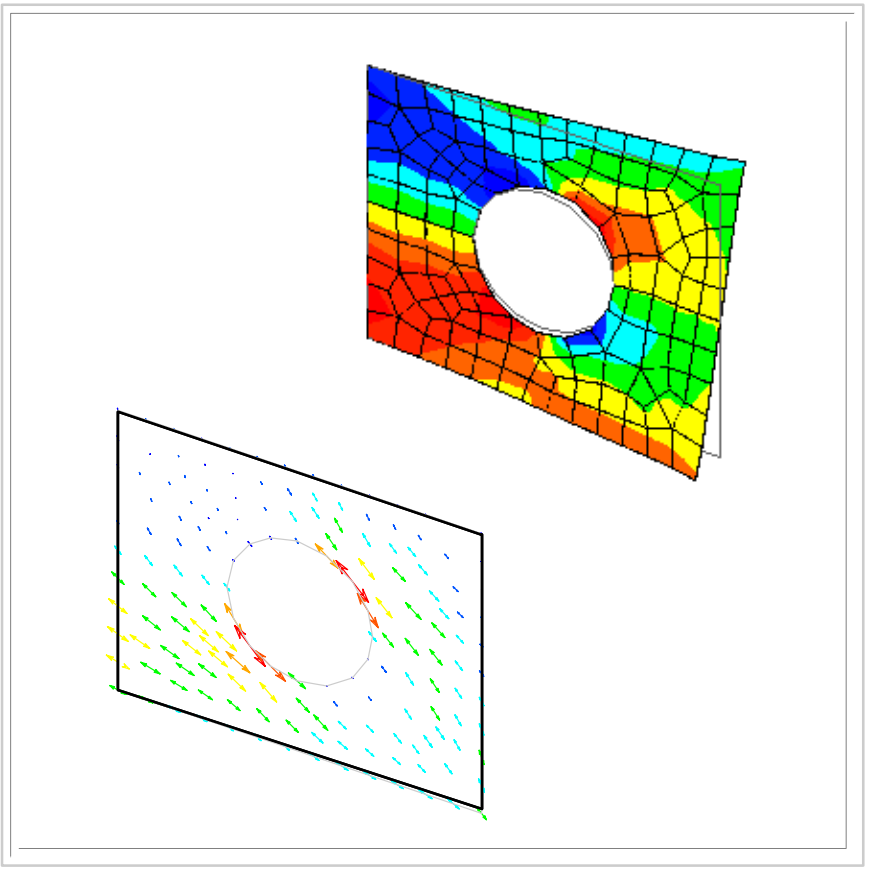


Displaying Results

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.

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

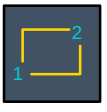
Set your units to mm.

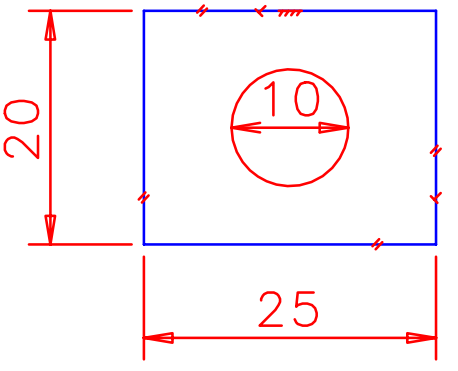
Options
Units

mm (*milli newton*)

What: Sketch this shape to the dimensions shown.

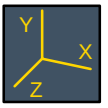
Hint

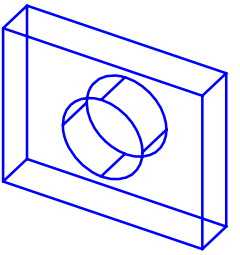




What: Extrude both the rectangle and circle 5mm.

Hint



What: Name the part.

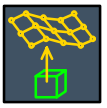
Hint


Name: Plate

What: Create an FE model associated to the part.

Hint

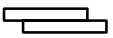

Boundary Conditions




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:

 *No*

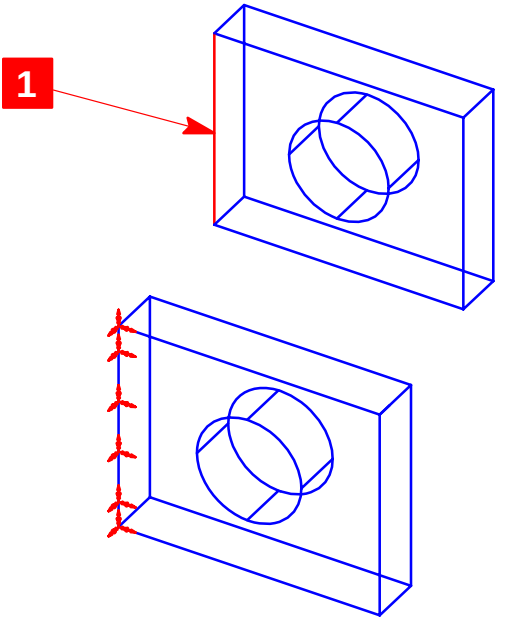
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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 front left edge.

Hint



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

Hint

1

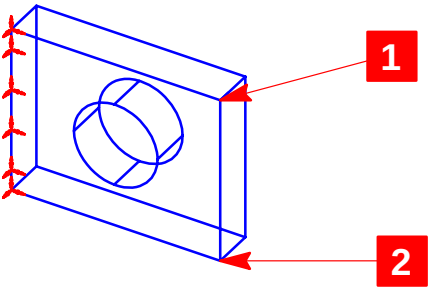
Force on Vertex/Location on Geometry form

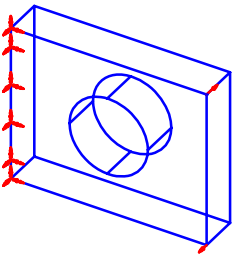
Z Force: –1000

2

Force on Vertex/Location on Geometry form

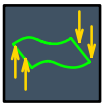
Z Force: 1000





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

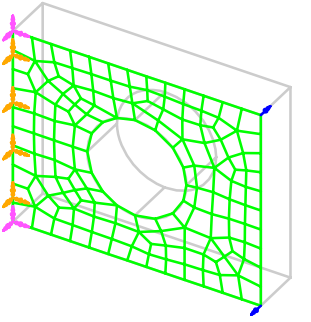
Warning!
If you enter more than one value, all 4 values must be non-zero. To enter a uniform thickness, you need to enter only the first value.

 *Done*

What: Mesh the front surface.

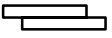
Hint

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

Select results to display 1 of 2

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

How:

Results Selection form


Stress_2... (select)

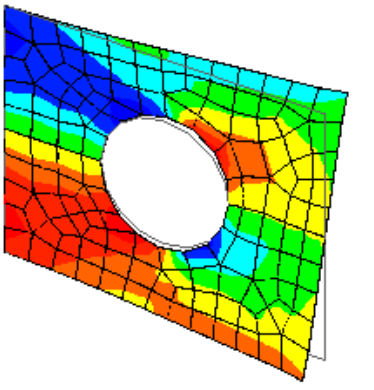

Display Results


Component: Maximum Principal


Displacement_1...


Deformation Results


OK



Select results to display 2 of 2

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...*

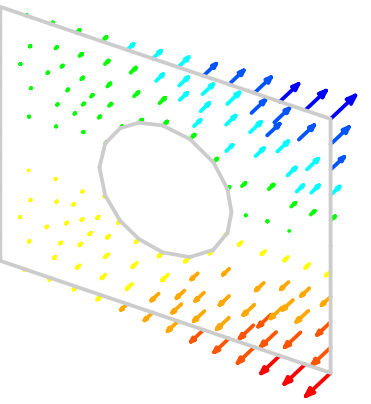
Arrow Options form

 *Longest Arrow (% of screen):* 30

 *OK*

Display Template form

OK



What: Display an arrow plot of maximum principal stress.

How:

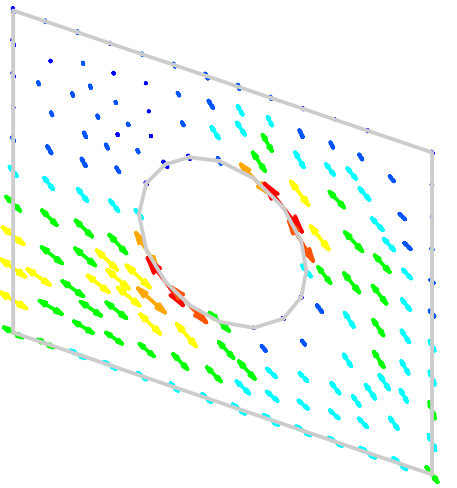
Results Selection form


Stress_2...


Display Results


Deformation Results: *Clear*


OK



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

How:

Display Template form

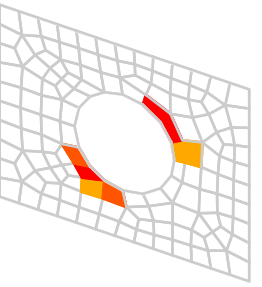

Element...(toggle on)


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

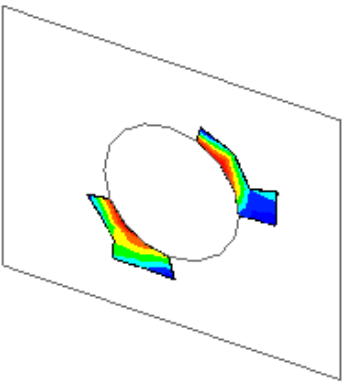
 *Contour...*

 *Stepped Shaded*

 *OK*

 *Use Group*

 *Output*



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

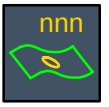
How:

Display Template form


Probe


Contour...


OK



Hint
Click the pointer on various nodes.


Done


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

What: Plot a line contour of stresses.

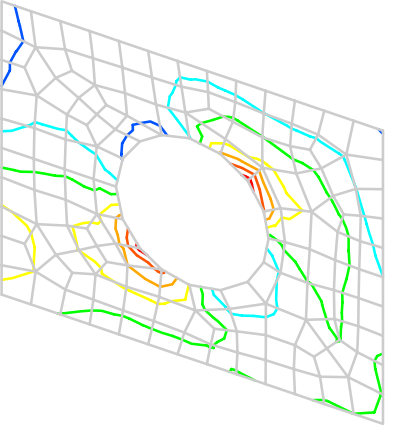
Display Template form

 *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

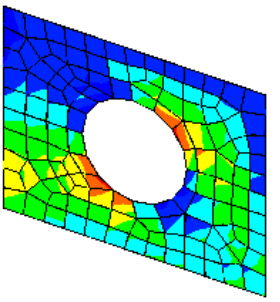

Elements


OK

Display Template form


Stepped Shaded


OK

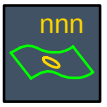


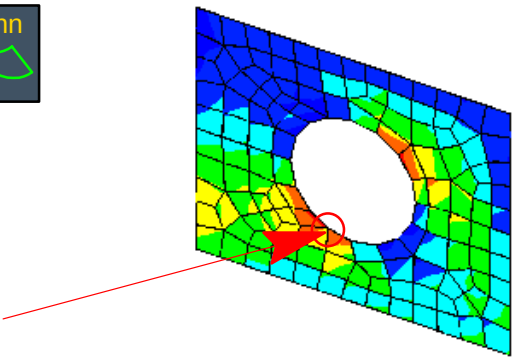
Things to notice

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?

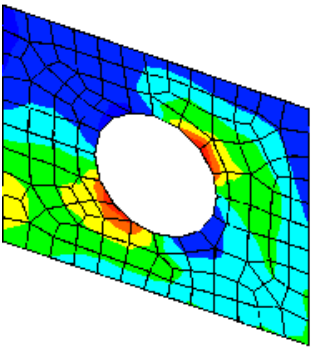
What: Return to normal averaging of stresses.

How:

Calculation Domain form

 *Do Not Average Across:* (toggle off)

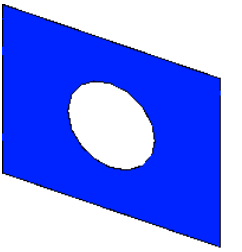
 *OK*



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

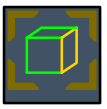
Hint

Shell: Middle

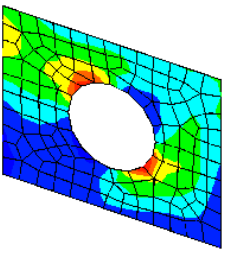


What: Display stresses on the bottom surface of the elements.

Hint



Shell: Bottom



Things to notice

Stresses vary through the shell element thickness.

What: Animate the stresses and displacements.

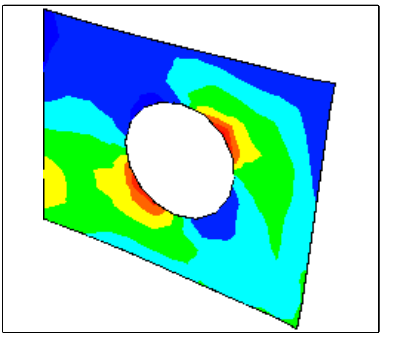
How:

Results Selection form

 Displacement_1...

 *Deformation Results*

 *OK*



 *End* (to stop animation)

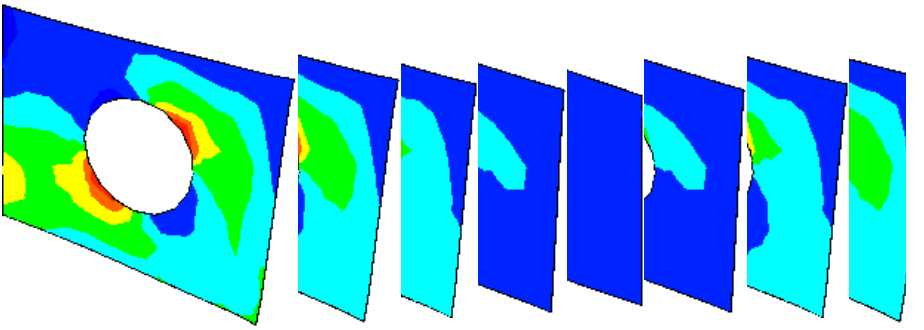
What: Change the number of frames of animation.

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)

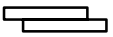
This results in a smoother transition between frames, but you lose some color detail.

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.”

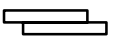
What: Save the model file before exiting this tutorial.



File
Save

See also...

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



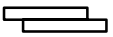
Help, Manuals, Table of Contents

Simulation: Finite Element Modeling User’s Guide
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:



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