

## Displaying Results

### I-DEAS™ Tutorials: Fundamental Skills

This tutorial shows the basics of displaying results in the Post Processing task.

#### Learn how to:

- select results to display
- use the display template
- control the calculation domain
- animate the results

# Before you begin...

---

## Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

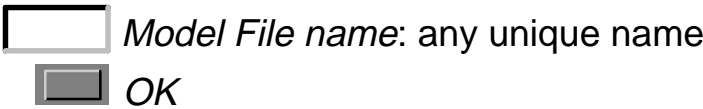
Creating Parts

- Introduction to Simulation
- Free Meshing
- Boundary Condition Sets

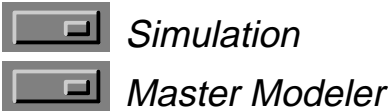
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



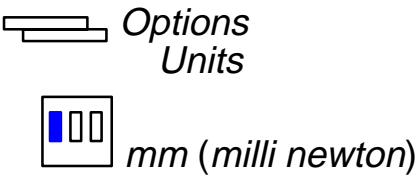
Open Model File form



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

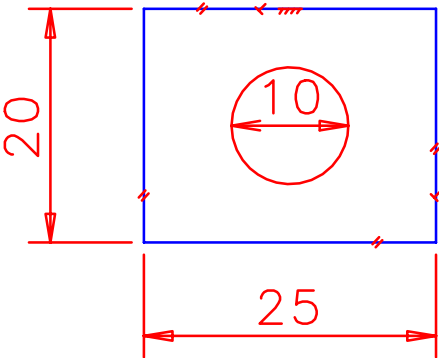


Set your units to mm.



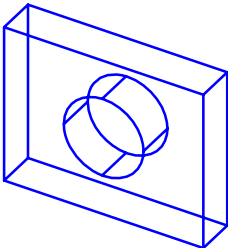
**What:** Sketch this shape to the dimensions shown.

Hint



**What:** Extrude both the rectangle and circle 5mm.

**Hint**



**What:** Name the part.

**Hint**



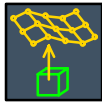
*Name: Plate*

**What:** Create an FE model associated to the part.

**Hint**



*Boundary Conditions*

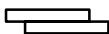


*FE Model Name:Thin-shell Model*



*Geometry Based Analysis Only*

**Save your model file.**



*File  
Save*

**Warning!**

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



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

**Why:**

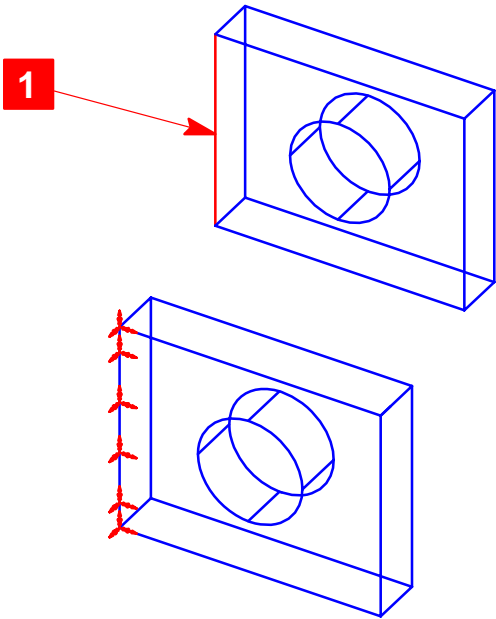
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

**Hint**

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

**What:** Fully restrain the front left edge.

**Hint**



**What:** Select the 2 front corners and apply equal but opposite forces.

**Hint**



1

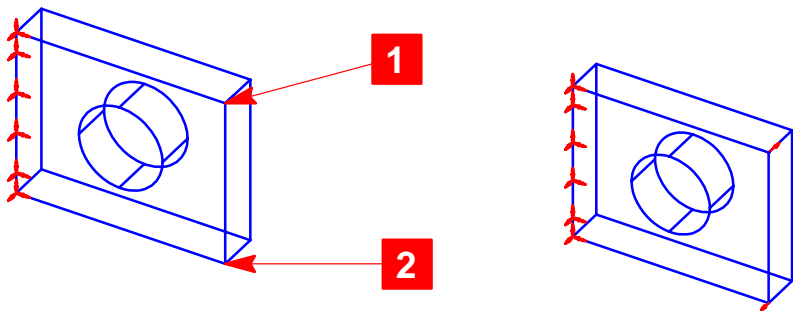
Force on Vertex/Location on Geometry form

Z Force: -1000

2

Force on Vertex/Location on Geometry form

Z Force: 1000



**What:** Create a boundary condition set.

**Hint**



☐ *Restraint Set*

☒ *Load Set 1*

**What:** Define a physical property table with a thickness of 5mm.

**How:**



*Meshing*



*Thin Shell*

*physical prop name:* Thin Shell-1



*No*



*Directory*



*TK THICKNESS [4V]*



There are four values of thickness (4V).



**Check I-DEAS Prompt.**

*1st value for thickness:* 5

<Return> accept all other defaults

**Warning!**

If you enter more than one value, all 4 values must be non-zero. To enter a uniform thickness, you need to enter only the first value.



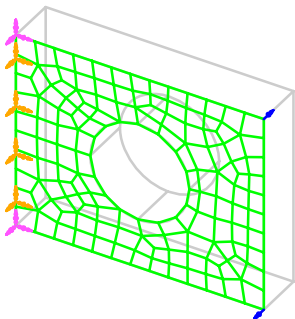
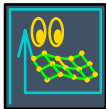
*Done*

**What:** Mesh the front surface.

**Hint**



*Element Length: 2*

☐

*Keep Mesh*

**Recovery Point**

*File  
Save*



**What:** Create a solution set.

**Hint**



*Model Solution*



---

**Manage Solution Sets form**



*Create...*

---

**Solution Set form**

*Name: Static Twist Solution*



*OK*

---

**Manage Solution Sets form**



*Dismiss*

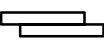
---

**What:** Solve the model.

**Hint**



**Recovery Point**



*File*

*Save*

**What:** Prepare for Post Processing by changing tasks.

**Hint**



*Post Processing*

The most important icons in Post Processing are the 3 across the top row.



**Results...**

Selects result sets:

- Display Results: result set to display in color.
- Deformation Results: result set to deform the model.



**Display Template...**

Selects the type of display and other options:

- Arrow
- Contour
- Element



**Calculation Domain...**

Selects calculation options:

- Elements to display
- Averaging
- Shell element layer to display

**What:** Display a deformed contour plot of the maximum principal stress.

**How:**



## Results Selection form



Stress\_2... (select)



*Display Results*



*Component: Maximum Principal*



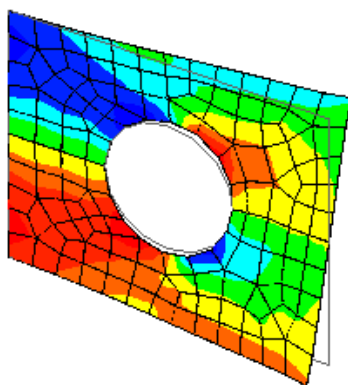
Displacement\_1...



*Deformation Results*



*OK*



**What:** Display a contour plot of the Z component of displacement.

**How:**



Results Selection form



Displacement\_1



Display Results



Component: Z



Deformation Results: Clear



OK



**What:** Display an arrow plot of deflections.

**How:**



Results Selection form



Displacement\_1



*Display Results*



OK



Display Template form



*Arrow... (toggle on)*



*Arrow...*

Arrow Options form



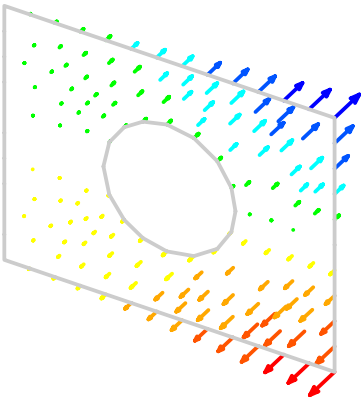
*Longest Arrow (% of screen): 30*



OK

Display Template form

 OK



**What:** Display an arrow plot of maximum principal stress.

**How:**



Results Selection form



Stress\_2...



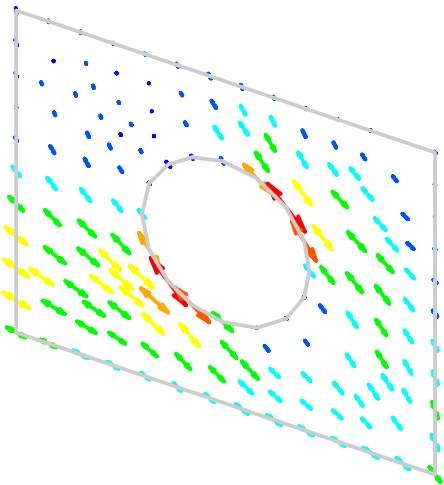
Display Results



Deformation Results: Clear



OK



**What:** Plot all elements with stresses over 75% of maximum stress.

**How:**





Display Template form

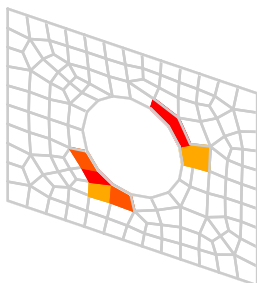
 *Element...(toggle on)*

 *Element...*

Element Criterion form

 *Above: 75*

 *OK (all forms)*



**Check I-DEAS List.**

A list of the elements selected is stored as an “output group,” which can then be used to re-select these elements.



**What:** Plot a shaded contour of just these elements.

**How:**



## Display Template form



*Contour...*



*Stepped Shaded*



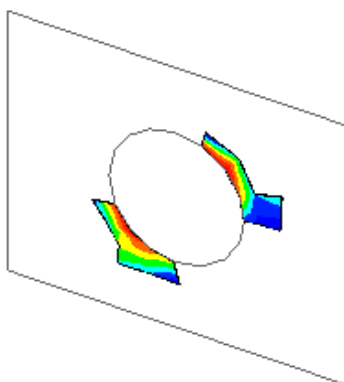
*OK*



*Use Group*



*Output*






**What:** Turn on the probe switch and use the *Probe* command to interrogate stresses.

**How:**



Display Template form

-  *Probe*
-  *Contour...*
-  *OK*



Hint

Click the pointer on various nodes.



**Check I-DEAS List.**  
Note the stress values listed.

**What:** Plot a line contour of stresses.



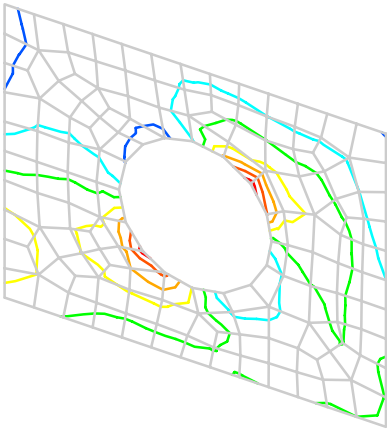
Display Template form

 *Probe*

 *Contour...*

 *Stepped Shaded  
Line*

 *OK*



**What:** Turn off averaging across elements, and generate the stress display with all elements.

**How:**



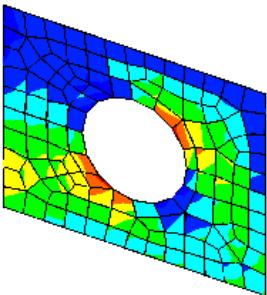
Calculation Domain form

- ☐ *Do Not Average Across*
- ☒ *Elements*
- ☐ *OK*



Display Template form

- ☒ *Stepped Shaded*
- ☐ *OK*

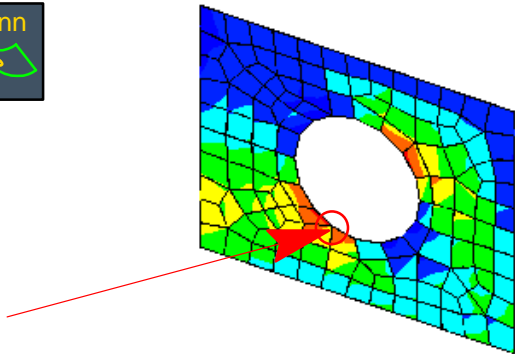


Things to notice

The stresses look rougher than when displayed with averaging, but this gives a more accurate indication of the solution accuracy.

**What:** Probe some of the node locations, particularly the node shown.

Hint

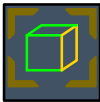


Things to notice

With averaging turned off, multiple stress values are listed for each node. Where can you find nodes with the highest difference?

**What:** Return to normal averaging of stresses.

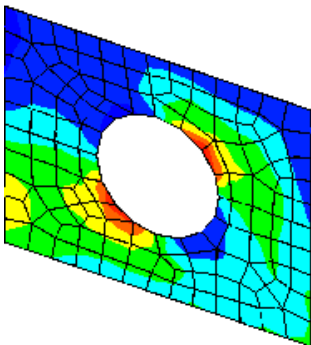
How:



Calculation Domain form

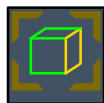
☐ Do Not Average Across: (toggle off)

☒ OK

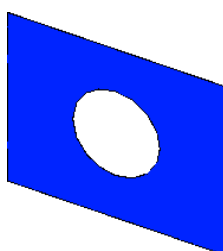


**What:** Display stresses in the middle of the shell elements.

**Hint**



*Shell: Middle*

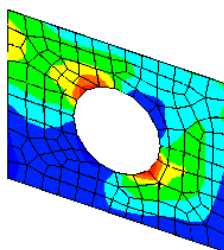


**What:** Display stresses on the bottom surface of the elements.

**Hint**



*Shell: Bottom*



**Things to notice**

Stresses vary through the shell element thickness.

**What:** Animate the stresses and displacements.

**How:**



## Results Selection form



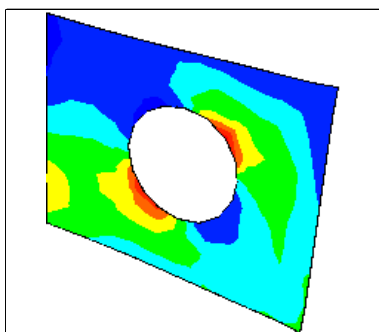
Displacement\_1...



*Deformation Results*



OK



*End (to stop animation)*

**What:** Change the number of frames of animation.

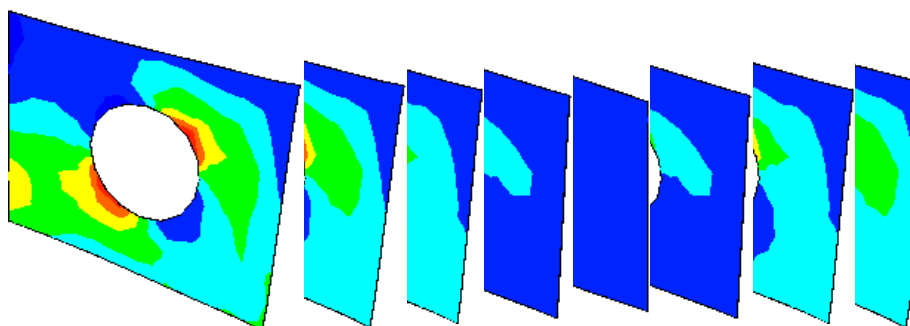
**How:**



## Animation form

*Number of Frames: 8*

*Animate*



*End (to stop animation)*

## Hint

If the animation flickers, change this option.

*Options, Preferences*



*Display*



*Double Buffering (toggle on)*

This results in a smoother transition between frames, but you lose some color detail.



## Tutorial wrap-up

---

You have completed the Displaying Results tutorial.



Do not delete the FE model or the part. They are used in the tutorial “Visualizer.”

**What:** Save the model file before exiting this tutorial.



### See also...

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

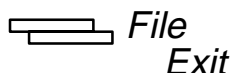


Simulation: Finite Element Modeling User's Guide  
Simulation Techniques and Examples  
Post Processing (picture file tips)  
Post-Processing Results  
Icon Overview for Post Processing  
Displaying Results

### What's next?

The Combining and Graphing Results tutorial covers additional Post Processing skills. The Visualizer tutorial covers additional display tools available on hardware displays.

To exit this tutorial, select:



### 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