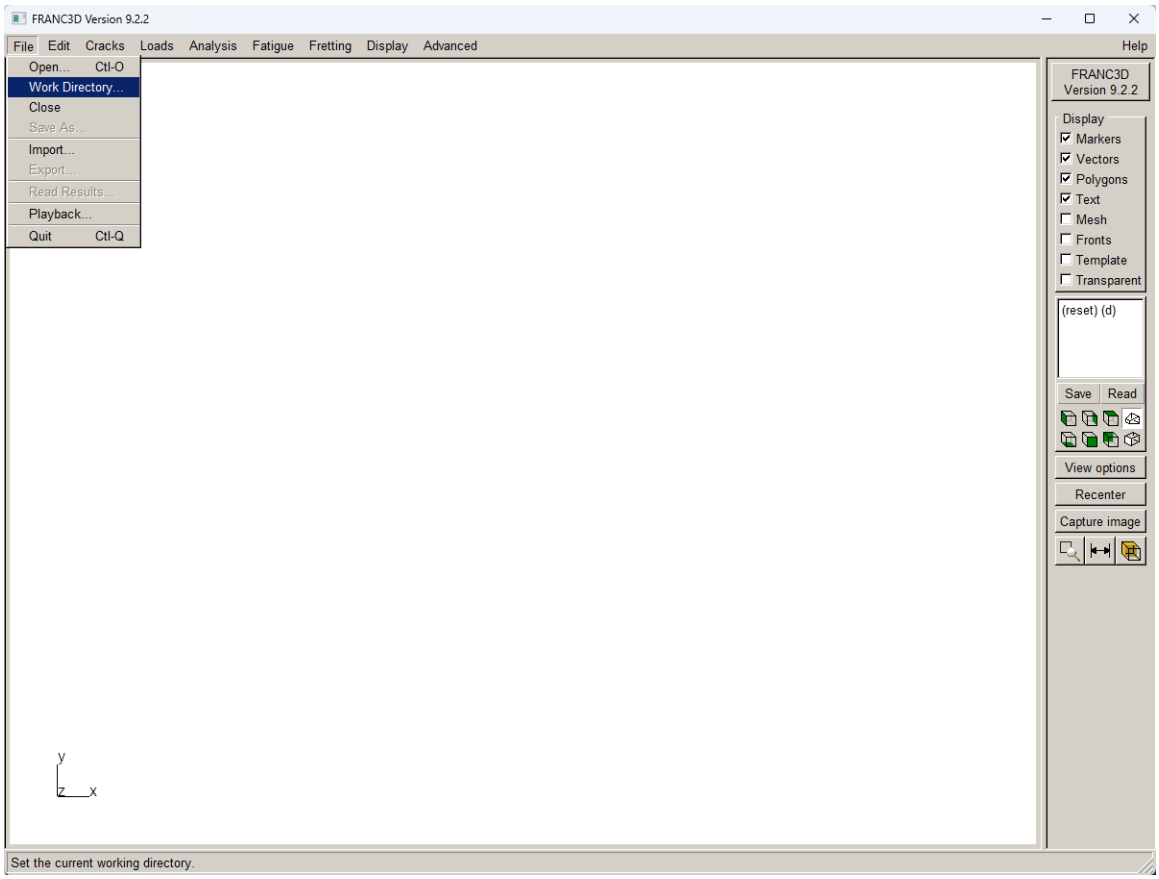


Figure 1.1 FRANC3D main window – Work Directory menu option.

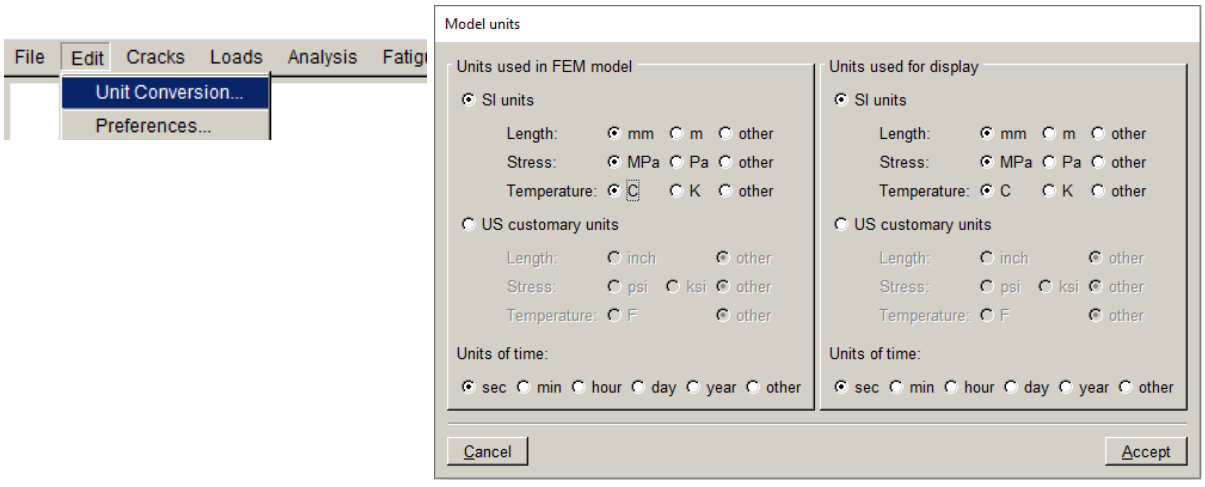


Figure 1.2 FRANC3D FE model unit conversion menu option and dialog.

2.0 Tutorial 1: Crack Insertion and Growth in a Cube

In this tutorial, a single surface crack in a cube under far-field tension is modeled. It is assumed that the user is familiar with a pre-processor for ABAQUS; we use ABAQUS CAE. Once the FE model is created, the FRANC3D steps to: 1) read the model, 2) insert a crack, 3) perform the ABAQUS analysis, and 4) compute stress intensity factors, are all described.

2.1 Step 1: Create the ABAQUS FE Model

First, you must create a cube model using any pre-processor for ABAQUS. We outline the steps to create the FE model with boundary conditions to ensure that you have a model that can be used with FRANC3D.

1. Create a 10x10x10 cube geometry; units of length are mm.
2. Subdivide the edges for meshing, using 10 to 20 subdivisions.
3. Define the element type as quadratic; use brick or tetrahedral elements to mesh the volume.
4. Define the material properties; elastic modulus (E) is 10000 and Poisson's ratio is 0.3; units for E are MPa.
5. Boundary conditions consist of displacement constraints on the bottom surface and uniform traction (a negative pressure) on the top surface. The bottom surface is constrained in the y-direction, the bottom left edge is also constrained in the x-direction, and the point at the origin is also constrained in the z-direction. The top surface traction is 10 MPa.
6. Write the *.inp* file (*Abaqus_Cube.inp*).

The resulting model is shown in Fig 2.1. The symbols for the boundary conditions are displayed attached to the model; the brick mesh is displayed in the lower right corner.

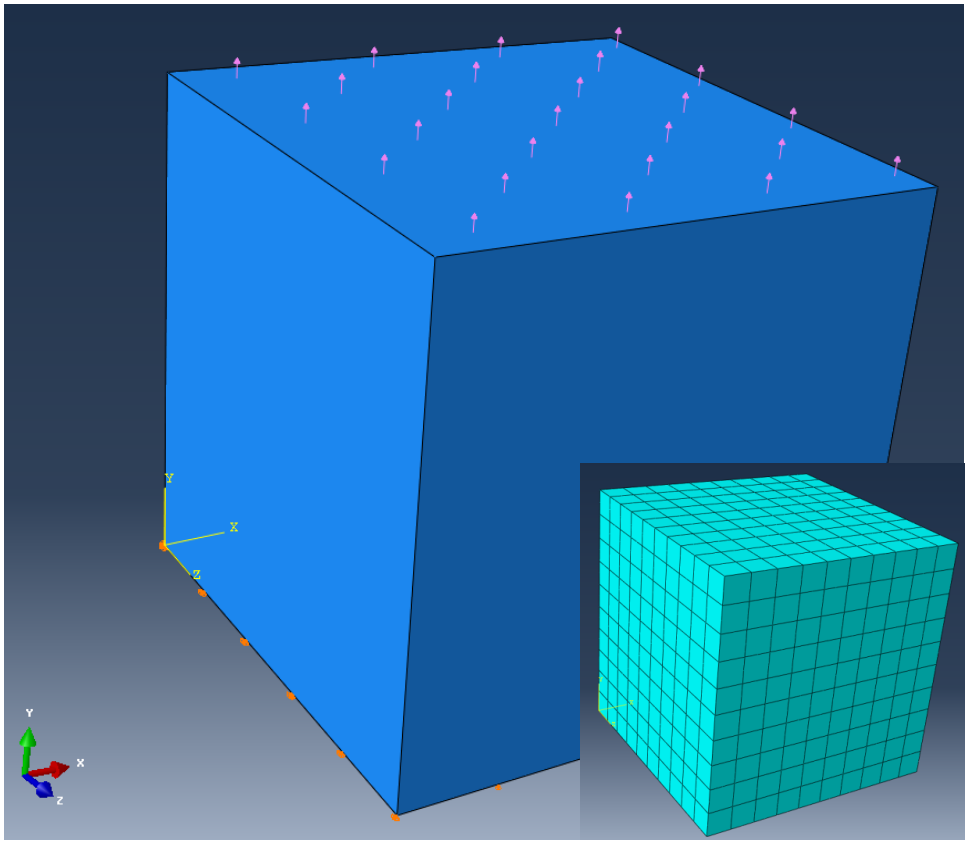


Figure 2.1 ABAQUS cube with boundary conditions; meshed with brick elements.

Note that ABAQUS CAE uses default set and surface names such as SET-1 at both the Part and Instance levels, Fig 2.2. Instance SET-1 can be both a node and an element set. If boundary conditions are applied in CAE using the Instance SET-1 (right side image), FRANC3D might apply the boundary conditions to both Part and Instance SET-1 nodes.

Use unique names for sets and surfaces in CAE, especially when defining boundary conditions.

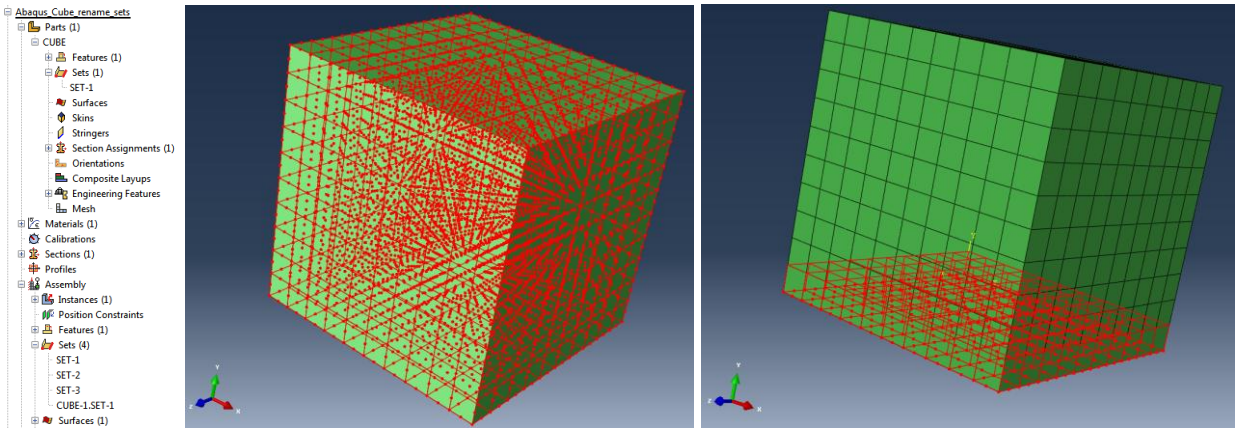


Figure 2.2 ABAQUS CAE Set-1 at both Part and Instance levels.

2.2 Step 2: Reading ABAQUS FE Model into FRANC3D

Once the ABAQUS FE model is created and the *.inp* file is saved, we proceed with importing the FE model into FRANC3D. We use the *Abaqus_Cube.inp* file from ABAQUS CAE that was created in the previous step.

You can choose to do either Step 2 or Step 3 (see Section 2.3).

Step 2.1: Importing Complete ABAQUS FE Model

Start with the FRANC3D graphical user interface (see Fig 1.1) and select **File →Import,** Fig 2.3. In the dialog shown in Fig 2.4, choose Complete Model and select **Next**.

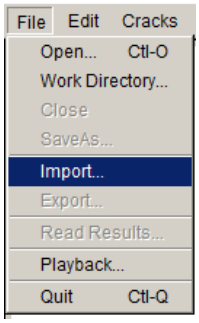


Figure 2.3 File Import menu option.

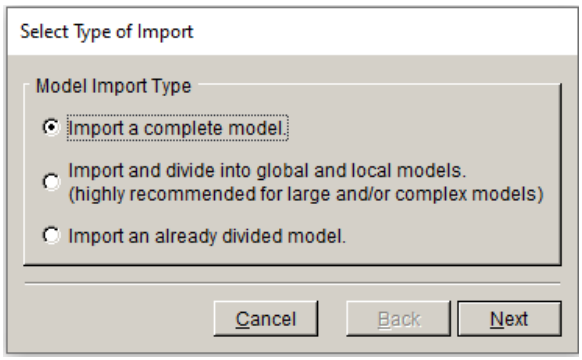


Figure 2.4 Model Import Type dialog.

Switch the Mesh File Type radio button in the Select Import Mesh File window, Fig 2.5, to ABAQUS and select the file name for the model, *Abaqus_Cube.inp*, and then select **Next**.

Note that the mesh file type can be set in the Preferences so that you do not have to change the mesh file type radio button if you always use ABAQUS.

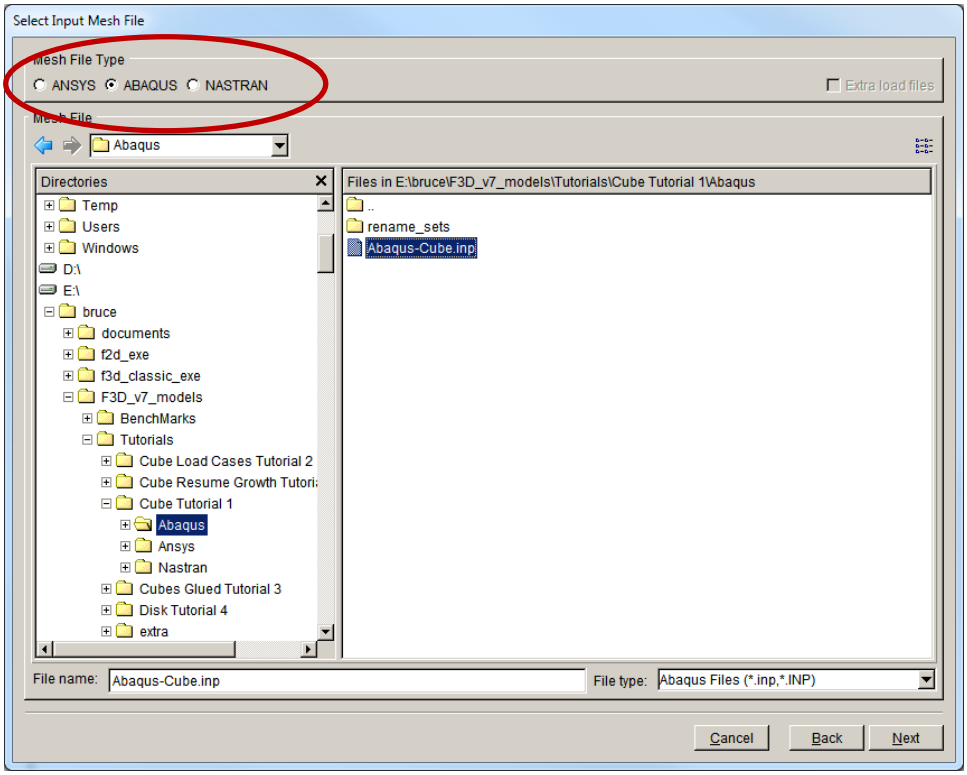


Figure 2.5 Select Input Mesh File dialog box.

Step 2.2: Select the Retained Items in the FE Model

The next panel, Fig 2.6, allows you to choose the mesh surface facets that are retained from the ABAQUS *.inp* file. Surfaces with boundary conditions are blue, Fig 2.7, and turn red when selected, Fig 2.8. You can rotate the model to view all surfaces. We retain the surfaces with boundary conditions (top and bottom of the cube) by choosing Select all. The boundary conditions will be transferred automatically to the new mesh once a crack is inserted.

Select **Next** to proceed. The Units dialog will be displayed; if you have already set the units using the **Edit** menu, just select **Finish**.

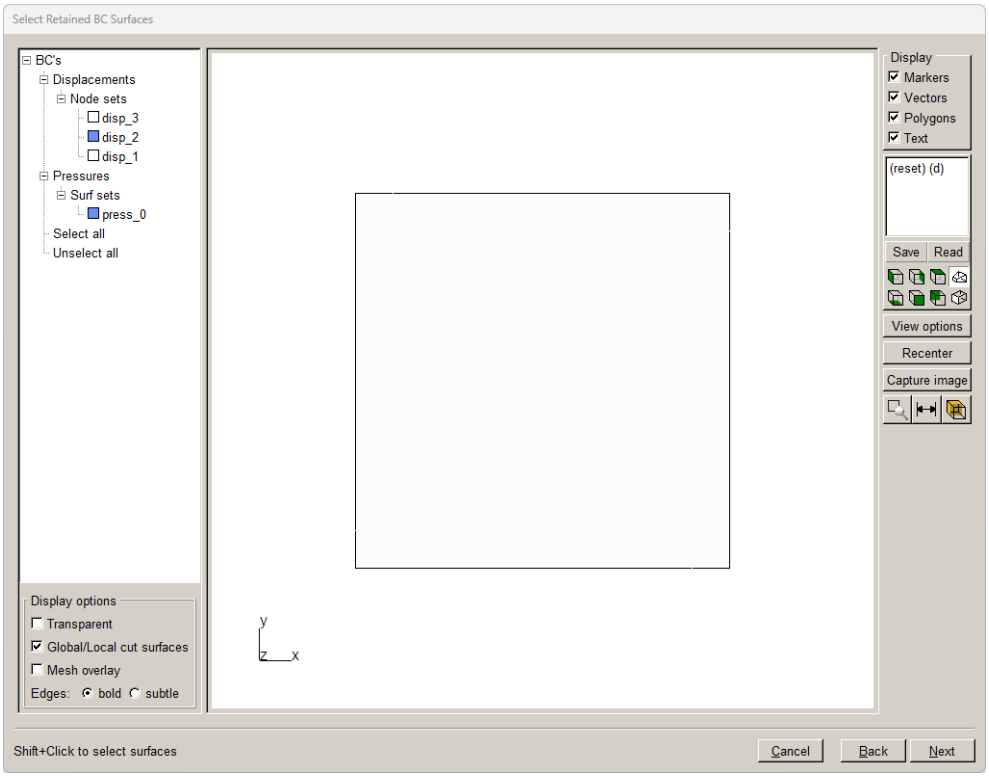


Figure 2.6 Select Retained BC Surfaces wizard panel.

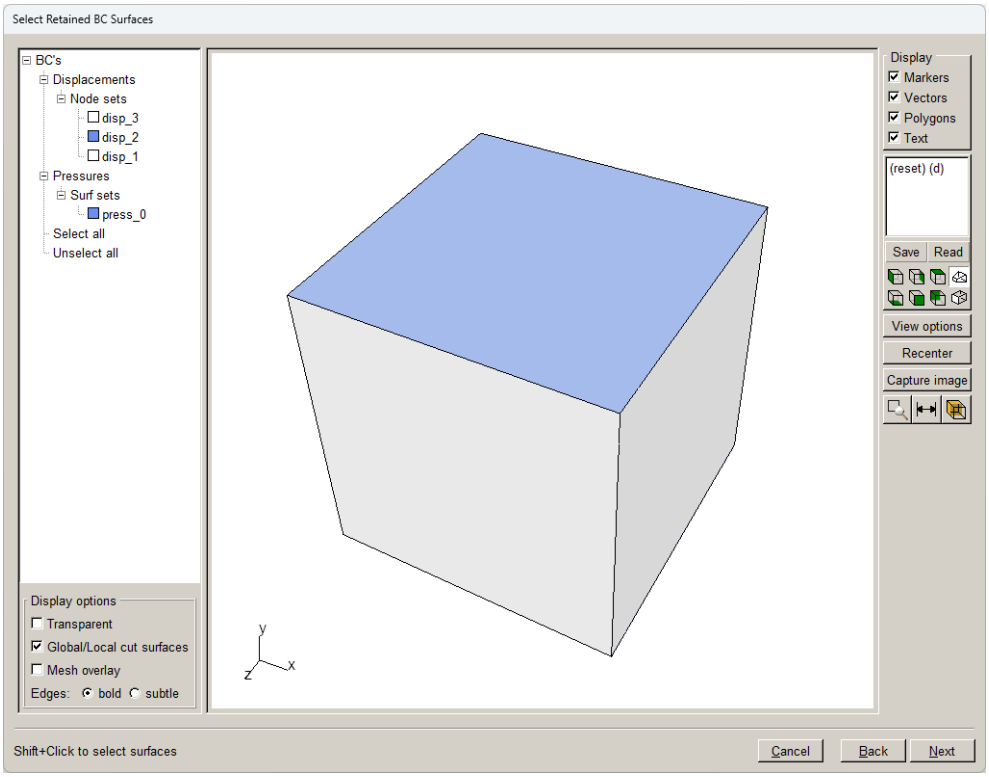


Figure 2.7 ABAQUS Model retain BC Surfaces wizard panel - unselected surface.

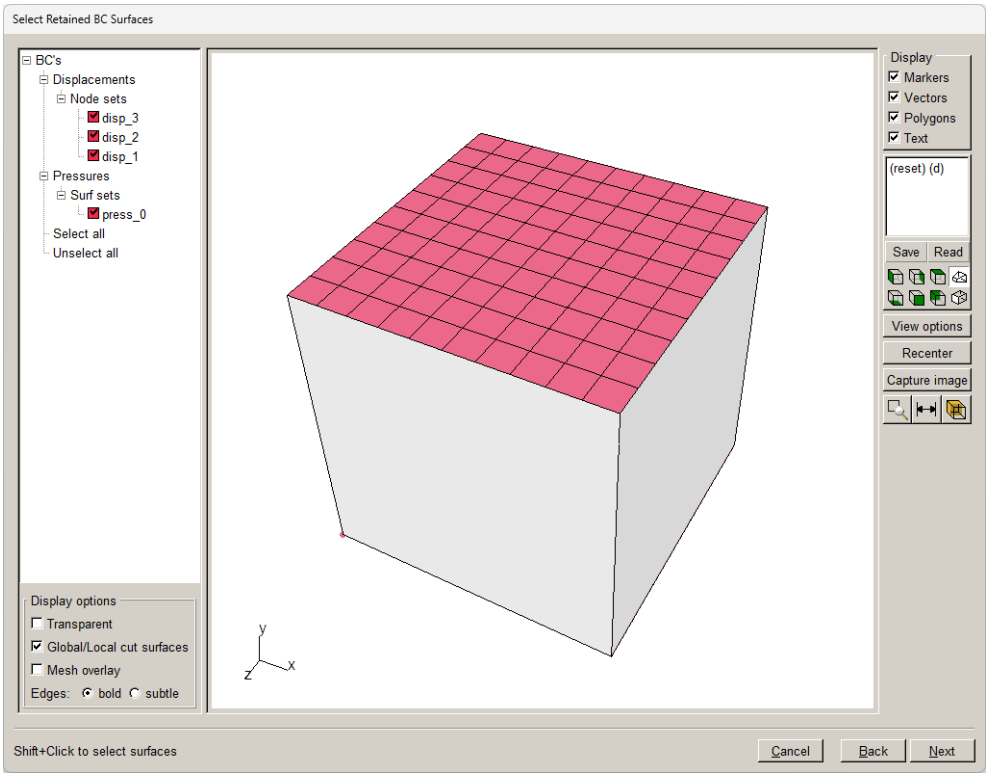


Figure 2.8 ABAQUS Model retain BC Surfaces wizard panel – selected surfaces.

Surface Mesh NOT Retained

The surface mesh facets do not have to be retained, and if a crack is inserted into a surface that has boundary conditions, then the surface mesh must not be retained. In such a case, the boundary condition data is mapped to the remeshed surface. In practice, transfer of boundary condition data is simpler and more precise if the surface mesh is retained, but sometimes this is not possible. FRANC3D will automatically transfer the boundary condition data to the new mesh in either case.

The model is displayed in the FRANC3D modeling window, Fig 2.9. You can turn on the surface mesh, or manipulate the model view by rotating, *etc*. Fig 2.9 shows the retained mesh on the top surface (retained bottom surface is not visible) where the traction boundary conditions are applied. See Section 2 of the Reference document for a description of the view manipulation.

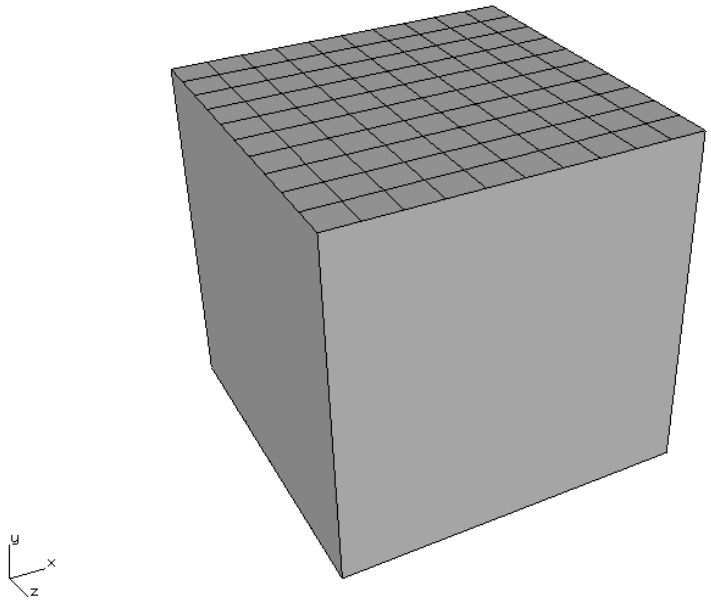


Figure 2.9 ABAQUS model imported into FRANC3D with retained facets on the top surface.

If you have not set a working directory (using the **File →Work Directory** menu option), FRANC3D might present the dialog shown in Fig 2.10 prior to displaying the model. Select **Yes** to set the work directory as the folder containing the ABAQUS *.inp* file. Many files can be created during a crack growth simulation, and it is best to keep them together in one folder.

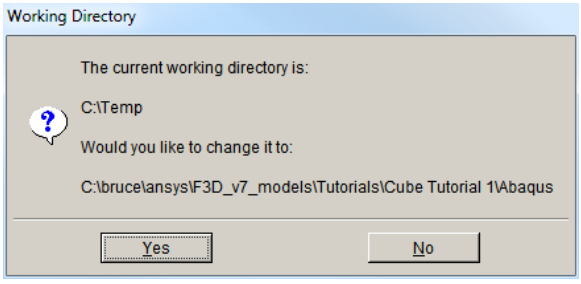


Figure 2.10 Set Working Directory dialog.

2.3 Step 3: Importing and Subdividing the Model

If you did Step 2, close the model using the **File →Close** menu option before starting this step.

The ABAQUS model can be split into smaller parts before inserting the crack. Go to **File →Import** and choose the Import and divide into global and local models radio button, Fig 2.11. Select **Next**, and then choose the *Abaqus_Cube.inp* model, Fig 2.12. Remember to set the Mesh File Type radio button to ABAQUS.

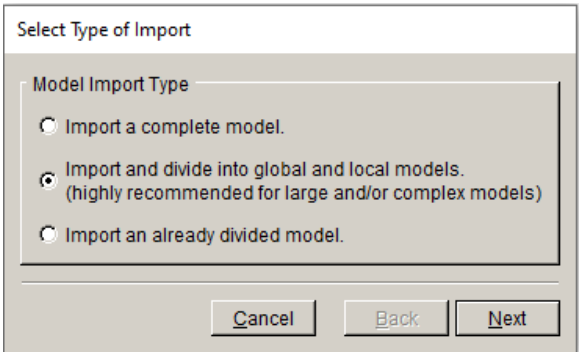


Figure 2.11 Model Import Type dialog with Import and divide selected.

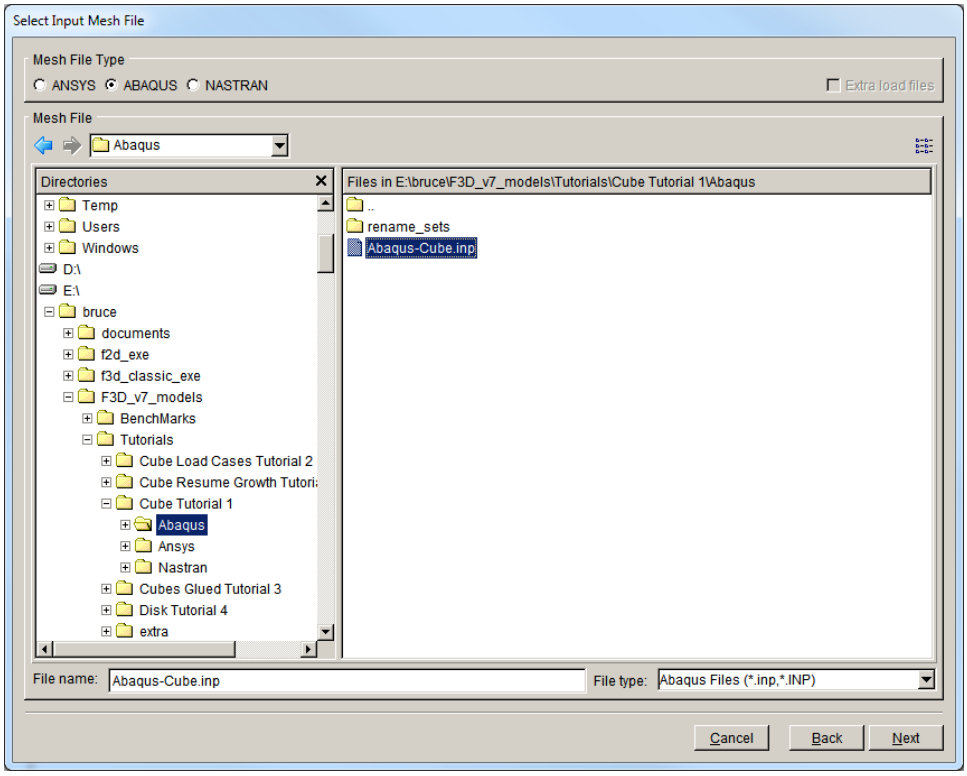


Figure 2.12 Select Input Mesh File selection.

Instead of the Retain BC Surfaces dialog (see Fig 2.6), the Define a Local Submodel window, Fig 2.13, appears. Figs 2.14 and 2.15 show two of the submodel selection tools. Using the options on the left, a submodel is selected; elements to be retained in the submodel appear red, and the selection is confirmed using the **Crop** button. See Section 4.5 of the Reference document for a more complete description of the submodeling tools.

For this tutorial, the rectangular Rubberband Box is used to create a smaller portion of the cube, Fig 2.16, which has an exposed face for crack insertion.

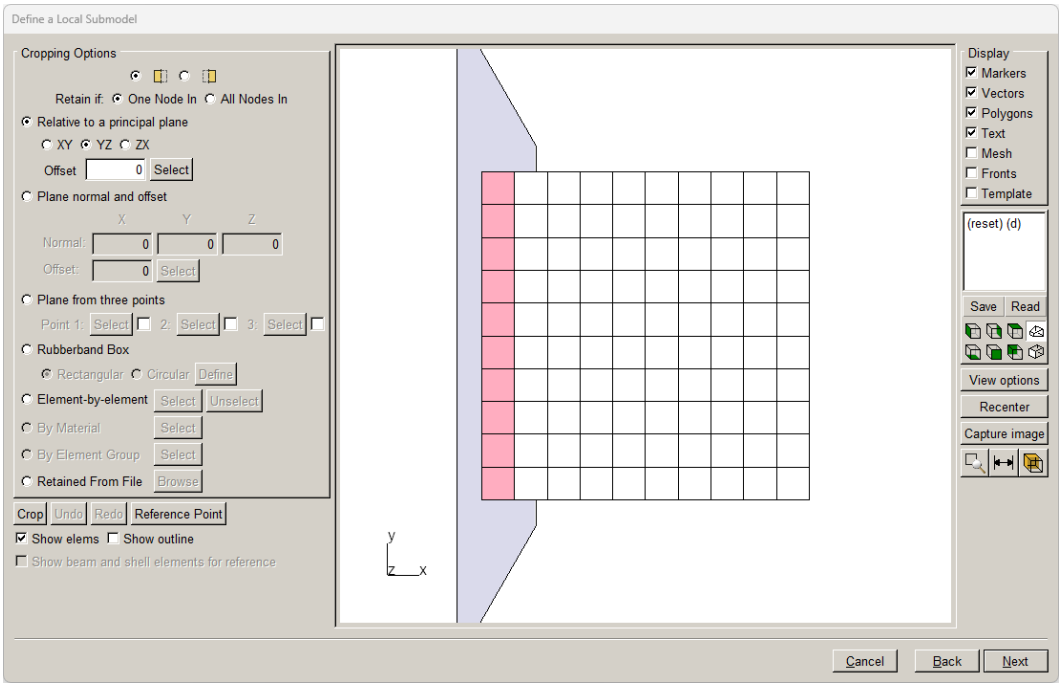


Figure 2.13 Define a Local Submodel Window

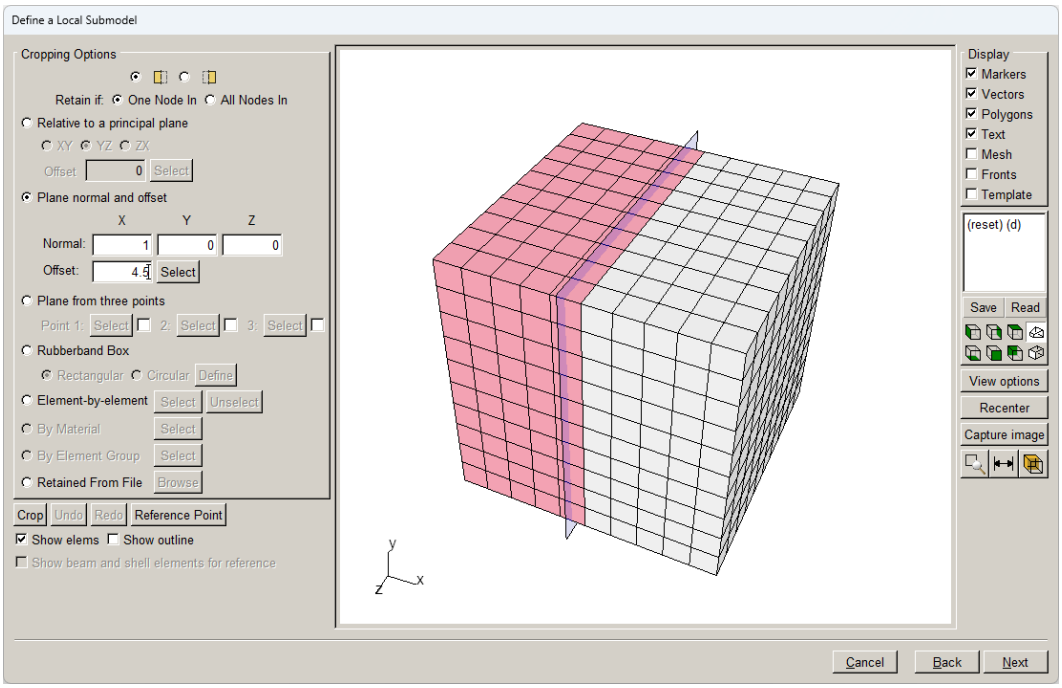


Figure 2.14 Plane Cutting tool – **Crop** will retain red elements.

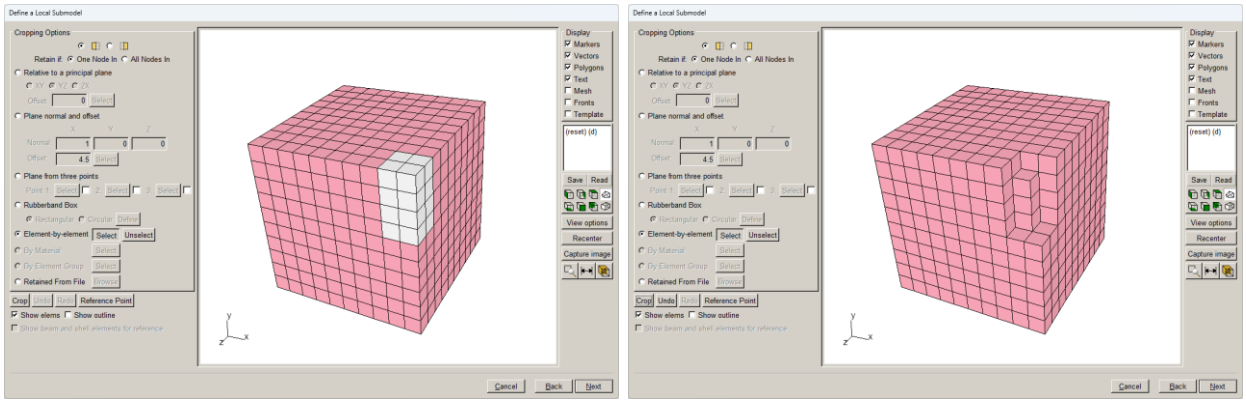


Figure 2.15 Element-by-Element cropping by selecting elements to remove.

We are inserting a surface crack that is normal to the y-axis (loading direction) and located on the +z surface, and the selection in Fig 2.16 is designed for this. Use the Rectangular option of the Rubberband Box to select and **Crop** a small portion of the model.

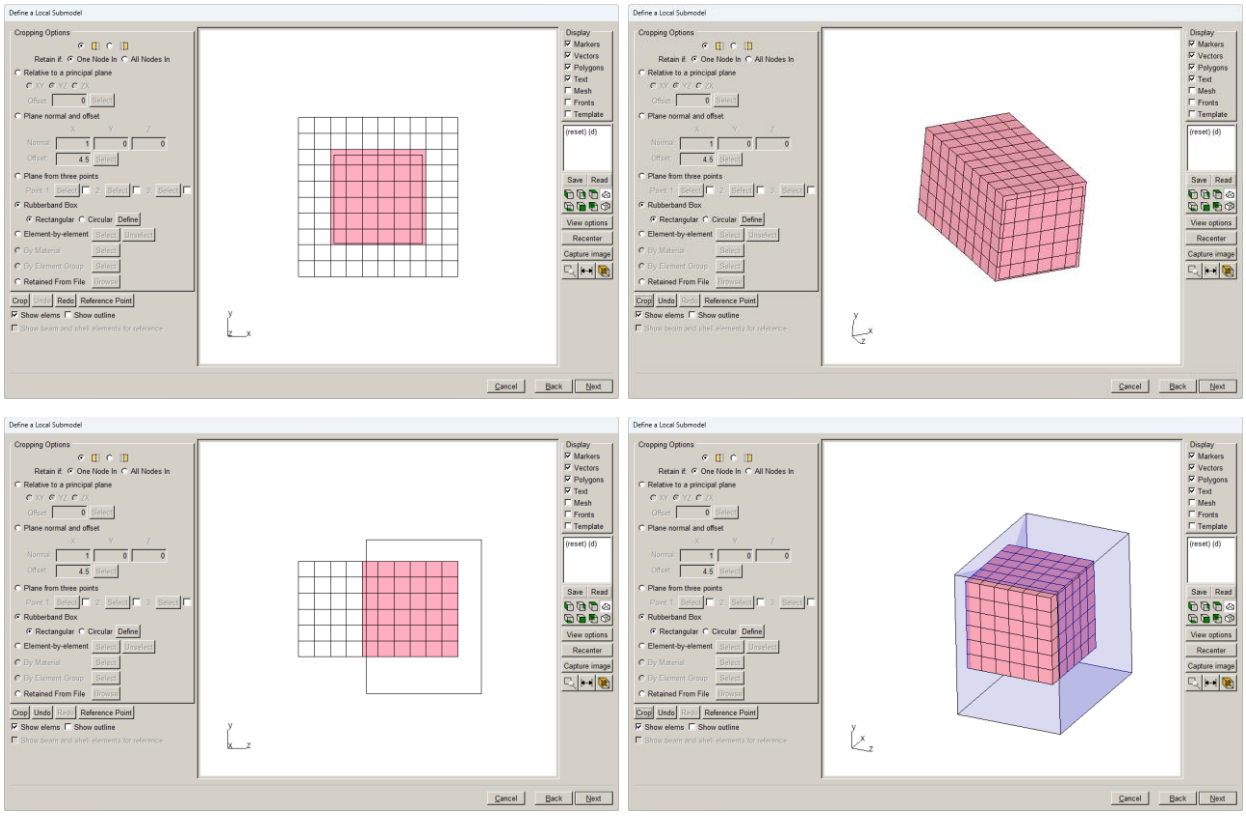


Figure 2.16 Rectangular Rubberband Box tool used to select a submodel.

Once the elements have been selected and cropped, select **Next**. The Save Files dialog appears, Fig 2.17. Choose the names of the local and global models and their location; we use the default file names and the working directory that was set earlier. Select **Next**.

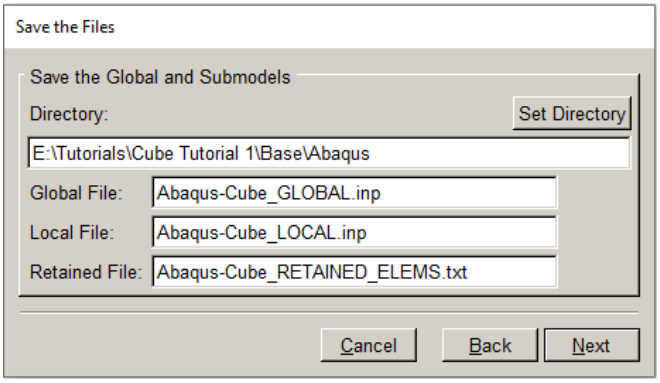


Figure 2.17 Local/Global model save dialog.

The Select Retained BC Surfaces dialog is displayed next, Fig 2.18, and shows the retained cut-surface. The cropped selection in Fig 2.16 avoids all surfaces with boundary conditions. Click **Next** to proceed.

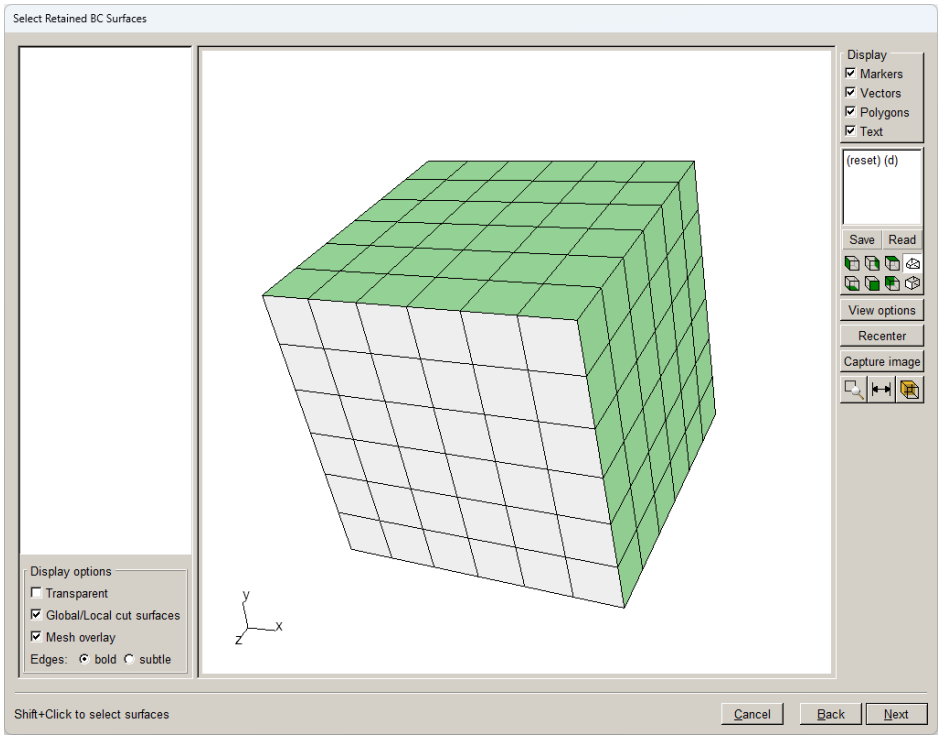


Figure 2.18 Select Retained BC Surfaces wizard panel – showing the retained cut-surface.

The model units, Fig 2.19, must be set. As defined in Step 1, the units for length are mm and the units for stress are MPa. The temperature and time units are not needed for this tutorial.

Select **Finish** and the local model is displayed in the main FRANC3D window, Fig 2.20. The cut-surface mesh facets are retained, while the front surface where the crack will be inserted does not have retained facets.

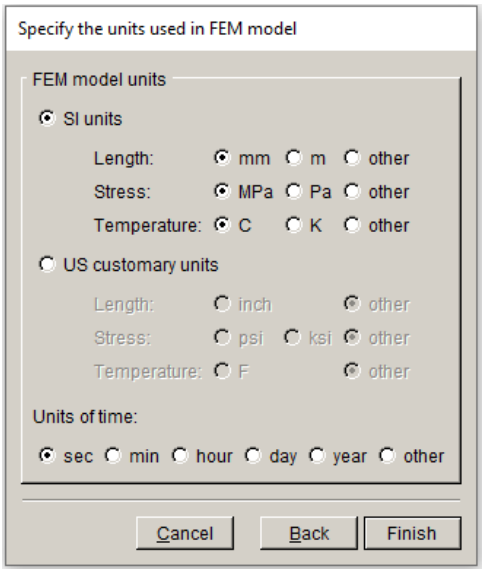


Figure 2.19 FE Model units

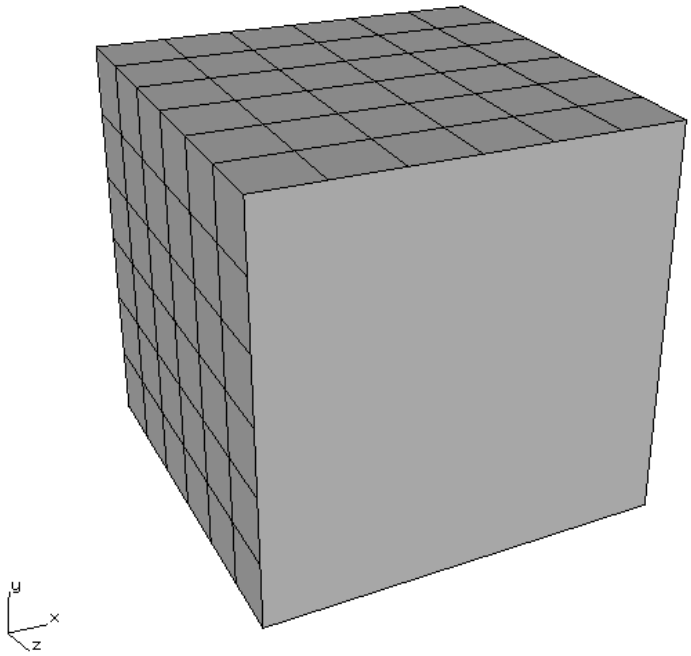


Figure 2.20 Local model with retained mesh facets on the cut-surface.

2.4 Step 4: Insert a Crack

We now insert a half-penny surface crack into the model. The local submodel from Section 2.3 is used, as opposed to the full model from Section 2.2. In Step 4.1, we describe how to define a new crack, and in Step 4.2, we describe how to insert a flaw from a file.

Note: if you want to try both types of crack insertion, you will have to re-import the uncracked model to insert a crack from a file; inserting a crack from a file will be used in subsequent steps and other tutorials.

Step 4.1: Define a new Crack from FRANC3D Menu

From the FRANC3D menu, select **Cracks →New Flaw Wizard,** Fig 2.21. The first panel of the wizard, Fig 2.22, allows you to select the flaw type; a crack with zero volume is the default. We select the Save to file and add flaw radio button so that the *.crk* file is saved for subsequent

tutorials. Note that the Symmetry surface crack option is only displayed if symmetry cracks are turned on in the Preferences (see Section 5.2 in the Reference).

Select **Next**. The second panel, Fig 2.23, lets you choose the crack shape; the default is the ellipse, which is what we want. Select **Next**. The crack is a circle with radius=1, centered on the cube's front (+z) face, and parallel to the xz-plane (normal to y).

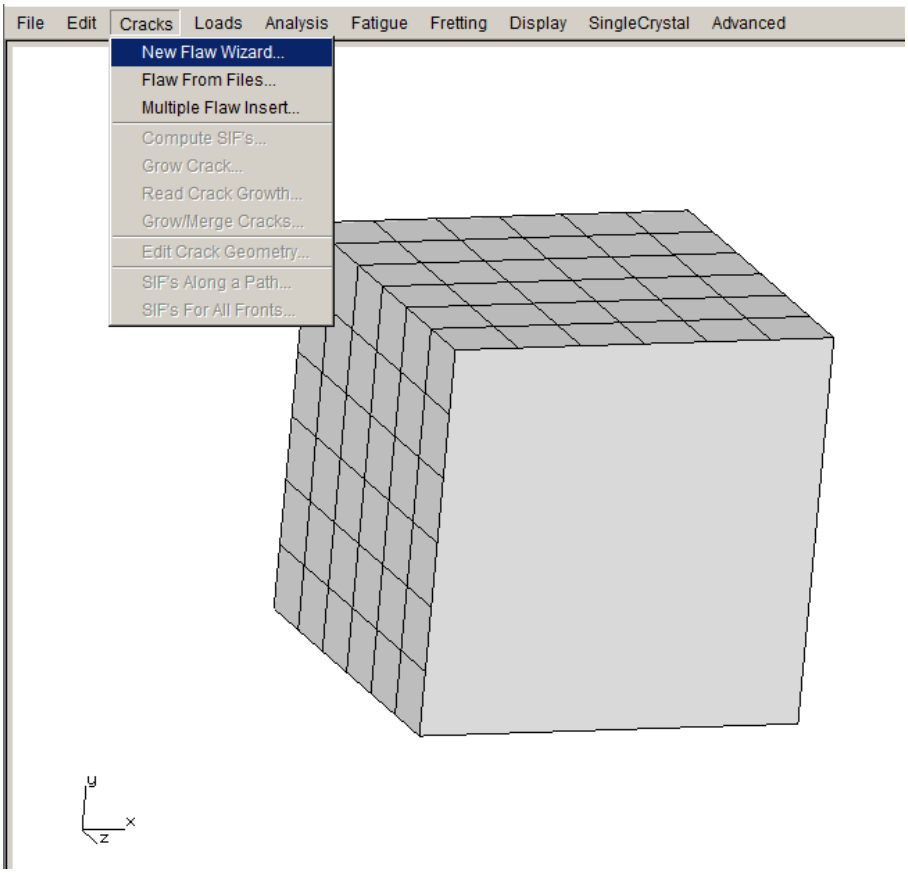


Figure 2.21 New Flaw Wizard menu item selected.

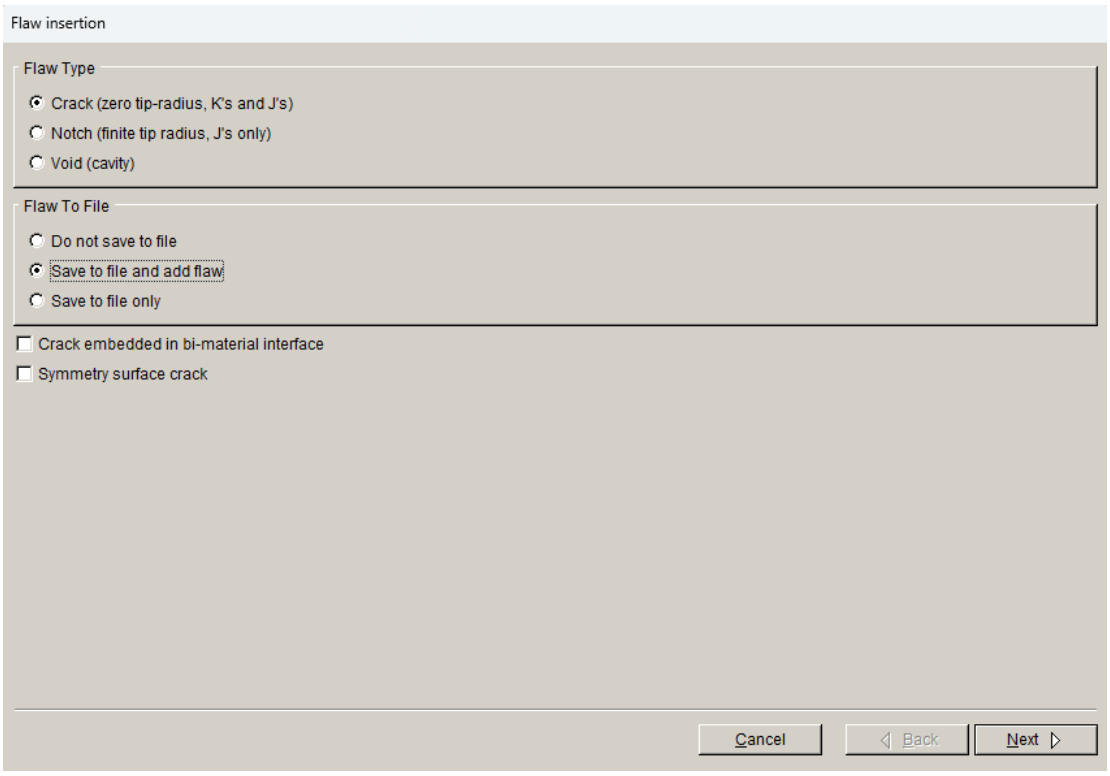


Figure 2.22 New Flaw Wizard first panel – Crack (zero volume flaw) selected.

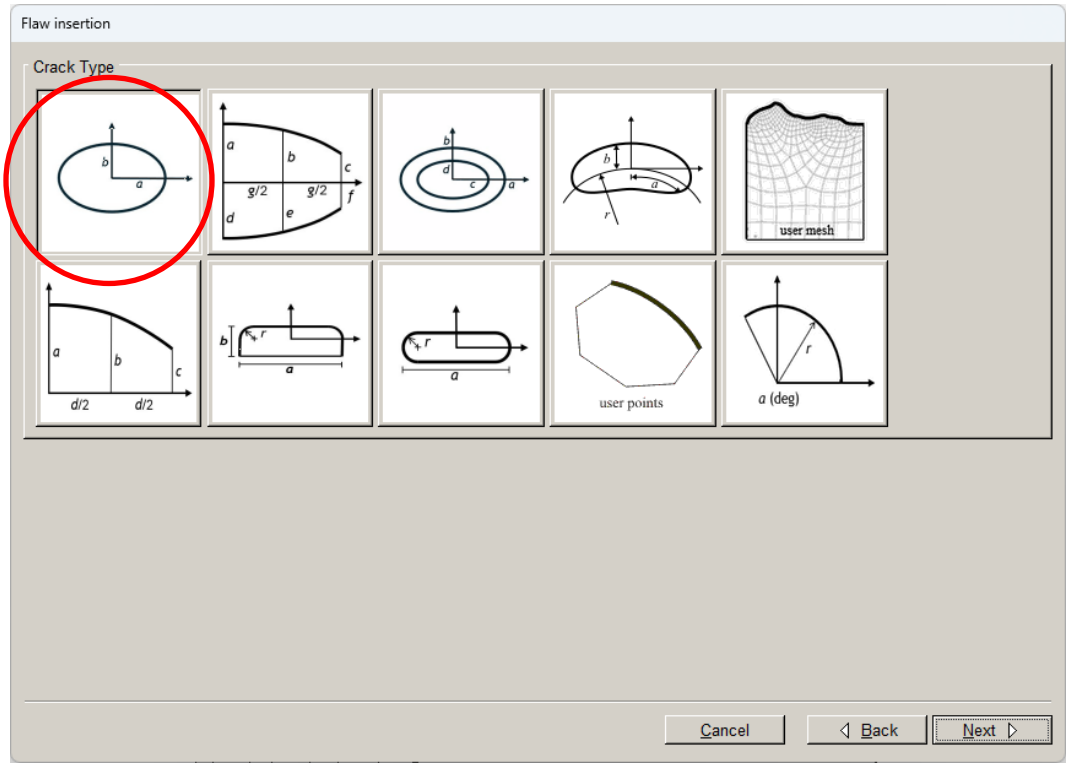


Figure 2.23 New Flaw Wizard second panel – choose ellipse shape.

The third wizard panel lets you set the dimensions for the ellipse, Fig 2.24; set a=1 and b=1. Select **Next**.

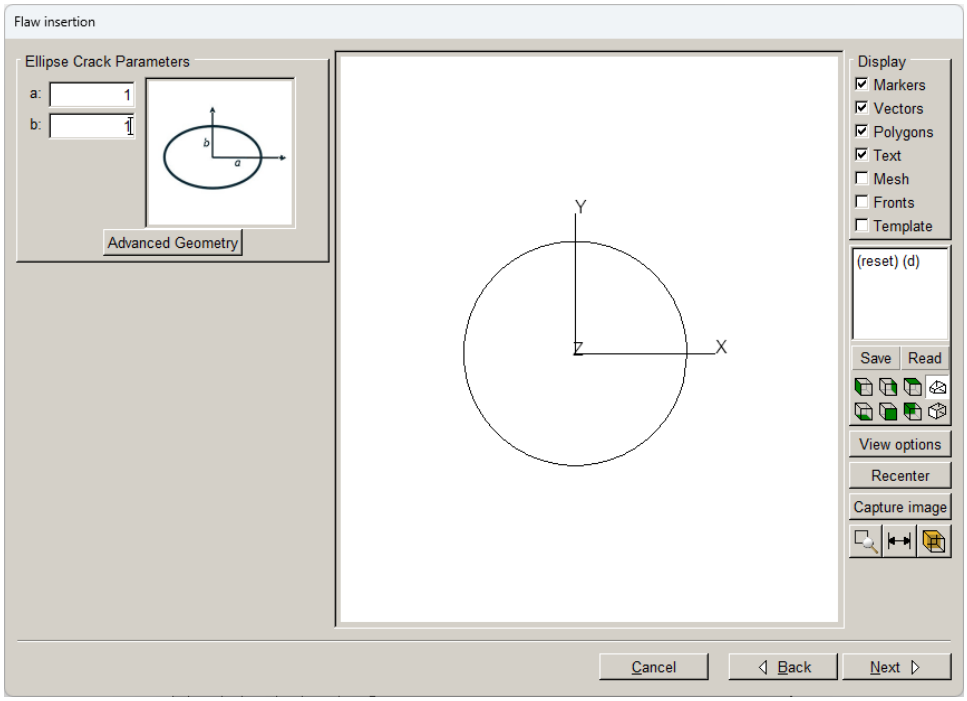


Figure 2.24 New Flaw Wizard third panel – set ellipse dimensions.

The fourth panel lets you set the crack location and orientation, Fig 2.25. We place the crack at the center of the front face at coordinates (5,5,10) and rotate the crack 90 degrees about the Global x-axis. Select **Next**.

The final panel allows you to set the crack front mesh template parameters, Fig 2.26; use the defaults. In practice, the default template radius is based on the crack dimensions and might need to be adjusted depending on the model and crack. Select **Finish**.

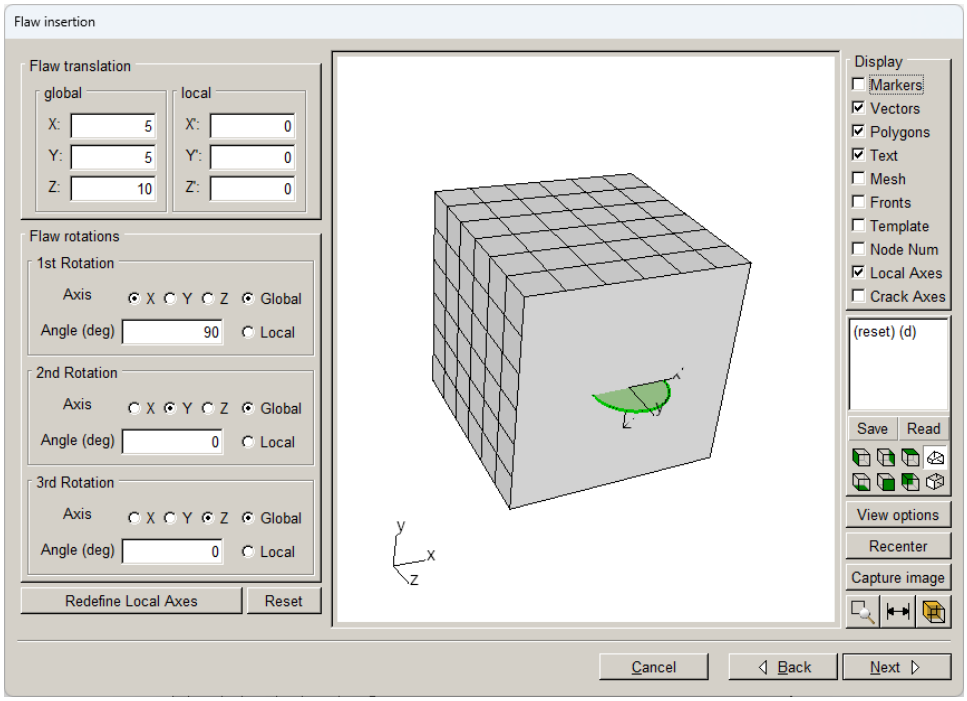


Figure 2.25 New Flaw Wizard fourth panel – set crack location and orientation.

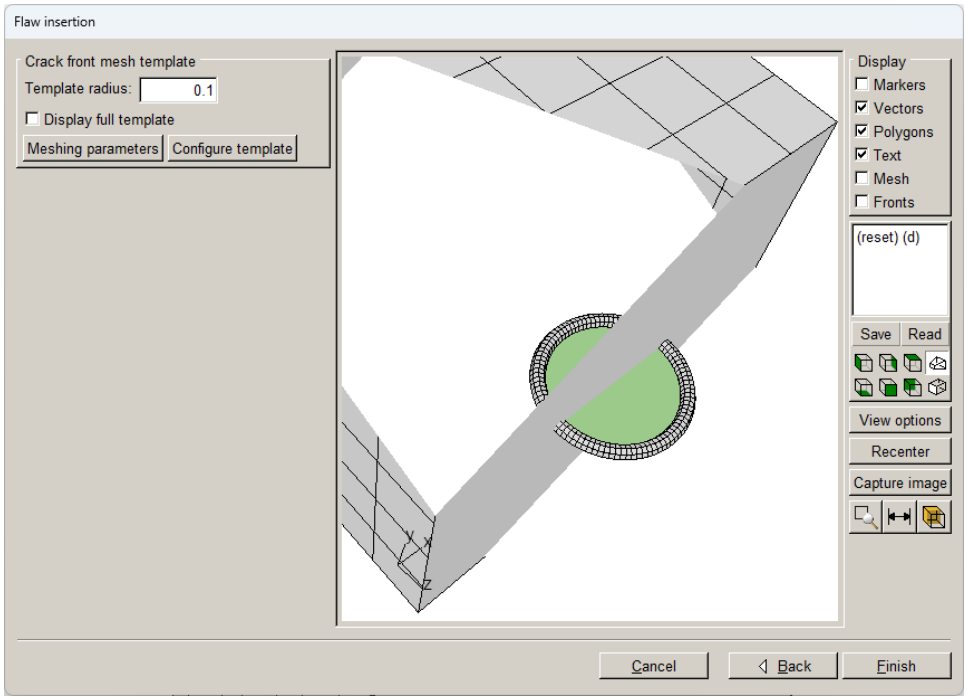


Figure 2.26 New Flaw Wizard final panel – set template mesh parameters.

At this point, you will be asked to specify the *.crk* file name, Fig 2.27; we call it *Cube_Crack.crk*. Select **Accept**.

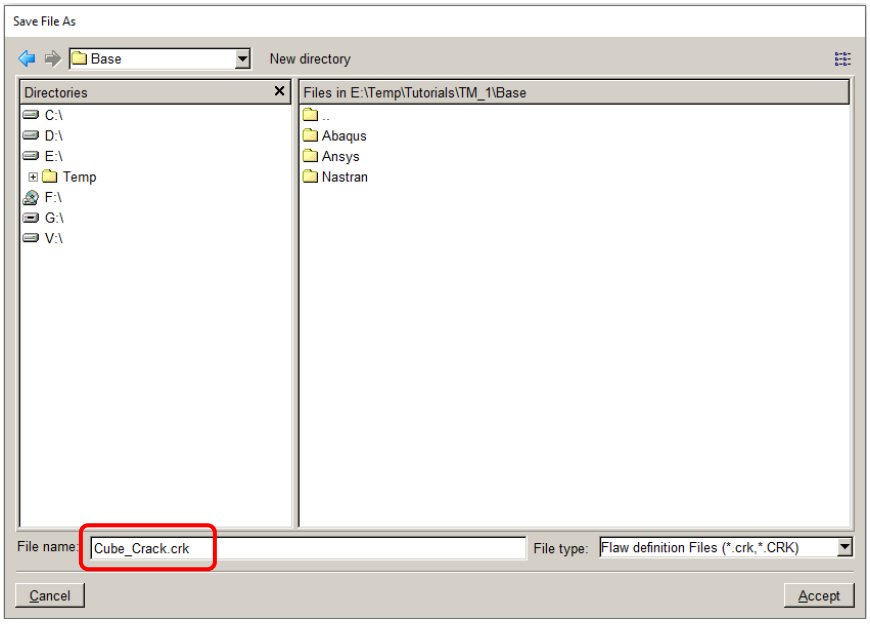


Figure 2.27 New Flaw Wizard final panel – save file as.

The crack is inserted into the model and remeshing occurs. A **Flaw Insertion Status** dialog is displayed during this process, Fig 2.28. There are three stages: geometric intersection of the crack with the model, surface meshing and volume meshing. The resulting model is displayed in the main FRANC3D window, Fig 2.29; the surface mesh is turned on in this image.

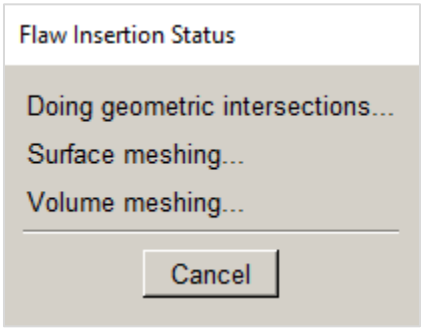


Figure 2.28 Flaw Insertion Status window.

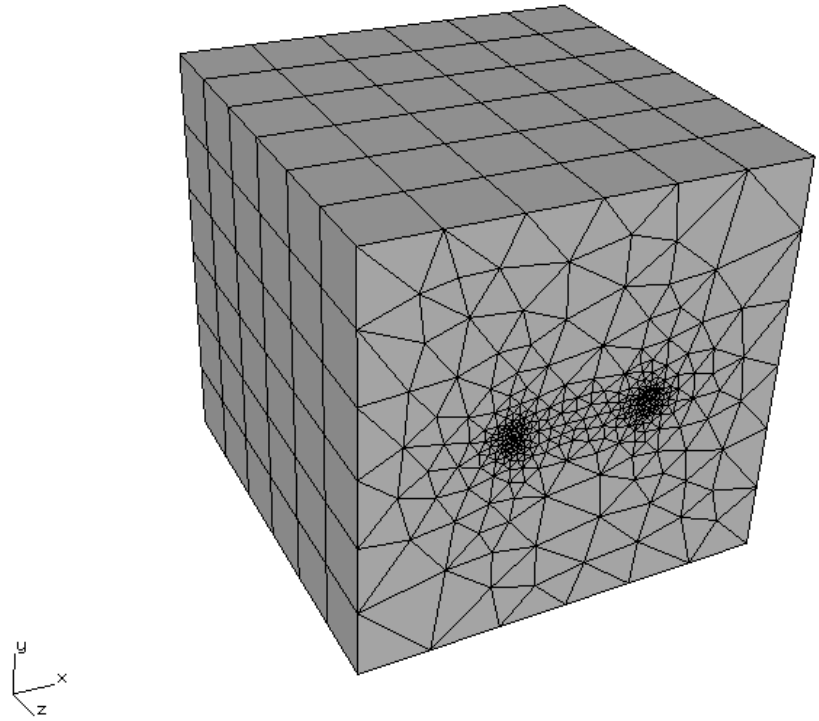


Figure 2.29 Remeshed cracked local model with surface mesh turned on.

Step 4.2: Insert Cracks from Files

If you want to try this step, you must **Close** the model from Step 4.1 and re-import the original uncracked (local) model.

From the FRANC3D menu, select **Cracks →Flaw From Files,** Fig 2.30. The first panel, Fig 2.31, lets you select the *.crk* file. Choose the *Cube_Crack.crk* file name and select **Accept**. This crack was created using the **New Flaw** Wizard with the save to file option in Step 4.1. The flaw being added is a circle with radius=1, centered on the cube's front face, and parallel to the xz-plane.

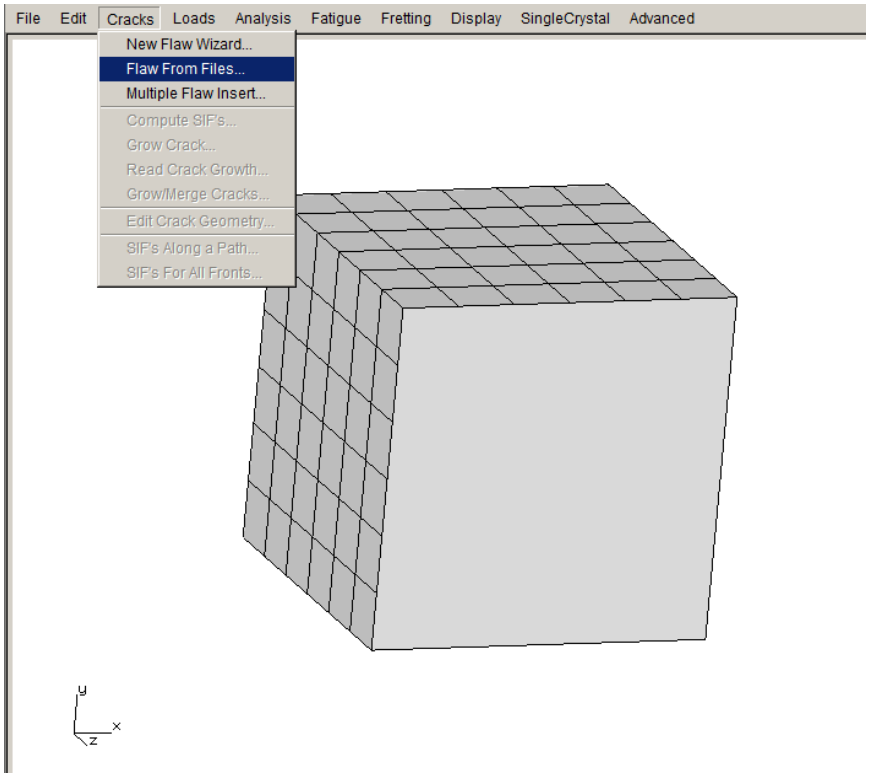


Figure 2.30 Flaw From Files option in Crack menu.

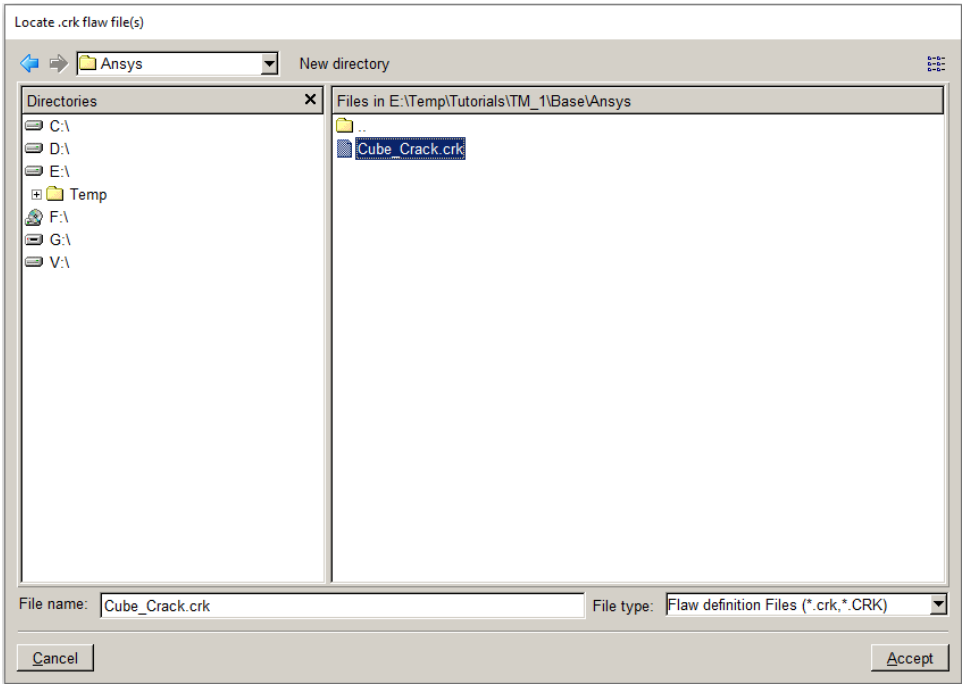


Figure 2.31 Flaw from file dialog to select .crk file.

The next panel, Fig 2.32, allows you to adjust the location of the flaw. The orientation cannot be changed. The final panel, Fig 2.33, allows you to set the template mesh parameters. Use the default settings and select **Finish** to complete the insertion and remeshing.

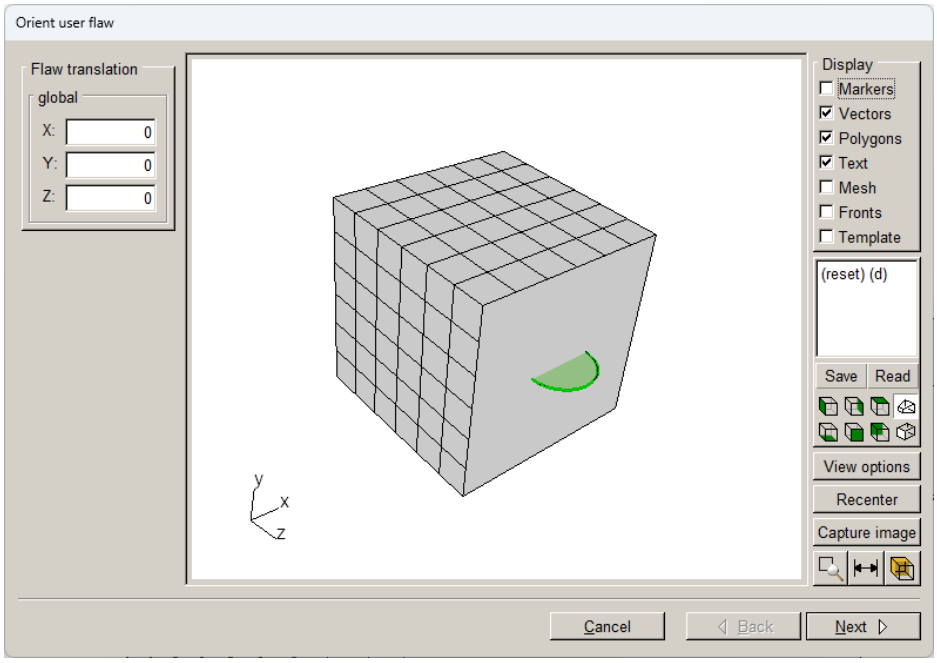


Figure 2.32 Flaw wizard panel to set location.

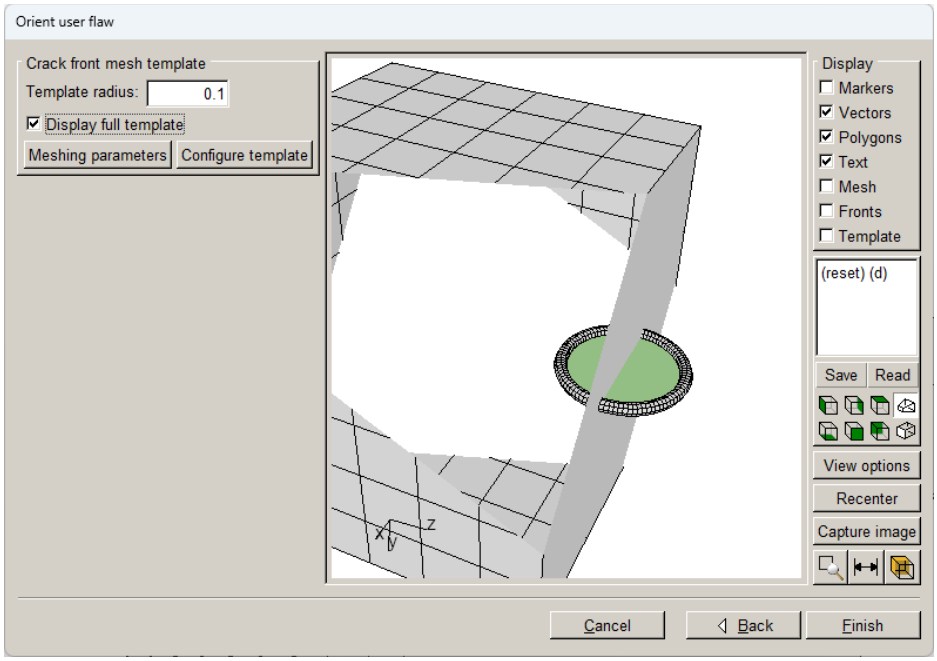


Figure 2.33 Flaw geometry shown in the model with crack template mesh.

2.5 Step 5: Static Crack Analysis

Once the crack is inserted and the model is remeshed, we perform a stress analysis using ABAQUS, which provides the displacement results that are needed to compute stress intensity factors (SIFs). Typically, you would run a "static crack analysis" of the initial crack prior to running automated crack growth; this allows you to verify that the crack model is consistent with the uncracked model. You can compare displacement contours, using the ABAQUS post-processor, for the original uncracked and cracked models to verify that the crack model is valid.

You might also insert the crack with a bigger or smaller template radius to verify that the computed SIFs are accurate; there should be no notable change in SIFs for a reasonable template.

Step 5.1: Select Static Crack Analysis

From the FRANC3D menu, select **Analysis →Static Crack Analysis**, Fig 2.34. The first panel, Fig 2.35, requests that you enter the file name for the FRANC3D data. We call it *Abaqus_Cube_Subdivide.fdb*; select **Next** once you enter a File Name.

Note that you cannot use the initial uncracked *Abaqus_Cube.inp* or *Abaqus_Cube_LOCAL.inp* file names. A new *.inp* file will be created during the analysis, and the original uncracked *.inp* files are reused for each step of crack growth.

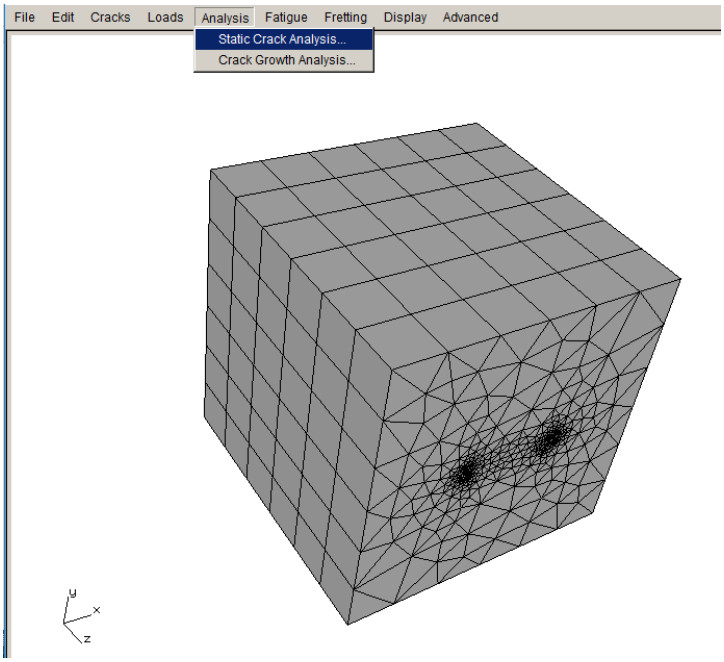


Figure 2.34 Static Crack Analysis menu

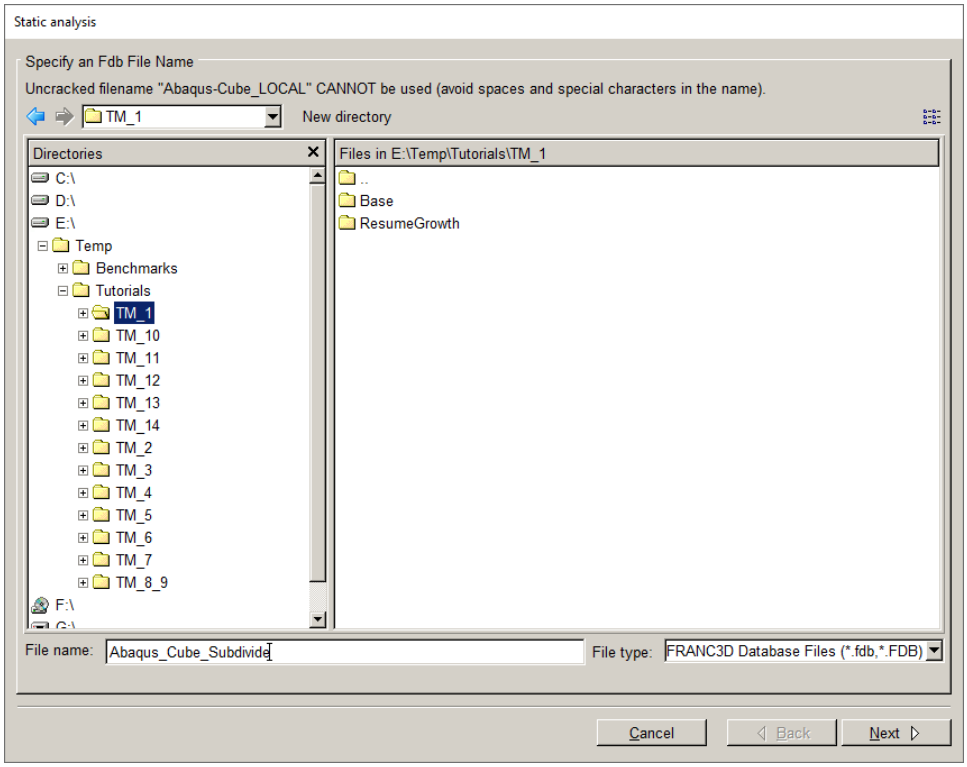


Figure 2.35 Static Analysis wizard first panel – File Name

Step 5.2: Select ABAQUS Analysis Options

The next panel, Fig 2.36, allows you to specify the ABAQUS output and analysis options. The model is a sub-model and must be connected to the global model, so the Connect to global model box is checked automatically if the FRANC3D submodeler tool was used. The next panel, Fig 2.37, allows you to set the local + global model connection. If the model being analyzed is a full/complete model, the panel in Fig 2.36 is the final panel as there is no global model, and the **Next** button will be a **Finish** button.

Fig 2.37 shows the options for joining the local and global models. AUTO_CUT_SURF and GLOBAL_CONNECT_SURF are the node sets created by the FRANC3D submodeling tool and should be selected automatically. Click **Finish** to start ABAQUS running (in batch/background mode). FRANC3D writes files and then attempts to execute ABAQUS based on the ABAQUS Executable information (Fig 2.36). You can monitor the FRANC3D CMD window and the ABAQUS *.dat* and *.msg* files for errors or messages.

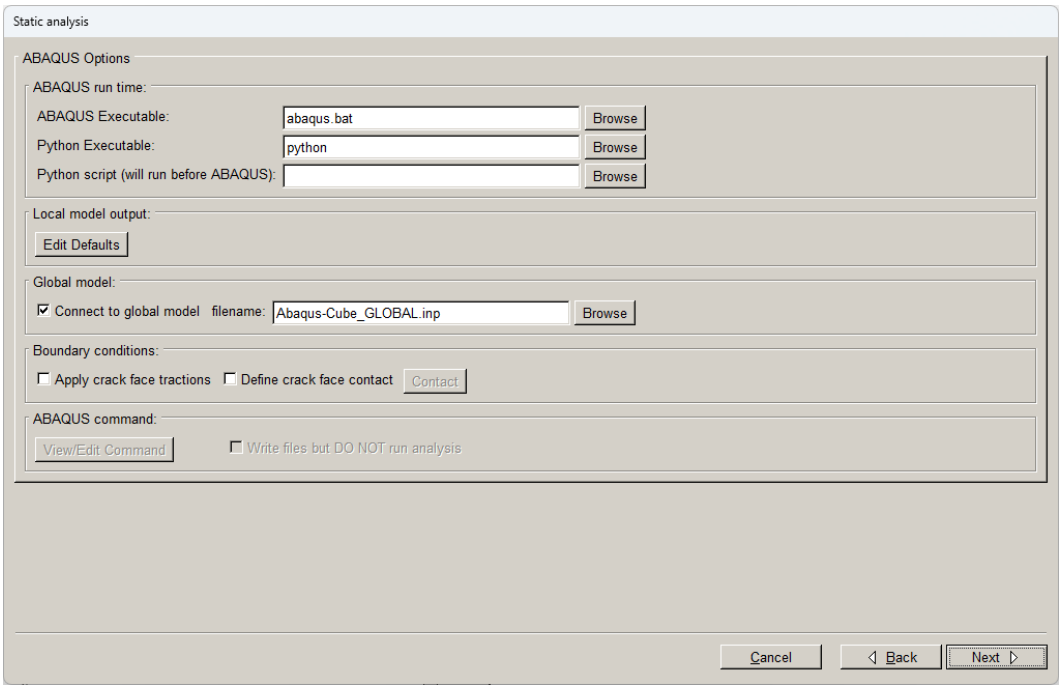


Figure 2.36 Static Analysis wizard third panel – ABAQUS output options

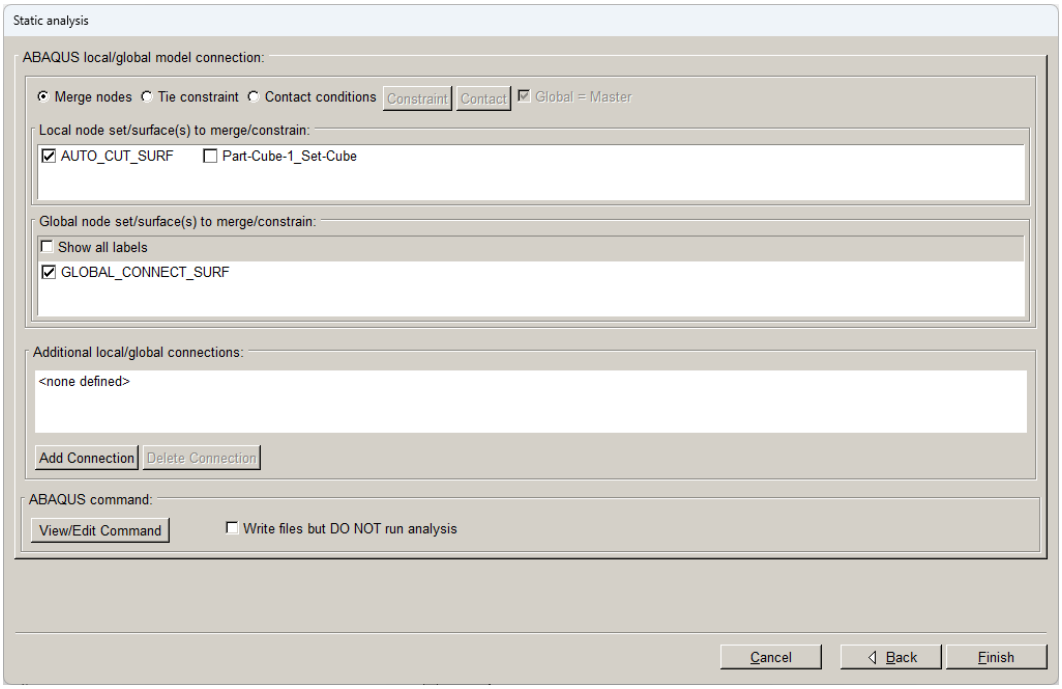


Figure 2.37 Local/Global model connection

Note that if ABAQUS does not run, you will see a message in FRANC3D indicating that SIFs cannot be computed because there are no displacements, or you might see a message indicating that FRANC3D cannot find or open the *.dtp* file. You can review the ABAQUS files for errors. The ABAQUS executable (and path to it) will depend on the installation and operating system. For MS Windows, for a typical installation, the executable is "abaqus.bat" and is in the folder "C:\SIMULIA\Commands".

Choosing Write files but DO NOT run analysis in Fig 2.37 will create all the files without running ABAQUS, if the analysis needs to be run later or on a different computer.

If you need to run ABAQUS on a different computer, you must bring the results file (*.dtp*) back to the working folder, and then you can use the **File →Read Results** menu option to import the results into FRANC3D. The *.dtp* file is created by running the *writeDtpFile.py* script, which FRANC3D writes; the script extracts data from the *.odb* file. Note that the 2024 ABAQUS version uses Python 3, whereas older versions used Python 2; the ABAQUS version must be set in the FRANC3D preferences.

In a CMD or Terminal window, the command is:

> abaqus python writeDtpFile.py

2.6 Step 6: Compute Stress Intensity Factors

When the ABAQUS analysis is done (and the *.dtp* file read back into FRANC3D), we can compute SIFs. From the FRANC3D menu, select **Cracks →Compute SIFs**, Fig 2.38. The Stress Intensity Factor dialog is displayed, Fig 2.39. You should use the M-integral, but you can check that the Displacement Correlation results are similar. Select **Accept** and the SIFs Plot window is displayed, Fig 2.41. You can view the three modes of stress intensity factors (SIFs) using the upper tabs: KI, KII and KIII; mode II and III values should be negligible compared to mode I.

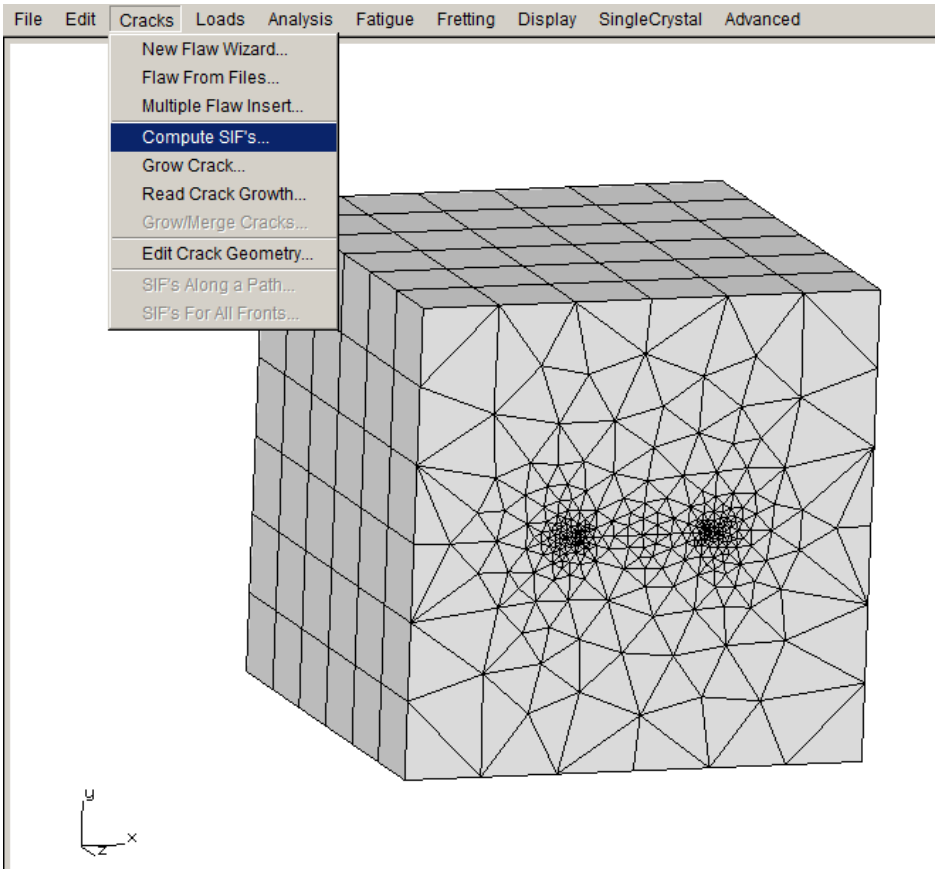


Figure 2.38 Compute SIFs selected from the Cracks menu.

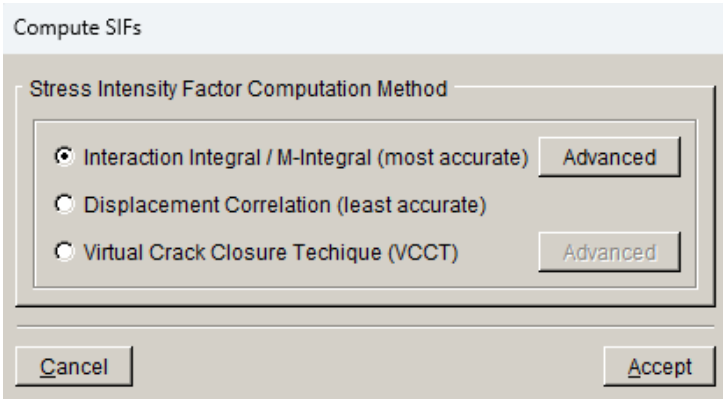


Figure 2.39 Compute SIFs dialog.

The units for SIFs are MPa√mm. The SIFs are plotted versus the normalized crack front length, starting from point A – see the image in the left pane of Fig 2.40. SIFs are computed at element midside nodes along the front, so the values do not start at exactly zero.

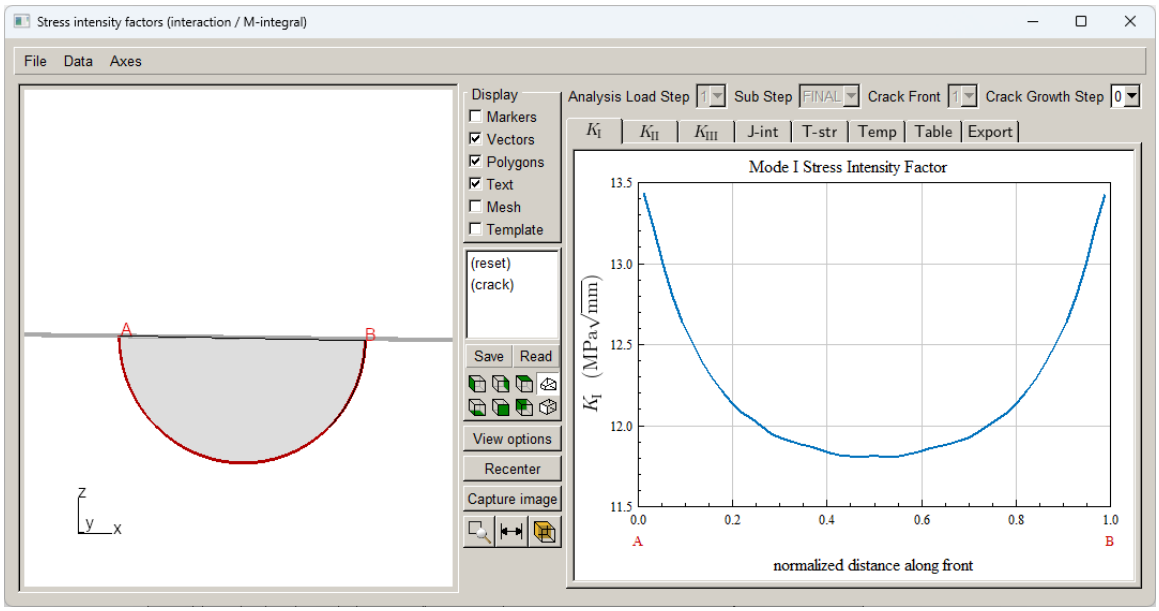


Figure 2.40 Stress Intensity Factor display

2.7 Step 7: Manual Crack Growth

We will "manually" propagate the crack front; this lets us examine the predicted crack growth to determine suitable parameters for fitting and extrapolation before proceeding with automated crack growth, which is described in Section 2.8.

Step 7.1: Select Grow Crack

From the FRANC3D menu, select **Cracks →Grow Crack**, Fig 2.41. In the first panel, Fig 2.42, switch to Quasi-Static crack growth and select **Next**. Use the Max Tensile Stress theory to determine the kink angle, Fig 2.43; in this model, crack growth is mostly planar. Select **Next**. Use a Power Law relationship to determine relative growth among points along the crack front, Fig 2.44; we will use the default power n=2. There is only one load step in the model. Select **Finish** to proceed. You will be asked if you want to save the growth parameters to a file, Fig 2.45, you can select **No**.

This quasi-static growth model can be considered a simplified Paris model; it is simpler to use than the subcritical growth model. Consult the Reference document for descriptions of crack extension models.

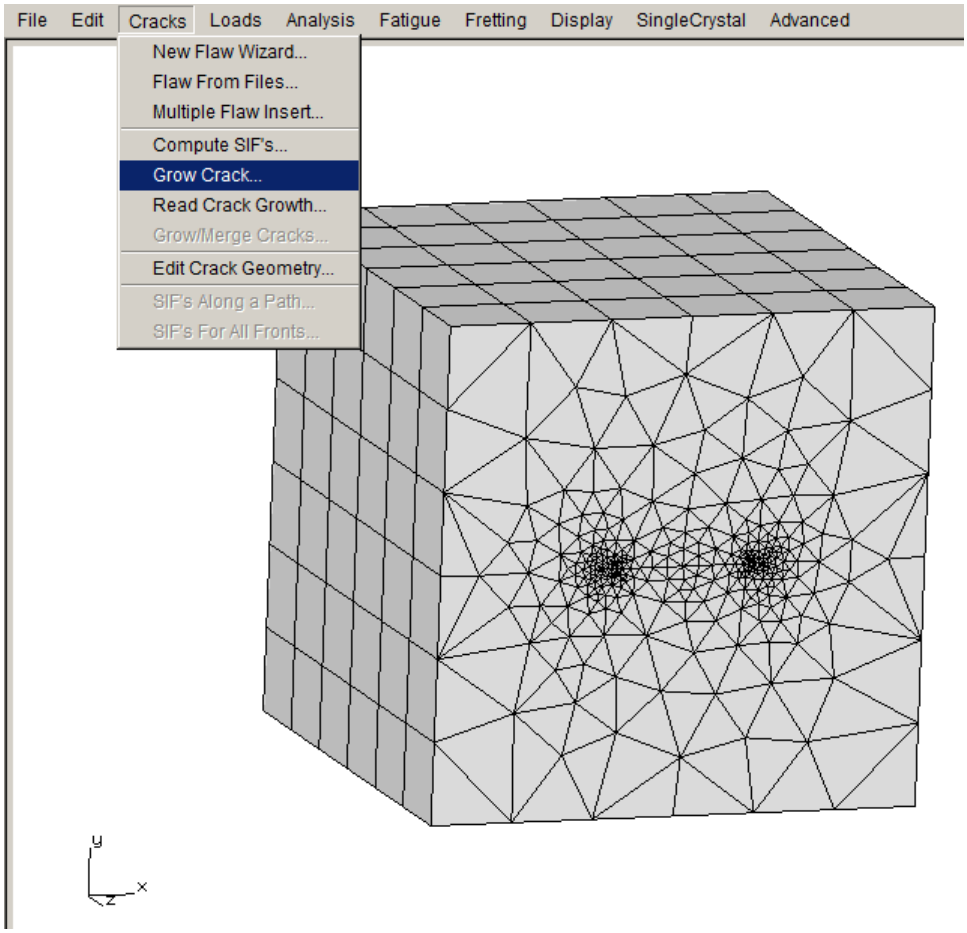


Figure 2.41 Grow Crack menu option.

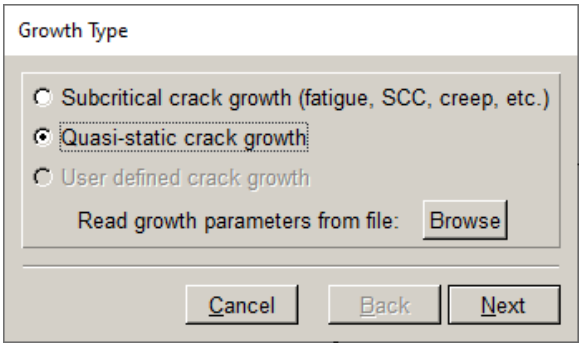


Figure 2.42 Crack Growth wizard – first panel.

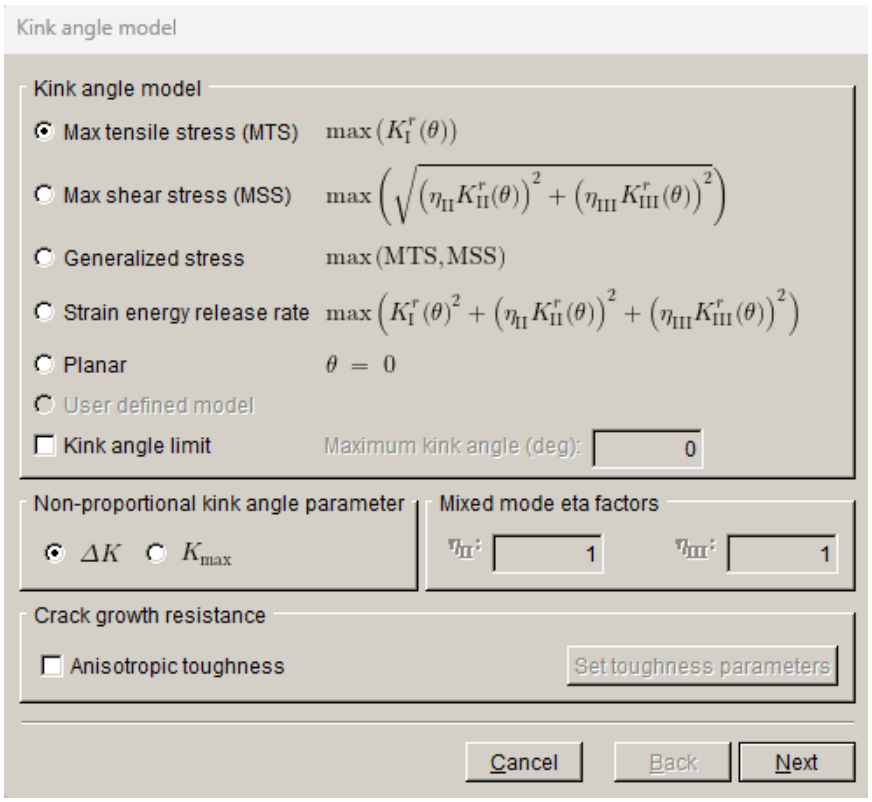


Figure 2.43 Crack Growth wizard – second panel using quasi-static growth.

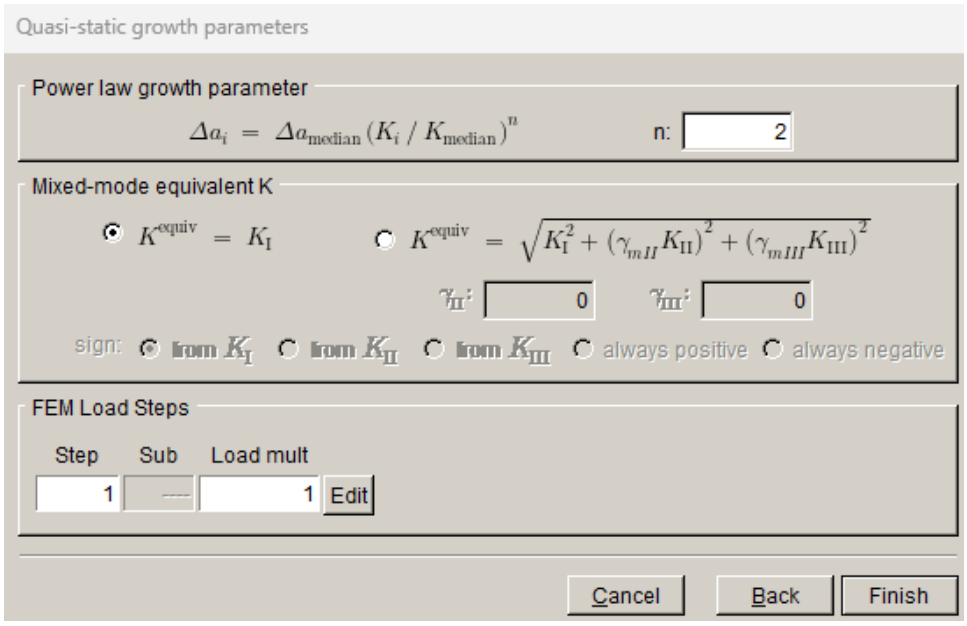


Figure 2.44 Crack Growth wizard – third panel using quasi-static growth.

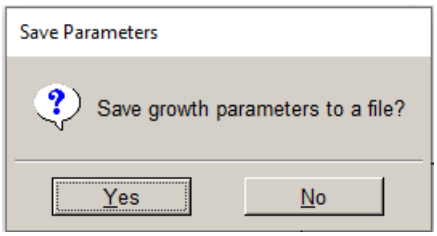


Figure 2.45 Save growth parameters dialog.

Step 7.2: Specify Fitting and Extrapolation

The next panel, Fig 2.46, allows you to specify the crack front point fitting parameters. You can double click on the (crack) view to see the crack surface; it should be the default view.

The median extension is set to 0.15; this value usually is set automatically based on the initial template radius, so your value might be different. We change it to 0.12 and turn on Mesh and Markers (top right checkbox in Fig 2.46). The green boxes represent the computed new front points, and the blue line is the curve-fit, Fig 2.47. Note that the scale node is 0.5; the computed extensions are sorted from minimum to maximum, and the scale node can vary from 0 to 1. For a scale node of 0.5, the median extension is set to the user-specified value, and all other extensions are scaled accordingly.

A Fixed order polynomial with order set to 3 and extrapolation set to 3% at both ends of the crack front is the default; the fitting parameters might need to be adjusted as the crack grows. A 4^{th} order polynomial might provide a slightly better fit in this case; the guideline for curve-fitting is to use the simplest fit that is reasonable. The extrapolation is automatically adjusted to make sure the end points of the fitted-front fall outside the model surface. We stick with the default 3^{rd} order polynomial; select **Next** to proceed.

Note that the fitted (blue) curve must be extrapolated so that both ends fall outside of the model, but it should not be extrapolated too much. Normally, FRANC3D will automatically find reasonable values for extrapolation if Allow fit adjustment is turned on.

The median extension must be enough to provide finite space between the current and new fronts along the entire front to define new crack geometry. It is best to specify the extension (and have cycles computed) as opposed to specifying the cycles and having extension computed.

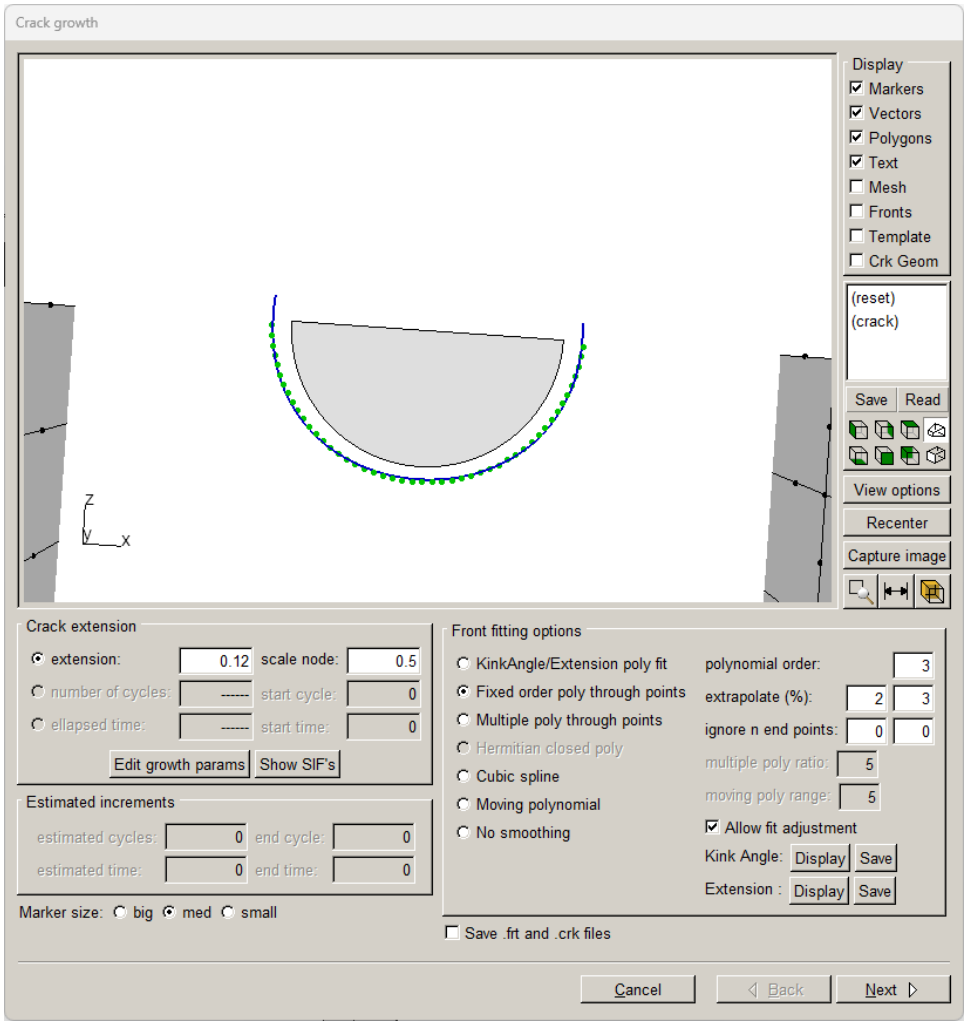


Figure 2.46 Crack growth wizard panel for crack front fitting options

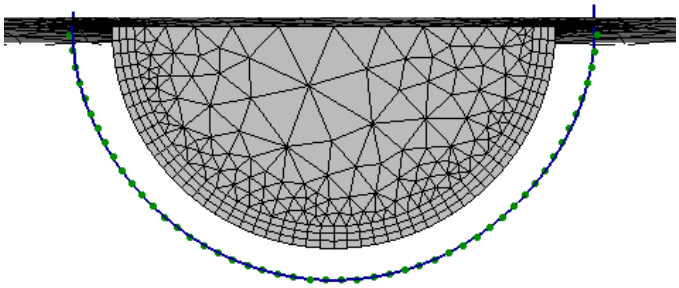


Figure 2.47 Curve fit (blue) through front (green) points.

Step 7.3: Specify Crack Front Template

The final panel, Fig 2.48, allows you to specify the crack front mesh template parameters. The crack front mesh template extends beyond the model surface, which corresponds with the front-fit extrapolation. The template radius can be set as an absolute value or as a % of the median extension; use an absolute value of 0.1, which is the same as the initial crack template radius. Select **Next** on this panel when ready to proceed with crack insertion and remeshing.

Note that the crack geometry for the extended crack is inserted into the original uncracked model, so do not overwrite, or remove the original uncracked model files.

A template radius that is slightly less than the crack extension, Fig 2.49, works best for cases where there is kinking or turning of the crack front.

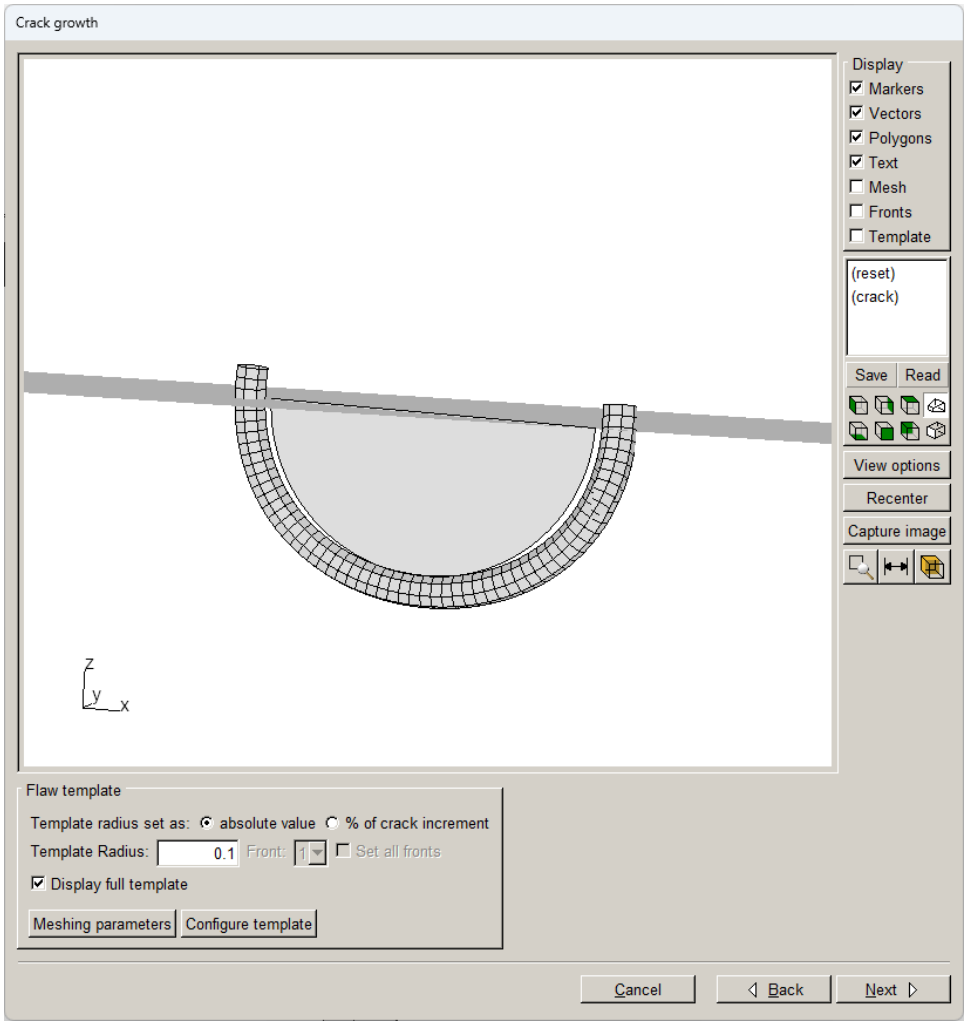


Figure 2.48 Crack growth wizard panel for mesh template options

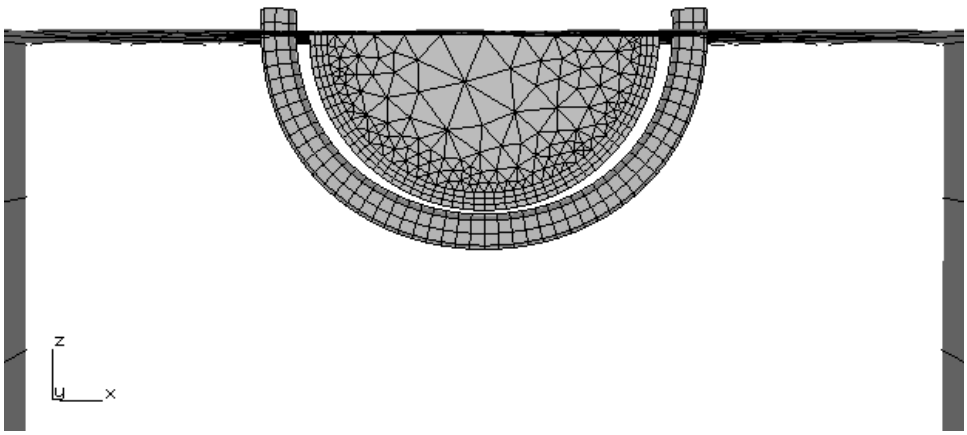


Figure 2.49 Template radius is smaller than extension.

The resulting model can be analyzed as was done for the initial crack (see Step 5 above). Note that you will want to give this model a different name, such as *Abaqus_Cube_ step_001*, so that you do not overwrite the initial crack files. You could continue with another manual step of growth followed by static analysis, or you can try automated crack growth, which is described next.

2.8 Step 8: Automatic Crack Growth

This section describes the steps for automatic crack growth starting from the initial crack model. We start with the existing FRANC3D model, using the model created in Sections 2.4 and 2.5. In most cases, we would simply do automatic growth after the manual step described in Section 2.7, but this will not show all the dialogs, so we restart from the initial crack model.

Step 8.1: Open FRANC3D Restart File

Start with the FRANC3D graphical user interface (see Fig 1.1) and select **File →Open**. Select the file name specified in Section 2.5, called *Abaqus_Cube_Subdivide.fdb*. Select **Accept**. The model will be read into FRANC3D (along with the results that were created when running the static analysis). We will ignore the fact that we already propagated the initial crack in the previous step (but we will use the settings). The next section describes the steps for automatic crack growth analysis.

Step 8.2: Select Crack Growth Analysis

From the FRANC3D menu, select **Analysis →Crack Growth Analysis**, Fig 2.50. The first panel of the wizard, Fig 2.51, allows you to choose the method for computing SIFs. We use the default M-integral and select **Next**. If this panel does not show up, the SIF method is chosen automatically based on your earlier choice when plotting SIFs.

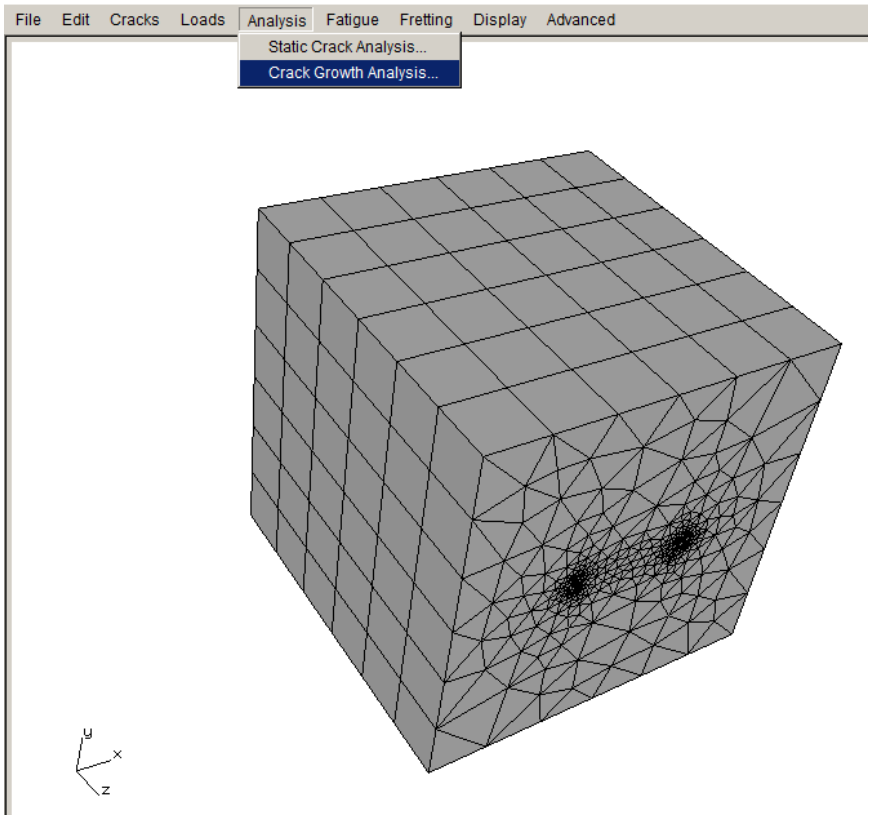


Figure 2.50 Crack Growth Analysis menu

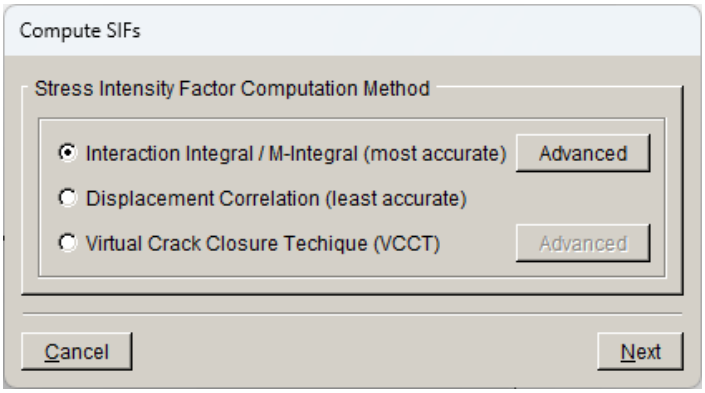


Figure 2.51 Crack Growth Analysis wizard – first panel.

Step 8.3: Specify Growth Rules

The second panel, Fig 2.52, lets us set the crack growth type; we use quasi-static growth. Select **Next** to display the third panel, Fig 2.53; use the default MTS theory to compute the kink angle. Select **Next** to display the fourth panel, Fig 2.54; use the default settings for the quasi-static growth; there is only one FEM load step to select. Select **Next**. You will be asked if you want to save the growth parameters to a file; you can select **No**.

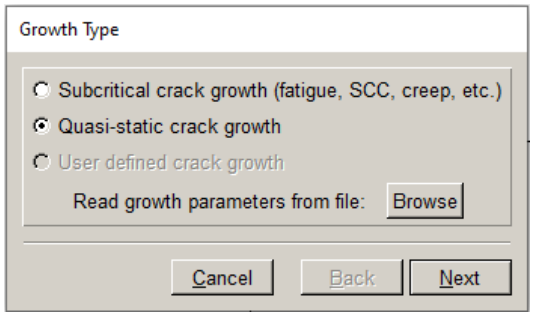


Figure 2.52 Crack Growth Analysis wizard – second panel.

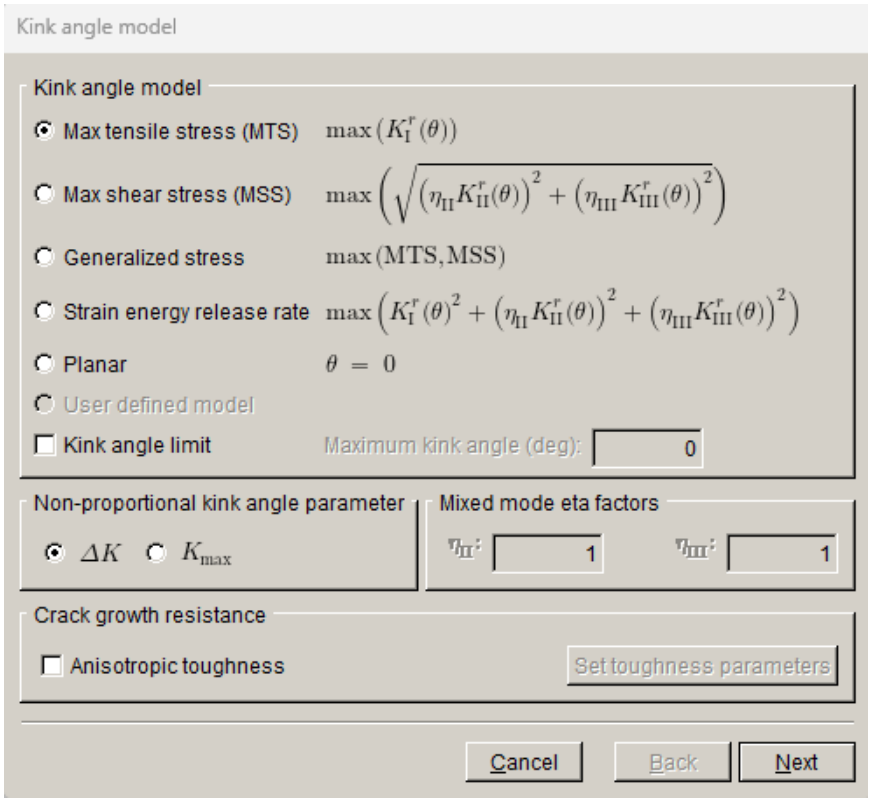


Figure 2.53 Crack Growth Analysis wizard – third panel.

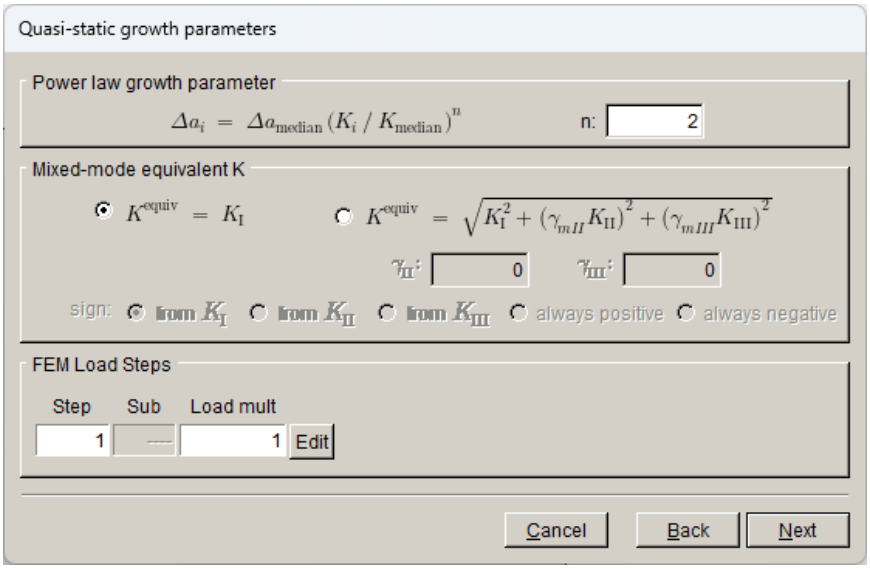


Figure 2.54 Crack Growth Analysis wizard – fourth panel.

Step 8.4: Specify Fitting and Template Parameters

The fifth panel, Fig 2.55, lets us set the crack front fitting and mesh template parameters. Use the default 3rd order polynomial and 2% extrapolation. Set the template radius to an absolute value of 0.1. Select **Next**.

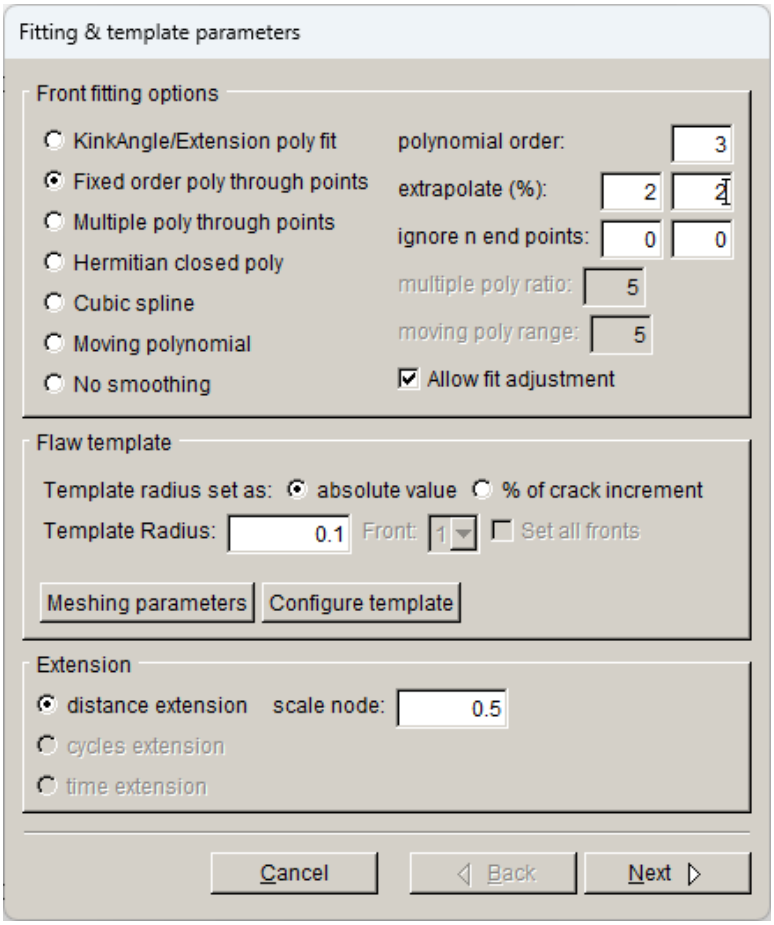


Figure 2.55 Crack Growth Analysis wizard – fifth panel.

Step 8.5: Specify Growth Plan

The sixth panel, Fig 2.56, lets us define the amount of extension for each step of crack growth. Set the number of steps to five and the constant median extension to 0.2. Select **Next**.

Note that we are taking a fairly large crack growth step; this is to limit the number of steps and time for this tutorial. Large steps can sometimes lead to issues with oscillations in the shape of the crack fronts (see Section 9 of the User's Guide), but for this model 0.2 is acceptable.

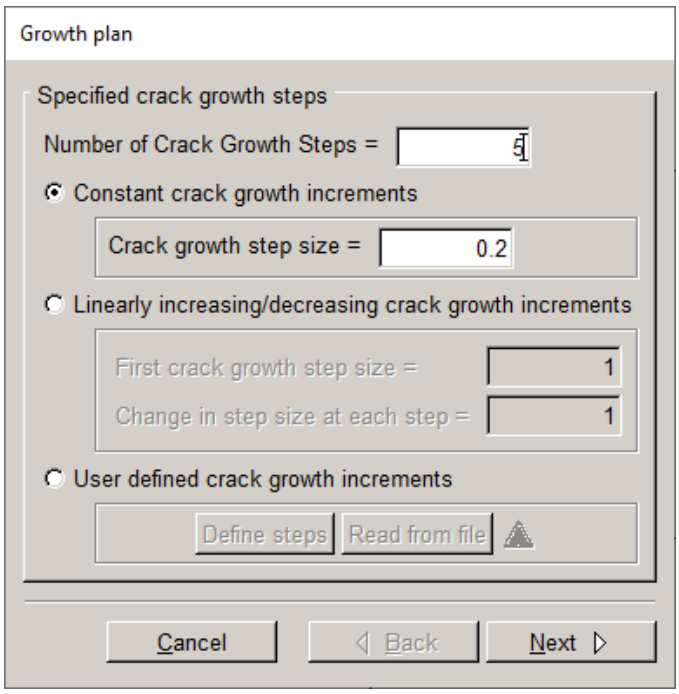


Figure 2.56 Crack Growth Analysis wizard – sixth panel.

Step 8.6: Analysis Code Settings

The seventh panel, Fig 2.57, shows the Base filename and analysis settings. The Current crack growth step defaults to 0 – representing the initial crack. If we "manually" propagated the crack first, we should set this to 1. FRANC3D will extend the initial crack based on the growth rule defined in the previous panels and then name the resulting set of files as *Abaqus_Cube_Subdivide_STEP_001*.*. Subsequent file names have the step number incremented as the automatic analysis proceeds.

Some options will be specific to your site. You should have already set the ABAQUS executable path in the FRANC3D Preferences. The Global model settings should be set automatically. The local + global connection is displayed in the next panel; select **Next**.

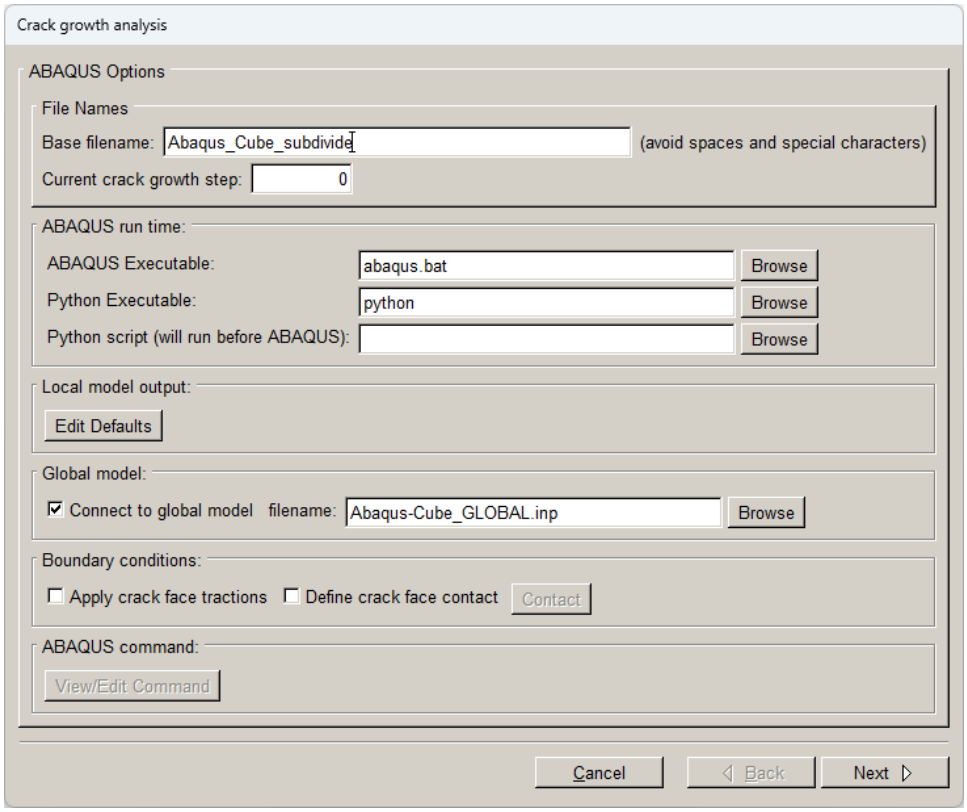


Figure 2.57 Crack Growth Analysis wizard – seventh panel.

The eighth panel, Fig 2.58, displays the local + global connection. The default settings use node-merging of the cut-surfaces between the local and global portions. Select **Finish** to start the automated crack growth process.

FRANC3D extends the initial crack first. It then saves *.fdb*, *.inp* and other files for the first crack step model, using the name *Abaqus_Cube_Subdivide_STEP_001*, and then ABAQUS starts in the background. If the analysis stops at any stage, it can be restarted from the last crack step. All the required restart *_STEP_#* files are retained. The model for any step can be read into FRANC3D to view the SIFs or to restart the analysis with a modified crack growth increment or modified fitting (for example).

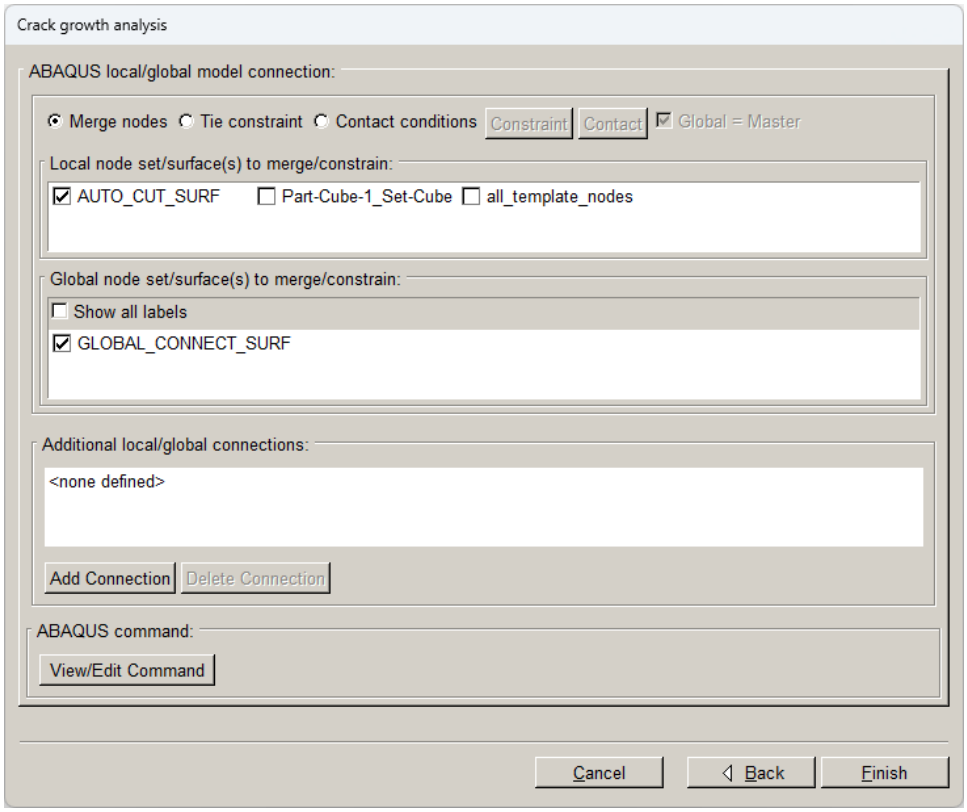


Figure 2.58 Local + Global connection options

A summary of the various files can be found in Section 3 of the User's Guide.

2.9 Step 9: SIF History and Fatigue Life

When the automatic crack growth analysis from Step 8 has finished, the SIF history can be displayed. If you still have FRANC3D open and the crack growth from Step 8 is done, you can proceed with Step 9.1. Otherwise, you can restart FRANC3D and read in the *.fdb* file for the last step, using the **File → Open** menu option.

Step 9.1: Select SIFs Along a Path

From the FRANC3D menu, select **Cracks →SIFs Along a Path**, Fig 2.59. If the SIFs have not been computed yet, the Compute SIFs dialog (see Fig 2.39) will be displayed; leave the defaults and select **Finish**.

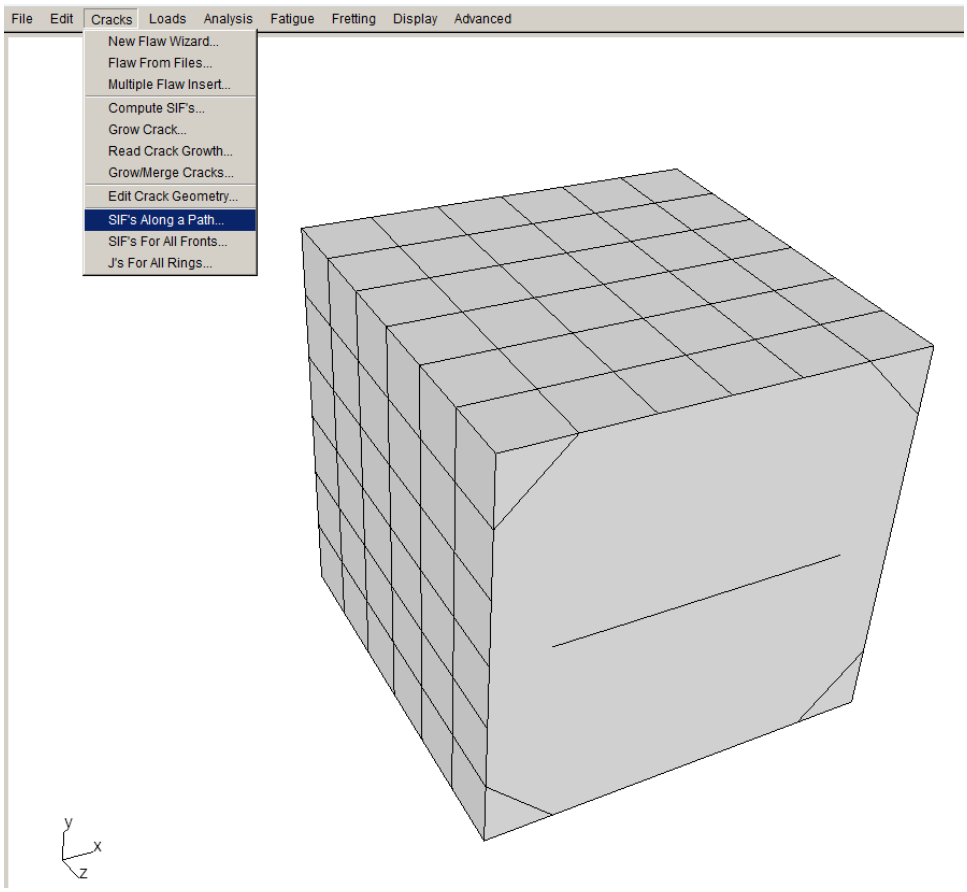


Figure 2.59 SIFs Along a Path menu option

The Stress Intensity Factors (along a path) dialog should appear, Fig 2.60. The crack fronts are displayed on the left along with the path through the fronts; the SIF history along the path is shown in the graph on the right. You can use the tabs above the graph to plot the Mode II and III SIFs as well as the elastic J-integral and T-stress values along the path. You can define a new path, Fig 2.61. You can export the data also; for example, you might need to export Mode I SIF

history (K vs a) to compute fatigue cycles using a different program. The exported data files are ASCII txt files.

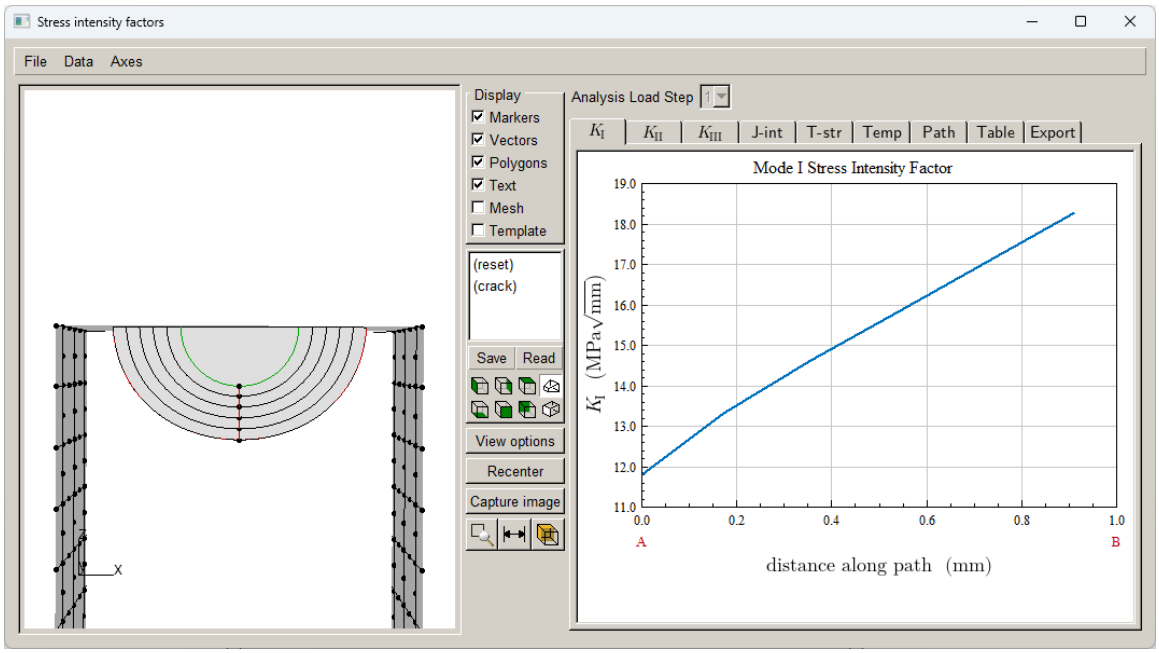


Figure 2.60 SIFs Along a Path dialog – KI plot

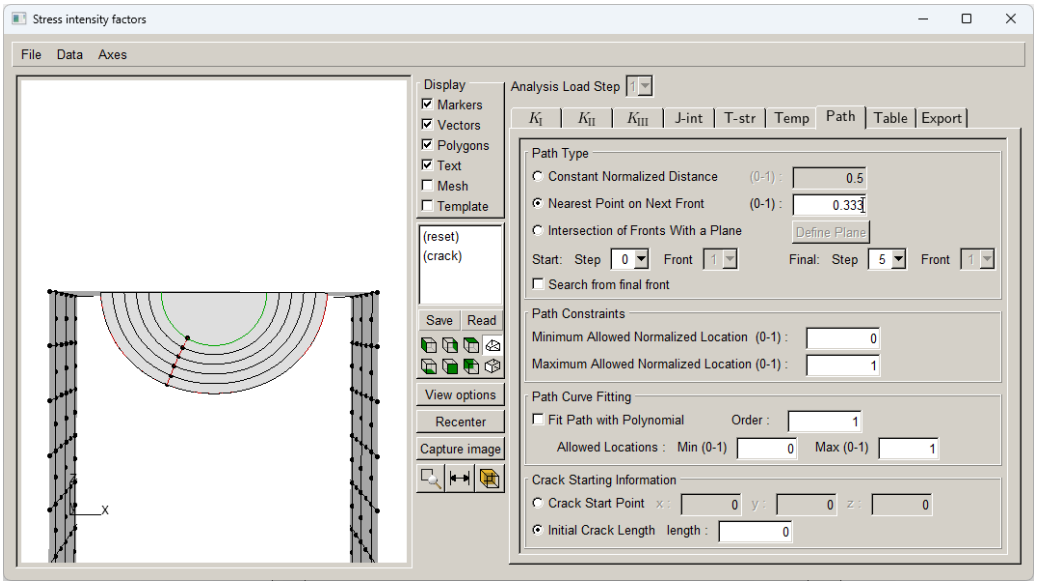


Figure 2.61 SIFs Along a Path dialog – Define Path

Step 9.2: Select SIFs For All Fronts

From the FRANC3D menu, select **Cracks →SIFs For All Fronts**. If the SIFs have not been computed yet, the Compute SIFs dialog will be displayed; leave the defaults and select **Finish**.

The Stress Intensity Factors (for all fronts) dialog should appear, Fig 2.62. The crack fronts are displayed on the left, and the SIFs for all crack fronts are displayed on the right. You can use the tabs above the plot to display Mode II and III SIFs as well as the elastic J-integral and T-stress values. If there are multiple crack fronts or multiple load steps, these can be selected using the drop-down lists; this will be seen in subsequent tutorials.

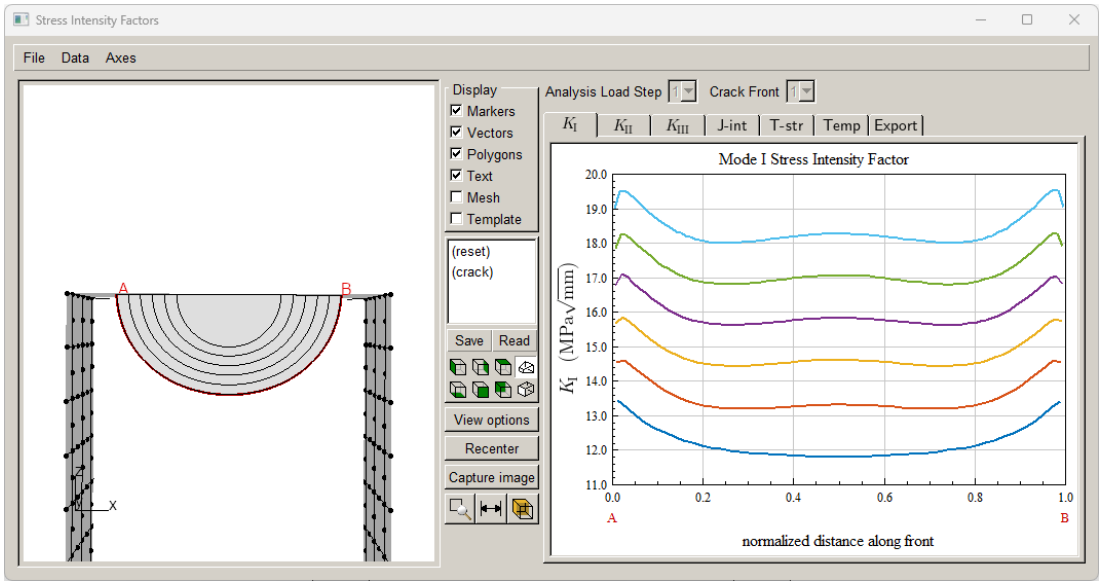


Figure 2.62 SIFs for all Fronts dialog – KI plot

Step 9.3: Select Fatigue Life Predictions

Fatigue life or cycles can be computed next. From the FRANC3D menu, select **Fatigue →Fatigue Life Predictions**, Fig 2.63. If the SIFs have not been computed yet, the Compute SIFs dialog will be displayed; leave the defaults and select **Finish**.

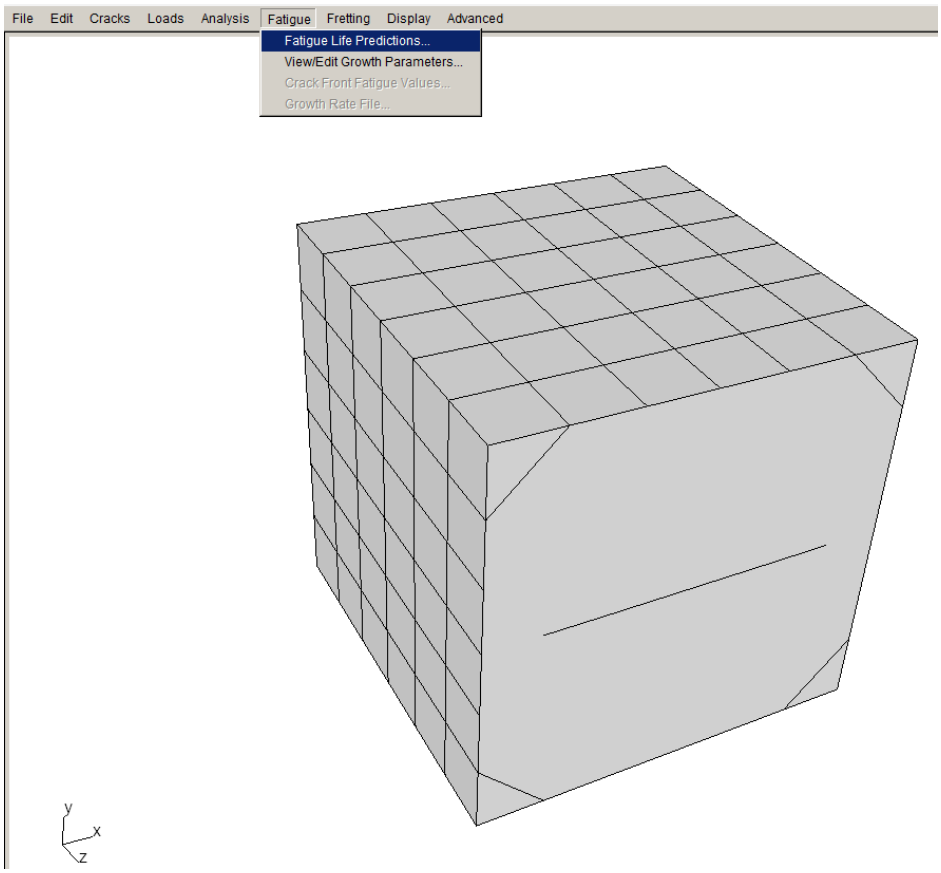


Figure 2.63 Fatigue Life Predictions menu option

The Fatigue Life dialog should appear as in Fig 2.64. 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 2.65.

The units are MPa and mm; select the **Change** button to display the dialog in Fig 2.66 and set the units to MPa and mm; select **Accept**.

The FEM units can also be set in the main FRANC3D window using the **Edit** menu.

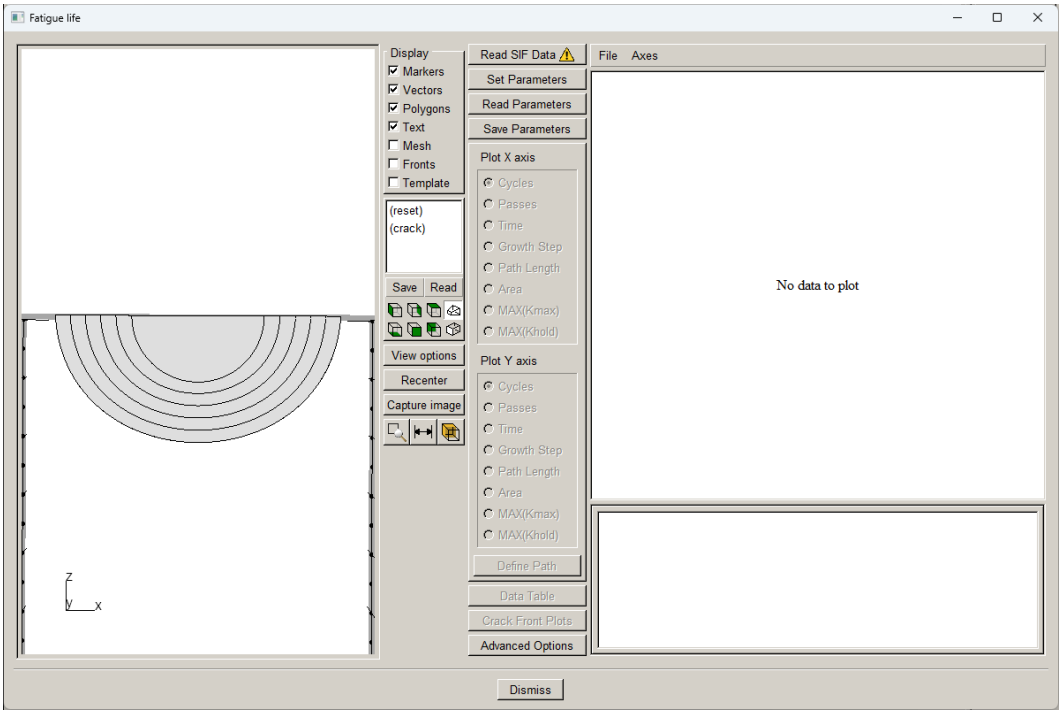


Figure 2.64 Fatigue Life dialog

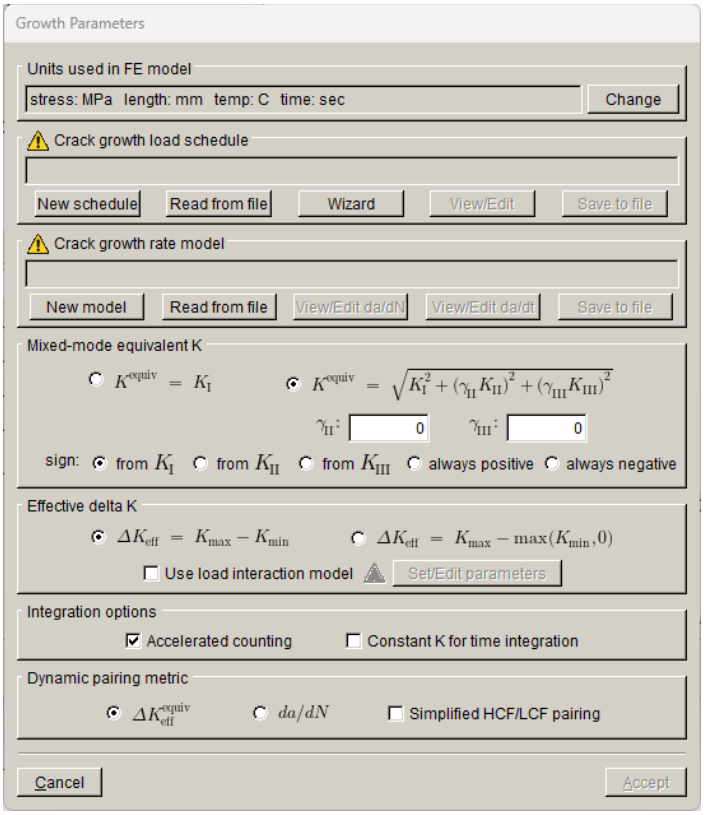


Figure 2.65 Fatigue Life parameters

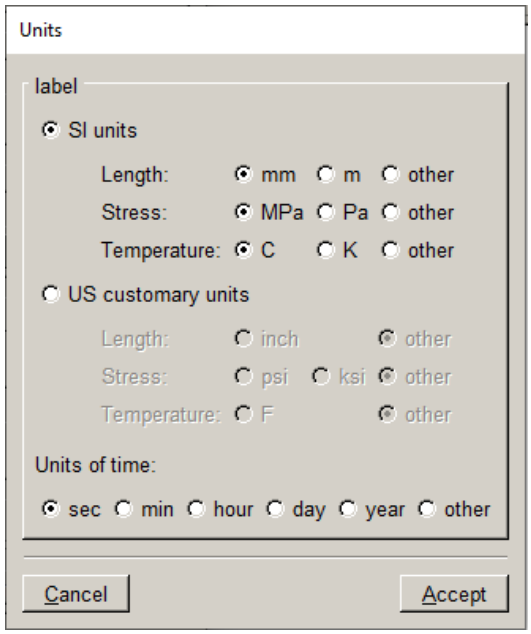


Figure 2.66 FEM units dialog

The crack growth load schedule is defined next. Select the **New Schedule** button (see Fig 2.65) to display the dialog in Fig 2.67. Select the Schedule and then select the **Add** button; see right side image of Fig 2.67. This will display the dialog shown in Fig 2.68; we use a simple cyclic load schedule where the applied load produces Kmax, and Kmin is zero. Select **Accept** to return to the Load Schedule, Fig 2.69.

The stress ratio is set to 0.0. There is only one load case, which represents the Kmax condition. The Repeat count is set to FOREVER. Select **Accept** to finish the load schedule. There is only one event in the schedule so the repeat forever can be set for either the event or the schedule; it is usually best to set Repeat FOREVER for the schedule.

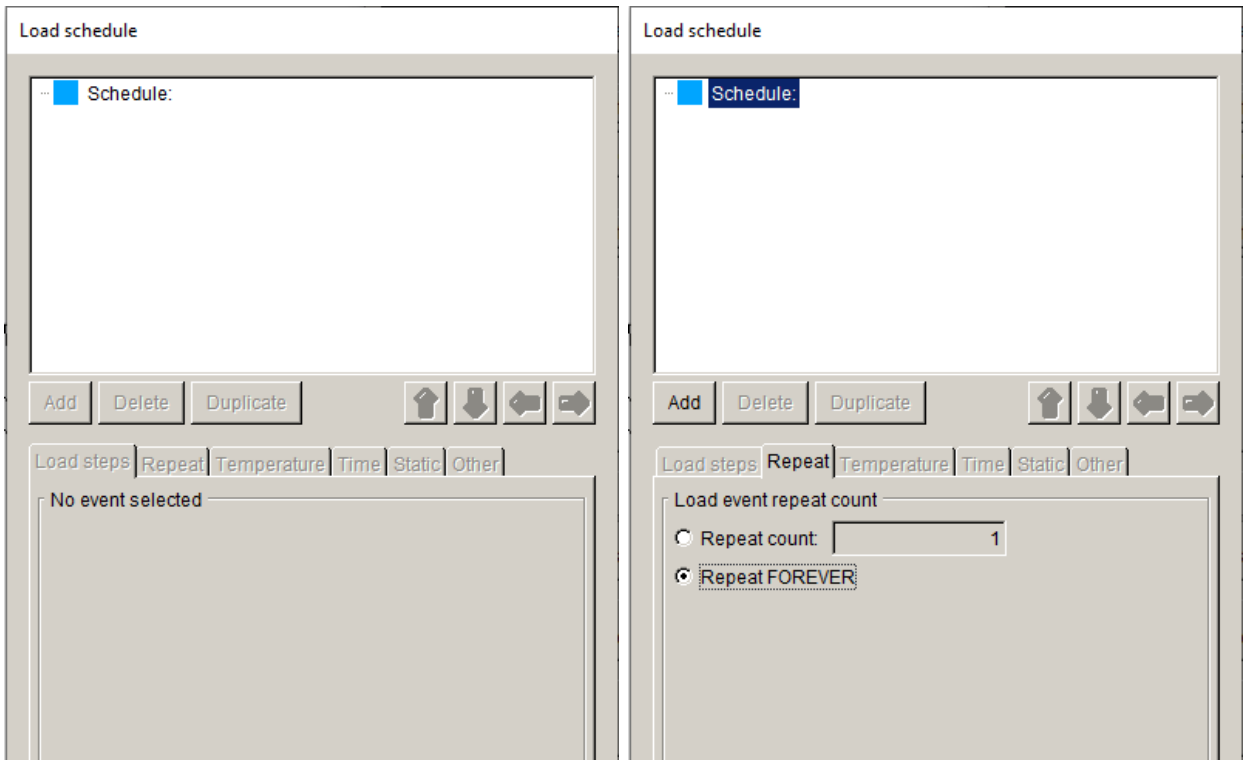


Figure 2.67 New Load Schedule dialog

Figure 2.68 Event type dialog

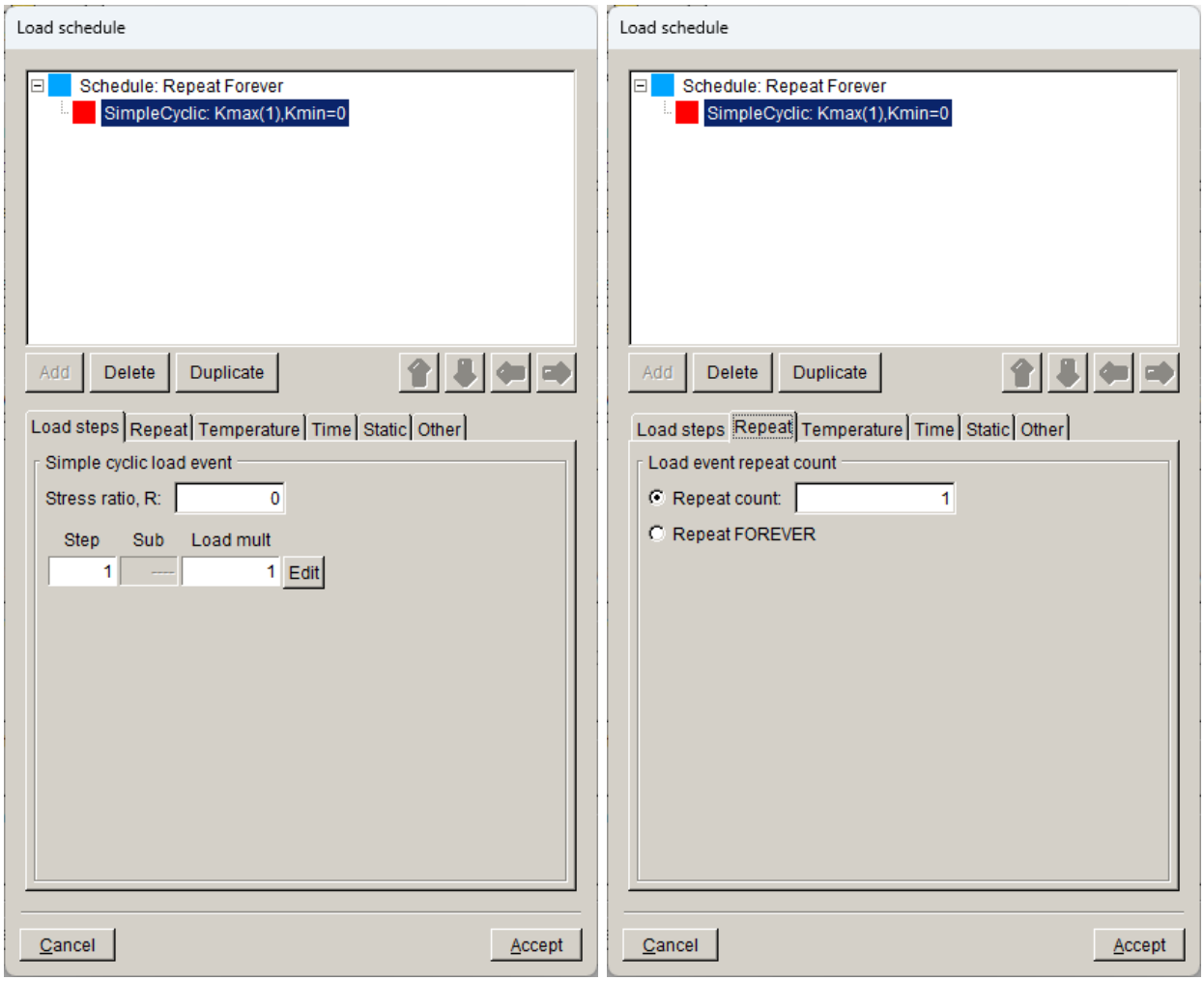


Figure 2.69 Simple cyclic load event repeated once and schedule repeated forever

The crack growth rate model is defined next. Select the **New Model** button (see Fig 2.65) to display the dialog in Fig 2.70. Choose the cyclic loading growth rate model, and select **Next** to display the dialog in Fig 2.71.

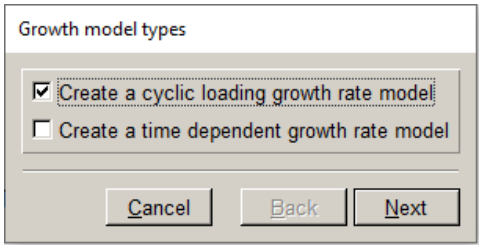


Figure 2.70 Growth model type dialog

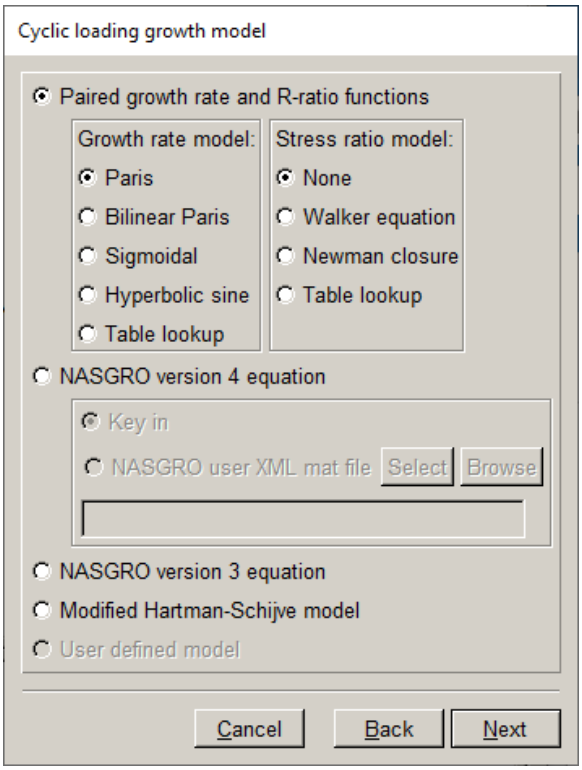


Figure 2.71 Cyclic loading growth model dialog

We use a Paris growth model, which is the default; select **Next** to display the dialog in Fig 2.72. The growth rate model will be temperature independent. Select **Next** to display the Paris growth model dialog, Fig 2.73.

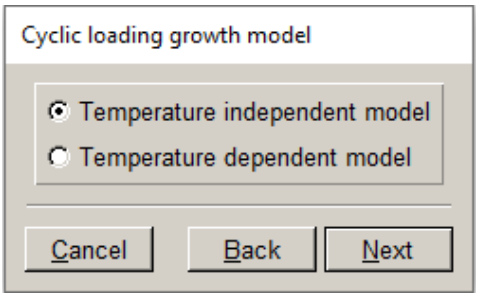


Figure 2.72 Temperature dependent or independent growth rate model dialog

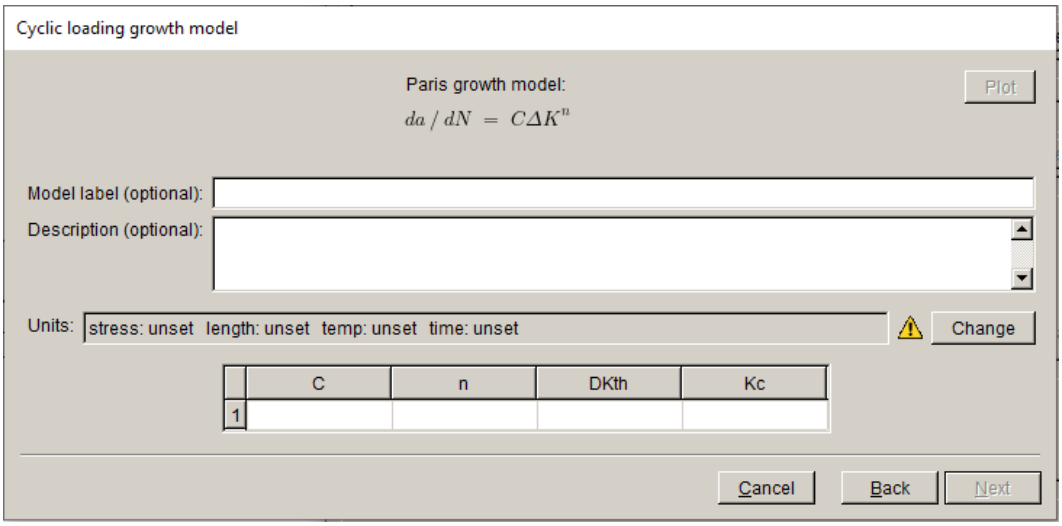


Figure 2.73 Paris growth rate model dialog

First, set the units for the Paris growth model. Select the **Change** button to display the Units dialog, Fig 2.74. Set the units the same as the FEM units that we set earlier, which were MPa and mm. If your growth model units are different from the FEM units, FRANC3D will do the unit conversions automatically. Select **Accept**.

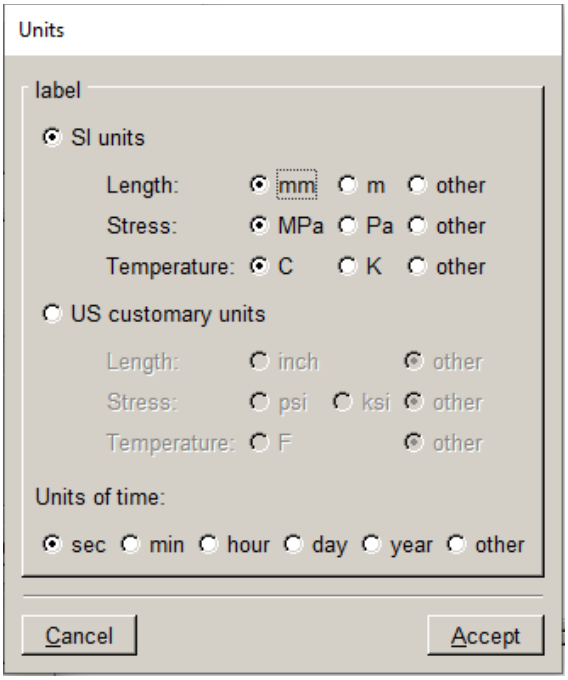


Figure 2.74 Paris growth rate model units

In the Paris growth model dialog, click on the fields for C, n, DKth and Kc and enter the values shown in Fig 2.75. Select **Next** to return to the main dialog (see Fig 2.65). Other settings and options (Fig 2.65) are left at their default values; select **Accept** (Fig 2.65) to return to the Fatigue Life dialog, Fig 2.76.

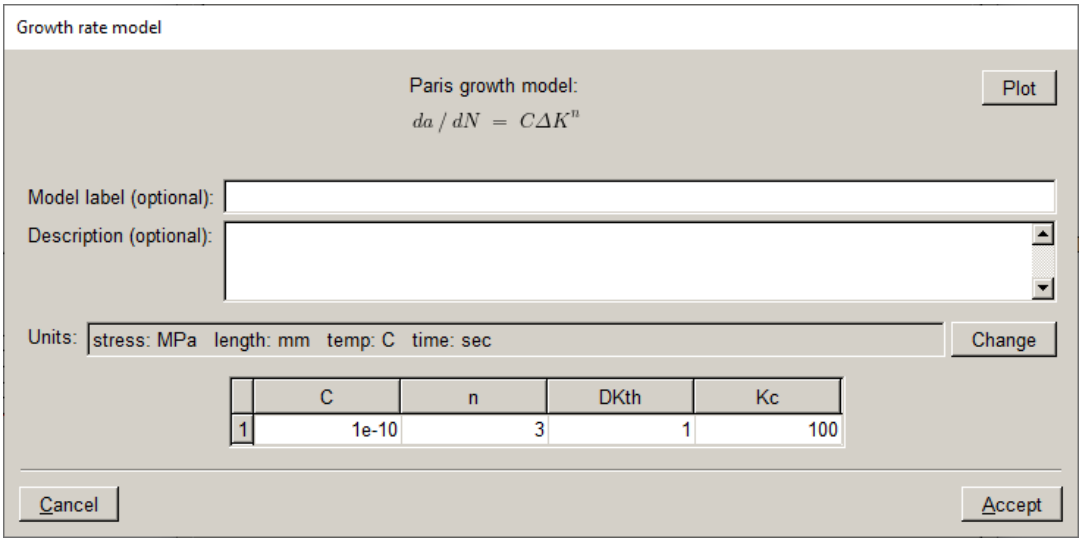


Figure 2.75 Paris growth rate model values

Note that the Paris exponent 'n' is set to 3 whereas we used n=2 for the quasi-static growth. In practice, you can use the "correct" fatigue model and material data during the growth, so that growth is consistent with the fatigue life computations. For a single crack front with relatively uniform K_I values along the front, the quasi-static growth option is acceptable. If there are multiple crack fronts with significantly different growth rates, the "correct" fatigue model should always be used.

If we choose to plot cycles based on a path, we must select the Path radio button and then use the **Define** button to create a path, Fig 2.77. The default path is through the crack front midpoints. Once a path is defined, the cycles versus crack path length is displayed, Fig 2.78.

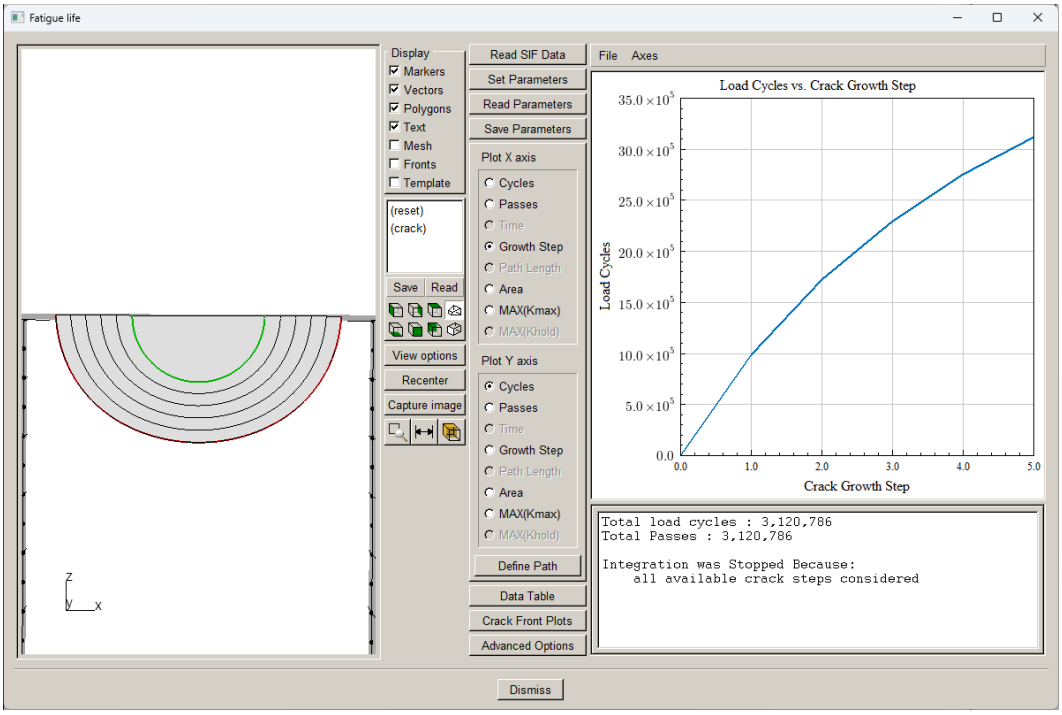


Figure 2.76 Fatigue life dialog showing Cycles vs Crack Growth Step

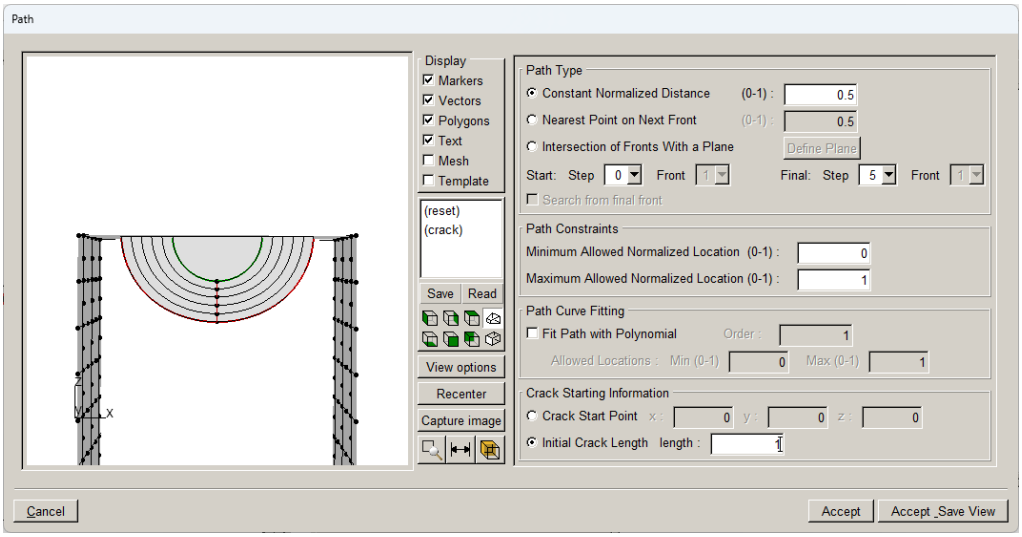


Figure 2.77 Define crack path dialog

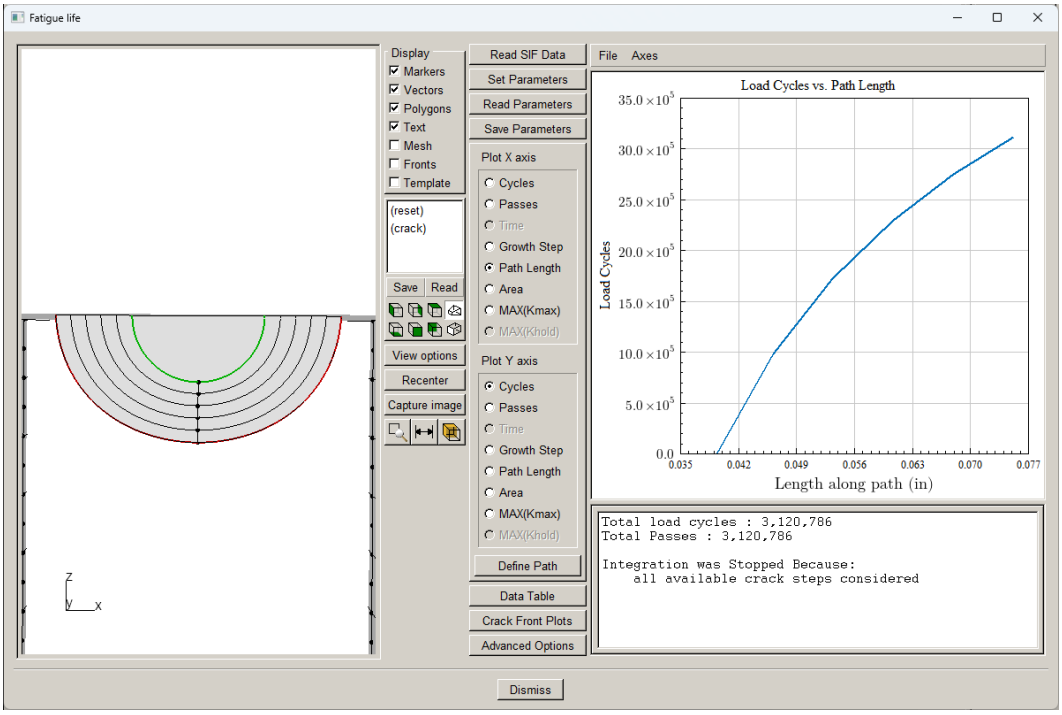


Figure 2.78 Fatigue cycles versus crack length

Appendix A:
ABAQUS local model between *Tie constraints

This appendix describes how one can use an already-divided ABAQUS model, where the local portion is tied to the global portion using *TIE constraints. Rather than dividing the model using the FRANC3D submodeling tools, one can divide the model and save the "local" and "global" *.inp* files using ABAQUS CAE. This appendix describes ABAQUS specific features that are not fully described in the Tutorials 2-14 document.

The complete model of a simple hinge is shown in Fig A-1. The local portion that will be imported into FRANC3D is located between two *TIE surfaces. The complete model can be copied to a local and a global model and the appropriate portions of the model saved as separate *.inp* files. In this example, the files are named: *hinge-global.inp* and *hinge-local.inp*. Once these two files are saved, we can proceed to FRANC3D.

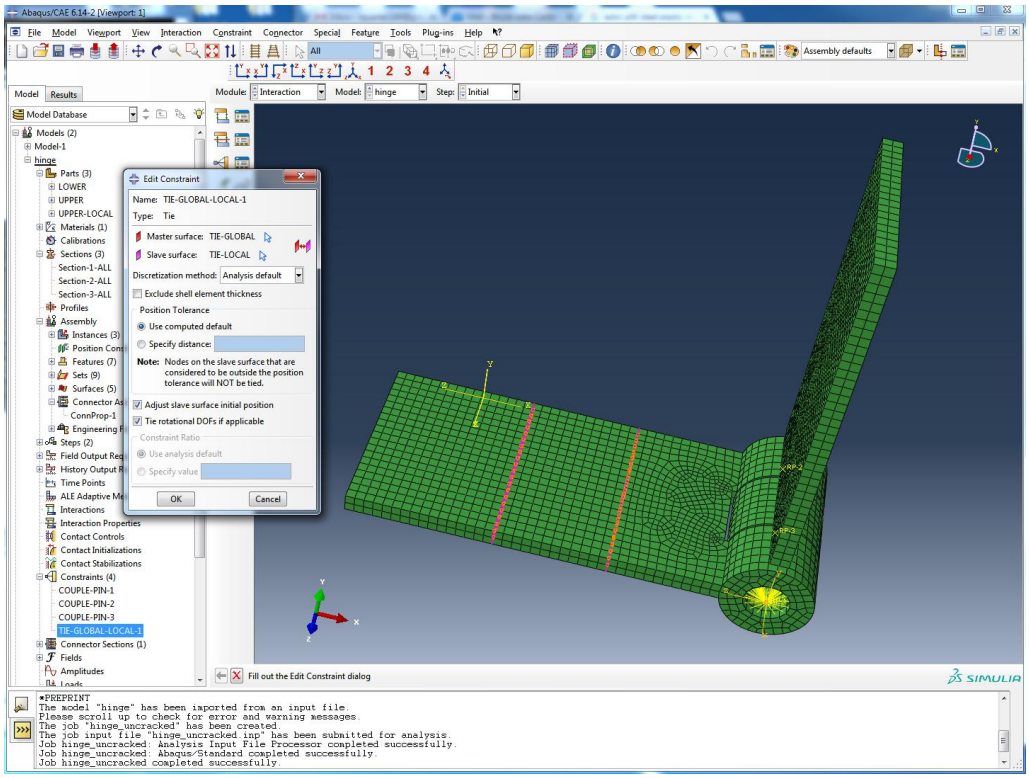


Figure A-1 ABAQUS hinge model; local portion is contained between two *TIE surfaces.

From the FRANC3D **File** menu, select Import. From the first dialog, Fig A-2, select the Import an already divided model option and click **Next**. The second dialog, Fig A-3, allows one to choose the global and local *.inp* files. Make sure the **ABAQUS** Mesh File Type is selected and then use the **Browse** button to specify the two files. Once the *hinge-global* and *hinge-local.inp* files are selected, click **Next**.

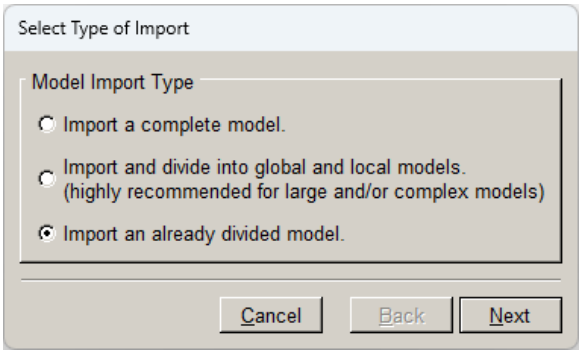


Figure A-2 Select type of import dialog.

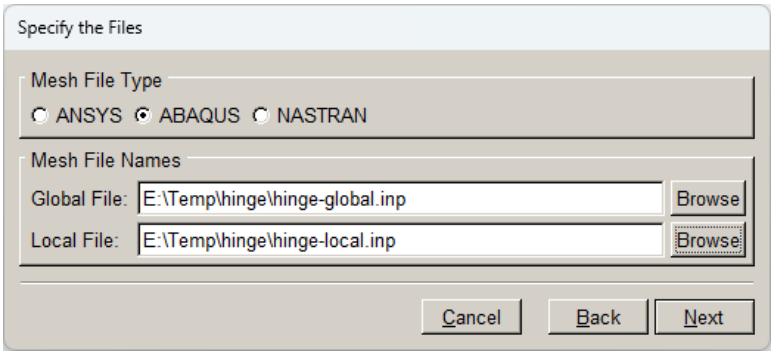


Figure A-3 Specify *.inp* files dialog for an already-divided model.

The next dialog, Fig A4, allows one to retain mesh facets. There are no boundary conditions, but we will retain the *TIE surfaces. Note that the tie-local surface (AUTO_CUT_SURF node set) was defined in ABAQUS CAE and is part of the local portion. There is a corresponding tie-global surface for the global portion. Click **Next** to set the units (default units can be used), and then click **Finish** to proceed to the FRANC3D main window, where the model will be displayed, Fig A-5.

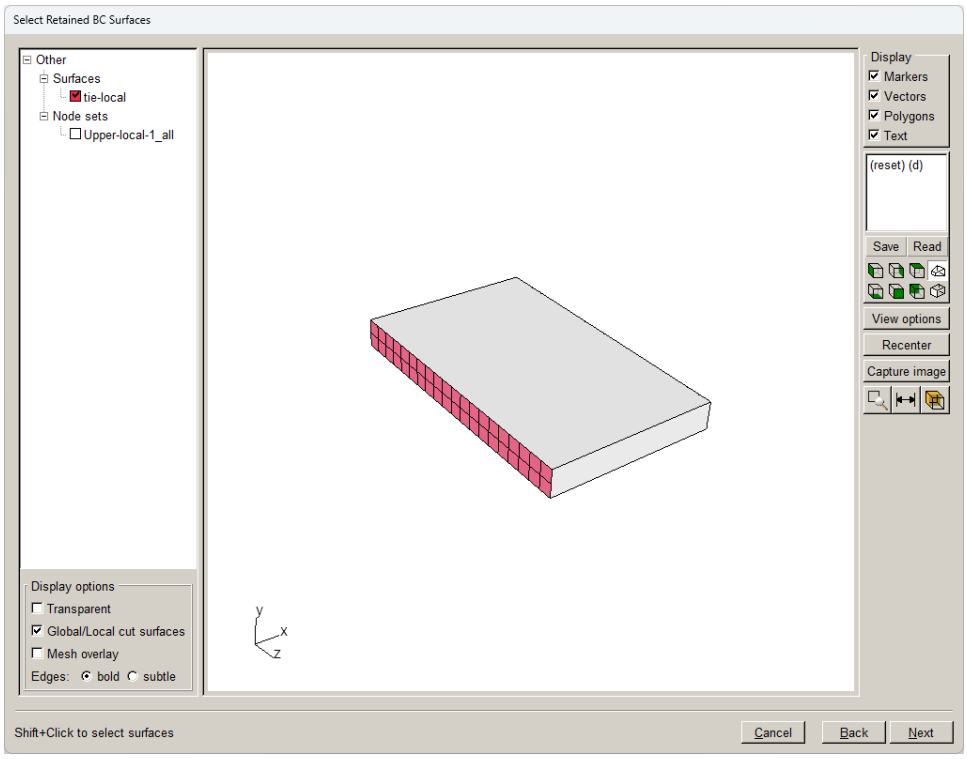


Figure A-4 Select retained surfaces dialog; the *TIE (local) constraint surfaces are selected.

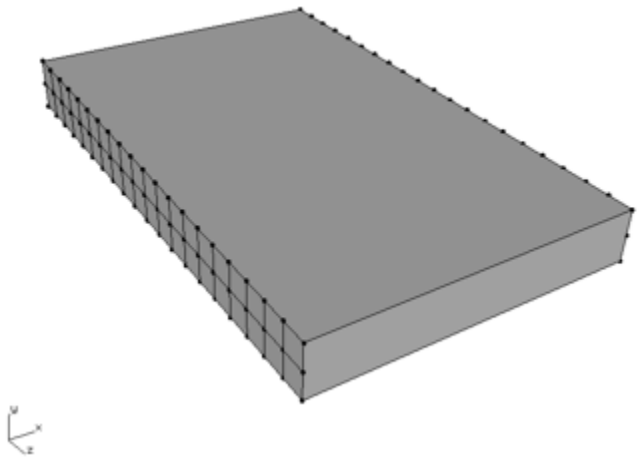


Figure A-5 FRANC3D local model imported with the *TIE surface mesh retained.

The next step is to define and insert a crack. We have defined and saved a crack to a file previously, so we use the **Cracks** menu and choose **Flaws from Files**. Fig A-6 shows the dialog box that allows us to choose the *hinge.crk* file. Once the file is selected, click **Accept**.

Fig A-7 shows the crack location in the local portion; we can move it if needed. The crack is a center-through type crack with two crack fronts; the distance between the two fronts is 2 units.

Click **Next** to set the crack front template mesh settings, Fig A-8. The template radius is saved in the *.crk* file as 0.1; we leave all the settings at their default values and click **Finish** to insert the crack and remesh. The remeshed local cracked model is shown in Fig A-9.

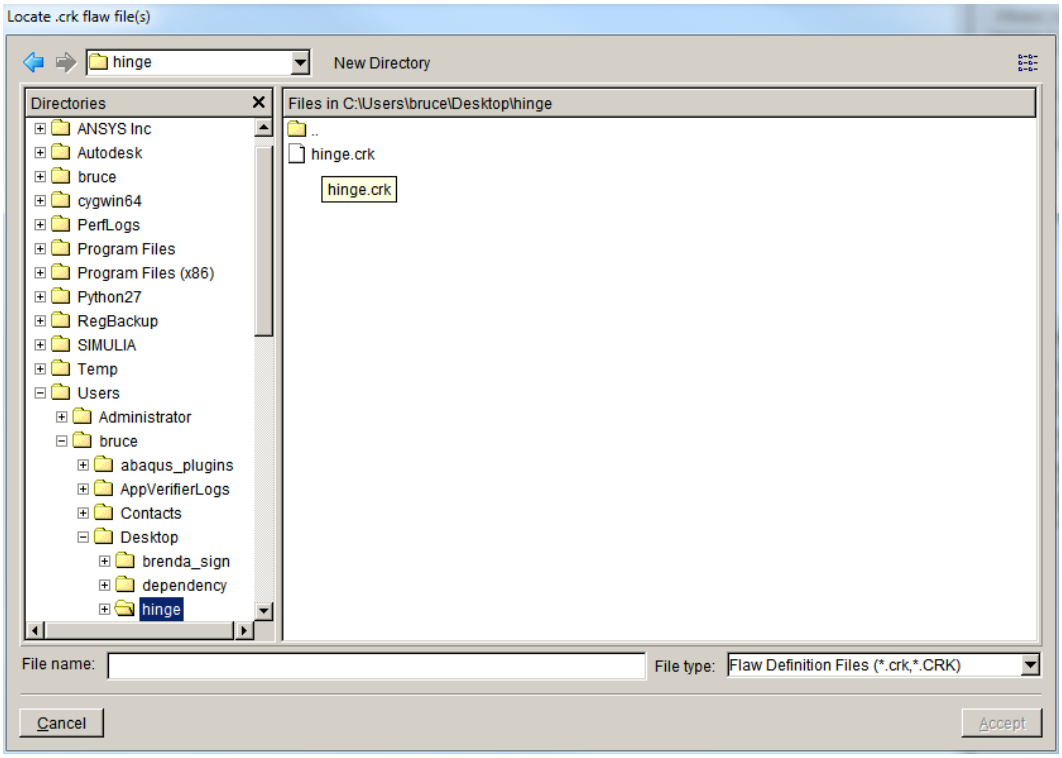


Figure A-6. Dialog to choose the .crk file.

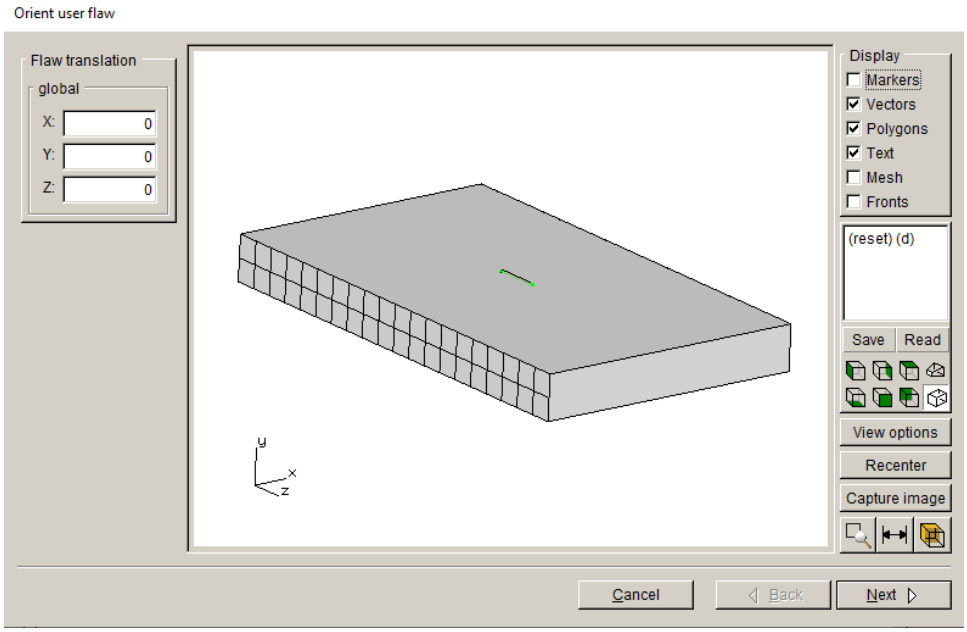


Figure A-7. Crack location dialog.

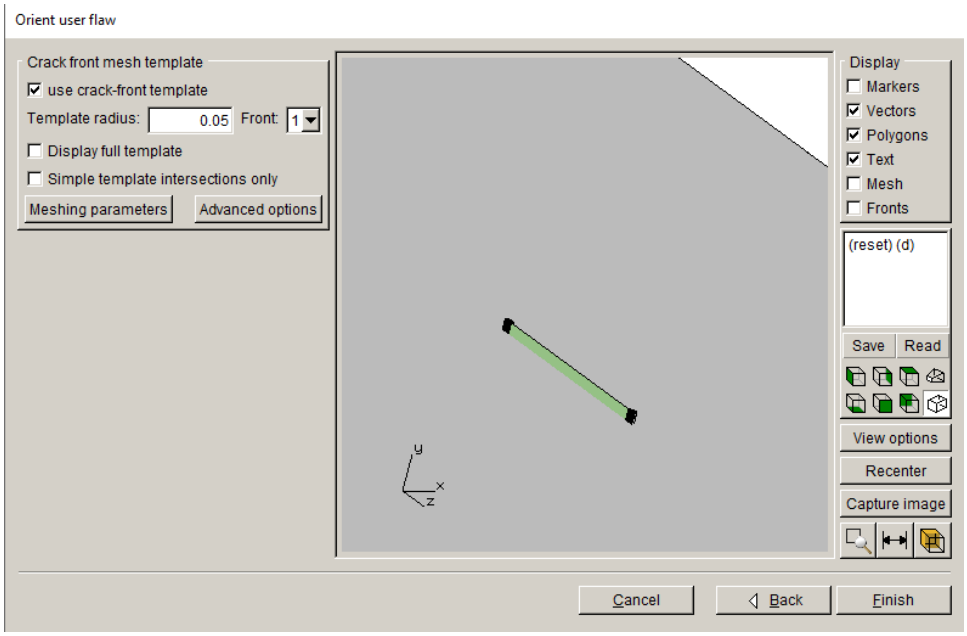


Figure A-8. Crack front template mesh settings dialog.

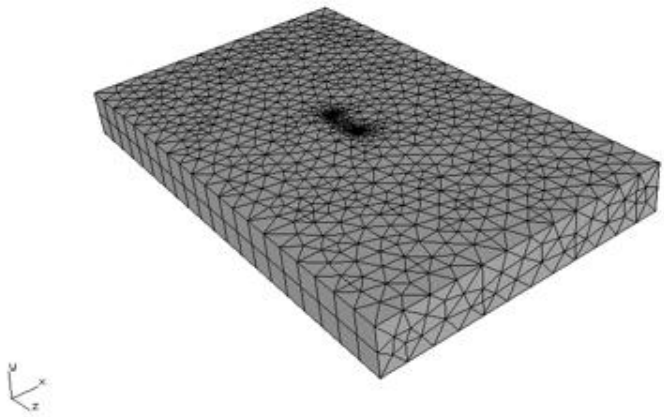


Figure A-9. Remeshed cracked local portion.

Once the crack is inserted, we can perform a static analysis. From the **Analysis** menu, select **Static Crack Analysis**. The first dialog, Fig A-10, allows us to specify the file name. We call it *hinge_crack*. Click **Next** and check that the solver is ABAQUS, Fig A-11. Click **Next** to set the ABAQUS solver options and to specify the global file name, Fig A-12.

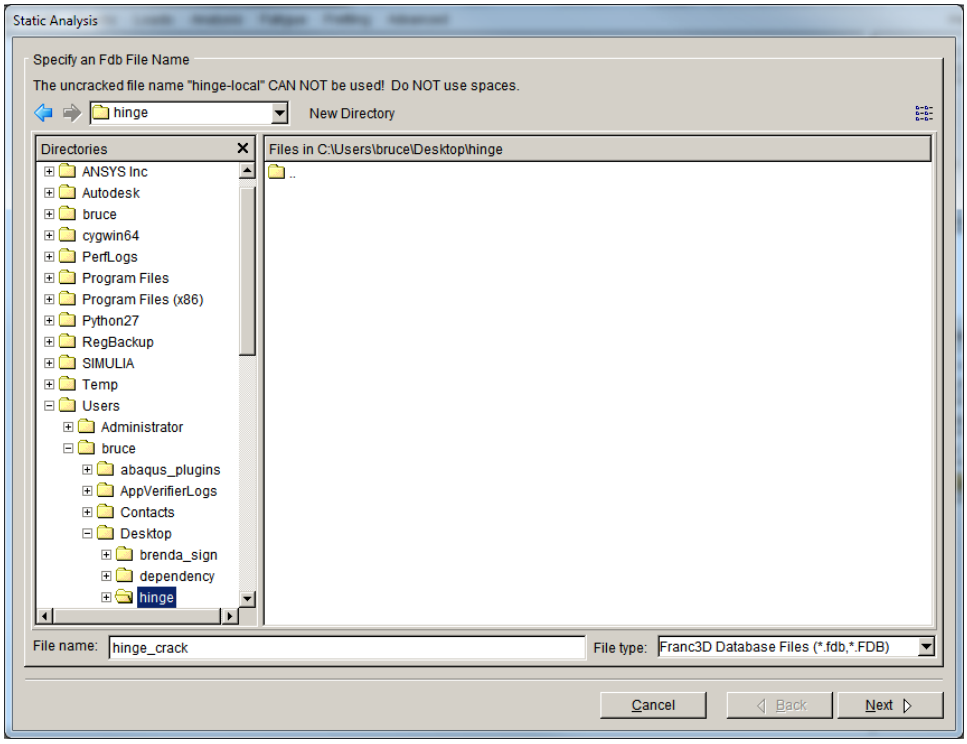


Figure A-10. Static analysis file name dialog.

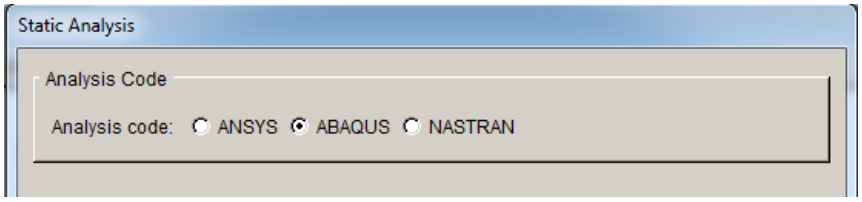


Figure A-11. Specify the solver as ABAQUS.

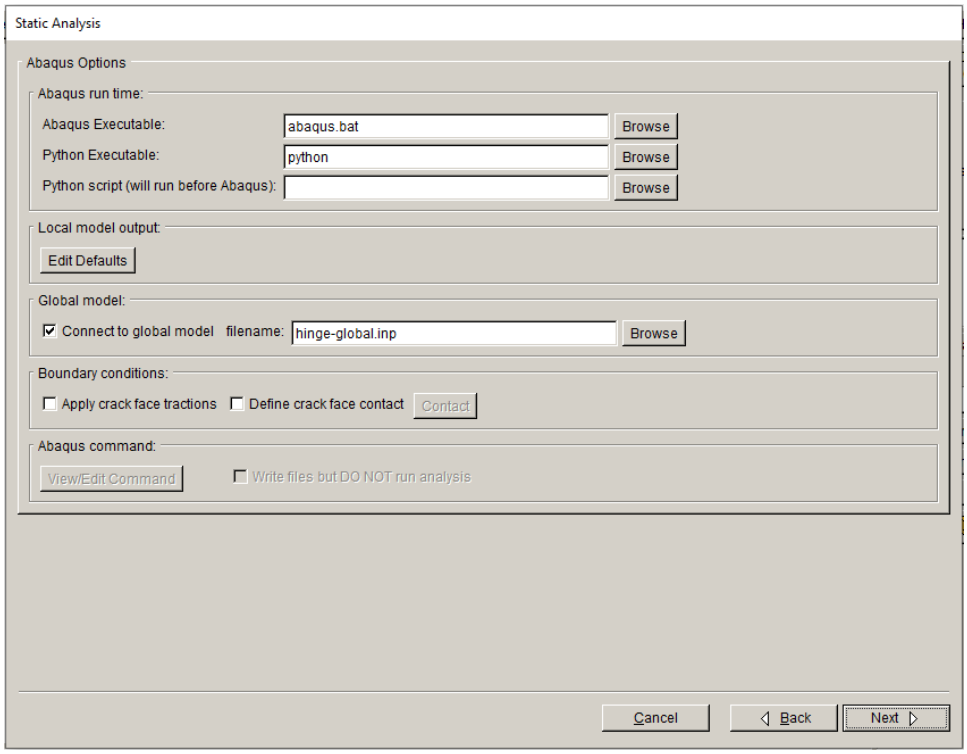


Figure A-12. Specify the ABAQUS analysis options.

The Global model file name is already filled in by default based on the imported file selection. Click **Next** to advance to the next dialog, Fig A-13, where we specify how the local and global model portions will be re-connected. The "tie-local" and "tie-global" surfaces must be selected along with the Tie constraint option.

Click **Finish** to start the process of saving the analysis files and running ABAQUS in the background. Once ABAQUS finishes running, we can compute the SIFs and analyze the results to verify that FRANC3D correctly processed all the ABAQUS information.

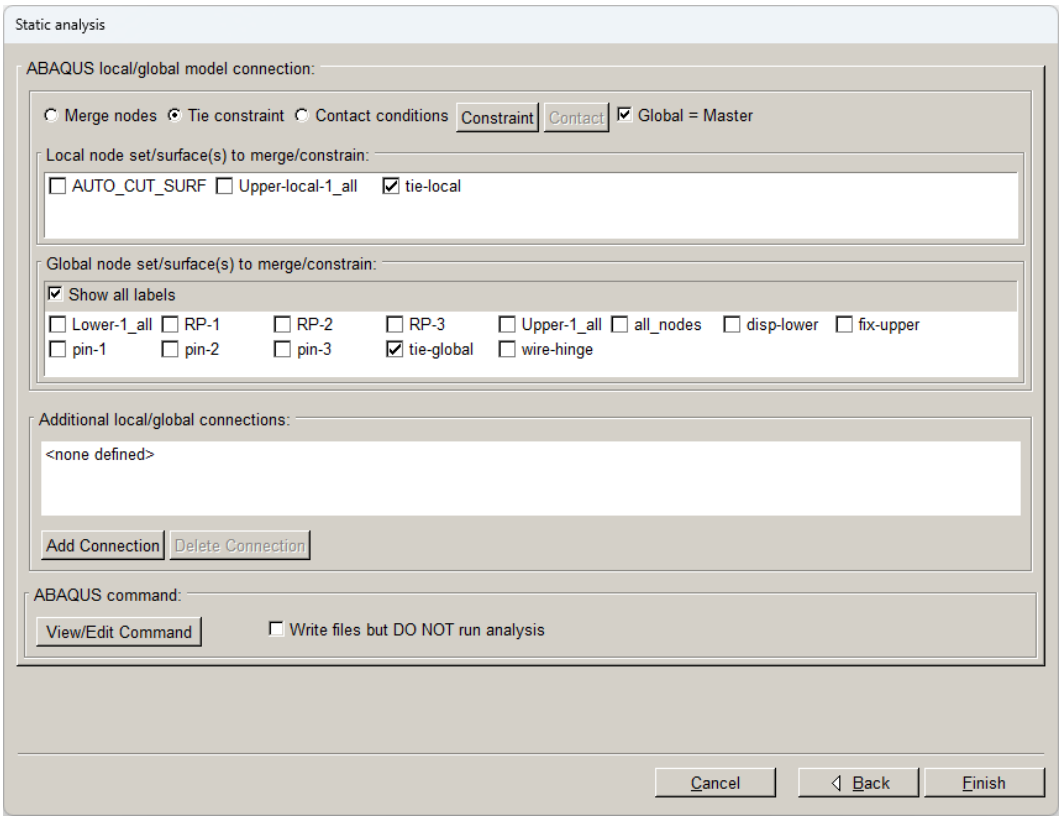


Figure A-13. Specify the ABAQUS local + global model connection.

ABAQUS creates a *.dtp* file and this file is automatically imported into FRANC3D (assuming that you are running ABAQUS on the same PC). Use the **Cracks** menu and select **Compute SIFs**, Fig A-14. Click **Finish** to display the SIFs for the two crack fronts, Fig A-15.

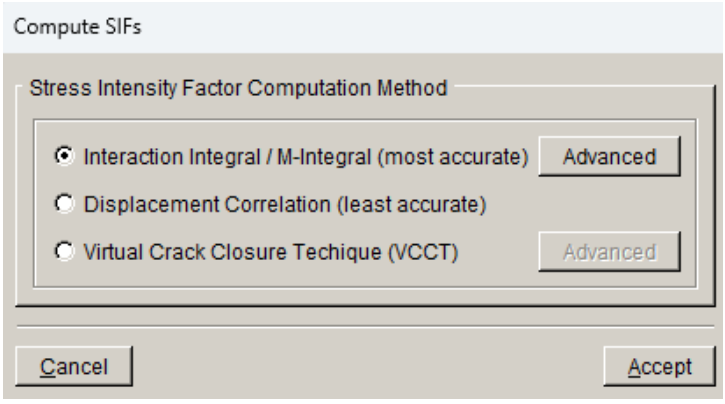


Figure A-14. Compute SIFs dialog.

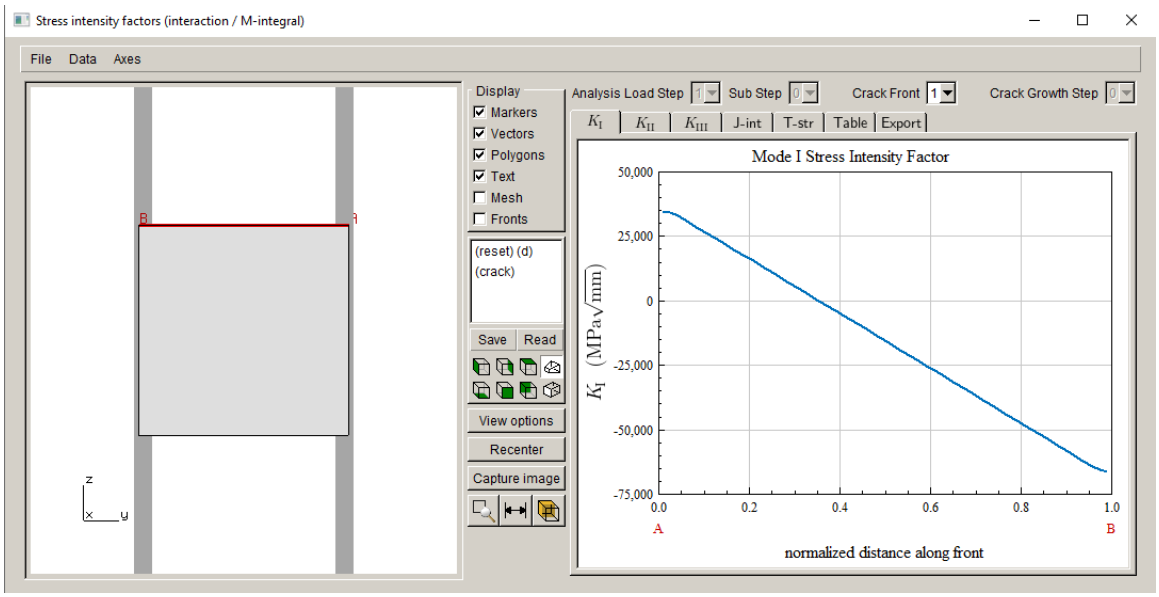


Figure A-15. Display SIFs dialog.

The SIFs indicate that the hinge is bending, which is what we expect. The deformed shape for the uncracked hinge is shown in Fig A-16. The contours show the displacement magnitude. Fig A-17 shows the same displacement magnitude contours for the cracked hinge model. The contours are almost identical, indicating that the cracked model is valid. Fig A-18 shows a close-up of the crack mouth; the displacement scaling is 10x and the contours show the maximum principal stress.

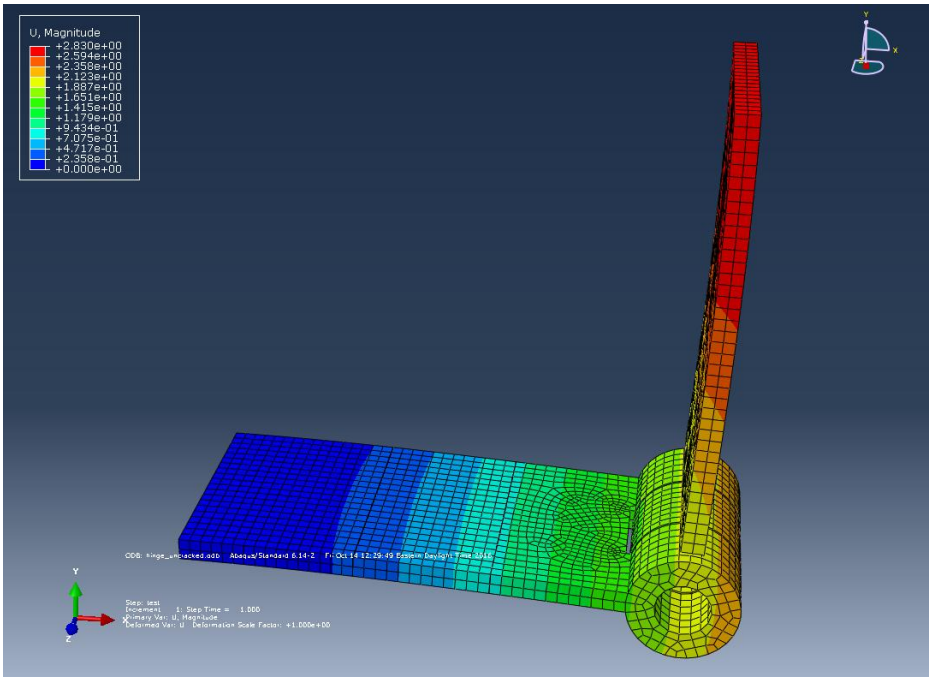


Figure A-16. Displacement magnitude contours for the uncracked hinge.

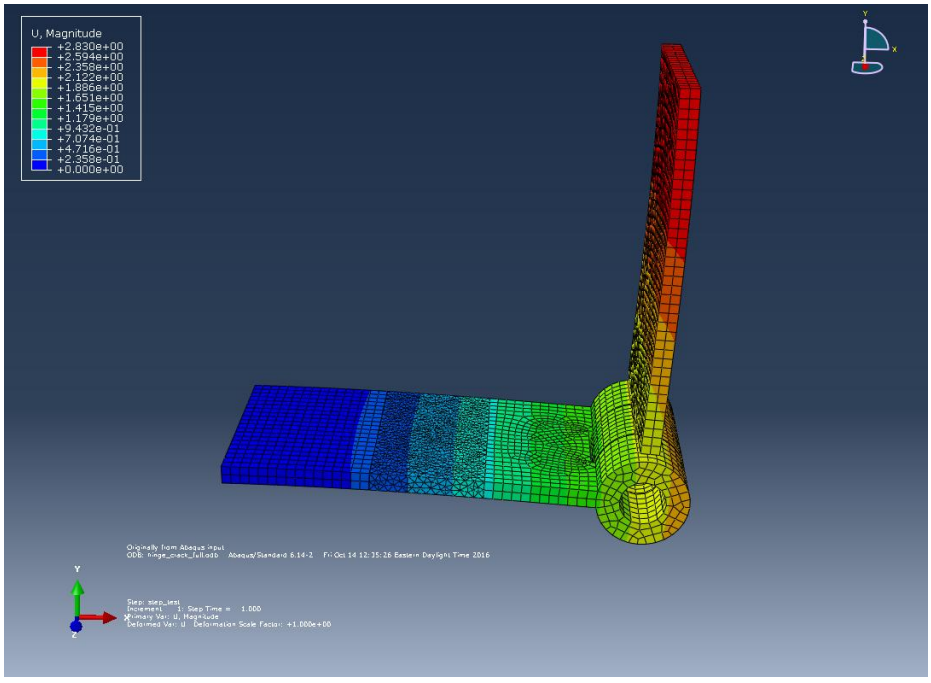


Figure A-17. Displacement magnitude contours for the cracked hinge.

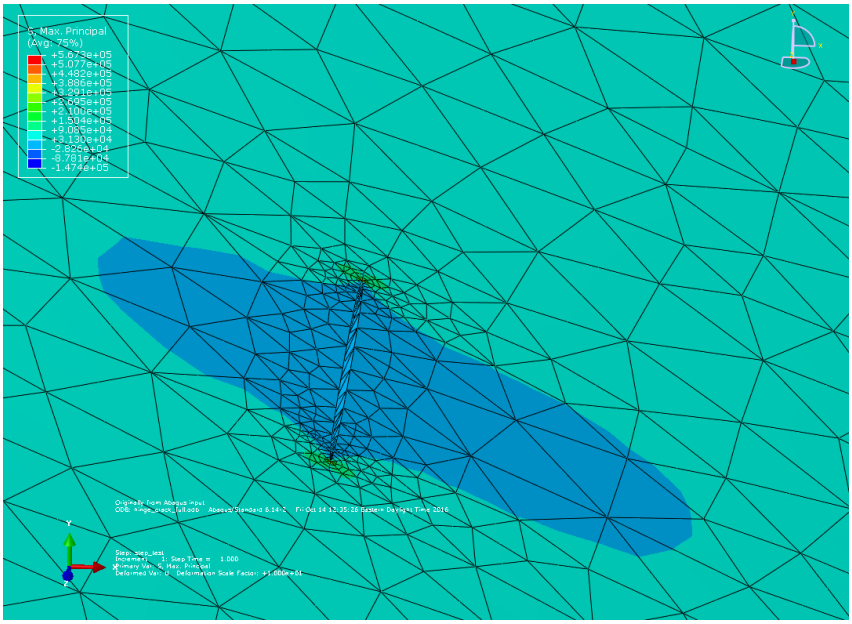


Figure A-18. Maximum principal stress contours for the cracked hinge.

Under these conditions, crack growth only occurs where the Mode I SIF is positive, Fig A-19.

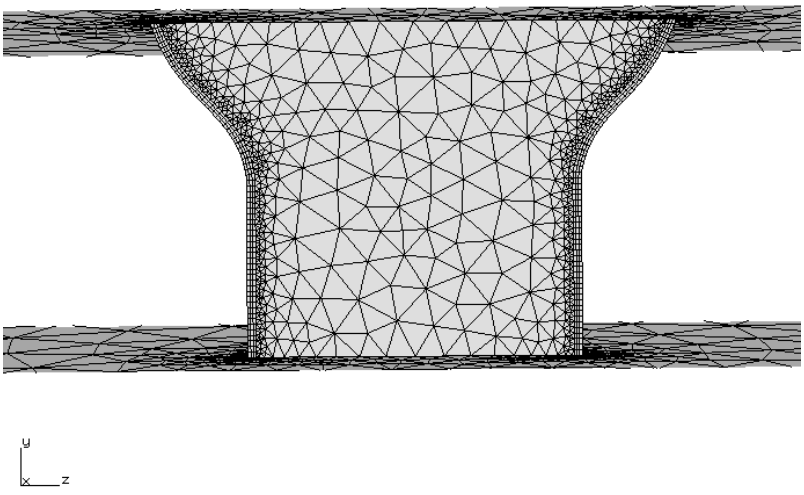


Figure A-19. Crack front partial growth.

Appendix B:
ABAQUS model with *Tied solid and shell elements.

This appendix describes how one can simulate crack growth in models with mixed solid and shell elements using *Tie constraints. A simple cube volume mesh with a shell surface attached is shown in Fig B-1. The cube mesh has two components, one for the region with the coarse mesh and one for the small portion with the more refined mesh. The two components are connected by *Tie constraints, as the meshes on the two adjacent surfaces do not match.

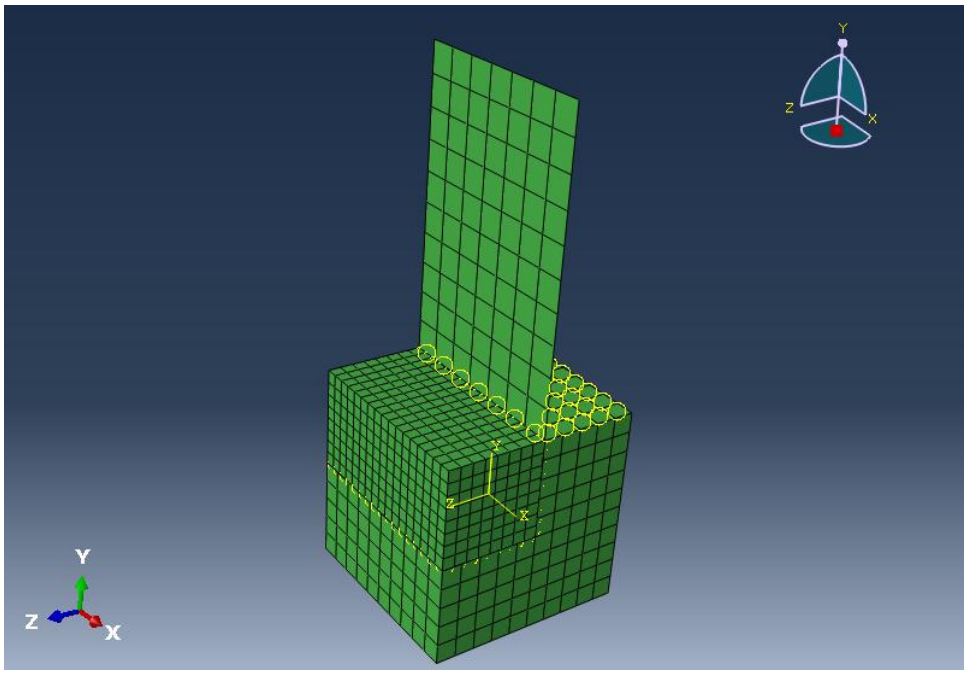


Figure B-1 ABAQUS model with mixed meshes and *Tie constraints.

From the FRANC3D **File** menu, select Import. In the first dialog, the "import and divide" option is the default, so just click **Next**. Select the *.inp* file and click **Accept** to display the dialog shown in Fig B-2. The shell elements will be displayed but cannot be selected; these elements will be part of the global *.inp* file. The local model should be selected so that the *Tie surface between the mismatched mesh surfaces stays with the global portion also. Fig B-2 shows the local portion highlighted; note that a layer of elements from the refined mesh adjacent to the Tie surface is not selected.

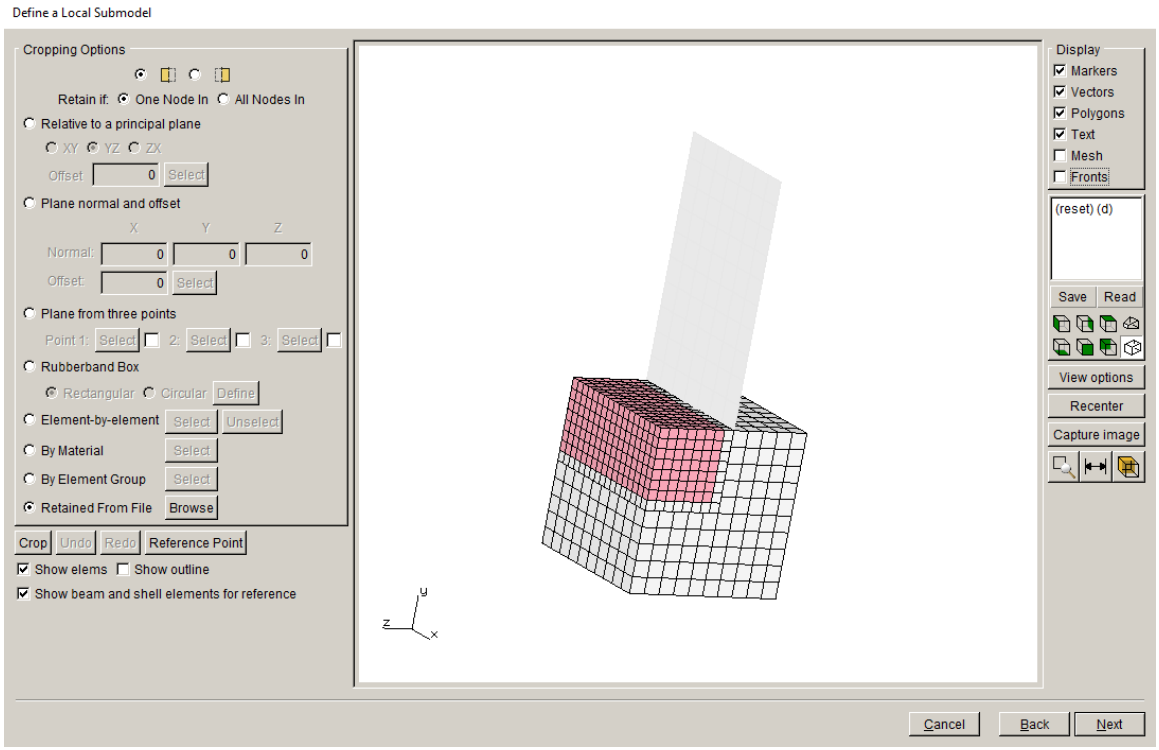


Figure B-2 ABAQUS model with local FRANC3D portion highlighted.

The local portion will be displayed in the subsequent dialog, Fig B-3. AUTO_CUT_SURF is created by FRANC3D to define the cut-surface. The mesh facets for this will be retained by default. Click **Next** and set the units, and the click **Finish** to proceed to the main FRANC3D window; the resulting local model should appear as in Fig B-4.

A surface elliptical crack is defined and inserted into the local model, Fig B-5. Once this crack is inserted and remeshing is completed, the model can be analyzed. The local portion is reconnected with the global portion. We left the existing *Tie surface between the mismatched meshes in the global portion. The original mesh was composed of linear brick elements, and the remeshed crack portion consists of quadratic elements (bricks, wedges, pyramids and tetrahedra). So, the local cracked portion is connected to the global using additional *Tie constraints, Fig B-6. AUTO_CUT_SURF and GLOBAL_CONNECT_SURF are defined by FRANC3D, and these are selected automatically along with the *Tie radio button. Select **Finish** to run the analysis.

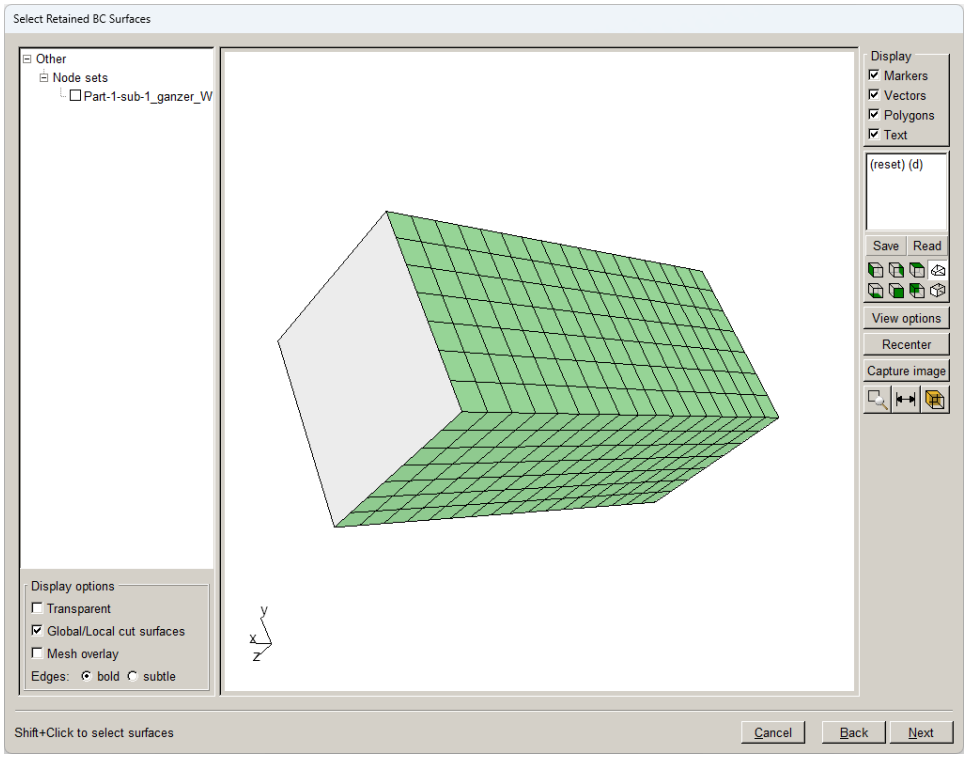


Figure B-3 Local model portion.

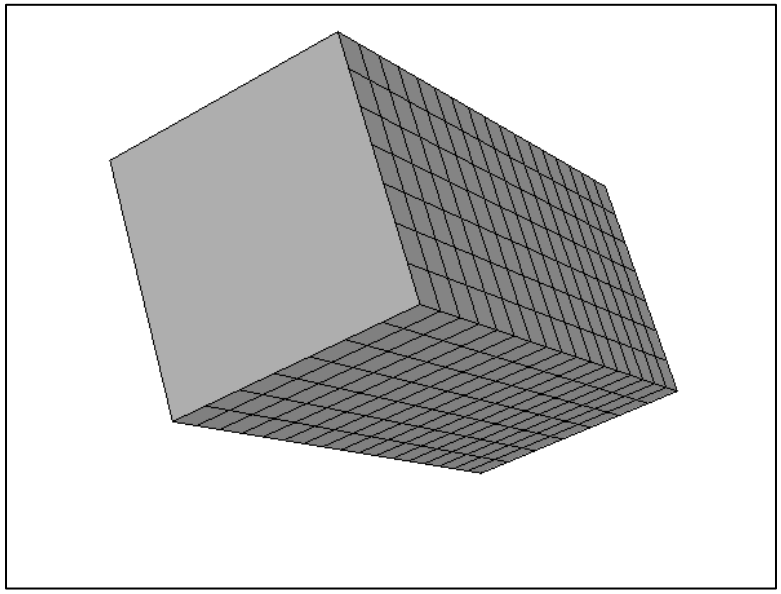


Figure B-4 Local model with retained mesh facets on the cut-surface.

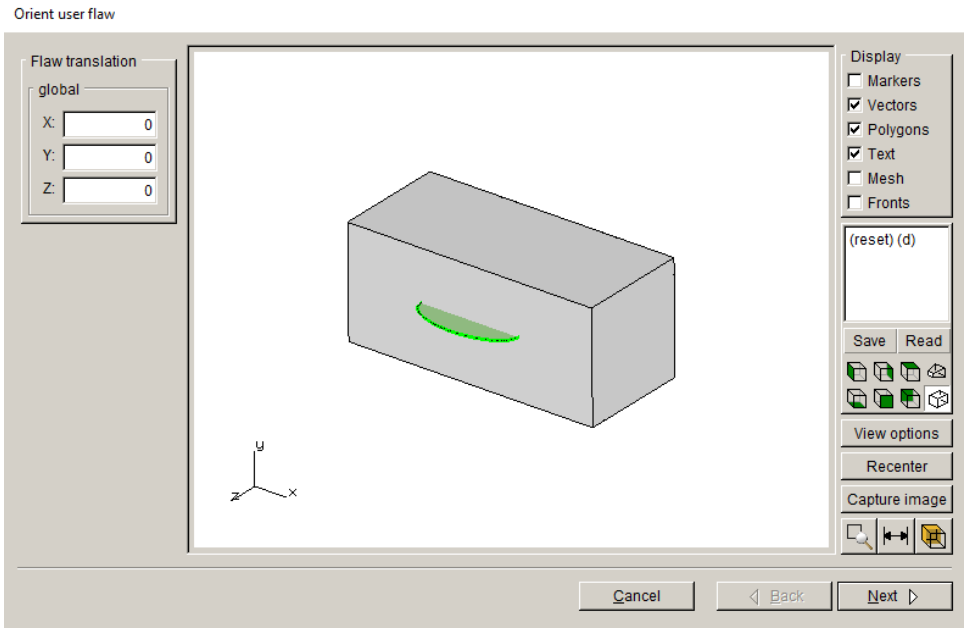


Figure B-5 Elliptical surface crack inserted into the local model.

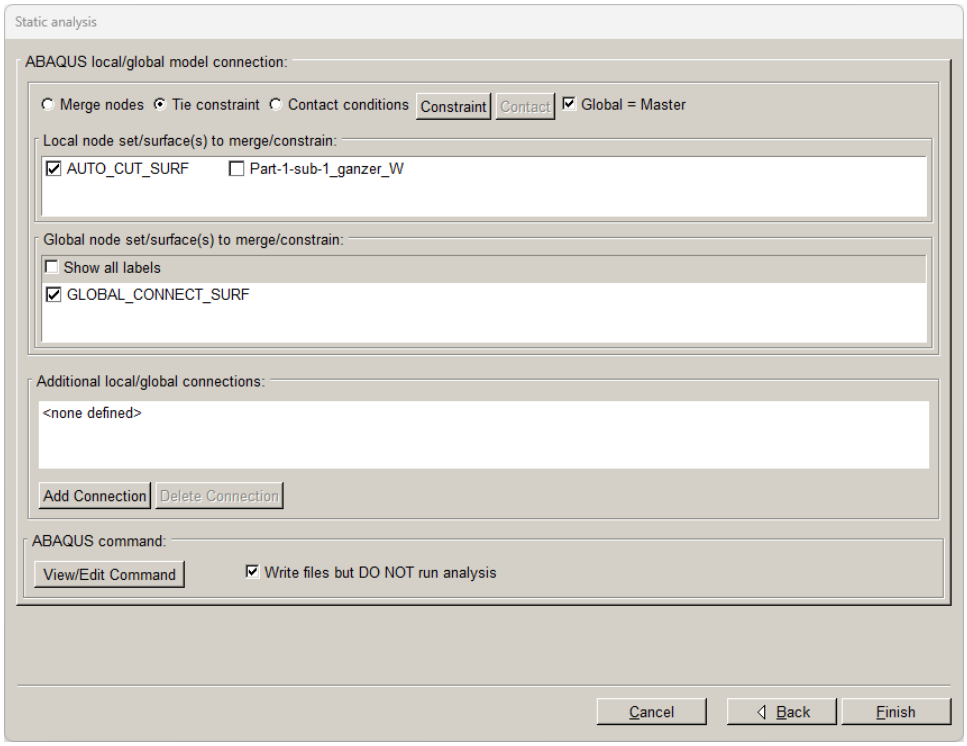


Figure B-6 Local + global model connection.

The resulting Mode I SIFs for the crack are shown in Fig B-7. The displacement magnitude contours for the original uncracked and the cracked model are shown in Fig B-8 and Fig B-9. The results match well; the results might be improved by using quadratic elements in the original model.

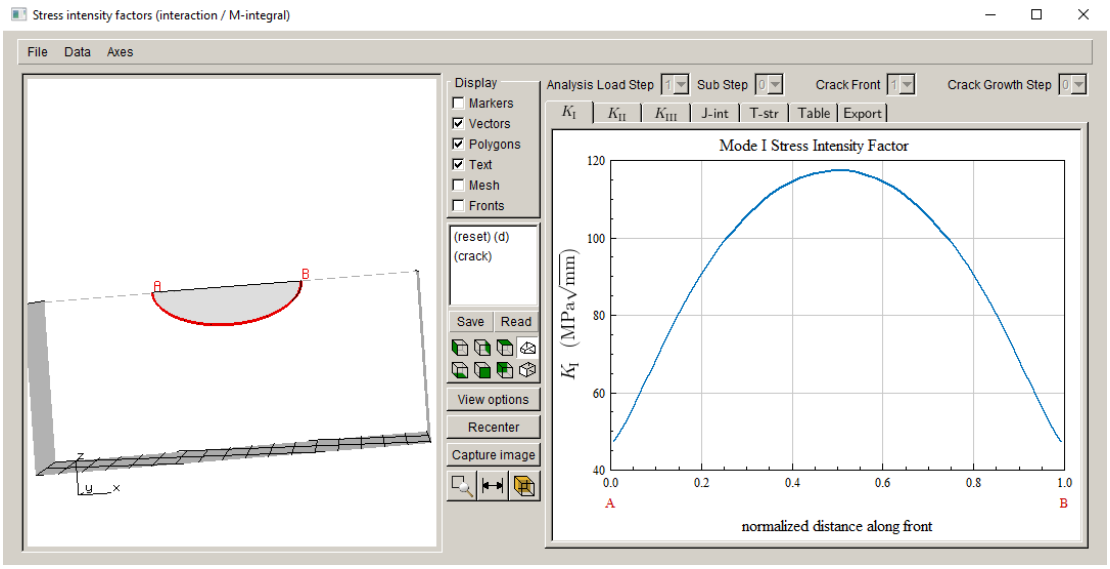


Figure B-7 Initial crack Mode I SIFs.

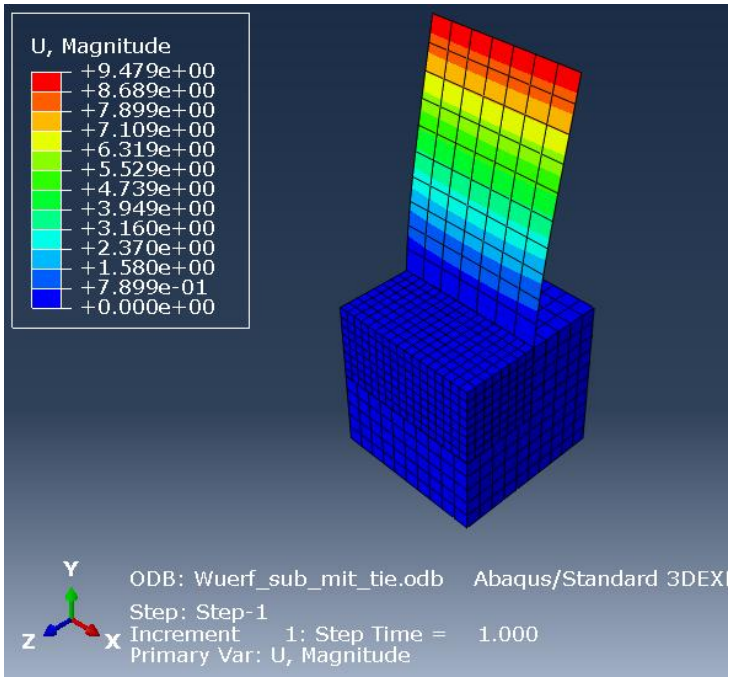


Figure B-8 Uncracked model displacement contours.

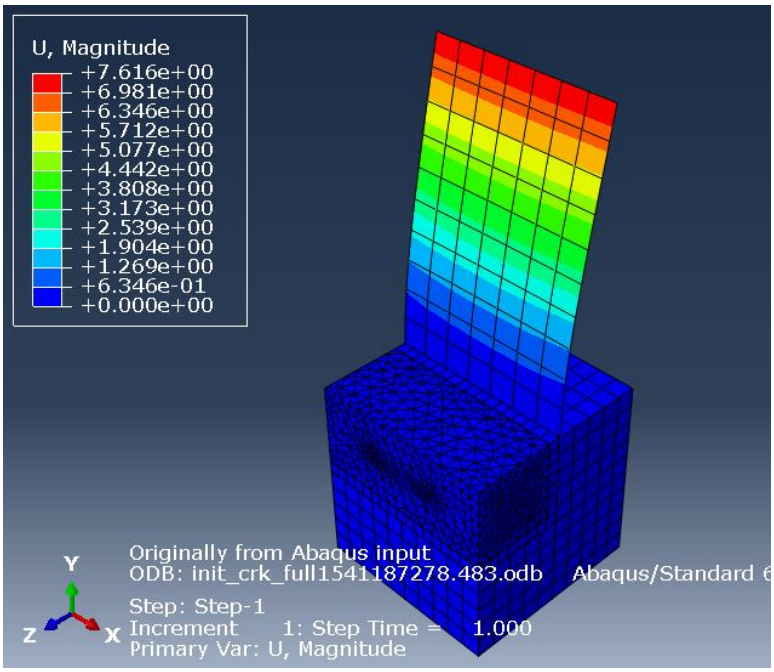


Figure B-9 Cracked model displacement contours.

Appendix C: ABAQUS Keywords

The list of ABAQUS keywords that FRANC3D recognizes is provided here. FRANC3D must process some of the data, but a large amount of the *.inp* file can be passed through to the *_GLOBAL.inp* and then to the *_full.inp*, which is the combination of the cracked/remeshed portion of the model and the *_GLOBAL.inp* portion.

```
static const char* reserved[] = {
   "*AMPLITUDE",
   "*ASSEMBLY",
   "*BEAM SECTION",
   "*BEAM GENERAL SECTION",
   "*BOUNDARY",
   "*CLEARANCE",
   "*CLOAD",
   "*CONDUCTIVITY",
   "*CONTACT PROPERTY ASSIGNMENT",
   "*CONTACT INCLUSIONS",
   "*CONTACT CONTROLS",
   "*CONTACT PAIR",
   "*CONTACT OUTPUT",
   "*CONTACT",
   "*CONTOUR INTEGRAL",
   "*CONTROLS",
   "*COUPLED THERMAL-ELECTRICAL",
   "*COUPLING",
   "*CYCLIC HARDENING",
   "*CYCLIC SYMMETRY MODEL",
   "*CYCLIC",
   "*DAMAGE EVOLUTION",
   "*DAMAGE INITIATION",
   "*DAMAGE STABILIZATION",
   "*DENSITY",
   "*DEPVAR",
   "*DLOAD",
   "*DSECURRENT",
   "*DSLOAD",
   "*EL FILE",
   "*EL PRINT",
   "*ELASTIC",
   "*ELECTRICAL CONDUCTIVITY",
   "*ELEMENT",
   "*ELEMENT OUTPUT",
   "*ELSET",
   "*END ASSEMBLY",
   "*END INSTANCE",
   "*END LOAD CASE",
   "*END PART",
   "*END STEP",
```

"*ENDSTEP",
"*END",
"*EQUATION",
"*EXPANSION",
"*FIELD",
"*FILE FORMAT",
"*FREQUENCY",
"*FRICTION",
"*GLOBAL DAMPING",
"*HEADING",
"*HYPERELASTIC",
"*HYPOELASTIC",
"*INCLUDE",
"*INITIAL CONDITIONS",
"*INITIAL CONDITION",
"*INITIAL COND",
"*INSTANCE",
"*KINEMATIC COUPLING",
"*LOAD CASE",
"*MASS",
"*MATERIAL",
"*MODAL DYNAMIC",
"*MODAL OUTPUT",
"*MODEL CHANGE",
"*MPC",
"*NCOPY",
"*NODE",
"*NODE FILE",
"*NODE PRINT",
"*NORMAL",
"*NSET",
"*ORIENTATION",
"*OUTPUT",
"*PART",
"*PLASTIC",
"*POROUS BULK MODULI",
"*POROUS ELASTIC",
"*POROUS FAILURE CRITERIA",
"*POROUS METAL PLASTICITY",
"*PRE-TENSION SECTION",
"*PREPRINT",
"*RESTART",
"*RIGID BODY",
"*ROTARY INERTIA",
"*SELECT EIGENMODES",
"*SHEAR FAILURE",
"*SOLID SECTION",
"*SPECIFIC HEAT",
"*SPRING",
"*STEP",
"*SUBMODEL",
"*SURFACE BEHAVIOR",
"*SURFACE INTERACTION",
"*SURFACE",
"*SYSTEM",

"*TEMPERATURE",
"*TIE",
"*TRANSFORM",
"*USER MATERIAL",
"ABSOLUTE EXTERIOR TOLERANCE",
"ADJUST",
"AMPLITUDE",
"BINC",
"BSTEP",
"CHANGE NUMBER",
"CONSTRAINT NAME",
"CONVERT SDI",
"CONTROLS",
"COORDINATES",
"CYCLIC SYMMETRY",
"CYLINDRICAL",
"DEFINITION",
"DISPLACEMENT",
"ELEMENT",
"ELSET",
"EXTERIOR TOLERANCE",
"FILE",
"FOLLOWER",
"GENERATE",
"INC",
"INPUT",
"INSTANCE",
"INTERPOLATE",
"ITERATIVE",
"MASTER",
"MATERIAL",
"MIDSIDE",
"MOD",
"NAME",
"NEW SET",
"NEW",
"NLGEOM",
"NO",
"NODE",
"NSET",
"OLD SET",
"OP",
"ORIENTATION",
"PART",
"PERTURBATION",
"PIN SET",
"POSITION TOLERANCE",
"RAMP",
"REF NODE",
"SHIFT",
"SLAVE",
"SOLVER",
"SPHERICAL",
"SPRING",
"STEP",

```
    "STRESS",
    "SUBMODEL",
    "SURFACE",
    "SYSTEM",
    "TEMPERATURE",
    "TIME",
    "TYPE",
    "UNSORTED",
    "USER",
    "VALUE",
    "VARIABLE",
    "VARIABLES",
    "YES",
    "Z RECTANGULAR",
} ;
```