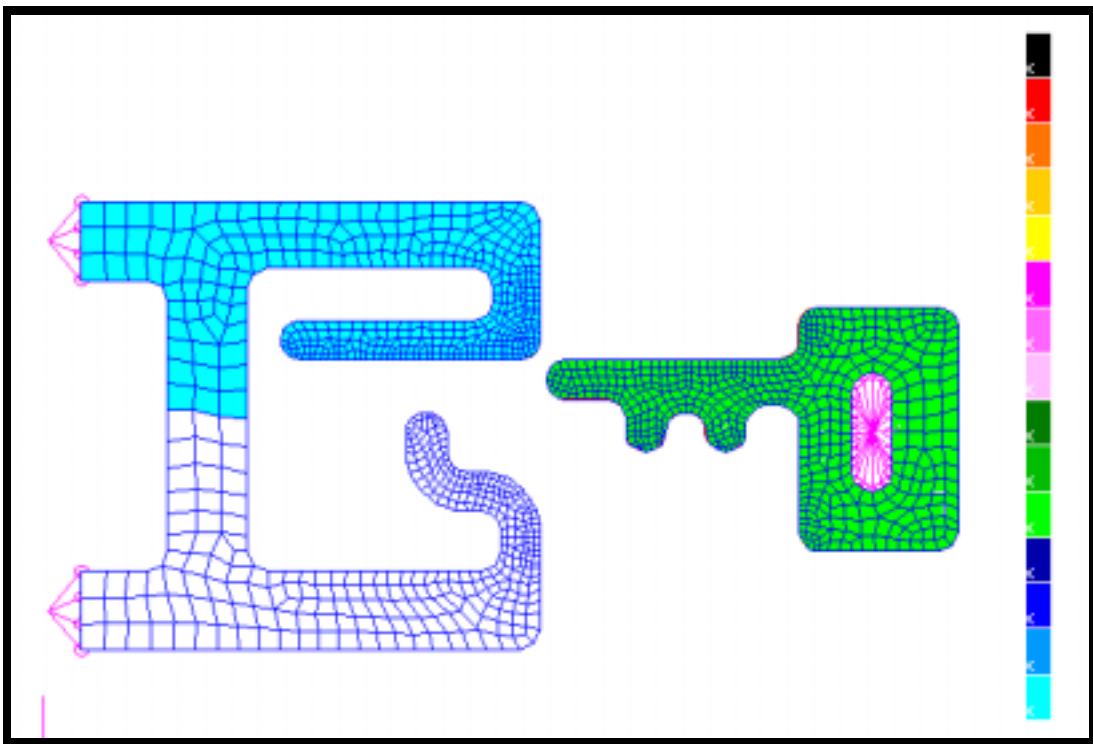

LESSON 16

Pin Insertion and Removal



Objectives:

- Resolve a difficult to converge contact problem.
- Plot Insertion and Removal Forces over the load history.
- Resolve snap-contact with static analysis.



Model Description:

This is a representative pin-clip ensemble of the sort widely used in various industries to attach separate components. Typically it is desired that the material remain in the elastic ranges and that the inserting force be smaller than the extracting force to facilitate the assembly and make an accidental dismembering difficult. The pin (shaped like a key) is pushed into and then pulled out of a clip attached to a wall.

Suggested Exercise Steps:

- Import mesh from a Patran neutral file.
- Create MPC to apply load/BCs to single nodes for easy reaction force recovery.
- Define materials and properties.
- Setup separate Insertion and Extraction load steps.
- Setup analysis with appropriate options as advised for job to converge.
- Run and monitor analysis.
- Import and post-process results.

Exercise Procedure:

1. Create a new database named **pin_insertion.db**

File/New ...

New Database Name:

pin_insertion.db

OK

In the New Model Preference form set the *Analysis Code* to **MSC.Marc**.

Analysis Code:

MSC.Marc

OK

2. Import the model geometry

File/Import ...

Change the Source to read from an **IGES** file:

Source:

IGES

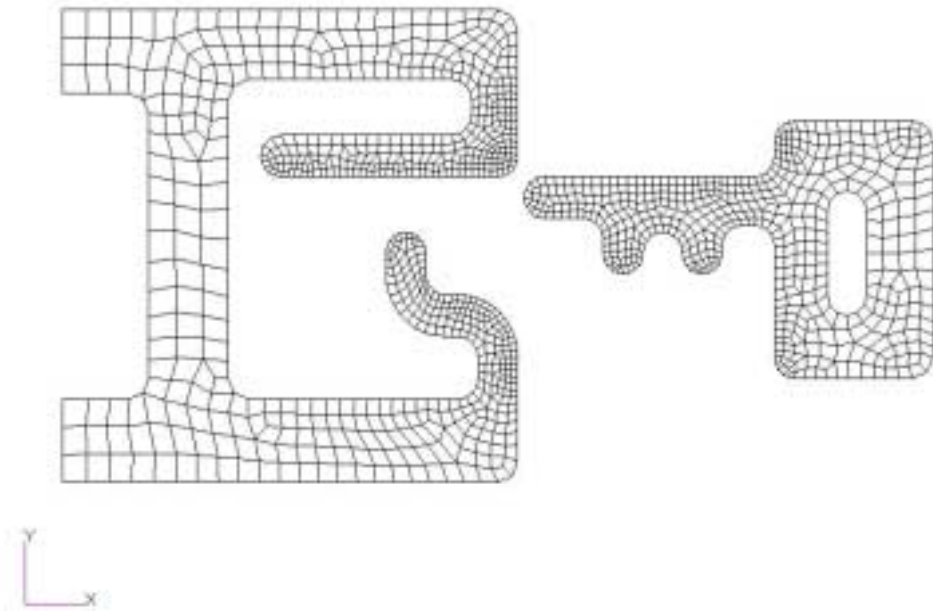
File Name:

clip.igs.01

Apply

Close the *IGES Import Summary* form that appears by clicking **OK**.

OK

**■ Elements***Action:***Create***Existing Properties***Mesh Seed***Type:***Uniform****Display Existing Seeds**

3. Click the Reset Graphics Icon.



4. Create the MPC's of the model.

■ Elements

Action: **Create**

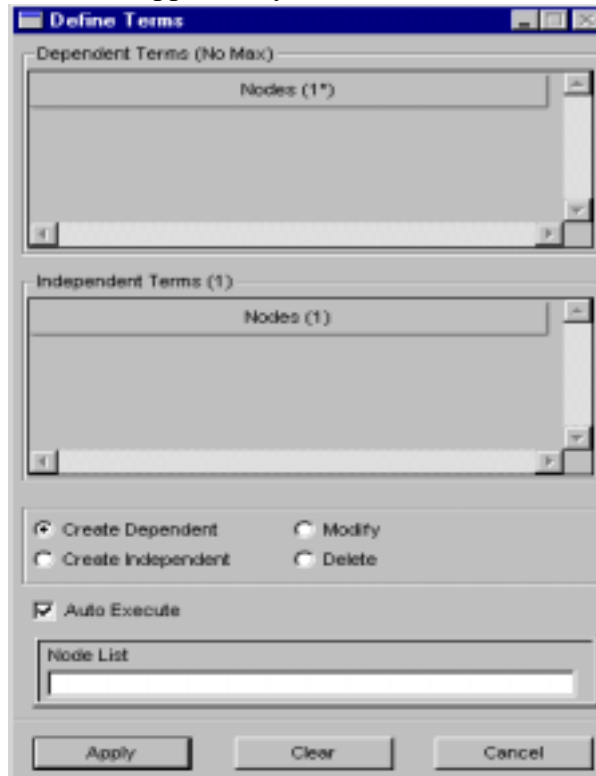
Object: **MPC**

Type: **Rigid (Fixed)**

Define Terms...

Begin with the MPC for the key hole.

This form will appear on your screen



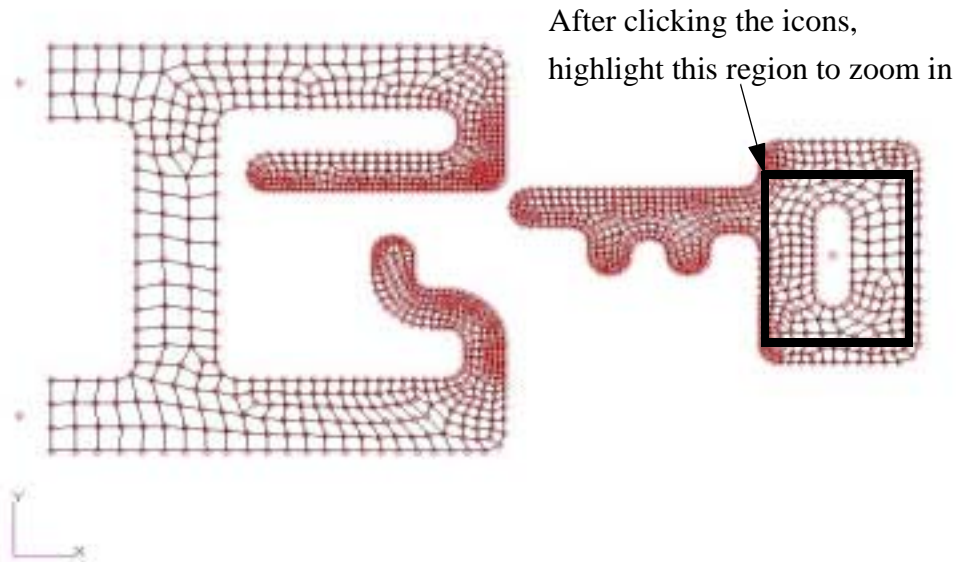
Creating the MPC for the key, select the Node Size icon, and select the View Corners icon to select the independent nodes when selecting.



Node Size



View Corners



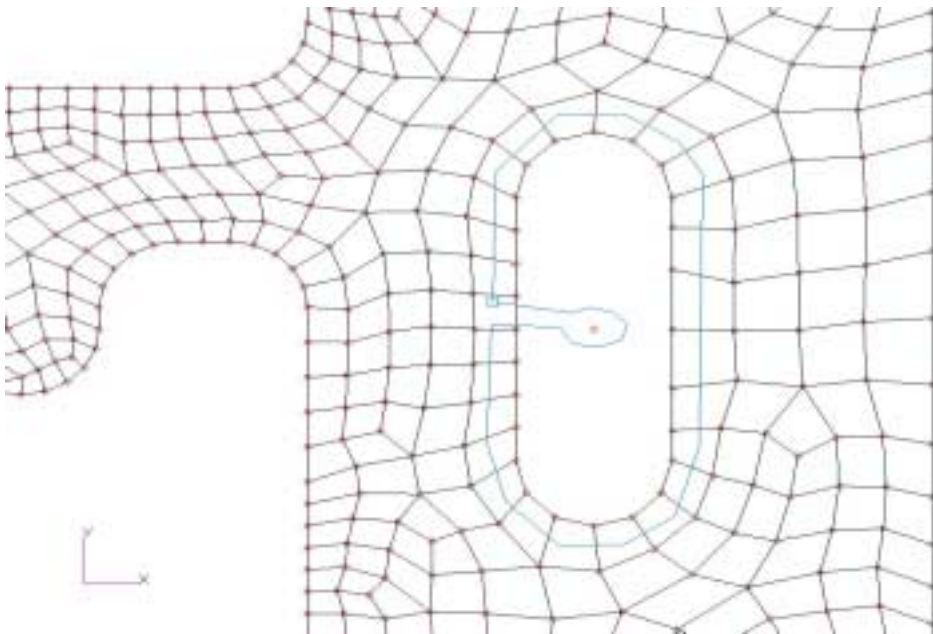
- **Create Dependent**

Node List:

(See Figure Below)

and hold down the Ctrl key to select Node 358:383

If *Auto Execute* is not selected, press *Apply*.



- **Create Independent**

Node List:

*(Select Node in the center of key hole, **Node 1629**)*

Cancel

Apply

Select the Fit View icon



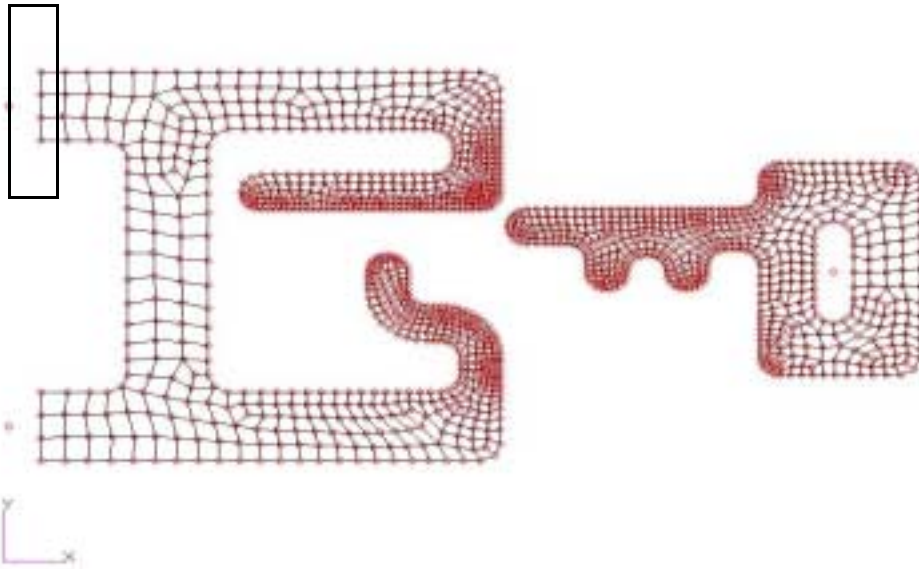
Continue with the MPC for the upper-left region of the clip.

Define Terms...

- **Create Dependent**

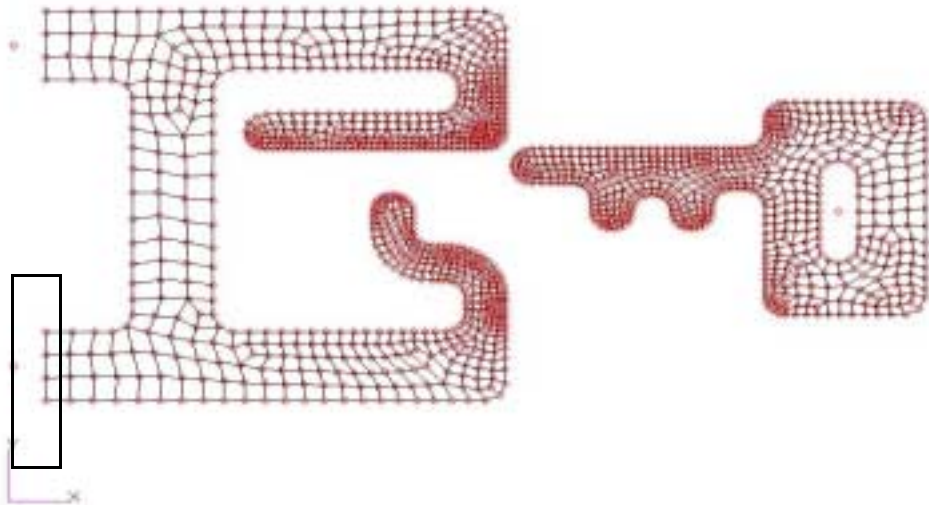
Node List:

*(See Figure Below, **Node 793:796**)*



● Create Independent*Node List:**(Select Node just left of the dependent nodes, **Node 1630**)***Cancel****Apply**

Continue with the MPC for the lower-left region of the clip.

Define Terms...**● Create Dependent***Node List:**(See Figure Below, **Node 815 818:820**)*

● **Create Independent**

Node List:

*(Select Node just left of the dependent nodes, **Node 1631**)*

Cancel

Apply

5. Define the key as a contact body.

Select the Node Size icon again



■ **Loads/BCs**

Action:

Create

Object:

Contact

Type:

Element Uniform

Option:

Deformable Body

New Set Name:

key

Target Element Type:

2D

Input Data...

Friction Coefficient (MU):

0

OK

Select Application Region...

Geometry Filter:

FEM

Select the View Corners icon to get a closer look at the key elements.



Select 2D Elements:

*(Select all Elements on key, holding the Ctrl button, **Element 193:693**)*

Add**OK****Apply**

Choose the Fit View to view the entire model



6. Likewise, create the contact definition for the key hole elements.

■ Loads/BCs

Action:

Create

Object:

Contact

Type:

Element Uniform

Option:

Deformable Body

New Set Name:

hole

Target Element Type:

2D**Select Application Region...**

Geometry Filter:

FEM

Choose the View Corners icon to zoom in on the hole elements.



Select 2D Elements:

*(Select all Elements on key hole, excluding outside independent nodes, by holding the Ctrl button and selecting **Element 694:1388**)*

Add**OK****Apply**

-
7. Create the necessary fixed displacements on the model.
Begin with the fixed displacement of the lower geometry region.

■ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Displacement"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name:</i>	<input type="text" value="fix_bottom"/>

<i>Translations <T1 T2 T3>:</i>	<input type="text" value="< 0, , >"/>
<i>Rotations <R1 R2 R3>:</i>	<input type="text" value="< >"/>

<i>Geometry Filter:</i>	<input type="text" value="FEM"/>
-------------------------	----------------------------------

Choose the independent node, to the left of the lower section of the key hole elements.

Select Nodes: **Node 1631**

Now create the fixed displacement for the top of the model.

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Displacement"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name:</i>	<input type="text" value="fix_top"/>

<i>Translations <T1 T2 T3>:</i>	<input type="text" value="< 0, 0, >"/>
---------------------------------------	--

*Geometry Filter:***FEM**

Choose the independent node, to the left of the upper section of the key hole elements.

*Select Nodes:***Node 1630****Add****OK****Apply**

8. Now create the displacement of the key towards the key hole.

Choose the Fit View to view the entire model

*Action:***Create***Object:***Displacement***Type:***Nodal***New Set Name:***insert****Input Data...***Translations <T1 T2 T3>:***< -3.9, 0, >****OK****Select Application Region...***Geometry Filter:***FEM**

Choose the independent node within the key.

*Select Nodes:***Node 1629****Add****OK****Apply**

9. Create the Load Case, **Removal**.

This step will distinguish the LBC's created to insert the key into the key hole and to remove the key. The default Load Case for the insertion of the key was automatically set up.

■ **Load Cases**

Load Case Name:

Removal

Apply

This Load Case is now the Current Load Case and will include the following LBC.

10. Set up the displacement of the key that removes it from the contact of the key hole.

■ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

remove

Input Data...

Translations <T1 T2 T3>:

< 0, 0, >

OK

Select Application Region...

Geometry Filter:

FEM

Select Nodes:

Node 1629

Add

OK

Apply

11. Rename the **Default** Load Case and modify the Load Cases for Analysis.

■ **Load Cases**

Action:

Modify

Select Load Case to Modify:

Close the **Assign/Prioritize Load/BCs** form that appears.

Rename Load Case as:

Now Modify the other Load Case to include the LBCs pertaining to the removal of the key from the key hole.

■ Load Cases

Action:

Select Load Case to Modify:

Select Individual Loads/BCs:

12. Create all the material, different for the key elements, top section of the key hole, and bottom section of the key hole.

First create the material for the bottom region of the key hole.

■ Materials

Action:

Object:

Method:

Material Name:

Constitutive Model:

<i>Elastic Modulus:</i>	<input type="text" value="4600"/>
<i>Poisson Ratio:</i>	<input type="text" value="0.3"/>
<i>Density:</i>	<input type="text" value="1e-6"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

Now create the material for the upper section.

<i>Material Name:</i>	<input type="text" value="top"/>
<input type="button" value="Input Properties ..."/>	
<i>Constitutive Model:</i>	<input type="text" value="Elastic"/>
<i>Elastic Modulus:</i>	<input type="text" value="2300"/>
<i>Poisson Ratio:</i>	<input type="text" value="0.3"/>
<i>Density:</i>	<input type="text" value="1e-6"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

And finally, the key material:

<i>Material Name:</i>	<input type="text" value="key"/>
<input type="button" value="Apply"/>	

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.

- Here, you will create the element properties. Begin with the bottom region, and include the material **bottom**.

■ Properties

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="2D"/>
<i>Type:</i>	<input type="text" value="2D Solid"/>
<i>Property Set Name:</i>	<input type="text" value="bottom"/>
<input type="button" value="Input Properties ..."/>	

Material Name:

(choose from databox
bottom)

Thickness:

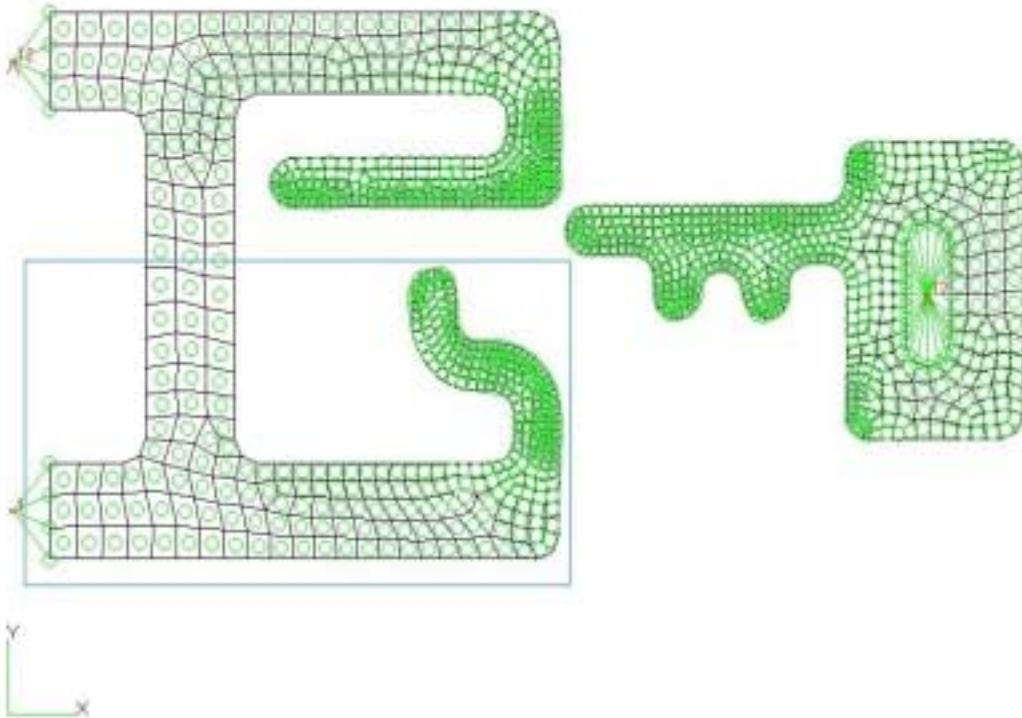
0.1

OK

Select Members:

(See Figure Below, and paste
following element ids,
Elm 694:813 959:971
1095:1207 1222:1284 1373
1379:1383:2 1384:1388)

Click Shell Element icon to choose members.



Add

Apply

Continue with the top section of the key hole elements, and include the material **top**.

Action:

Object:

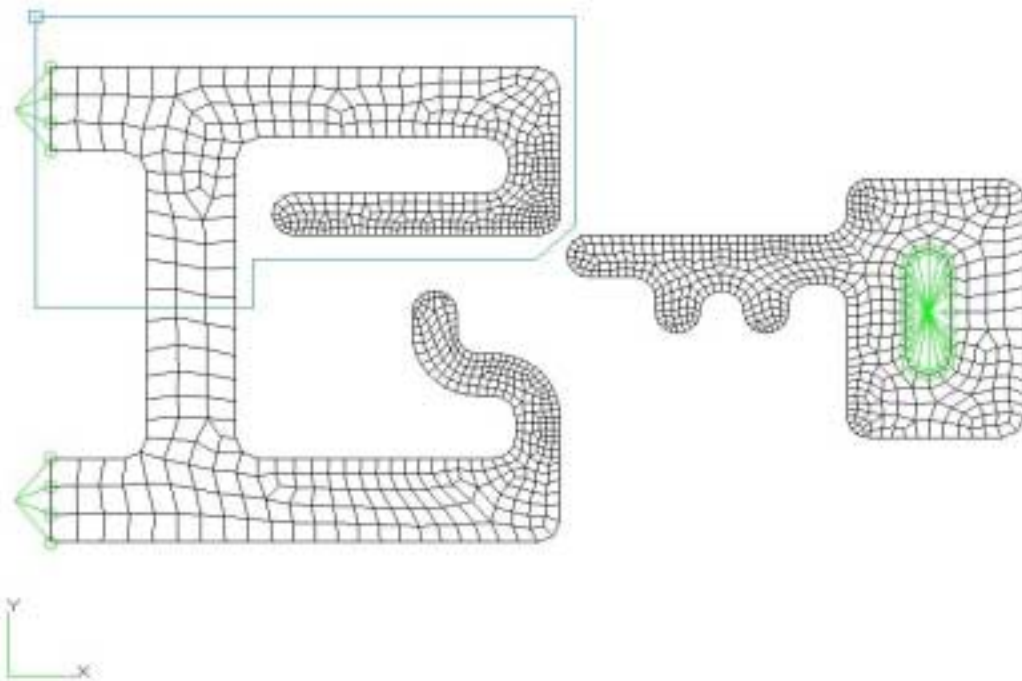
Type:

Property Set Name:

Material Name:

Thickness:

Select Members:



Add**Apply**

Repeat to create the key properties.

*Action:***Create***Object:***2D***Type:***2D Solid***Property Set Name:***key****Input Properties ...***Material Name:**(choose from databox
key)**Thickness:***0.1****OK***Select Members:**(Select all key elements,
Elm 193:693)***Add****Apply**

14. Now ready the model for Analysis with two steps, beginning with the **Insertion** step, followed by the **Removal** step.

■ Analysis

*Action:***Analyze***Object:***Entire Model***Method:***Full Run****Translation Parameters ...****Contact Parameters ...***Deformable-Deformable
Method:***■ Single-Sided**

Friction Parameters ...

Type:

Coulomb for Rolling

Method:

Nodal Stress

Relative Sliding Velocity:

0.0

OK

OK

Tolerances/ Writing:

1e-8

OK

Load Step Creation ...

Job Name:

Insertion

Solution Parameters ...

Load Increment Parameters ...

Trial Time Step Size:

0.01

Max # of Steps:

1000

Total Time:

1.0

OK

Iteration Parameters ...

Tolerance Method:

Incremental Displacement

OK

OK

Select Load Case ...

Available Load Cases:

Insertion

OK

Apply

Job Name:

Removal

Solution Parameters ...

Load Increment Parameters ...

<i>Trial Time Step Size:</i>	<input type="text" value="0.01"/>
<i>Max # of Steps:</i>	<input type="text" value="1000"/>
<i>Total Time:</i>	<input type="text" value="1.0"/>
<input type="button" value="OK"/>	
<input type="button" value="OK"/>	
<input type="button" value="Select Load Case ..."/>	
<i>Available Load Cases:</i>	<input type="text" value="Removal"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	
<input type="button" value="Load Step Selection ..."/>	
<i>Selected Job Steps:</i>	<input type="text" value="Insertion
Removal"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

15. Read the results back into Patran.

■ Analysis

<i>Action:</i>	<input type="text" value="Read Results"/>
<i>Object:</i>	<input type="text" value="Result Entities"/>
<i>Method:</i>	<input type="text" value="Attach"/>
<i>Select Results File...:</i>	<input type="text" value="pin_insertion.t16"/>
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

16. Here, you will post process the Results.

■ Results

<i>Action:</i>	<input type="text" value="Create"/>
----------------	-------------------------------------

Object:

Quick Plot

Click on the **Select Results** icon.



Select Result Cases:

select removal step around time
Removal,A1:...,Time=1.2748

Select Fringe Result:

Strain, Total

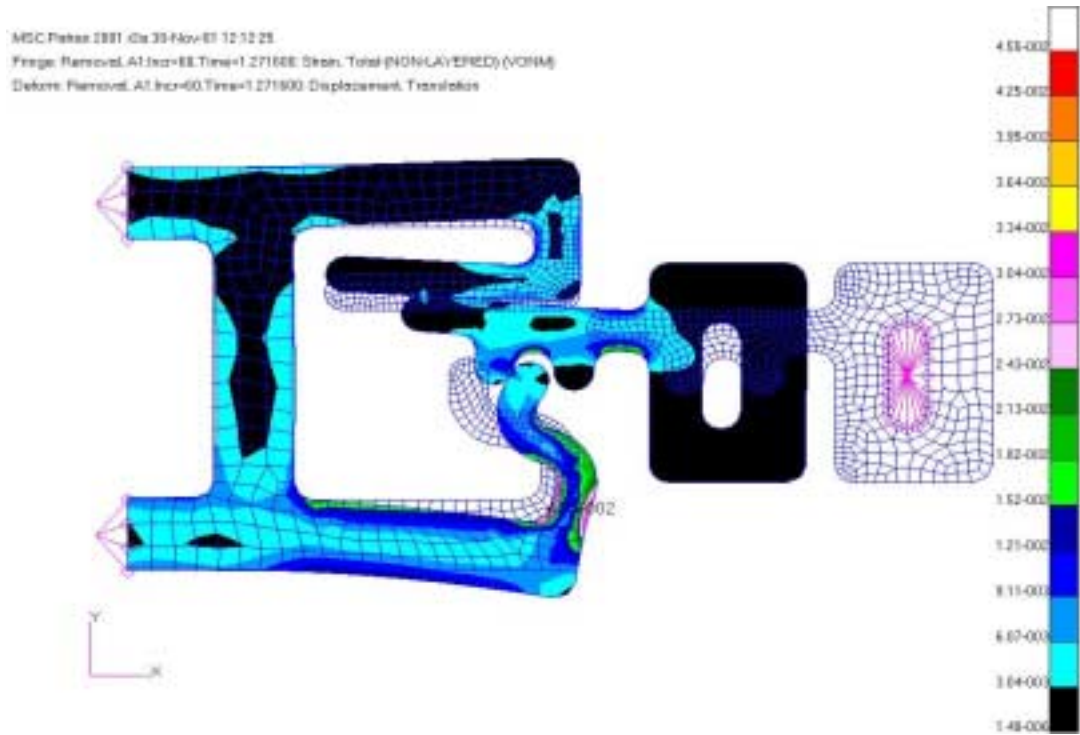
Quantity:

von Mises

Select Deformation Result:

Displacement, Translation

Apply



Close the database and quit PATRAN.

This concludes this exercise.