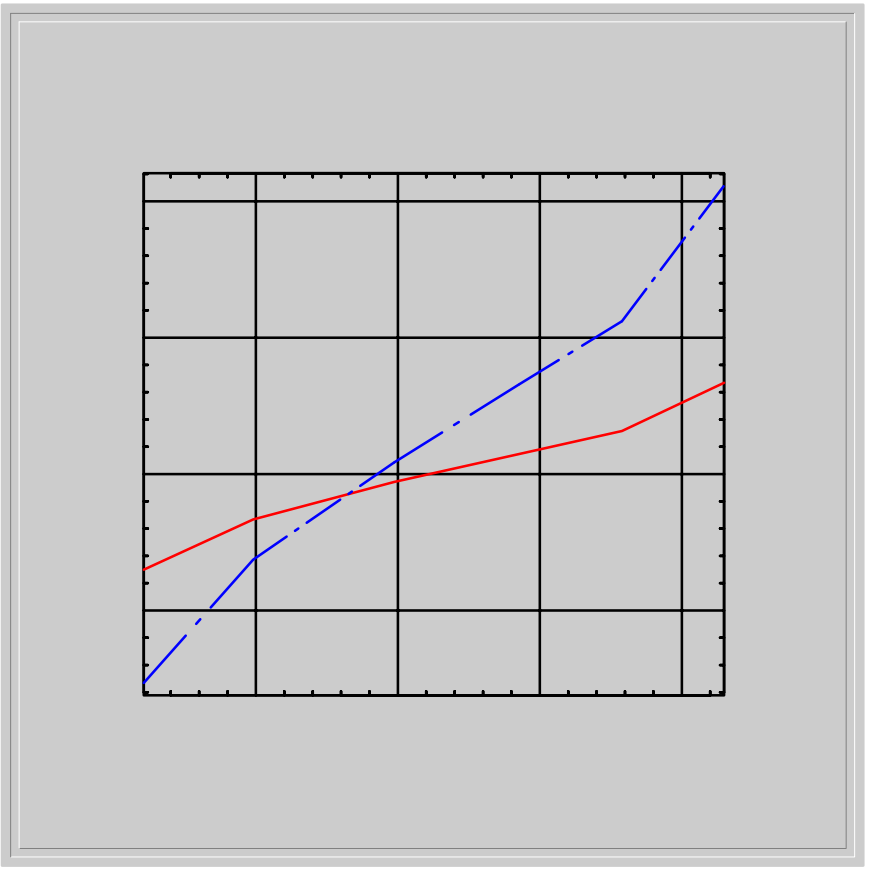


Graphing and Combining Results

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- graph results
- combine results
- compare results

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Quick Tips to Using I-DEAS
  –and–
  Creating Parts
- Introduction to Simulation
- Free Meshing
- Boundary Condition Sets
- Displaying Results

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

File
Open

Open Model File form

Model File name: any unique name
OK

Make sure you're in the following application and task:

Simulation
Master Modeler

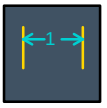
Set your units to mm.

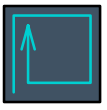
Options
Units

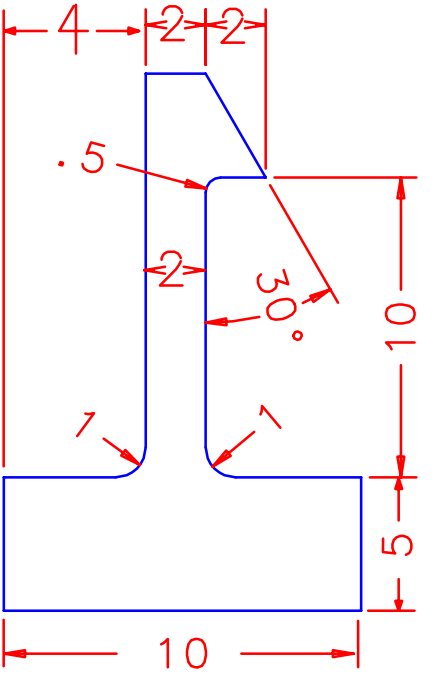
mm (*milli newton*)

What: Sketch and dimension the shape shown. (Dimensions are critical to get the same results.)

Hint



Hint

Since this part is very small, it may help to zoom in before sketching on the workplane.

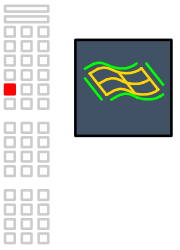
Hint

You may also want to autoscale dimension sizes to make them smaller.

Autoscale

What: Create a part using surface by boundary.

How:



Options...

Surface by Boundaries Options form

Stitch/Join Edges

Autochain Wireframe

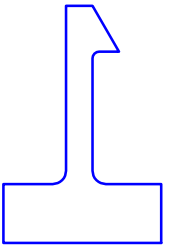
OK

---

pick anywhere on wireframe

Done

Yes



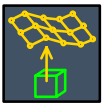
What: Name the part any unique name.

Hint

What: Create a geometry-based FE model associated to the part.

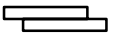
Hint


Boundary Conditions




Geometry Based Analysis Only

Save your model file.


File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:


No

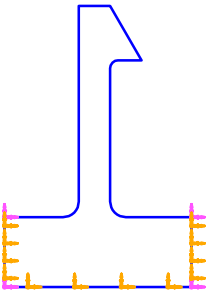
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

Why:
If you make a mistake at any time between saves and can’t recover, you can reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-Z.

What: Fully restrain the lower three edges.

Hint



What: Create a force to represent the insertion of the latch.

How:

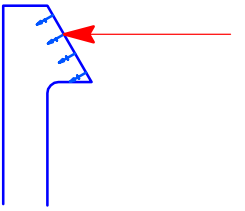
Force on Edge form

Load Set: INSERTION LOAD

Total Force

In Plane Force: 1000

OK



What: Create a second force to represent the retraction load pulling against the latch.

How:

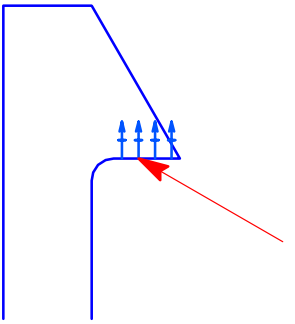
Force on Edge form

Load Set: RETRACTION LOAD

Total Force

In Plane Force: 10000

OK



Things to notice

When you entered a name for the load set, this new load set was created and made current. The previous load set is not displayed because only one load set at a time can be current.

Recovery Point

File
Save

What: Create a boundary condition set containing both load sets.

Why: When a boundary condition set with multiple load sets is solved, results are created for each individual load set as if multiple solves were performed.

How:

Boundary Condition Set Management form

 Boundary Condition Set 1

 *Restraint Set*

 RESTRAINT SET 1

Load Sets

 INSERTION LOAD (select)

 RETRACTION LOAD (shift-select)

 *OK*

What: Create a table of physical properties.

How:


Meshing


Thin Shell


No


Directory


TK THICKNESS[4V]


1st value for thickness: 2


<Return> accept all other defaults


Done

What: Define the mesh.

How:

 pick surface

Define Mesh form

 *Mesh Type*: Free

 *Element Length:* 1

 *Free Options*...

Define Free Meshing Options form

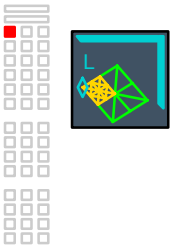
Curvature Based Length:

 *None*

 *OK* (all forms)

What: Refine the mesh along the fillets.

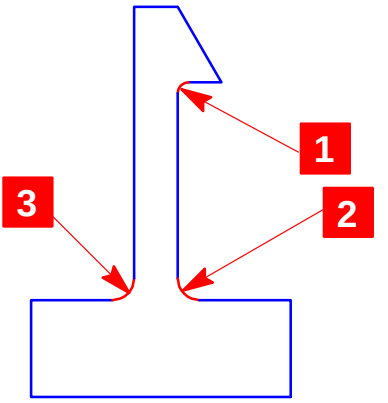
How:



1 pick curve

2 shift-pick

3 shift-pick



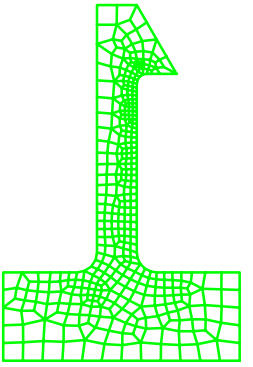
number of elements on edge: 4
(repeat for the other 2 edges)

Done

What: Generate the mesh.

Hint

 pick surface



 *Yes*

What: Solve the model.

How:


Model Solution

Manage Solution Sets form


Create...

Solution Set form


Name: Static Solve

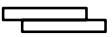

Boundary Condition Set:
Boundary Condition Set 1

Boundary Condition Set Management form

Make sure Boundary Condition Set 1 consists of Restraint Set 1 and both load sets.


OK or *Dismiss* (all forms)

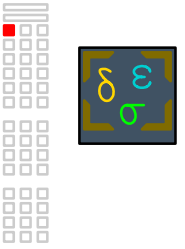
Recovery Point


File
Save

What: Display stresses from the insertion load set.

How:

Post Processing



Results Selection form

STRESS_3,INSERTION LOAD

Display Results

Component: *Von Mises*

DISPLACEMENT_1,INSERTION LOAD

Deformation Results

OK

What: Configure the Display Template.

How:

Display Template form


Results Display


Contour...(toggle on)

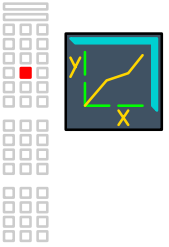

Deformed Model...


OK

What: Display the results.

What: Use the *Setup XY Graph* command to set up the XY graph values to graph the stress along a line of nodes across the latch.

How:



Select Results To Plot

Results Selection form

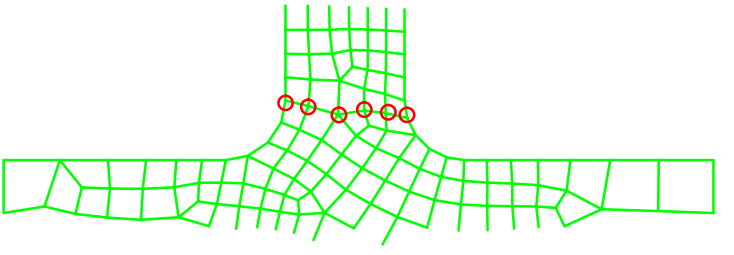
Display Results

Component: Y

OK

Single Result Set

Pick Nodes/Elements: (shift-pick nodes in a line between the tops of the two bottom fillets)



Done

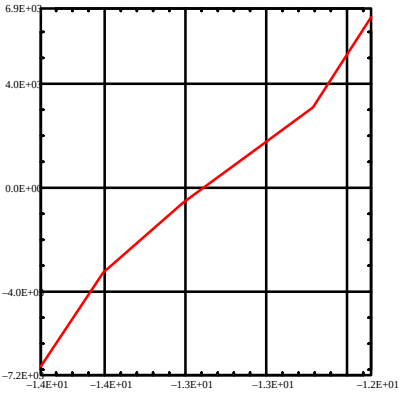
Continued on next page ...

Node Data

Part Coordinate

X Direction

Execute Graph



Store Current Graph

Function Dataset name or no.:
Insertion Dataset

id line 1: <Return>

id line 2,3,4,5: <Return> for all the rest

Done

What: Display the stresses from the retraction load (load set 2).

How:

Results Selection form


STRESS_4,RETRACTION LOAD

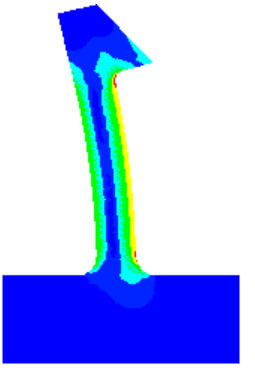

Display Results


Component: *Von Mises*


DISPLACEMENT_2,RETRACTION LOAD


Deformation Results


OK



What: Set up the XY graph values to graph stress along the same line of nodes on the latch.

How:

 *Select Results to Plot*

Results Selection form

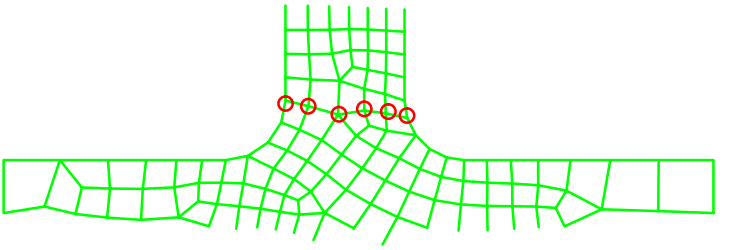
Display Results

 *Component:* Y

 *OK*

 *Single Result Set*

 *Pick Nodes/Elements*: (shift-pick nodes in a line between the tops of the two bottom fillets)



 *Done*

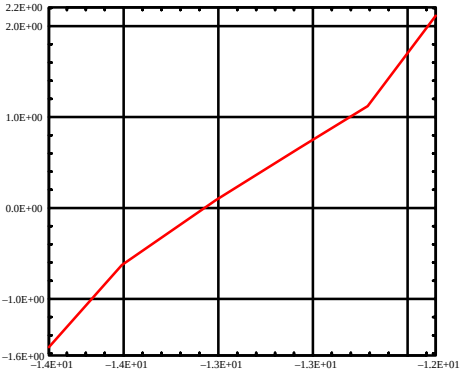
Continued on next page ...

Node Data

Part Coordinate

X Direction

Execute Graph



Store Current Graph

Function Dataset name or no.:
Retraction Dataset

id line 1,2,3,4,5: <Return> for all

Done

What: Display both graphs together.

How:

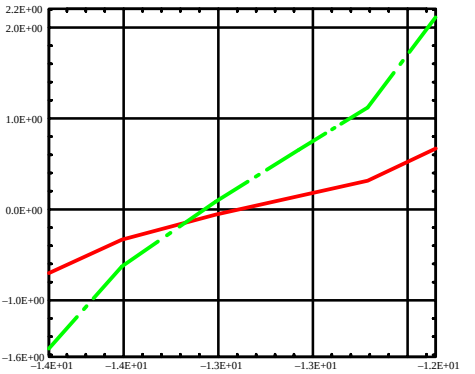

Directory


INSERTION DATASET


Directory

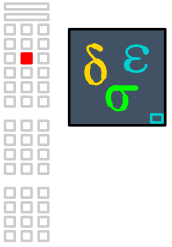

RETRACTION DATASET


Done



What: Algebraically combine both results together to display a stress condition of the insertion load plus half the retraction load.

Why: This combination represents the load to disassemble the latch by bending the latch while pulling.



Results name or no.: Disassembly Stress

Directory

STRESS 3,INSERTION LOAD

Independent variable to represent Results 1: I (for Insertion stress)

Directory

STRESS 4,RETRACTION LOAD

Independent variable to represent Results 2: R (for Retraction stress)

Done

Continued on next page ...

function: .5*R+I

id line 1,2,3,4,5: <Return> for all

STRESS

Check *I-DEAS List*.
Notice the variables are defined along with a list of elements processed.

What: Display the resulting combined stress.

How:

Results Selection form

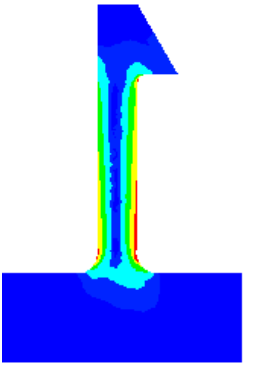
DISASSEMBLY STRESS

Display Results

Component: *Von Mises*

Deformation Results : *Clear*

OK



What: Create a dataset sorting the maximum stresses from the first two sets.

How:

Options
Preferences...

Preferences form


Menus...


Menu Display Type


All


OK (all forms)

Results
Create
Sort for Extremes
Result Type


STRESS

Data Component
Von Mises
Sort


Directory


STRESS 3,INSERTION LOAD

Continued on next page ...

Directory

STRESS 4,RETRACTION LOAD

Done

Maximum

<Return>

Maximum Stresses

Next

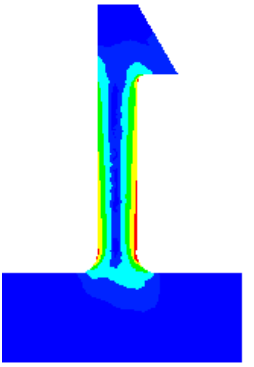
STRESS COMPONENT

Results Selection form

MAXIMUM STRESSES

Display Results

OK



What: Plot by element criteria, showing elements with stresses over 70% of the maximum.

Why: It doesn't make sense to display an average of these stresses, because the maximum stresses could have come from different load cases.

How:

Display Template form.

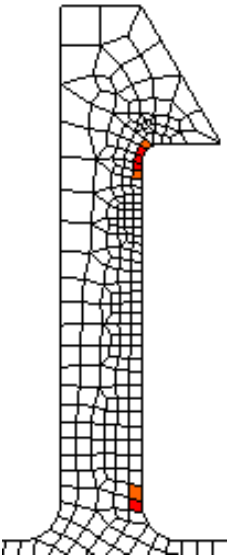
 *Element...*(toggle on)

 *Element...*

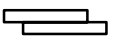
Element Criterion form

 *Above*: 70

 *OK* all forms



Hint

 *Options*
Menus ON/OFF

What: Display the retraction load stresses.

Why: Results can be compared by using the same color bar.

How:

Results Selection form

 STRESS_4,RETRACTION LOAD

 *Display Results*

 *Component*: *Von Mises*

 *Deformation Results*: *Clear*

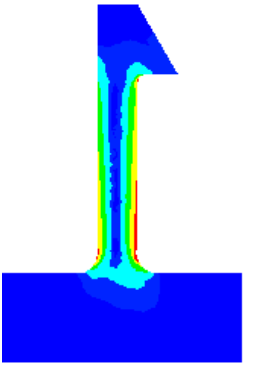
 *OK*

Display Template form

 *Contour...*(toggle on)

 *Stepped Shaded*

 *OK*



What: Convert the color bar to absolute model values.

How:

Color Bar form


Color Bar: Comparison Scale


Model Units

Compute Min/Max From


Display Results


Min of Bands: 0


OK

Compare results 3 of 3

What: Display the insertion stresses using the same color bar.

How:

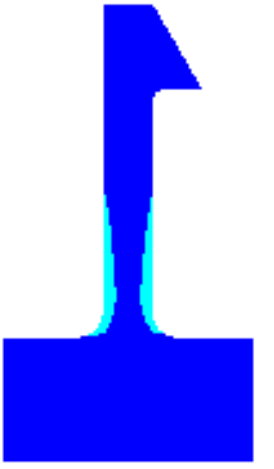
Results Selection form


STRESS_3,INSERTION LOAD


Display Results


Component: *Von Mises*


OK




To quickly interpret and compare results, set the color bar maximum value to your maximum allowable stress.

Tutorial wrap-up

You have completed the Graphing and Combining Results tutorial.

Delete the FE model, then delete the part. This part is not used in any other tutorial.

Hint

See also...

For additional information on the concepts covered in this tutorial and additional material not covered, see the following:

Help, Manuals, Table of Contents

Simulation: Finite Element Modeling User's Guide
- Post–Processing Results
  - Icon Overview for Post Processing
  - Displaying Results
  - Generating Special Results Displays

What's next?

The Visualizer tutorial covers additional display tools available on hardware displays. After completing the Fundamental Skills tutorials, begin the Advanced Projects tutorials, which cover other solution methods.

To exit this tutorial, select:

File
Exit

Warning!
Do not use the menu in the *I-DEAS Icons* window to exit. Use the menu in the Acrobat Reader window.

I-DEAS Master Series™ Online Tutorials

Structural Dynamics Research Corporation
2000 Eastman Drive
Milford, Ohio 45150
(513) 576−2400