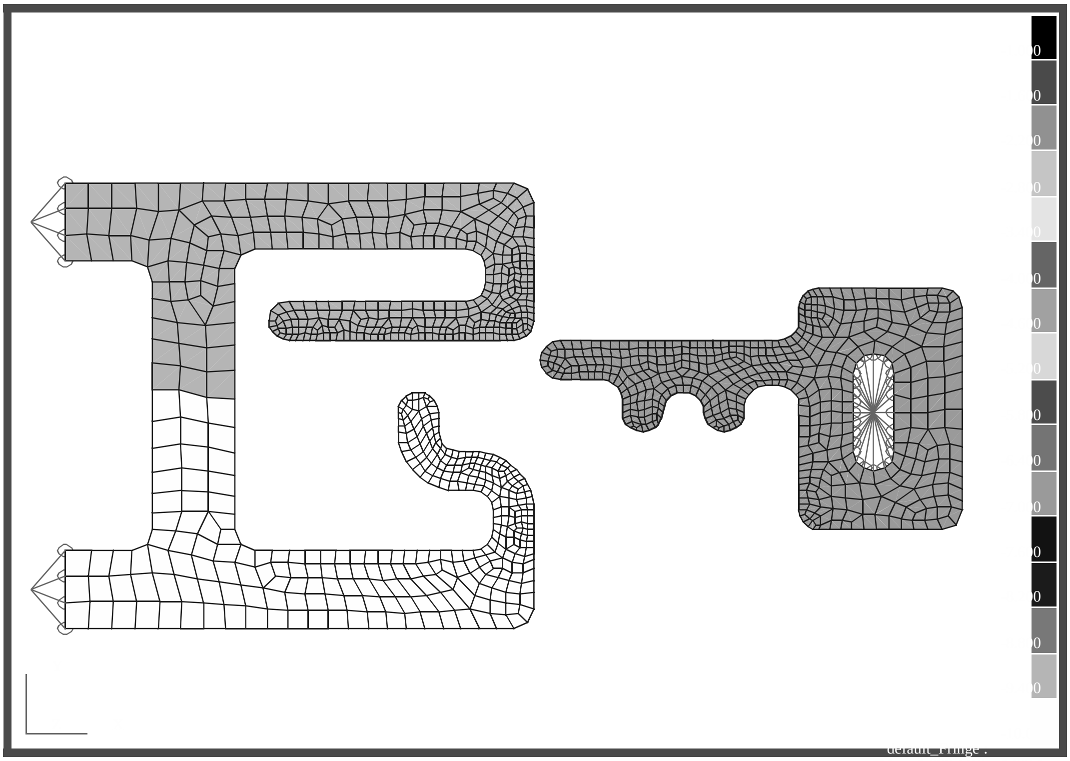
LESSON 12

Pin Insertion



Objectives:

- Resolve nonconvergence issues in to contact analysis
- Use Tools/Lists as aid to quickly modify element properties associated with contact areas.
- Use Animation tools to understand the nature of difficulties found in nonlinear static contact analysis

Model Description:

In this lesson, you will inspect a model prepared by another engineer, run the analysis as set, find that it does not run fully because of nonconvergence of the nonlinear static procedure due to difficulties in resolving contact, understand the reasons for such difficulties, modify the model so that the trouble may be overcome, run the analysis for the whole model to completion, and display the results including a full animation of insertion and extraction.

Suggested Exercise Steps:

- Recover the database from its machine-independent version file as provided.
- Inspect the database as originally setup to simulate the pin insertion and subsequent extraction.
- Run the analysis as originally setup. This job will fail to complete the extraction part of the analysis.
- Modify the model so that it will complete the whole analysis.
- Import the results and postprocess them.
- In particular run a nonlinear animation sequence exhibiting fringe plots of von Mises stresses on the deformed shapes.

Exercise Procedure:

1. Open a the database **clip.db**

File/Open ...

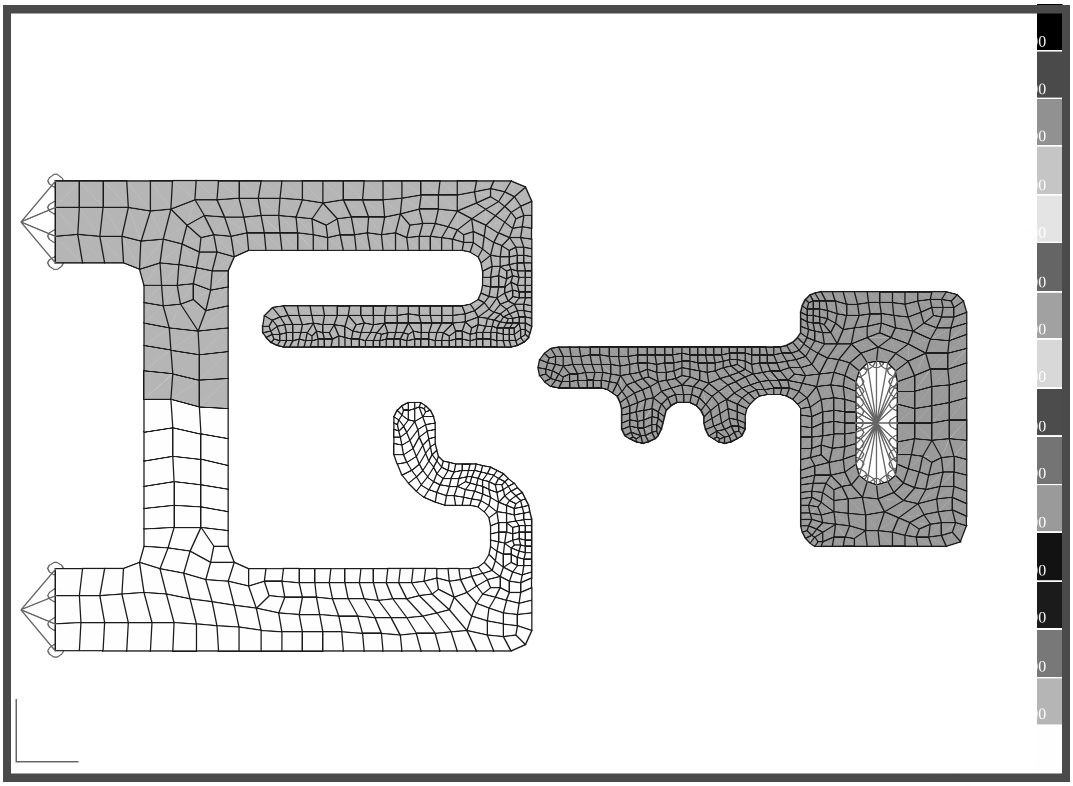
Open Database Name:	**clip.db**
OK	

2. Set up the properties by following these steps.

◆ **Properties**

Action:	**Show**
Existing Properties	**Property Set Name**
Display Method:	**Scalar Plot**
Select Groups	**fem**
Apply	

Notice the pin has one property but the clip has two different properties, one for the bottom half and one for the top half.



3. Now show the material name.

Each property is assigned a different material.

Action:	**Show**
Existing Properties	**Material Name**
Display Method:	**Scalar Plot**
Select Groups	**fem**

Apply

4. Show the thickness of the model. You can see the thickness is uniform and equal to 0.1 for all parts.

Action:	**Show**
Existing Properties	**Thickness**
Display Method:	**Scalar Plot**
Select Groups	**fem**

Apply

5. Show the MPC's of the model.

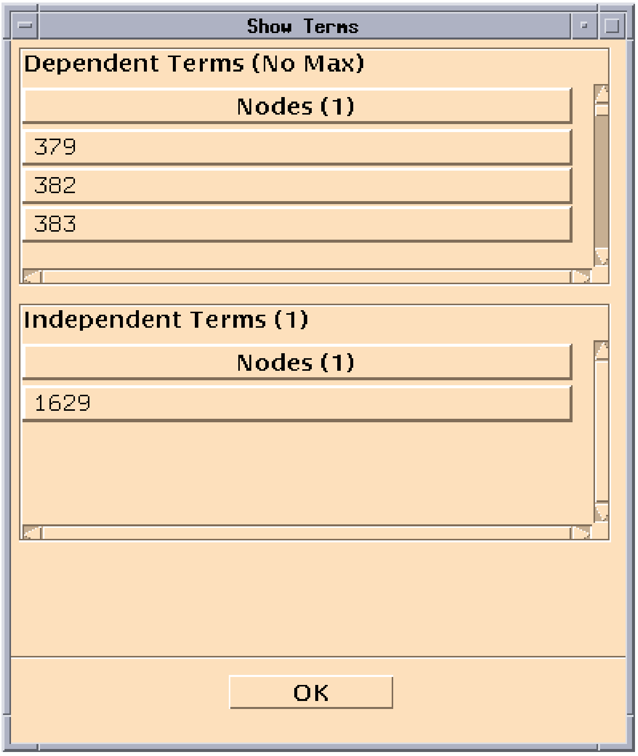
◆ **Finite Elements**

Action:	**Show**
Object:	**MPC**
MPC ID:	Select the Center of the Pin Hole

Notice this is MPC 1 and it is of the Rigid (fixed) type.

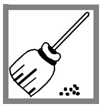
Show Terms...

This form will appear on your screen



The independent node is the one in the center. Write down the node number. This is the node that we will use to drive the pin. Similarly inspect the other two MPC's and write down the node number. The advantage of using these MPC's for providing loads and boundary conditions is that we may request reaction forces and thus obtain the forces at these three independent nodes as functions of time without having to add the forces at individual mesh nodes.

6. Click the Reset Graphics Icon.



7. Create a new group called "slines."

Group/Create...

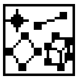
New Group Name | **slines**

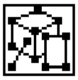
■ **Make Current**

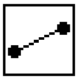
■ **Unpost All Other Groups**

Add Entity Selection

Click on the FEM Entity Icon in the select menu. Then Click on the Element Icon and finally the Beam Element Icon.





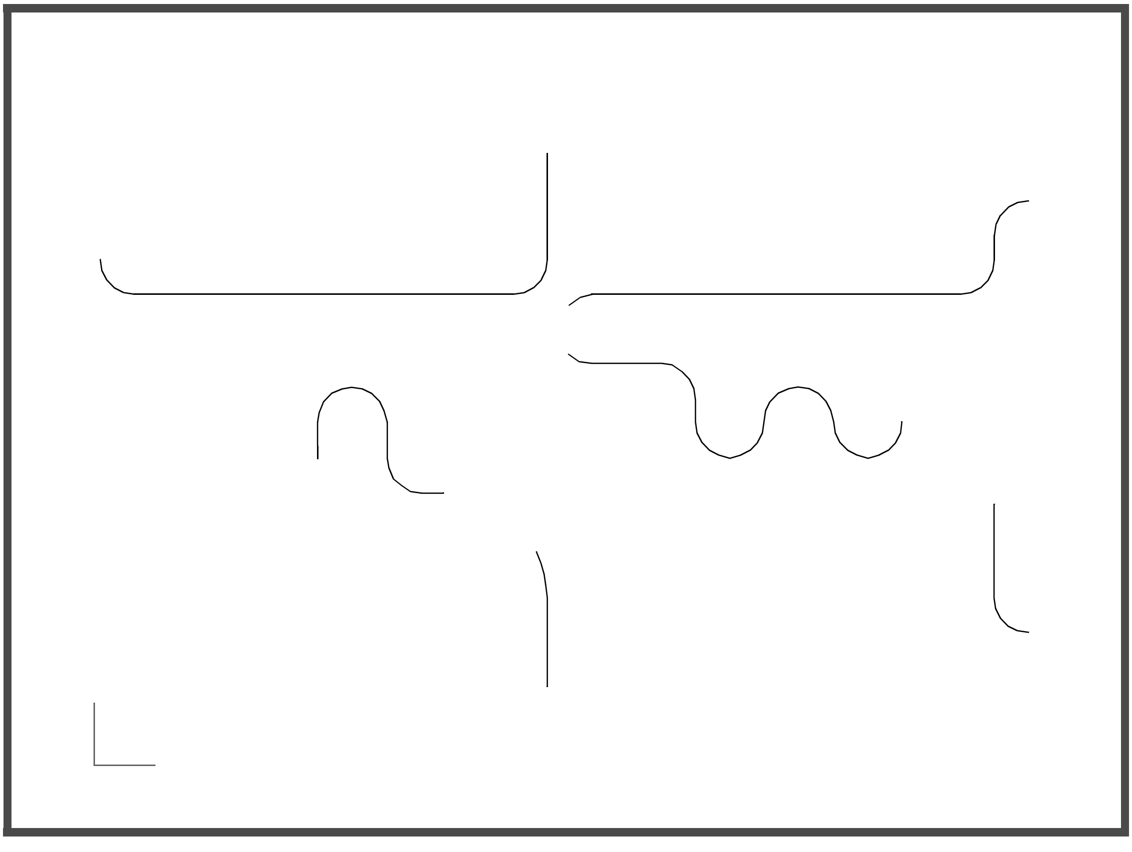


Highlight everything in the viewport using the rectangle select.

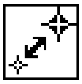
Entity Selection	Select all the beam elements

Apply

The viewport now has the slines group which has all 1D elements used to define three different slide line contact sets.



8. Click on the Node Size Icon in the Main Form. Notice that nothing happens because there are no nodes in the slines group.



9. Use the tools to associate the nodes to the slide line elements.

Tools/List/Create...

Model:	**FEM**
Object:	**Node**
Method:	**Association**
Association	**Element**
Element	Select All the Elements in the viewport

Apply

Add To Group

Apply

You should now see all the nodes in the 1D ISL and slide line elements.

10. Show the properties of the slines that were just created

◆ **Properties**

Action:	**Show**
Existing Properties	**Property Set Name**
Display Method:	**Marker Plot**
Select Groups	**slines**

Apply

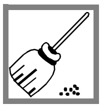
Zoom in to see the name of the property. The ones around the pin are the Slide Lines (named sl-top, sl-middle, and sl-bottom). The others are ISL elements (named isl-top, isl-middle, and isl-bottom).

11. Look at the 1D Properties to see their definitions. DO NOT click on Apply, for all you are doing is inspecting them. Notice the type of property is shown when you click on one of these properties. Notice the ELSET name pairing each slide line to one ISL set.

◆ **Properties**

Action:	**Modify**
Dimension	**1D**
Type:	**ISL (in plane)**
Select Prop. Set To Modify	Select the Property for the definitions

12. Notice the Start Point of the Slide Lines. Identify it on screen by using the Preferences/ Picking and setting both Label and Entity Highlighting to ON. Move about with your mouse (without clicking) and find where the Start Point is for each Slide Line. First **Reset Graphics**



Preferences/Picking...

■ **Label Highlighting**

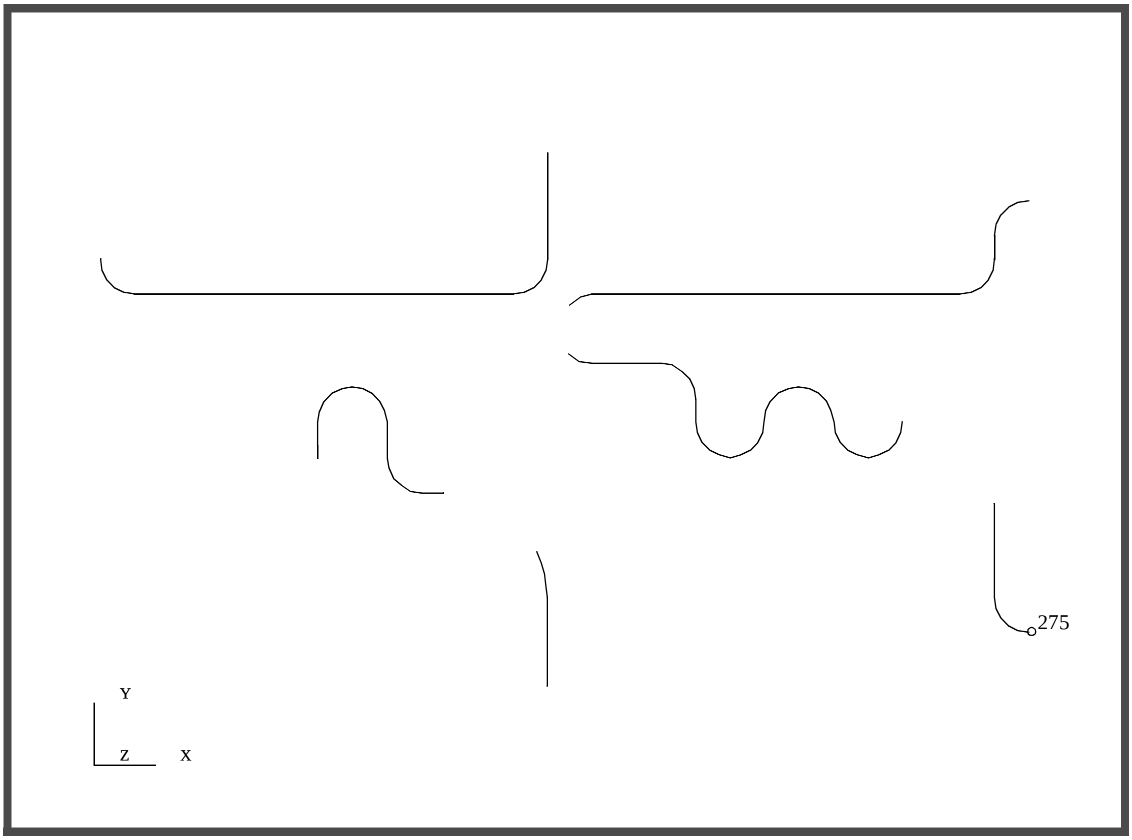
■ **Entity Highlighting**

Close

Hint 1. The start point should be at one end of the slide line.

Hint 2. This is a 2D solid model. As you move along the slide line, the body (master) the slide line bounds should be on your right, that is you should be moving clockwise around the master surface. The nodes on the ISL act as slave nodes, sliding over the slide line. As you inspect the ISL sets, notice the Width parameter has been set equal to the thickness of the plane stress, quad4 elements. This is relevant only for recovering the contact stresses.

This figure shows one such **Start Point**



Look at the contact options (Planar, Elastic Slip Hard Contact) used in this exercise. Inspect other options but do not apply.

13. Click into the properties named quads-bottom, quads-top, and quads-key.

◆ **Properties**

Action:	**Modify**
Dimension:	**2D**
Type:	**2D Solid**
Select Prop. Set to Modify:	**quads-bottom**

In all cases the options adopted are Plane Stress and Standard Formulation, but the material changes. You have seen a plot of properties at the beginning of the exercise.

14. Inspect the properties of plastic-bottom, plastic-top, and plastic-key.

◆ **Materials**

Action:	**Show**
Object:	**Isotropic**
Method:	**Tabular**
Existing Materials:	**plastic-bottom**

Show Properties ...

Notice the key and upper part have the same properties but are still defined with different names so in the future you could easily try different pin materials.

15. Post the group **fem** only.

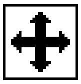
Group/Post ...

Select Groups to Post:	**fem**

Apply

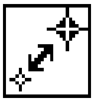
Cancel

Click on the **Fit View** button



Fit View

Click on the Node size icon so you don't see the nodes.



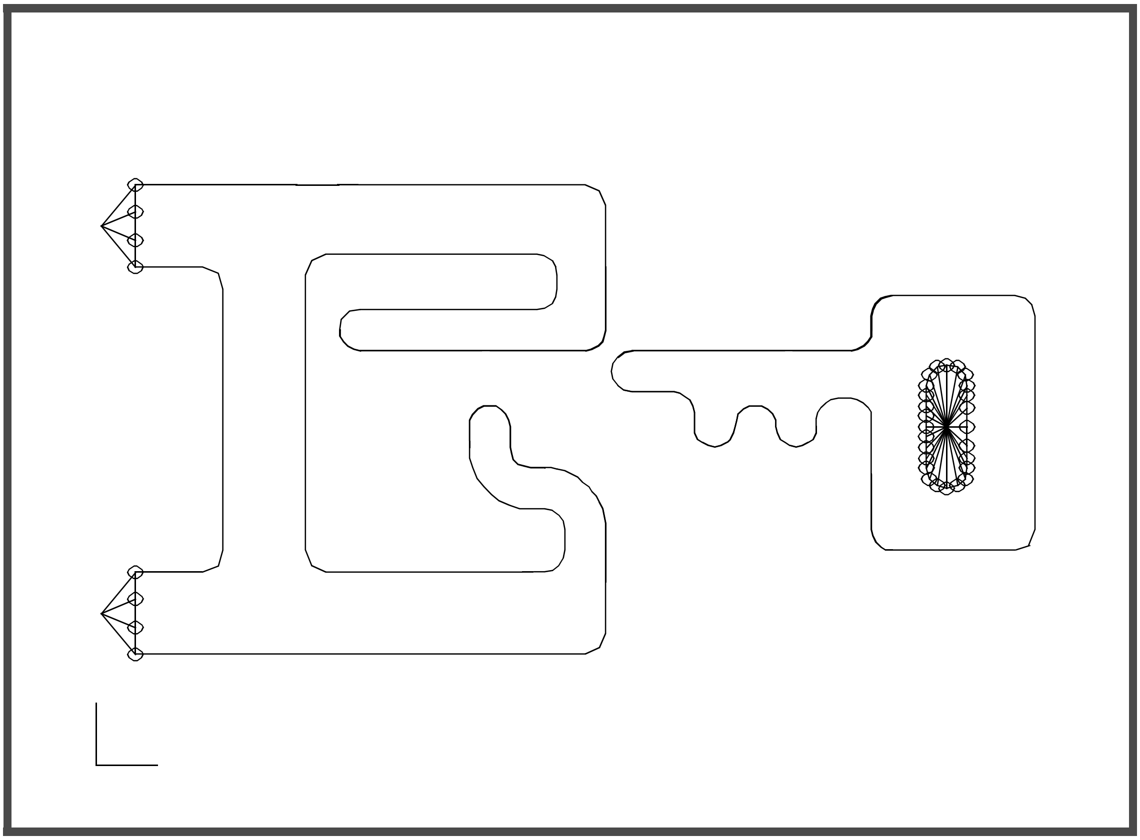
Node Size

Display/Finite Elements ...

Show Only Free	◆ **Edges**

Apply

Cancel



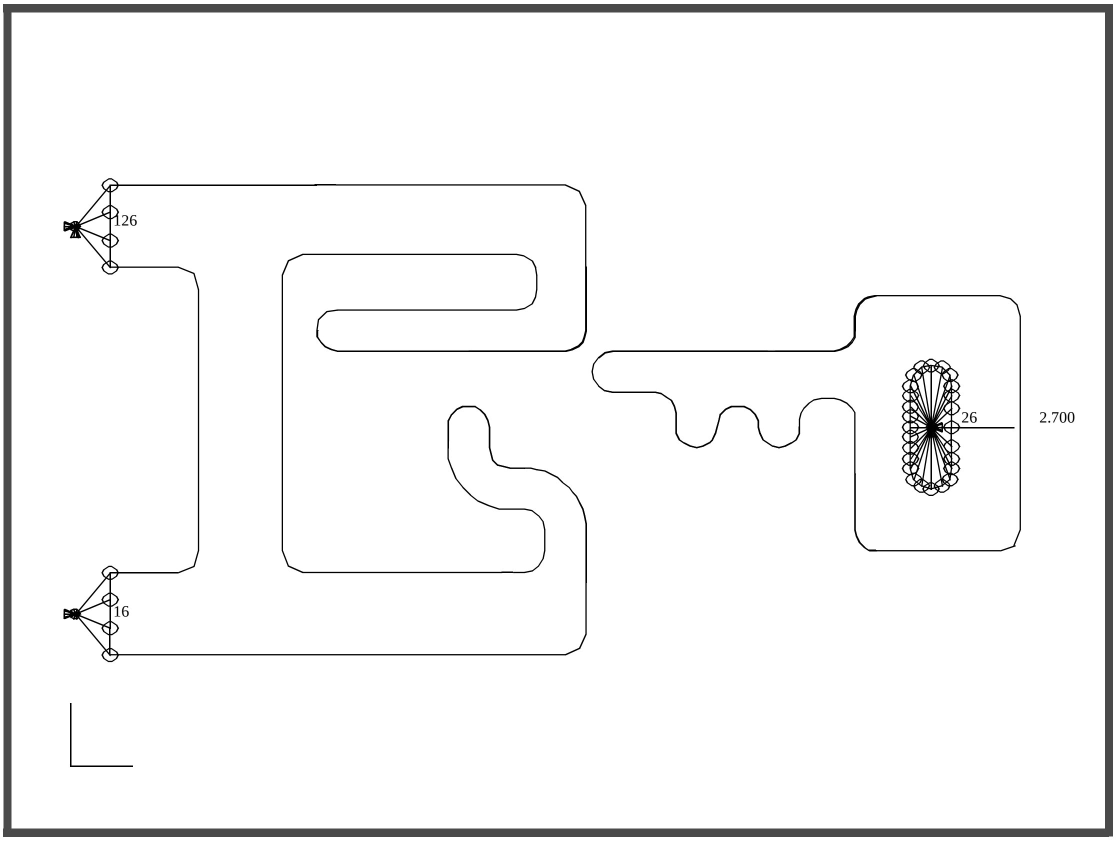
16. Plot the markers.

◆ **Load/BCs**

Action:	**Plot Markers**
Assigned Load/BC Sets:	highlight all three
Select Groups	**fem**

Apply

Your viewport appears as follows



17. Remember nodes 1629, 1630, and 1631 were the three independent nodes in the MPCs. These are the nodes on which the load/BC markers are set. Node 1629 shows a displacement marker of <-2.7, 0, 0> and the 2 and 6 DOFs restricted (fixed to be zero). Node 1630 has its 1, 2, and 6 DOFs restricted to nil, and node 1631 has its 1 and 6 DOFs restricted to nil. Look into the form: It indicates that the current Load Case is "step1".

18. Click into "step1" and change the current case to "step2".

◆ **Load/BCs**

Action:	**Plot Markers**
Current Load Case:	**step 1...**
Existing Load Cases	**step 2**

OK

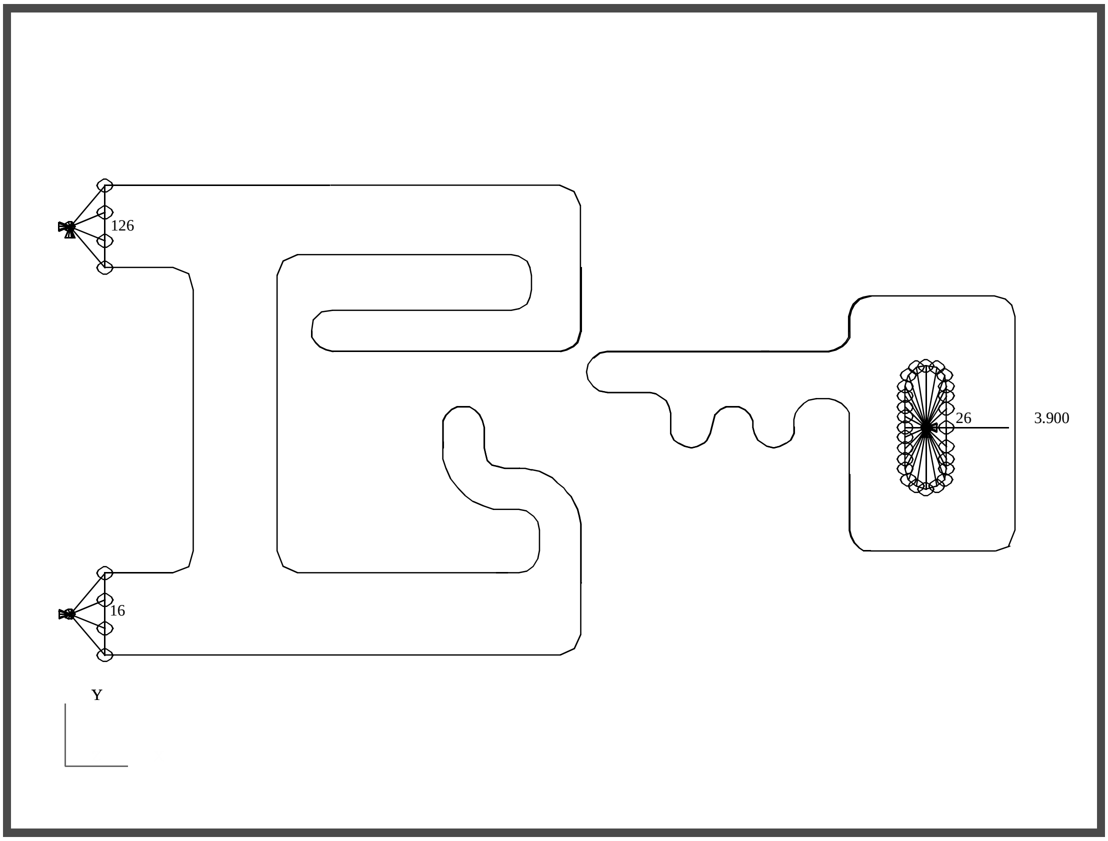
Plot markers again.

◆ **Load/BCs**

Action:	**Plot Markers**
Assigned Load/BC Sets:	highlight all three
Select Groups	**fem**

Apply

The total displacement of the key is now <-3.9, 0, 0>.



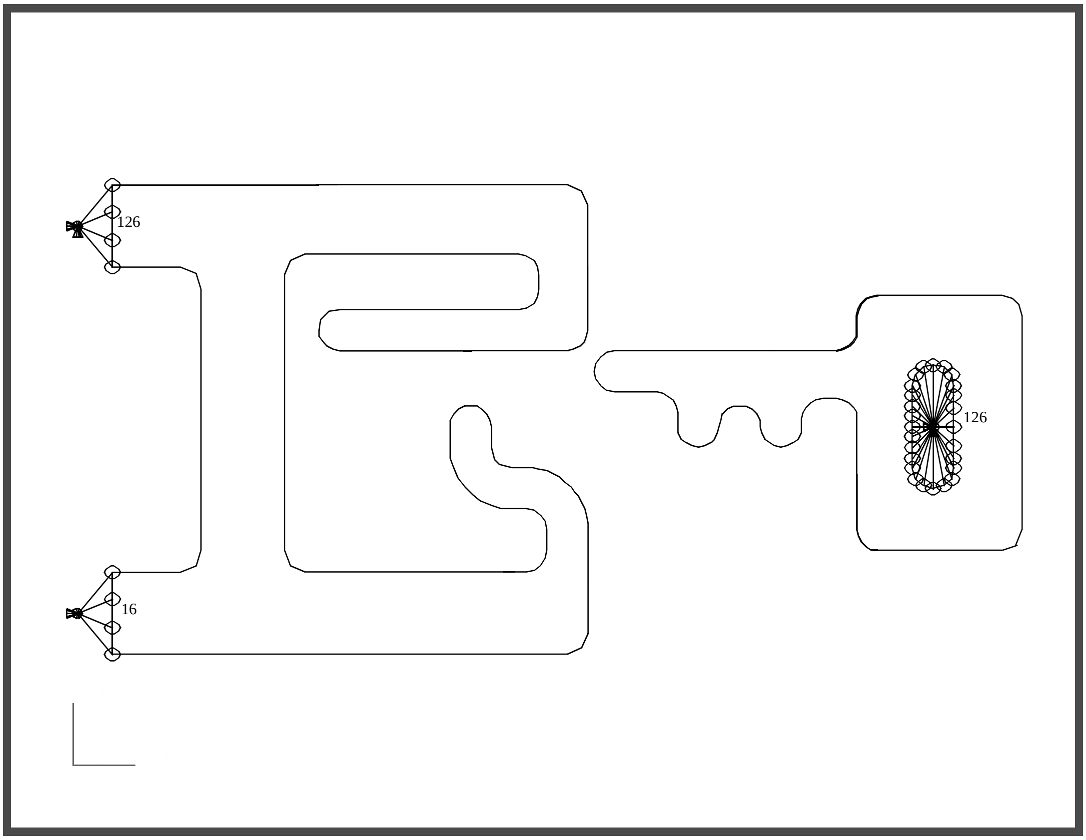
Click into "step2" and change the current case to "step3".

◆ **Load/BCs**

Action:	**Plot Markers**
Current Load Case:	**step 2...**
Existing Load Cases	**step 3**

OK

Your viewport appears as follows



The key is forced to move <0 0 0>. These are total displacements, so if we run the three step analysis with the step1, step2, step3 sequence, the key will be moved a distance of 2.7 towards the clip, then an additional distance of 1.2 in the same direction and finally a distance of 3.9 away from the clip. (Back to the original position, that is Xdisp = - 2.7 - 1.2 + 3.9 = 0).

19. Go to Analysis form and inspect the analysis setup.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Available Jobs:	**STD3**

Step Selection...

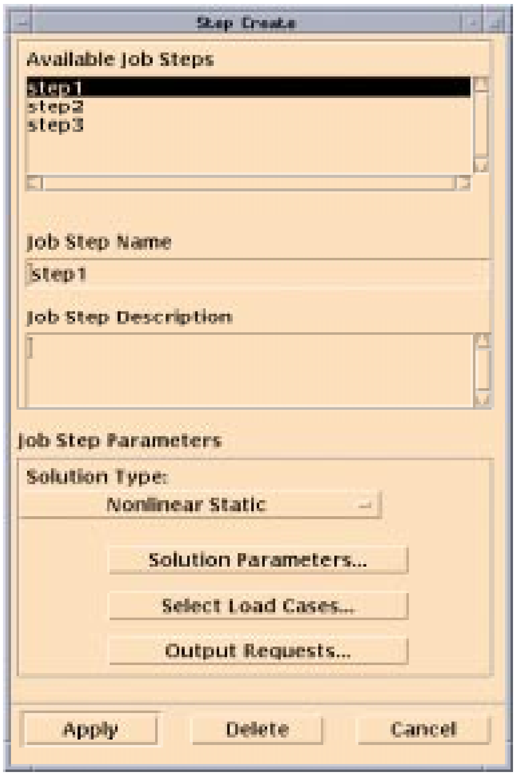
Notice that the three steps discussed above are in fact the Selected Job Steps.

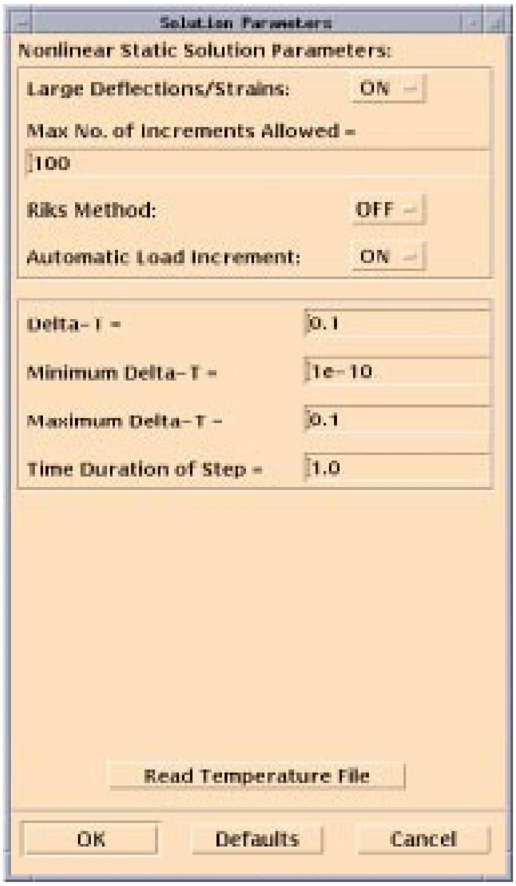
Cancel

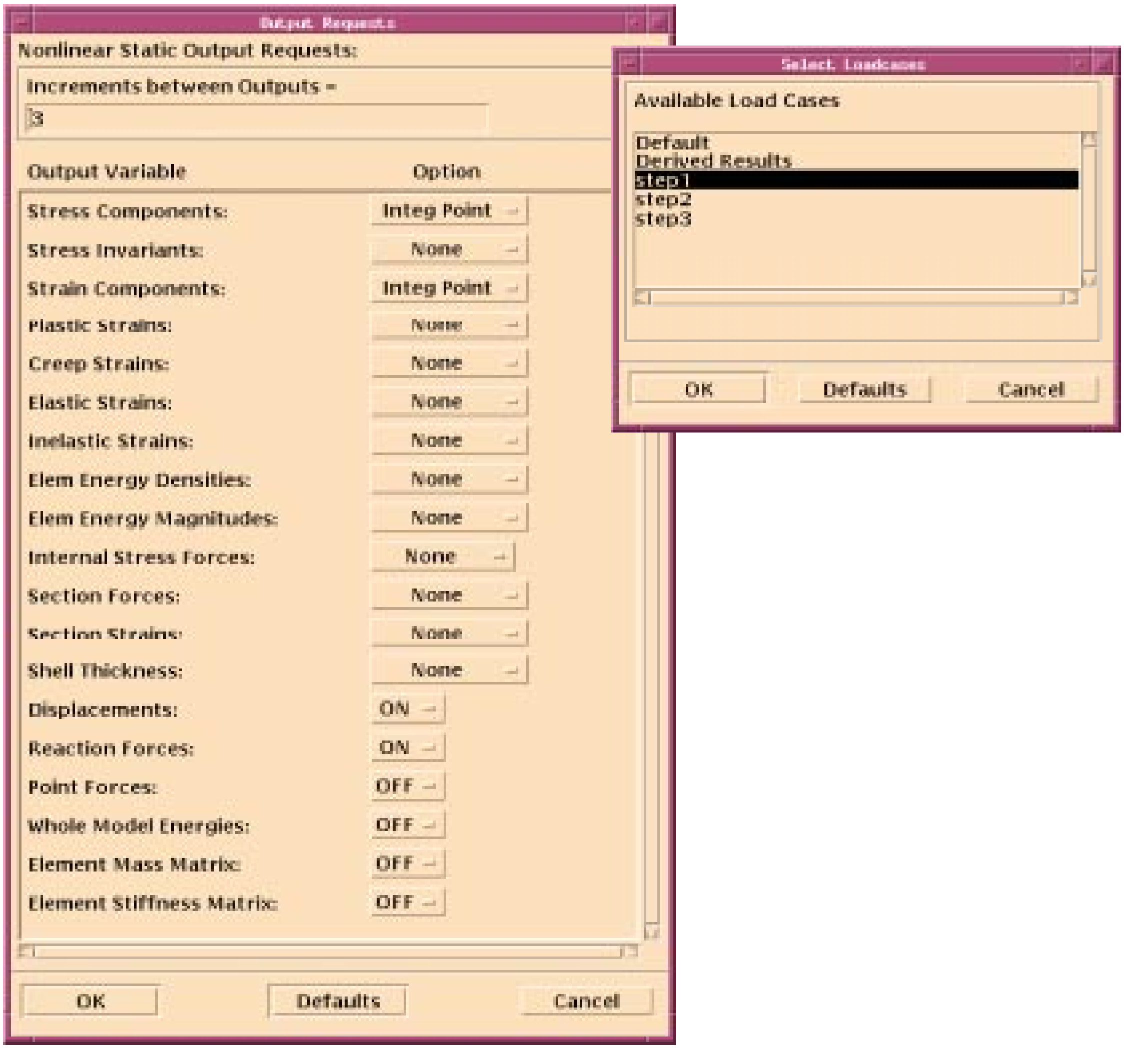
Step Creation...

Available Job Steps: **step1**

Notice the solution type switched to Nonlinear Static. Click on the three buttons: Solution Parameters, Select Loadcases, Output Request. Move the forms on the screen so you may see the whole of these three forms and the Step Create form.







You are seeing the whole setup for the step1. (The one highlighted in the Step Create form.) Inspect them, see how the time increments are defined. You may change things momentarily but please do not apply, OK, or Cancel any form. At any time you may go back to the original (not the default!) definition by clicking on "step1" in the Available Job Steps panel of the Step Create form. In fact, alternatively click on "step2" and "step3" in the same form. You will see that the highlighted load case in the Available Load Cases form changes but non of the other forms changes. This is because that is the way those job steps have been created. In fact, this is a nice way to quickly inspect all job steps you may have defined in your database for a certain preference and type. (Not all of which are in general used in a given job.) Cancel (do not apply) the Step Create form. (This closes the other, subordinate forms.)

Cancel

20. Run the analysis. Just click Apply on the Analysis form.

Apply

21. The viewport is gone while the job is launched. This particular analysis takes about 900 seconds of CPU time on an SGI Indigo2 (R10000). Therefore you will keep working on the exercise while the job runs, making modifications in the database while the analysis runs in the background
    You will see that the job fails to run to completion. It will crash while running the third, or "pull" step. Because of the shape of the pin and clip, pulling the key out in the actual, physical structure is more difficult that pushing it in. Also the numerical simulation finds the task more difficult and in fact the procedure fails to converge to the point of maximum effort when the key is being pulled out. One might try different ways of tackling the problem, like remeshing (the mesh used is rather crude,) or softening the contact, or both.

Allowing even a smaller time increment than the one attempted by the automatic incrementation algorithm will not help though. Yet in this exercise we will use a nice technique that takes into account what happens in the area of contact, namely the fact that there is a point, when the clip protrusion that has previously fallen in between the teeth of the key is about to snap out while the key is being pulled out. In the actual device, there is a sudden acceleration of this protrusion, which, if we run a Nonlinear Transient procedure instead of the attempted Nonlinear Static procedure we will help stabilize the numerical algorithm, and the mass matrix will have a larger influence precisely at that moment. However, we intend to complete our nonlinear static procedure. At this critical moment the few elements involved in the contact between one tooth in the key and the clip’s lobe have the largest rate of change of nodal displacements and therefore of stress distributions, likewise resulting in sudden changes of contact forces, leading to instability. This happens on the face of and element while the opposite face remains relatively quiet. This translates into an oscillation of the element shape associated with shear stresses, which we know Incompatible Mode (IM) elements have been made to minimize. While we would like to retain the Standard Integration type of elements because of their greater accuracy, we want these IM elements at the contact faces. What follows of the exercise takes advantage of MSC/PATRAN tools that let you easily and successfully implement this technique.

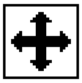
22. Post the Slines group.

Group/Post...

Action:	**Post**
Select Groups to Post	**slines**

Apply

Click on Fit View Icon in the Main Form.



23. These are the 1D elements used to define the contact. We want to change the type of elements connected to them. Clear the List A contents panel.

Tools/List/Create...

Model:	**FEM**
Object:	**Element**
Method:	**Association**
Association	**Nodes**

Click in the Node panel and rectangle pick everything in the viewport.

Node	Select All the Nodes in the viewport

Apply

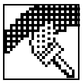
Add To Group

Existing Groups	**slines**

Apply

Cancel

Click on the Refresh Graphics Icon.



24. Show only the free edges of the elements.

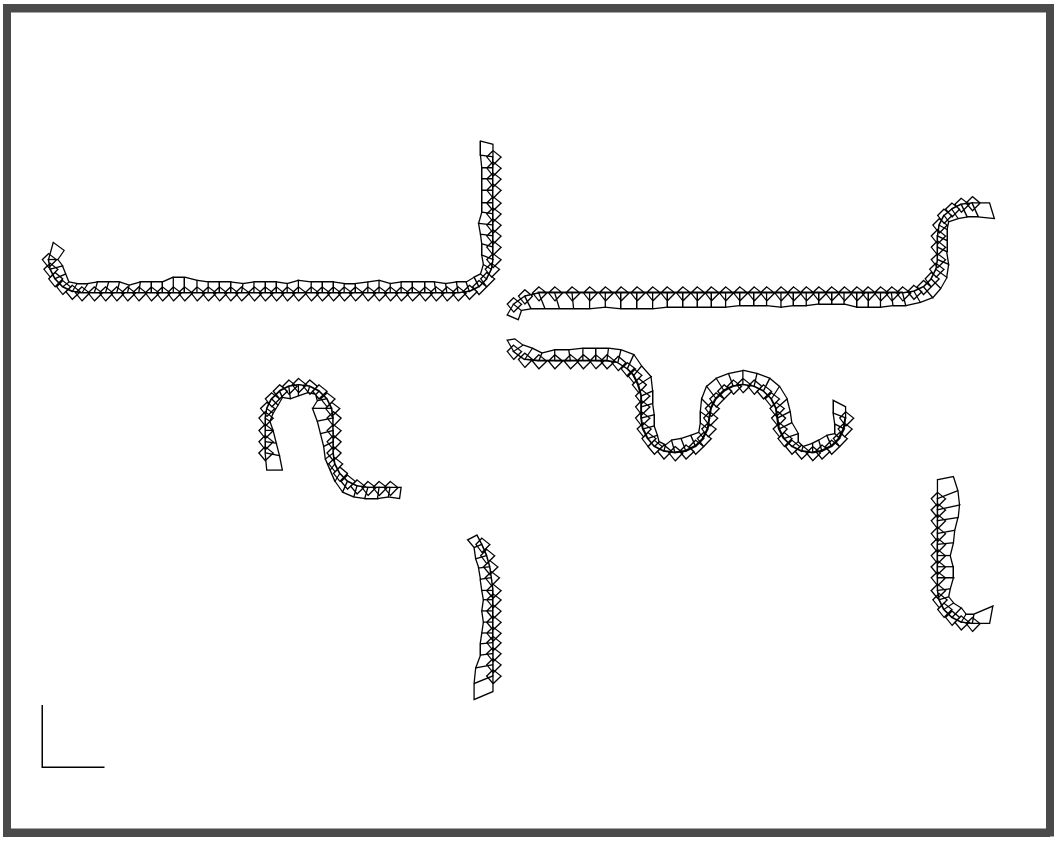
Display/Finite Elements...

Show Only Free ❑ **Edges**

Apply

Cancel

These are elements you want to transform to Incompatible Modes.



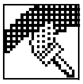
25. Create the properties for the 2D Solids.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
New Property Set Name	**quads-key-IM**
Option(s):	**Plane Stress**

	Incompatible Modes
Input Properties...	
Material Property Sets	**plastic-key**
Thickness	**0.1**
OK	

Refresh the Graphics by clicking on the Refresh Graphics Icon.



Clear the Application Region so that it appears empty. Click in the Select Members panel and rectangel pick ONLY the key elements, the two set of elements on the right half of the viewport.

Select Members	Select the members on the right side of the viewport
Add	
Apply	

MSC/PATRAN will warn you that those elements already have a property because they currently already have the Standard Integration Type. Click on the YES for All button.

Yes For All

26. Likewise create a "quads-top-IM" property with the uppermost left set of elements in the viewport, and a "quads-bottom-IM" property with the two sets on the lower left quarter of the viewport. Be careful choosing the corresponding materials, which are different for the top and bottom of the clip.

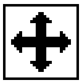
27. Post the FEM group.

Group/Post...

Action:	**Post**
Select Groups to Post	**FEM**

Apply

Now click on the Fit View Icon.



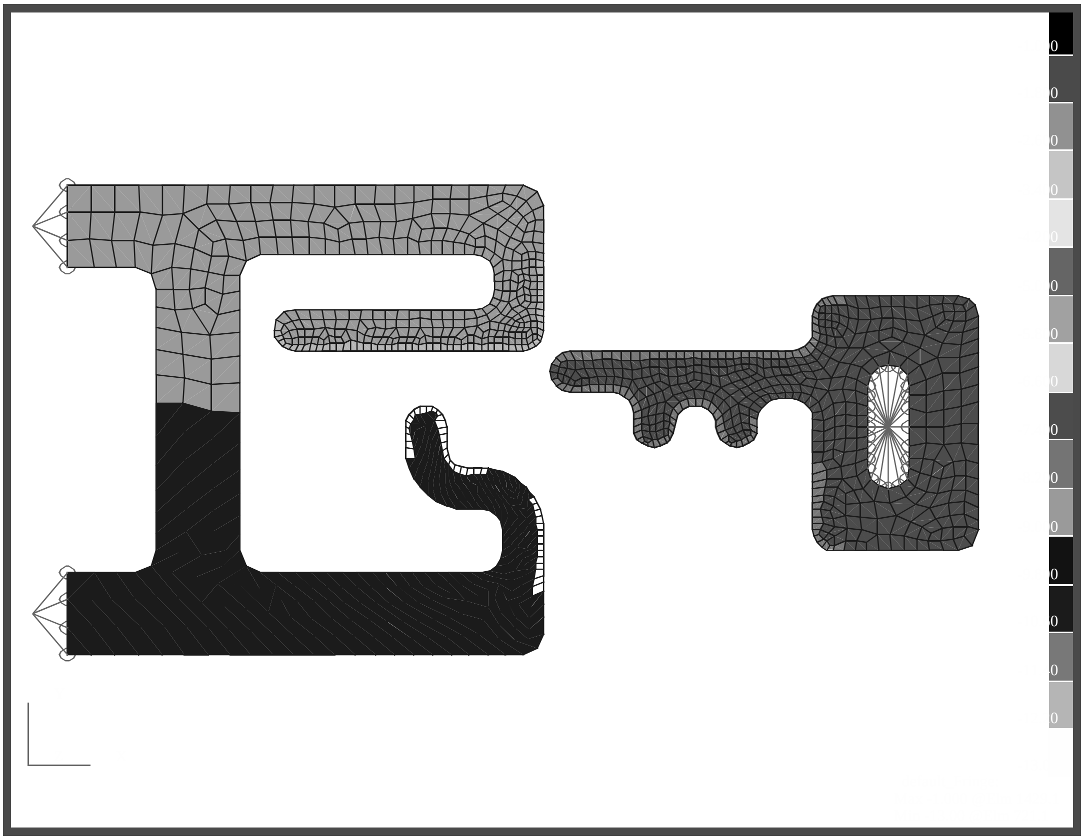
28. Show the property set names.

◆ **Properties**

Action:	**Show**
Existing Properties	**Property Set Name**
Display Method:	**Scalar Plot**
Select Groups	**FEM**

Apply

Notice there are now six different properties.

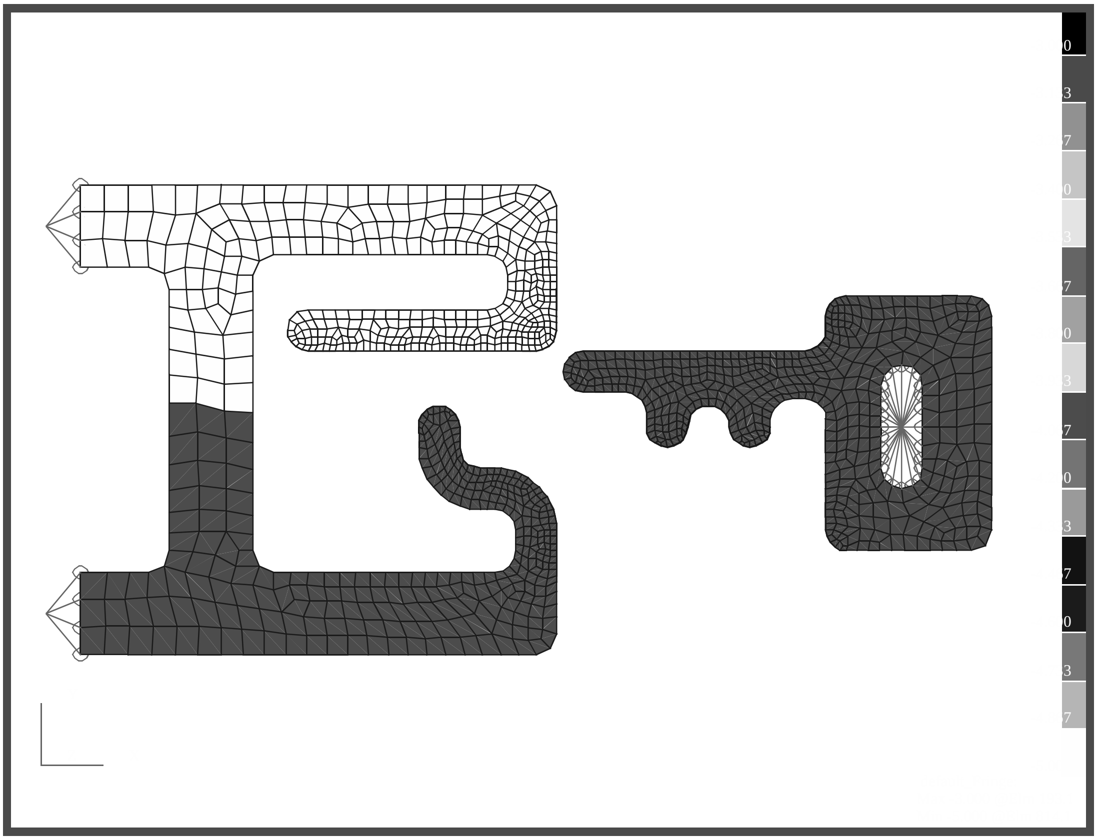


29. Show the material name.

Action:	**Show**
Existing Properties	**Material Name**
Display Method:	**Scalar Plot**
Select Groups	**FEM**

Apply

Notice there are still three materials, same as before, if you did you job well. In fact the two pin properties have been assigned the same material, and similarly for the two clip halves.



30. Show the thickness of the model.

Action:	**Show**
Existing Properties	**Thickness**

Display Method: **Scalar Plot**

Select Groups **FEM**

Apply

You should have the same 0.1 thickness throughout. You are now done modifying your model.

31. Set up the analysis for the model.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Available Jobs **STD3**

Rename the job IM3.

Job Name **IM3**

Apply

This will copy all the existing properties of STD3 to Im3

Enter "ps" in the Unix window (Windows NT users would proceed differently) you started MSC/PATRAN from. If there is no response, this is because MSC/PATRAN is running in the foreground. In that case, enter "Control,z" and then "bg" (no comma, no quotations.) This sends the execution to interactive background.mode. Now enter "ps". When "ps" executes you should see the "p3" process and also the "afea" and "afeapre" or "afeamain" process, and a process ID for each one. (The number at foremost left.) If you want to kill the MSC/ADVANCED FEA process use the ID to the left of "afeamain" to kill it gracefully by entering "kill -9 ID". (Replace "ID" with the appropriate number.) Do not kill the "afea" process as this will clean the temporary files and directories and die by itself soon enough. (Thus "gracefully".) Now and then you may check how many minutes of execution afeamain (responsible of the main or "history" calculations) has taken up to that point. Although the fact that you see the process means that the analysis is still carrying on.

Notice that while the analysis run a file named jobid.sta (in this case IM3.sta,) accumulates a summary of steps, time increments, et cetera as the job run. You might enter "tail IM3.sta") from time to time monitor the progress of the job. Also the file jobid.msg accumulates much more detailed information. You might want to see the last page of that file at any time by entering "tail -66 IM3.sta"

Because we implemented a separate job, you still have all the output files from the previous job. If you use the tail command on the STD3 job the way you just did with the IM3 job you will see that the STD3 job crashed before completion. If the IM3 job is still running you may nonetheless continue the exercise using the output from the STD3, although the animation that we will produce will not be able to show the complete pull out of the pin. It will, however, show the complete insertion and part of the extraction, and for our exercise purposes this is enough.

When an analysis job has finished you will see at the tail of the jobid.msg file a "JOB TIME SUMMARY" (unless the job has crashed ungracefully,) a message indicating "Successful termination of afeacontrol" and the release of the MSC/PATRAN ADVANCED_FEA license.Then you may read the results file jobid.fil -actually you may read partial results any time during the execution as well but until you are well familiar with managing -including deleting- results in MSC/PATRAN it would be better fro you to wait for the job completion. Make sure the current group is the "fem" group.

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Available Files: **IM3.fil**

OK

Apply

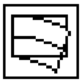
This reads the results into the database. The viewport disappears momentarily and the heartbeat remains blue while this happens.

◆ **Results**

Action: **Create**

Object: **Quick Plot**

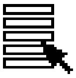
Select the **Deformation Attributes** icon



Scale Factor:	**1.0**
Scale Interpretation	◆ **True Scale**
Render Style:	**Free Edge**

❒ **Show Undeformed**

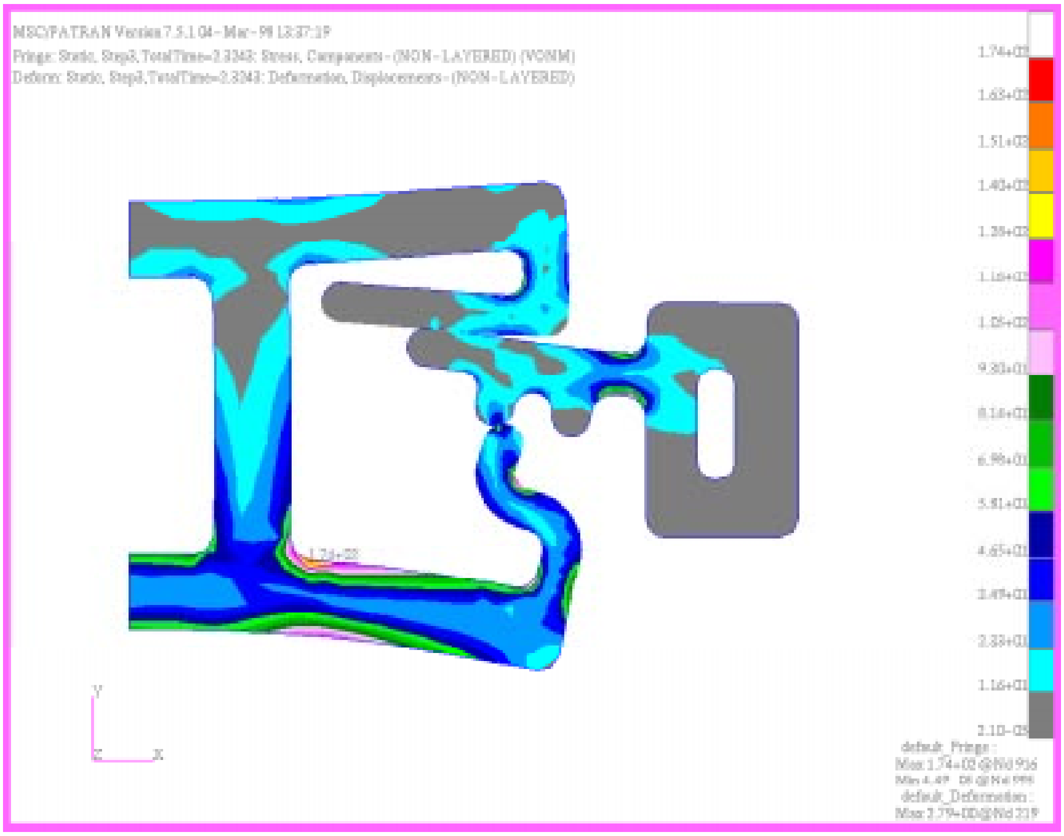
Click on the **Select Results** icon.



Select Result Case:	**Static, Step 3, TotalTime=2.324**
Select Fringe Result:	**Stress, Component**
Quantity:	**Von Mises**
Select Deformation:	**Deformation, Displacement**

Apply

This subcase corresponds to the point during the extraction step in which the vertical (Y) displacements are the largest. Notice how low the lower, right corner of the clip went.



Viewing/Named View Options ...

Create View...

Create New View **Step 3, TotalTime=2.3245 image**

Apply

Close

Select another result case

Select Result Case: **Static, Step 2, TotalTime=1.674**

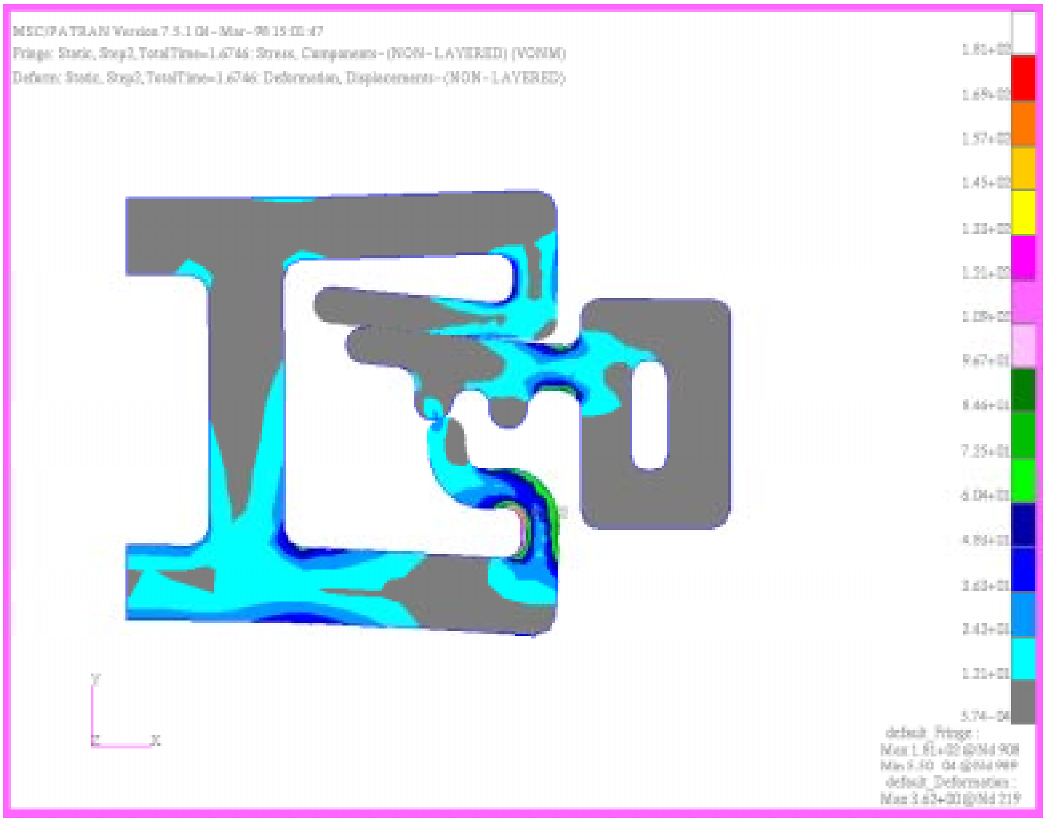
Select Fringe Result: **Stress, Component**

Quantity: **Von Mises**

Select Deformation: **Deformation, Displacement**

Apply

This subcase corresponds to the point during the insertion step in which the vertical (Y) displacements are the largest. Notice how little the lower, right corner of the clip went down. (about a third than the displacement exhibited during the extraction step.) The following figure corresponds to this moment.



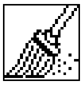
Display/ Plot/Erase ...

Select Entities **mpc1:#**

Erase

OK

The MPCs are gone from the viewport but they still belong to the group.

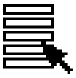


Reset Graphics

◆ **Results**

Object:	**Deformation**

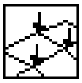
Click on the **Select Results** icon.



Select Result Case(s):	**highlight all**
Select Fringe Result:	**Stress, Component**
Select Deformation Result	**Deformation, Displacement**
Show As:	**Resultant**

■ **Animate**

Select the **Display Attributes** icon:.



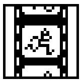
Render Style	**Free Edge**
Scale Interpretation	◆ **True Scale**

❒ **Show Undeformed**

Label Style...

Label Color:	**Red**
Label Format:	**Fixed**
Significant figures	**3**

Select the **Animation Options** icon



Animation Method	**Global Variable**
Global Variable	**Time**

If you are on a work station with a graphics card and enough swap space use 3D Animation Graphics. This allows you to view the model from different angle while the animation is happening. If not, use 2D.

Animation Graphics ■ **3D**

Apply

You will see the frames being created and then the animation starting. An Animation Control form will open which let you pause the animation, slow it down, advance it frame by frame and stop the animation.

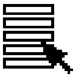
Select Stop Animation so we can add a fringe plot to the animation.

Stop Animation

◆ **Results**

Object: **Fringe**

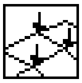
Click on the **Select Results** icon.



Select Result Case(s):	**highlight all**
Select Fringe Result:	**Stress, Component**
Quantity	**Von Mises**

■ **Animate**

Select the **Display Attributes** icon:.



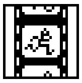
Render Style **Discrete/Smooth**

Scale Interpretation ◆ **True Scale**

■ **Show Undeformed**

Label Style...

Select the **Animation Options** icon.



Animation Method **Global Variable**

Global Variable **Time**

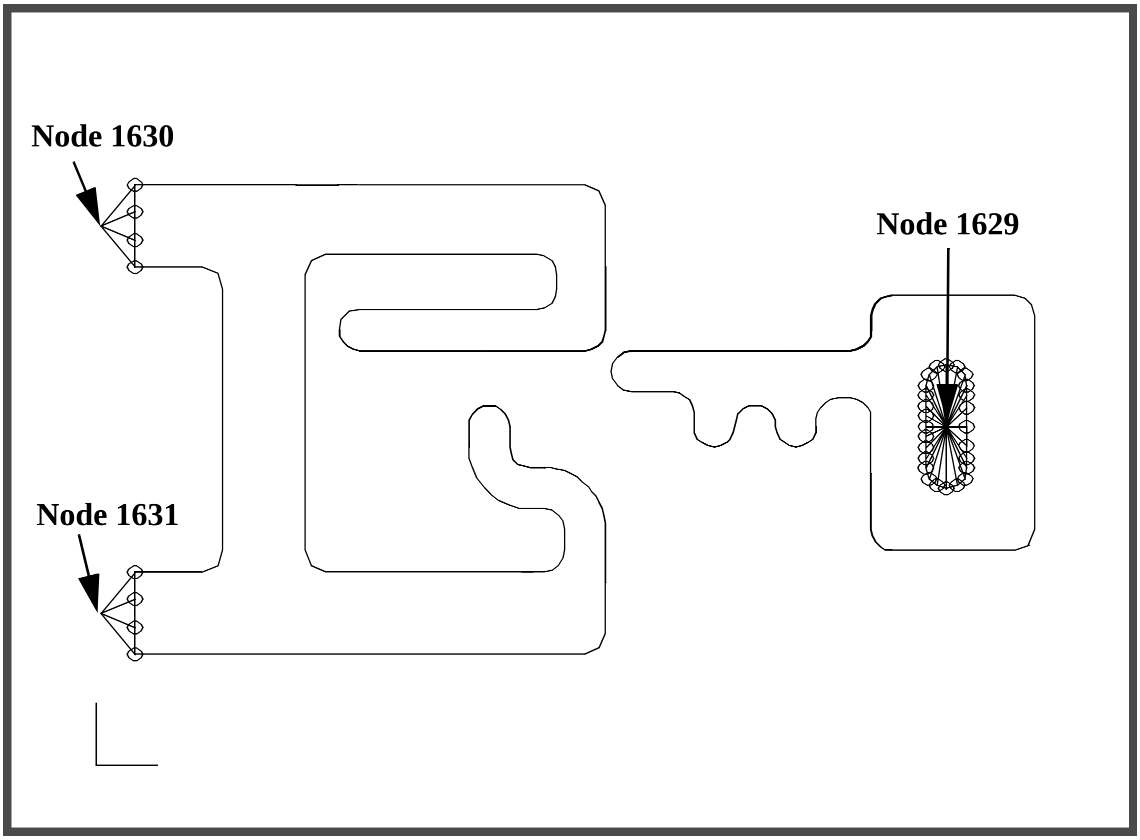
Animation Graphics ■ **3D**

Number of Frames **36**

Apply

When you are through viewing the animation select Stop Animation.

32. Create XY plots of reaction forces at each of the MPC's reference nodes. The figure below shows the location of the nodes

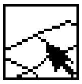


First we will look at the reaction force in the X-direction

◆ **Results**

Action:	**Create**
Object:	**Graph**
Type:	**X vs Y**
Select Result Case(s):	**highlight all**
Y:	**Result**
Select Y Result	**Reaction, Force**
Quantity:	**X Component**
X:	**Global Variable**
Variable	**Time**

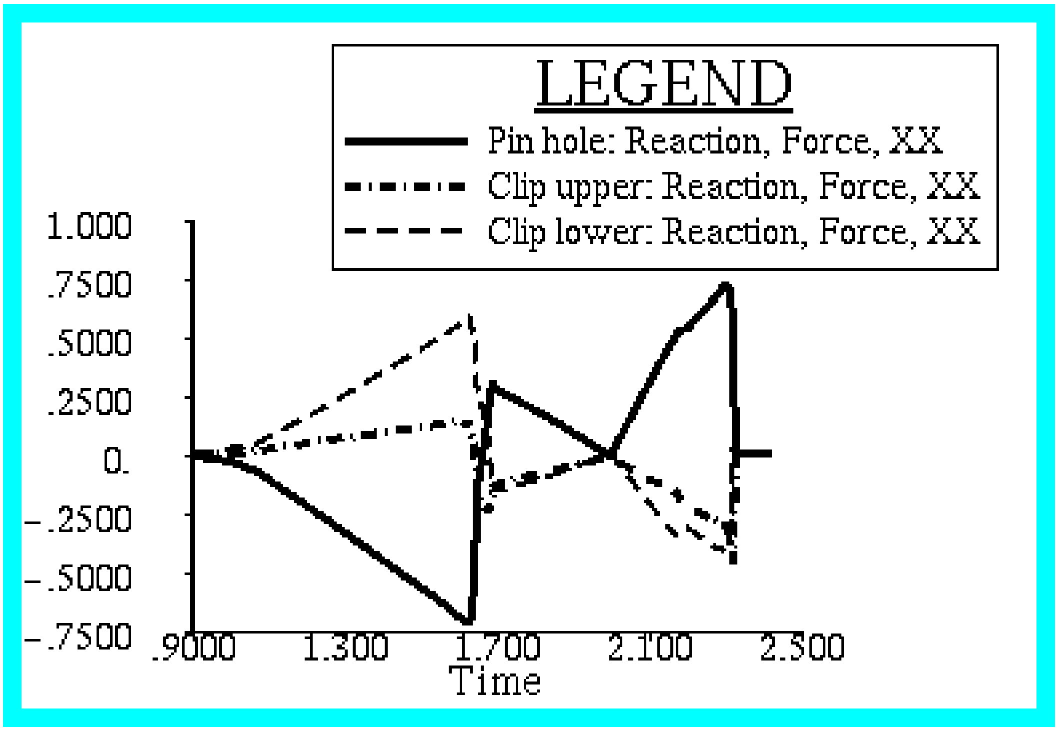
To select the nodes click on **Target Entities**



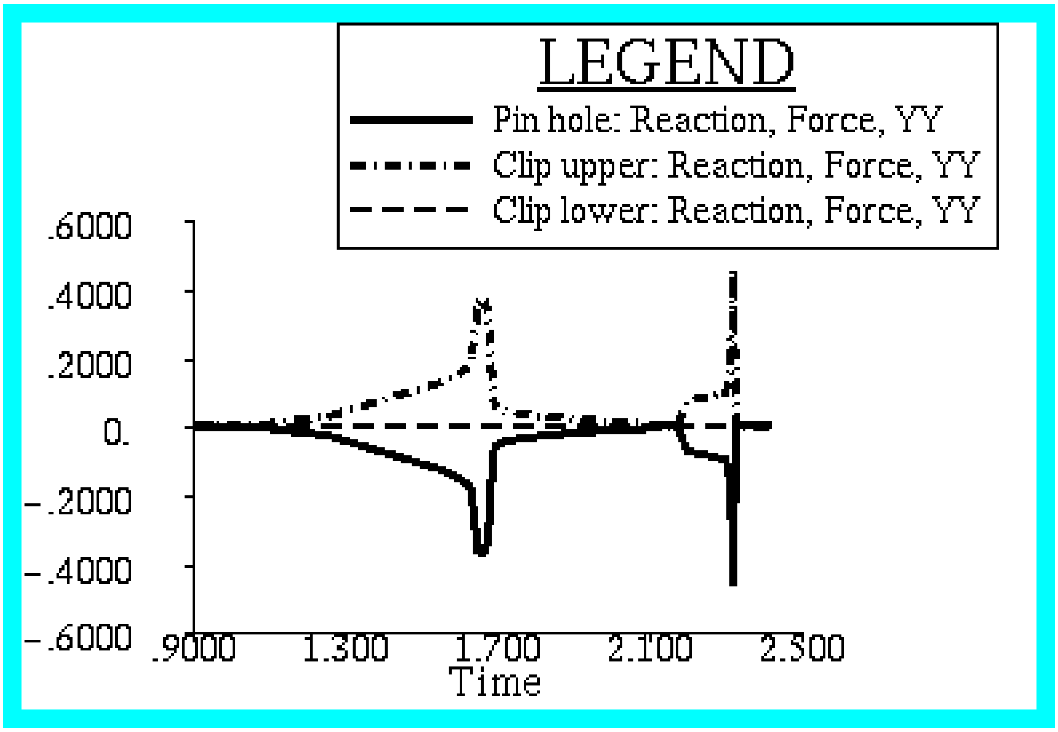
Target Entities	**Nodes**
Select Nodes	**Node 1629:1631**

Apply

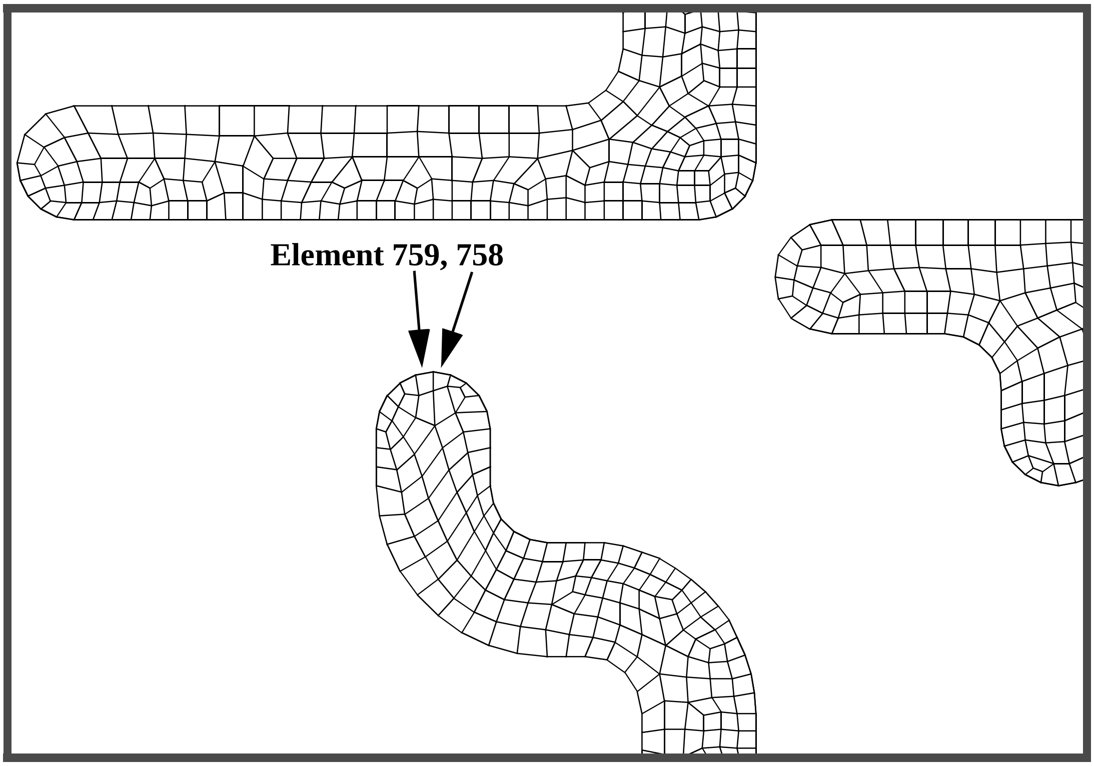
An XY Plot will appear on your screen that looks similar to the one shown below. This one has been modified to differentiate between lines and the nodes have been given their corresponding title. It is not necessary for you to do the same. Simply compare the shape of the curves to the one on your screen



Repeat the same procedure with the *Quantity* changed to **Y Component.** The plot should appear as below.

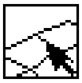


33. Now create a plot of the stresses on the top most elements that come into contact with the pin. These elements are shown below



Action:	**Create**
Object:	**Graph**
Type:	**X vs Y**
Select Result Case(s):	**highlight all**
Y:	**Result**
Select Y Result	**Stress, Component**
Quantity:	**Von Mises**
X:	**Global Variable**
Variable	**Time**

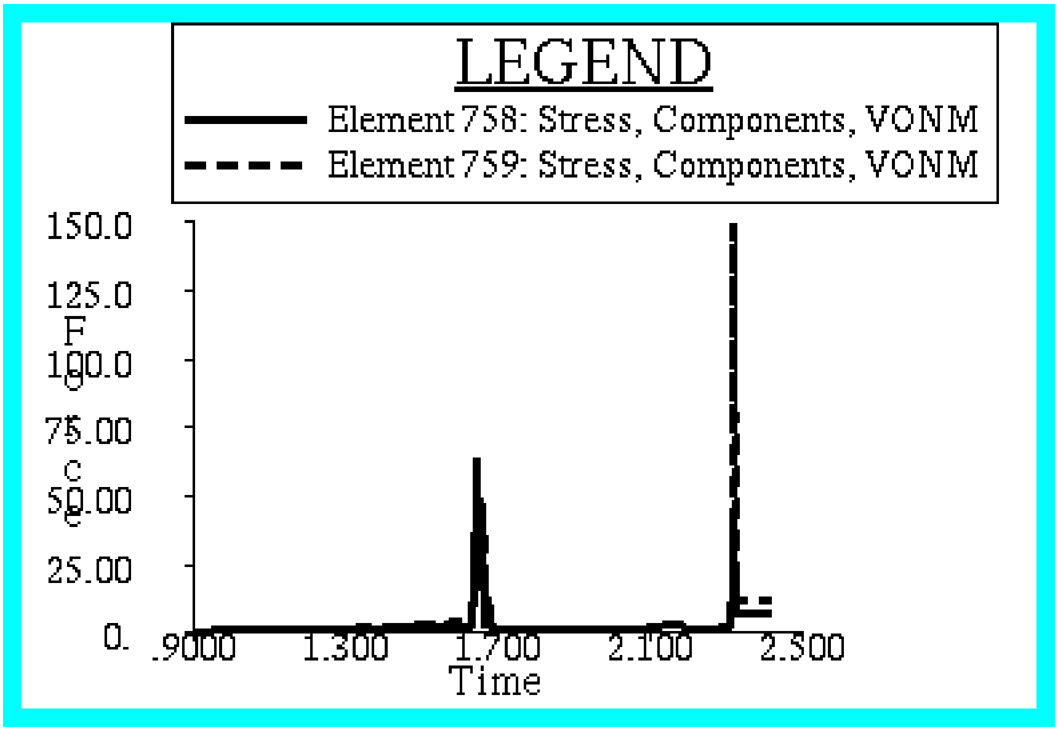
To select the nodes click on **Target Entities**



Target Entities	**Elements**
Select Nodes	**Element 758 759**

Apply

The graph should appear as below



This ends the exercise, close the database and quit PATRAN.