

Parameter Studies

I-DEAS™ Tutorials: Fundamental Skills

A parameter study is a quick way to perform a series of solutions using different values as a design parameter. In this tutorial, you'll study the effect of changing the stress relief radius on a hinge snap.

Learn how to:

- create a design
- create a design parameter
- create output monitors
- solve the parameter study
- display results sets
- graph monitor output

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

Creating Parts

- Introduction to Simulation
- Managing Parts in Model Files
- Free Meshing
- Boundary Condition Sets
- Boundary Condition Surface Loads
- Boundary Condition Symmetry
- Displaying Results



If you're using the model file saved in the tutorial "Optimization Redesign," **skip to page 14.**

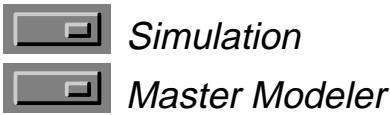
Otherwise, open a new model file and give it a unique name. Then create the part shown on the following Setup pages.

Hint

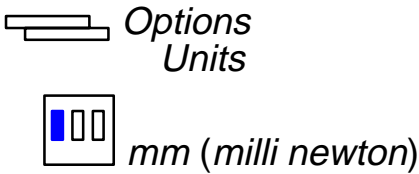


Model File name: any unique name

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

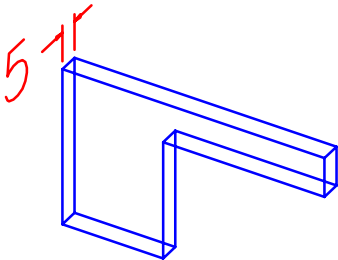
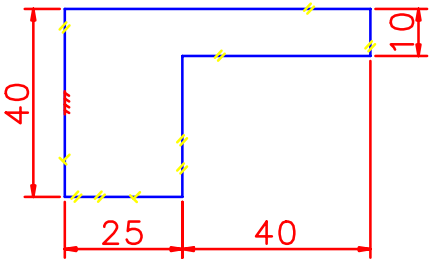


Set your units to mm.



What: Create this part to the dimensions shown.

Hint

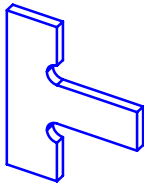


What: Name the part “Hinge Snap.”

Hint



Why: This part represents one half of the plastic tab that acts as a spring in a self-closing hinge. You will use symmetry to reduce the model size.



The goal is to design the tab to give the desired stiffness. The deflection should be 3mm with a force of 10,000 mN. The stress must be less than 20,000 mN/mm**2. A design goal is to minimize the amount of material to reduce cost, weight, and part size.

Save your model file.



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.

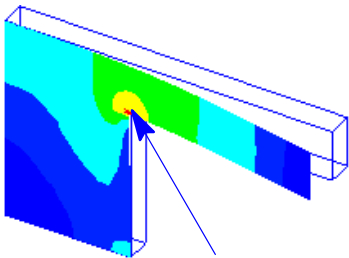
Hint

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

A first analysis shows high stresses in the square corner. (This square corner is actually an infinite singularity.)



You don't need to perform this analysis. This result is shown only to explain why you're doing the step below.



What: Cut a stress-relief in the corner.

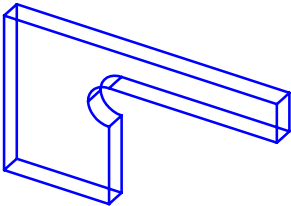
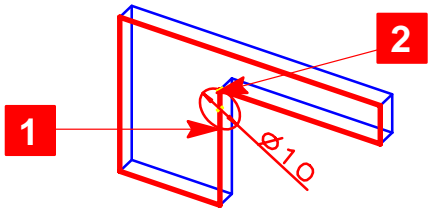
Hint

Sketch a circle on the face, and make a cutout. Make sure that the circle edge is tangent to the corner.



1

2



Why: For parameter studies or optimization redesigns, the program will modify dimensions of the part. It is important that the part is constructed properly so it maintains design intent when it is modified.

What: Verify that your part will update properly.

How:



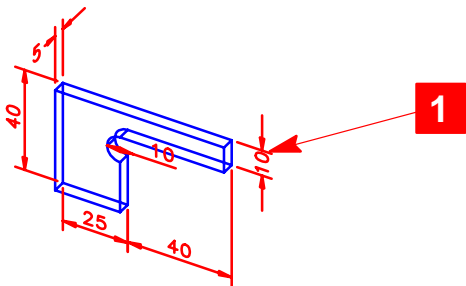
pick part



Show Dimensions



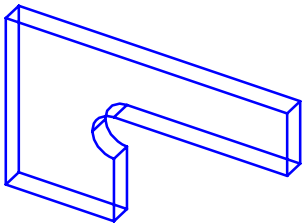
pick dimension



15

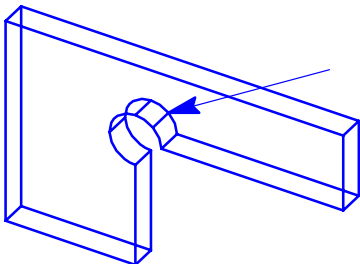


OK





If your part looks more like the one below than the one on the previous page, do not continue. Open your model file to the last save and redo the cutout making sure the stress relief remains tangent in the corner.



Hint

Control-Z

What: If your part is constructed properly, modify the dimension back to 10 before continuing.

How:

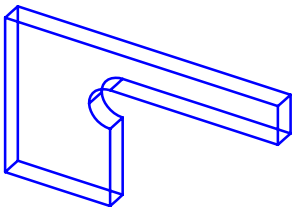
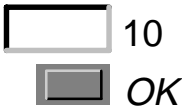
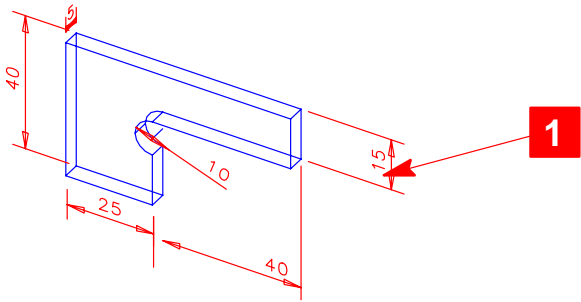


pick part



Show Dimensions

1 pick dimension



What: Create an FE model associated with the part.

Hint

 *Boundary Conditions*

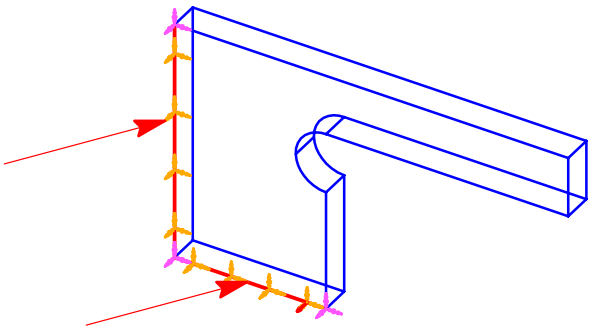


 *Geometry Based Analysis Only*

Warning! Make sure you turn on geometry based analysis, or you won't be able to use geometry parameters for optimization.

What: Fully restrain the two edges shown.

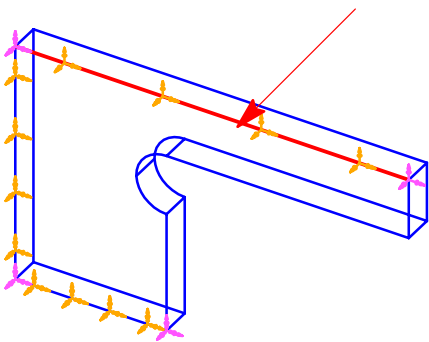
Hint



Why: Since you'll use thin-shell elements to model the part, only the front edge needs to be restrained.

What: Restrain the edge shown to impose symmetric boundary conditions.

Hint



X Translation: Free



Z Translation: Free



Y Rotation: Free



OK

What: Apply a force on the edge shown.

Hint



1 pick edge



2 pick front surface

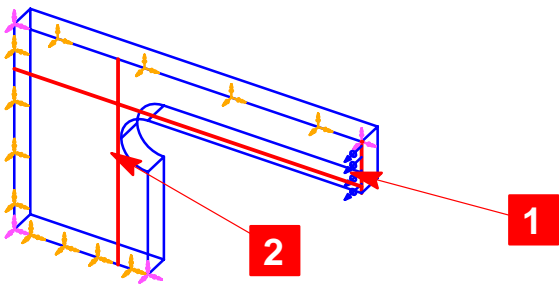


Total Force



Out of Plane Force: -10000

Why: Applying a total force instead of an intensity means that the force will remain constant if the tab width changes. The direction of the force is important if you set a displacement limit as part of an optimization redesign.



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

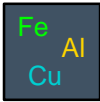
Hint



What: Create a material for the analysis.

How:

 *Meshing*



 *Quick Create*

 Hinge Material

 *Modulus of Elasticity: 2E6*


 *Modify Value*

 *Mass Density: 1.5E-6*

 *Modify Value*

 *Shear Modulus*

 *Null Property*

 *OK (all forms)*

What: Create a physical property defining the thin-shell element thickness.

Hint



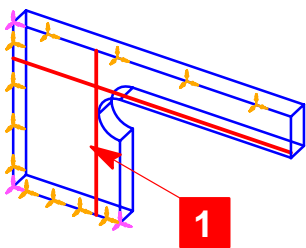
 *1st value for thickness: 5*

What: Define the mesh for the front surface using thin-shell elements.

Hint



1



Define Mesh form

Element Length: 4



Percent Deviation

Percent Deviation: 5

Material



Other

Materials form



Hinge Material

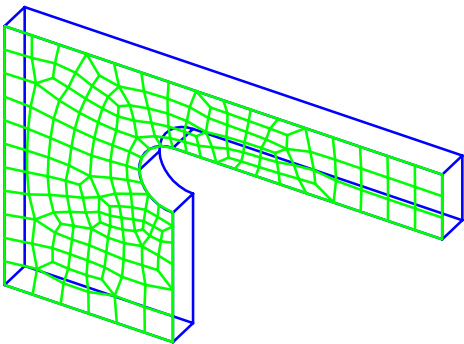
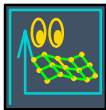


Do not close the Define Mesh form.

What: Generate the mesh on the front surface.

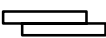
Hint

Define Mesh form



Keep Mesh

Recovery Point

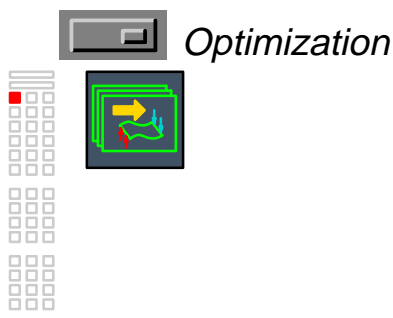


*File
Save*

What: Create a design to perform a parameter study on the effects of changing the stress relief size.

Why: A design is the container for a parameter study or optimization redesign. It may contain multiple solution sets, as well as design parameters and limits.

How:



Manage Designs form



Design form

Name: Radius Study

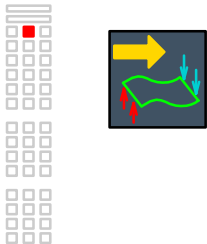
Description: vary stress relief size

☒ *Parameter Study*



What: Create a solution set to describe the analysis to be solved at each step of the parameter study.

How:




Manage Solution Sets form

 *Create...*

Solution Set form

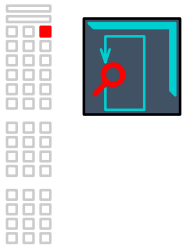
 *OK*

 *Dismiss*

 Parameter studies can include linear statics, normal mode dynamics, linear buckling, and heat transfer. You may create more than one solution set to perform different types of solutions for each step of the design variable.

What: Define the number of steps for the parameter study.

How:



Step Control form

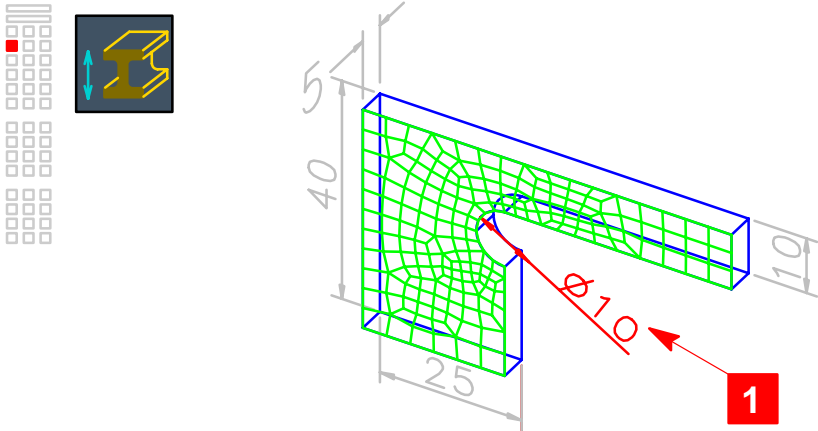
Total Number of Steps: 5

Things to notice

Starting from the present value, the parameter being studied will be changed two steps in each direction.

What: Create a design parameter.

How:



Manage Design Parameters form



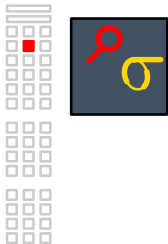
Design Parameter form



Why: With two steps in either direction (set on the previous page), this step size will result in values of 2, 6, 10, 14, and 18 for the stress-relief diameter.

What: Create a stress monitor to track the maximum stress in all the elements for each parameter step.

How:



Manage Stress Monitors form



Stress Monitor form



Things to notice

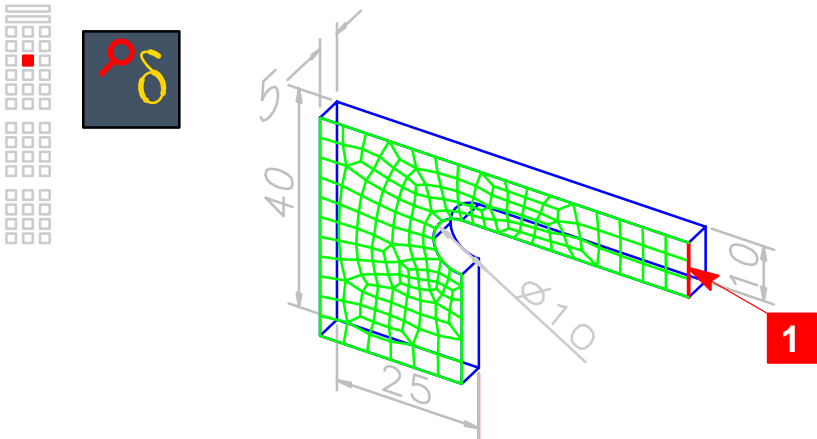
You can create multiple stress monitors to graph the stress in different locations in the model as the design parameter changes.



In a parameter study, you can only monitor stress. In an optimization redesign, you can create limits on stress.

What: Create a displacement monitor to track the displacements for each parameter step.

How:



Manage Displacement Monitors form

 Create...

Monitor Displacement form

 Displacement Direction: Z

 Select Geometry

1

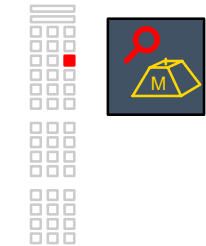
 (Done)

 OK

 Dismiss

What: Create a mass monitor.

How:



Manage Mass Monitors form



Mass Monitor form



Things to notice

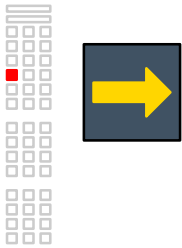
Mass is useful to monitor, since it represents the amount of material used in the part. Reducing material reduces both cost and size.

Recovery Point



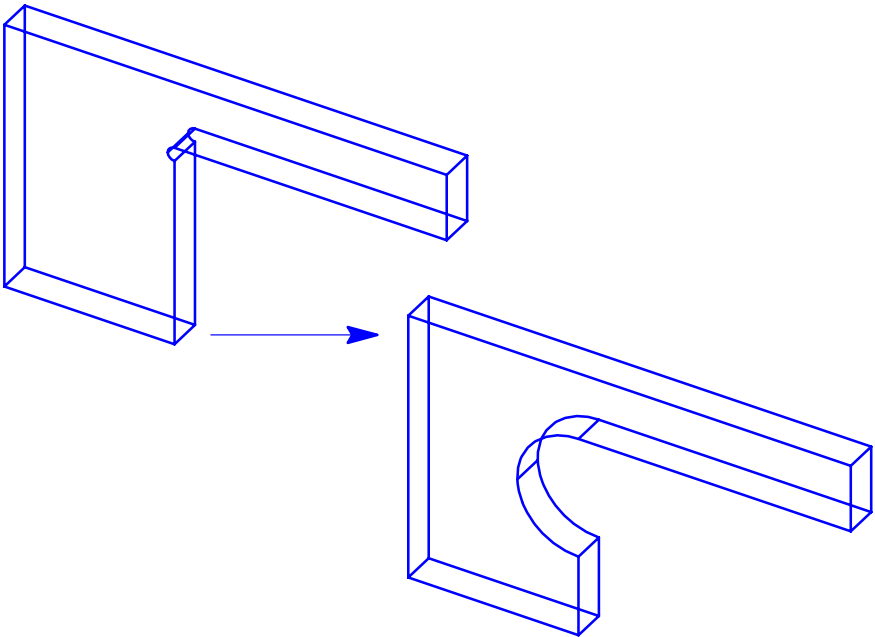
What: Solve the parameter design study.

How:



Things to notice

During the solve, you can watch the part shape change during each step.



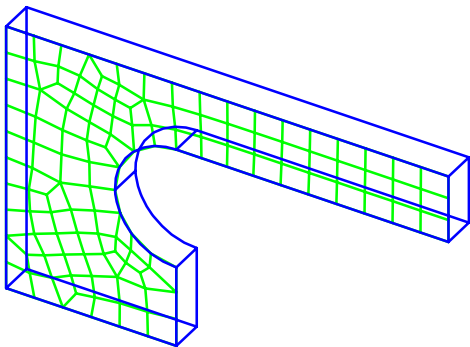
What: After the solve, report any errors and warnings.

How:



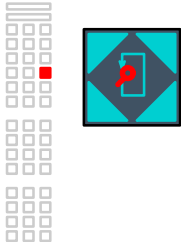
Check I-DEAS List.

Notice: The warnings are informational only.



What: Examine some of the stored steps. See how the model and the mesh change with each value of the parameter.

How:



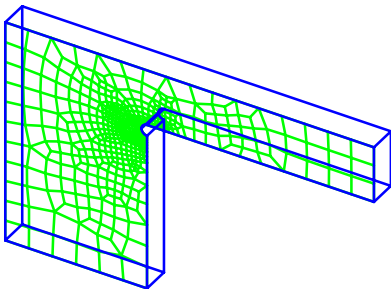
Manage Steps form



Step1



Current



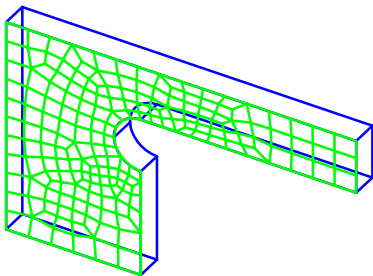
Step3 (Original)



Current



Dismiss



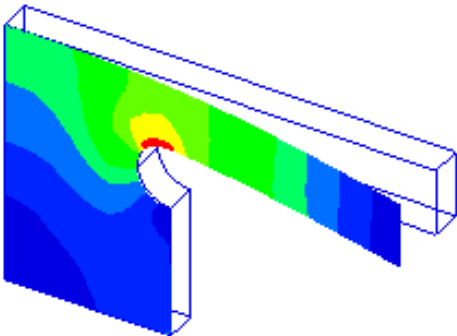
Things to notice

You can display the stress and displacement results for any step containing stored results by making the step current and then switching to the *Post Processing* task.

Hint



Post Processing



If you want to manipulate the part model, leave the original step as the current step on the display before you switch out of the *Optimization* task. The other step configurations are stored only in *Optimization*, not in the bins like other parts.

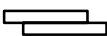
What: Switch back to the *Optimization* task before continuing the tutorial.

Hint



Optimization

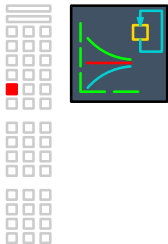
Recovery Point



*File
Save*

What: Display a graph of the stress as a function of the stress-relief diameter.

How:

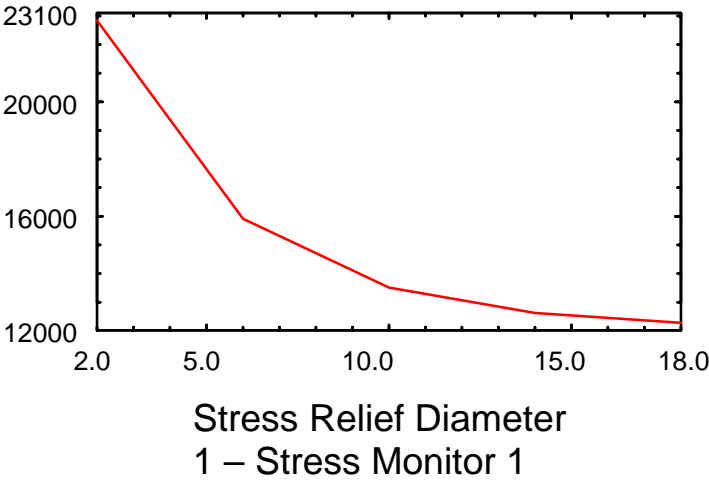


Parameter Study Histories form



Stress Monitor 1

Parameter Study Stress History



Things to notice

The stress-relief diameter has a large effect on the stress. If the diameter is above 4mm, the stress is below the allowable value of 20,000.

What: Graph all three monitors together.

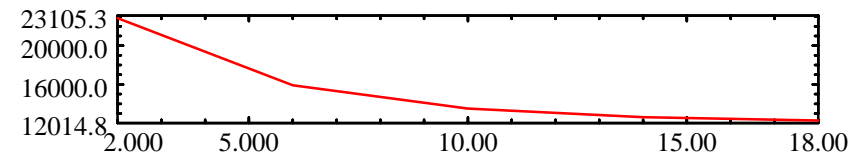


Stress Monitor 1
Displacement Monitor 1 (Control-pick)
Mass Monitor 1 (Control-pick)

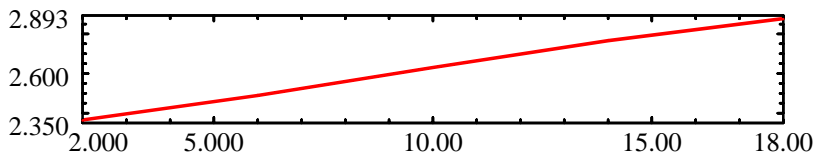


Dismiss

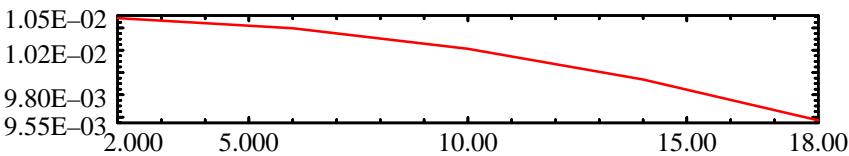
Stress



Displacement



Mass



Things to notice

The displacement is lower than the target of 3mm for all values of the diameter. Other parameters will need to be modified to raise the displacement to the desired value. In the tutorial “Optimization Redesign,” you’ll perform a redesign analysis to vary the design parameters to increase this displacement.

Hint

To display the part again, click on any display-type icon, such as *Line* or *Hidden*.

Tutorial wrap-up

You have completed the Parameter Studies tutorial.



Don't delete the FE model or part.

What: Make sure you save your model file. This part is also used in the tutorial “Optimization Redesign.”

See also...

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

 *Help, Manuals, Table of Contents*

Simulation: Model Solution/Optimization User's Guide
Using Optimization

Overview and Concepts...

Definitions and Concepts of Parameter Study

Understanding Differences between
Redesign and Parameter Study

What's next?

Other Advanced Projects tutorials introduce different element types and solution methods. The Optimization Redesign tutorial continues with the part from this tutorial.

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