

INTRODUCTION

This tutorial provides step by step instructions to begin cohesive crack modeling using FRANC2D and the defined crack path strategy. A familiarity with CASCA and FRANC2D is assumed in the presentation of information. For information on the basics of using CASCA and FRANC2D, please refer to the FRANC2D primer.

Tulio Bittencourt performed much of the work to allow cohesive crack modeling in FRANC2D as part of his Ph.D. research. For more technical information and background on the cohesive crack modeling capabilities in FRANC2D, please refer to his thesis,

Bittencourt, T. N., 1993, "Computer Simulation of Linear and Nonlinear Crack Propagation in Cementitious Materials," Ph. D. Dissertation, Department of Civil and Environmental Engineering, Cornell University, Ithaca, NY.

Although Bittencourt describes a few strategies for solving cohesive crack problems with FRANC2D, this tutorial only focuses on the "Defined Crack Path Strategy". The limitation of this strategy is that the crack path must be defined before the simulation begins. For computational efficiency, the program uses a "dynamic relaxation" solution scheme that is explained in detail in the thesis.

The cohesive capabilities in FRANC2D are applicable to any material exhibiting cohesive crack behavior. In this tutorial, cohesive crack behavior means that once the stress ahead of the crack tip reaches a limiting tensile value, stress is transferred across the crack according to a function of the crack opening. This generic model can be considered a general Dugdale-Barenblatt model.

To illustrate the steps required to implement the defined crack path strategy in FRANC2D, this tutorial focuses on a concrete single edge notched beam in three point bending (Figure 1).

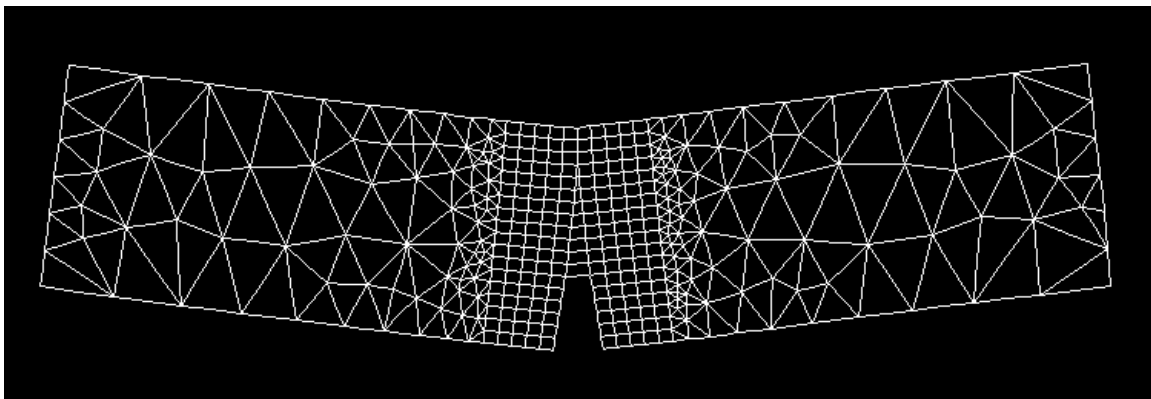


Figure 1. Single edge notched beam.

At this time, the cohesive cracking aspects of FRANC2D are not very user friendly. Until this is updated, follow the steps outlined in this tutorial closely. The "NOTE:" comments relate to especially important information to avoid problems.

Save the model often, and use different checkpoint names so that you can go back to a particular step if there is an irrecoverable mistake.

MODELING THE BEAM

Use the CASCA program to generate the geometry and mesh information for the model. Ensure that the crack path consists of element boundaries. If the path is a straight line, a "subregion" line will ensure that the path is a boundary between finite elements.

For the beam in this example, the dimensions are as follows:

$$\begin{array}{ll} L = 27.5 & d = 6.0 \\ S = 24.0 & a_o = 2.0 \\ b = 3.125 & t = 0.125 \end{array}$$

FRANC2D uses the zoom level in the main program window to determine the tolerance distance for nearest nodes and edges. Because of this, I initially had difficulty generating two different points to be the tip of the notch. By zooming in, I was able to create the two points by typing in coordinates.

The mesh used for this model is shown in Figure 2. Once the model is meshed, you can generate the file to be read into FRANC2D.

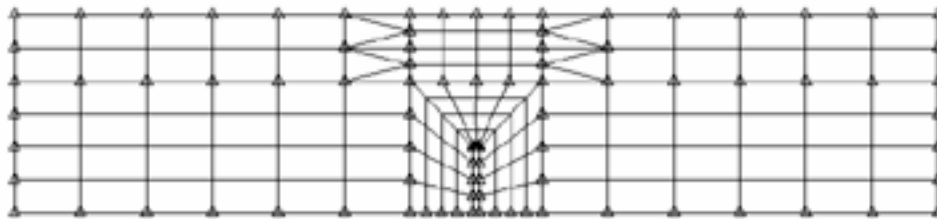


Figure 2. Single edge beam mesh.

FRANC2D PRE-PROCESSING

The main menu that displays when FRANC2D starts is shown in Figure 3a. From this menu, select the "Pre-process" button. The menu in Figure 3b will appear.

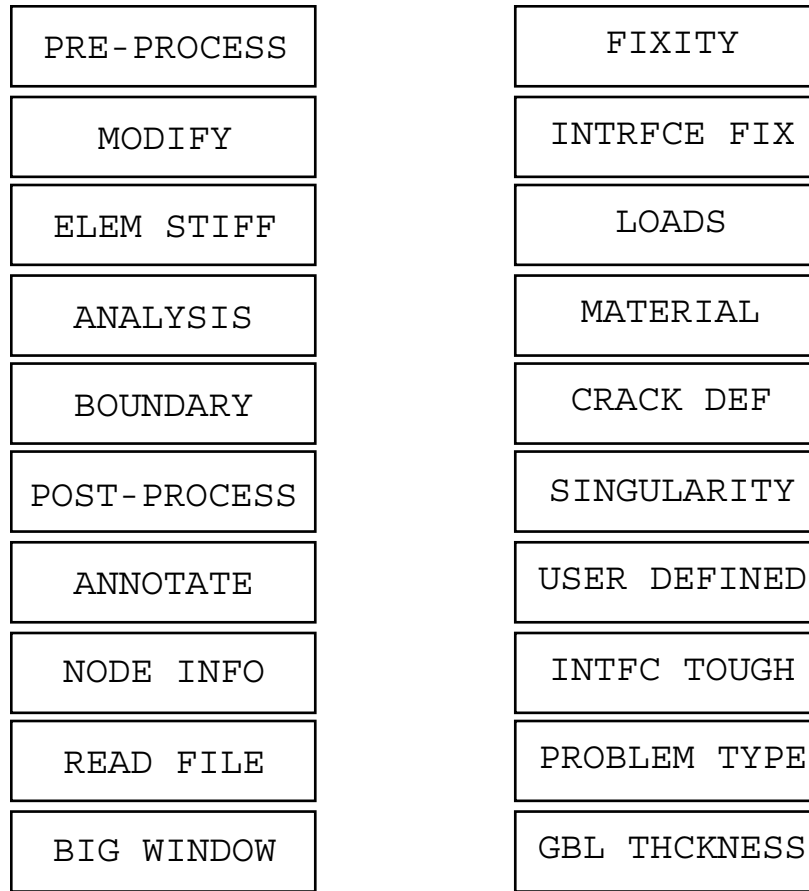


Figure 3. (a) Main menu. (b) Pre-process menu.

The first two pre-processing tasks are set the boundary conditions and input the material properties. It does not matter which task comes first. The boundary conditions are set through the "Fixity" menu. This is described in the FRANC2D primer. The material properties are set through the "Material" menu.

After selecting the "Material" button, the menu changes to the one shown in Figure 4 and an auxiliary window activates. By default the model has the properties of Material 1. By selecting the "E", "Nu" and "Thickness" buttons, you can change the properties of Material 1.

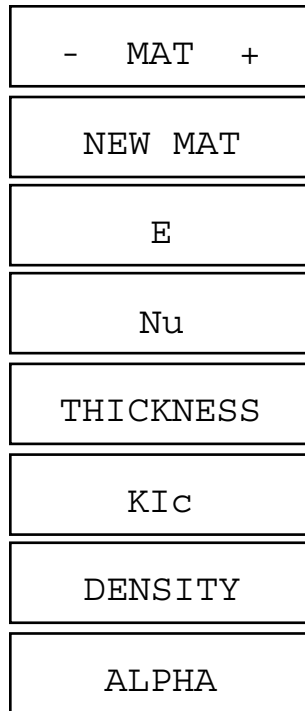


Figure 4. Material menu.

For this example, I set the properties to $E = 4e6$, $Nu = 0.18$ and Thickness = 3.125. Once I set the properties, the text in the auxiliary window read as follows.

```
Total number of materials : 1
Material number : 1
Material type : Isotropic
  Young's Modulus      : 0.400E+07
  Poisson ratio        : 0.180
  Thickness            : 3.13
  KIc                  : 1.00
  Density              : 0.868E-1
  Alpha                : 0.000E+00
```

For the cohesive interface, we generate a new material by selecting the "New Mat" button. The new material menu is shown in Figure 5a. From it, select the "NL Interface" button. The non-linear interface menu is shown in Figure 5b.

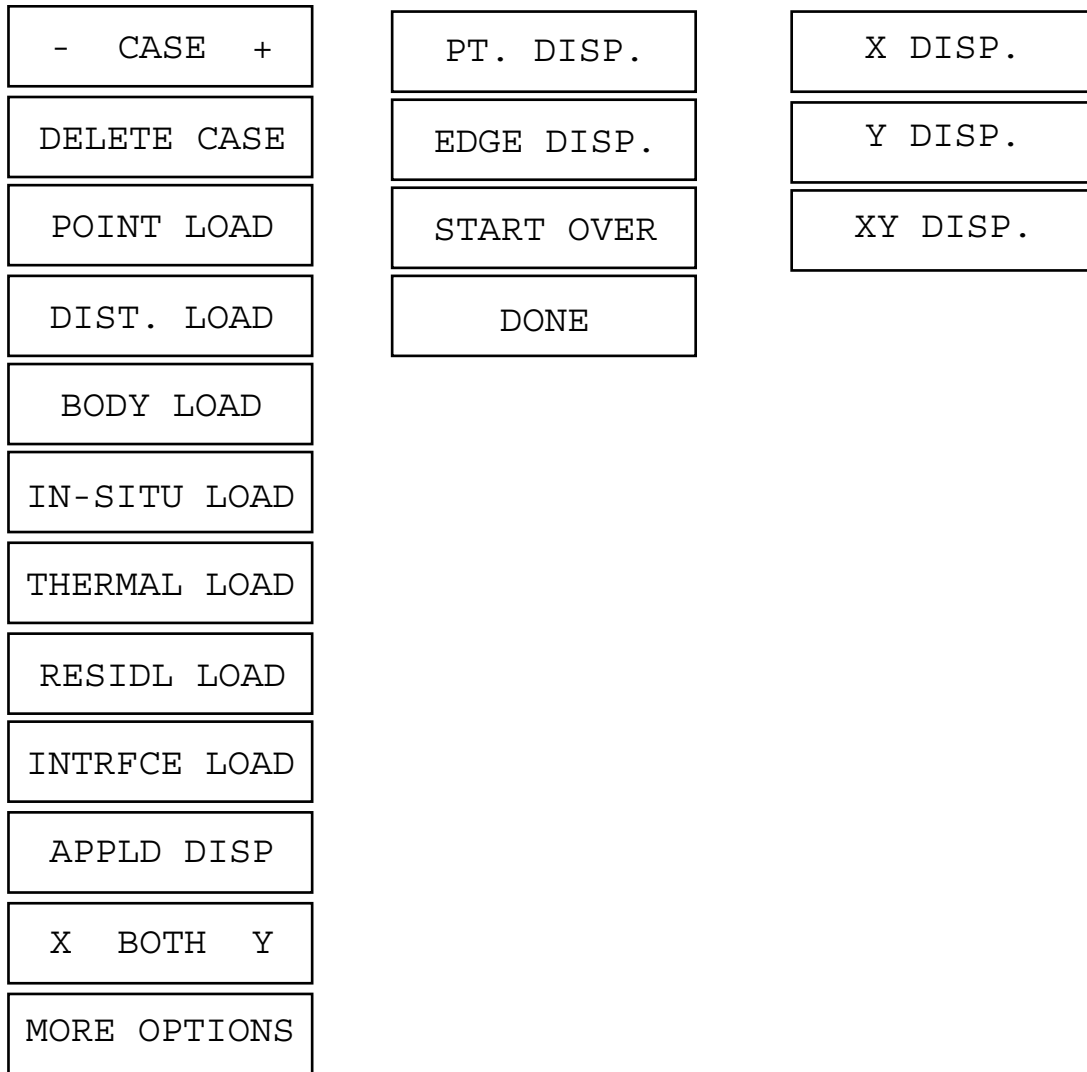


Figure 7. (a) Loads menu. (b) Applied displacement menu. (c) Point displacement menu.

NOTE: The applied displacement must be the 1st load case.

NOTE: If you need to change the location of an applied displacement or otherwise need to remove an applied displacement, you must also go back to the "Fixity" menu, remove the fixities and reapply them.

NOTE: The program converts the applied displacement to nodal equivalent loads. Therefore, the material properties must be specified before applying the displacement.

The solution scheme used in FRANC2D does not "propagate" the crack. The dynamic solver finds the equilibrium configuration for the model for the applied displacement. To "propagate" the crack, apply a larger displacement and rerun the analysis. This method can be used to generate points on the load versus displacement or load versus CMOD graph. The solution to each applied displacement is one data point.

ADD THE NL INTERFACE

Once the pre-processing is complete, we are ready to add the non-linear interface. From the main menu, select the "Modify" button. The modify menu is shown in Figure 8. From the modify menu, select the "Add NL Intfc" button.

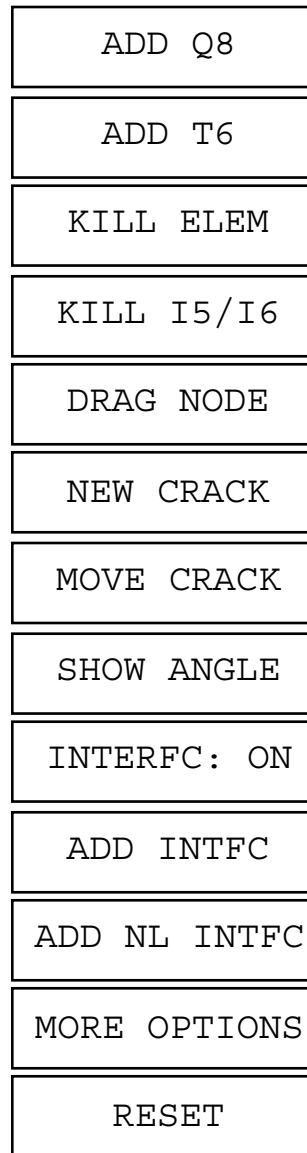


Figure 8. Modify menu.

This is the most temperamental part of the program. Follow these next instructions exactly. Input the interface one element at a time. Begin at the outer edge of the model and build the elements in the direction that the crack will grow.

After selecting the "Add NL Intfc" button, the program window will give you the command to "Select the first node of the first int elem". Select the first node of the interface. It must be the node at the outer face where the crack begins (node "A" in Fig. 9). Hit the "Done" button.

The program window will give you the command to "Specify end node of elem & hit DON". Select the node at the other end of the first element (node "B" in Fig. 9). Hit the "Done" button.

The program window will give you the command "To end interface, specify previous nod". Select the same node you just selected (node "C" in Fig. 9). "Previous node" means the second node. Hit the "Done" button. If everything worked correctly, you will be automatically taken to the "Modify" menu again.

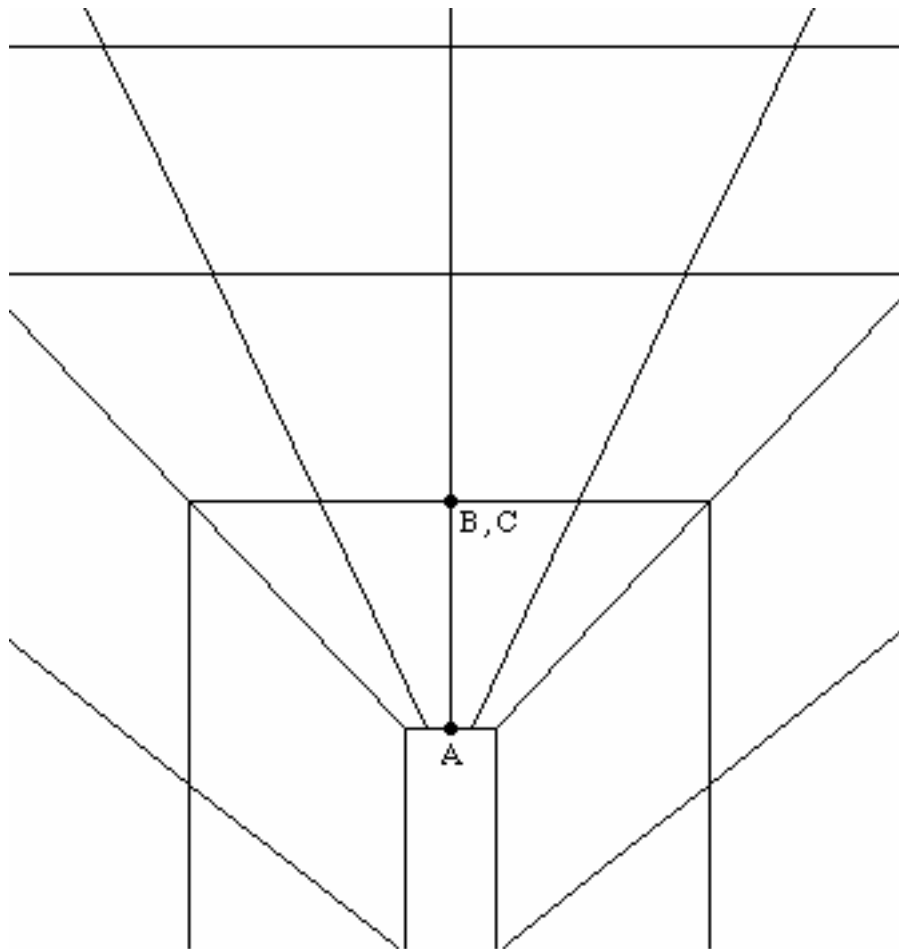


Figure 9. Sequence of node selection for the first NL interface element.

From here you can perform the same process over and over to extend the interface one element at a time. You must maintain the direction of insertion; the first node chosen for any element must be the one closest to the surface node where the crack begins (Figure 10).

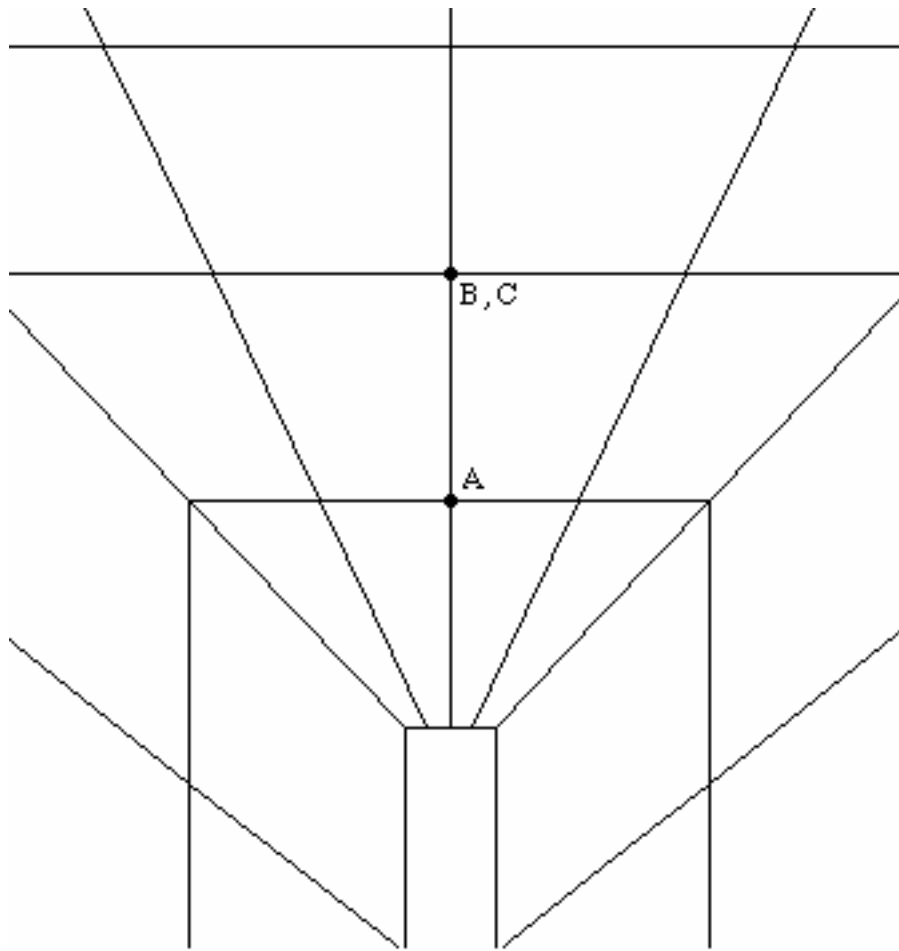


Figure 10. Node selection sequence for subsequent NL interface elements.

To check that the interface has been inserted, return to the "Main" menu and select the "Boundary" button. This button toggles off the mesh. The nonlinear interface should be shown as a thin line inside the geometry borders (Fig. 11). You may select the "Mesh" button to toggle the mesh back on.



Figure 10. Correctly inserted interface shown by toggling off the "Boundary" button.

NOTE: Do not put interface elements through the entire height of the model such that the last element reaches a free surface. This may become unstable.

Once the you insert the interface, it remains. Therefore, you do not need to replace the interface after changing the applied displacement for subsequent solutions.

ANALYSIS

To begin solving the model, select the "Analysis" button from the main menu. From the analysis menu (Fig. 11a) select the "Linear" button. From the linear solution menu (Fig. 11b) select the "Dyn Relax" button.

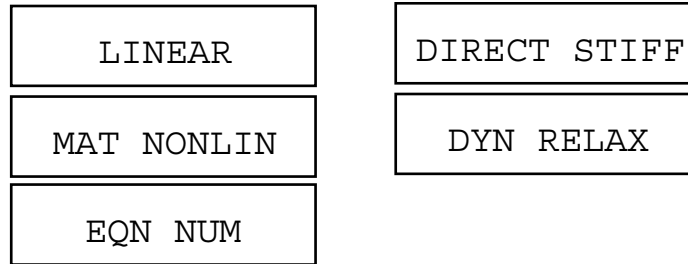


Figure 11. (a) Analysis menu. (b) Linear menu.

The program provides the following cues in the terminal window and waits for your responses. Once you have input the solver criteria, the program begins the solution. Once the solution residual becomes less than the tolerance you input or the solver exceeds the maximum number of iterations you specified, the program provides a summary of the solution.

DYNAMIC RELAXATION ANALYSIS:

```

Enter the tolerance (fraction of appl. load):
1e-3
Enter the max number of iterations:
5000
  
```

```

Starting iterations:
Iteration Max Acc*Mass Residual Norm Target Max delta u
100      242.7      239.2      18.47      0.1937E-05
200      88.37      87.67      18.47      0.9850E-06
300      57.52      57.73      18.47      0.6182E-06
400      68.93      68.96      18.47      0.4420E-06
500      68.50      68.47      18.47      0.3533E-06
600      64.69      64.65      18.47      0.3232E-06
700      60.46      60.42      18.47      0.3523E-06
800      56.40      56.36      18.47      0.3799E-06
900      52.52      52.49      18.47      0.4014E-06
1000     48.77      48.74      18.47      0.4179E-06
1100     45.12      45.09      18.47      0.4301E-06
1200     41.57      41.54      18.47      0.4382E-06
1300     38.13      38.10      18.47      0.4418E-06
1400     34.81      34.77      18.47      0.4406E-06
1500     31.62      31.59      18.47      0.4345E-06
1600     28.60      28.57      18.47      0.4249E-06
1700     25.75      25.73      18.47      0.4097E-06
1800     23.08      23.05      18.47      0.3907E-06
1900     20.59      20.56      18.47      0.3688E-06
Converged in      1993 iterations
Load Residual Norm = 20.56
  
```

```

*****
*****
  
```

FRANC Analysis Report

2286 Equations
0 Nonlin Interface Eq.

Total Time (inc overhead) : 27 seconds

Analysis done

If the dynamic relaxation analysis crashes with a "floating error" or the residual error keeps growing while the solver is working, the "shear" stiffness is probably too high. Try a lower value. You do not need to rebuild the interface, just change the material property.

If the dynamic relaxation analysis oscillates and does not converge or is slow to converge, the compressive stiffness of the "normal" stresses law is probably too high. Try a lower value. You do not need to rebuild the interface, just change the material property.

POST PROCESSING

To review the results, select the "Post-Process" button on the main menu. Select "Deformed Mesh" to see the displaced shape. To determine the applied load, check the reactions and sum up the forces.

If you can not select the entire length of the non-linear interface to display crack opening while post processing, you may have a discontinuous interface. Regenerate the model from the last checkpoint file before adding the interface.