

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**.

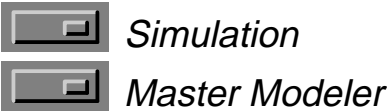
Otherwise, open a new model file and give it a unique name.



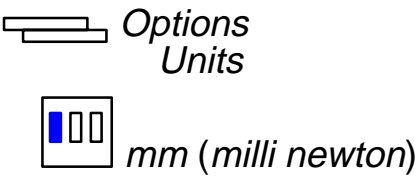
Open Model File form

Model File name: any unique name

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

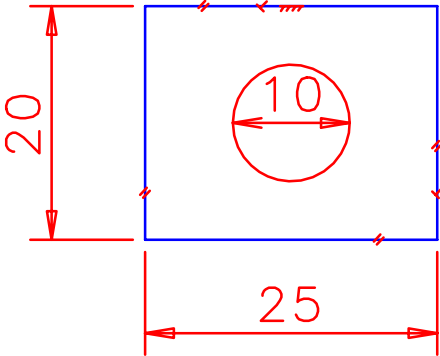
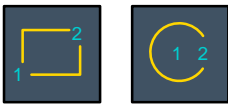


Set your units to mm.



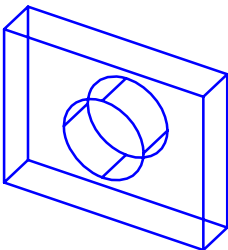
What: Sketch this shape to the dimensions shown.

Hint



What: Extrude the rectangle and circle 5mm.

Hint



What: Name the part.

Hint



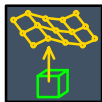
Name: Plate

What: Create an FE model associated to the part.

Hint



Boundary Conditions

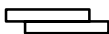


FE Model Name: Thin-shell Model



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:

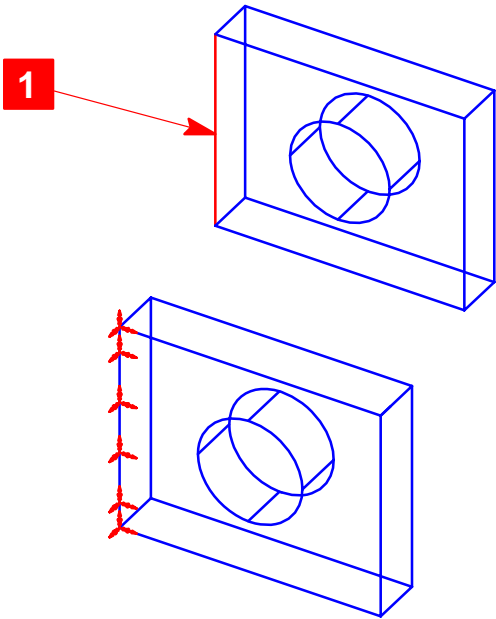
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

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

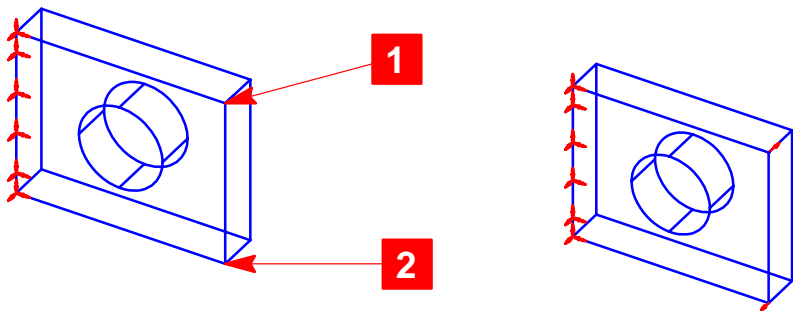
Force on Vertex/Location on Geometry form

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)—one value at each corner.

1st value for thickness: 5

<Return> accept all other defaults



Done

Warning!

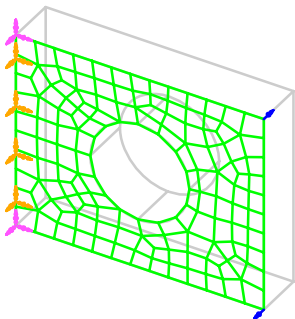
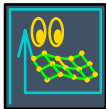
If you enter more than one value, all four values must be non-zero. To enter a uniform thickness at each corner, you need to enter only the first value.

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



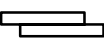
Dismiss

What: Solve the model.

Hint



Recovery Point



File

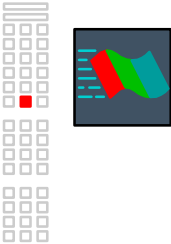
Save

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.

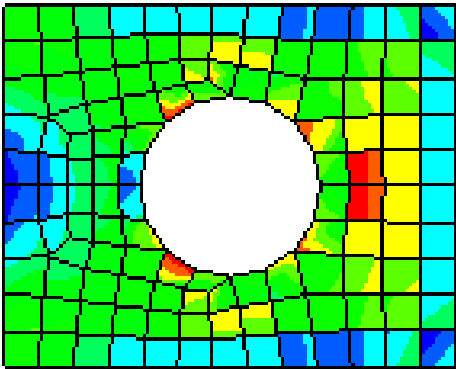
Create Display form



STATIC TWIST SOLUTION



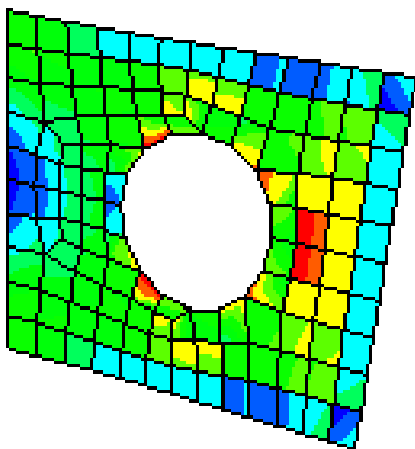
OK




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

Hint

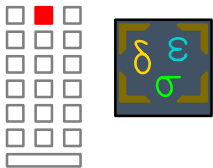
F1, F2, and F3



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

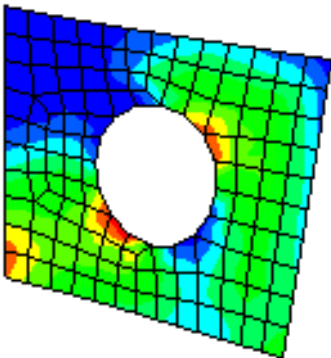
What: Select the averaged maximum principal stresses.

How:



Select Results form

- ☒ *Maximum Principal*
- ☐ *Averaged*
- ☐ *OK*

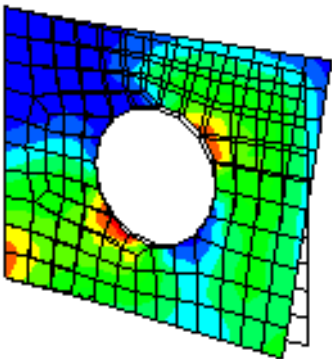
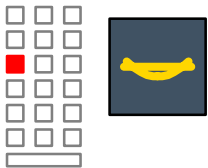


Things to notice

The display automatically changes.

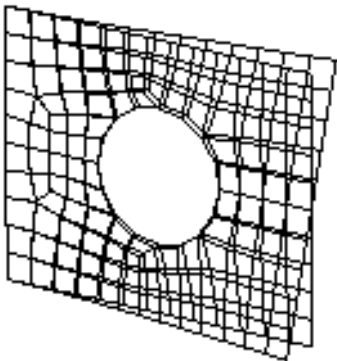
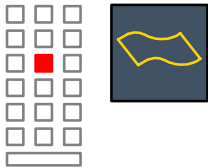
What: Display the undeformed model with the deformed results.

How:



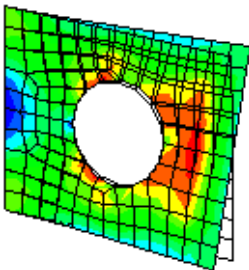
What: Display the deformed and undeformed model without the results.

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:



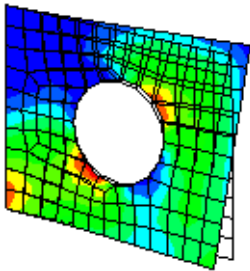
Select Results form



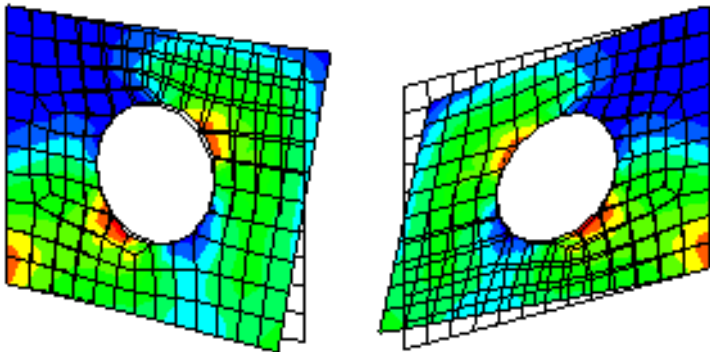
Maximum Principal



OK



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

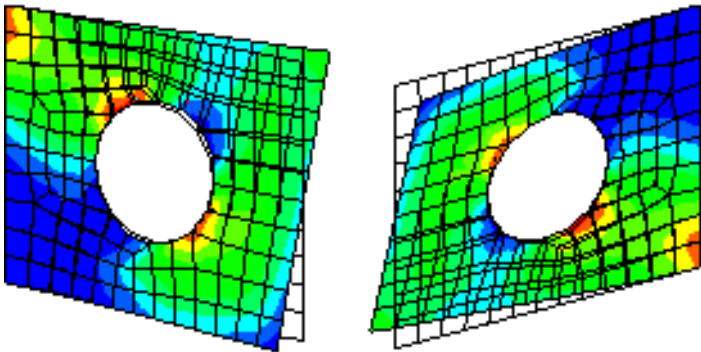


Things to notice

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

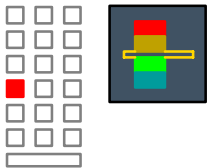


Things to notice

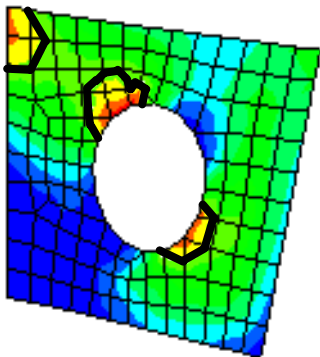
The stresses are different on each side of the model, showing the correct stresses on the top or bottom from the direction of view.

What: Highlight the stress values.

How:



Iso Cursor form



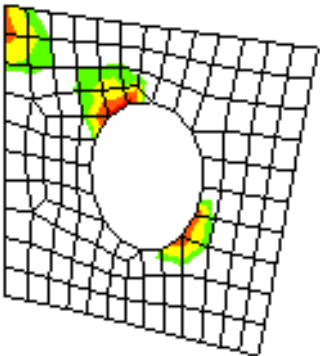
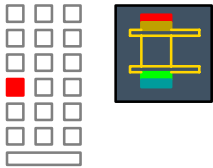
Things to notice

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



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

How:



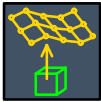
What: Close the Visualizer subpanel.



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

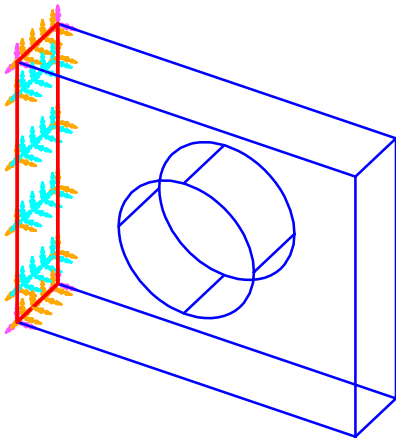
Why: You do this so you can compare a model using solid elements with the thin-shell results.

Hint



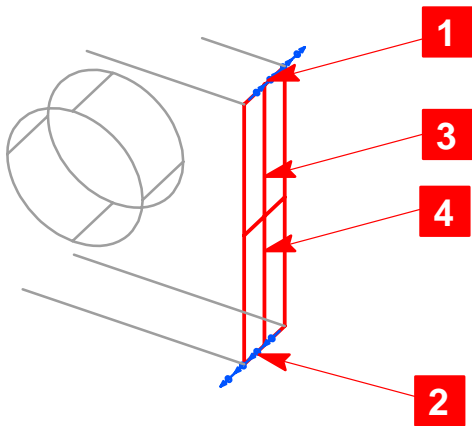
What: Restrain the face shown.

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



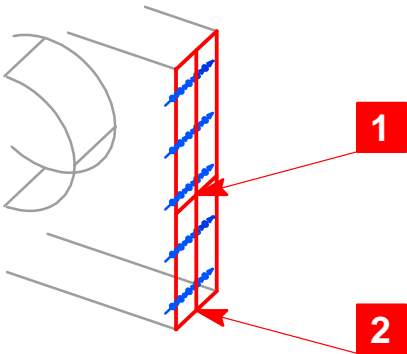
Shear Force: 1000



OK

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

Hint



1



Load Set: Bending Load



Vector



2



Yes



Total Force



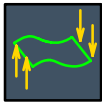
Traction: 1000



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.

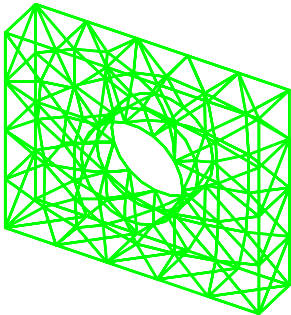
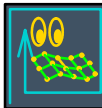
Hint



Meshing



Element Length: 5



Keep Mesh

What: Create a solution set.

Hint



Model Solution



Manage Solution Sets form



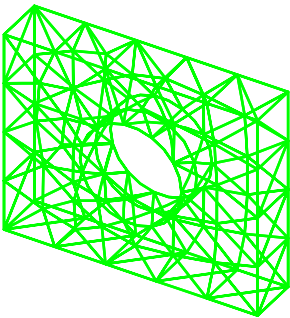
Create



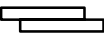
Name: Twisting and Bending

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



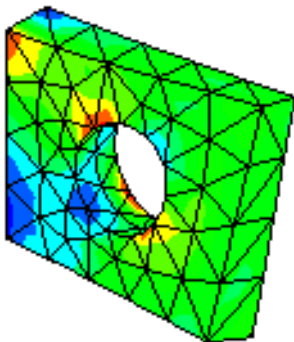
Averaged



STRESS, TWISTING LOAD

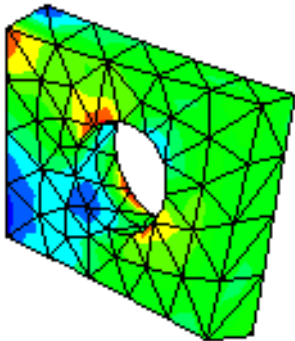
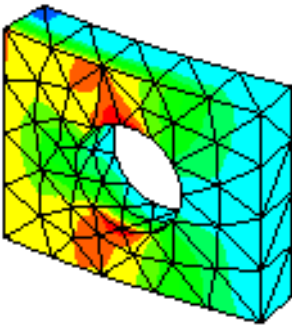


OK



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

How:

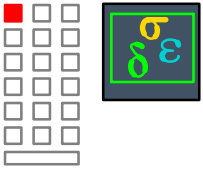


Things to notice

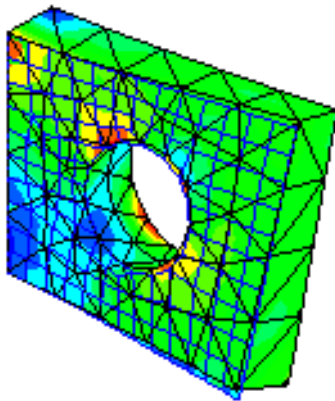
With multiple results from the same solution set, you can easily switch results using these two icons. This automatically switches stresses and displacements.

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

How:



STATIC TWIST SOLUTION



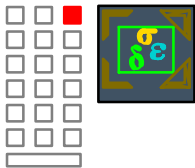
There are now two displays on top of each other in the same viewport. The element border color in each display matches the text color for that display.



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



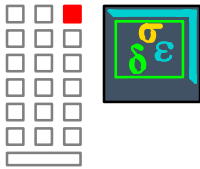
Display 1

OK

 Commands to manipulate the display work on the current display.

What: Translate the model in Display 1.

How:

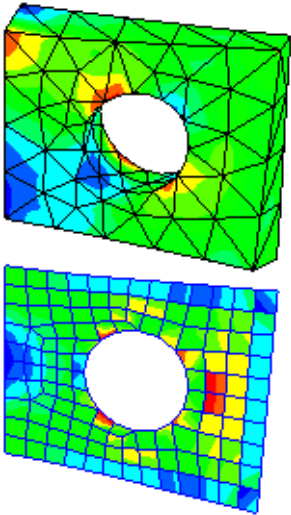


Display Settings form

Translate: 0, 25, 0



OK



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

Hint



Current Display form



Display 2



OK



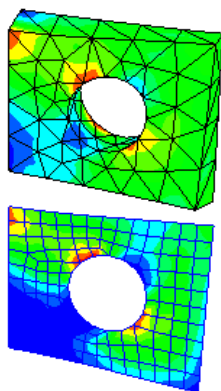
Select Results form



Maximum Principal



Averaged

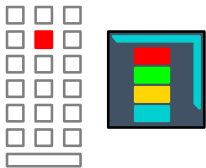


Things to notice

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.

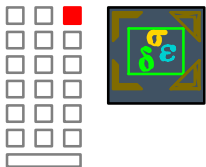
How:



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

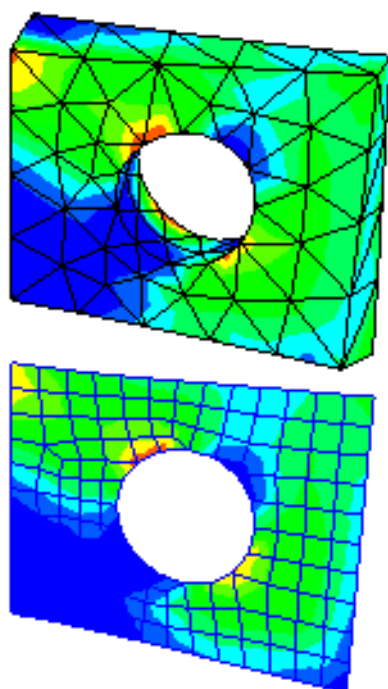


Overflow: Above & Below



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

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