

## 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—

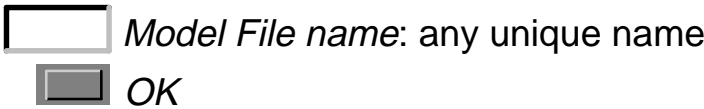
Creating Parts

- Introduction to Simulation
- Free Meshing
- Boundary Condition Sets
- Displaying Results

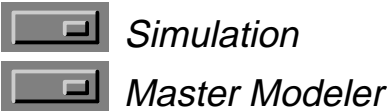
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



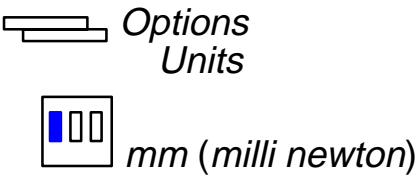
Open Model File form



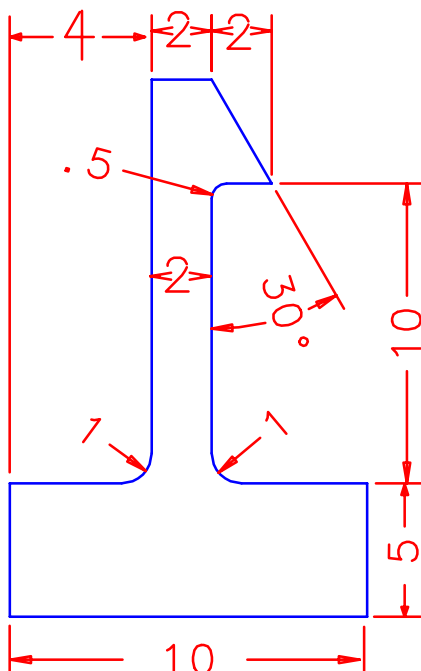
Make sure you're in the following application and task:



Set your units to mm.



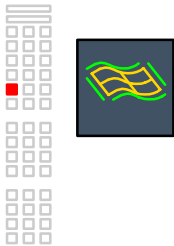
### Hint



## Autoscale

**What:** Create a part using surface by boundary.

**How:**



Options...

Surface by Boundaries Options form



Stitch/Join Edges



Autochain Wireframe



OK



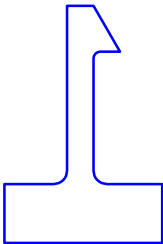
pick anywhere on wireframe



Done



Yes



**What:** Name the part any unique name.

**Hint**

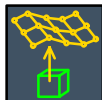


**What:** Create a geometry-based FE model associated to the part.

**Hint**

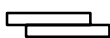


*Boundary Conditions*



*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:**

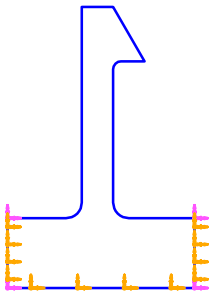
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

**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:**

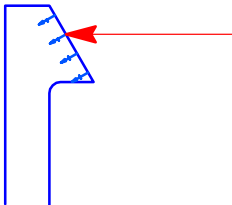


**Force on Edge form**

*Load Set:* INSERTION LOAD

☒ *Total Force*

*In Plane Force:* 1000



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

**How:**

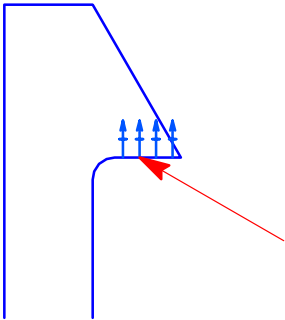


**Force on Edge form**

*Load Set: RETRACTION LOAD*

☒ *Total Force*

*In Plane Force: 10000*



**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.

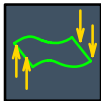
**Recovery Point**



**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:*



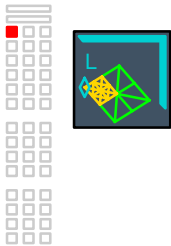
*None*



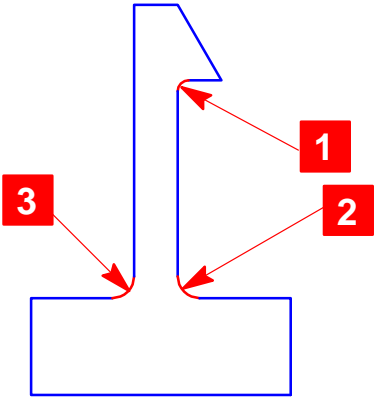
*OK (all forms)*

**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)*



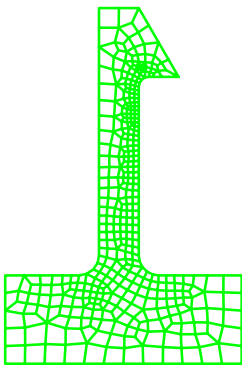
*Done*

**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.

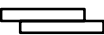


*OK or Dismiss (all forms)*



---

**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.

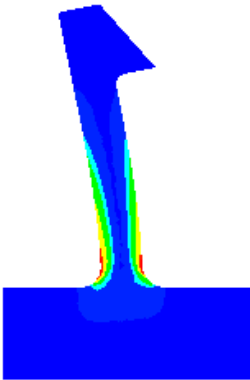
**How:**



Display Template form

- ☒ *Results Display*
- ☒ *Contour...(toggle on)*
- ☒ *Deformed Model...*
- ☐ *OK*

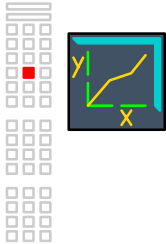
**What:** Display the results.





**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



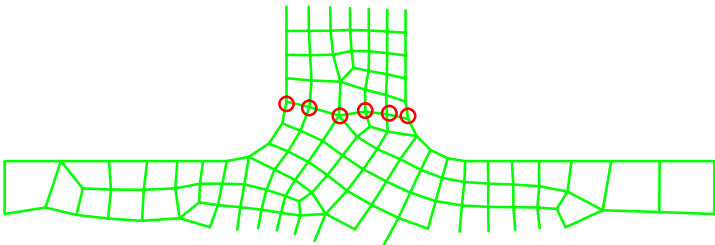
OK



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



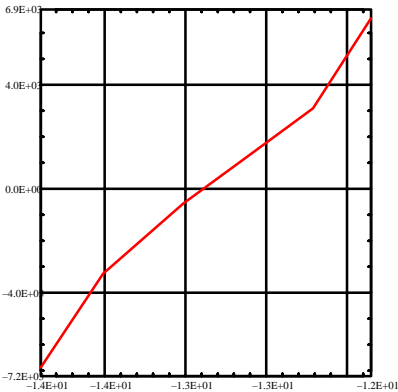
Part Coordinate



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*

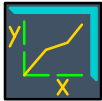


*OK*



**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



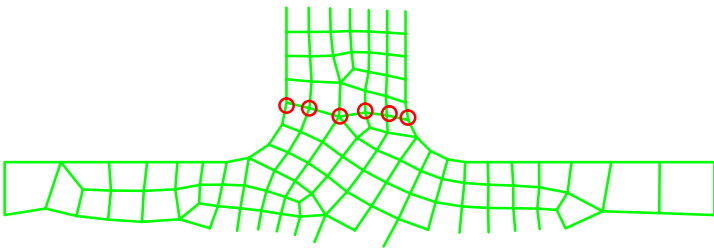
OK



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



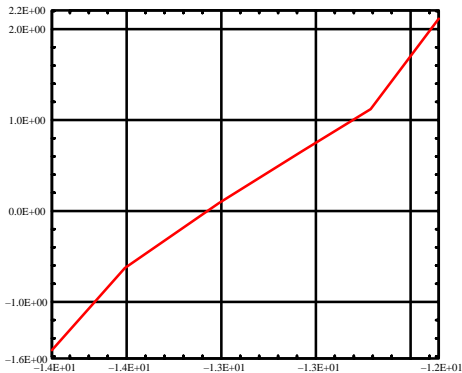
Part Coordinate



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



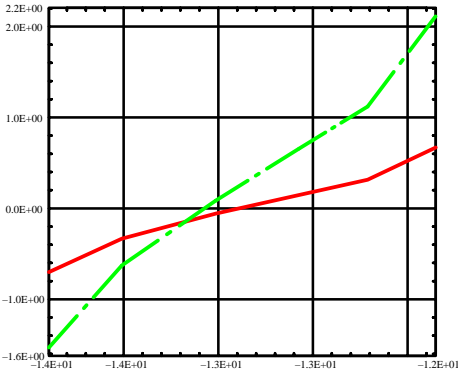
Directory



RETRACTION DATASET

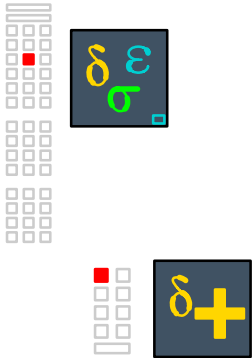


Done



**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

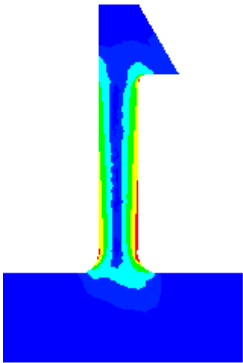
 DISASSEMBLY STRESS

 *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...

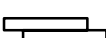
---

 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



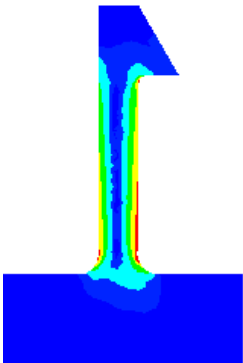
MAXIMUM STRESSES



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.

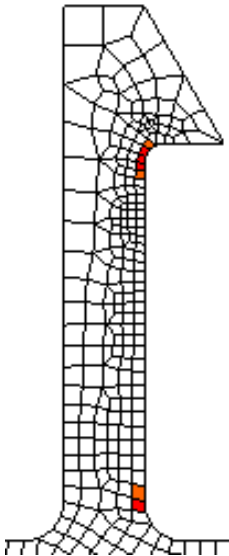
☒ *Element...(toggle on)*

☐ *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



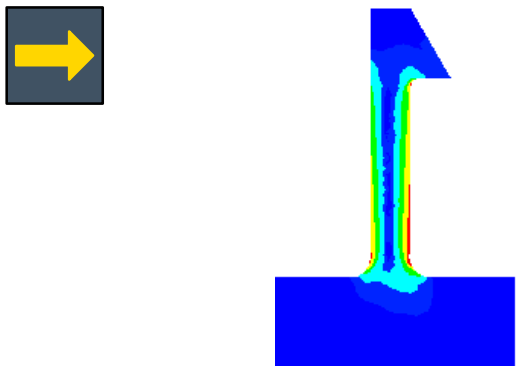
*Contour...(toggle on)*



*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*

**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

This online information content, is licensed to the user for the period set forth in the applicable license agreement, subject to termination of the license by Structural Dynamics Research Corporation (SDRC®) at any time, and at all times remains the intellectual property of SDRC. The information contained herein is confidential to SDRC and shall not be copied or reproduced in any form whatsoever, nor be disclosed to anyone other than an authorized representative of the user's employer who is contractually obligated not to disclose same, without express prior written consent of SDRC. The user of this tutorial and the computer program(s) referred to herein retains full control over and is solely responsible for the mechanical design of the user's equipment, machinery, systems, and products. SDRC makes no warranties of any kind, including the warranty of merchantability or fitness for a particular purpose in respect to the equipment, machinery, systems, and products derived or resulting hereunder, and the user assumes all risks and liability for results obtained by the manufacturing, use or implementation of the computer program(s) described herein, whether used singly or in combination with other designs or products. SDRC shall not be liable for any special or consequential damages. SDRC makes no warranty that the equipment, machinery, systems, and products derived or resulting hereunder will not infringe the claims of domestic or foreign patents and further does not warrant against infringement by reason of the use thereof in combination with other design, products, or materials or in the operation of any process. Users shall protect, indemnify and hold harmless SDRC of and from any loss, cost, damage or expense arising from any claim that is in any way associated with the computer program(s) described in this tutorial. Data presented in examples do not necessarily reflect actual test results and should not be used as design criteria.

By acceptance of I-DEAS Master Series, the user agrees to the above conditions and further agrees that this intellectual property will not be exported (or reexported from a country of installation), directly or indirectly, separately or as part of a system, without user or user's employer, at its own cost, first obtaining all licenses from the United States Department of Commerce and any other appropriate agency of the United States government as may be required by law.

© Structural Dynamics Research Corporation 1979, 1980, 1983, 1984, 1986, 1988, 1990, 1991, 1992, 1993, 1994, 1995, 1996, 1997, 1998

© Maya Heat Transfer 1990, 1991, 1992, 1993, 1994, 1995, 1996, 1997, 1998

All rights reserved. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, or by any information storage or retrieval system without permission in writing from SDRC.

Federal Acquisitions: Commercial Computer Software  
Use governed by terms of SDRC's Software License and Service Agreement.

SDRC has worked to verify the accuracy of the information contained in this manual as of its publication date; however, such information is subject to change without notice and SDRC is not responsible for any errors that may occur in this document.

This software is a Licensed Product of and distributed by SDRC and may only be used according to the terms of that license on the system identified in the License Agreement.

**SDRC and SDRC I-DEAS are registered trademarks of Structural Dynamics Research Corporation.**

**The following are trademarks of Structural Dynamics Research Corporation**

I-DEAS, I-DEAS Master Series

All other trademarks or registered trademarks belong to their respective holders. All questions or requests should be addressed to:

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