

Boundary Condition Surface Loads

I-DEAS™ Tutorials: Fundamental Skills

This tutorial shows techniques to apply loads to the surface of a hole, such as caused by a bolt or a shaft. These techniques include partitioning the part to load a portion of the surface and applying loads in a radial coordinate system.

Learn how to:

- create surface traction loads
- partition a part to apply loads
- use a coordinate system

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

Creating Parts

- Managing Parts in Model Files
- Introduction to Simulation
- Free Meshing
- Boundary Condition Sets



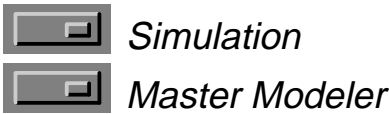
If you completed the tutorial “Boundary Condition Sets” and saved the part, you can continue to use that model file with the same part. Make sure you delete all FE models associated with the part, or you may not be able to modify the part later in this tutorial. **Skip to the page 5.**

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

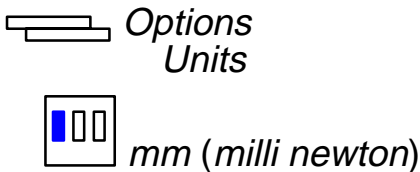
Hint



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

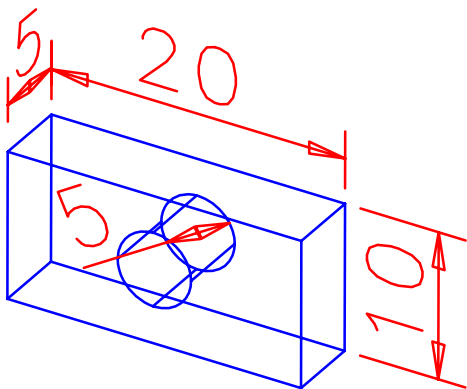
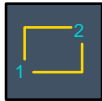


Set your units to mm.



What: Create the part shown.

Hint



What: Name the part.

Hint



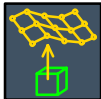
Name: Link Part

What: Create an FE model named “FEM3” associated with the part.

Hint

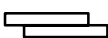


Boundary Conditions



FE Model Name: FEM3

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:

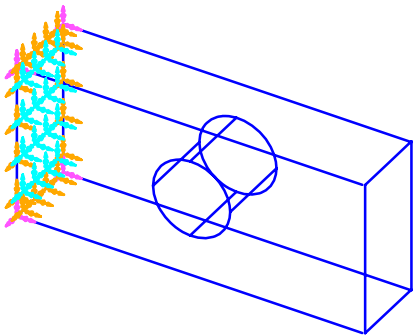
If you make a mistake at any time between saves and can’t recover, you can reopen your model file to the last save and start over from that point.

Hint

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

What: Restrain the face shown.

Hint



What: Name the restraint set “Clamped End Restraint.”

How:



Set Management form



Restraint



Rename...

Rename form



Clamped End Restraint



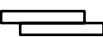
OK

Set Management form



Dismiss

Recovery Point

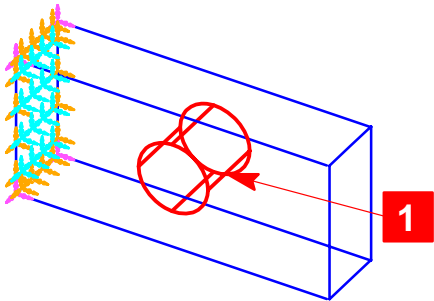


File

Save

What: Create a force on the cylindrical surface.

Hint



1



Done

Traction on Surface form



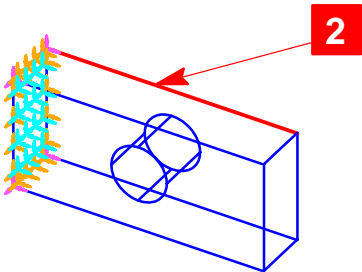
Load Set: Pin Force



Vector




2




No or Yes (to switch vector in +X direction)

Traction on Surface form

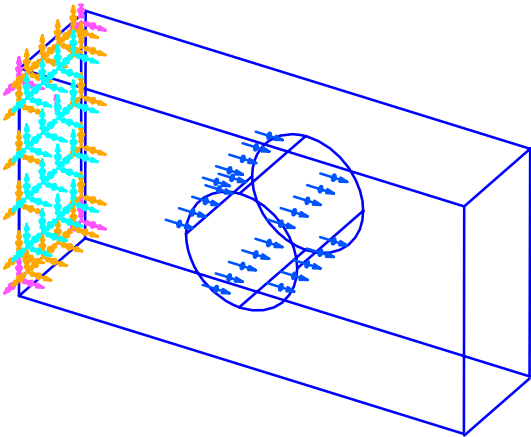


Total Force

Traction: 1000



OK



Recovery Point



File



Save

What: Create a boundary condition set.

How:



Boundary Condition Set Management form

 Tension – Pin Force

 *Linear Statics*

 *Restraint Set*

 *Load Sets: Pin Force*

Warning! Make sure the load set is selected.

 *OK*

Recovery Point

 *File*
Save

What: Create a solid mesh.

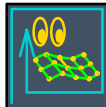
Hint



Meshing



Element Length: 2



What: Solve the model.

Hint



Model Solution



Create

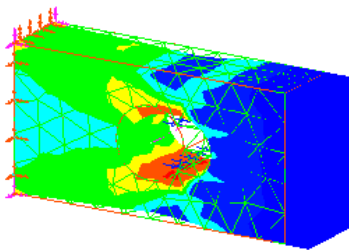


What: Display the results.

Hint

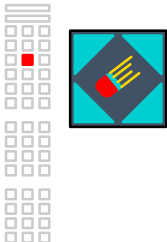


Post Processing



What: Before continuing with the tutorial, delete the results from the solve.

How:



All



Why: If finite element models contain results, the part becomes write-locked, preventing any changes.

What: Put away the FE model.

How:



Manage form



Link Part... (double-click if ... is showing)



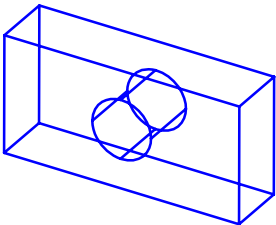
FEM3 (select)



Put Away



Dismiss



Why: When you modify a part, the FE model is also automatically updated if it's on the workbench. By putting it away while you make changes, you avoid updating the model after each change.

Recovery Point



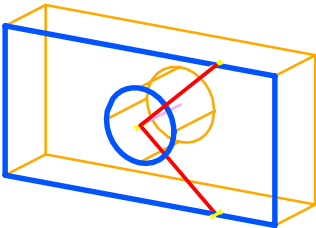
What: Partition the part to create a smaller surface to which to apply forces.

Hint



Master Modeler

Sketch two lines.



Extrude and partition.



Section Options...

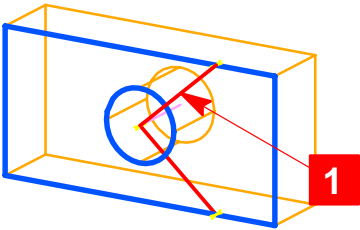


Stop at intersections (off)



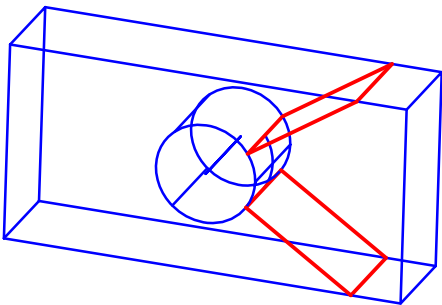
Partition

1



OK

Result

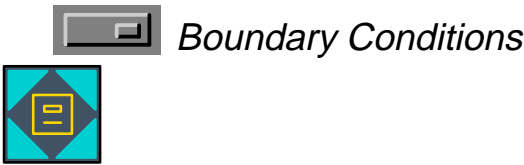


Recovery Point



What: Get the FE model “FEM3” back onto the workbench.

Hint



Things to notice

The FE model is updated to match the part. However, the force on the cylindrical surface is not available since the surface topology has changed.

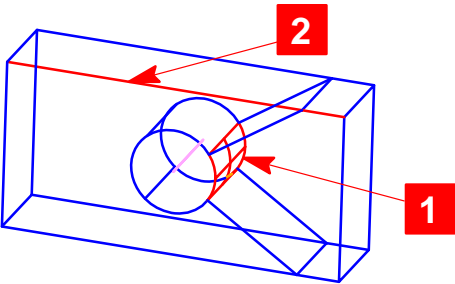
What: Create a force on one partition of the cylindrical surface.

Hint

Use the Display Filter if you have trouble picking the surface.



1 pick surface



Done



Load Set: Pin Force



2 pick edge



No or Yes (to switch vector in +X direction)



Total Force

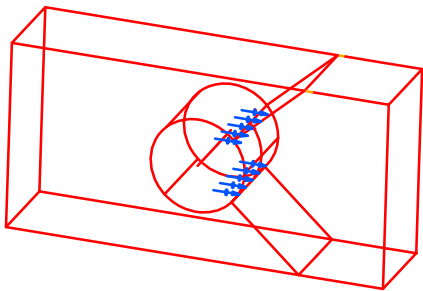


Traction: 1000



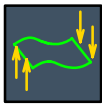
OK

Result



What: Check the boundary condition set.

Hint



Boundary Condition Set Management form

- ☐ Tension – Pin Force
- ☒ *Restraint Set*
- ☐ *Load Sets: Pin Force*
- ☐ *OK*

Things to notice

The boundary condition set created earlier is still available for use.

Recovery Point



What: Solve the model.

Hint



Model Solution

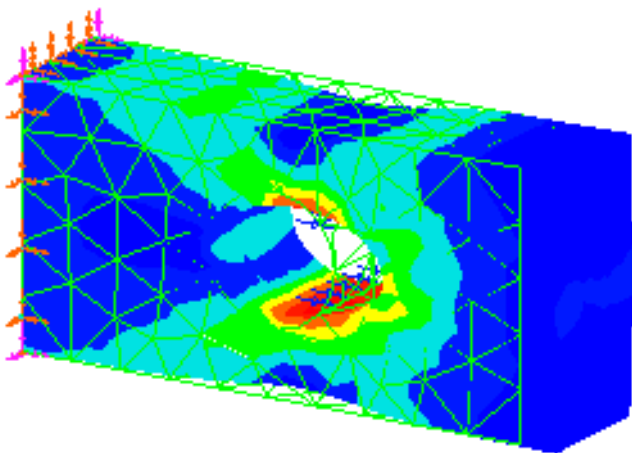


What: Display the results.

Hint



Post Processing



One way to model the condition of rotation around a pin is to use a local cylindrical coordinate system (R, Theta, Z) in the center of the hole, to restrain each node on the edge of the hole in the R direction.

What: Delete the results.

Hint



All



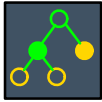
Yes



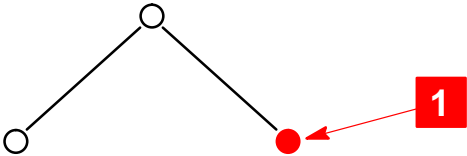
What: Delete the results, the partition, and the traction force before continuing.



Master Modeler



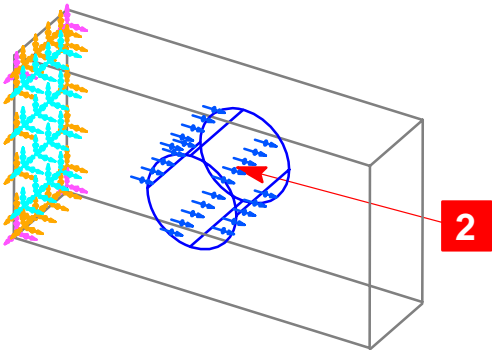
1



Boundary Conditions



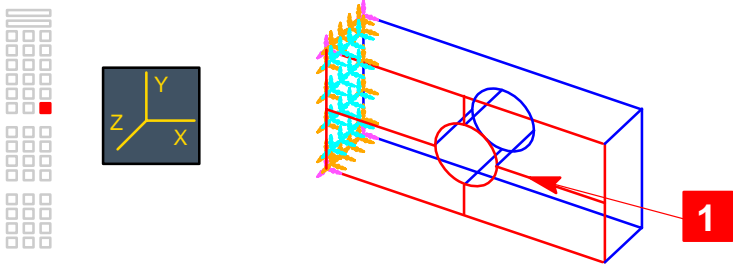
2



What: Create a cylindrical coordinate system at the hole center.

Hint
Make sure the *Centerpoints*, and *Centerlines* are on in the Part Display Filter form, using *Display Filter*.

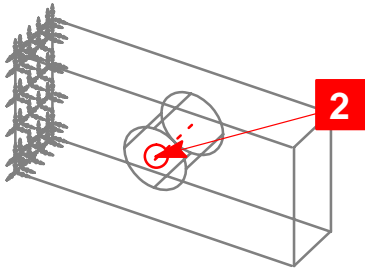
How:



1



2 pick the hole center



Why: The coordinate system will be created in the center without the *Origin* selection, but it may not remain there if you move or size the part. Selecting *Origin* ensures the coordinate system stays in the center.



Coordinate System Options form

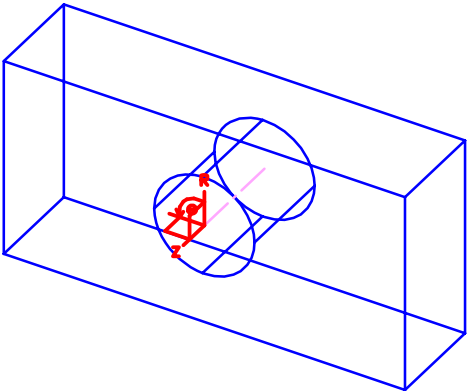
☒ *Coordinates: Cylindrical*


☐ *OK*

☒ *Done*

Things to notice

The coordinate system is not fully defined in all 6 DOF. This is not a problem in this tutorial because you will not modify the part.



 You can restrain one node against translation and still allow rotation. However, when you want to allow a **group** of nodes to be free to rotate as a unit but not translate, don't restrain each node against translation. Restraining multiple nodes against translation in effect restrains overall rotation.

Recovery Point

☐ *File*

☐ *Save*

What: Restrain the cylindrical surface in the radial direction to simulate a press fit of .5mm.

How:



1 pick cylinder surface



Done

Displacement Restraint on Surface form



Select

2 pick coordinate system

Displacement Restraint on Surface form



X translation: .5



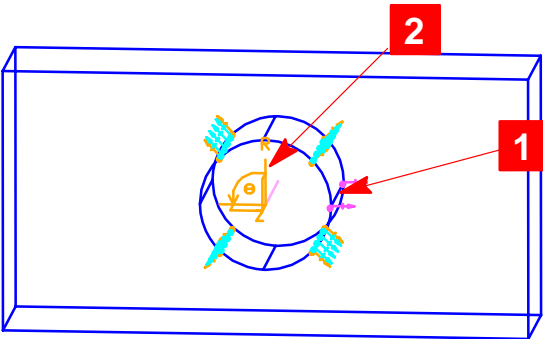
X translation: Constant



(all others free)



OK



What: Solve the model and display results.

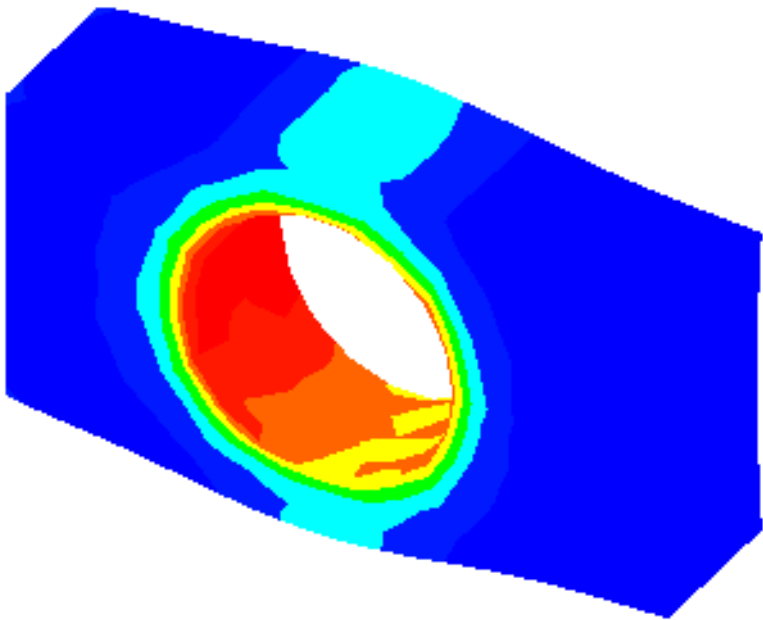
Hint



Model Solution



Post Processing



You could also use contact elements to simulate a press fit.

You have completed the Boundary Condition Surface Loads tutorial.



Do not delete the part “Link Part.” It is used in a later tutorial.

What: Delete any FE models. They won’t be needed in any other tutorials.

Hint



Save your model file.



See also...

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



Simulation: Finite Element Modeling User’s Guide
Simulation Techniques and Examples
Applying Boundary Conditions
Applying Boundary Conditions
Icon Overview for Boundary Conditions
Applying Boundary Conditions
Defining Boundary Conditions
Applying Structural Loads
Defining Contact

What's next?

Continue with the tutorial “Boundary Condition Symmetry” for more information on boundary conditions. The tutorial “Preparing a Model for Analysis” has related information on partitioning part volumes.

To exit this tutorial, select:



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

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