

Thermal Analysis

I-DEAS™ Tutorials: Fundamental Skills

This tutorial demonstrates using linear conduction thermal analyses on a circular saw blade.

Learn how to:

- create a thermal model
- solve and display results
- calculate thermal stresses
- analyze thermal buckling

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

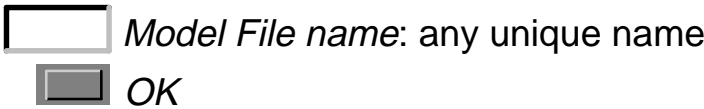
Creating Parts

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

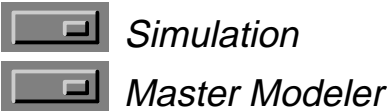
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



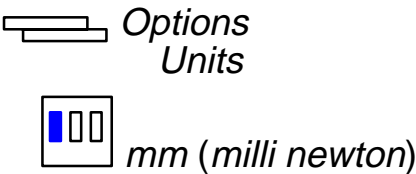
Open Model File form



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

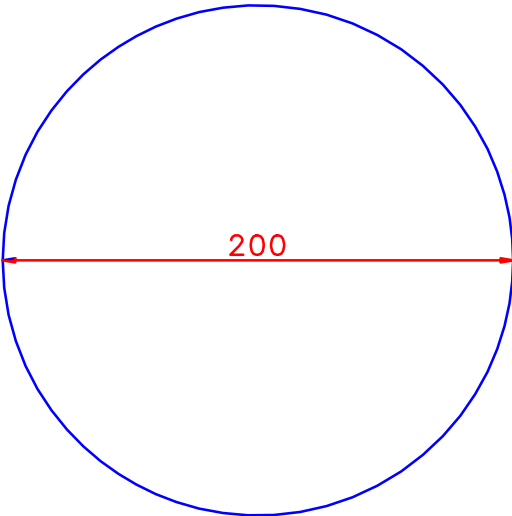


Set your units to mm.



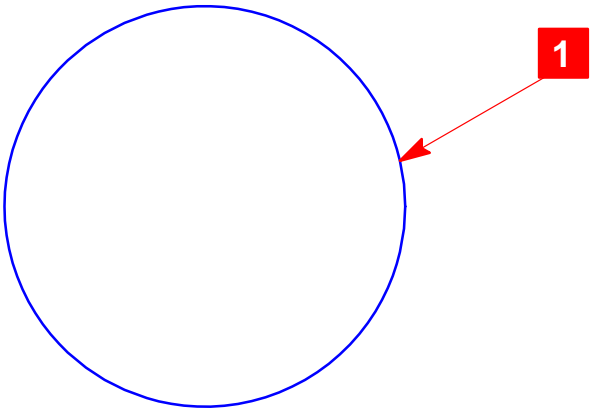
What: Sketch a circle with a radius of 100mm.

Hint



What: Create a surface bounded by the circle.

Hint



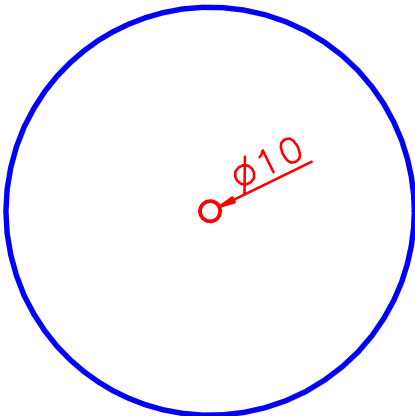
1



Yes

What: Sketch an inner circle on the surface with a radius of 5mm.

Hint



What: Cut the inner circle from the part.

Hint



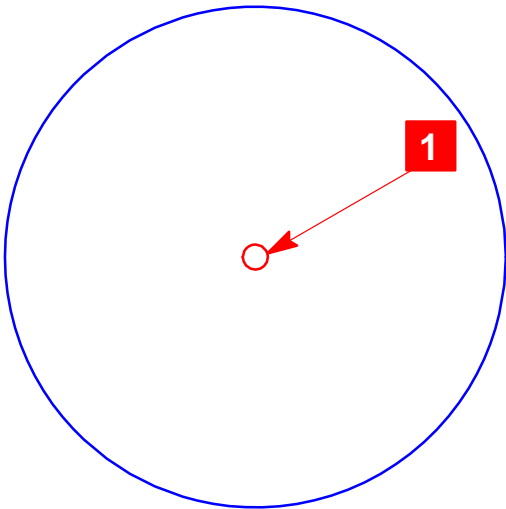
1



Done



Cutout



What: Name the part.

Hint



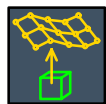
Name: Blade

What: Create an FE model.

Hint

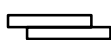


Boundary Conditions



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:

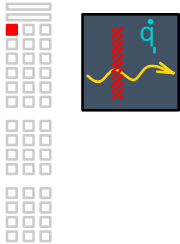
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

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

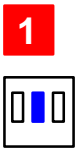
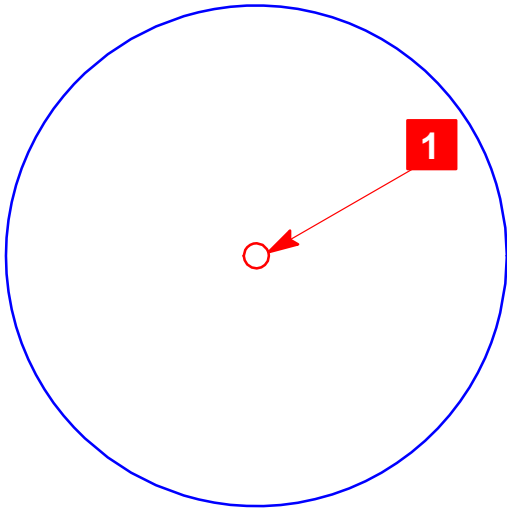
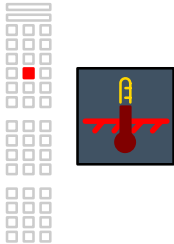
What: Create a temperature restraint on the inner edge.

How:



Things to notice

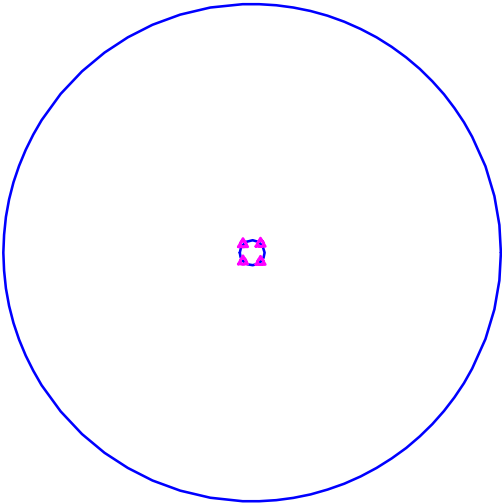
By defining the type of analysis first, icons that don't apply are made inactive.



Continued on next page...

Temperature Restraint on Edge form

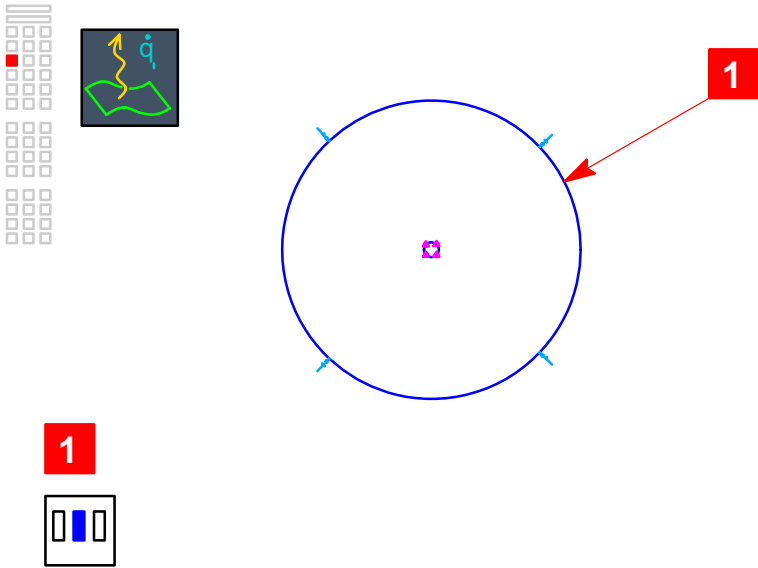
- ☐ *Restraint Set: Thermal Restraint*
- ☒ *On Edge of...: Thin Shell*
- Temperature (Middle): 20*
-



Why: The blade is bolted tightly to a solid shaft which will conduct heat from the blade. This is modeled as a temperature restraint.

What: Create a heat flux on the outer edge.

How:




Heat Flux on Edge form

Load Set: Thermal Load

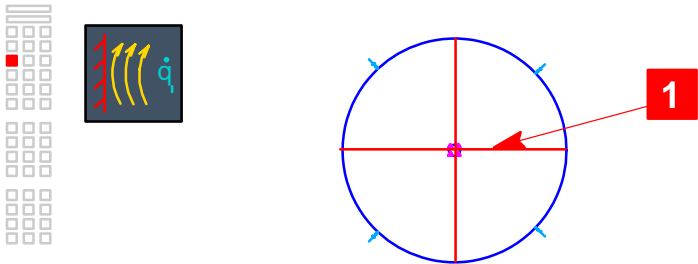
Heat Flux: 400000

Why: The heat flux value for the edge represents 250 watts of power being dissipated by the cutting friction divided by a circumference of (.628m) = 400 W/(m sec) in SI units. In (mm–milli Newton) units, this value is 400,000.

 Check the element library documentation for a description of units for the type of element you're using. Heat flux is expressed differently on surfaces or edges.

What: Define convection on both sides of the surface.


How:



1 pick surface



Convection on Surface form

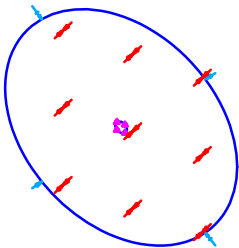
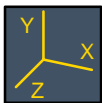
 *On Face of...: Thin Shell*

Convection Coef (Top): 100

Surrounding Temp(Top): 20

Convection Coef (Bottom): 100

Surrounding Temp(Bottom): 20



For help calculating convection values, see Maya's Thermal Lab at www.mayahtt.com.

What: Create a thermal analysis boundary condition set.

How:



Boundary Condition Set Management form



Thermal



Heat Transfer



Restraint Set



Restraint Set: Thermal Restraint

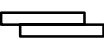


Load Sets: Thermal Load



OK

Recovery Point



File
Save

What: Create a physical property table.

Hint



Meshing



1st value for thickness: 1.4

What: Mesh the surface.

Hint



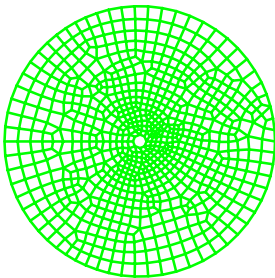
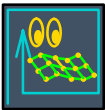
Element Length: 10



Free Options...

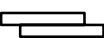


Percent Deviation: 10



Keep Mesh

Recovery Point



*File
Save*

What: Create a solution set.

Hint



Model Solution



Create

Solution Set form



Name: Thermal Solution



Type of Solution: Heat Transfer

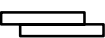


OK



Dismiss

Recovery Point



*File
Save*

What: Solve the model.

Hint




What: Check for errors.

Hint



Things to notice

The informational warning states that a temperature load set has been stored for subsequent structural analysis. This set will be used to compute stresses and buckling load factors later in this tutorial.

 To modify the model, you'll need to delete this temperature set as well as the result sets.

What: Display the resulting temperatures.

Hint



Post Processing



Temperature



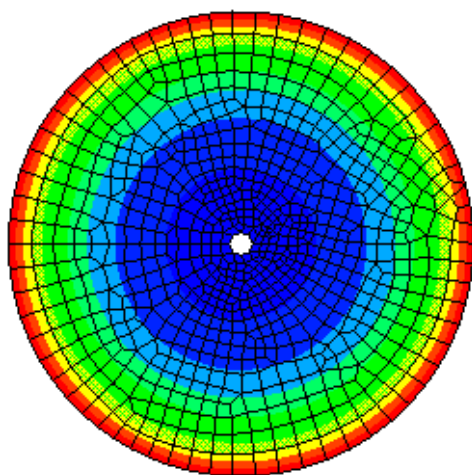
Display Results



Deformation Results: Clear



OK



Things to notice

What's the maximum temperature at the edge of the blade?

What: Display the heat flux as an arrow plot.

Hint



Heat Flux



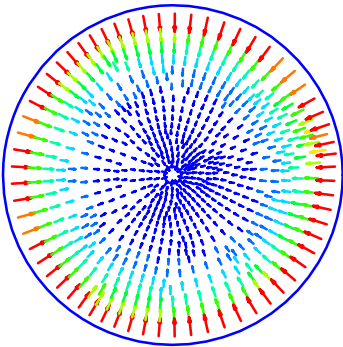
Display Results



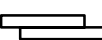
OK



Arrow



Recovery Point



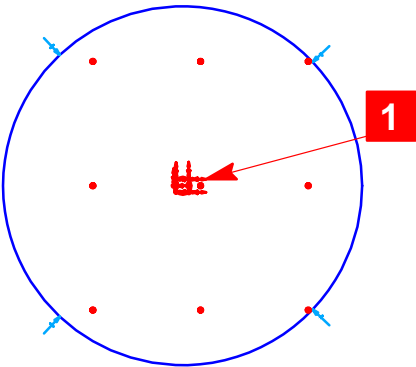
File
Save

What: Restrain the center edge for the static analysis.

Hint



Boundary Conditions



1 pick center edge



Displacement Restraint on Edge form



Restraint Set: Static Restraint



OK

Why: You must enter a new name for the restraint set, to create a new set. Otherwise, you'll be attempting to add the static restraint to the thermal restraint set, which is locked by the results.

Things to notice

Elements are not shown here to simplify the illustration.

What: Create a boundary condition set for linear statics.

How:



Boundary Condition Set Management form



Static



Linear Statics



Restraint Set



Restraint Set form



Static Restraint



OK

Temperature Set



Temperature Set form



Load 1 Temps/Grads



OK



Don't select any load set.



OK

What: Create a solution set.

Hint



Model Solution



Manage Solutions Sets form



Create

Solution Set form

Name: Static Solution



Type of Solution: Linear Statics

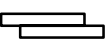


OK



Dismiss

Recovery Point



*File
Save*

What: Solve the static analysis.

Hint



What: Display the resulting stresses.

Hint



Post Processing



Stress



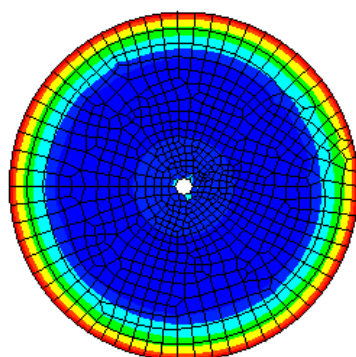
Display Results



Deformation Results: Clear



Contour



Things to notice

Is the blade in compression or tension in the outer edge?
(Display maximum and minimum principal stresses to see the sign.)

What: Display the displacements.

Hint



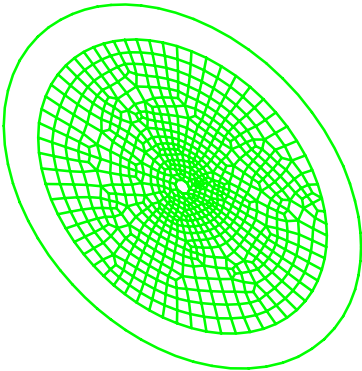
Display Results: Clear



Displacement



Deformation Results

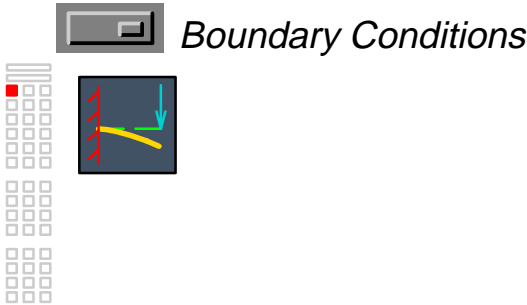


Things to notice

The blade expands in the plane. Buckling behavior isn't predicted unless you perform a buckling analysis.

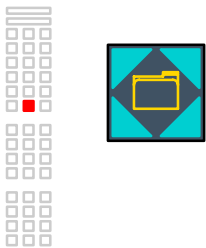
What: Prepare to create a boundary condition set for linear buckling.

How:

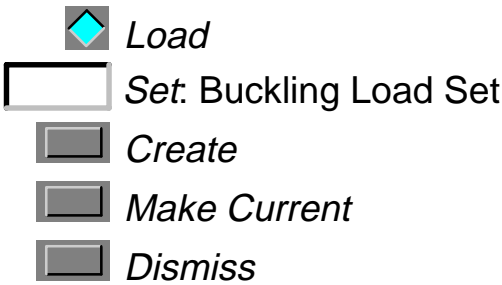


What: Create a new load set.

How:



Set Management form



Why: A load set is required for buckling, even though no loads will be supplied in addition to the thermal load set.

What: Create a boundary condition set for linear buckling.

Hint



Boundary Condition Set Management form



Buckling



Linear Buckling



Restraint Set



Restraint Set form



Static Restraint



OK

Temperature Set



Temperature Set form



Load 1 Temps/Grads



OK



Load Sets: Buckling Load Set



OK

What: Create a solution set.

Hint



Model Solution



Manage Solutions Sets form



Create

Solution Set form

Name: Buckling Solution

Warning!

Be sure to define the type of solution, since buckling may not be the default in this case.



Type of Solution: Linear Buckling



OK



Dismiss

What: Solve the buckling analysis.

Hint



What: Display the buckling shapes and load factors.

Hint



Post Processing



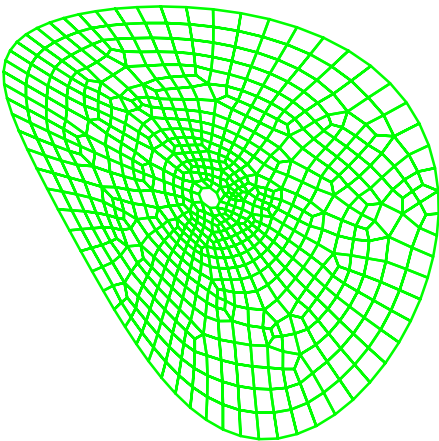
Display Results: Clear



Normal Mode 1



Deformation Results

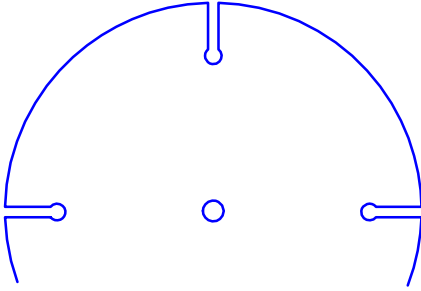


Things to notice

What's the buckling load factor? If it's less than 1.0, buckling is predicted.

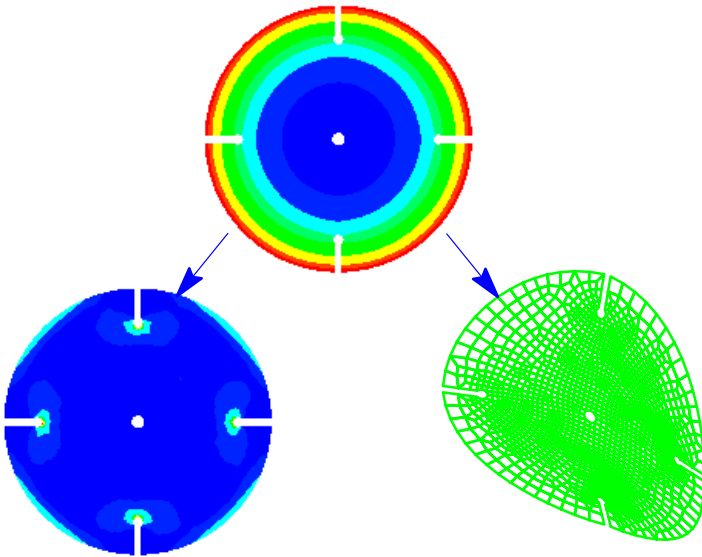
On your own...

What: To review the steps covered in this tutorial, modify the blade and repeat the analysis.



Hint

Make a copy of the blade part and add stress relief cuts. Rerun the thermal analysis, then the static and buckling analyses, using the new temperature loads.



Things to notice

Although the stress relief cuts will increase the buckling load factor, you must also make sure that the cuts don't introduce high stress concentrations.

Tutorial wrap-up

You've completed the Thermal Analysis tutorial.

Delete or put away the FE model and the part. This part is not used in any other tutorials.

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
Solving (heat transfer structural loads)

Applying Boundary Conditions

Applying Heat Transfer Loads

Defining Temperatures (as a structural load)

Simulation: Model Solution/Optimization User's Guide

Using the Solvers

Using Heat Transfer Analysis

Related information for TMG and ESC is found in:

Simulation: Thermal Analysis User's Guide

TMG Thermal Analysis,

ESC I-DEAS Electronic System Cooling

What's next?

Other Advanced Projects tutorials introduce different element types and solution methods, including a series of tutorials on TMG and ESC.

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