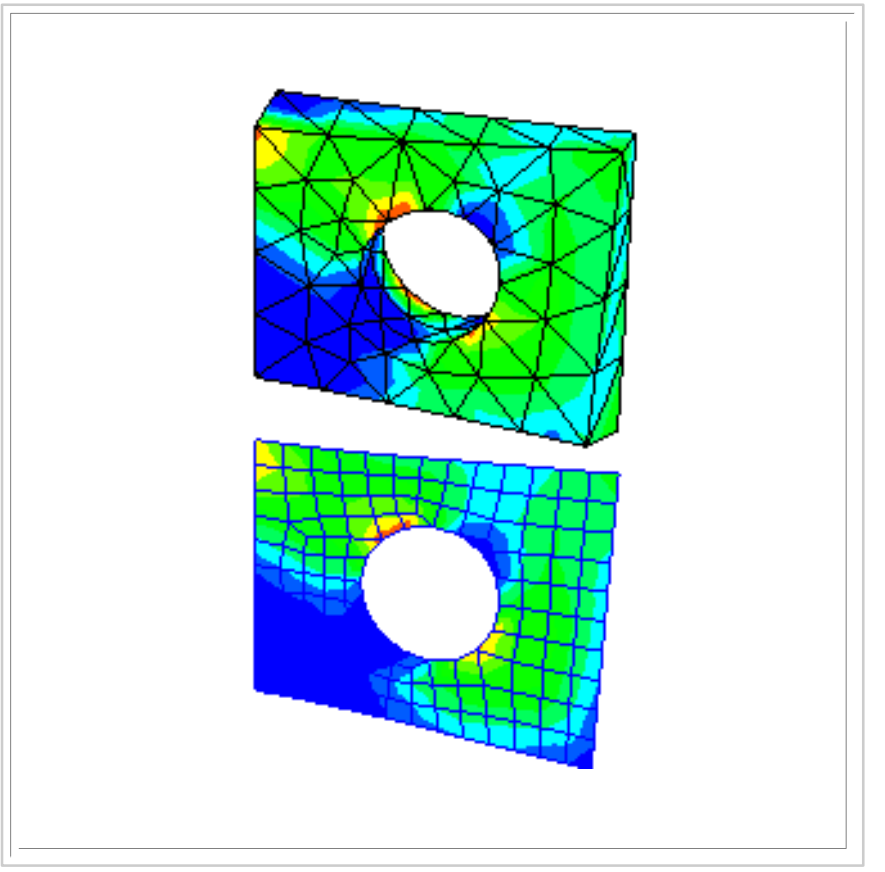


Visualizer

I-DEAS™ Tutorials: Fundamental Skills

This tutorial covers the Visualizer, which is available only if you're using a graphics hardware display such as PEX or OGL (not X3D).

Learn how to:

- create a display
- select results
- manipulate the display
- compare multiple displays

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
- Boundary Condition Surface Loads
- Displaying Results

If you're using the model file saved in the "Displaying Results" tutorial, **skip to page 10**.

Otherwise, open a new model file 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

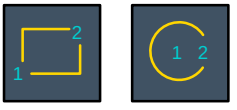
Set your units to mm.

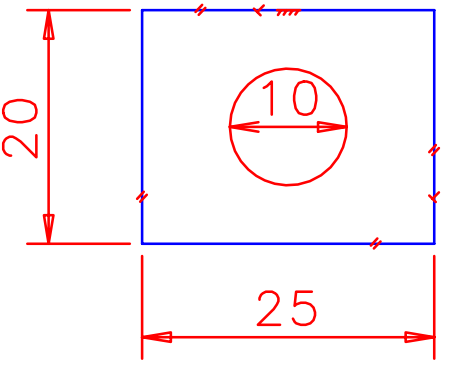
Options
Units

mm (*milli newton*)

What: Sketch this shape to the dimensions shown.

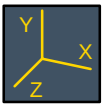
Hint

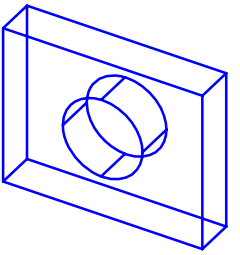




What: Extrude the rectangle and circle 5mm.

Hint



What: Name the part.

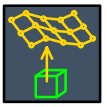
Hint


Name: Plate

What: Create an FE model associated to the part.

Hint

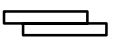

Boundary Conditions




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*

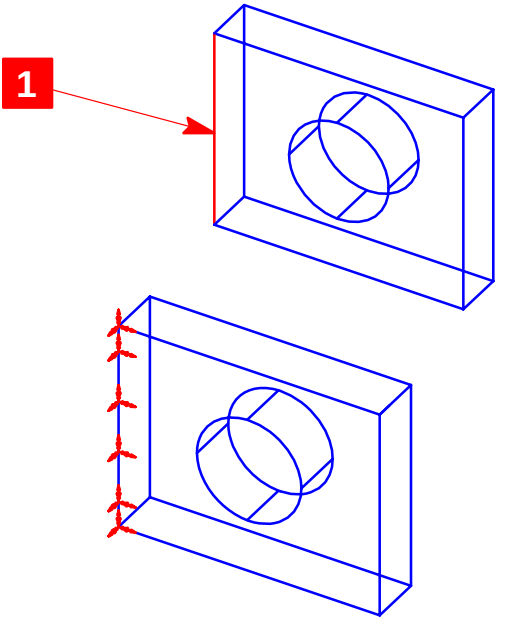
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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

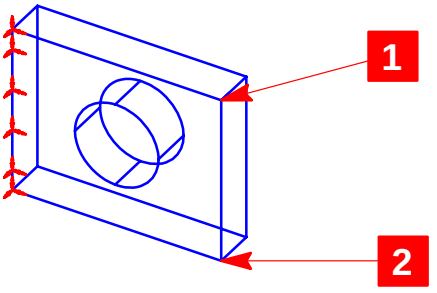
Force on Vertex/Location on Geometry form

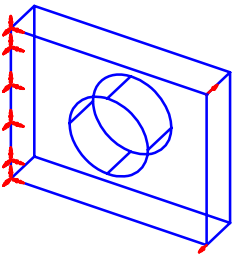
Z Force: –1000

2

Force on Vertex/Location on Geometry form

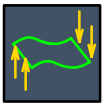
Z Force: 1000





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)—one value at each corner.

 *1st value for thickness*: 5

 <Return> accept all other defaults

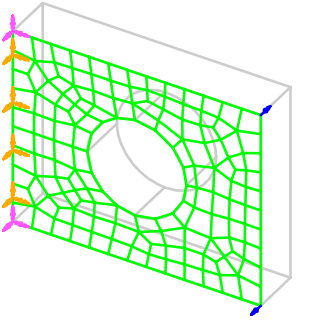
 *Done*

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

What: Mesh the front surface.

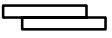
Hint

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

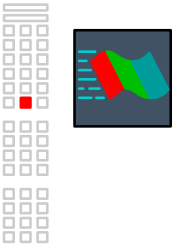
Recovery Point

File
Save

What: Start the Visualizer.

How:


Post Processing



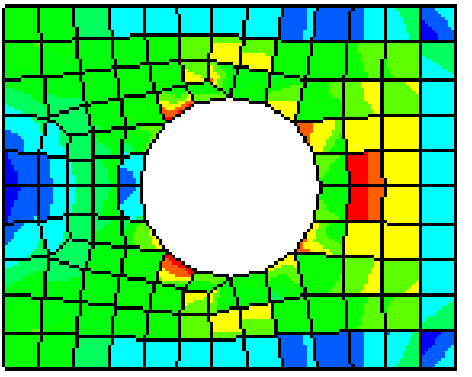
Things to notice

A new window is displayed along with the Visualizer subpanel. Since there's no display defined, the Create Display form is automatically generated.

Create Display form


STATIC TWIST SOLUTION

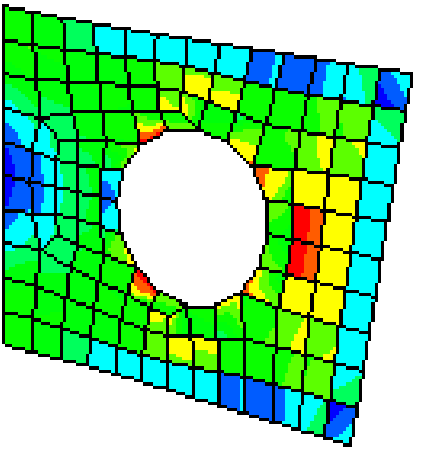

OK



Create a display 2 of 2

What: Use dynamic viewing to orient the display. Only the Visualizer icons work in this special display window.

Hint
F1, F2, and F3



If dynamic viewing doesn't work, try clicking in the Graphics window to establish focus.

What: Select the averaged maximum principal stresses.

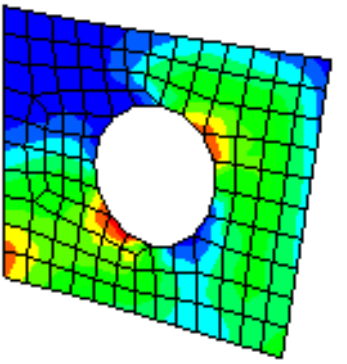
How:

Select Results form


Maximum Principal


Averaged


OK

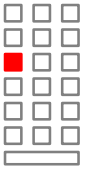


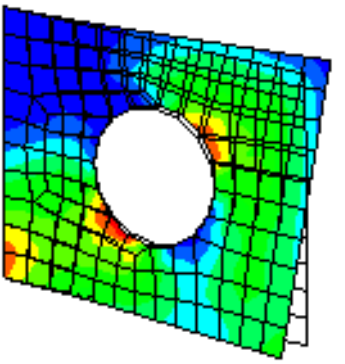
Things to notice

The display automatically changes.

What: Display the undeformed model with the deformed results.

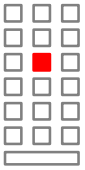
How:

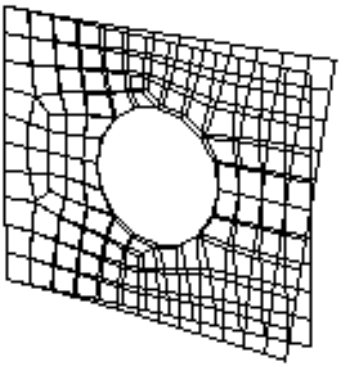




What: Display the deformed and undeformed model without the results.

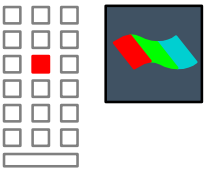
How:

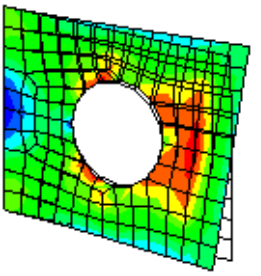




What: Display a contour plot of the stresses.

How:



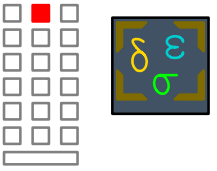


Things to notice

After turning off results, the stress component is reset to *Von Mises*.

What: Select the maximum principal stress component.

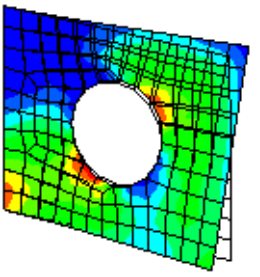
How:



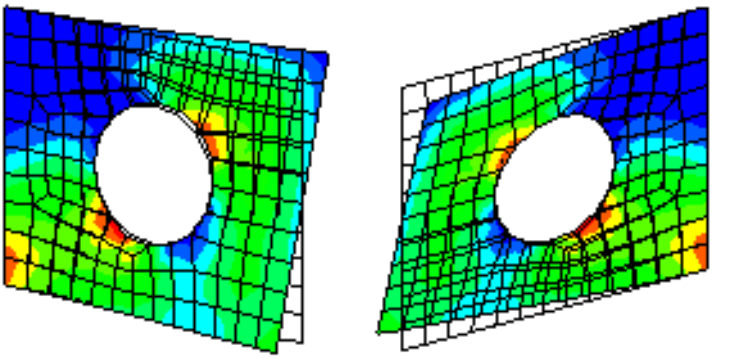
Select Results form

Maximum Principal

OK



What: Rotate the model to see the stresses on the back.

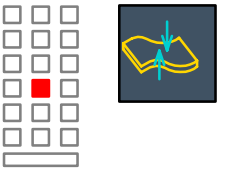


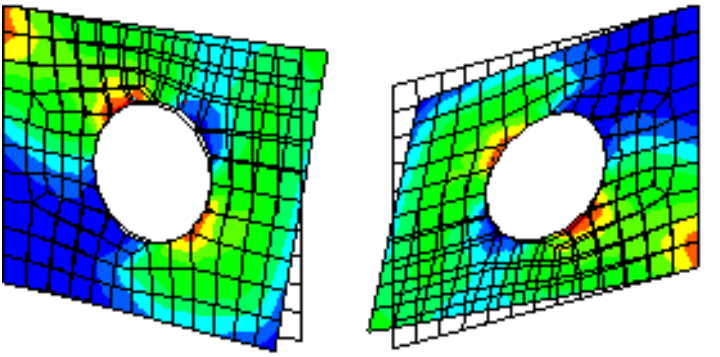
Things to notice

The stresses on the back are the same as the front.

What: Display the top and bottom stresses of the elements at the same time. Rotate the model to view both sides.

Hint





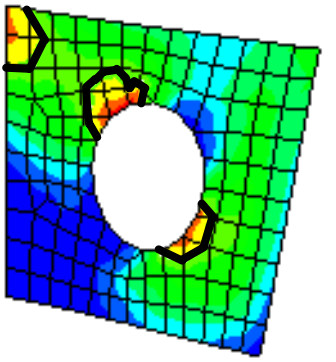
Things to notice

The stresses are different on each side of the model, showing the correct stresses on the top or bottom from the direction of view.

What: Highlight the stress values.

How:


Iso Cursor form



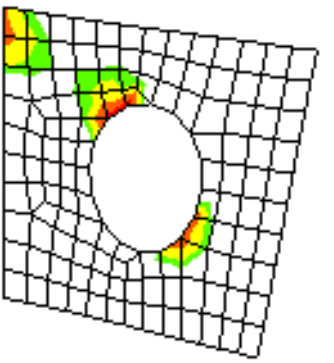
Things to notice

Slide the bar side to side. Notice the black band on the display.

 *Dismiss*

What: Highlight the stresses that are above a given value.

How:

What: Close the Visualizer subpanel.

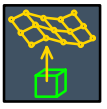
Close

What: Create a second FE model, so there will be two models to compare.

Why: You do this so you can compare a model using solid elements with the thin-shell results.

Hint

Boundary Conditions

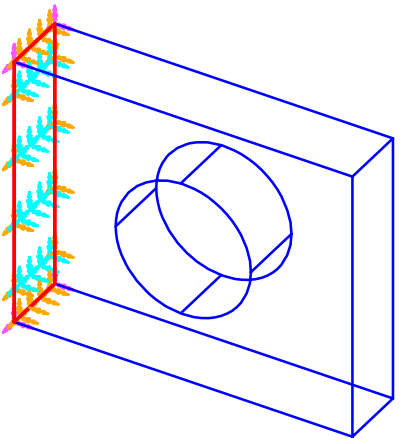


FE Model Name: Solid Model

Geometry Based Analysis Only

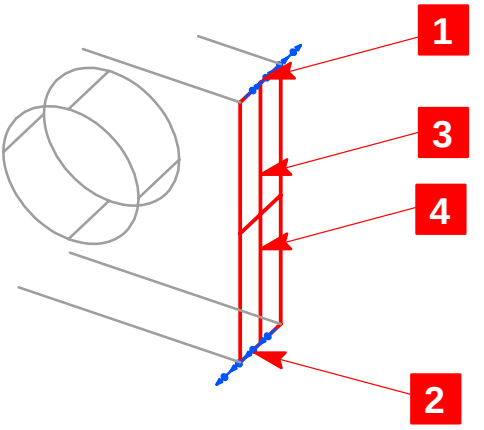
What: Restrain the face shown.

Hint



What: Create twisting loads on the two edges shown.

Hint



1

2 shift–pick

3 pick end face

4 pick end face

 *Load Set*: Twisting Load

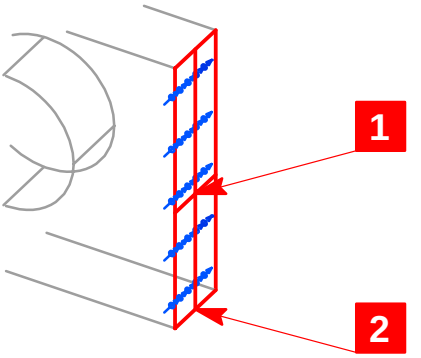
 *Total Force*

 *Shear Force*: 1000

 *OK*

What: Create a second load set to model a bending load.

Hint



1


Load Set: Bending Load


Vector

2


Yes

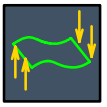

Total Force


Traction: 1000


OK

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

How:



Boundary Condition Set Management form

 *Restraint Set*

 TWISTING LOAD

 BENDING LOAD (Control-pick)

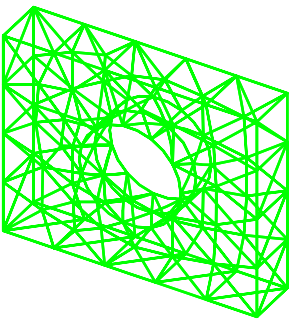
 *OK*

What: Mesh the volume.

Hint

 *Meshing*

 *Element Length*: 5



 *Keep Mesh*

What: Create a solution set.

Hint

Model Solution

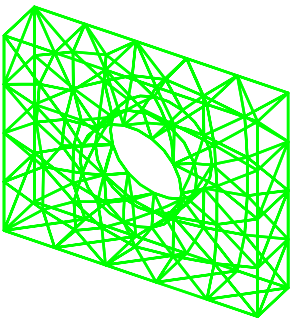
Manage Solution Sets form

Create

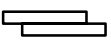
Name: Twisting and Bending

What: Solve the model.

Hint



Recovery Point



File
Save

What: Display the results using the Visualizer.

How:

Post Processing

Create Display form

TWISTING AND BENDING

OK

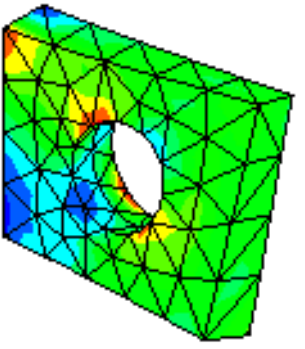
Select Results form

Maximum Principal

Averaged

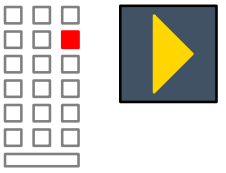
STRESS, TWISTING LOAD

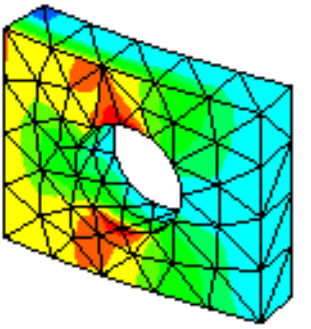
OK

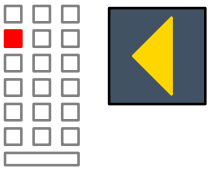


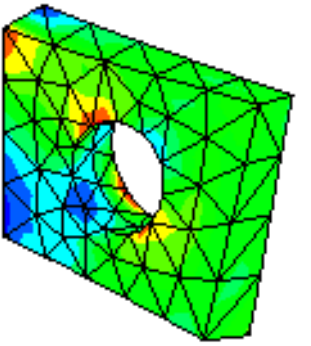
What: Switch to the display of bending stress and deflection (Load 2) and then back to the twisting results.

How:









Things to notice

With multiple results from the same solution set, you can easily switch results using these two icons. This automatically switches stresses and displacements.

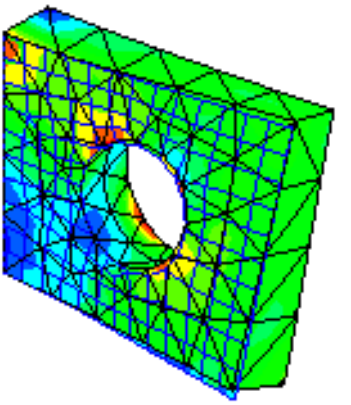
What: Leave the twist results on the display for the next step.

What: Create a second display, adding the results from the first FE model to the same viewport.

How:

Create Display form

STATIC TWIST SOLUTION



OK

Things to notice

There are now two displays on top of each other in the same viewport. The element border color in each display matches the text color for that display.

When you add multiple displays, you can add them to the same viewport, or different viewports. Displays in the same viewport rotate together with dynamic viewing. If you want independent view control, use two viewports.

What: Make the first display the current display.

How:

Current Display form

Display 1

OK

Commands to manipulate the display work on the current display.

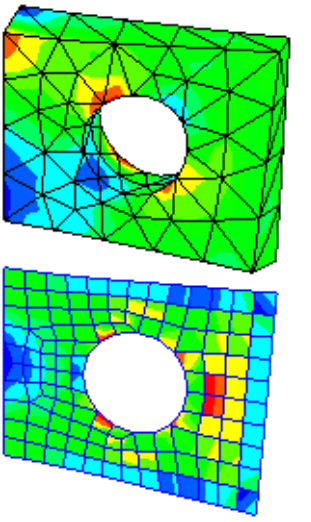
What: Translate the model in Display 1.

How:

Display Settings form

Translate: 0, 25, 0

OK



What: Make Display 2 current, and select the same stress display.

Hint

Current Display form

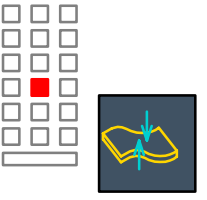
Display 2

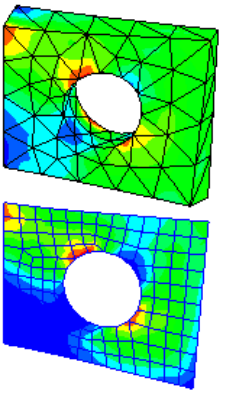
OK

Select Results form

Maximum Principal

Averaged



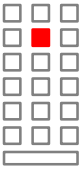


Things to notice

Record the maximum and minimum values. Your values may be different from those shown in the next step if you modeled the part with different dimensions.

What: Set the color bar to the same values in both displays.

How:



Color Bar form

 *Min/Max*: *User Defined*

 *Minimum*: 0

 *Maximum*: 360

 *Overflow*: *Above & Below*

 *OK*

The *Overflow* option means that if there are stresses above or below the color bar, display them as red or blue instead of not displaying them.


Current Display form


Display 1


OK

Color Bar form


Min/Max: *User Defined*

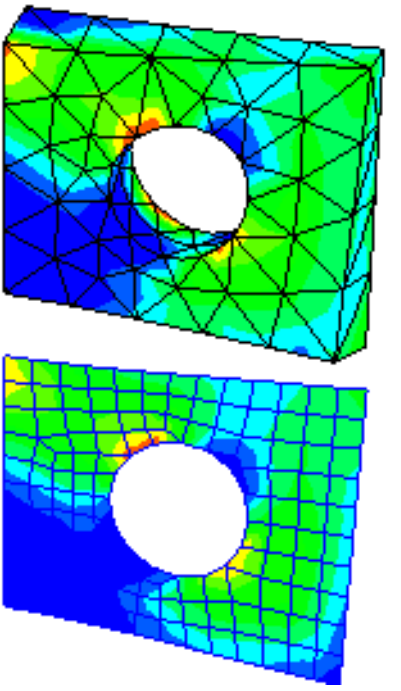

Minimum: 0


Maximum: 360


Overflow: *Above & Below*


OK

Result



Things to notice
The stress contour display now gives a more accurate comparison between the two modeling methods.

What: Close the Visualizer.

How:

Close

Tutorial wrap-up

You have completed the Visualizer tutorial.

You can delete or put away the FE models and the part. This part is not used in any other tutorials.

Hint

See also...

For additional information on the concepts covered in this tutorial, see the following:

Help, Manuals, Table of Contents

Simulation: Finite Element Modeling User's Guide
Viewing Results with the I-DEAS Visualizer
Using the I-DEAS Visualizer
I-DEAS Visualizer Icon and Form Descriptions

What's next?

The remaining Fundamental Skills tutorials present more modeling techniques. After completing these tutorials, use the Advanced Projects tutorials to get an introduction to other element types and 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 File, Exit 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