

Boundary Condition Sets

I-DEAS™ Tutorials: Fundamental Skills

You should already be familiar with simple forces and restraints. This tutorial covers the more advanced types of boundary condition sets (loads and restraints).

Learn how to:

- create restraints and restraint sets
- create loads and load sets
- create a boundary condition set
- create a pressure
- compare boundary condition types

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

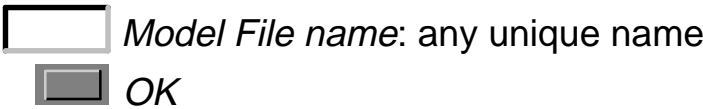
Creating Parts

- Managing Parts in Model Files
- Introduction to Simulation
- Free Meshing

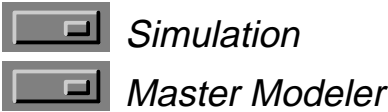
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



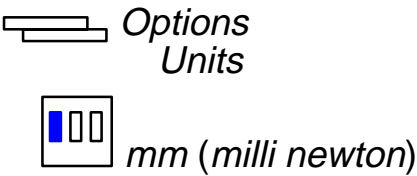
Open Model File form



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

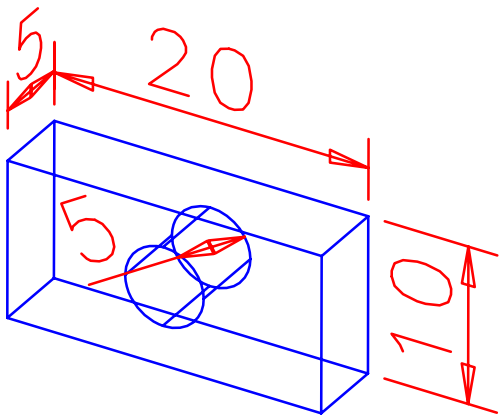


Set your units to mm.

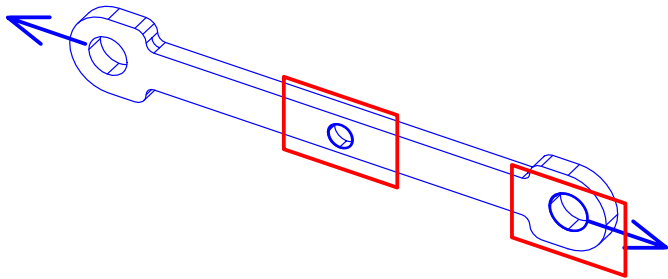


What: Sketch and extrude the shape to the dimensions shown.

Hint



Why: This part is used to represent different sections of typical machine linkage.



What: Name the part.

Hint



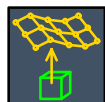
Name: Link Part

What: Create an FE model named “FEM1” associated with the part.

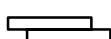
Hint



Boundary Conditions



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:

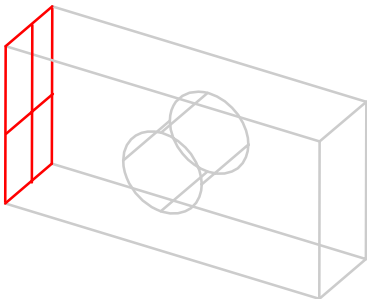
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

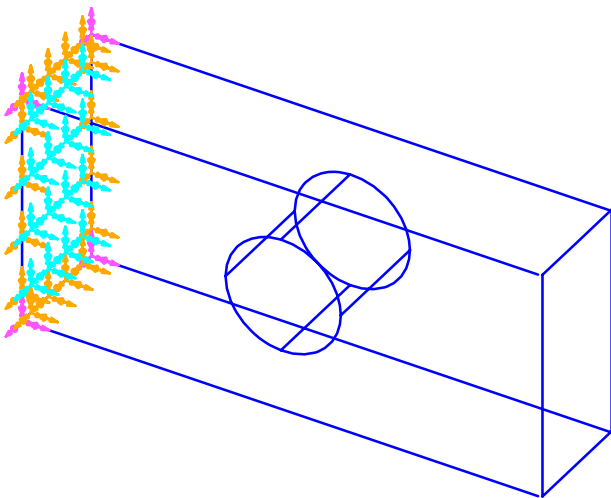
To reopen your model file to the previous save, press Control-Z.

What: Restrain the face shown.

Hint

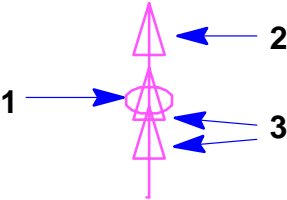


Displacement Restraint on Surface form



Things to notice

- Restraint symbols are created.
- Restraints are created on the edges, face, and vertices in different colors, and with a variety of symbols.

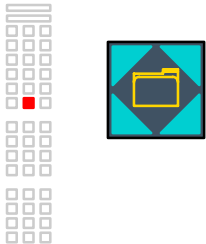


1. Circles indicate geometry-based restraints.
2. Single arrowheads indicate translational directions.
3. Double arrowheads indicate rotations.

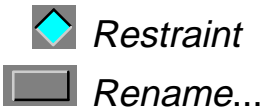
What: Rename the restraint set.

Why: When you use multiple restraint and load set combinations, it helps to use meaningful names.

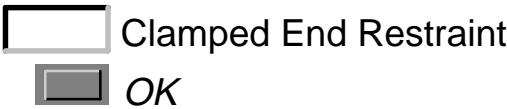
How:



Set Management form



Rename form



Set Management form



Recovery Point

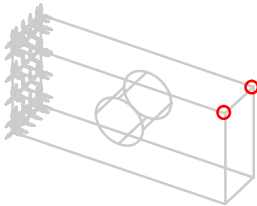


You can create forces on:

- vertices
- edges
- surfaces
- specific points on edges and surfaces

What: Create forces on the two vertices shown.

How:



shift-pick two vertices

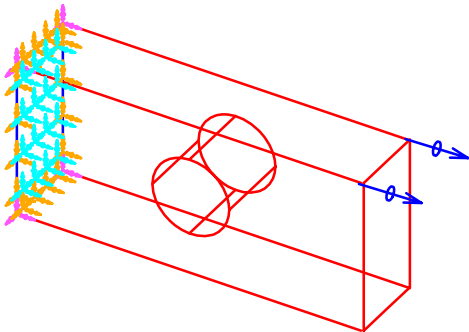


Done

Force on Vertex/Location on Geometry form

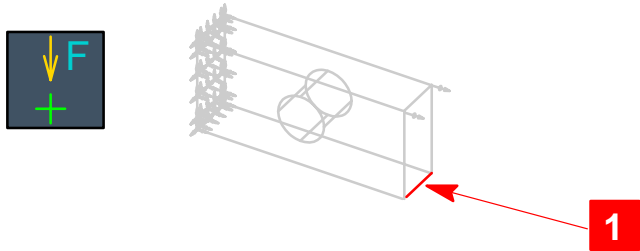
X Force: 250

OK



What: Create a force on the bottom edge.

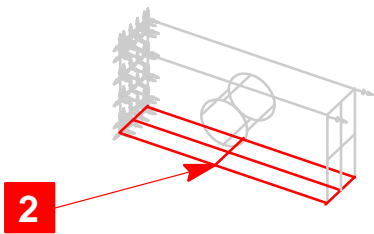
How:



1



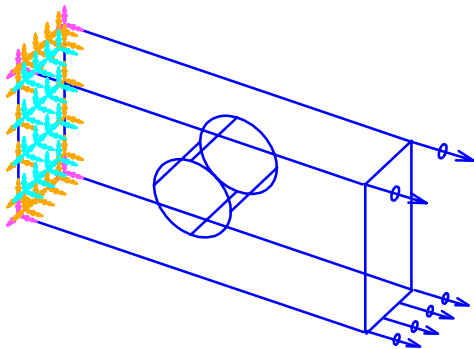
2



Force on Edge form

☒ Total Force

In Plane Force: -500



What: Rename the load set.

Why: As mentioned earlier, it helps to use meaningful names when you use multiple restraint and load set combinations.

How:



Set Management form



Load



Rename...

Rename form



Point and Edge Loads



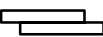
OK

Set Management form



Dismiss

Recovery Point

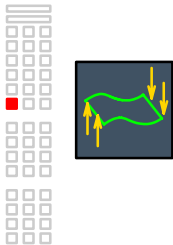


*File
Save*

A boundary condition set contains one restraint set and multiple load sets. Depending on the solution type you plan to use, other sets, such as temperature sets, may also be included in the boundary condition set.

What: Create a boundary condition set to analyze the effects of the edge and vertex forces.

How:



Boundary Condition Set Management form

Hint

Name the boundary condition set in the initial name field in the upper-left corner of the form:

Tension – Point and Edge Loads

☐

Linear Statics

☒

Restraint Set

☒

Load Sets: Point and Edge Loads
(select)

Recovery Point



What: Create a solid mesh.

Hint



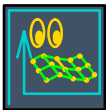
Meshing



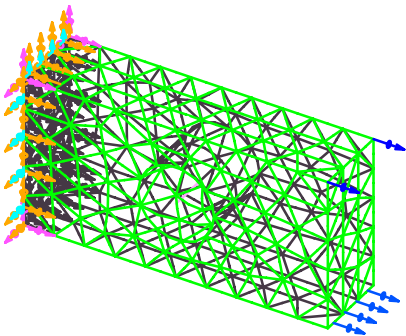
pick part



Element Length: 2



Modify Mesh Preview form



Keep Mesh

What: Solve the model.

Hint

 *Model Solution*



 *Create*

 *OK*

 *Dismiss*



Check I-DEAS List.

Look for any warning or error messages.

What: Display results with all elements selected.

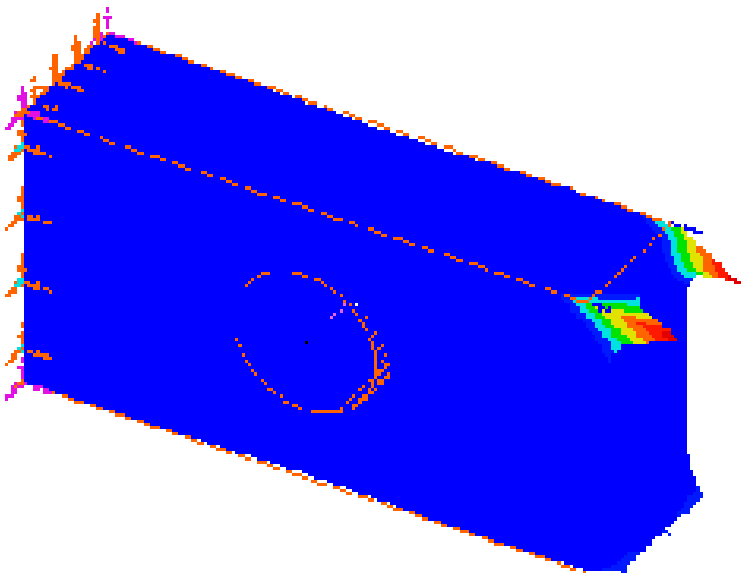
Hint



Post Processing



(default is all)



Things to notice

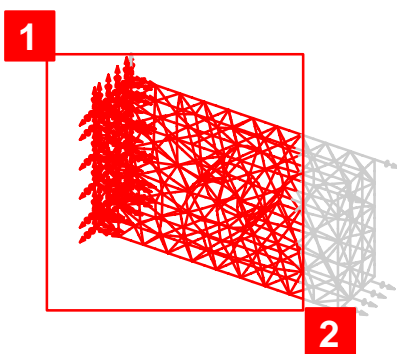
The forces at the top create considerable local stress at the vertices, while the force applied to the bottom edge has a less dramatic effect. The deformation and stress about the hole are somewhat masked by the high stresses at the vertices.

What: Display results with elements nearer the hole (away from the forces) selected.

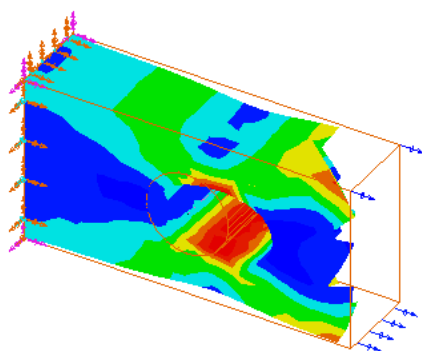
Hint



use area-select to pick most of the elements.



Done

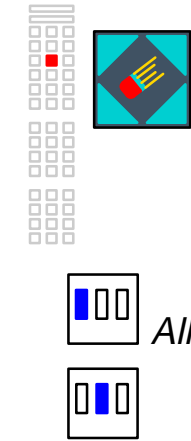


Things to notice

When the local effect of the forces is not displayed, the deformation and stress about the hole are more visible. (This is an example of St. Venant's principle.)

What: Delete the results from the solve.

How:



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

Recovery Point

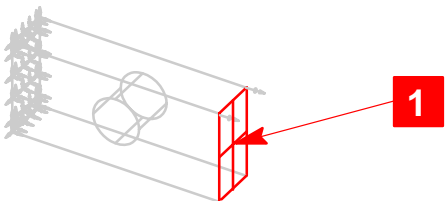


What: Create a pressure on a face, creating a new load set at the same time.

How:



Boundary Conditions



1



Done

Pressure on Surface form



Load Set: Face Pressure



Total Force

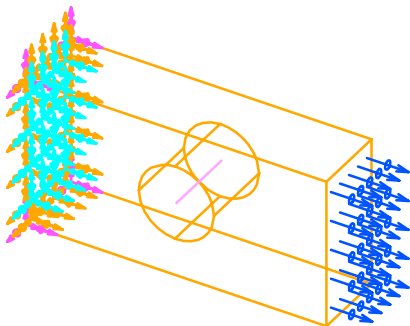


Force: -1000



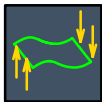
OK

Things to notice By entering a new load set name, you created a new load set.



What: Create a new boundary condition set to analyze the effects of the face pressure.

Hint



Tension – Face Load



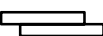
Load Sets: Face Pressure

Warning! Make sure the Face Pressure load set is selected, not the “Point and Edge Loads” load set.



OK

Recovery Point



File

Save

What: Solve the model.

Hint



Model Solution



Create

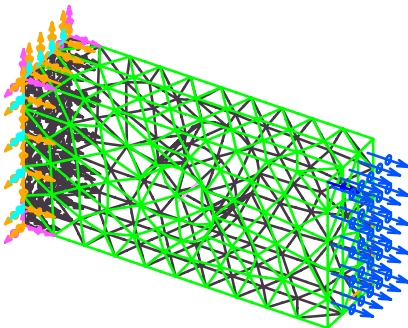
*Boundary Condition Set:
Tension – Face Load*



OK



Dismiss

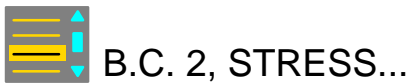


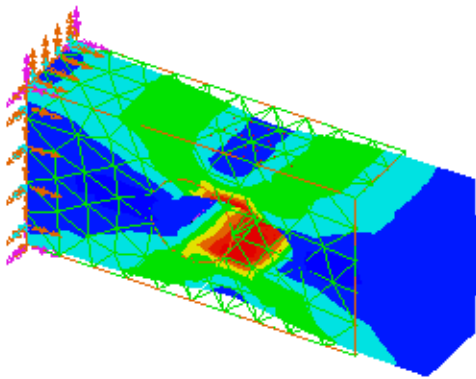
What: Display the results.

How:



Results Selection form



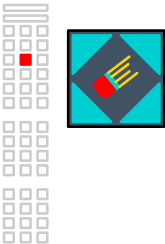


Things to notice

In both this case and in the preceding example, the stresses around the hole were valid. But in this example, you have avoided singularity where the load is applied.

What: Delete the results.

Hint



Recovery Point

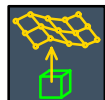


What: Create another FE model (FEM2) associated with the same part.

Hint



Boundary Conditions



Do not turn on the toggle:



Geometry Based Analysis Only

Why: With this toggle on, you can't create FE-based boundary conditions.

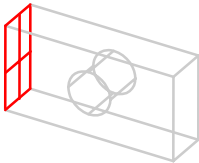
- Boundary conditions created on part geometry are called geometry-based. The software converts these to nodal and element values to solve the model.
- Boundary conditions created directly on nodes and elements are called FE-based.



Apply geometry-based boundary conditions before meshing. FE-based boundary conditions must be applied after meshing.

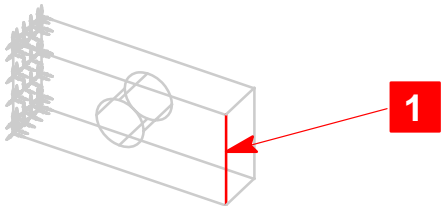
What: Restrain the face shown.

Hint



What: Create a force on the edge shown.

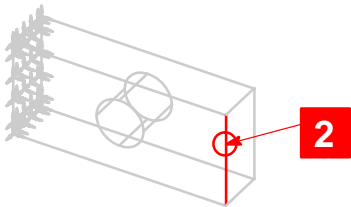
Hint



1



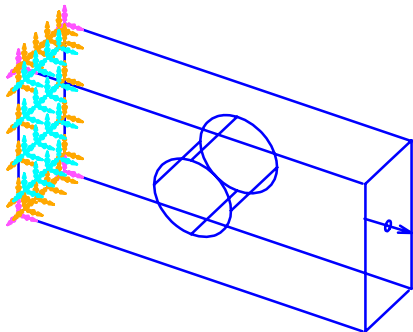
Locations on Edge



2



X Force: 500

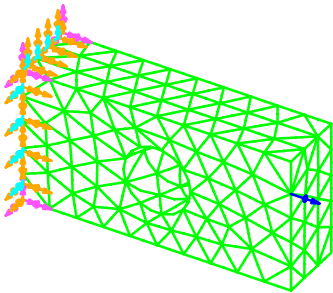
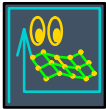


What: Create a solid mesh.

Hint



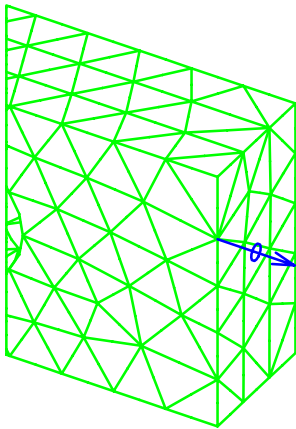
Meshing



Keep Mesh

Things to notice

The mesh automatically placed a node where the force was placed on the edge.



What: Create a force on a node as shown.

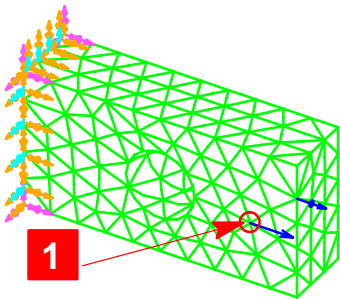
Hint



Boundary Conditions



1 pick a node (zoom in if necessary)



X Force: 1000

Warning! Before going to the next page, delete FEM1.

Why: If you performed the optional solution pages, FEM1 will contain results, which will lock the part model so you can't modify it.

Hint



Link Part... (double click)



FEM1 (select)



What: Modify the part's wireframe, then update.

Hint



Master Modeler

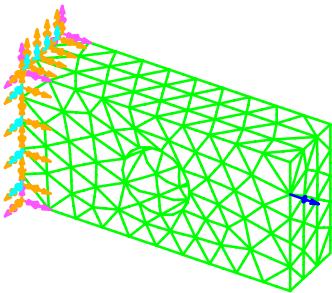


pick the part



Wireframe

Change any dimension.



FE-based boundary conditions are lost if the mesh is deleted or if geometry changes. To prevent this, remember to select this toggle when creating an FE model:



Geometry Based Analysis Only

Tutorial wrap-up

You have completed the Boundary Condition Sets 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

- Applying Boundary Conditions

 - Icon Overview for Boundary Conditions

 - Applying Boundary Conditions

 - Defining Boundary Conditions

 - Applying Restraints

 - Applying Structural Loads

What’s next?

For more information on boundary conditions, continue with the tutorials “Boundary Condition Surface Loads” and “Boundary Condition Symmetry.”

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